Autodesk® Mechanical Desktop®

autodesk®

User's Guide

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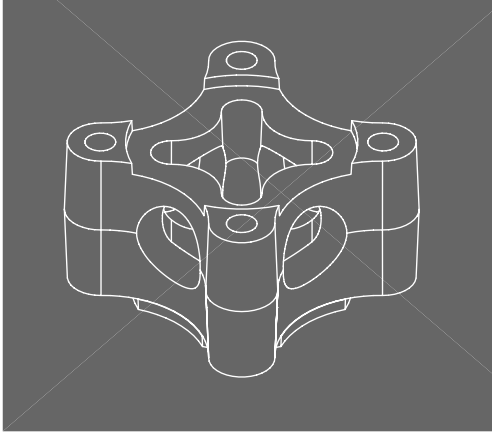
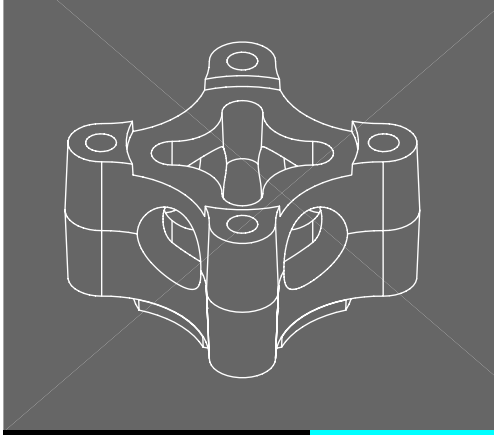


Part I
Getting Started with Autodesk® Mechanical Desktop®

Part I provides information for getting started with your Mechanical Desktop 6 software. It includes information to help in the transition from AutoCAD® and the migration of files from previous releases. It explains the user interface and the basics of modeling in the different work environments in Mechanical Desktop.

In addition, Part I provides a guide to both the print and online documentation that you received with your Mechanical Desktop software. Information about training courseware and Internet resources are also included.

Welcome



In This Chapter

1

This chapter provides an overview of the capabilities of Autodesk® Mechanical Desktop® 6 software. You learn about the transition from AutoCAD®, data exchange, and the migration of files from previous releases with the Mechanical Desktop Migration Assistance.

- About Mechanical Desktop
- Making the transition from AutoCAD
- Migrating files from previous releases

What is Autodesk Mechanical Desktop?

Mechanical Desktop is a powerful and easy-to-use 3D parametric modeler used in mechanical design. Built on AutoCAD 2002, the Mechanical Desktop 6 design software package includes:

- AutoCAD Mechanical 6 with the power pack (2D Parts and Calculations)
- Mechanical Desktop 6 with the power pack (Mechanical Desktop 6, 3D Parts and Calculations)
- AutoCAD 2002

When you start Mechanical Desktop 6, you have the option to run it with or without the power pack.

The Mechanical Desktop software provides design tools to

- Create parts from sketched and placed features
- Combine parts and toolbodies
- Build assemblies and subassemblies
- Define scenes for drawing views
- Set up drawing sheets and views
- Annotate drawings for final documentation
- Manage and reuse design data
- Migrate and edit legacy solids data

Productivity and collaboration tools in Mechanical Desktop enable you to improve workflows and comply with company practices.

Web tools are provided in a design portal called the Today page. From the Today page, you can

- Start a new drawing or open an existing drawing
- Access symbol libraries
- Communicate to design team members through a Web page you create from a template provided
- Link directly to design information on the Web
- Link directly to Autodesk Web pages

For more information about the Today page, see “Mechanical Desktop Today” on page 14.

Making the Transition from AutoCAD

Mechanical Desktop 6 is built on AutoCAD 2002 and uses many of the tools you may already be familiar with. Because Mechanical Desktop is a parametric modeling program, exercise care in using standard AutoCAD commands.

In the sketching stage, you can use any AutoCAD command to create the geometry for your sketch. You can use AutoCAD drawing and editing tools to edit sketch geometry after it has been consumed by a feature.

In general, follow these rules:

- Use Mechanical Desktop dimensions. AutoCAD dimensions are not parametric and cannot control the size, shape, or position of Mechanical Desktop parts and features.
- Use sketch planes and work planes to control the UCS orientation. Using the AutoCAD UCS command does not associate the current plane with your part.
- Do not use the command EXPLODE. Exploding a part deletes the part definition from a Mechanical Desktop drawing.
- Use the Assembly Catalog or the Browser to insert external part files into drawings and externalize part files. Using the AutoCAD INSERT, WBLOCK, XREF, and XBIND commands could corrupt Mechanical Desktop data.
- Use the Mechanical Desktop drawing view commands to create drawing views. The AutoCAD MVIEW command does not create associative views of your parts.

Migrating Files from Previous Releases

In Mechanical Desktop 6, you can add more than one part to a part file for creating combined parts. The first part becomes the part definition, while all other parts become unconsumed toolbodies. You combine toolbodies with each other and the first part to create a complex part.

To migrate parts from a part file that contains more than one part and was created before Mechanical Desktop Release 2, you need to follow specific procedures. See "Running the Desktop File Migration Utility" in the *Autodesk Mechanical Products Installation Guide* on the product CD.

The File Migration Tool (FMT) is a component of Mechanical Desktop Migration Assistance, an independent Visual Basic (not VBA) application located on your product CD. The FMT migrates multiple files from previous releases of Mechanical Desktop to the current format. You can install Mechanical Desktop Migration Assistance during or after the installation of your Autodesk mechanical product.

To install the Mechanical Desktop Migration Assistance from your product CD

1. Hold down the SHIFT key while you insert the product CD into the CD-ROM drive. This prevents Setup from starting automatically.
2. In the file tree of the CD-ROM drive, navigate to the *Migrate* folder and click *setup.exe*.
3. Respond to the directions in the Mechanical Desktop Migration Assistance installation dialog boxes.

NOTE For more information about installing the Migration Assistance and running the FMT, see "Mechanical Desktop Migration Assistance" in the *Autodesk Mechanical Products Installation Guide* on your product CD.

Data Exchange

During your design process, you may want to complement Mechanical Desktop with other computer-aided design (CAD) software. Mechanical Desktop 6 includes the STEP translator and the IGES Translator. The Standard for the Exchange of Product Model Data (STEP) is International Standards Organization (ISO) 10303. The Initial Graphics Exchange Specification (IGES) is the ANSI standard for data exchange between CAD systems and is supported by many CAD vendors.

The IGES Translator is compliant with the most recent version of IGES and related standards. It supports both the United States Department of Defense Continuous Acquisition and Life-cycle Support initiative (CALS) and the Japanese Automotive Manufacturers Association subset of IGES (JAMA).

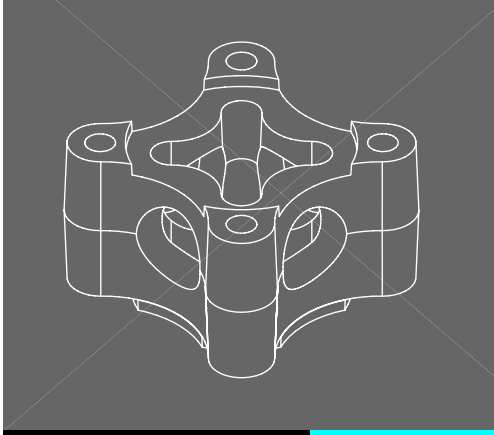
Besides creating and maintaining a flexible CAD tool environment, the Translator preserves the investment you have made in previous designs developed with other CAD systems.

The Translator supports the following types of design objects:

- 2D and 3D wireframe geometry
- Ruled, parametric, and NURBS surfaces
- Mechanical Desktop and AutoCAD native solids, and IGES boundary representation solids (B-rep).

For more information, see STEP and IGES in the Mechanical Desktop Help.

Modeling with Autodesk® Mechanical Desktop®



In This Chapter

2

- Mechanical Desktop basics
- Mechanical Desktop work environments

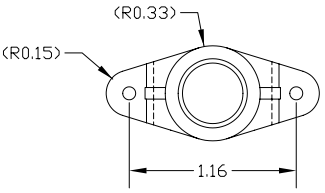
This chapter describes the basic concepts of mechanical design with Autodesk Mechanical Desktop software, including fundamentals of parametric design.

If you understand the underlying concepts in this chapter, you can become proficient in using the Mechanical Desktop software.

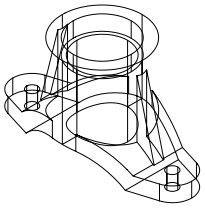
Mechanical Desktop Basics

Mechanical Desktop is an integrated package of advanced 3D modeling tools and 2D drafting and drawing capabilities that helps you conceptualize, design, and document your mechanical products.

You create models of 3D parts, not just 2D drawings.
You use these 3D parts to create 2D drawings and 3D assemblies.

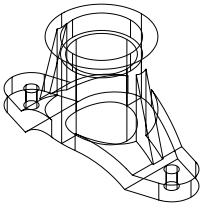


2D drawing

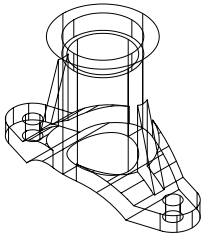


3D part

Mechanical Desktop, a dimension-driven system, creates parametric models.
Your model is defined in terms of the size, shape, and position of its features. You can modify the size and shape of your model, while preserving your design intent.

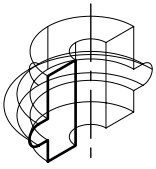


original part

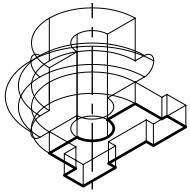


revised part

You build parts from features—the basic shapes of your part.
Building blocks like extrusions, lofts, sweeps, bends, holes, fillets, and chamfers are parametrically combined to create your part.

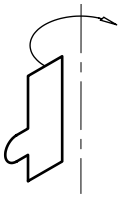


revolved feature

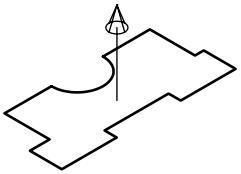


extruded feature

You create most features from sketches.
Sketches can be extruded, revolved, lofted, or swept along a path to create features.

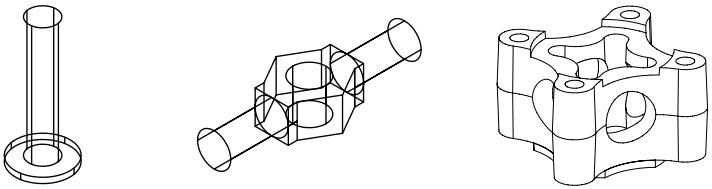


sketch for revolved feature



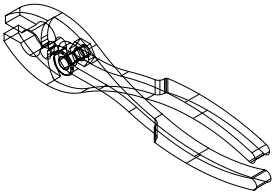
sketch for extruded feature

You work in the Part Modeling environment to create single parts.
In this environment, only one part can exist in a drawing. Additional parts become unconsumed toolbodies for the purpose of creating a combined part. Use part files to build a library of standardized parts.



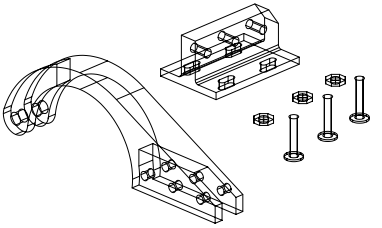
examples of single part files

You work in Assembly Modeling to create multiple parts and assemblies.
In this environment, any number of parts can exist in one drawing. Parts can be externally referenced from part and assembly files, or localized in the assembly drawing.

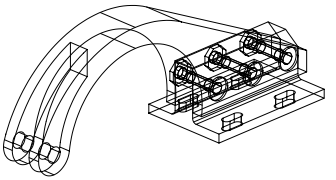


assembly file containing four external part files

Individual parts can be fit together to create subassemblies and assemblies.
Assembly files contain more than one part. Parts are fit together using assembly constraints to define the positions of the individual parts that make up your final product.



individual parts in an assembly file



completed assembly

For standard parts, you can define different versions using a spreadsheet.
Instead of a large library of parts that differ only in size, like springs, bolts, nuts, washers, and clamps, you can create one part and define different versions of that part in a spreadsheet that is linked to your drawing.

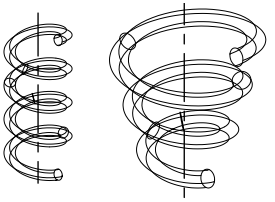
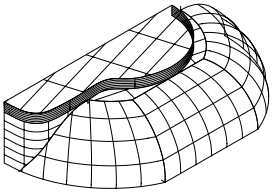
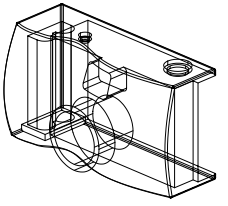


table driven part versions

You can also create 3D surface models.
Surface modeling is useful in the design of stamping dies, castings, or injection molds. You can also use surfaces to add to or cut material from a solid part to create hybrid shapes.

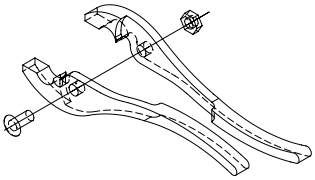


surfaces used to create a part



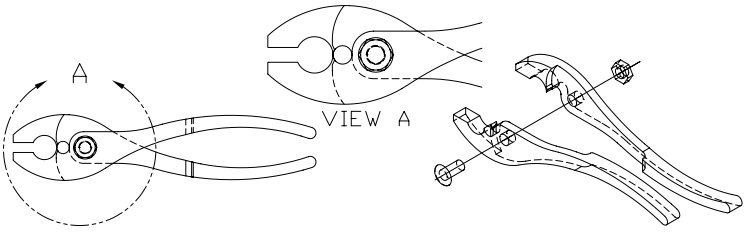
surface cut applied to a part

You can create scenes to define how your design fits together.
To better conceptualize the position of the parts in your assembly, you define scenes using explosion factors, tweaks, and trails that illustrate how your design is assembled.



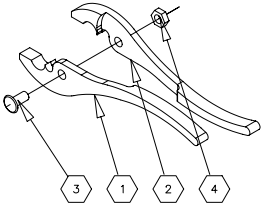
exploded scene

You can create base, orthogonal, isometric, section, and detail views.
To document your design, drawing views can be created from scenes, parts, or groups of selected objects. Any design changes are automatically updated in these drawing views.



parametric drawing views

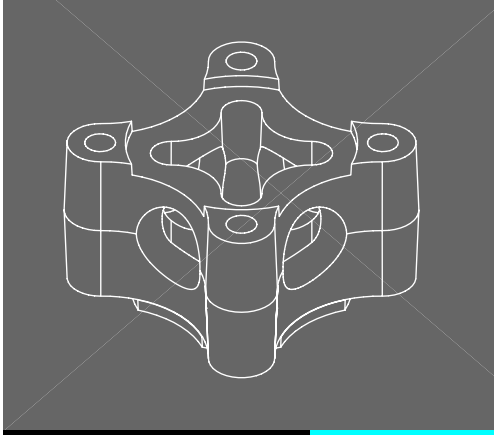
Add annotations and additional dimensions to finalize your documentation.
After you have created drawing views, finalize your design by adding balloons, bills of material, notes, reference dimensions, and mechanical symbols.



Parts list			
Item	Part No.	Vendor	Qty.
1	PLIERT	ABC	1
2	PLIERB	ABC	1
3	HEXBOLT	XYZ	1
4	HEXNUT	XYZ	1

annotations added to drawing

The User Interface



In This Chapter

3

- The Today window
- Work environments
- Mechanical Desktop interface
- Working in the Browser
- Methods for issuing commands

When you start the Autodesk® Mechanical Desktop® 6 software, a page called the Today window is displayed. This chapter provides an overview of the options on the Today window to help manage your work, collaborate with others, and link to information on the Web.

Information about the work environments and the user interface are included to help you get started using the Mechanical Desktop software.

Mechanical Desktop Today

The first time you open the Mechanical Desktop 6 program, the Today window is displayed on top of the program interface, along with instructions about how to use it. The Today feature is a powerful tool that makes it easy to manage drawings, communicate with design teams, and link directly to design information.

In the Today Window, you can expand the following options for access to the the services you require.

My Workplace	Connect directly to files on your computer and your local network.
My Drawings	Open existing drawings, create new ones, or access symbol libraries.
Bulletin Board	Post your own Web page with links to block libraries, CAD standards, or other folders and directories on your company network. CAD managers can use the Bulletin Board to communicate with their design teams. An HTML bulletin board template is provided.
The Web	Connect directly to the Internet.
Autodesk Point A	Link directly to design information and tools such as Buzzsaw.com on the Web. Use the units converter, link to Autodesk Web sites, and much more. Login and create your free account. Customize the information in Autodesk Point A for your specific needs.

You can close the Today Window and use the File menu to create new drawings or open existing drawings.

To reopen Today, in the Assist menu choose Mechanical Desktop Today.

If you prefer not to see the Today Window when you start Mechanical Desktop, you can turn it off in Assist ➤ Options ➤ System ➤ Startup.

Mechanical Desktop Environments

Mechanical Desktop has two working environments: Assembly Modeling and Part Modeling.

Assembly Modeling Environment

This is the environment Mechanical Desktop uses when you start the program or create a new file by using File ➤ New. Any number of parts and subassemblies can coexist in the same drawing.

The advantages of the Assembly Modeling environment are

- More than one part can be created in the same drawing.
- Individual part files, and other assemblies or subassemblies, can be externally referenced or localized and used to build a complex assembly.
- Different versions of a part can be displayed in the same file.
- Scenes containing explosion factors, tweaks, and trails can be created.

There are three modes in the Assembly Modeling environment: Model, Scene, and Drawing.

Model Mode

In Model mode, you create as many parts as you need. Parts may be local or externally referenced. Create subassemblies and save them for use in larger assemblies. Build assemblies from any number of single part files, subassemblies, and assemblies. You can also generate a BOM (Bill of Material) database so a list of parts can be placed in your final drawing.

Scene Mode

In Scene mode, you set explosion factors for your assembled parts and create tweaks and trails. These settings govern how your drawing views represent your assemblies.

Drawing Mode

In an assembly file, you can place balloons to reference the parts in your assembly. You can create a parts list with as much information as you need to define your parts. To illustrate how parts in an assembly fit together, you can create base views on exploded scenes.

Part Modeling Environment

To begin a new drawing in the Part Modeling environment, choose File ➤ New Part File. Only one part may exist in the drawing. If you add more parts, they automatically become unconsumed toolbodies. You use toolbodies to create complex combined parts.

The advantages of the Part Modeling environment are

- A library of standard parts can be created for use in assembly files.
- The interface is streamlined to allow only those commands available in a part file.
- File sizes are minimized because the database doesn't need additional assembly information.

There are two modes in the Part Modeling environment: Model and Drawing.

Model Mode

In Model mode, you build and modify your design to create a single parametric part. The part takes the name of the drawing file.

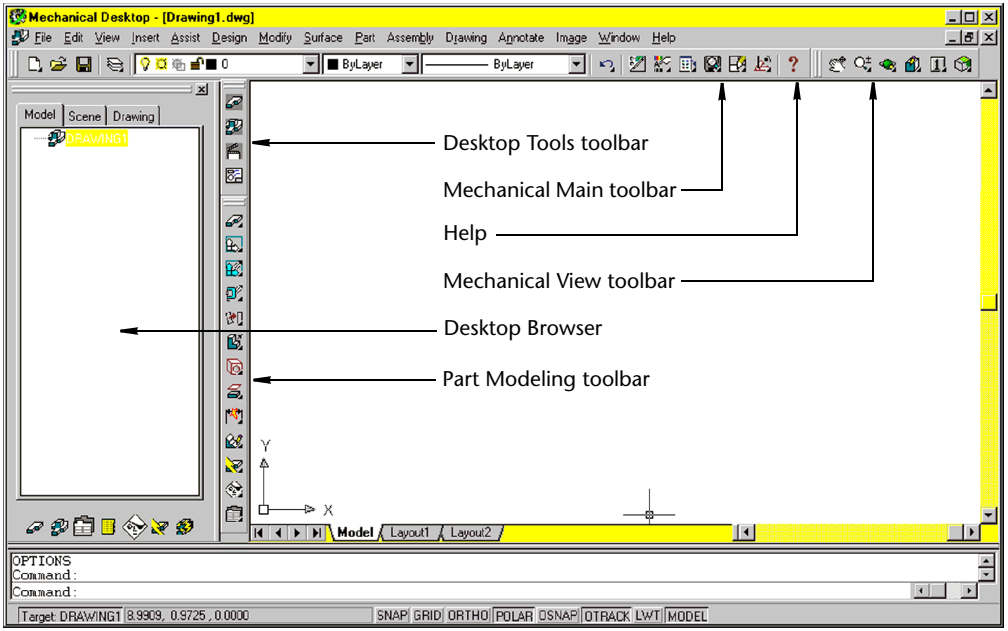
Drawing Mode

In Drawing mode, you define views of your part and place annotations for documentation. You can also create a parts list and balloons to reference a combined part and its toolbodies.

Mechanical Desktop Interface

When you open a new or existing drawing in Mechanical Desktop 6, four toolbars and the Desktop Browser are displayed.

- The Mechanical Main toolbar provides quick access to select commands from the AutoCAD Standard and the Object Properties toolbars, some Mechanical Desktop commands, and the Web. Icons are available for direct links to Mechanical Desktop Today window and Web tools such as, Point A, Streamline, RedSpark, MeetNow, Publish to Web, and eTransmit.
- The Desktop Tools toolbar acts as a toggle, giving you quick access to Part Modeling, Assembly Modeling, Scenes, and Drawing Layout.
- The Part Modeling toolbar is the default, but, when you use the Desktop Tools toolbar or the Desktop Browser to switch modes, the toolbar representing the mode you have chosen is displayed.
- The Mechanical View toolbar is designed to give you full control over how you view your models, including real-time pan, zoom, dynamic 3D rotation, and shading commands.
- The Desktop Browser is docked at the left side of the screen.



There are four main toolbars controlled by the Desktop Tools toolbar: Part Modeling, Assembly Modeling, Scene, and Drawing Layout.

Part Modeling
Assembly Modeling
Scene
Drawing Layout

If you begin a drawing in the Part Modeling environment, the Desktop Tools toolbar changes to display three buttons that control the Part Modeling, Toolbody Modeling, and Drawing Layout toolbars.

Part Modeling
Toolbody Modeling
Drawing Layout

In addition to controlling the Mechanical Desktop toolbars, the Desktop Tools toolbar switches between Part, Toolbody/Assembly, Scene, and Drawing modes.

For a complete description of Mechanical Desktop toolbars, see appendix A, "Toolbar Icons."

Desktop Browser

When you start Mechanical Desktop 6, the Desktop Browser is displayed in the default position at the left of your screen.

Docking the Desktop Browser

Right-click the gray area at the top of the Browser for a context menu of docking controls. You can turn the following Browser docking options on and off.

Allow Docking

With Docking on, you can drag a corner of the Browser to change its shape and size, and you can drag the Browser to a new location on your screen.

To return the Browser to its default position, turn on Allow Docking, and double-click the Browser title bar.

AutoHide

With AutoHide on, choose Collapse to minimize the Browser. When you move the cursor over and off of the Browser, it expands and collapses.

Choose Right or Left to hide the Browser off a side of the screen. When you move your cursor to the corresponding edge of the screen, the Browser is displayed. Move the cursor off the Browser, and it is hidden again.

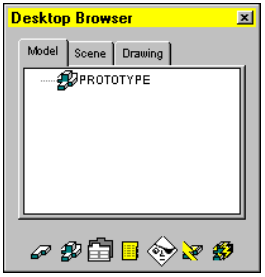
To turn AutoHide off, in the Browser docking menu choose AutoHide ➤ Off.

Hide

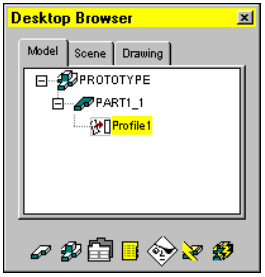
Hides the Browser entirely. To restore it, in the Desktop menu choose View ➤ Display ➤ Desktop Browser.

Working with the Desktop Browser

When you begin, Mechanical Desktop starts a new drawing in the Assembly Modeling environment. The assembly is named for the current file.



When you create the first sketch, a part is automatically named, numbered, and represented in the Browser. Because the first thing you create is a sketch, it is nested under the part. As these objects are created, they are displayed automatically in a hierarchy.

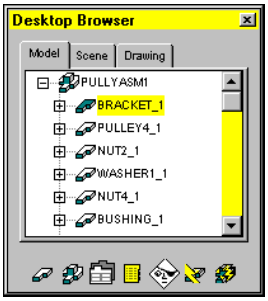


In the Browser, you can show as much or as little detail as you wish. When there is more information, a plus sign is shown beside an object. You click the plus sign to reveal more levels.

You collapse levels by clicking the minus sign beside an object, or collapse the entire hierarchy by right-clicking the assembly name and choosing Collapse from the menu.

When you start a new drawing in the Part Modeling environment, or open an existing part file, the Desktop Browser contains two tabs: Model and Drawing. In the Assembly Modeling environment, the Browser contains three tabs: Model, Scene, and Drawing. You can choose the tabs at the top of the Browser window to navigate from one mode to another.

Part Modeling environment



Assembly Modeling environment

Icons at the bottom of the Browser provide quick access to frequently-used commands.

Using the Browser in Part Modeling

When you are working in the Part Modeling environment, the Browser contains two tabs: Model and Drawing.

Model Mode in Part Modeling

In Model mode, seven icons are displayed at the bottom of the Browser.

The two at the left are quick filters. These filters are available so that you can control the visibility of features and assembly constraints in the Browser when you are creating combined parts.

The first icon, the Part filter, controls the display of assembly constraints attached to a part and its toolbodies. If the Part filter is selected, only the features of your part and its toolbodies are visible in the Browser. If it is not selected, assembly constraints are also visible.

The second icon is the Assembly filter. If you select this filter, only assembly constraints that are attached to your part and its toolbodies are visible.

The third icon accesses the Desktop Options dialog box where you control the settings for your part, surfaces, drawing views, and miscellaneous desktop preferences.

The middle icon provides immediate access to the Part Catalog. You use the Part Catalog to attach and localize external part files, and instance external and local parts in your current file for the purpose of creating combined parts.

The fifth icon opens the Desktop Visibility dialog box where you control the visibility of your part, toolbodies, and drawing objects. The sixth icon updates your part after you have made changes to it, and the last icon updates assembly constraints if you are working with a combined part.

Drawing Mode in Part Modeling

In Drawing mode, six icons are displayed at the bottom of the Browser.

The first two icons on the left are toggles to control automatic updating of your drawing views or part. The last four icons access desktop options, control visibility, and manually update your drawing views or part.

Using the Browser in Assembly Modeling

In the Assembly Modeling environment, the Browser displays three tabs: Model, Scene, and Drawing. With these tabs, you can create multiple parts, assemblies, scenes, BOMs, and documents, and you can reorder assemblies. You can localize and externalize parts in the Browser without opening the Assembly Catalog.

Model Mode in Assembly Modeling

Model mode in the Assembly Modeling environment has the same icons at the bottom of the Browser as Model mode in the Part Modeling environment. Because you are working in the Assembly environment, these icons provide more functionality.

The first icon is the Part filter which you use to control the display of the features that make up your parts. If the Part filter is selected, only part features are visible in the Browser. If it is not selected, assembly constraints are also visible.

The second icon is the Assembly filter. When you select this filter, only the assembly constraints attached to your parts are visible.

The third icon opens the Mechanical Options dialog box. From this dialog box you can manage your settings and standards for parts, assemblies, surfaces, drawings, shaft generators, calculations, standard parts, and various desktop preferences.

The middle icon provides access to the Assembly Catalog, a powerful interface for attaching and localizing external part and assembly files as well as instancing both external and local parts in your current assembly.

The fifth icon controls the visibility of parts, assemblies, drawing entities, layers, and linetypes. The sixth icon updates the active part after you have made changes to it, and the last icon updates the active assembly or subassembly.

Scene Mode in Assembly Modeling

In Scene mode, three icons are displayed at the bottom of the Browser.

The first icon accesses Desktop Options, where you can control the settings for scenes. The second icon accesses Desktop Visibility, where you can control the visibility of your parts, assemblies, and individual drawing objects. The last icon updates the active scene.

Drawing Mode in Assembly Modeling

In Drawing mode, six icons perform the same functions as those in Drawing mode in the Part Modeling environment.

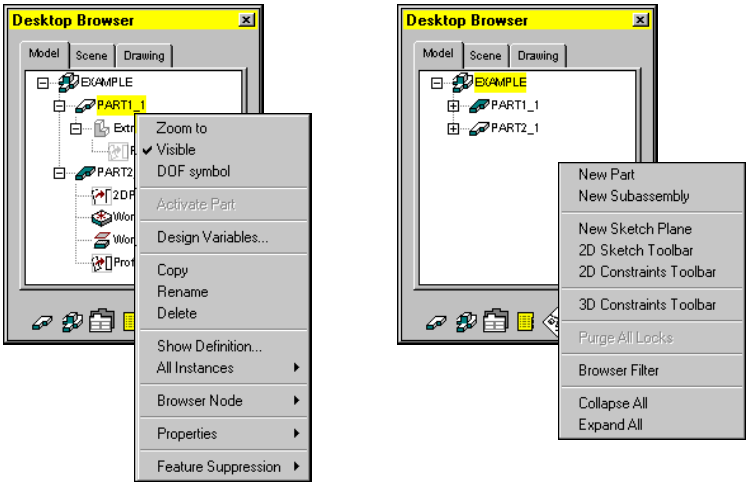
Issuing Commands

You can issue commands in several ways:

- Select an option from a right-click menu in the Desktop Browser.
- Select an option from a right-click menu in the active screen area of your drawing.
- Select a toolbar icon.
- Select an option from a pull-down menu.
- Enter the command name on the command line.
- Use an abbreviation of the command, called an accelerator key, on the command line.

Using Command Menus in the Desktop Browser

Many of the commands in Mechanical Desktop can be accessed using the Browser menus. The Browser has two types of menus. One you activate by right-clicking an existing object in the Browser. The other you activate by right-clicking the Browser background. Options that are not available are gray.



The type of object you select with a right-click determines the menu displayed. The mode you are in, Model, Scene, or Drawing, when you right-click the Browser background determines the menu displayed.

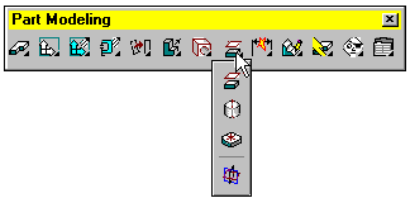
Using Context Menus in the Graphics Area

In addition to the Browser menus, context-sensitive menus are available in the graphics area during the modeling process. When you start Mechanical Desktop, the Part menu is available in the graphics area. You can toggle between the Part and Assembly menus as you build your models. When you are in Scene mode, the Scene menu is available. In Drawing mode, you can toggle between the Drawing and Annotate menus.

Using Toolbars

Toolbars have icons to represent frequently-used commands, settings, and environments. You can choose an icon instead of selecting a command from a menu or entering its name on the command line. When you pause with the mouse selection arrow on an icon, the command action is shown at the bottom of the screen. A tooltip also appears under the cursor. Click the left mouse button to select the command.

Some icons have a subtoolbar (flyout) with related icons. If the icon has a small arrow in the lower right corner, drag the mouse to reveal the additional icons, and then select one.



To hide a toolbar, click the button in its upper right corner. To unhide it, right-click any toolbar. In the pop-up menu, select the toolbar to redisplay. The toolbar is automatically redisplayed.

To reorient the Mechanical Desktop toolbars to their default positions, choose View ➤ Toolbars ➤ Desktop Express (Left). If you prefer the toolbars at the right of your screen, choose Desktop Express (Right).

You may want to view larger toolbar icons. To do so, right-click any toolbar and choose Customize. Select Large Buttons at the bottom left of the Toolbars dialog box.

If you choose Large Buttons and then dock the toolbars in the screen header area above the command line or at either side of the screen, some icons may not be visible. In that case, you can drag the toolbar onto the screen.

Using Pull-down Menus

To select a menu option, or access a submenu, hold down the left mouse button while you navigate through the menu. When you find the command you want to use, release the mouse button.

You can also access menu commands by using the keyboard. Hold down ALT while selecting the underlined letter of the menu option. For example, to select AMPROFILE from the keyboard, press ALT, then P, S, P.

Selecting Command Options from Dialog Boxes

Many commands have options within dialog boxes. As the term *dialog box* suggests, you interact by selecting options to make a particular setting active, display a list from which to choose an option, or enter a specific value. If a command has a dialog box, it is displayed when you access the command, regardless of whether you did so on the command line or from a menu or toolbar icon.

When you need information about a dialog box you are working with, click the Help button located in the dialog box.

NOTE If the Mechanical Desktop dialog boxes do not display, on the command line enter CMDDIA, and change the system variable to **1**.

Using the Command Line

You can access a command or system variable directly by entering its name on the command line. Many experienced users prefer this method because it is faster than using menus. Some experienced users are familiar with specifying command options from the command line and prefer to turn off the display of dialog boxes.

However, because many Mechanical Desktop commands require input through their dialog boxes, it is recommended that you use the dialog boxes instead of the command line to ensure that you have access to the full functionality of each feature.

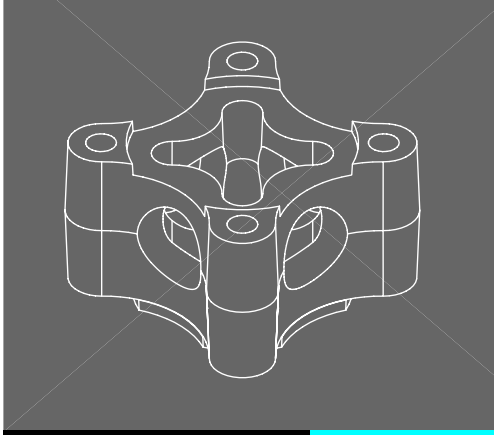
All the commands and system variables for Mechanical Desktop and AutoCAD are documented in Help.

Using Accelerator Keys

Many commands also have shortcuts called accelerator keys. To issue a command using an accelerator key, simply enter the command alias on the command line.

For a complete list of Mechanical Desktop accelerator keys, see "Accelerator Keys" in the Command Reference in Help.

Documentation and Support



In This Chapter

4

This chapter provides an overview of the printed and online documentation provided with Autodesk® Mechanical Desktop® 6. It guides you to resources for product learning, training, and support.

Read this section so that any time you need product information, you will know where to locate it.

- Mechanical Desktop print documentation
- Mechanical Desktop online documentation
- Product Support Assistance in Help
- Mechanical Desktop learning and training
- Your Internet resources

Printed and Online Manuals

The extensive set of printed and online documentation provided with your purchase of Mechanical Desktop 6 software includes the printed *Autodesk Mechanical Desktop 6 User's Guide*, *AutoCAD Mechanical 6 User's Guide*, and the *AutoCAD 2002 User's Guide*.

The online *AutoCAD Mechanical 6 and Mechanical Desktop 6 Installation Guide* is provided on the product CD.

All of the Mechanical Desktop 6 manuals are available in PDF format on the product CD, and on the Mechanical Desktop product page of the Autodesk Web site at *http://www.autodesk.com/mechdesktop* ➤ Product Information ➤ Online and Print Manuals.

Mechanical Desktop Printed Manual

The printed *Autodesk Mechanical Desktop 6 User's Guide* is divided into two parts.

Part I	An introduction to the product and information you need to get started using the software.
Part II	A set of tutorials to expand your skills in using Mechanical Desktop and understanding mechanical design. Chapters 5 through 21 focus on Mechanical Desktop, while chapters 22 through 24 focus on Mechanical Desktop with the power pack.

AutoCAD Printed Manual

The printed *AutoCAD User's Guide* contains comprehensive information and instructions for using AutoCAD. This manual is also available online in the AutoCAD Help.

Online Installation Guide

The *AutoCAD Mechanical 6 and Mechanical Desktop 6 Installation Guide* is available on the product CD. It provides the following information:

Introduction	What's in the software.
Chapter 1	System requirements and recommendations for installing and running the software.

Chapter 2	Procedures to install, upgrade, authorize, and maintain the software for a single user, and information you need to know before you begin your installation.
Chapter 3	Information for network administrators. Instructions for installing and configuring for a network environment.
Chapter 4	Technical information about environment variables and performance enhancements to optimize performance of the software.
Chapter 5	Information about cabling and option settings, plus other information necessary to link and configure plotters and printers with AutoCAD Mechanical/Mechanical Desktop.
Chapter 6	Instructions to uninstall the software, maintain your hard disk, and recover data in case of a system failure.

AutoCAD 2002 Documentation

You should be familiar with AutoCAD before you use Mechanical Desktop. The complete set of AutoCAD 2002 documentation is available in the AutoCAD Help. It includes:

- *User's Guide**
- *Command Reference**
- *Customization Guide**
- *ActiveX® and VBA Developer's Guide**
- *ActiveX® and VBA Reference*
- *AutoLISP® Reference*
- *Visual LISP™ Developer's Guide**
- *Visual LISP™ Tutorial**
- *DXF™ Reference*
- *Driver Peripheral Guide*
- *Connectivity Automation Reference*
- *Network Administrator's Guide*

AutoCAD 2002 manuals marked with an asterisk can be ordered in print from your local reseller.

The AutoCAD 2002 Learning Assistance CD that is included in your package is a multimedia learning tool for intermediate to experienced AutoCAD users.

If you currently own a valid license for an Autodesk product and require replacement media or documentation, please call the Customer Service Center at 1-800-538-6401 to order.

Mechanical Desktop Help

The Help in Mechanical Desktop provides integrated information about AutoCAD Mechanical and Mechanical Desktop.

The Help is formatted for easy navigation, and includes:

- Content organized by the major functional areas of Mechanical Desktop, with How To, Reference, and Learn About pages for each functional area
- Specific information about each of the features in the program
- Concepts and procedures for the new features in this release
- A keyword index, search function, and Favorites tab
- Printable Command Reference
- Guides to system variables and accelerator keys
- Access to Support Assistance with integrated links to solutions

For access to Help, you can choose from the following methods:

- From the Help menu, select Mechanical Help Topics.
- Select the Help button in the standard toolbar.
- Press F1. This opens the topic for an active button or command.
- Click the Help button within a dialog box.

Updating Help Files

If you have access to the Internet, you can download updated Help files from the Autodesk Web site.

To update your Help files

1. In Mechanical Desktop Today, choose Autodesk Point A. In Useful Autodesk Links, choose Autodesk Product Support Index.
2. Follow the links to Mechanical Desktop 6 product support and updates.

Product Support Assistance in Help

When you need product support, refer to Support Assistance in the Help menu. Support Assistance ensures quick access to technical support information through an easy-to-use issue/solution format with self-help tools and a knowledge base.

Product Support Assistance provides information about support options available from resellers, Autodesk System Centers (ASCs), user groups in your area, and those available directly from the Autodesk Web pages, including the Autodesk Product Support Index.

Updating the Support Assistance Knowledge Base

You can update your Support Assistance knowledge base with the latest support information about Mechanical Desktop by using the Documentation Update utility in the Support Assistance Welcome.

To update your Support Assistance Knowledge Base

1. From the Help menu, choose Support Assistance, then choose Download.
2. Follow the prompts to update your knowledge base.

Learning and Training Resources

Many sources for learning and training are listed on the Mechanical Desktop Learning and Training Web page. From the Mechanical Desktop Web site at *http://www.autodesk.com/mechdesktop,* navigate to Learning and Training. You can link directly to sources for

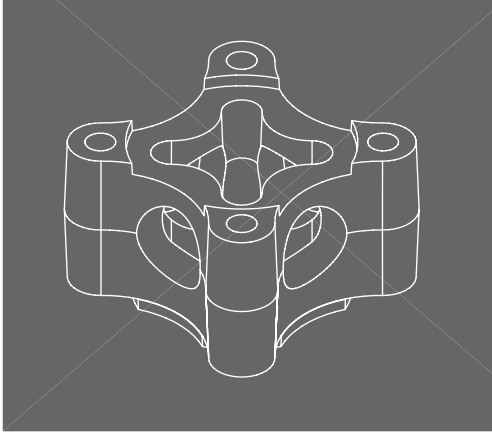
- Online courses and tutorials
- The Autodesk Official Training Courseware (AOTC)
- A list of Autodesk authorized resellers and trainers

Autodesk Official Training Courseware (AOTC) is the Autodesk-endorsed courseware for instructor-led training. To register for a training course, contact an Authorized Autodesk Training Center, Authorized Autodesk Reseller, or Autodesk System Center.

Internet Resources

Following are resources for information about Autodesk products and assistance with your Mechanical Desktop questions.

- Autodesk Web site: *http://www.autodesk.com*
- Mechanical Desktop home page at the Autodesk Web site: *http://www.autodesk.com/mechdesktop*
- AutoCAD Mechanical home page at the Autodesk Web site *http://www.autodesk.com/autocadmech*
- Mechanical Desktop discussion groups: *http://www.autodesk.com/mechdesktop-discussion*
- AutoCAD Mechanical discussion groups: *http://www.autodesk.com/autocadmech-discussion*
- To locate an authorized reseller in your area, go to: *http://www.autodesk.com/support*.



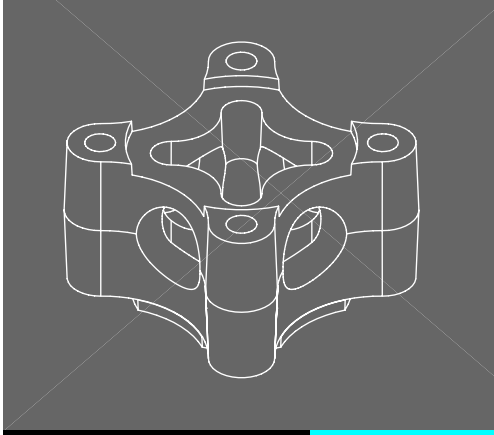
Part II
Autodesk® Mechanical Desktop® Tutorials

The tutorials in this section teach you how to use Mechanical Desktop 6, and provide a comprehensive overview of mechanical design. The lessons range from basic to advanced, and include step-by-step instructions and helpful illustrations.

You learn how to create parts, surfaces, assemblies, table driven parts, and bills of material. You will also learn how to prepare your designs for final documentation. Specific drawing files for each lesson are included with the program. These drawing files provide design elements that help you understand and learn mechanical design concepts.

There are lessons designed for learning to model with Mechanical Desktop, and others designed specifically for learning to use Mechanical Desktop with the power pack.

Using the Tutorials



In This Chapter 5

- Finding the right tutorial
- Accessing commands
- Controlling the appearance of the Desktop Browser
- Backing up tutorial files

This Introduction presents information that is useful to know before you start performing the tutorials for Autodesk® Mechanical Desktop®. It provides a summary of how the tutorials are structured, and the methods you can use to issue commands. You learn how to manipulate the position of the Browser to best suit your work space.

As you work through the tutorials, you use a set of drawing files that are included with your software. In this section, you learn how to locate, back up, and maintain these drawings.

How the Tutorials are Organized

Read the Key Terms and Basic Concepts sections at the beginning of each tutorial before you begin the step-by-step instructions. Understanding this information before you begin will help you learn.

Key Terms	Lists pertinent mechanical design terms and definitions for the lesson.
Basic Concepts	Gives you an overview of the design concepts you learn in the lesson.

The tutorials begin with basic concepts and move toward more advanced design techniques. They are presented in three design categories: part modeling, assembly modeling, and surface modeling.

For best results, run Mechanical Desktop 6 to perform the tutorials in chapters 1 through 16, and Mechanical Desktop 6 with the power pack to perform chapters 17 through 19.

Chapters 6 Through 15 Part Modeling

These tutorials guide you through the basics of part modeling. Starting with a basic sketch, you learn how to create fully parametric feature-based models and generate drawing views.

Chapters 16 Through 18 Assembly Modeling

The assembly modeling tutorials show you how to create, manage, and document complete assemblies and subassemblies, and create exploded views of your assembly design. You also learn how to use assembly techniques to build a combined part in the Part Modeling environment.

Chapters 19 Through 21 Surface Modeling

These tutorials cover the techniques of surface modeling. You start by learning how to create and edit different types of surfaces. Then you create a surface and use it to cut material from a parametric part. You also learn how to surface a wireframe model from the ground up.

Chapters 22 Through 24 2D and 3D Parts and Calculations

These tutorials focus on features in the Mechanical Desktop 6 with the power pack. Included are tutorials working with standard parts and the shaft generator and 3D finite element analysis (FEA) features. The exercises in these tutorial chapters are designed to help you understand and use the power pack features to simplify your work.

Accessing Mechanical Desktop Commands

Mechanical Desktop provides several methods to access commands and manage your design process.

The following are samples of the access methods available to you:

Browser Right-click the window background and choose New Part.

Context Menu In the graphics area, right-click and choose Part ➤ New Part.

Toolbutton New Part

Desktop Menu Part ➤ Part ➤ New Part

Command AMNEW

The step-by-step procedures in the tutorials indicate the command name in the opening procedural text. The appropriate toolbutton is displayed in the margin next to the preferred access method. In the tutorials, the context menu method is used when the menus are sensitive to what you are doing. The Browser method is used when you can save time and steps. You can use any of the alternate methods as well.

If you are in Model mode, you can toggle between the Part and Assembly context menus. If you are in Scene mode, the Scene menu is available. When you are working in Drawing mode, you can toggle between the Drawing and Annotate context menus.

Here is an example of how methods are used in the tutorials:

3 Use AMNEW to create a new part.

Context Menu In the graphics area, right-click and choose Part ➤ New Part.

NOTE To find the location of a particular toolbutton, refer to Appendix A.

Positioning the Desktop Browser

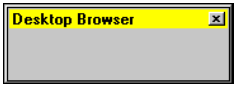
The Desktop Browser is a graphical interface that is useful in both creating and modifying your designs. You can do much of your work in the Browser as you proceed through the lessons in the tutorials.

By default, the Browser is located on the left side of your screen. You may want to move, resize, or hide the Browser to suit your working conditions. This section provides instructions to control the size, shape, and location of the Browser, and to return it quickly to the default location.

The Browser behaves differently when it is in the Auto Hide state. The following are procedures for positioning the Browser both in and out of the Auto Hide state.

To minimize and expand the Desktop Browser

To minimize the Browser double-click the gray area above the tabs.



To expand the Browser, double-click the gray area again.

To minimize the Browser in the Auto Hide state, right-click the gray area and choose Auto Hide ➤ Collapse.

After you minimize the Browser in Auto Hide, you control the expand and collapse function by moving your cursor onto and off of the Browser.

To turn off Auto Hide, right-click the gray area and choose Auto Hide ➤ Off. With Auto Hide off, the Browser remains expanded when you move your cursor away from it.

To move the Browser out of the default position

To move the Browser to another location on the screen, right-click the title bar and choose Move. Click the title bar and drag the Browser to a location on your screen.

To return the Browser to the default position

To return the Browser to the default position, double-click the title bar. The Browser is docked in the default position along the left side of the graphics screen.

To return to the previous location, right-click the gray area and turn off Allow Docking.

To hide and unhide the Browser

To hide the Browser, right-click the gray area above the tabs and choose Hide. To unhide the Browser, choose View ➤ Display ➤ Desktop Browser.

To move the Browser off the screen with Auto Hide, right-click the gray bar above the tabs and choose Auto Hide ➤ Left (or Right).

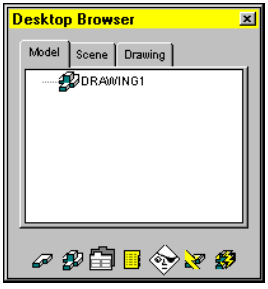
After you move the Browser off the left or right side of the screen with Auto Hide, if you move your mouse to the corresponding edge of the screen, the Browser is displayed along that edge. Move your mouse off the Browser, and the Browser returns to the location off the screen.

To turn off Auto Hide, right-click the gray area and choose Auto Hide ➤ Off. The Browser remains positioned on the screen when you move your cursor away from it.

To move the Browser directly from Auto Hide to another location on your screen, choose Auto Hide ➤ Allow Docking. Click the title bar and drag the Browser to a new location. The Browser is docked in the new location.

To resize the Browser

Right-click the title bar and choose Size. Then drag a corner to resize the Browser.



To return the Browser to its previous size, double-click the title bar.

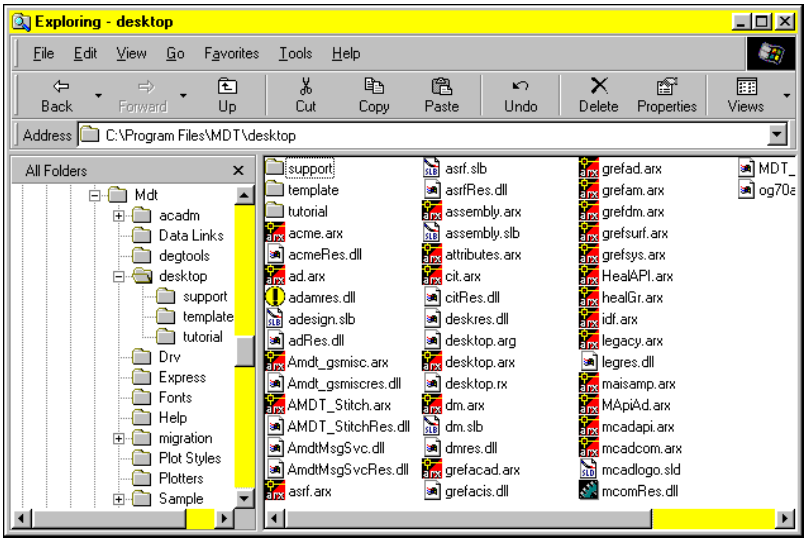
Backing up Tutorial Drawing Files

For each tutorial, you use one or more of the master drawing files that contain the settings, example geometry, or parts for the lesson. These files are included with Mechanical Desktop. Before you begin the tutorials, back up these drawing files so you always have the originals available. Any mistakes you make while you are learning will not affect the master files.

To back up tutorial drawing files

1 From the Windows Start menu, choose Programs ➤ Windows Explorer.

2 In the folder where Mechanical Desktop is installed (by default this is *Program Files\Mdt\desktop*), choose File ➤ New ➤ Folder.



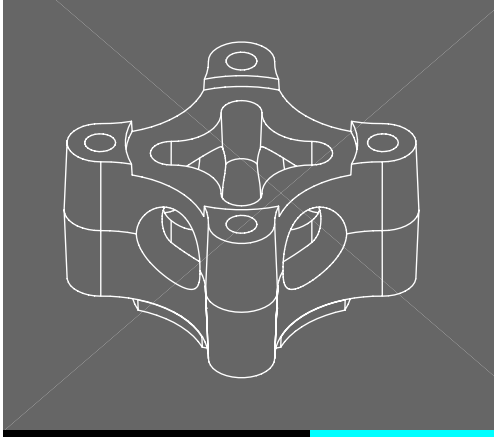
3 Create a new folder called *tutorial backup*.

4 Open the *desktop\tutorial* folder that contains all the tutorial drawing files and copy them into your new folder.

Now you can use the tutorial drawings in the *desktop\tutorial* folder as you work through the tutorials in this book.

NOTE Keep your working tutorial files in the *desktop\tutorial* folder so that external references in the assembly tutorials can update correctly.

Creating Parametric Sketches



In This Chapter

6

- Analyzing a design and creating a strategy for sketching
- Text sketch profiles
- Open profile sketches
- Closed profile sketches
- Path sketches
- Cut line sketches
- Split line sketches
- Break line sketches

Autodesk® Mechanical Desktop® automates your design and revision process using parametric geometry. Parametric geometry controls relationships among design elements and automatically updates models and drawings as they are refined.

The sketch is the basic design element that defines the approximate size and shape of features in your part. As the name implies, a sketch is a loose approximation of the shape that will become a feature. After a sketch is solved, you apply parametric constraints to control its shape.

After you learn to create sketches, move on to chapter 2 to learn how to add constraints to sketches.

Key Terms

Term	Definition
2D constraint	Defines how a sketch can change shape or size. Geometric constraints control the shape and relationships among sketch lines and arcs. Dimensional constraints control the size of sketch geometry.
closed loop	A polyline entity, or group of lines and arcs that form a closed shape. Closed loops are used to create profile sketches.
closed profile	A constrained sketch that is a cross section or boundary of a shape, such as an extrusion, a revolved feature, or a swept feature.
construction geometry	Any line or arc created with a noncontinuous linetype. Using construction geometry in paths and profiles may mean fewer constraints and dimensions are needed to control size and shape of symmetrical or geometrically consistent sketches.
cut line	Used to specify the path of a cross-section drawing view. Unlike a profile sketch, the cut line sketch is not a closed loop. There are two types of cut line sketches—offset and aligned.
feature	An element of a parametric part model. You can create extruded features, revolved features, loft features, and swept features using profiles and paths. You can also create placed features like holes, chamfers, and fillets. You combine features to create complete parametric part models.
nested loop	A closed loop that lies within the boundary of another closed loop. Nested loops are used to create more complex profile sketches.
open profile	A profile created from one or more line segments sketched to form an open shape. Open profiles are used in bend, rib, and thin wall features.
path sketch	A constrained sketch that is a trajectory for a swept feature.
sketch	A planar collection of points, lines, arcs, and polylines used to form a profile, path, split line, break line, or cutting line. An unconstrained sketch contains geometry and occasionally dimensions. A constrained sketch, such as a profile, path, split line, cut line, or break line that contains “real” and construction geometry, and is controlled by dimensions and geometric constraints.
sketch tolerance	Tolerance setting that closes gaps smaller than the pickbox and snaps lines to horizontal, vertical, parallel, or perpendicular.
split line	A sketch, either open or closed, used to split a part into two distinct parts. Also known as a parting line.
text sketch profile	A profile created from a single line of text in a selected font and style. Text-based profiles are used to emboss parts with text.

Basic Concepts of Parametric Sketching

You create, constrain, and edit sketches to define a

- Profile that governs the shape of your part or feature
- Location for a bend feature in a part design
- Path for your profile to follow
- Cut line to define section views
- Split line to split a face or part
- Break line to define breakout section views

After you create a rough sketch with lines, polylines, arcs, circles, and ellipses to represent a feature, you solve the sketch. Solving a sketch creates a parametric profile, path, cut line, split line, or break line from your sketched geometry.

When you solve a sketch, Mechanical Desktop converts it to a parametric sketch by applying two-dimensional constraints to it, according to internal rules. This reduces the number of dimensions and constraints you need to fully constrain it. In general, a sketch should be fully constrained before it is used to create a feature.

You can control the shape and size of the parametric sketch throughout multiple design revisions.

In this tutorial, you learn to create and solve sketches. Chapter 7, "Constraining Sketches," introduces you to creating, modifying, and deleting the constraints and parametric dimensions that control a sketch.

Sketching Tips

Some of these tips do not apply to this chapter, but you will see their usefulness when you use sketches to create complex parts.

Tip	Explanation
Keep sketches simple	It is easier to work with a single object than a multiple-object sketch. Combine simple sketches for complex shapes.
Repeat simple shapes	If a design has repeating elements, sketch one and then copy or array as needed.
Define a sketching layer	Specify a separate layer and color for sketching. Your sketch is visible with other part geometry but easy to identify when you need to modify it.
Preset sketch tolerances	Define characteristics, such as sketch precision and angular tolerance of sketch lines, if the default values are not sufficient.
Draw sketches to size	When your sketches are roughly correct in size and shape, your design is less likely to become distorted as dimensions or constraints are added. Sketch a rectangle to serve as a boundary for the base feature to set relative size. Sketch the feature, but delete the rectangle before you create a profile.
Use PLINE	Whenever possible, use the PLINE command to create your sketches. With PLINE, you can easily draw tangent lines and arcs.

Creating Profile Sketches

In Mechanical Desktop, there are three types of profile sketches:

- Text-based profiles, used to create parametric 3D text-based shapes
- Open profile sketches, used to define features on parts
- Closed profile sketches, used to outline parts and features

You can solve and apply parametric constraints and dimensions to all three of these profile sketch types.

Creating Text Sketch Profiles

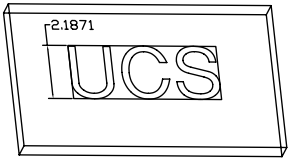
A text sketch profile is a line of text displayed in a rectangular boundary. You extrude a text sketch profile to create the emboss feature on part models.

To create a text sketch profile, you use the command AMTEXTSK. A dialog box opens where you can enter text and choose a font style and size, or you can enter the information on the command line.

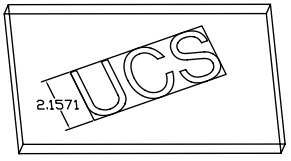
You define an anchor point for the rectangle on your part and a point to define the height of the text. You have the option to define a rotation value on the command line to position the text at an angle. As you move your cursor to define the anchor and height points, the rectangular boundary scales appropriately to accommodate the size of the text.

You can change the size of the text by changing the value of the height dimension. You can apply typical parametric dimensions and constraints between the rectangular boundary and other part edges or features.

When the text sketch profile is sized correctly and in the right position on your part, you extrude it to create the emboss feature.



text sketch



text sketch with rotation defined

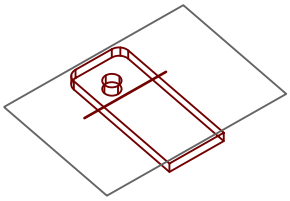
To learn more about using text sketch profiles in the emboss feature, see “Creating Emboss Features” on page 140.

Creating Open Profile Sketches

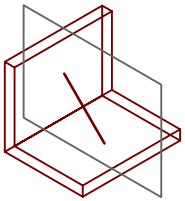
You can create an open profile from single or multiple line segments, and solve it in the same way as you solve a closed profile.

An open profile constructed with one line segment is used to define the location of a bend feature on a flat or cylindrical part model. To bend an entire part, you sketch the open profile over the entire part. If you sketch the open profile over a portion of a part, only that portion of the part bends.

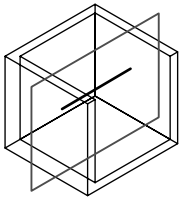
Open profiles constructed with one or multiple line segments are extruded to form rib features and thin features. For a rib feature, the open profile defines the outline of the rib, and is sketched from the side view. For a thin feature, the open profile defines the shape of a wall and is extruded normal to the work plane.



profile for bend feature



profile for rib feature



profile for thin feature

To learn more about open profiles in features, see “Creating Bend Features” on page 163, “Creating Rib Features” on page 133, and “Creating Thin Features” on page 136.

Creating Closed Profile Sketches

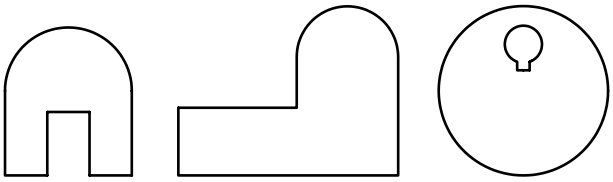
A profile sketch is a two-dimensional outline of a feature. Closed profile sketches are continuous shapes, called loops, that you construct from lines, arcs, and polylines. You use closed profile sketches to create features with custom shapes (unlike standard mechanical features such as holes, chamfers, and fillets).

Profile sketches can be created from a set of objects, or a single polyline, that defines one or more *closed loops*. You can use more than one closed loop to create a profile sketch if the loops are *nested* within each other.

You cannot create profile sketches with loops that are

- Self-intersecting
- Intersecting
- Tangential
- Nested more than one level deep

In this section, you create three profile sketches.

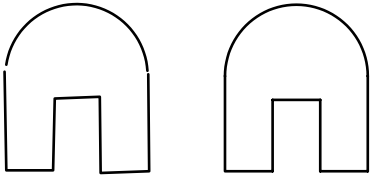


Open the file *sketch1.dwg* in the *desktop\tutorial* folder. This drawing file is blank but it contains the settings you need to create these profiles.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

Using Default Sketch Rules

Mechanical Desktop analyzes individual geometric elements, and operates on a set of assumptions about how they should be oriented and joined.



rough sketch profile sketch

Before you begin, look at the Desktop Browser. It contains an icon with the drawing file name. There are no other icons in the Browser, which indicates that your file contains no parts.

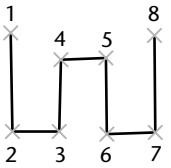
You can move the Browser on your desktop and resize it to give yourself more drawing area. See "Positioning the Desktop Browser" on page 38.

To create a profile sketch from multiple objects

1 Use LINE to draw this shape, entering the points in the order shown.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Line.

Specify first point: *Specify a point (1)*
Specify next point or [Undo]: *Specify a second point (2)*
Specify next point or [Undo]: *Specify a third point (3)*
Specify next point or [Close/Undo]: *Specify a fourth point (4)*
Specify next point or [Close/Undo]: *Specify a fifth point (5)*
Specify next point or [Close/Undo]: *Specify a sixth point (6)*
Specify next point or [Close/Undo]: *Specify a seventh point (7)*
Specify next point or [Close/Undo]: *Specify an eighth point (8)*
Specify next point or [Close/Undo]: *Press* ENTER

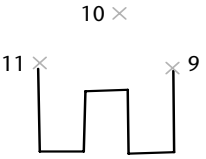


You do not need to make the lines absolutely vertical or horizontal. The objective is to approximate the size and shape of the illustration.

2 Using ARC, sketch the top of the shape, following the prompts on the command line.

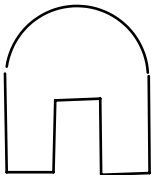
Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Arc.

Specify start point of arc or [CEnter]: *Specify the start point (9)*
Specify second point of arc or [CEnter/ENd]: *Specify the second point (10)*
Specify end point of arc: *Specify the endpoint (11)*



You do not need to use OSNAP to connect the arc to the endpoints of the lines.

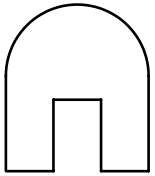
Your sketch should look like this.



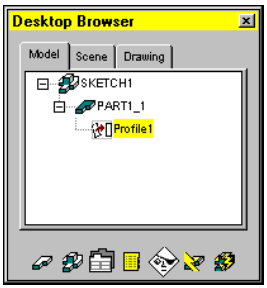
3 Create a profile sketch from the rough sketch, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Profile.

Select objects for sketch: *Select the arc and the lines*
Select objects for sketch: *Press* ENTER



As soon as the sketch is profiled, a part is created. The Browser contains a new icon labelled PART1_1. A profile icon is nested under the part icon.



According to internal sketching rules, Mechanical Desktop determines whether to interpret the sketch geometry as rough or precise and whether to apply constraints.

By default, Mechanical Desktop interprets the sketch as rough and applies constraints, redrawing the sketch. You can customize these default settings with Mechanical Options.

When redrawing, Mechanical Desktop uses assumed constraints in the sketch. For example, lines that are nearly vertical are redrawn as vertical, and lines that are nearly horizontal are redrawn as horizontal.

After the sketch is redrawn, a message on the command line tells you that Mechanical Desktop needs additional information:

Solved under constrained sketch requiring 5 dimensions or constraints.

Depending on how you drew your sketch, the number of dimensions required to fully constrain your sketch may differ from that in this exercise.

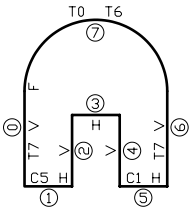
This message tells you that the sketch is not fully defined. When you add the missing dimensions or constraints, you determine how the sketch can change throughout design modifications. Before you add the final constraints, you need to show the assumed constraints.

4 Use AMSHOWCON to show the existing constraints, following the prompt.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

Enter an option [All/Select/Next/eXit] <eXit>: *Enter* **a**

The constraint symbols are displayed.



NOTE The numbers in your sketch might differ, depending on the order in which you created the geometric elements.

The sketch has eight geometric elements, seven lines and an arc, each identified by a number in a circle. Four lines have a V symbol (vertical) and three lines have an H symbol (horizontal). Two of the horizontal lines have constraints denoted by symbols that begin with the letter C (collinear), and three of the elements have constraints denoted by symbols that begin with the letter T (tangent).

If your sketch does not contain the same constraints, redraw it to more closely resemble the illustrations in steps 1 and 2.

Notice the letter F, located at the start point of line 0. It indicates that a fix constraint has been applied to that point. When Mechanical Desktop solves a sketch, it applies a fix constraint to the start point of the first segment of your sketch. This point serves as an anchor for the sketch as you make changes. It remains fixed in space, while other points and geometry move relative to it.

You may delete this constraint if you wish, and apply one or more fix constraints to the endpoints of sketch segments, or to the segments themselves, in order to make your sketch more rigid.

5 To hide the constraints, respond to the prompt as follows:

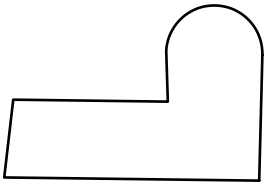
Enter an option [All/Select/Next/eXit] <eXit>: *Press* ENTER

Save your file.

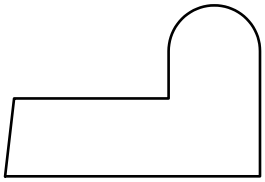
You have successfully created a profile sketch. In chapter 7, "Constraining Sketches," you learn to create, modify, and delete constraints and parametric dimensions.

Using Custom Sketch Rules

Custom settings affect how Mechanical Desktop analyzes rough sketches. In this exercise, you sketch with PLINE and convert your drawing to a profile sketch. You will modify one of the Mechanical Options sketch rule settings and see its effect on the sketch.



rough sketch



profile sketch

Before you begin the next exercise, create a new part definition.

To create a new part definition

1 Use the context menu to initiate a new part definition.

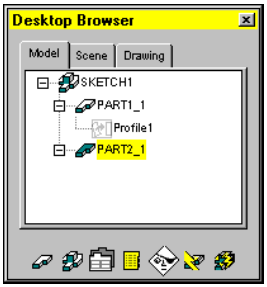
Context Menu In the graphics area, right-click and choose Part ➤ New Part.

2 Respond to the prompts as follows:

Select an object or enter new part name <PART2>: *Press* ENTER

NOTE The command method you use determines which prompts appear.

A new part definition is created in the drawing and displayed in the Browser. The new part automatically becomes the active part.



3 Pan the drawing so you have room to create the next sketch.

Context Menu In the graphics area, right-click and choose Pan.

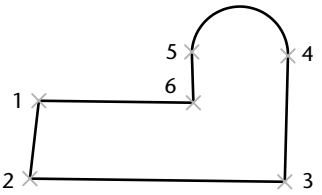
You are ready for the next exercise.

To create a profile sketch from a single polyline

1 Use PLINE to draw this rough sketch as a continuous shape, following the prompts for the first four points.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.

Specify start point: *Specify a point (1)*
Current line-width is 0.0000
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
Specify a second point (2)
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
Specify a third point (3)
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
Specify a fourth point (4)



2 Following the prompts, switch to Arc to create the arc segment, then switch back to Line. Switch to Close to finish the sketch.

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: *Enter* **a**
Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:
Specify a fifth point (5)
Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:
Enter **l**
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
Specify a sixth point (6)
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: *Enter* **c**

3 Use AMPROFILE to create a profile sketch from the rough sketch.

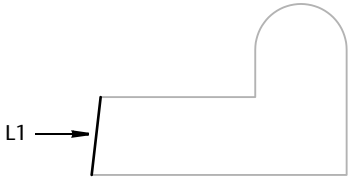
Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

NOTE If you used line segments and an arc to draw your sketch you cannot use Single Profile. This command profiles single object sketches only. For sketches containing more than one object, use Profile.

When you use Single Profile, you are not prompted to select the sketch geometry. Mechanical Desktop looks for the last entity you created. If it is a valid closed loop, Mechanical Desktop analyzes the sketch, redraws it, and displays the following message:

Solved under constrained sketch requiring 5 dimensions or constraints.

All lines were redrawn as horizontal or vertical except one. L1 remains angled because the angle of the line exceeds the setting for angular tolerance. By default, this rule makes a line horizontal or vertical if the angle is within 4 degrees of horizontal or vertical.

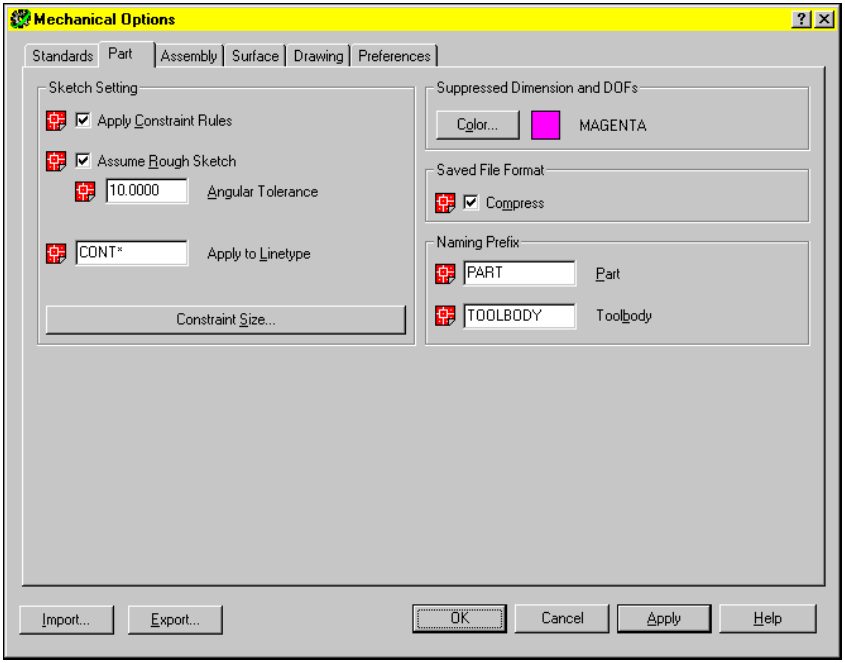


You can modify this and other sketch tolerance settings to adjust the precision of your sketch analysis.

4 Change the angular tolerance setting.

Browser Click the Options button below the window.

5 In the Mechanical Options dialog box, choose the Part tab and change the angular tolerance from 4 degrees to **10** degrees, the maximum value.



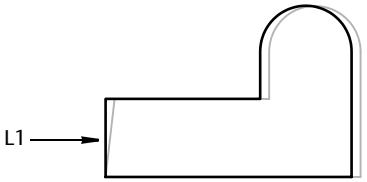
Choose OK.

6 Reprofile the sketch, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Profile.

NOTE You cannot use Single Profile to reprofile a sketch.

Select objects for sketch: *Use a crossing window to specify the sketch*
Select objects for sketch: *Press* ENTER



If your sketch shows line L1 unchanged, the angle was greater than 10 degrees. You need to edit or redraw the shape and append the sketch.

NOTE When adding geometry or changing a sketch, you must append the new geometry so that the sketch is reanalyzed and constraints are reapplied. See chapter 7, "Constraining Sketches," to append geometry to a sketch.

When L1 was made vertical, it required one less dimension or constraint to fully solve the sketch. The following message is displayed on the command line.

Solved underconstrained sketch requiring 4 dimensions or constraints.

Save your file.

You can adjust sketch rules that determine how precisely you need to draw. For most sketching, you should use the default settings. However, you can change the default settings as needed.

Using Nested Loops

You can select more than one closed loop to create a profile sketch. A closed loop must encompass the nested loops. They cannot overlap, intersect, or touch. With nested loops you can easily create complex profile sketches.

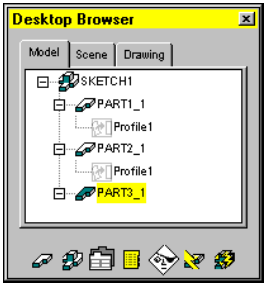
To create a profile sketch using nested loops

1 Use AMNEW to create a new part definition.

Context Menu In the graphics area, right-click and choose Part ➤ New Part.

2 Accept the default part name on the command line.

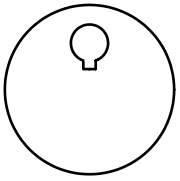
The Browser now contains a third part.



3 Pan the drawing so you have room to create the next sketch.

Context Menu In the graphics area, right-click and choose Pan.

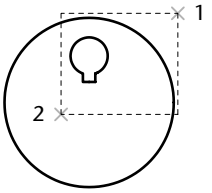
4 Create the following sketch using lines or polylines, and circles. Then, in the graphics area, right-click and choose 2D Sketching ➤ Trim and follow the prompts on the command line to remove the section from the smaller circle.



5 Profile the sketch, following the prompts to select the objects with a crossing window.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Profile.

Select objects for sketch: *Specify a point to the right of the sketch (1)*
Specify opposite corner: *Specify a second point (2)*
5 found
Select objects for sketch: *Press* ENTER



Mechanical Desktop calculates the number of dimensions or constraints required to fully constrain the profile.

Solved underconstrained sketch requiring 7 dimensions or constraints.

NOTE You may need more dimensions or constraints, depending on how you created your sketch.

Save your file.

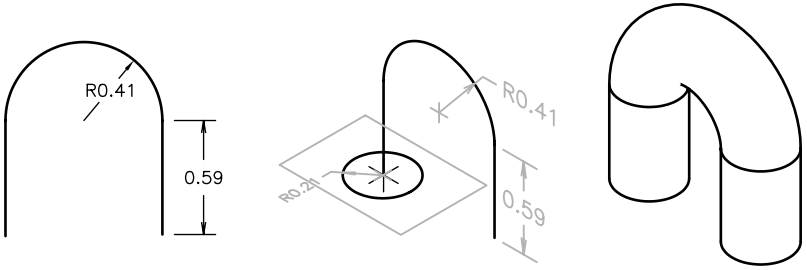
This simple cam illustrates how you can easily create complex shapes to define parts and features. Experiment on your own to create profiles from nested loops.

Creating Path Sketches

Path sketches can be both two dimensional and three dimensional. Like open profile sketches, they can be open shapes. In this exercise, you create only the path sketches, but not the profiles that would sweep along the paths.

Creating 2D Path Sketches

A 2D path sketch serves as a trajectory for a *swept feature*. You create a swept feature by defining a path and then a profile sketch of a cross section. Then, you sweep the profile along the path.



path sketch profile sketch swept feature

The geometry for the 2D path must be created on the same plane.

Valid geometry that can be used to create a 2D path includes

- Lines
- Arcs
- Polylines
- Ellipse segments
- 2D splines

When you solve a 2D path sketch, you can automatically create a work plane normal to the start point of the path. You use this work plane to create a profile sketch for the swept feature, and then constrain the profile sketch to the start point of the path.

To create a 2D path sketch

1 Create a new part definition.

Context Menu In the graphics area, right-click and choose Part ➤ New Part.

2 Press ENTER on the command line to accept the default part name.

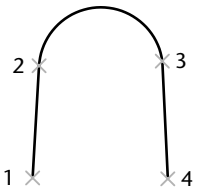
3 Pan the drawing so you have room to create the next sketch.

Context Menu In the graphics area, right-click and choose Pan.

4 Use PLINE to draw the rough sketch as a continuous shape, responding to the prompts to specify the points in the following illustration.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.

Specify start point: *Specify a point (1)*
Current line-width is 0.0000
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
Specify a second point (2)
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
Enter **a** *to create an arc segment*
Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:
Specify a third point (3)
Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:
Enter **l** *to create a line segment*
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
Specify a fourth point (4)
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: *Press* ENTER



Make sure to switch between drawing lines and arcs at points (2) and (3).

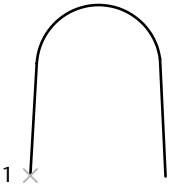
5 Use AM2DPATH to convert the rough sketch to a path sketch, following the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ 2D Path.

Select objects: *Specify the polyline shape*
Select objects: *Press* ENTER

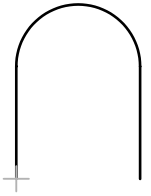
At the prompt for the start point of the path, you select the point where the path begins. This determines the direction to sweep the profile of the cross section.

Select start point of the path: *Specify the start point (1)*



You can also specify whether a work plane is created perpendicular to the path. In this example, a work plane is not required.

Create a profile plane perpendicular to the path? [Yes/No] <Yes>: *Enter* **n**



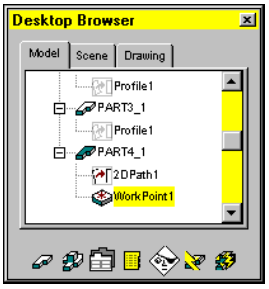
NOTE If you choose to create a sketch to sweep along the path, Mechanical Desktop can automatically place a work plane perpendicular to the path.

Press the F2 function key to activate the AutoCAD Text window. Examine the prompts for the AM2DPATH command. The following line is displayed:

Solved underconstrained sketch requiring 3 dimensions or constraints.

The sketch analysis rules indicate that the path sketch needs three more dimensions or constraints to fully define the sketch.

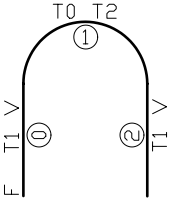
A work point is automatically placed at the start point of the path. The Browser displays both a 2DPath icon and a work point icon nested below the part definition.



6 Use AMSHOWCON to display the existing constraints, responding to the prompt.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

Enter an option [All/Select/Next/eXit] <eXit>: *Enter* **a**



The start point of the path is fixed. Both lines are vertical and are tangent to the endpoints of the arc. The missing information is the length of each line and the radius of the arc. Given these values, the sketch would be fully constrained.

Enter an option [All/Select/Next/eXit] <eXit>: *Press* ENTER

Save your file.

Next, you create a three-dimensional path.

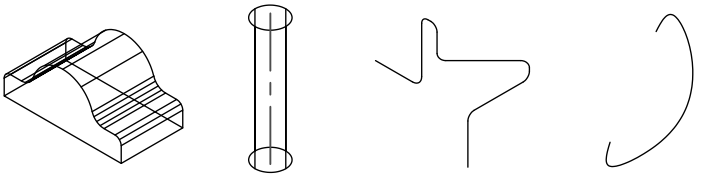
Creating 3D Path Sketches

3D path sketches are used to create

- A 3D path from existing part edges
- A helical path
- The centerline of a 3D pipe
- A 3D spline path

3D paths are used to create swept features that are not limited to one plane. See chapter 8, "Creating Sketched Features," to learn more about sweeping features along a 3D path.

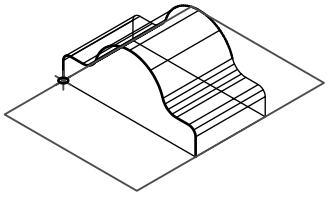
Open the file *sketch2.dwg* in the *desktop\tutorial* folder. The drawing contains four part definitions and the geometry you need to create the 3D paths.



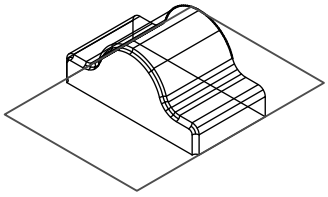
NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

Creating a 3D Edge Path

A 3D edge path is used to create a path from existing part edges. After you create the path, you can sweep a profile and use a Boolean operation to combine the feature with the existing part.



3D edge path and profile sketch



3D sweep along edge path

Before you can work on a part, it must be active. Activate PART1_1, responding to the prompts.

Context Menu In the graphics area, right-click and choose Part ➤ Activate Part.

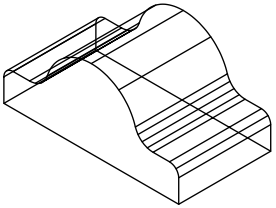
Select part to activate or [?] <PART1_1>: *Enter* **PART1_1**

PART1_1 is activated, and highlighted in the Browser.

Use Pan to center PART1_1 on your screen.

Context Menu In the graphics area, right-click and choose Pan.

PART1_1 contains an extruded part.



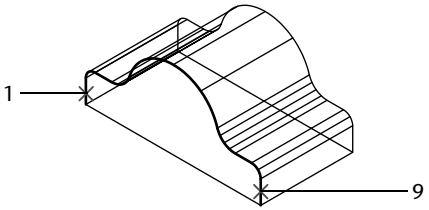
To create a 3D edge path

1 Use AM3DPATH to define the 3D edge path, following the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ 3D Edge Path.

Select model edges (to add): *Specify the first part edge (1)*
Select model edges (to add): *Specify the next edges in a clockwise sequence*
Select model edges (to add): *Specify the last edge (9)*
Select model edges (to add): *Press* ENTER
Specify start point: *Specify start point (1)*
Create workplane? [Yes/No] <Yes>: *Press* ENTER

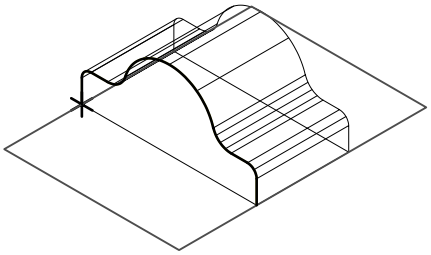
The command method you use determines the prompts that are displayed.



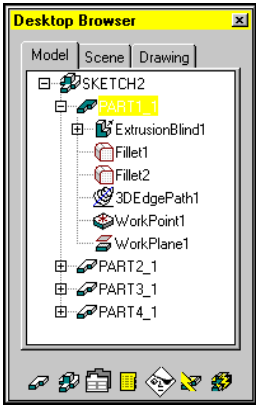
2 Continue on the command line to place the work plane.

Plane=Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <accept>: *Press* ENTER

The path is created, and a work point is located at the start point. A work plane is placed normal to the start of the path so you can sketch the profile for the sweep feature.



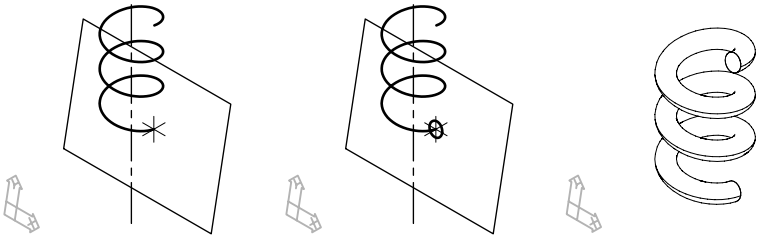
In the Browser, the new geometry is nested below the extrusion and fillets in PART1_1.



Save your file.

Creating a 3D Helical Path

A 3D helical path is used for a special type of swept feature. Helical sweeps are used to create threads, springs, and coils. You create a 3D helical path from an existing work axis, cylindrical face, or cylindrical edge.



When you create a 3D helical path, you can specify whether a work plane is also created. The work plane can be normal to the path, at the center of the path, or along the work axis. You use this work plane to draw the profile sketch for the helical sweep.

Before you begin, activate PART2_1, responding to the prompts.

Context Menu In the graphics area, right-click and choose Part ➤ Activate Part.

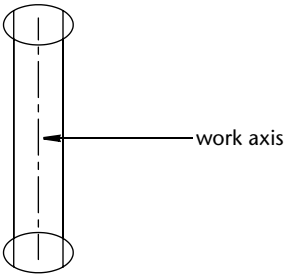
Select part to activate or [?] <PART1_1>: *Enter* **PART2_1**

PART2_1 is highlighted in the Browser and on your screen.

Use Pan to center PART2_1 on your screen.

Context Menu In the graphics area, right-click and choose Pan.

PART2_1 contains a cylinder and a work axis.



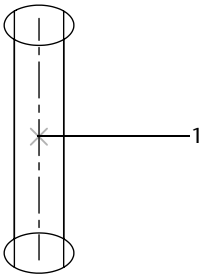
To create a 3D helical path

3 Use AM3DPATH to define the 3D helical path, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ 3D Helix Path.

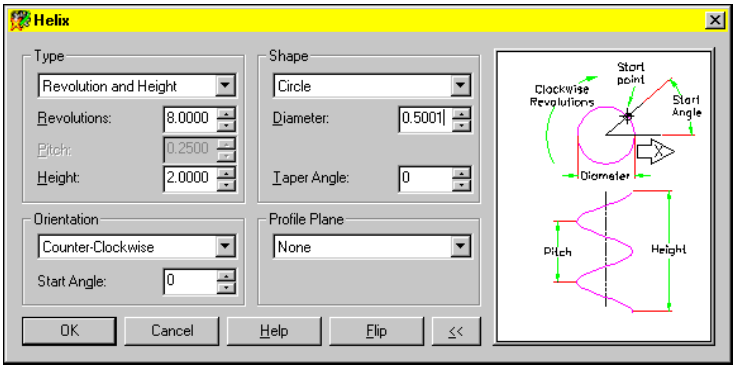
Enter path type [Helical/Spline/Edge/Pipe] <Edge>: *Enter* **h**
Select work axis, circular edge, or circular face for helical center:
Select the work axis (1)

The command method you use determines the prompts that are displayed.



4 In the Helix dialog box, specify the following:

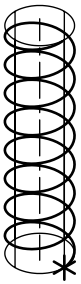
Type: Revolution and Height
Revolutions: *Enter* **8**
Height: *Enter* **2**
Diameter: *Enter* **.5**
Orientation: Counter-Clockwise

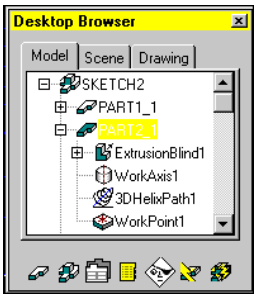


Choose OK.

NOTE The path is automatically constrained with the parameters defined in the Helix dialog box. You can edit the path at any time with AMEDITFEAT.

The 3D helix path is created. A work point is placed at the beginning of the path.



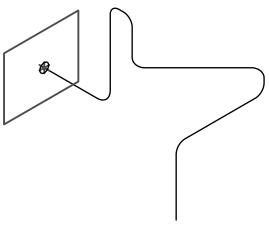


You can also specify that a work plane is placed normal to the start point of the 3D path, at the center of the path, or along the work axis. This option makes it easier for you to create the sketch geometry for the profile you sweep along the path.

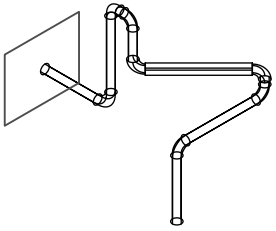
Save your file.

Creating a 3D Pipe Path

A 3D pipe path is used to sweep a feature along a three-dimensional path containing line and arc segments or filleted polylines. You can modify each of the control points and the angle of the segments in the 3D Pipe Path dialog box.



3D pipe path and profile sketch



3D sweep along pipe path

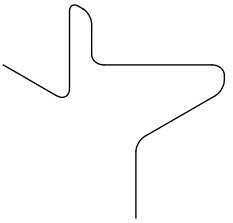
Before you begin, activate PART3_1. This time use the Browser method to activate the part.

Browser In the graphics area, double-click PART3_1.

PART3_1 is activated, and highlighted in the Browser.

Use Pan to center PART3_1 on your screen.

PART3_1 contains an unsolved sketch of line segments and arcs.



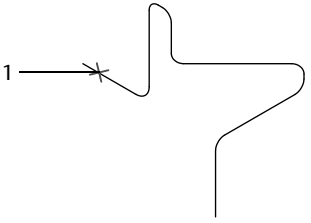
To create a 3D pipe path

1 Use AM3DPATH to define the 3D pipe path, responding to the prompts.

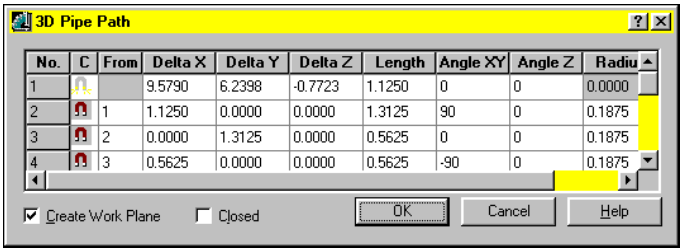
Context Menu In the graphics area, right-click and choose Sketch Solving ➤ 3D Pipe Path.

Select polyline path source: *Select the first line (1)*
Select polyline path source: *Select the remaining arcs and lines in sequence*
Select polyline path source: *Press* ENTER
Specify start point: *Specify a point near the start of the first line (1)*

The command method you use determines the prompts that are displayed.



2 In the 3D Pipe Path dialog box, examine the vertices and angles of the path. Verify that Create Work Plane is selected.



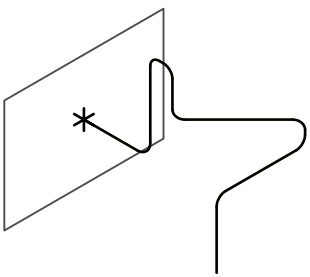
NOTE Your numbers might not match the illustration above.

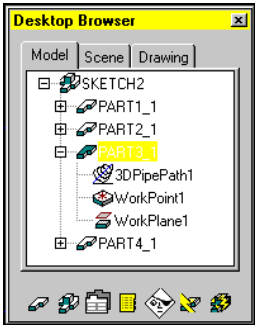
Choose OK to exit the dialog box.

3 Place the work plane, following the prompts.

Plane=Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <accept>: *Press* ENTER

The Desktop Browser now contains a 3D Pipe icon, a work plane, and a work point nested below the PART3_1 definition.



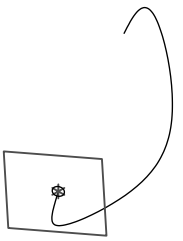


Save your file.

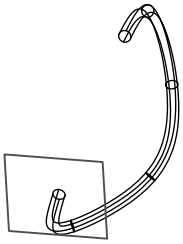
Creating a 3D Spline Path

In this type of path, you sweep a feature along a 3D spline created with fit points or control points. Working in one integrated dialog box, you can modify any fit point or control point in a 3D spline path, and you can convert fit points to control points, and control points to fit points.

In this exercise, you work with a fit point spline.



3D spline path and profile sketch



3D sweep along spline path

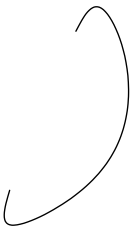
Before you begin, activate PART4_1 from the Browser.

Browser In the graphics area, double-click PART4_1.

PART4_1 is highlighted in the Browser and on your screen.

Use Pan to center PART4_1 on your screen.

PART4_1 contains an unsolved spline sketch.



To create a 3D spline path

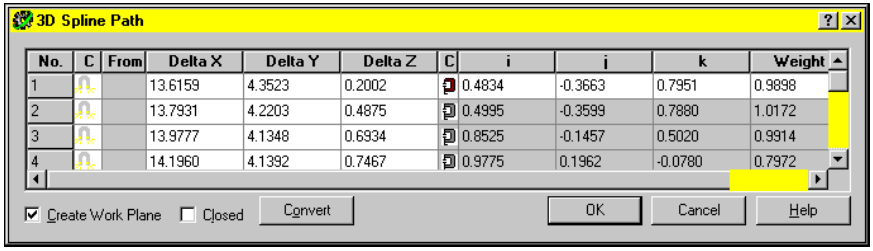
1 Use AM3DPATH to define the 3D spline path, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ 3D Spline Path.

Select 3D spline path source: *Specify the spline*
Specify start point: *Specify the start point*

The command method you use determines the prompts that are displayed.

2 In the 3D Spline Path dialog box, examine the vertices of the spline, and verify that Create Work Plane is selected.



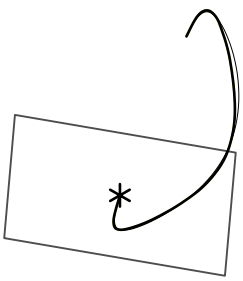
NOTE Your numbers might not match the illustration above.

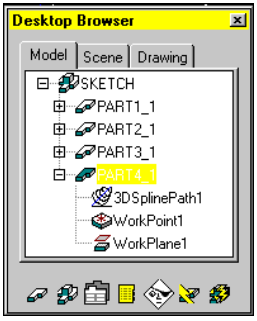
Choose OK to exit the dialog box.

3 Create the work plane, responding to the prompts.

Plane=Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <accept>: *Press* ENTER

The path is created, and a work point is located at the start point. A work plane is placed normal to the start of the path so you can begin to sketch the profile for the sweep feature.





Save your file.

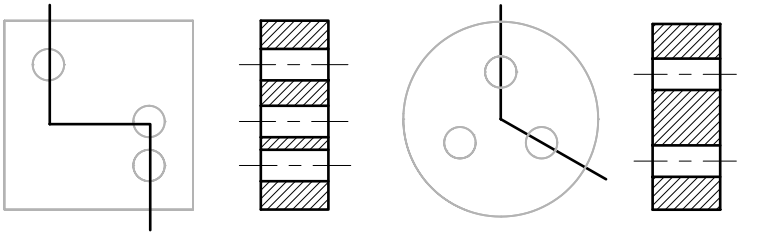
Creating a path sketch is similar to creating a profile sketch. The difference between the two sketch types is their purpose.

- Profile sketches provide a general way to create a variety of features.
- Path sketches are used exclusively for creating trajectory paths for 2D and 3D swept features.

Creating Cut Line Sketches

When you create drawing views, you might want to depict a cut path across a part for offset, cross-section views. After you have extruded or revolved a profile sketch to create a feature, you can return to an original sketch and draw the cut line across the features you want to include in the cross section.

There are two types of cut line sketches: offset and aligned. An offset cut line sketch is a two-dimensional line constructed from orthogonal segments. An aligned cut line sketch is a two-dimensional line constructed from non-orthogonal segments.



Two general rules govern cut line sketches:

- Only line and polyline segments are allowed.
- The start and end points of the cut line must be outside the part.

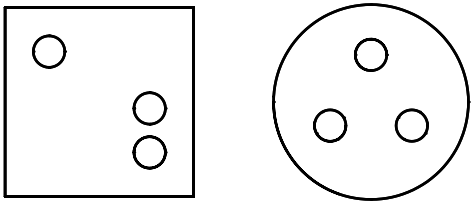
These additional rules apply to cut line sketches:

- The first and last line segments of an offset cut line must be parallel.
- Offset cut line segments can change direction in 90-degree increments only.
- Only two line segments are allowed in an aligned cut line.
- Line segments of aligned cut lines can change direction at any angle.

In the following exercise, after you create a cut line sketch on these models, the resulting cross-section drawing views can be generated in Drawing mode.

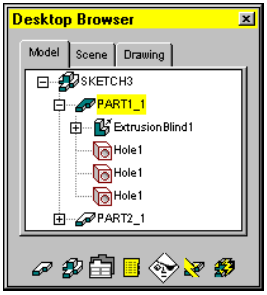
A cut line sketch is needed when you want to define a custom cross-section view only, but not for a half or full cross-section view.

Open the file *sketch3.dwg* in the *desktop\tutorial* folder. The drawing contains two parts.



NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

Before you begin, click the plus signs in front of SKETCH3 and PART1_1 to expand the Browser hierarchy.



To create an offset cut line sketch

1 Use PLINE to sketch through the center of the holes on the square part.

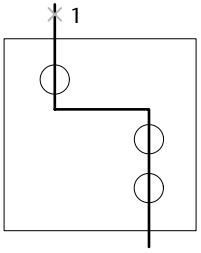
Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.

Next, analyze the cut line sketch according to internal sketching rules.

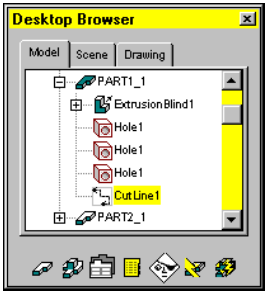
2 Use AMCUTLINE to solve the cut line, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Cut Line.

Select objects to define the section cutting line: *Select the polyline (1)*
Select objects to define the section cutting line: *Press* ENTER



A new icon called CutLine1 is added to the PART1_1 hierarchy in the Browser.



Save your file.

As with the other sketches you created, a message tells you how many dimensions and constraints are needed to fully solve the sketch. In this case, you need five dimensions or constraints to complete the definition of the sketch: three to define the shape of the sketch, and two to constrain it to the part.

When you create a cross-section drawing view, this sketch defines the path of the cut plane. If you change the size of the part or holes, or their placement, the cut line is updated to reflect the new values.

For the next exercise, you use the circular part. In the Browser, click the minus sign in from of PART1_1 to collapse the part hierarchy. Then click the plus sign in front of PART2_1 to expand the circular part hierarchy.

Before you begin, you need to activate the circular part.

Browser Double-click PART2_1.

PART2_1 is activated, and highlighted in the Browser and on your screen.

NOTE Before you can work on a part, it must be active.

To create an aligned cut line sketch

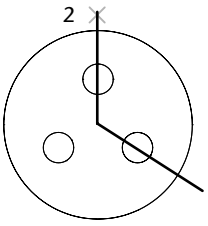
1 Use PLINE to sketch through the centers of two of the holes on the circular part.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.

2 Define a cut line on your sketch, responding to the prompts.

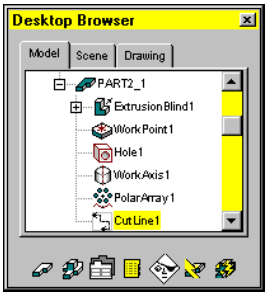
Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Cut Line.

Select objects to define the section cutting line: *Select the polyline (2)*
Select objects to define the section cutting line: *Press* ENTER



A message states that you need five dimensions or constraints to fully solve this sketch.

3 In the Browser, the new CutLine1 icon is part of the PART2_1 hierarchy.



Save your file.

Creating Split Line Sketches

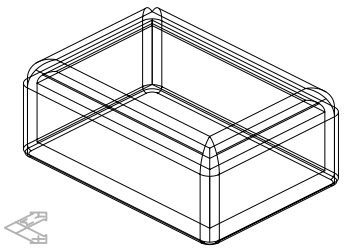
A molded part or casting usually requires two or more shapes to define the part. To make a mold or a cast, you create the shape of your part and then apply a split line to split the part into two or more pieces. You may also need to apply a small draft angle to the faces of your part so that your part can be easily removed from the mold.

Split lines can be as simple as a planar intersection with your part, or as complex as a 3D polyline, or spline, along planar or curved faces.

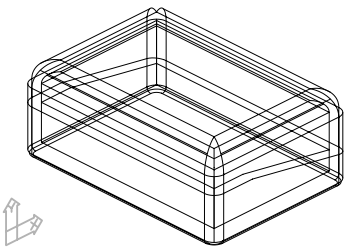
You can also split parts using either

- A selected planar face or a work plane
- A sketch projected onto a selected set of faces

In this exercise, you create a split line to split a shelled part into two separate parts.



shelled part



split part

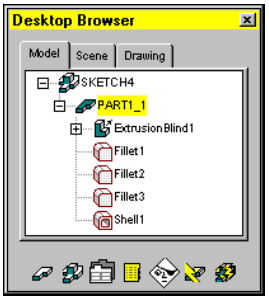
Open the file *sketch4.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing file contains a simple shelled box. Two viewports have been defined: the right side of the part, and an isometric view. You'll define a new sketch plane in the right viewport and sketch a split line in the left viewport.

To create a split line

1 Expand the Browser hierarchy of SKETCH4 and PART1_1.

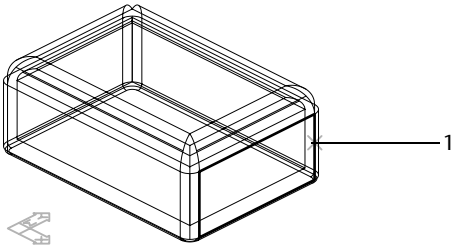


The part consists of an extrusion, three fillets, and a shell feature. Next, you create a sketch plane on the outside right face of the part.

2 In the right viewport, define a new sketch plane, responding to the prompts.

Context Menu In the graphics area, right-click and choose New Sketch Plane.

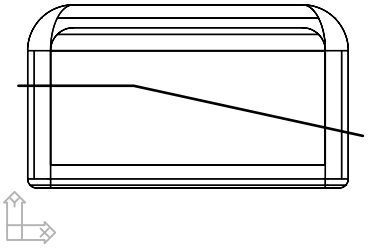
Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Specify the outside right face of the part (1)
Enter an option [Accept/Next] <Accept>: *Press* ENTER
Plane = Parametric
Select edge to align X axis [Flip/Rotate/Origin] <accept>: *Press* ENTER



Next, create a sketch and convert it to a split line.

3 In the left viewport, use PLINE to sketch the split line.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.



4 Use AMSPLITLINE to create a split line from your sketch, responding to the prompts.

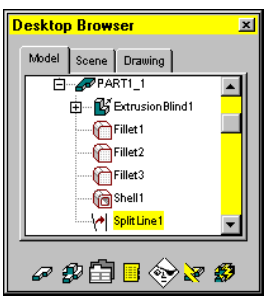
Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Split Line.

Select objects for sketch: *Select the polyline*
Select objects for sketch: 1 found
Select objects for sketch: *Press* ENTER
Select edge to include in split line or press <ENTER> to accept: *Press* ENTER

Mechanical Desktop solves the sketch and displays the number of constraints required to fully constrain it.

Solved underconstrained sketch requiring 5 dimensions or constraints.

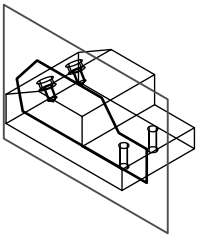
5 Look at the Browser. SplitLine1 is now nested under the part definition.



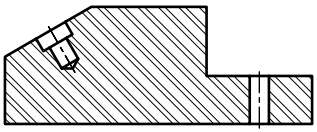
Save your file.

Creating Break Line Sketches

When you want to document complex assemblies, it is not always easy to display parts and subassemblies that are hidden by other parts in your drawing views. By creating a break line sketch, you can specify what part of your model will be cut away in a breakout drawing view so that you can illustrate the parts behind it.



break line path

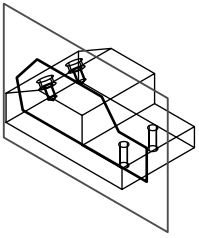


breakout drawing view

Open the file *sketch4a.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing file contains a simple part. An unsolved sketch lies on a work plane. You create a break line from this sketch.

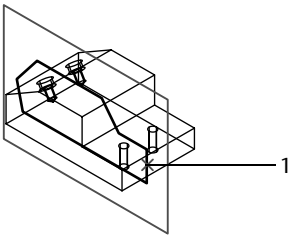


To create a break line

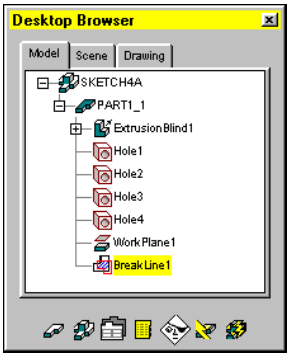
1 Use AMBREAKLINE to define the break line sketch, following the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Break Line.

Select objects for sketch: *Specify the sketch (1)*
Select objects for sketch: *Press* ENTER



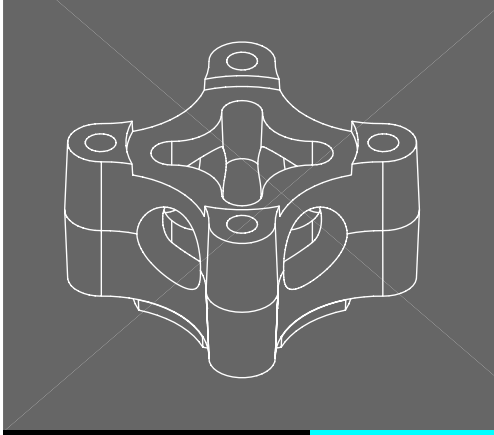
The break line is created. The Browser contains a break line icon nested below the work plane.



Save your file.

Now that you have learned the basics of creating sketches, you are ready to constrain them by adding geometric and parametric dimension constraints.

Constraining Sketches



In This Chapter

7

- Creating a strategy for constraining and dimensioning
- Defining sketch shape and size with dimensions and geometric constraints
- Using construction lines, arcs, and circles to create and control sketches
- Modifying a design
- Re-creating a constrained sketch

When you solve a sketch in Autodesk® Mechanical Desktop®, geometric constraints are applied in accordance with internal rules. To fully constrain the sketch, you apply the remaining parametric dimensions and geometric constraints that are necessary to meet your design goals.

Any time you modify a sketch, the parametric geometry retains the relationships among design elements.

To reduce the number of constraints required to fully constrain a sketch, you can use construction geometry. Construction geometry becomes part of the sketch, but is ignored when the sketch is used to create a feature.

In the next chapter, you learn to add sketched features to your constrained sketches.

Key Terms

Term	Definition
2D constraint	Defines how a sketch can change shape or size. Geometric constraints control the shape and relationships among sketch lines and arcs. Dimensional constraints control the size of sketch geometry.
degree of freedom	In part modeling, determines how a geometric object such as a line, arc, or circle can change shape or size. For example, a circle has two degrees of freedom, center and radius. When these values are known, degrees of freedom are said to be eliminated.
dimensional constraint	Parametric dimension that controls the size of a sketch. When changed, the sketch resizes. May be expressed as a constant value, a variable in an equation, a variable in a table, or in global parameter files.
geometric constraint	Controls the shape and relationships among geometric elements in a sketch.
parametrics	A solution method that uses the values of part parameters to determine the geometric configuration of the part.

Basic Concepts of Creating Constraints

A sketch needs geometric and dimensional constraints to define its shape and size. These constraints reduce the degrees of freedom among the elements of a sketch and control every aspect of its final shape.

When you solve a sketch, Mechanical Desktop applies some geometric constraints. In general, use the automatically applied constraints to stabilize the sketch shape.

Depending upon how accurately you sketch, you may need to add one or more constraints to fully solve a sketch. You can also add construction geometry to your sketch to reduce the number of additional constraints required. After you add further constraints, you might need to delete some of the applied constraints.

In most cases, you need to fully constrain sketches before you use them to create the features that define a part. As you gain experience, you will be able to determine which constraints control the sketch shape according to your design requirements.

Constraining Tips

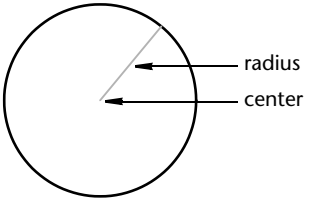
Tip	Explanation
Determine sketch dependencies	Analyze the design to determine how sketch elements interrelate; then decide which geometric constraints are needed.
Analyze automatically applied constraints	Determine the degrees of freedom not resolved by automatic constraints. Decide if any automatic constraints need to be deleted in order to constrain elements as you require.
Use only needed constraints	Replace constraints as needed to define shape. Because constraints often solve more than one degree of freedom, use fewer constraints than degrees of freedom.
Stabilize shape before size	If you apply geometric constraints before dimensions, your sketch shape is less likely to become distorted.
Dimension large before small	To minimize distortion, define larger elements that have an overall bearing on the sketch size. Dimensioning small elements first may restrict overall size. Delete or undo a dimension if the sketch shape is distorted.
Use both geometric constraints and dimensions	Some constraint combinations may distort unconstrained portions of the sketch. If so, delete the last constraint and consider using a dimension or a different constraint combination.

Constraining Sketches

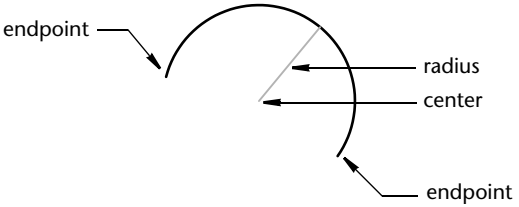
Constraining a sketch defines how a sketch can change shape or size. In addition to the inferences by the software, you often need additional dimensions or constraints.

Constraints may be fixed or variable, but they always prevent unwanted changes to a feature as you make modifications.

The ways a sketch can change size or shape are called *degrees of freedom*. For example, a circle has two degrees of freedom—the location of its center and its radius. If the center and radius are defined, the circle is fully constrained and those values can be maintained.



Similarly, an arc has four degrees of freedom—center, radius, and the endpoints of the arc segment.



The degrees of freedom you define correspond to how fully the sketch is constrained. If you define all degrees of freedom, the arc is *fully constrained*. If you do not define all degrees of freedom, the sketch is *underconstrained*.

Mechanical Desktop does not allow you to define a degree of freedom in more than one way and thus prevents you from *overconstraining* a sketch.

Before you add constraints, study your sketch, and then decide how to constrain it. Usually, you need both geometric constraints and dimensions. See "Constraining Tips" on page 86.

You should fully constrain sketches so that they update predictably as you make changes. As you gain experience, you may want to underconstrain a sketch while you work out fine points of a design, but doing so may allow that feature to become distorted as you modify dimensions or constraints.

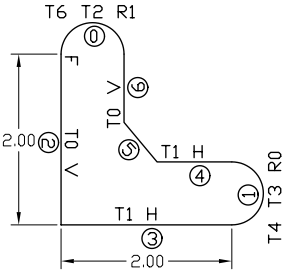
Applying Geometric Constraints

When constraining a sketch, begin by defining its overall shape before defining its size. Geometric constraints specify the orientation and relationship of the geometric elements. For example

- Constraints that specify orientation indicate whether an element is horizontal or vertical.
- Constraints that determine relationships specify whether two elements are perpendicular, parallel, tangent, collinear, concentric, projected, joined, have the same *X* or *Y* coordinate location, or have the same radius.

Mechanical Desktop displays geometric constraints as letter symbols. If the constraint specifies a relationship between two elements, the letter symbol is followed by the number of the sketch element to which the constraint is related. In the example below,

- The start point of the arc (0) has a fix constraint. This point is anchored and will not move when changes are made to the sketch constraints.
- The lines (2, 3, 4, and 6) have constraint symbols of either H (horizontal) or V (vertical).
- All lines except one are tangent to at least one of the arcs (0 and 1). Each symbol T (tangent) is followed by the number of the arc to which it is tangent.
- Each arc is tangent to its connecting lines, as shown by T constraint symbols, and the arcs have the same radius, as indicated by the R constraint symbols.

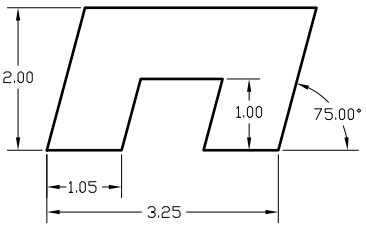


As you apply geometric constraints, you should continue to analyze your sketch, reviewing and replacing constraints.

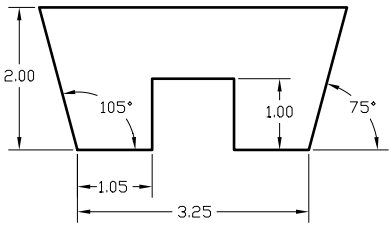
In the next exercise, you gain experience with constraining techniques by analyzing and then modifying geometric constraints to reshape the sketch.

Open the file *sketch5.dwg* in the *desktop\tutorial* folder. Use the before-and-after sketches below to determine what changes you must make. Then change the constraints and see the results of your analysis.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.



before geometric constraints



after geometric constraints

In the before-and-after sketches, you can see that the constraints and dimensions differ, but you cannot discern which geometric constraints Mechanical Desktop has assumed. You will notice that

- The linear dimensions are the same for both sketches.
- The angular relationships of the vertical lines differ.

Showing Constraint Symbols

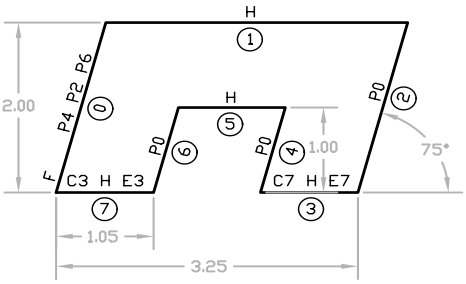
You can change the parametric relationships of the lines by modifying geometric or dimensional constraints. Because geometric constraints control the overall shape of the sketch, you cannot safely make any changes until you know the current geometric constraints. Therefore, the next step is to show the symbols.

To show constraint symbols

1 Use AMSHOWCON to display constraint symbols, responding to the prompt.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

Enter an option [All/Select/Next/eXit] <eXit>: *Enter* **a**

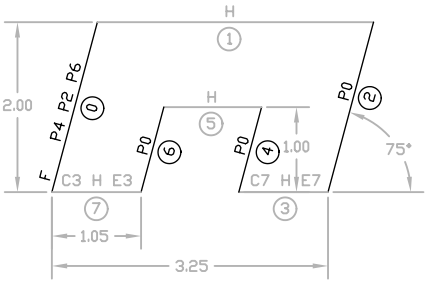


Parallel constraints exist between lines 0, 2, 4 and 6. Lines 1, 3, 5, and 7 have horizontal constraints. Lines 3 and 7 are also collinear and equal in length.

You begin reshaping your sketch by removing the parallel constraints.

To understand the constraints, look at symbol P0 (on line 2). This symbol indicates that line 2 is parallel to line 0.

Similarly, the constraint symbols (P2, P4, and P6) show that line 0 is parallel to lines 2, 4 and 6.



2 Hide the constraint symbols.

Enter an option [All/Select/Next/eXit] <eXit>: *Press* ENTER

Replacing Constraints

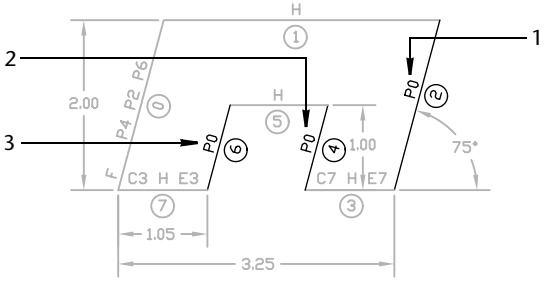
After you delete the unwanted constraints, you can add constraints to reshape the sketch. In this exercise, you delete the parallel constraints that control the inner and outer angled lines in the sketch and replace them with vertical constraints.

To replace a constraint

1 Use AMDELCON to replace the constraints, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Delete Constraints.

Select or [Size/All]: *Select the parallel constraint symbols (1), (2), and (3)*
Select or [Size/All]: *Press* ENTER

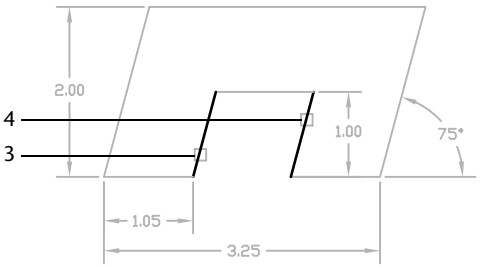


The parallel constraints are deleted. The sketch shape looks the same until you add constraints or change dimensions.

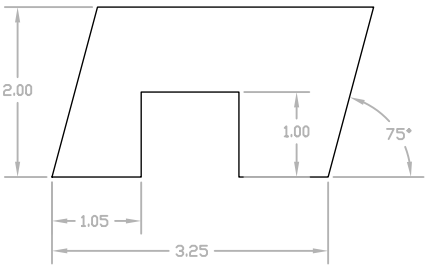
2 Use AMADDCON to add vertical constraints to the two inner angled lines, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Vertical.

Valid selection(s): line, ellipse or spline segment
Select object to be reoriented: *Specify line (3)*
Solved under constrained sketch requiring 2 dimensions or constraints.
Valid selection(s): line, ellipse or spline segment
Select object to be reoriented: *Specify line (4)*
Solved under constrained sketch requiring 1 dimensions or constraints.
Valid selection(s): line, ellipse or spline segment
Select object to be reoriented: *Press* ENTER
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix] <eXit>: *Press* ENTER



The vertical constraints are applied, and your sketch should look like this.



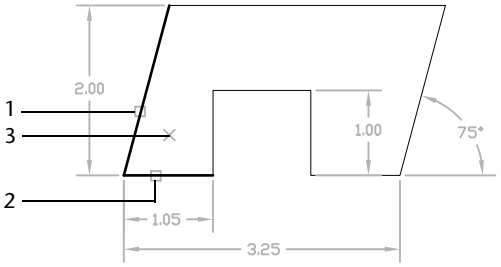
You removed the constraints that forced these lines to be parallel to one another. In order to force the outer lines to be complementary angles to one another, you need to add an angular dimension to the leftmost line.

3 Use AMPARDIM to add an angular dimension, responding to the prompts.

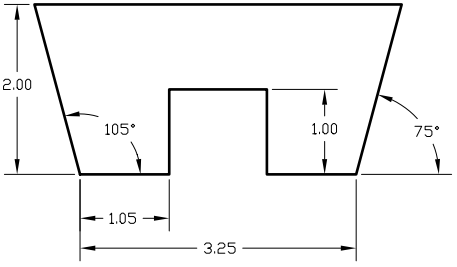
Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Select near the middle of line (1)*
Select second object or place dimension: *Select near the middle of line (2)*
Specify dimension placement: *Place the dimension (3)*
Enter dimension value or [Undo/Placement point] <75>: *Enter* **105**
Solved fully constrained sketch.
Select first object: *Press* ENTER

NOTE If you do not select the lines near their midpoints, you may be prompted to specify the type of dimension to create. Choose Angular.



You have modified the geometric constraint scheme to reshape the sketch.



Save your file.

Next, you learn to use parametric dimensions to constrain the shape of a sketch.

Applying Dimension Constraints

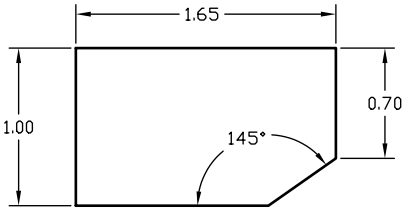
It is good practice to stabilize the shape of a sketch with geometric constraints before you specify size with dimensional constraints.

Dimensions specify the length, radius, or rotation angle of geometric elements in the sketch. Unlike geometric constraints, dimensions are parametric; changing their values causes the geometry to change.

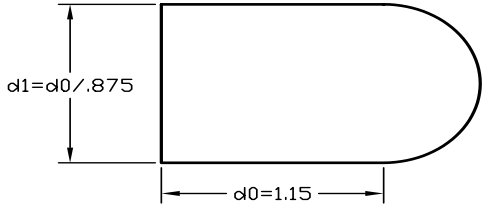
Dimensions can be shown as numeric constants or as equations. Although you can use them interchangeably, they each have specific uses.

- Numeric constants are useful when a geometric element has a static size and is not related to any other geometric element.
- Equations are useful when the size of a geometric element is proportional to the size of another element.

In the following illustration, all of the lines and the angles are constant, and stated as numeric values.



In the next illustration, the dimensions are expressed as equations.



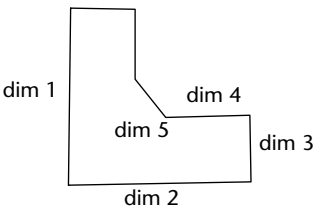
In this case, the height of the sketch must maintain the same proportion to the length, even if you change dimensions later. In an equation, you can state the height relative to the length. The dimension for the vertical line is defined as an equation of d1 = d0/.875 where d1 is the parameter name for the vertical line and d0 is the parameter name for one of the horizontal lines.

The *d* variables in the equations are parameter names assigned by Mechanical Desktop when you define the parameters. The letter *d* indicates that the parameter is a dimension. The number signifies the dimension number relative to the beginning of the dimensioning sequence.

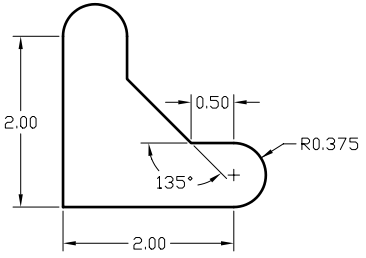
Open the file *sketch6.dwg* in the *desktop\tutorial* folder. Add and modify dimensions to complete the definition of the following sketch.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The before-and-after sketches reveal where dimensions are needed and in what order you should place them. The dimensions needed here have already been identified and are expressed as numeric constants.



original sketch



profiled sketch

To keep the sketch shape from becoming distorted as the dimensions resize it, define larger dimensions first: the left vertical line (dim 1) and the bottom horizontal line (dim 2).

By adding geometric constraints, you can reduce the number of dimensions you need. Later, you can modify the sketch with fewer changes.

After the basic shape has been defined, you replace the rightmost vertical line and the top horizontal line with fillets, and add geometric constraints and dimensions to finish the profile.

Creating Profile Sketches

First, convert the unconstrained sketch to a profile sketch before you add dimensions. Then examine the default geometric constraints.

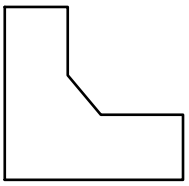
To create a profile from a sketch and examine constraints

1 Use AMPROFILE to create a profile from the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Mechanical Desktop redraws the sketch and reports that it still needs six dimensions or constraints to solve the sketch:

Solved under constrained sketch requiring 6 dimensions or constraints.

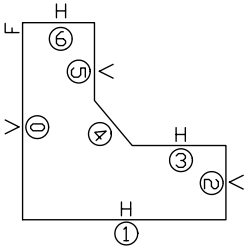


Examine the inferred geometric constraints and determine if the default constraints are correct or whether they inhibit the dimensions you want to add.

2 Use AMSHOWCON to display the constraints, responding to the prompt.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

Enter an option [All/Select/Next/eXit] <eXit>: *Enter* **a**



Mechanical Desktop recalculates the sketch and displays the constraints.

- A fix constraint is added to the start point of the first line of the sketch. This point is anchored and will not move when changes are made to the sketch constraints.
- Nearly horizontal and vertical lines have been assigned horizontal (H) and vertical (V) constraints.
- Nearly vertical lines are assumed to be parallel (P) to one another.

For this exercise, all of the assumed geometric constraints are correct and none of them restrict the dimensioning scheme shown earlier.

Exit from Show Constraints, responding to the prompt as follows:

Enter an option [All/Select/Next/eXit] <eXit>: *Press* ENTER

Adding Dimensions

The rough sketch is converted to a profile sketch, and default geometric constraints are applied. Now you need to fully constrain the sketch by adding four dimensions and two geometric constraints. Parts are resized as you change parametric dimensions to refine your design, while all geometric relationships are maintained.

Keep the following points in mind as you are adding dimensions:

- Select the elements to dimension and choose where to place the dimension.
- Dimension type depends on the element you choose and where you place the dimension. The current size of the selected element is shown.
- You can accept the calculated size or specify a new value.
- The sketch element is resized according to the dimension value and the dimension is placed at the location you chose.

It is good practice to accept the automatically calculated dimensions to stabilize the sketch shape, particularly large outer dimensions. When you later modify dimensions to exact sizes, the sketch shape is less likely to become distorted.

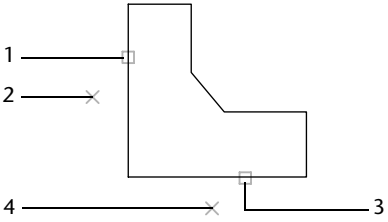
In this exercise, you create horizontal and vertical dimensions. Then you modify the sketch by appending geometry, and applying angular and radial dimensions.

To add a dimension to a profile

1 Use AMPARDIM to add dimensions to your profile, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify the line (1)*
Select second object or place dimension: *Place the dimension (2)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <1.9606>: *Enter* **2**
Solved under constrained sketch requiring 5 dimensions or constraints.



The sketch is updated with the new dimension value.

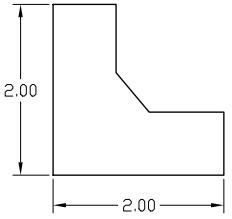
The command line lists several options. These options and the number of elements you select determine the type and placement of dimensions.

In this example, you choose a line and the placement of the dimension. If you selected two elements and specified a location, Mechanical Desktop would place a dimension that gives the distance between the two elements.

2 Continue dimensioning the sketch by choosing the bottom horizontal line.

Select first object: *Specify the line (3)*
Select second object or place dimension: *Place the dimension (4)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <2.1123>: *Enter* **2**
Solved under constrained sketch requiring 4 dimensions or constraints.
Select first object: *Press* ENTER

Mechanical Desktop redraws the sketch according to the new dimension value.



Now that the default constraints and larger dimensions have stabilized the sketch shape and size, you can begin to make changes to the sketch. To practice changing and updating the sketch, you add fillets to the two legs of the sketch.

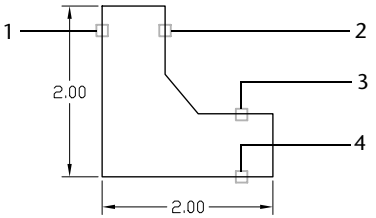
To add a fillet to a sketch

1 Use AMFILLET to apply a fillet, entering the points in the order shown.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Fillet.

Current settings: Mode = TRIM, Radius = 0.1250
Select first object or [Polyline/Radius/Trim]: *Specify the line (1)*
Select second object: *Specify the line (2)*

NOTE Because you selected parallel lines, FILLET ignores the radius value and joins the endpoints of the selected lines with a continuous arc.

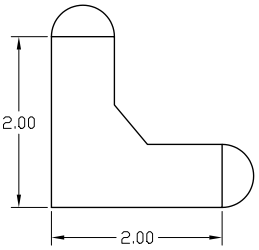


2 Apply a fillet to the other leg of the sketch.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Fillet.

Current settings: Mode = TRIM, Radius = 0.1250
Select first object or [Polyline/Radius/Trim]: *Specify the line (3)*
Select second object: *Specify the line (4)*

Your sketch should now look like this.

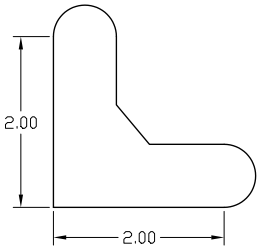


Before you continue defining your sketch, erase the horizontal line and the vertical line joining the endpoints of the new arcs.

3 Erase the two lines.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Erase.

Your drawing should look like this.



Because you have changed the sketch, you must re-solve it before you can use it to create a feature.

Appending Sketches

By adding the fillets and removing the lines, you have changed the sketch geometry. Whenever you add, modify, or remove geometry you must append the changed geometry to the profile sketch. You will be prompted to select any new geometry you have created. Mechanical Desktop appends the new geometry and recalculates the sketch, assigning new geometric constraints.

After appending the sketch, re-examine the geometric constraints to see if they affect your dimensioning scheme.

To append a profile sketch and re-examine geometric constraints

1 Expand the hierarchy of PART1_1.

2 Use AMRSOLVESK to append the existing fillets, responding to the prompts.

Context Menu In the graphics area, right-click and choose Append Sketch.

Select geometry to append to sketch: *Specify the first arc*
Select geometry to append to sketch: *Specify the second arc*
Select geometry to append to sketch: *Press* ENTER
Redefining existing sketch.
Solved under constrained sketch requiring 4 dimensions or constraints.

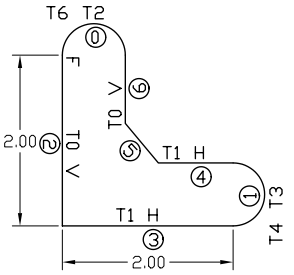
Mechanical Desktop analyzes and redraws the profile in accordance with its sketch analysis rules. Four additional constraints are needed to fully constrain the sketch.

3 Use AMSHOWCON to display the constraint symbols.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

Press ENTER to exit the command.

4 Display all of the symbols. Several tangent (T) constraints are added to the original geometric constraints.



The tangent constraints join the arcs to their adjoining lines. Notice that although the sketch segment numbers have changed because of the new geometry, the fix constraint remains in the same location.

For this exercise, do not delete any constraints because the tangent constraints do not adversely affect the dimensioning scheme. Now that you have recreated the profile sketch, you can continue to add geometric constraints and dimensions to the sketch, starting with a radial constraint to the two arcs.

Depending on how you drew your sketch, your default dimension values may differ from those in this exercise.

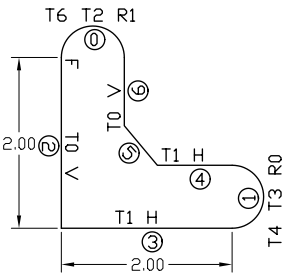
To add constraints to a re-created profile sketch

1 Use AMADDCON to add a radial constraint to the two arcs, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Radius.

Valid selections: arc or circle
Select object to be resized: *Specify an arc*
Valid selections: arc or circle
Select object radius is based on: *Specify the other arc*
Solved under constrained sketch requiring 3 dimensions or constraints.
Valid selections: arc or circle
Select object to be resized: *Press* ENTER
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix] <eXit>: *Press* ENTER

Mechanical Desktop adds radius constraints to the two arcs.

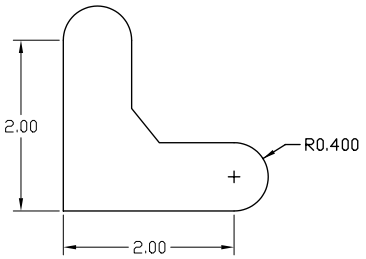


Finish constraining the sketch by adding three dimension constraints.

2 Use AMPARDIM to dimension the leftmost arc, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

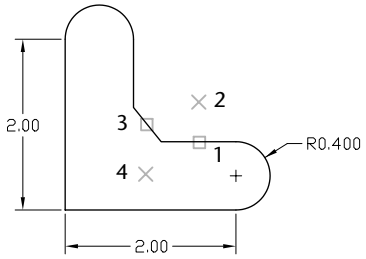
Select first object: *Specify the lower arc*
Select second object or place dimension: *Place the dimension*
Enter dimension value or [Undo/Diameter/Ordinate/Placement point] <0.3687>: *Enter* **.4**
Solved under constrained sketch requiring 2 dimensions or constraints.



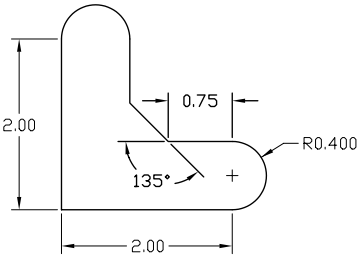
After you enter the new radius value, the arcs are updated because the radius constraint makes both arcs equal.

3 Add the final two dimensions by responding to the prompts as follows:

Select first object: *Specify the line (1)*
Select second object or place dimension: *Place the dimension (2)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.8753>: *Enter* **.75**
Solved under constrained sketch requiring 2 dimensions or constraints.
Select first object: *Specify near the middle of line (1)*
Select second object or place dimension: *Specify near the middle of line (3)*
Specify dimension placement: *Place the dimension (4)*
Enter dimension value or [Undo/Placement point] <138>: *Enter* **135**
Solved fully constrained sketch.
Select first object: *Press* ENTER



The dimensions are placed. Your sketch should be fully constrained..



Save your file.

Modifying Dimensions

Because your design changes during development, you must be able to delete or modify dimension values. Mechanical Desktop parametric commands ensure that relationships among geometric elements remain intact.

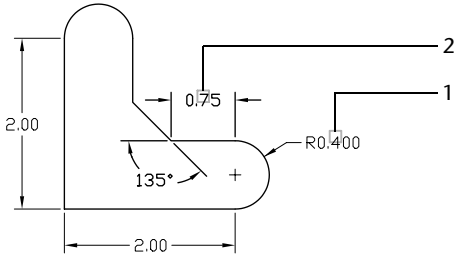
To finish the sketch, change the dimension of the top horizontal line and the angular dimension.

To change a dimension

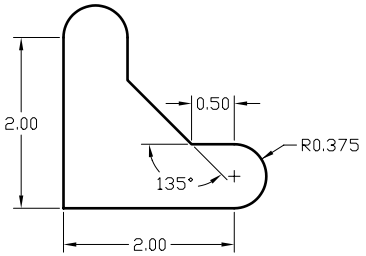
1 Use AMMODDIM to modify the dimensions, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Specify the dimension (1)*
New value for dimension <.4>: *Enter* **.375**
Solved fully constrained sketch.
Select dimension to change: *Specify the dimension (2)*
New value for dimension <.75>: *Enter* **.5**
Solved fully constrained sketch.
Select dimension to change: *Press* ENTER



Your finished sketch should now look like this.



Save your file.

Using Construction Geometry

Construction geometry can minimize the number of constraints and dimensions needed in a sketch and offers more ways to control sketch features. Construction geometry works well for sketches that are symmetrical or have geometric consistencies. Some examples are sketches that have geometry lying on a radius, a straight line, or at an angle to other geometry.

Construction geometry is any line, arc, or circle in the sketch profile or path that is a different linetype from the sketch linetype. By default, construction geometry is placed on the AM_CON layer. To make construction geometry easier to see, you can change its color, linetype, or linetype scale.

Construction geometry can be used to constrain only the sketch it is associated with. When you create a feature from a sketch, you also select the construction geometry with the path or profile sketch. After the feature is created, the construction geometry is no longer visible.

Creating Profile Sketches

In this exercise, you follow a typical sequence. As always, study the sketch to determine what constraints and dimensions you need and decide where to place construction geometry to make solving the sketch easier.

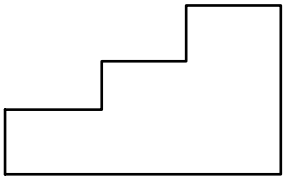
Open the file *sketch7.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

To create a single profile sketch

1 Use PLINE to draw the rough sketch.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.



2 Use AMSOLVE to solve the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

The polyline is automatically selected.

Mechanical Desktop applies constraints according to how you sketch and then reports that the sketch needs six or more additional constraints. A fix constraint is automatically applied to the point where you started your sketch.

3 Use AMSHOWCON to display the existing constraints.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

4 Display all of the assumed constraint symbols. Each of the eight lines should have a vertical or horizontal constraint.

Next, create a construction line to assist in constraining the sketch.

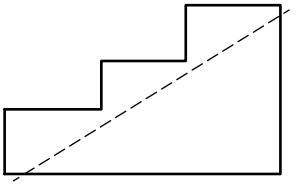
NOTE If necessary, remove the fix constraint using AMDELCON. This constraint prevents you from projecting the sketch to the construction line.

To create a construction line

1 Create a construction line.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Construction Line.

2 Draw the line diagonally across the sketch.



Mechanical Desktop draws the line on a new layer called AM_CON. The line is yellow and drawn with the HIDDEN linetype. Because the linetype is different from the one used to draw the sketch, the line is considered construction geometry. It is used only in this sketch.

3 Use AMRSOLVESK to append the profile.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Append.

4 Select the construction line.

5 Re-examine the assumed constraints.

Adding Project Constraints

Mechanical Desktop recognizes nine lines in the sketch. The sketch requires two more constraints because you added a construction line.

Next, project the construction line to each vertex that serves as an inner corner of a stair.

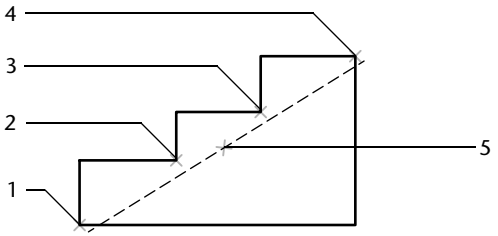
To place a project constraint, specify a vertex and then select the construction line. Depending on how closely you drew the construction line to the vertices, some constraints may have already been applied.

To add a project constraint

1 Use AMADDCON to add the project constraints, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Project.

Valid selections: line, circle, arc, ellipse or spline segment
Specify a point to project: *Enter* **end**
of: *Specify point (1)*
Valid selections: line, circle, arc, ellipse, work point or spline segment
Select object to be projected to: *Specify the construction line (5)*
Valid selections: line, circle, arc, ellipse or spline segment
Specify a point to project:
Repeat this process for points (2) through (4), then press ENTER

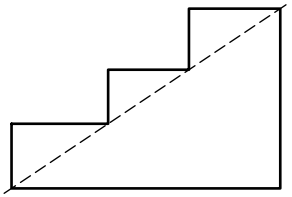


NOTE If you do not use the endpoint object snap, you will not be able to correctly constrain the sketch.

By defining the slope of the stairs with the construction line, you have reduced the number of required constraints and dimensions to four.

2 Use REDRAW to clean up the screen display.

Desktop Menu View ➤ Redraw



Adding Parametric Dimensions

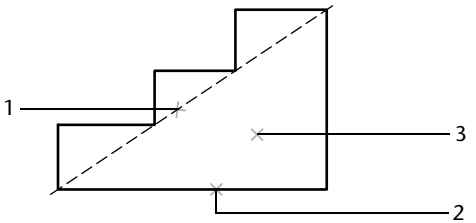
To fully define the sketch, dimension one of the risers and apply a slope angle for the construction line. Each step is equal in height, so you can add equal length constraints to the remaining steps later.

To add a parametric dimension

1 Use AMPARDIM to dimension the slope angle, responding to the prompts.

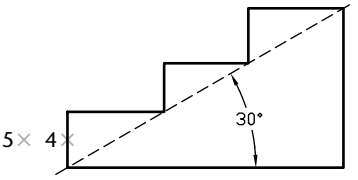
Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify near the middle of the construction line (1)*
Select second object or place dimension:
Specify near the middle of the bottom horizontal line (2)
Specify dimension placement: *Specify a point to right (3)*
Enter dimension value or [Undo/Placement point] <31>: *Enter* **30**
Solved under constrained sketch requiring 3 dimensions or constraints.



2 Continue, adding dimensions to the first vertical riser.

Select first object: *Specify a point near the center of the lower left vertical line (4)*
Select second object or place dimension: *Specify a point to left of first point (5)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.9463>: *Enter* **1**
Solved under constrained sketch requiring 2 dimensions or constraints.
Select first object: *Press* ENTER



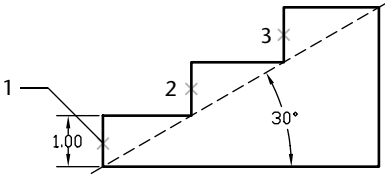
To finish constraining the sketch, add equal length dimensions to the remaining two risers.

To add an equal length constraint

1 Use AMADDCON to add an equal length constraint, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Equal Length.

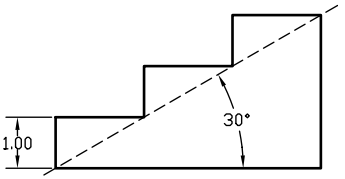
Valid selections: line or spline segment
Select object to be resized: *Specify the second riser (2)*
Valid selections: line or spline segment
Select object to base size on: *Specify the dimensioned riser (1)*
Solved under constrained sketch requiring 1 dimensions or constraints.



2 Continue on the command line to place the last constraint.

Valid selections: line or spline segment
Select object to be resized: *Specify the third riser (3)*
Valid selections: line or spline segment
Select object to base size on: *Specify the dimensioned riser (1)*
Solved fully constrained sketch.

You should now have a fully constrained sketch. Exit the command by pressing ENTER twice.



3 Use AMMODDIM to change the angular dimension, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Specify the angular dimension*
New value for dimension <30>: *Enter* **25**
Select dimension to change: *Press* ENTER

Save your file.

Constraining Path Sketches

Construction geometry helps you constrain sketches that may be difficult to constrain with only the geometry of the sketch shape. In this exercise, you create a path sketch, add a construction line, and constrain the sketch to the line.

Before you begin this exercise, create a new part definition for the sketch.

To create a new part definition

1 Use AMNEW to create a new part definition.

Context Menu In the graphics area, right-click and choose Part ➤ New Part.

2 Press ENTER on the command line to accept the default part name.

3 Pan the drawing so you have room to create the next sketch.

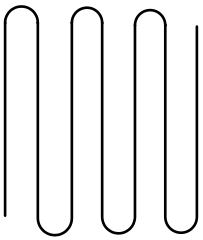
Context Menu In the graphics area, right-click and choose Pan.

You are ready for the next exercise.

To use construction geometry in a swept path

1 Use PLINE to draw the following sketch.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.



Use the arc/direction option of PLINE to draw the arcs. You can also use your cursor crosshairs to visually align the endpoints of each arc as you sketch.

NOTE To enlarge the crosshairs, choose Assist ➤ Options. Under Crosshair Size, set the size to 15 or larger.

---

2 Use AM2DPATH to create a 2D path from your sketch, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ 2D Path.

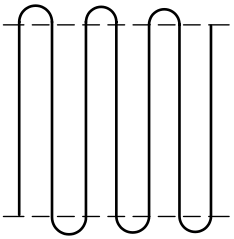
Select objects: *Specify the polyline*
Select objects: *Press* ENTER
Specify the start point of the path: *Specify one of the ends of the path*
Solved under constrained sketch requiring 10 dimensions or constraints.
Create a profile plane perpendicular to the path? [Yes/No] <Yes>: *Enter* **n**

You can use either end for the start point.

Mechanical Desktop reports that the sketch needs ten or more additional constraints, depending on how you drew the sketch.

3 Draw two construction lines. The goal is to have each of the ends of the arcs meet the construction lines.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Construction Line.



4 In the Desktop Browser, expand the PART2_1 hierarchy.

5 Use AMRSOLVESK to append the construction lines to your sketch, following the prompts.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Append.

Select geometry to append to sketch: *Specify a construction line*
Select geometry to append to sketch: *Specify the other construction line*
Select geometry to append to sketch: *Press* ENTER
Redefining existing sketch.
Specify start point of path: *Specify one of the ends of the path*
Solved under constrained sketch requiring 6 dimensions or constraints.

The construction lines have reduced the number of constraints or dimensions needed by constraining the arc endpoints and centers to the line. The construction lines have been made horizontal as well.

To check for and add missing constraints

1 Use AMSHOWCON to check for constraints that are still needed.

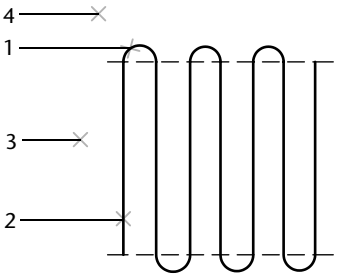
Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

2 Display all the constraints and press ENTER to exit the command.

3 Use AMADDCON to add constraints and dimensions to the sketch, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify the upper left arc (1)*
Select second object or place dimension:
Specify the vertical line on the left below its midpoint (2)
Specify dimension placement: *Specify a point to the left of the sketch (3)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace]
<3.1069>: *Enter* **3**
Solved under constrained sketch requiring 5 dimensions or constraints.



4 Add a second dimension.

Select first object: *Specify the upper left arc (1)*
Select second object or place dimension:
Specify a point above and left of sketch (4)
Enter dimension value or [Undo/Diameter/Ordinate/Placement point]
<0.2788>: *Enter* **.25**
Solved under constrained sketch requiring 4 dimensions or constraints.
Select first object: *Press* ENTER

Next, you fully solve the path by adding 2D constraints.

5 Constrain all the arcs with the same radius as the one you just dimensioned, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Radius.

Valid selections: arc or circle
Select object to be resized: *Specify the lower left arc*
Valid selections: arc or circle
Select object radius is based on: *Specify the arc with the radial dimension*
Solved under constrained sketch requiring 3 dimensions or constraints.
Valid selections: arc or circle
Select object to be resized: *Specify the upper arc that is second from the left*
Valid selections: arc or circle
Select object radius is based on: *Specify the arc with the radial dimension*
Solved under constrained sketch requiring 2 dimensions or constraints.
Valid selections: arc or circle
Select object to be resized: *Specify the lower arc that is second from the left*
Valid selections: arc or circle
Select object radius is based on: *Specify the arc with the radial dimension*
Solved under constrained sketch requiring 1 dimensions or constraints.
Valid selections: arc or circle
Select object to be resized: *Specify the upper right arc*
Valid selections: arc or circle
Select object radius is based on: *Specify the arc with the radial dimension*
Solved fully constrained sketch.
Valid selections: arc or circle
Select object to be resized: *Press* ENTER
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix] <eXit>: *Press* ENTER

Your sketch should now be fully constrained. You may need to use the Equal Length constraint for the beginning and end vertical line segments of your sketch. Experiment with this sketch by changing the values of the two dimensions.

If arc centers do not lie on the construction line, use the project constraint. Add project constraints until the sketch is fully constrained.

NOTE Depending on how accurately you sketched the path, you may need to add other constraints. Experiment until your sketch is fully constrained. If you have difficulty, delete the sketch and try again.

Save your file.

Controlling Tangency

A single piece of construction geometry can manage the size and shape of entire sketches. Circles and arcs are particularly useful for constraining the perimeter shapes of nuts, knobs, multisided profiles, and common polygons.

In this exercise, you create a triangular sketch and then constrain the sides of the triangle and the internal angles to remain equal. In this manner, you could form the basis for a family of parts in which the only variable is a single diameter dimension.

Create a new part definition for the next sketch.

To create a new part definition

1 Use AMNEW to create a new part definition.

Context Menu In the graphics area, right-click and choose Part ➤ New Part.

2 Accept the default part name.

3 Pan the drawing so you have room to create the next sketch.

Context Menu In the graphics area, right-click and choose Pan.

You are ready to create the next sketch.

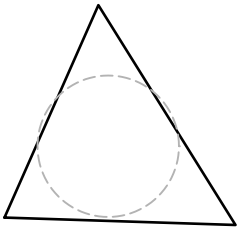
To control tangency with construction geometry

1 Use PLINE to create the triangular shape.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.

2 Draw a circle inside the triangle.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Construction Circle.



3 Use AMPROFILE to turn the sketch into a profile sketch, making sure to select both the polyline and the circle.

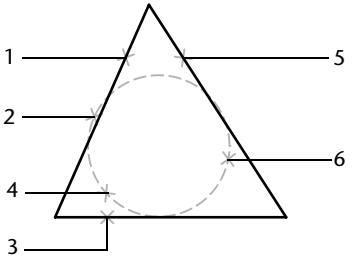
Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Profile.

At this point, the circle may be tangent to some or all of the sides of the triangle.

4 Use AMADDCON to add Tangent constraints to the sketch, following the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Tangent.

Valid selections: line, circle, arc, ellipse or spline segment
Select object to be reoriented: *Specify the line (1)*
Valid selections: line, circle, arc, ellipse or spline segment
Select object to be made tangent to: *Specify the circle (2)*
Solved under constrained sketch requiring 5 dimensions or constraints.
Valid selections: line, circle, arc, ellipse or spline segment
Select object to be reoriented: *Specify the line (3)*
Valid selections: line, circle, arc, ellipse or spline segment
Select object to be made tangent to: *Specify the circle (4)*
Solved under constrained sketch requiring 4 dimensions or constraints.
Valid selections: line, circle, arc, ellipse or spline segment
Select object to be reoriented: *Specify the line (5)*
Valid selections: line, circle, arc, ellipse or spline segment
Select object to be made tangent to: *Specify the circle (6)*
Solved under constrained sketch requiring 3 dimensions or constraints.
Valid selections: line, circle, arc, ellipse or spline segment
Select object to be reoriented: *Press* ENTER
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix] <eXit>: *Press* ENTER



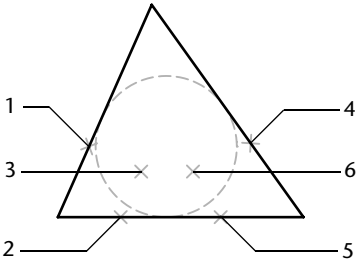
Mechanical Desktop now needs three or more dimensions or constraints to fully solve the sketch.

To add a dimension to an angle

1 Use AMPARDIM to apply angular dimensions to the triangle, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify near the middle of the line (1)*
Select second object or place dimension: *Specify near the middle of the line (2)*
Specify dimension placement: *Place the dimension (3)*
Enter dimension value or [Undo/Placement point] <67>: *Enter* **60**
Solved under constrained sketch requiring 2 dimensions or constraints.

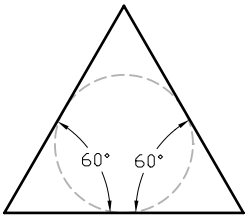


2 Continue on the command line.

Select first object: *Specify near the middle of the line (4)*
Select second object or place dimension: *Specify near the middle of the line (5)*
Specify dimension placement: *Place the dimension (6)*
Enter dimension value or [Undo/Placement point] <78>: *Enter* **60**
Solved under constrained sketch requiring 1 dimensions or constraints.
Select first object: *Press* ENTER

NOTE If you do not select the lines near their midpoints, you may be prompted to specify the type of dimension to create. Choose Angular.

The angular dimensions should look like these.



To add a dimension to a circle

1 Add a dimension to the diameter of the construction circle, following the prompts.

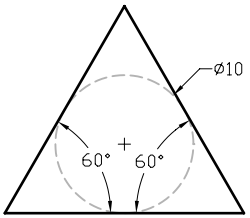
Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify a point on the circle*
Select second object or place dimension: *Specify a point outside of the triangle*
Enter dimension value or [Undo/Radius/Ordinate/Placement point] <3.1541>: *Enter* **10**
Solved fully constrained sketch.
Select first object: *Press* ENTER

The sketch should now be fully constrained.

2 Zoom out to view the entire sketch.

Context Menu In the graphics area, right-click and choose Zoom.



NOTE If the bottom segment of your triangle is still not horizontal, you will need to add a Horizontal constraint to fully constrain the sketch.

3 Experiment with the size of the sketch. Use AMMODDIM to change the diameter dimension of the circle, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

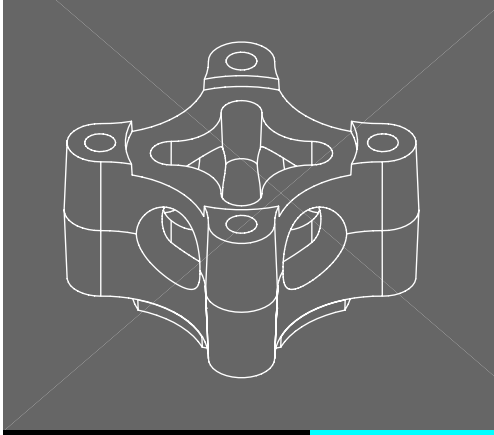
Select dimension to change: *Specify the diameter dimension*
New value for dimension <10>: *Enter* **5**
Solved fully constrained sketch.
Select dimension to change: *Press* ENTER

Save your file.

All sides remain equal in length and tangent to the circle, and the bottom of the triangle remains horizontal. If you used this sketch as a base feature of a part, you could change the overall size of the part simply by changing the diameter of the construction circle.

This technique could be applied to more complex geometry such as pentagons, octagons, and odd-shaped polygons. These shapes can form the base feature for a family of nuts, bolts, fittings, and so on. Try these types of sketches on your own.

Creating Sketched Features



In This Chapter

8

- Extruded features
- Loft features
- Revolved features
- Face splits
- Sweep features

Features are the parametric building blocks of parts. By creating and adding features you define the shape of your part. Because features are parametric, any changes to them are automatically reflected when the part is updated.

In Autodesk® Mechanical Desktop®, there are three types of features—sketched, work and placed.

In this tutorial, you learn to create and modify sketched features. In chapter 4, you learn about work features.

Key Terms

Term	Definition
base feature	The first feature you create. As the basic element of your part, it represents its simplest shape. All geometry you create for a part depends on the base feature.
Boolean modeling	A solid modeling technique in which two solids are combined to form one resulting solid. Boolean operations include cut, join, and intersect. Cut subtracts the volume of one solid from the other. Join unites two solid volumes. Intersect leaves only the volume shared by the two solids.
consumed sketch	A sketch used in a feature, for example, an extruded profile sketch. The sketch is consumed when the feature is created.
cubic loft	A feature created by a gradual blending between two or more planar sections.
draft angle	An angle applied parallel to the path of extruded, revolved, or swept surfaces or parts. A draft angle is used to allow easy withdrawal from a mold or easy insertion into a mated part.
extrude	In part modeling, to create a geometric sketch defined by a planar profile extended along a linear distance perpendicular to the profile plane.
feature	An element of a parametric part model. You can create extruded features, revolved features, loft features, and swept features using profiles and paths. You can also create placed features like holes, chamfers, and fillets. You combine features to create complete parametric part models.
helical sweep	A geometric feature defined by the volume from moving a profile along a 3D path about a work axis.
linear loft	A feature created by a linear transition between two planar sections.
lofted feature	A parametric shape created from a series of sketches defining the cross-sectional shape of the feature at each section.
revolve	In part modeling, to create a feature by revolving a profile about an axis of revolution.
sketch plane	A temporary drawing surface that corresponds to a real plane on a feature. It is an infinite plane with both *X* and *Y* axes on which you sketch or place a feature.
sketched feature	A three-dimensional solid whose shape is defined by constrained sketches and located parametrically on a part. Sketched features are extrudes, lofts, revolves, sweeps, or face splits.
sweep	A geometric sketch feature defined by the volume from moving a profile along a path.
swept profile	A special parametric sketch used to create a swept feature from the cross section of a profile.

Basic Concepts of Sketched Features

Features are the building blocks you use to create and shape a part. Because they are fully parametric, they can easily be modified at any time.

The first feature in a part is called the *base feature*. As you add more features, they can be combined with the base feature or each other to create your part.

Boolean operations, such as cut, join, and intersect, can be used to combine features after a base feature has been created.

You create a sketched feature from a profile, which is an open or closed parametric sketch that has been solved. You can also create a feature from a text-based sketch. In most cases, you fully constrain the profile before you create a feature. Because a sketch is parametric, you can easily modify it to change the shape of the feature. When you update your part, the changes you made are displayed automatically.

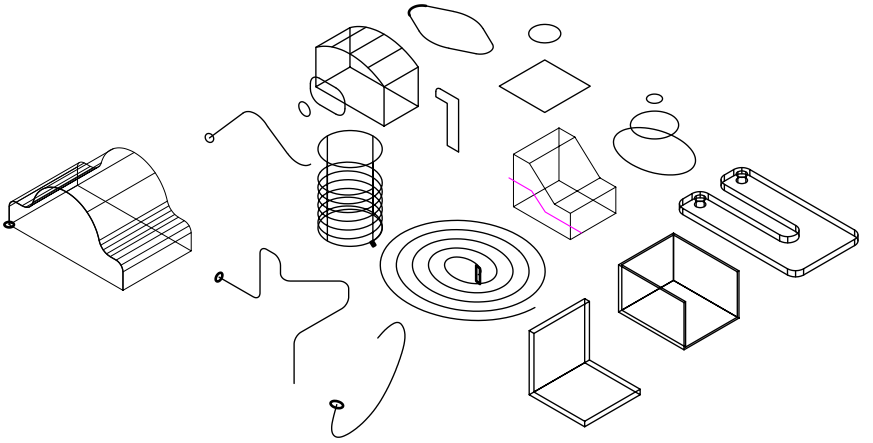
Sketched features include extrusions, lofts, revolutions, sweeps, and embossing. Face splits are also considered sketched features, but they are created by splitting a part face using an existing face, a work plane, or a split line. If you choose the split line method, you are using a sketched feature to split the face.

In this tutorial you learn how to create and edit sketched features. Later you learn how to create and edit work features and placed features.

Open the file *s_feat.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing file includes fifteen parts which contain the geometry you need to create the sketched features in this section.



NOTE For clarity, the work features are not shown.

First, you create an extruded feature.

Creating Extruded Features

Extrusions are the most common sketched features. An extruded feature can be created from a closed profile, an open profile, or a text-based profile.

Extruding Closed Profiles

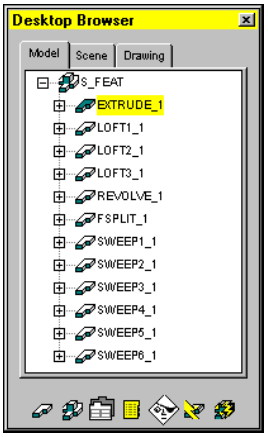
A closed profile is used to create a base feature, or in Boolean modeling to cut, intersect, and join with other features.

In the first exercise, you use the part EXTRUDE_1. Activate the part, and expand the hierarchy of EXTRUDE_1.

To activate a part

Browser Double-click EXTRUDE_1.

Click the plus sign in front of EXTRUDE_1 to expand the hierarchy.



Clear the visibility of the other parts, and display the dimensions and work features of the active part.

To turn off the visibility of multiple parts

Browser Select EXTRUDERIB_1, then hold down SHIFT as you select BEND_1. Right-click the selected block and choose Visible.

NOTE Because most of the parts do not contain features yet, you cannot use the toolbutton, menu, or command methods to make the part instances invisible.

Click the plus sign in front of EXTRUDE_1 to expand the hierarchy.

To thaw dimension and work layers

Desktop Menu Assist ➤ Format ➤ Layer

The Layer Properties Manager dialog box is displayed.

In the AM_PARDIM layer, select the On icon and the Freeze icon to unthaw the layer. Repeat for the AM_WORK layer.

Choose OK to exit the dialog box.

The parametric dimensions and work features for each part are now visible.

To zoom in to a part

Browser Right-click EXTRUDE_1, and choose Zoom to.

The EXTRUDE_1 part is positioned on your screen.

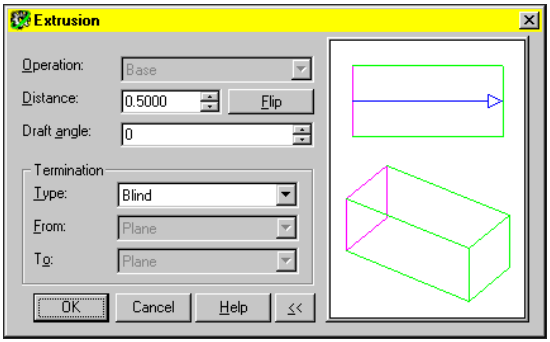
To create an extruded feature

1 Use AMEXTRUDE to create an extruded feature from Profile1.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

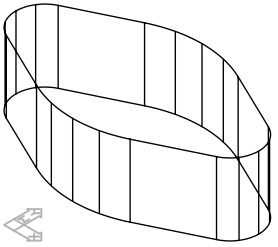
Distance: *Enter* **0.5**
Termination: Type: Blind



The image tile indicates the direction of the extrusion.

Choose OK.

The profile is extruded perpendicular to the plane of the profile.



Next, you create and constrain another profile, and extrude it to cut material from the base feature.

To create a profile sketch

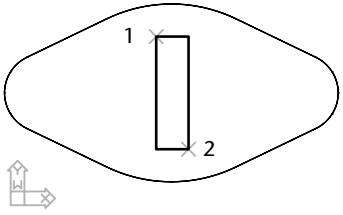
1 Change to the top view of your part.

Desktop Menu View ➤ 3D Views ➤ Top

2 Use RECTANGLE to sketch a rectangle as shown in the following illustration, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Rectangle.

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
Specify a point (1)
Specify other corner point: *Specify a second point (2)*



3 Use AMRSOLVESK to solve the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

The command line indicates the number of constraints required to fully constrain the profile.

Solved underconstrained sketch requiring 4 dimensions or constraints.

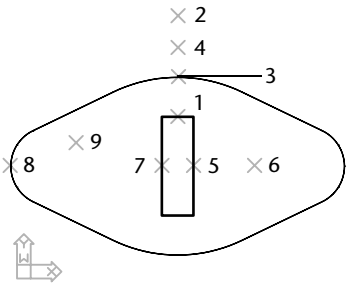
Before you extrude the profile, fully constrain it by adding four dimensional constraints.

To constrain a sketch

1 Use AMPARDIM to add parametric dimensions to fully constrain the sketch, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify the top edge (1)*
Select second object or place dimension: *Place the dimension (2)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.1574>: *Enter* **.16**
Solved underconstrained sketch requiring 3 dimensions or constraints.
Select first object: *Specify the top edge again (1)*
Select second object or place dimension: *Specify the top arc (3)*
Specify dimension placement: *Place the dimension (4)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.0730>: *Enter* **.08**
Solved underconstrained sketch requiring 2 dimensions or constraints.



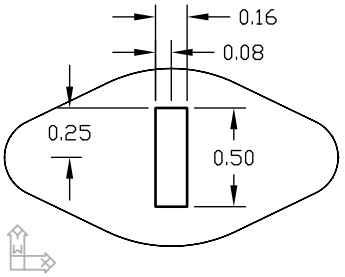
2 Continue creating the parametric dimensions.

Select first object: *Specify the right edge (5)*
Select second object or place dimension: *Place the dimension (6)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.4500>: *Enter* **.5**
Solved underconstrained sketch requiring 1 dimensions or constraints.
Select first object: *Specify the left edge (7)*
Select second object or place dimension: *Specify the left arc (8)*
Specify dimension placement: *Place the dimension (9)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.2430>: *Enter* **v**
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.2430>: *Enter* **.25**

After you finish dimensioning, the following message is displayed on the command line:

Solved fully constrained sketch.
Select first object: *Press* ENTER

Your sketch should look like this.



NOTE For clarity, the parametric dimensions controlling Profile1 are not shown.

Now that the profile is fully constrained, you extrude it into the base feature to cut material from your part.

To add an extruded feature to a part

1 Change to an isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

2 Extrude the profile.

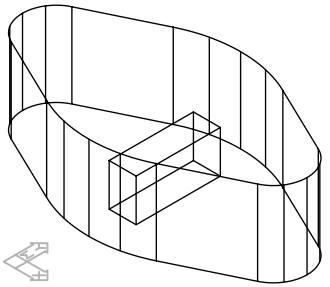
Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

3 In the Extrusion dialog box, specify the following:

Operation: Cut
Distance: *Enter* **0.25**
Termination: Blind

4 Choose OK to exit the dialog box.

Your part should look like this.



Save your file.

Editing Extruded Features

Because an extruded feature is controlled by parametric dimensions, you can easily make changes to it by modifying the values of the profiled sketch, or the extruded feature itself.

To modify a consumed profile

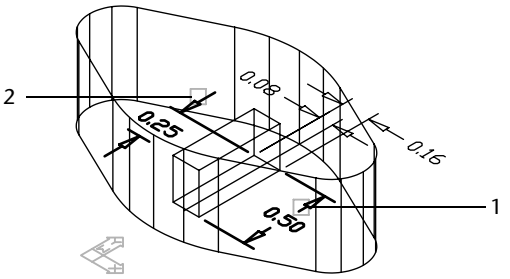
1. Expand ExtrusionBlind2 in the Browser.
2. Edit the dimensions of the profile used to define the shape of the extrusion, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Select the cut extrusion
Enter an option [Next/Accept] <Accept>: ExtrusionBlind2: *Press* ENTER

3. Choose OK to exit the Extrusion dialog box, then continue on the command line.

Select object: *Select the 0.5 dimension (1)*
Enter new value for dimension <.5>: *Enter* **1**
Solved fully constrained sketch.
Select object: *Select the 0.25 dimension (2)*
Enter new value for dimension <.25>: *Enter* **.5**
Solved fully constrained sketch.
Select object: *Press* ENTER

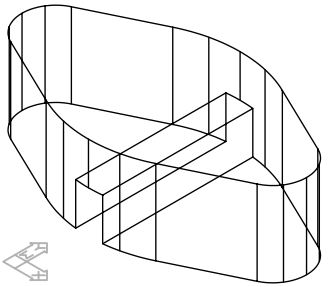


NOTE For clarity, the taper and depth dimensions are not illustrated.

4 Use AMUPDATE to update your part.

Context Menu In the graphics area, right-click and choose Update Part.

The part now reflects the changes to the profile that controls the shape of the extrusion you used to cut material from the part.



Next, modify the extrusion feature to change the depth of the cut.

To modify a feature

1 Select the cut extrusion to modify, responding to the prompt.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Press ENTER, *and select the cut extrusion*
Enter an option [Next/Accept] <Accept>: *Press* ENTER

2 In the Extrusion dialog box, specify a distance of **.15** and Choose OK.

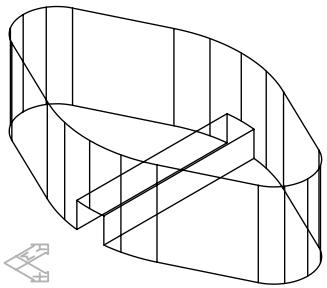
3 Continue on the command line.

Select object: *Press* ENTER

4 Use AMUPDATE to update the part.

Context Menu In the graphics area, right-click and choose Update Part.

Your part should look like this.



Save your file.

Extruding Open Profiles

You extrude open profiles to create rib features and thin features.

For more information about sketching open profiles, see “Creating Open Profile Sketches” on page 46.

Creating Rib Features

To create a rib feature on a part model, you sketch an open profile to shape the rib, define the thickness of the rib, and extrude it to part surfaces. Observe these rules when you sketch open profiles for ribs:

- Sketch the side view of the rib.
- The sketch can have any number of segments.
- The ends of the sketch need not touch surfaces the rib will attach to, but when extended must meet valid active part surfaces, without holes in the extrusion path.

You solve the sketch to create an open profile, and apply parametric constraints and dimensions as with any other profile sketch.

Like other features, the rib feature can be edited and it has dependencies. If you delete something in your model that a rib feature depends upon, such as a face that a profile plane is based on, you delete the rib feature as well.

In this exercise, you extrude a rib feature to two perpendicular walls of a part.

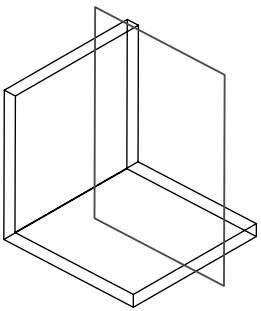
Turn off visibility for EXTRUDE_1, and make EXTRUDERIB_1 visible.

Browser Right-click EXTRUDE_1 and choose Visible. Then right-click EXTRUDERIB_1 and choose Visible.

Activate EXTRUDERIB_1 and position it on your screen.

Browser Double-click EXTRUDERIB_1. Then right-click EXTRUDERIB_1 and choose Zoom to.

In the previous exercise, you made the work feature layer visible.



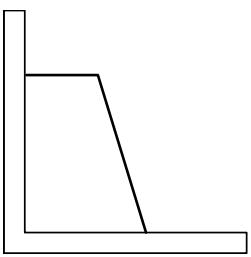
To create a rib feature

1 Change to the front view so you can sketch the rib from its side.

Desktop Menu View ➤ 3D Views ➤ Front

2 Use PLINE to sketch a rough outline of the rib, as shown in the following illustration. The sketch doesn't have to be touching the surfaces.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.



NOTE For clarity, the work plane is not shown.

3 Use AMPROFILE to solve the sketch, responding to the prompt.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Select part edge to close the profile <open profile>: *Press* ENTER

An icon for the open profile is displayed in the Browser.

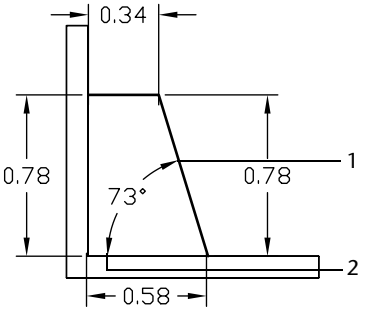
4 Use AMPARDIM to add an angular dimension between the lower wall and the lower section of the rib, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify a point on the rib (1)*
Select second object or place dimension: *Specify a point on the lower wall (2)*
Specify dimension placement: *Specify a point to place the dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <71>: *Enter a value of* **73**
Select first object: *Specify the next dimension to add*

Continue adding dimensions, as shown in the illustration, to fully constrain the open profile sketch.

Solved fully constrained sketch.
Select first object: *Press* ENTER

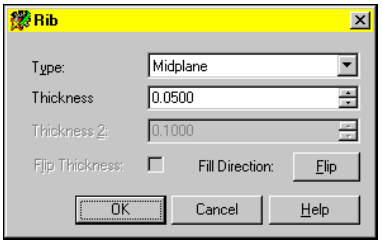


5 Use AMRIB to create the rib.

Browser In the Browser, right-click the open profile icon, and choose Rib.

In the Rib dialog box, specify:

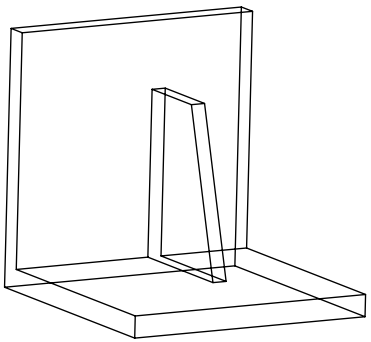
Type: Midplane
Thickness: **.05**



Choose OK.

6 Use 3DORBIT to rotate your part so you can see the rib feature.

Your part should look like this.



Creating Thin Features

To create a thin feature, you sketch an open profile and extrude it to part surfaces. When you extrude an open profile, the Extrusion dialog box includes the options for defining a thin wall feature. When you sketch open profiles for thin features

- Sketch must be an open profile from the front view
- Sketch is extruded normal to the sketch plane
- Ends of the open profile need not touch surfaces, but when extended must meet valid active part surfaces, without holes in the extrusion path

For more information about sketching open profiles, see “Creating Open Profile Sketches” on page 46.

In this exercise, you create a thin wall in a shell. In the Browser, turn off visibility for EXTRUDERIB_1, and make EXTRUDETHIN_1 visible.

Browser Right-click EXTRUDERIB_1 and choose Visible. Then right-click EXTRUDETHIN_1 and choose Visible.

Activate EXTRUDETHIN_1 and position it on your screen.

Browser Double-click EXTRUDETHIN_1. Then right-click EXTRUDETHIN_1 and choose Zoom to.

To create a thin feature

1 Use AMWORKPLN to create a work plane for the profile sketch.

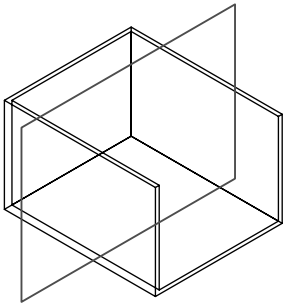
Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Plane.

In the Work Plane dialog box, specify:

1st Modifier: Planar Parallel
2nd Modifier: Offset
Offset: *Enter* **.75**

Choose OK.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the back face of the shell
Enter an option [Next/Accept] <Accept>: *Press* ENTER
Enter an option [Flip/Accept] <Accept>:
Flip to point arrow to back face, or press ENTER
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>: *Press* ENTER



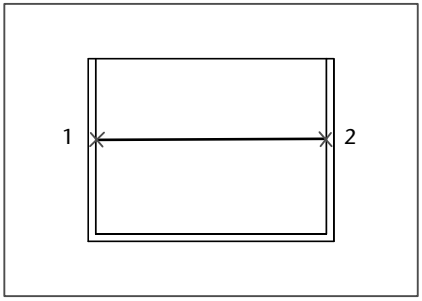
2 Change to the Right view to sketch the thin feature.

Desktop Menu View ➤ 3D Views ➤ Right

3 Use LINE to sketch the thin feature.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Line

Specify first point: *Specify the start point of the line (1)*
Specify next point or [Undo]: *Specify the end point of the line (2) and press* ENTER



NOTE Turn OSNAP off so that you will not snap to the back face when you pick.

4 Use AMRSOLVESK to solve the sketch, responding to the prompt.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Select part edge to close the profile <open profile>: *Press* ENTER

In the Browser, an open profile icon is displayed.

5 Change to the front right isometric view to extrude the profile.

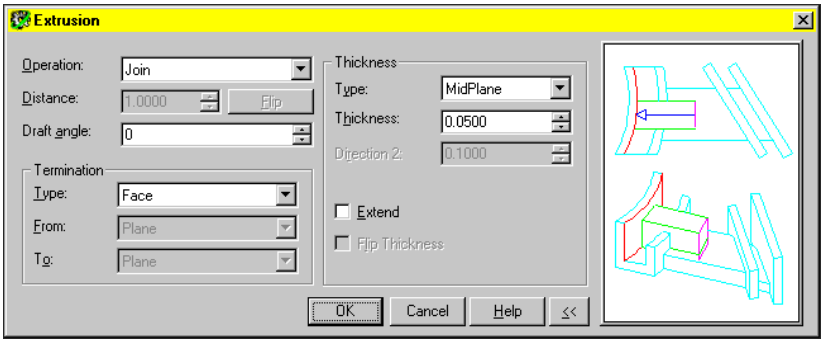
Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

6 USE AMEXTRUDE to extrude the open profile.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify

Operation: Join
Termination: Type: Face
Thickness: Type: Midplane
Thickness: *Enter* **.05**

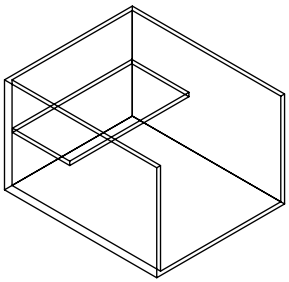


Choose OK.

7 Respond to the prompt:

Select Face: *Select the back face of the shell and press* ENTER
Enter an option [Next/Accept] <Accept>: *Press* ENTER

Your part should look like this.



A thin wall is created with equal thickness on each side of the profile. In the Browser, an icon is displayed for the thin extrusion.

NOTE When you extrude an open profile, the Extrusion dialog box contains options for defining a thin feature.

Save your file with a new name so you can use the same shell part for the next exercise.

Creating Emboss Features

Emboss features are text sketch profiles extruded on part models. A text sketch profile is one line of text displayed in a rectangular boundary.

To create a text sketch profile, you define a font and a style, and enter one line of text. Then you place the text on an active sketch plane on your part, and extrude it to emboss the surface of your part with the text.

Delete the thin extrusion from your shell part.

Browser Right-Click ThinExtrusionToFace1 and choose Delete.

Highlighted features will be deleted. Continue? [Yes/No] <Yes>: *Press* ENTER

Change to the Front view to create the emboss feature.

Desktop Menu View ➤ 3D Views ➤ Front

To create an emboss feature

1 Use AMWORKPLN to create a work plane for the text sketch.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Plane.

In the Work Plane dialog box, specify:

1st Modifier: On Edge/Axis
2nd Modifier: Planar Parallel

Choose OK.

2 Respond to the prompts:

Select work axis, straight edge or [worldX/worldY/worldZ]:
Select the top edge of the shell
Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the front face of the shell and press ENTER
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>: *Press* ENTER

NOTE Verify that CMDDIA is set to 1 so that the Text Sketch dialog box will be displayed. On the command line, enter **CMDDIA**, then enter **1**.

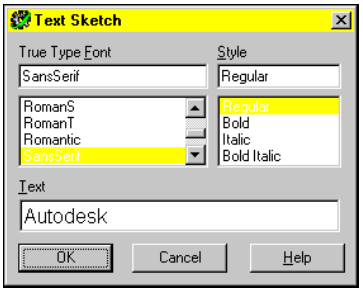
---

3 Use AMTEXTSK to create a text sketch profile.

Command AMTEXTSK

In the Text Sketch dialog box, specify:

True Type Font: Sans Serif
Style: Regular
Text: *Enter* **Autodesk**

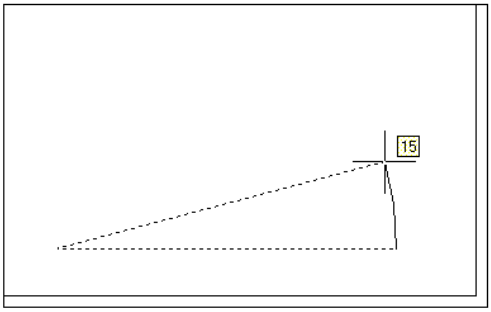


Choose OK.

4 Define a location for the text sketch with a rotation angle of 15, responding to the prompts.

Specify first corner: *Specify a point in the lower left corner of the shell*
Specify opposite corner or [Height/Rotation]: *Enter* **r** *and press* ENTER
Specify second angle endpoint or [Direction] <0>:
Move the cursor to the right and specify a rotation angle of 15

Hold the cursor in one location momentarily to display the angle dimension.

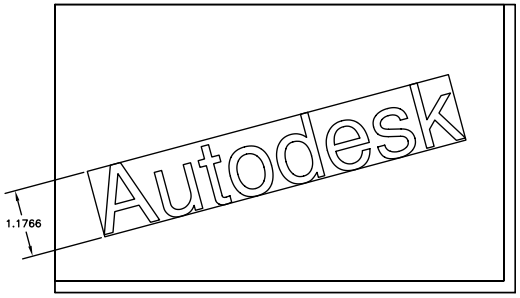


NOTE For clarity, the work plane is not shown.

Continue on the command line.

Specify opposite corner or [Height/Rotation]: *Specify a point for the height*

As you move the cursor, the rectangular border adjusts to accommodate the size of the text.



In the Browser, an icon is displayed for the text sketch.

You can change the parametric dimension for the height, and you can control the placement of the text object with typical 2D constraints and parametric dimensions between the rectangular boundary and other edges or features on your part.

After the text sketch is positioned on the part, you can extrude it.

5 Use AMEXTRUDE to extrude the text sketch.

Browser Right-click the text sketch icon and choose Extrude.

In the Extrusion dialog box, specify:

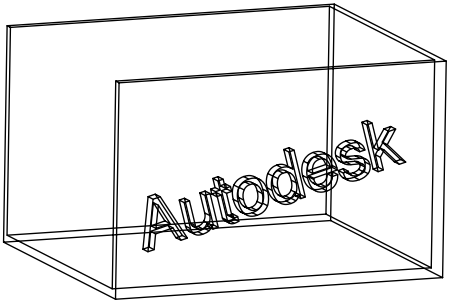
Operation: Join
Distance: *Enter* **.5**
Termination: Type: Blind

Choose OK.

6 Use 3DORBIT to rotate your part so you can see the emboss feature.

Toolbutton

Your part should look like this.



Editing Emboss Features

You can edit the text in an emboss feature using the Text Sketch dialog box before the text sketch is consumed. After a text sketch is consumed by a feature, you can edit the feature dimensions or the sketch font and style.

Creating Loft Features

You create loft features by defining a series of cross sections through which the feature is blended. Lofts may be linear or cubic. Both types can be created with existing part faces as the start and end sections.

Creating Linear Lofts

A linear loft is a feature created by a linear transition between two planar sections.

First, activate the next part in your drawing.

To activate a part

1 Make LOFT1_1 visible.

Browser Right-click LOFT1_1 and choose Visible.

NOTE Because LOFT1_1 does not contain any features, you cannot use the toolbutton, menu, or command methods to make it visible.

---

2 Activate LOFT1_1.

Browser Right-click LOFT1_1 and choose Activate Part

3 Make EXTRUDE_1 invisible.

Browser Right-click EXTRUDE_1 and choose Visible.

4 In the Desktop Visibility dialog box, select the Assembly tab.

5 Choose Select and continue on the command line.

Select assembly objects to hide: *Select* EXTRUDE_1
Select assembly objects to hide: *Press* ENTER

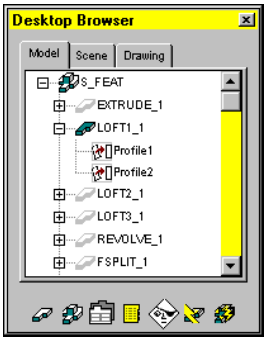
6 Choose OK to exit the dialog box.

If you choose the Browser method, the dialog box is not displayed.

Next, create the lofted feature.

To create a linear loft

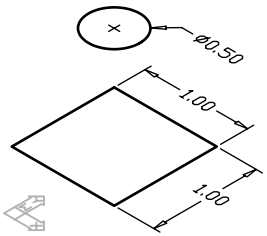
1 Expand LOFT1_1 in the Browser. Minimize EXTRUDE_1.



2 Zoom in to LOFT1_1.

Desktop Menu View ➤ Zoom ➤ All

The LOFT1 part contains two planar sections you use to create a linear lofted feature.



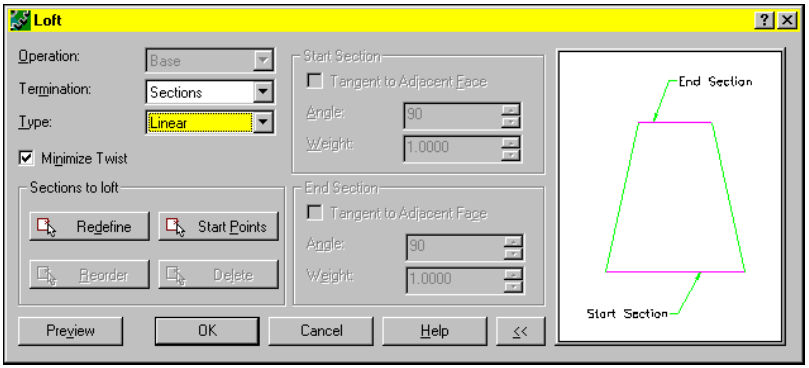
3 Create the loft feature, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Loft.

Select profiles or planar faces to loft: *Specify the bottom profile*
Select profiles or planar faces to loft: *Specify the top profile*
Select profiles or planar faces to loft or [Redefine sections]: *Press* ENTER

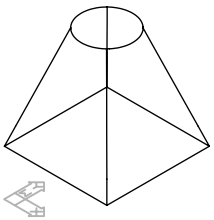
4 In the Loft dialog box, specify:

Type: Linear



5 Choose OK to exit the Loft dialog box.

Mechanical Desktop® calculates and displays the loft feature.



Save your file.

Next, you create a cubic loft blended through three planar sections.

Creating Cubic Lofts

A cubic loft is created by a gradual blend between two or more planar sections. Before the loft begins blending with the next section, you can control the tangency and the take-off angle at the start and end sections, and the distance the loft follows the tangent or angle options.

To create a cubic loft

1 Make LOFT2_1 visible.

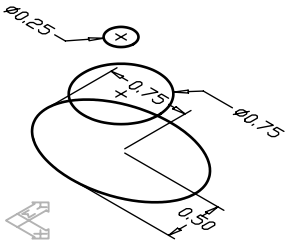
2 Activate LOFT2_1.

3 Make LOFT1_1 invisible.

4 Zoom in to LOFT2_1.

Browser Right-click LOFT2_1, and choose Zoom to.

LOFT2 contains three profiles defining the sections you use for the loft feature.



5 Create the loft, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Loft.

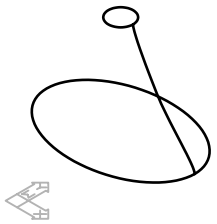
Select profiles or planar faces to loft: *Select the bottom profile*
Select profiles or planar faces to loft: *Select the middle profile*
Select profiles or planar faces to loft or [Redefine sections]: *Select the top profile*
Select profiles or planar faces to loft or [Redefine sections]: *Press* ENTER

6 In the Loft dialog box specify:

Type: Cubic

7 Choose OK to exit the Loft dialog box.

The loft is displayed with isolines because it is created from elliptical and circular sections. The default isoline setting displays the loft as in the following illustration.



For a better view of the loft, increase the number of isolines defining the feature.

To change the number of isolines

1 Modify the ISOLINES system variable.

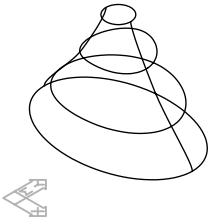
Command ISOLINES

New value for ISOLINES <4>: *Enter* **6**

2 Regenerate your drawing.

Desktop Menu View ➤ Regen

Mechanical Desktop regenerates the drawing and displays the loft using more isolines.



NOTE A higher value for ISOLINES increases the time it takes to recalculate a part. In general, keep ISOLINES at its default value (4).

3 Reset the value of ISOLINES to its default setting.

Command ISOLINES

New value for ISOLINES <6>: *Enter* **4**

4 Regenerate your drawing.

Desktop Menu View ➤ Regen

Save your file.

In the next exercise you create a cubic loft using an existing part face as the start section of the loft.

To create a cubic loft from an existing face

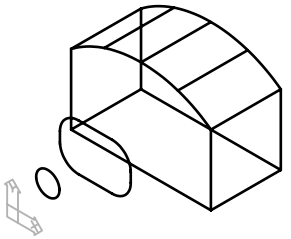
1 Make LOFT3_1 visible.

2 Activate LOFT3_1.

3 Make LOFT2_1 invisible.

4 Zoom in to LOFT3_1.

LOFT3 contains an existing extrusion and two profiles parametrically constrained to it.

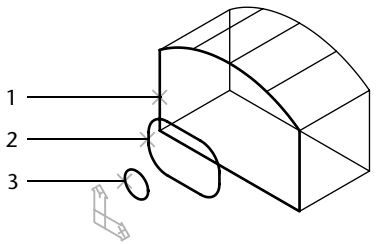


NOTE For clarity, the parametric dimensions are not shown.

5 Select the profiles to use for the cubic loft, following the prompts, and join the loft to the existing extrusion.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Loft.

Select profiles or planar faces to loft: *Select the front planar face (1)*
Enter an option [Accept/Next] <Accept>:
Highlight the front face and press ENTER
Select profiles or planar faces to loft: *Select the first profile (2)*
Select profiles or planar faces to loft or [Redefine sections]:
Select the second profile (3)
Select profiles or planar faces to loft or [Redefine sections]: *Press* ENTER

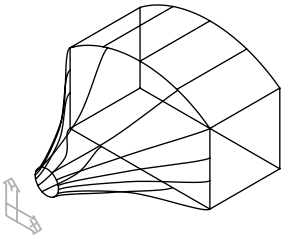


6 In the Loft dialog box, specify:

Operation: Join
Type: Cubic

Choose OK to exit the Loft dialog box.

Your drawing should look like this.



Save your file.

Editing Loft Features

You edit loft features the same way extruded features are edited—change the profiles or modify the loft feature itself.

Try editing the loft features you created in this section.

Creating Revolved Features

You create revolved features by revolving a closed profile about an axis. The axis may be a work axis or a part edge.

To create a revolved feature about a work axis

1 Make REVOLVE_1 visible.

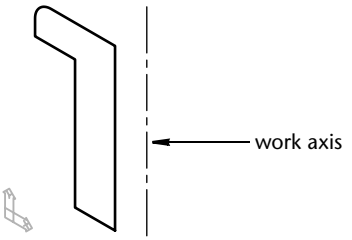
2 Activate REVOLVE_1.

3 Expand REVOLVE_1 and make Work Axis1 visible.

4 Make LOFT3_1 invisible.

5 Zoom in to REVOLVE_1.

REVOLVE_1 contains a profile parametrically constrained to a work axis.



NOTE For clarity, the parametric dimensions are not shown.

6 Create a revolved feature.

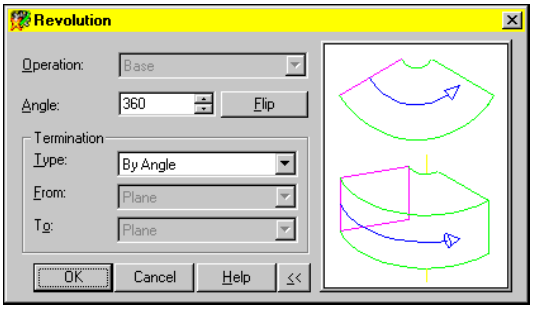
Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Revolve.

7 Respond to the prompt as follows:

Select revolution axis: *Specify the work axis*

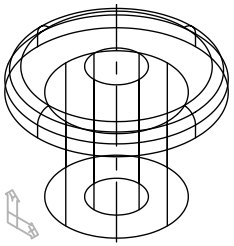
8 In the Revolution dialog box, specify:

Angle: *Enter* **360**
Termination: By Angle



Choose OK.

Mechanical Desktop calculates and displays the feature.



Save your file.

Editing Revolved Features

Edit a revolved feature by making changes to the profile, or by modifying the feature itself (like editing extruded and lofted features).

Try editing your revolved feature following the procedures for editing extruded features you learned earlier in this tutorial.

Creating Face Splits

Use face splits to split existing part faces. They can be created with

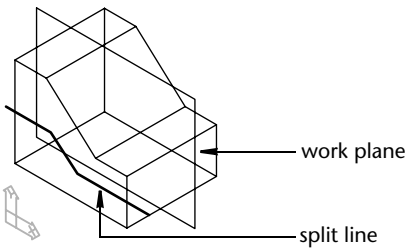
- An existing part face
- A work plane
- A split line

First, use one of the part's existing faces to split a face.

To split a face using an existing part face

1. Make FSPLIT_1 visible.
2. Activate FSPLIT_1.
3. Make REVOLVE_1 invisible.
4. Zoom in to FSPLIT_1.

   FSPLIT_1 contains a part, a work plane, and a split line.

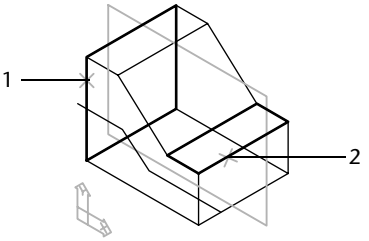


NOTE For clarity, the parametric dimensions are not shown.

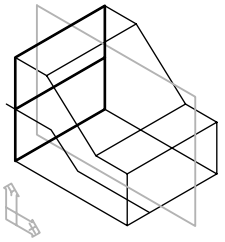
5. Create the face split, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Face Split.

Enter facesplit type [Planar/pRoject] <pRoject>: *Enter* **p**
Select faces to split or [All]: *Specify the left back face (1)*
Enter an option [Accept/Next] <Accept>:
Enter **n** *to flip to the back face or press* ENTER *to continue*
Select faces to split or [All/Remove]: *Press* ENTER
Select planar face or work plane for split: *Specify the top right face (2)*
Enter an option [Accept/Next] <Accept>:
Enter **n** *to flip to the top face or press* ENTER



Mechanical Desktop splits the back face into two faces.



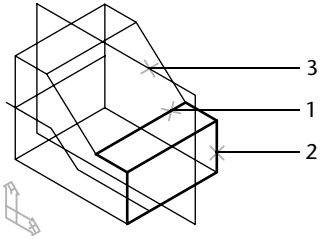
Next, split a face using a work plane.

To split a face using a work plane

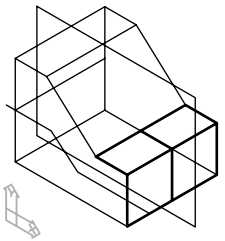
1 Create the face split, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Face Split.

Enter facesplit type [Planar/pRoject] <pRoject>: *Enter* **p**
Select faces to split or [All]: *Specify the top right face (1)*
Enter an option [Accept/Next] <Accept>:
Enter **n** *to flip to the top face or press* ENTER *to continue*
Select faces to split or [All/Remove]: *Specify the right front face (2)*
Enter an option [Accept/Next] <Accept>:
Enter **n** *to flip to the front face or press* ENTER *to continue*
Select faces to split [All/Remove]: *Press* ENTER
Select planar face or work plane for split: *Specify the work plane (3)*



Your drawing should look like this.



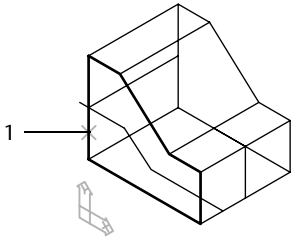
Now split the front face using the split line sketch.

To split a face using a split line

1. Make Work Plane2 invisible.
2. Create the face split, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Face Split.

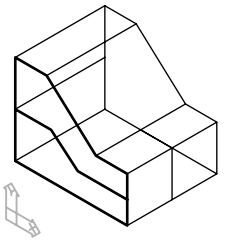
Enter facesplit type [Planar/pRoject] <pRoject>: *Press* ENTER
Select faces to split or [All]: *Specify the left front face (1)*
Enter an option [Accept/Next] <Accept>:
Enter **n** *to flip to the front face or press* ENTER *to continue*
Select faces to split or [All/Remove]: *Press* ENTER



If you use the Browser method, the prompts are not displayed.

When you choose the Project option, Mechanical Desktop automatically looks for an unconsumed split line. If more than one split line exists, you are prompted to select the split line for the face split.

Mechanical Desktop displays the new face split.



The Browser contains three face split features.

Save your file.

Editing Face Splits

Face splits created from an existing planar face can be edited by modifying the position of the face on the part. Face splits created from a work plane can be edited by modifying the dimensions controlling the location of the work plane. Face splits created from a split line can be modified by editing the parametric dimensions that control the split line.

Try editing the face splits you just completed in this exercise.

Creating Sweep Features

Sweep features can be either 2D or 3D. Both are created by sweeping a closed profile along a path.

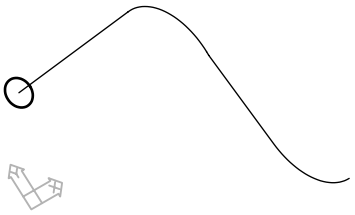
Creating 2D Sweep Features

You create a 2D sweep feature by sweeping a profile along a path that lies on a 2D plane. The feature may be the base feature of your part, or you can use Boolean operations to cut, intersect, split, or join the feature to your part.

To create a 2D sweep

1. Make SWEEP1_1 visible.
2. Activate SWEEP1_1.
3. Make FSPLIT_1 invisible.
4. Zoom in to SWEEP1_1.

   SWEEP1_1 contains a solved profile constrained to the start of a 2D path.

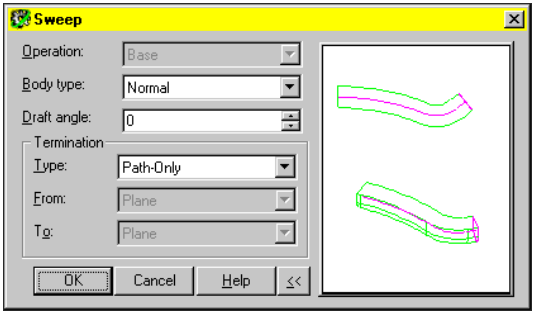


NOTE For clarity, the parametric dimensions and the work point are not shown.

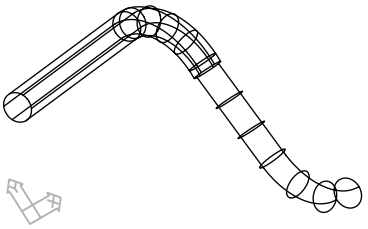
5. Create the 2D sweep.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Sweep.

6. In the Sweep dialog box, choose OK to accept the settings.



Your drawing should look like this.



NOTE Increase the value of ISOLINES for a more accurate display of the sweep.

Save your file.

Creating 3D Sweep Features

With Mechanical Desktop, you can also sweep profiles along a variety of 3D paths. Use these paths to create a feature swept along

- A helical path
- A spiral path
- A path defined by a 3D spline
- A path created from filleted 3D polylines and lines
- A path created from existing part edges

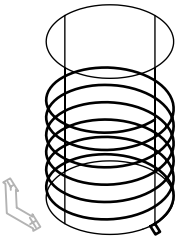
For more information about creating 3D paths, see chapter 6, "Creating Parametric Sketches."

First, create a 3D helical sweep.

To create a 3D helical sweep

1. Make SWEEP2_1 visible.
2. Activate SWEEP2_1.
3. Make SWEEP1_1 invisible.
4. Zoom in to SWEEP2_1.

SWEEP2_1 contains a cylinder and a helical path. A solved profile is constrained to the start of the path.



5 Create the 3D helical sweep.

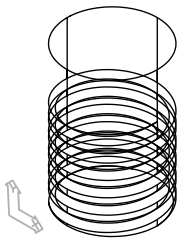
Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Sweep.

6 In the Sweep Feature dialog box, choose OK to accept the settings.

You can create a cut, join, intersection, or split feature. These options are available because there is a base feature in the part definition.

Choose OK to exit the dialog box.

Mechanical Desktop calculates the sweep and displays your part.



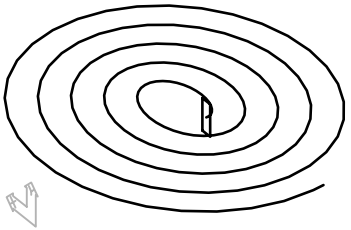
Save your file.

Next, create a spiral 3D sweep.

To create a spiral 3D sweep

1. Make SWEEP3_1 visible.
2. Activate SWEEP3_1.
3. Make SWEEP2_1 invisible.
4. Zoom in to SWEEP3_1.

   SWEEP3_1 contains a spiral helical path and a solved profile constrained to the start of the path. The spiral path is elliptical.

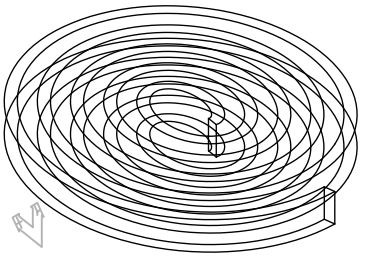


5. Create the 3D sweep.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Sweep.

6. In the Sweep Feature dialog box, choose OK to accept the settings.

   Your drawing should look like this.



Save your file.

Next, create a sweep using a 3D edge path.

To create a sweep from a 3D edge path

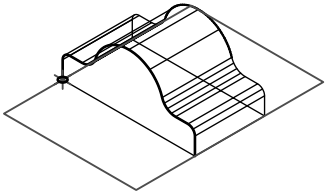
1 Make SWEEP4_1 visible.

2 Activate SWEEP4_1.

3 Make SWEEP3_1 invisible.

4 Zoom in to SWEEP4_1.

SWEEP4_1 contains a 3D edge path and a solved profile constrained to the start of the path.

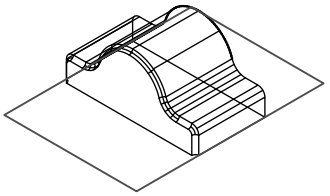


5 Create the sweep.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Sweep.

6 In the Sweep Feature dialog box, choose OK to accept the settings.

Your drawing should look like this.



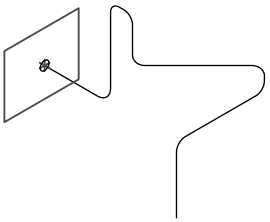
Save your file.

Next, sweep a feature along a path created from non-planar lines and arcs.

To create a sweep from a 3D pipe path

1. Make SWEEP5_1 visible.
2. Activate SWEEP5_1.
3. Make SWEEP4_1 invisible.
4. Zoom in to SWEEP5_1.

   SWEEP5_1 contains a 3D pipe path and a solved profile constrained to the start of the path.

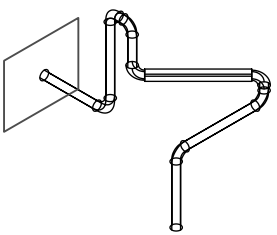


5. Create the sweep.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Sweep.

6. In the Sweep dialog box, choose OK to accept the settings.

   Your drawing should look like this.



Save your file.

Finally, create a swept feature using a path created from a 3D spline.

To create a sweep from a 3D spline path

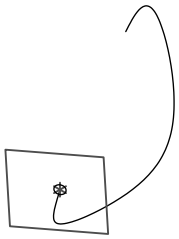
1 Make SWEEP6_1 visible.

2 Activate SWEEP6_1.

3 Make SWEEP5_1 invisible.

4 Zoom in to SWEEP6_1.

SWEEP6_1 contains a 3D spline path and a solved profile constrained to the start of the path.

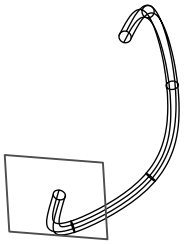


5 Create the sweep.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Sweep.

6 In the Sweep Feature dialog box, choose OK to accept the settings.

Your drawing should look like this.



Save your file.

Editing Sweep Features

As with all sketched features, sweep features can be edited by modifying the profile, the path, or the feature itself.

Try modifying the sweep features you just created.

Creating Bend Features

The bend feature is for bending flat or cylindrical parts.

To create a bend feature, you sketch a single line segment on your part and create an open profile to define the tangency location where the part transitions from its current shape to the final bent shape.

To bend an entire flat part, sketch the open profile to extend over the entire part. To bend only a portion of a flat part, sketch the open profile over only the portion you want to bend.

By choosing options and entering values in the Bend dialog box, you design a theoretical cylinder tangent to the open profile, about which the part bends. The bend feature is placed automatically in one operation.

In the next exercise, you create a bend feature on a portion of a flat part. Make the BEND_1 part visible. Then activate it and use ZOOM to position the part on your screen.

To create a bend feature on a flat part

1 Use LINE to sketch a line on one side of the plate, responding to the prompts.

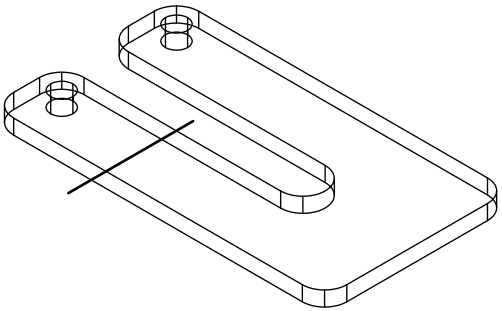
Context Menu In the graphics area, right click and choose 2D Sketching ➤ Line.

Specify first point: *Select the start point of the line*
Specify next point [or Undo]: *Select the end point of the line, and press* ENTER

2 Use AMPROFILE to create an open profile, responding to the prompt.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Select part edge to close the profile <open profile>: *Press* ENTER



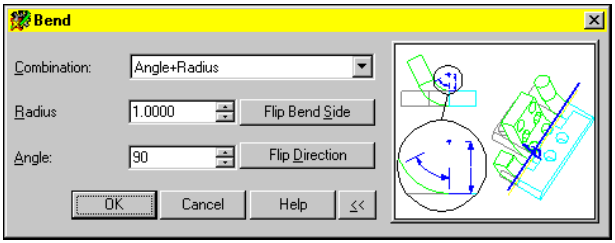
In the Browser, an icon for the open profile is displayed.

3 Use AMBEND to create the bend feature.

Context Menu In the graphics area, right-click and choose Sketched Work Features ➤ Bend.

4 In the Bend dialog box specify:

Combination: Angle+Radius
Radius: 1.0
Angle: 90
Flip Bend Side: *Verify that the direction arrow points toward the hole*
Flip Direction: *Verify that the arrow points up*



Choose OK.

Hide the hidden lines to see your part better. To display silhouette edges, you set the DISPSILH system variable to 1 first.

5 Change the setting for DISPSILH.

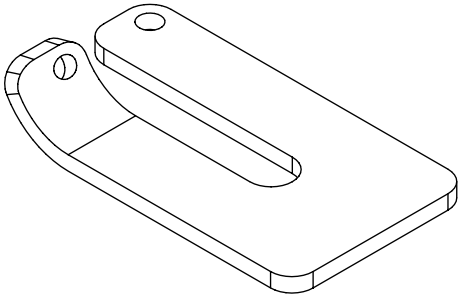
Command DISPSILH

New value for DISPSILH <0>: *Enter* **1**

6 Use HIDE to hide the hidden lines.

Desktop Menu View ➤ Hide

Your part should look like this.



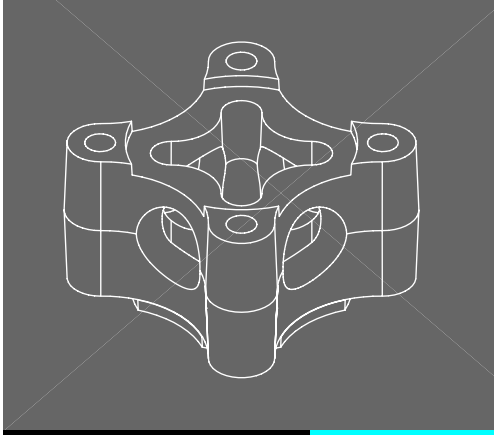
The bend is completed, and an icon for the bend feature is displayed in the Browser. Save your file.

Editing Bend Features

Use typical editing methods to edit a profile for a bend feature or to redefine the bend.

Try redefining the bend feature you just created.

Creating Work Features



In This Chapter

9

- Work planes
- Work axes
- Work points

In Autodesk® Mechanical Desktop®, work features are special construction features that you use to place geometry that would otherwise be very difficult to position parametrically.

By constraining sketched and placed features to a work feature, that is in turn constrained to your part, you can easily control their location by changing the position of the work feature.

This tutorial teaches you how to use work features to control the position of sketched features. You learn about each of these features as you work through the tutorial.

Key Terms

Term	Definition
nonparametric work plane	A work plane fixed in location with respect to a part. If the part geometry is parametrically changed, the work plane is unaffected.
parametrics	A solution method that uses the values of part parameters to determine the geometric configuration of the part.
parametric work plane	A work plane associated with and dependent on the edges, faces, planes, vertices, and axes of a part.
sketch plane	A temporary drawing surface that corresponds to a real plane on a feature. It is an infinite plane with both *X* and *Y* axes on which you sketch or place a feature.
work axis	A parametric construction line created along the centerline of a cylindrical feature, or sketched on the current sketch plane. A work axis can be used as the axis of revolution for a revolved or swept feature, an array of features, to place a work plane, and to locate new sketch geometry. It can be included in dimensions.
work feature	Planes, axes, and points used to place geometric features on an active part.
work plane	An infinite plane attached to a part. A work plane can be designated as a sketch plane and can be included in a constraint or dimension scheme. Work planes can be either parametric, or nonparametric.
work point	A parametric work feature used to position a hole, the center of an array, or any other point for which there is no other geometric reference.

Basic Concepts of Work Features

When you build a parametric part, you define how the part's features are associated. Changing one feature directly affects all the features related to it.

Work features are special construction features that help you define the relationships between the features on your part. They provide control when placing sketches and features. Any changes to the position of a work feature directly affect the placement of the sketches and features constrained to it.

You use work features to define

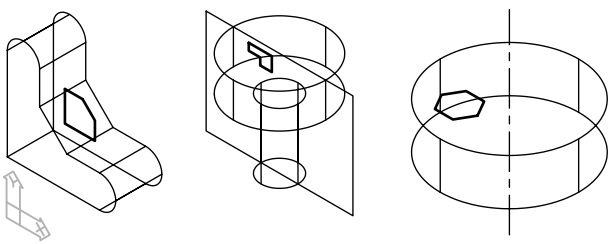
- A plane to place sketches and features
- A plane or edge to place parametric dimensions and constraints
- An axis or point of rotation for revolved, swept, and array features

There are three types of work features: work planes, work axes, and work points. In this tutorial, you learn the basics of creating and modifying each of these work features.

Open the file *w_feat.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing contains three simple parts.



Each part has a profile sketch associated with it. You create work features to control the behavior of each of the sketches.

Creating Work Planes

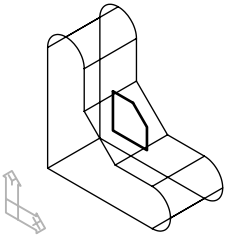
A work plane is an infinite plane that you attach to your part. It can be either parametric or nonparametric. A work plane can also be used to define a sketch plane for new geometry. To position a feature that does not lie on the same plane as your base feature, you define a new plane and then create the feature. If the plane is parametric, any changes to it affect the position of the feature.

Work planes are defined using two *modifiers*. The modifiers determine how the plane will be oriented. By selecting the right modifiers, you can create a work plane wherever you need a plane to place geometry.

Parametric work planes can be created by specifying edges, axes, or vertices, and defining whether the plane is normal, parallel, or tangent to selected geometry. Nonparametric work planes can be created on the current coordinate system (UCS), or on any of the three planes of the World Coordinate System (WCS).

For more information about creating work planes, see AMWORKPLN in the online Command Reference.

PART1_1 contains an extrusion with a profile constrained to its back face.



NOTE For clarity, the parametric dimensions are not shown.

In this tutorial, you use this profile to cut material from the part. By extruding the profile to a work plane, you can easily control the depth of the extrusion by changing the position of the plane.

First, you create a work plane through the midplane of the part and extrude the profile to it. Later, you edit the position of the work plane to modify the depth of the new extrusion.

Activate PART1_1 and use ZOOM to position it on your screen.

Browser Double-click PART1_1. Now right-click PART1_1 and choose Zoom to.

To create a work plane

1 Use AMWORKPLN to create a work plane through the midplane of PART1_1.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Plane.

2 In the Work Plane dialog box, specify:

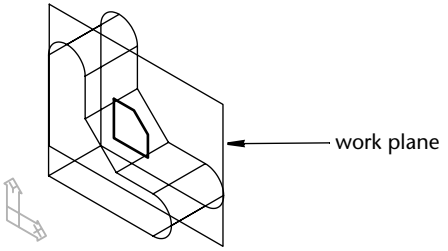
1st Modifier: Planar Parallel
2nd Modifier: Offset
Offset: *Enter* **.5**

Choose OK.

3 Continue on the command line.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Specify the front face
Enter an option [Next/Accept] <Accept>:
Enter **n** *to cycle to the front face or press* ENTER
Enter an option [Flip/Accept] <Accept>:
Enter **f** *to point direction arrows into the part*
Enter an option [Flip/Accept] <Accept>: *Press* ENTER
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <accept>: *Press* ENTER

A work plane now bisects the part.



Next, extrude the profile to the work plane.

To extrude a profile to a plane

1 Use AMEXTRUDE to extrude the profile.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

2 In the Extrusion dialog box, specify the following:

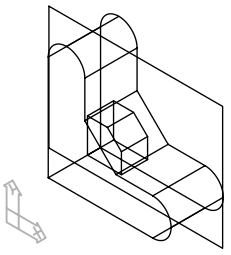
Operation: Cut
Termination: Plane

Choose OK to exit the dialog box.

3 Continue on the command line.

Select face or work plane: *Specify the work plane*

The profile is extruded to the work plane.



Now edit the location of the work plane to control the depth of the extrusion you just created.

Editing Work Planes

Because a nonparametric work plane is static, any features constrained to it are restricted to the original plane. If you change the position or orientation of your part, the features remain associated with the work plane and your part could fail to update.

Whenever possible, locate your features on parametric work planes. When you change the location of a parametric work plane, you change the position of any features created on it or constrained to it.

To modify the position of a work plane

1 Use AMEDITFEAT to reposition the work plane, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <selectFeature>: *Press* ENTER
Select feature: *Specify the work plane*
Select object: *Specify the 0.5 dimension*
Enter dimension value <.5>: *Enter* **.15**
Select object: *Press* ENTER

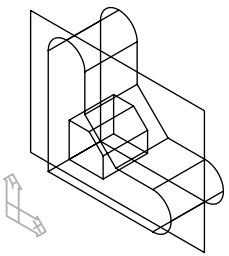
2 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Enter an option [active Part/aLl parts] <active Part>: *Press* ENTER

If you use the Browser, the prompt is not displayed.

Your part should look like this.

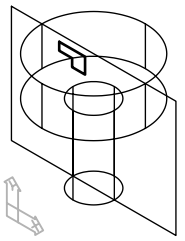


Save your file.

Creating Work Axes

A work axis is a parametric construction line used as the axis of revolution for a revolved or swept feature, or an array of features; it is also used to place a work plane, and to locate new sketch geometry. You can create a work axis through the center of a cylindrical edge, or draw it on the current sketch plane by specifying any two points.

PART2 contains a simple revolved feature, a work plane, and a partially constrained profile. You create a work axis through the center of the part. Then you constrain the profile to the work axis so you can cut material from the base feature.



NOTE For clarity, the parametric dimensions are not shown.

Activate PART2 and use ZOOM to position it on your screen.

Browser Double-click PART2_1. Now right-click PART2_1 and choose Zoom to.

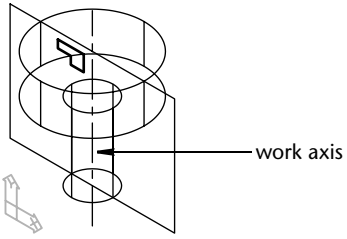
To create a work axis

1 Use AMWORKAXIS to create a work axis, responding to the prompt.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Axis.

Select cylinder, cone or torus [Sketch]: *Select a cylindrical edge*

Because the work axis is created through the center of the part, no constraints are necessary.



Next, constrain the profile to the new work axis and create a revolved feature from it.

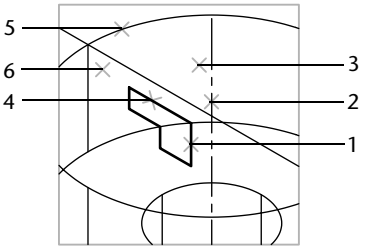
Depending on your drawing, your default dimension values may differ from those in this exercise.

To constrain and revolve a profile

1 Use AMPARDIM to constrain the profile to the work axis. Add two dimensions, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify the right edge of the profile (1)*
Select second object or place dimension: *Specify the work axis (2)*
Specify dimension placement: *Place the dimension (3)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.8324>: *Enter* **h** *to force a horizontal dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.1671>: *Enter* **0**
Solved underconstrained sketch requiring 1 dimensions or constraints.
Select first object: *Specify the top edge of the profile (4)*
Select second object or place dimension: *Specify the top edge of the part (5)*
Specify dimension placement: *Place the vertical dimension (6)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.1333>: *Enter* **0**
Solved fully constrained sketch.
Select first object: *Press* ENTER



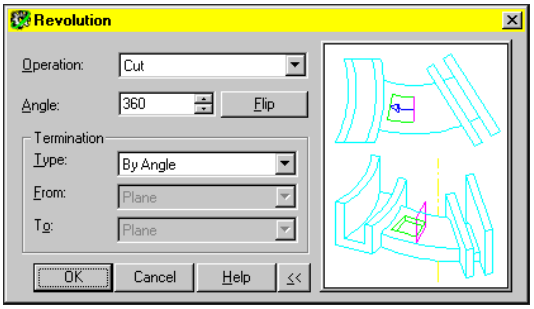
2 Use AMREVOLVE to revolve a feature from the profile, responding to the prompt.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Revolve.

Select revolution axis: *Select the work axis*

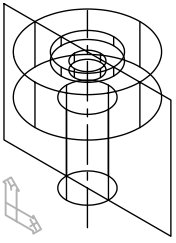
3 In the Revolution dialog box, specify:

Operation: Cut
Angle: *Enter* **360**
Termination: By Angle



Choose OK to exit the dialog box.

Your drawing should look like this.



Save your file.

Editing Work Axes

Work axes are parametric, so any changes to the parameters controlling a work axis affect the location of features constrained to it.

In this exercise, the work axis was created through the center of a cylindrical object and cannot be repositioned. But by changing one of the dimensions that constrains the profile to the axis, the revolved feature changes.

To modify the revolved feature, you change the horizontal dimension constraining the profile to the work axis. In this exercise, because the value of the dimension is 0, modifying it forces the profile in the wrong direction. To relocate the profile correctly, erase the dimension, move the profile slightly, and then add a new horizontal dimension.

To reposition a profile constrained to a work axis

1 Edit the revolved feature with the Browser.

Browser Right-click Revolution Angle1 and choose Edit Sketch.

2 Use ERASE to erase the 0.00 dimension constraining the profile to the work axis.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Erase.

3 Use MOVE to move the profile and its dimensions, following the prompts.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Move.

Select objects: *Enter* **w**
Specify first corner: *Specify a point above and left of the 0.15 dimension*
Specify opposite corner: *Specify a point below and right of the 0.30 dimension*
11 found
Select objects: *Press* ENTER
Specify base point or displacement: *Specify any point*
Specify second point of displacement or <use first point as displacement>:
Specify a point to the left of the base point

NOTE Press F8 to turn on orthographic mode before you specify the base and second points.

---

4 Use AMPARDIM to create a new parametric dimension, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

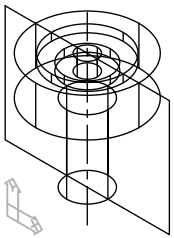
Select first object: *Specify the right edge of the profile*
Select second object or place dimension: *Specify the work axis*
Specify dimension placement: *Place the dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.4347>: *Verify that the dimension is horizontal, then enter* **.15**
Solved fully constrained sketch.
Select first object: *Press* ENTER

5 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Enter an option [active Part/aLl parts] <active Part>: *Press* ENTER

Your drawing should look like this.



Save your file.

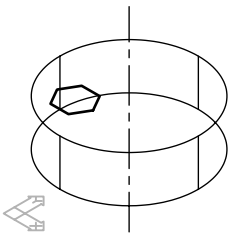
Creating Work Points

A work point is a parametric point for positioning features that cannot easily be located on a part. By constraining a feature to a work point and then constraining the work point to the part, you control the position of the feature.

Use work points to

- Position sketched features
- Create centers for polar arrays
- Place surface cut features
- Place holes when concentric cylindrical edges, or two planar edges, are not available

PART3 contains a simple cylindrical extrusion with a work axis at its center, and a sketch on its top face.



You place a work point on the sketch plane and profile the sketch. Then, you constrain the profile to the work point, and the work point to the work axis.

Activate PART3, and use ZOOM to position it on your screen.

Browser Double-click PART3_1. Then right-click PART3_1 and choose Zoom to.

To create and constrain a work point

1 Use AMWORKPT to create a work point, responding to the prompt.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Point.

Specify the location of the workpoint: *Specify a point near the center of the sketch*

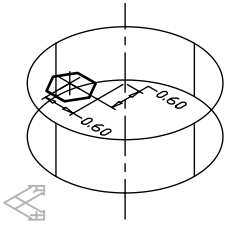
NOTE You may prefer to turn OSNAP off before you create and constrain the work point. Click the OSNAP button at the bottom of your screen.

---

2 Use AMPARDIM to constrain the work point to the work axis, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify the work point*
Select second object or place dimension: *Specify the work axis*
Specify dimension placement: *Place the dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.5890>: *Verify the dimension is horizontal and enter* **.6**
Solved underconstrained sketch requiring 1 dimensions or constraints.
Select first object: *Specify the work point*
Select second object or place dimension: *Specify the work axis*
Specify dimension placement: *Place the dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.6130>: *Verify the dimension is vertical and enter* **.6**
Solved fully constrained sketch.
Select first object: *Press* ENTER



Solve the sketch and constrain it to a work point.

Change to a top view of your part.

Desktop Menu View ➤ 3D Views ➤ Top

To solve a sketch and constrain it to a work point

1 Use AMPROFILE to solve the sketch, responding to the prompts.

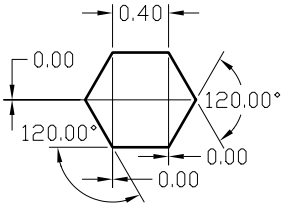
Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Profile.

Select objects for sketch: *Specify the polygon sketch*
Select objects for sketch: *Press* ENTER
Solved underconstrained sketch requiring 8 dimensions or constraints.

NOTE Although the polygon is a single object, you cannot use Single Profile to solve it because it was not the last object created.

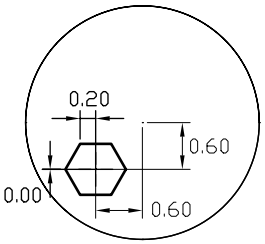
The profile requires eight constraints: six to solve it, and two to constrain it to the work point.

2 Zoom in to the profile and constrain it using the dimensions in the following illustration.



You could also use Equal Length constraints on the line segments to reduce the number of dimensions required.

3 Constrain the profile to the work point as in the following illustration.



NOTE For clarity, the dimensions of the profile are not shown.

The profile is now fully constrained. Next, you create an extrusion to cut material from the base feature.

To extrude a feature through a part

1 Change to an isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

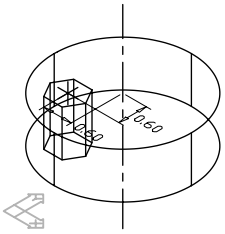
2 Use AMEXTRUDE to extrude the profile through the part.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Operation: Cut
Termination: Through

Choose OK.



The dimensions controlling the work point are still visible because the work point has not been consumed by a feature.

Save your file.

Editing Work Points

Next, to relocate the extrusion you change the dimensions constraining the work point to the work axis. The extrusion and the work point are parametrically associated; any change to the position of the work point causes the extrusion to move.

To edit a work point

1 Use AMMODDIM to modify the vertical sketch dimension controlling the work point, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Specify the vertical dimension*
New value for dimension <0.6000>: *Enter* **0**
Solved fully constrained sketch.

2 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Enter an option [active Part/aLl parts] <active Part>: *Press* ENTER

3 Use AMMODDIM to modify the horizontal sketch dimension controlling the work point, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Specify the horizontal dimension*
New value for dimension <0.6000>: *Enter* **.75**
Solved fully constrained sketch.

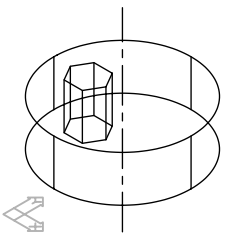
4 Update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Enter an option [active Part/aLl parts] <active Part>: *Press* ENTER

5 Turn off the visibility of the work point and its dimensions.

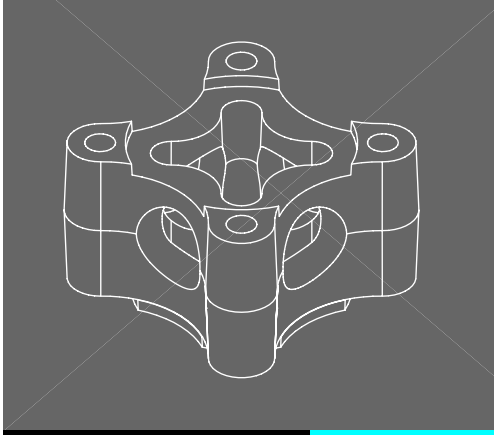
Browser Right-click WorkPoint1 and choose Visible.



Save your file.

You learn more about creating work features as you go through the rest of the tutorials in this book.

Creating Placed Features



In This Chapter

10

This tutorial introduces you to placed features, and builds on what you learned in previous tutorials. A placed feature is a well-defined common shape, such as a hole or a fillet. To create a placed feature, you only need to supply its dimensions. Autodesk® Mechanical Desktop® creates the feature for you.

In this lesson, you learn how to create and modify placed features.

- Holes
- Face drafts
- Fillets
- Chamfers
- Shells
- Surface cuts
- Patterns
- Copied features
- Combined features
- Part splits

Key Terms

Term	Definition
chamfer	A beveled surface between two faces.
combine feature	A parametric feature resulting from the union, subtraction, or intersection of a base part with a toolbody part.
draft angle	An angle applied parallel to the path of extruded, revolved, or swept surfaces or parts. A draft angle is used to allow easy withdrawal from a mold or easy insertion into a mated part.
face draft	A part face that has a draft angle applied to it. Used to create an angle on a face that will be needed when pulling a part out of a mold.
fillet	A curved transition from one part face or surface to another. The transition cuts off the outside edge or fills in the inside edge. The fillet can have a constant or variable radius.
hole	A geometric feature with a predefined shape: drilled, counterbore, or countersink.
pattern feature	A parameter-driven collection of duplicate features. You can create rectangular, polar, and axial patterns. If you change the original patterned feature, all the elements in the pattern change.
placed feature	A well-defined mechanical shape that does not require sketches, such as a hole, chamfer, or fillet. Placed features are constrained to the feature on which they are placed, and they are geometrically dependent.
shell	A Mechanical Desktop feature that cuts portions of the active part by offsetting its faces.
surface cut	A feature on a part created when a surface is joined to the solid. Where the surface cuts the part or protrudes, the part face assumes the curved shape of the surface. The surface, like other features, is parametric; both the surface and the part retain their parametric relationship whenever either is modified.

Basic Concepts of Placed Features

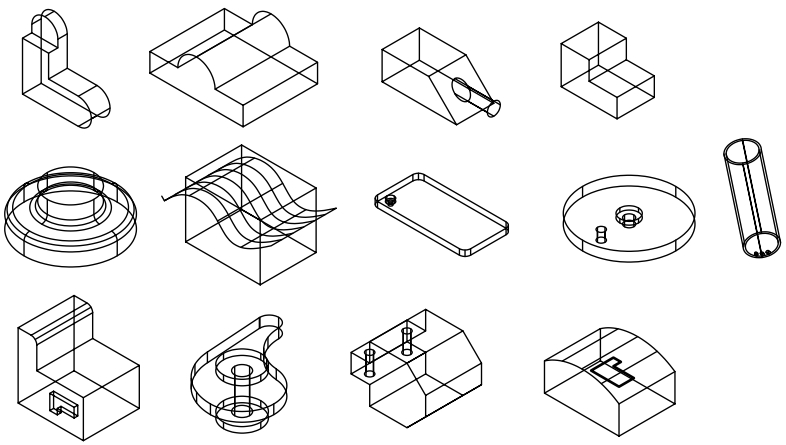
Placed features are well defined features that you don't need to sketch, such as fillets, holes, chamfers, face drafts, shells, surface cuts, patterns, combined features, and part splits.

You specify values for their parameters and then you position them on your part. To modify placed features, you simply change the parameters controlling them.

Open the file *p_feat.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing includes thirteen parts which contain the geometry you need to create the features in this tutorial. If you are interested in how the parts in this drawing were created, activate a part and use AMREPLAY.



Before you begin, expand the Browser hierarchy by clicking the plus sign in front of P_FEAT. Expand the hierarchy of the active part HOLE_1.

NOTE For clarity, the work features are not shown.

Creating Hole Features

You can create drilled, counterbore, and countersink hole features. Each may be assigned tapped hole information. Holes can extend through the part, stop at a defined plane, or stop at a defined depth. You can change a hole from one type to another at any time.

When you create a hole, you can use the Thread tab in the Hole dialog box to include threads. Threads can also be added to existing holes.

Instead of creating a custom hole, you can specify a standard hole from an external file. Standard holes can be tapped or untapped.

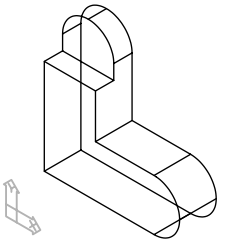
In this exercise, you create hole features first. Then you add thread data to the hole you created.

To create a hole feature

1 Activate HOLE_1 part, and zoom in to it.

Browser In the Browser, double-click HOLE_1. Now right-click HOLE_1 and choose Zoom to.

HOLE_1 is created from two extrusions.



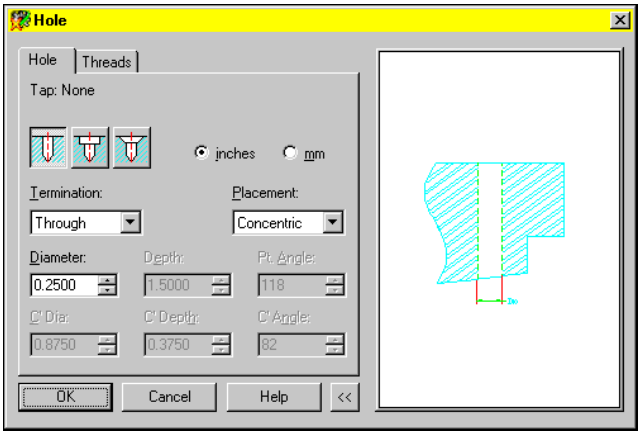
2 Use AMHOLE to create two drilled holes.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Hole.

NOTE Hold your cursor over an icon to see a tooltip that identifies the icon.

In the Hole dialog box, on the Hole tab, select the Drilled hole type icon, and specify:

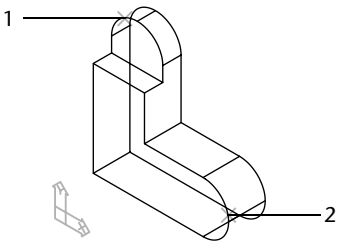
Termination: Through
Placement: Concentric
Diameter: *Enter* **.25**



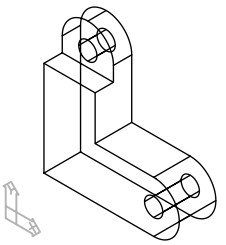
Choose OK to exit the dialog box.

3 Define the locations for the holes, responding to the prompts.

Select work plane or planar face [worldXy/worldYz/worldZx/Ucs]:
Specify a face (1)
Select concentric edge: *Specify an edge (1)*
Select work plane or planar face [worldXy/worldYz/worldZx/Ucs]:
Specify a face (2)
Select concentric edge: *Specify an edge (2)*
Select work plane or planar face [worldXy/worldYz/worldZx/Ucs]:
Press ENTER



Your drawing should look like this.



Next, add internal threads to the HOLE_1.

Creating Thread Features

You can create internal or external threads on cylindrical, conical, and elliptical shapes. You edit existing threads from within the Thread dialog box. As with holes, you can specify standard threads from an external file.

In the following exercise, you add an external thread to one of the cylindrical holes you created.

To create a thread feature

1 In the Browser, select the hole to add threads.

Browser Select Hole1.

2 Define the thread for Hole1

Context Menu In the graphics area, right-click and choose Placed Features ➤ Thread

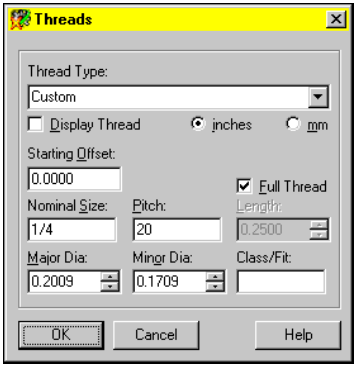
Respond to the prompts:

Select cylindrical/conical edge or face: *Select the circular edge of Hole1*
Enter an option [Next/Accept] <Accept>: *Press* ENTER

In the Threads dialog box, specify:

Thread Type: Custom
Full Thread: Select the check box
Major Dia: **0.2009**
Minor Dia: **0.1709**

Choose OK.



The thread feature is placed on Hole1 and an icon representing the external thread is added to the Browser hierarchy.

Next, you change one of the drilled holes to a counterbore hole, and change the minor diameter of the thread feature.

Editing Hole Features

You can change a hole feature from one type of hole to another by modifying the parameters defining the hole.

To edit a hole feature

1 Use AMEDITFEAT to change the second hole to a counterbore hole, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Specify Hole2
Enter an option [Next/Accept] <Accept>: *Press* ENTER

2 In the Hole dialog box, select the Counterbore icon, and specify:

Termination: Through
Dia: *Enter* **.2**
C'Dia: *Enter* **.375**
C'Depth: *Enter* **.15**

Choose OK to exit the dialog box.

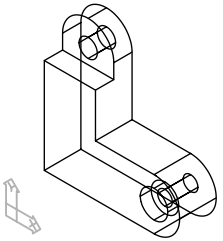
3 Continue on the command line.

Select object: *Press* ENTER

4 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Your part should look like this.



Save your file.

Editing Thread Features

You can redefine the size of an existing thread. If you need to change the thread type, it is necessary to delete the existing thread and create a new one.

To edit a thread feature

1 Use AMEDITFEAT to change and display the thread feature, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Press ENTER
Select feature: *Select the ExternalThread1 feature*

2 In the Threads dialog box, specify:

Thread Type: Custom
Display Thread: Select the check box.
Minor Diameter: **0.1805**

Choose OK.

3 Continue on the command line.

Select object: *Press* ENTER

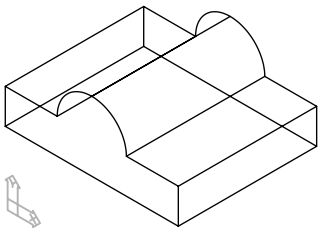
The thread feature is displayed, and reflects the new minor diameter value. Next, you learn how to create and edit face drafts.

Creating Face Drafts

Face drafts are used to add a small angle to one or more faces of a part; then the part can be easily extracted from a mold after it is manufactured.

Face drafts can be applied from a specified plane, an existing part face, or a part edge. You can also create a shadow draft from a circular face. If you are creating a face draft from a plane, the plane can be either an existing face, or a work plane offset from the part.

First, activate F-DRAFT_1 and zoom in on the part. Turn off the visibility of HOLE_1.



The part contains a simple extrusion.

To create a face draft from a plane

1 Use AMFACEDRAFT to create a face draft.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Face Draft.

In the Face Draft dialog box, specify:

Type: From Plane
Angle: *Enter* **10**

2 Choose Draft Plane and continue on the command line.

Select draft plane (planar face or work plane): *Specify the bottom face*
Draft direction [Accept/Flip] <Accept>: *Enter* **f** *to flip the direction arrow up*
Draft direction [Accept/Flip] <Accept>: *Press* ENTER

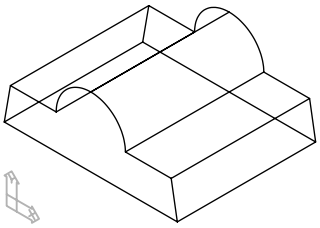
3 In the Face Draft dialog box, in Faces to Draft, press Add.

4 Continue on the command line.

Select faces to draft (ruled faces only): *Specify the left side face*
Select faces to draft (ruled faces only): *Specify the right side face*
Select faces to draft (ruled faces only): *Press* ENTER

NOTE Refer to the UCS icon to orient yourself when selecting faces.

5 Choose OK to exit the Face Draft dialog box. Draft is applied to the two faces.



A face draft can also be applied from an existing edge.

To create a face draft from a fixed edge

1 Create a face draft.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Face Draft.

In the Face Draft dialog box, specify:

Type: From Edge
Angle: *Enter* **10**

Choose Draft Plane.

2 Respond to the prompts as follows:

Select draft plane (planar face or work plane): *Specify the back face*
Enter an option [Next/Accept] <Accept>:
Enter **n** *to cycle to the back face, or press* ENTER
Draft direction [Flip/Next] <Accept>:
Enter **f** *to flip the arrow away from the part, or press* ENTER

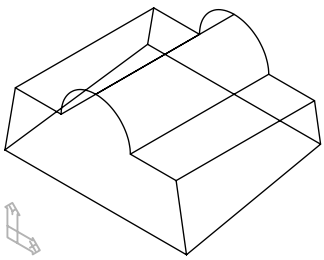
3 In the Face Draft dialog box, specify:

Faces to Draft: Add

4 Continue on the command line.

Select faces to draft (ruled faces only): *Specify the bottom face*
Select faces to draft (ruled faces only): *Press* ENTER
Select fixed edge: *Specify the bottom edge of the back face*
Select fixed edge: *Press* ENTER

5 In the Face Draft dialog box, choose OK to exit.



Draft is applied to the bottom face.

Next, create a shadow draft along the circular face of the part.

To create a shadow draft

1 Create the shadow draft.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Face Draft.

In the Face Draft dialog box, specify:

Type: Shadow
Angle: *Enter* **45**

Choose Draft Plane.

2 Respond to the prompts as follows:

Select draft plane (planar face or work plane): *Specify the top right face*
Enter an option [Next/Accept] <Accept>:
Enter **n** *to cycle to the top right face or press* ENTER
Draft direction [Flip/Accept] <Accept>:
Enter **f** *to flip the arrow away from the part or press* ENTER

3 In the Face Draft dialog box, specify:

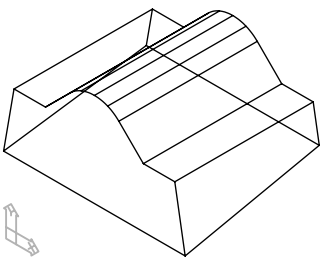
Faces to Draft: Add

4 Continue on the command line.

Select faces to draft (ruled faces only): *Specify the cylindrical face*
Enter an option [Next/Accept] <Accept>:
Enter **n** *to cycle to the cylindrical face or press* ENTER
Select faces to draft (ruled faces only): *Press* ENTER

5 In the Face Draft dialog box, choose OK to exit.

Your part should look like this.



The Browser contains three face draft icons nested below the FDRAFT_1 part definition.

Save your file.

Next, you modify one of the face drafts you just created.

Editing Face Drafts

To modify a face draft, you change the parameters that control it.

To edit a face draft

1 Use AMEDITFEAT to change FaceDraft2, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Specify the part
Enter an option [Next/Accept] <Accept>: ExtrusionBlind1: *Enter* **n**
Enter an option [Next/Accept] <Accept>: FaceDraft1: *Enter* **n**
Enter an option [Next/Accept] <Accept>: FaceDraft2: *Press* ENTER

2 In the Face Draft dialog box, change the Angle to **5**. Choose OK.

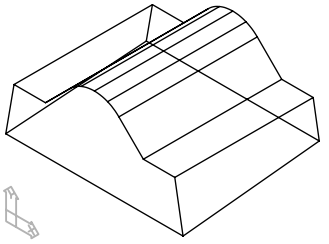
3 Continue on the command line.

Select object: *Press* ENTER

4 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Your part should look like this.



Save your file.

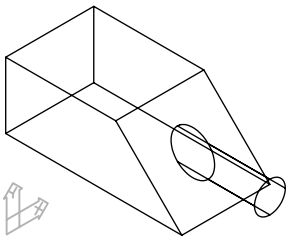
Creating Fillet Features

Fillet features can range from simple constant fillets to complex cubic fillets. Mechanical Desktop creates the following fillet types:

- Constant
- Fixed width
- Linear
- Cubic

A constant fillet has one radius defining it. A fixed width fillet is controlled by a chord length. Linear and cubic fillets have a radius at each vertex of the selected edges that you are filleting. A linear fillet has a straight transition from one vertex to the next. A cubic fillet has a continually changing radius from one vertex to the next.

Activate FILLET_1, and zoom to it. Turn off the visibility of F-DRAFT_1.

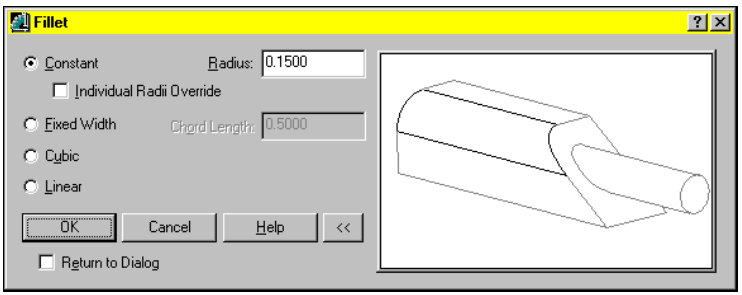


To create a constant radius fillet

1 Use AMFILLET to create a constant radius fillet.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Fillet.

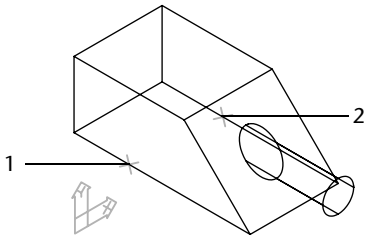
In the Fillet dialog box, choose Constant and specify a radius of **.15**.



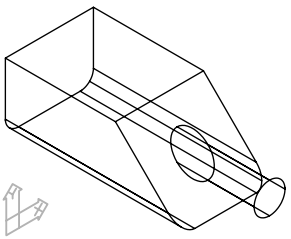
Choose OK.

2 Continue on the command line.

Select edges or faces to fillet: *Specify an edge (1)*
Select edges or faces to fillet: *Specify an edge (2), and press* ENTER



The fillets are applied to your part.



Next, create a fixed width fillet where the cylindrical extrusion meets the angled face.

To create a fixed width fillet

1 Use AMFILLET to create a fixed width fillet.

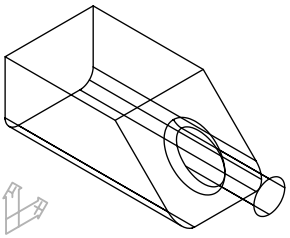
Context Menu In the graphics area, right-click and choose Placed Features ➤ Fillet.

In the Fillet dialog box, choose Fixed Width and specify a chord length of **.1**. Then choose OK.

2 Continue on the command line.

Select edges: *Specify the circular edge on the angled face*

Your part should look like this.



Create a linear fillet along the top left edge.

To create a linear fillet

1 Create a linear fillet.

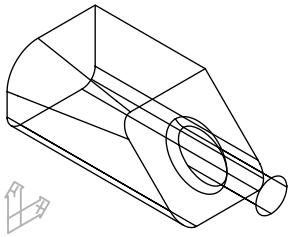
Context Menu In the graphics area, right-click and choose Placed Features ➤ Fillet.

In the Fillet dialog box, choose Linear, then choose OK.

2 Continue on the command line.

Select edge: *Specify the top left edge*
Select radius: *Specify the back radius symbol*
Enter radius <0.5000>: *Enter* **.35** *and press* ENTER
Select radius: *Specify the front radius symbol*
Enter radius <0.0000>: *Enter* **.15** *and press* ENTER
Select radius: *Press* ENTER

Your part should look like this.



You create a cubic fillet in the same way you create a linear fillet. Cubic and linear fillets differ because a cubic fillet is a blend on constantly changing radii from one vertex to the next.

To create a cubic fillet

1 Create a cubic fillet.

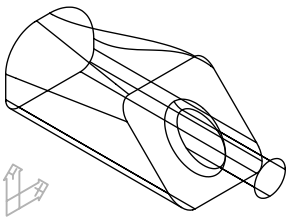
Context Menu In the graphics area, right-click and choose Placed Features ➤ Fillet.

In the Fillet dialog box, choose Cubic, then choose OK.

2 Continue on the command line.

Select edge: *Specify the top right edge at the back of the part*
Select radius or [Add vertex/Clear/Delete vertex]: *Specify the back radius symbol*
Enter radius <0.5000>: *Enter* **.5** *and press* ENTER
Select radius or [Add vertex/Clear/Delete vertex]: *Specify the front radius symbol*
Enter radius <0.0000>: *Enter* **.1** *and press* ENTER
Select radius or [Add vertex/Clear/Delete vertex]: *Press* ENTER

Your part should look like this.



The Desktop Browser contains four fillet icons nested under FILLET_1.

Save your file.

Editing Fillet Features

Like all placed features, fillets are modified by changing the parameters that control them.

To edit a fillet

1 Use AMEDITFEAT to modify the cubic fillet, responding to the prompts.

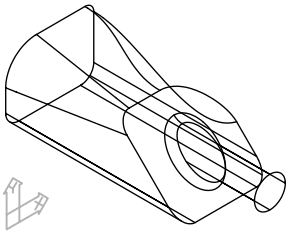
Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Specify Fillet4
Enter an option [Next/Accept] <Accept>:
Enter **n** *to cycle to Fillet4 or press* ENTER
Select object: *Specify the .1 radius*
Enter Radius <0.100000000>: *Enter* **.5** *and press* ENTER
Select object: *Specify the original .5 radius*
Enter Radius <0.500000000>: *Enter* **.1** *and press* ENTER
Select object: *Press* ENTER

2 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Your part should look like this.



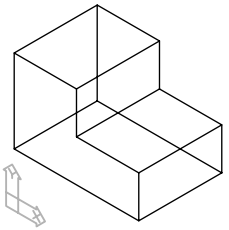
Save your file.

Delete some or all of the fillets you created in these procedures, and replace them with your own fillets to change the shape of your part.

Creating Chamfer Features

A chamfer feature is a bevelled face created between two existing faces on a part. Chamfers can be created with an equal distance, two different distances, or a distance and an angle. You can select an edge or a face to place a chamfer. If one or more of the edges of a face you want to chamfer have been altered, you need to use the edge selection method to place chamfers around that face.

First, activate CHAMFER_1 and zoom in on the part. Turn off the visibility of FILLET_1.



The part contains a simple extrusion.

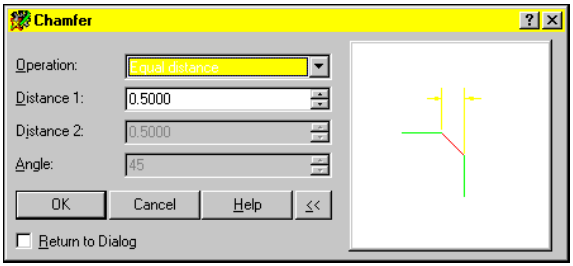
To create a chamfer defined by an equal distance

1 Use AMCHAMFER to create the chamfer.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Chamfer.

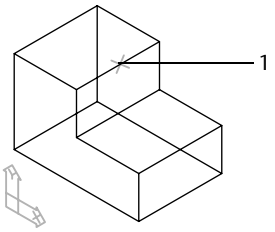
In the Chamfer dialog box, specify:

Operation: Equal Distance
Distance1: *Enter* **.5**

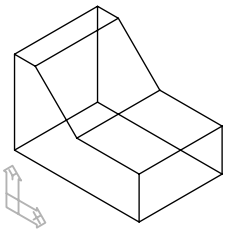


2 Choose OK and respond to the prompts as follows:

Select edges or faces to chamfer: *Specify an edge (1)*
Select edges or faces to chamfer <continue>: *Press* ENTER



Mechanical Desktop creates the chamfer along the edge you selected.



You can also create chamfers by specifying two different distances. After you select the edge, you specify a face for Distance 1, called the base distance. Distance 2 is applied to the other face.

To create a chamfer defined by two distances

1 Use AMCHAMFER to create the chamfer.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Chamfer.

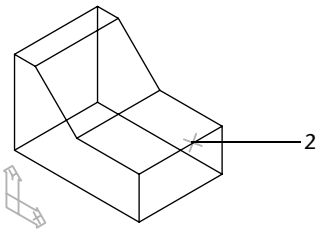
In the Chamfer dialog box, specify:

Operation: Two Distances
Distance1: *Enter* **.25**
Distance2: *Enter* **.15**

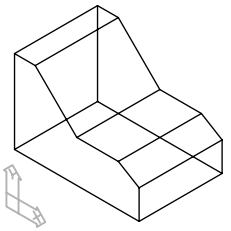
Choose OK.

2 Respond to the prompts as follows:

Select an edge or face to chamfer: *Specify the edge (2)*
Press <ENTER> to continue: *Press* ENTER
The specified face will be used for base distance.
Specify face for first chamfer distance (base) [Next/Accept] <Accept>:
Press ENTER



Mechanical Desktop calculates and displays the chamfer. Your drawing should look like this.



You can create a chamfer defined by a distance and an angle. You select an edge, and then specify the face for the angle. The distance is applied to the other face.

To create a chamfer defined by a distance and angle

1 Define the chamfer.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Chamfer.

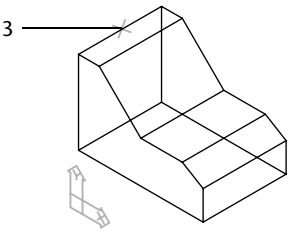
In the Chamfer dialog box, specify:

Operation: Distance and Angle
Distance1: *Enter* **1**
Angle: *Enter* **10**

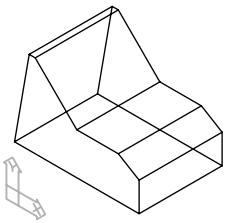
Choose OK.

2 Continue on the command line.

Select an edge or face to chamfer: *Specify the edge (3)*
Press <ENTER> to continue: *Press* ENTER
The specified face will be used for base distance.
Specify face for chamfer distance (base) [Next/Accept] <Accept>:
Press ENTER



Mechanical Desktop calculates and displays the chamfer.



If you need to place a chamfer on all sides of a face, you can select the face and place a chamfer on all of the edges in one operation. This works on faces where none of the edges to be chamfered have been altered.

To create a chamfer on all edges of a face

1 Define the chamfer.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Chamfer.

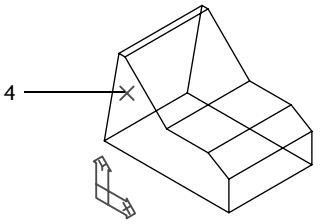
In the Chamfer dialog box, specify:

Operation: Equal Distance
Distance1: *Enter* **.04**

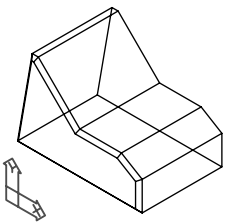
Choose OK.

2 Continue on the command line.

Select edges or faces to chamfer: *Select the face (4)*
Enter an option [Next/Accept] <Accept>: *Press* ENTER
Select edges or faces to chamfer <continue>: *Press* ENTER



A chamfer is placed on all edges of the face you selected.



Four chamfer icons are nested below the CHAMFER_1 part definition in the Browser.

Save your file.

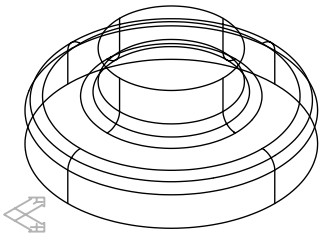
Editing Chamfer Features

As with all placed features, chamfers can be edited by selecting the feature, changing parameters, and updating the part. Try editing some of the chamfer features you created in this section.

Creating Shell Features

You use shell features to hollow parts that are used in a variety of industrial applications. For example, you shell parts to create molds, castings, containers, bottles, and cans.

Activate SHELL_1 and zoom in on it. Turn off the visibility of CHAMFER_1.



The part is constructed from two extrusions and one fillet feature.

Next, you shell the part, and then modify it to exclude the top and bottom faces.

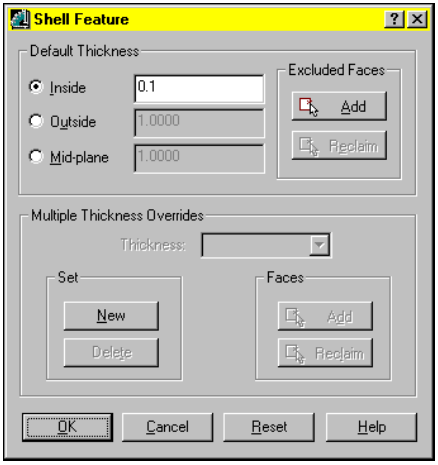
To create a shell feature

1 Use AMSHELL to create a shell.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Shell.

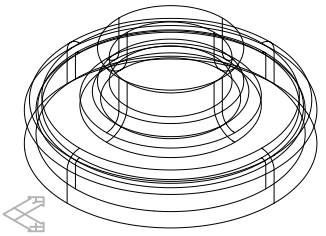
In the Shell Feature dialog box, specify:

Default Thickness: Inside: *Enter* **.1**



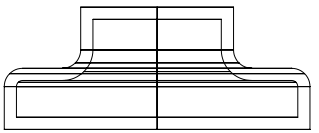
Choose OK to exit the dialog box.

Mechanical Desktop offsets all faces by the thickness you specified in the Shell Feature dialog box.



2 Change to a front view for a better view of the feature.

Desktop Menu View ➤ 3D Views ➤ Front



Save your file.

Next, you edit the feature to exclude the top and bottom faces from the shell.

Editing Shell Features

You modify shell features by changing the parameters that control them. Shells can also have multiple thickness overrides applied to individual faces. You learn to use multiple thickness overrides in chapter 14, "Creating Shells."

To edit a shell feature

1 Return to an isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

2 Use AMEDITFEAT to modify the shell feature, responding to the prompt.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Specify the shell feature

3 In the Shell Feature dialog box under Excluded Faces, choose Add.

4 Continue on the command line.

Select faces to exclude: *Specify the bottom face*
Enter an option [Accept/Next] <Accept>: *Enter* **n** *to cycle to the bottom face*
Enter an option [Accept/Next] <Accept>: *Press* ENTER
Select faces to exclude: *Specify the top face*
Enter an option [Accept/Next] <Accept>: *Enter* **n** *to cycle to the top face*
Enter an option [Accept/Next] <Accept>: *Press* ENTER
Select faces to exclude: *Press* ENTER

Choose OK to exit the Shell Feature dialog box.

5 Use AMUPDATE to update your part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Hide the hidden lines to see your part better. Because the part is cylindrical, to display silhouette edges, you set the DISPSILH system variable to 1 first.

6 Change the setting for DISPSILH.

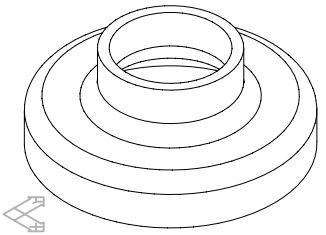
Command DISPSILH

New value for DISPSILH <0>: *Enter* **1**

7 Use HIDE to hide the hidden lines.

Desktop Menu View ➤ Hide

Your part should look like this.



8 Return to wireframe display.

Desktop Menu View ➤ Shade ➤ 3D Wireframe

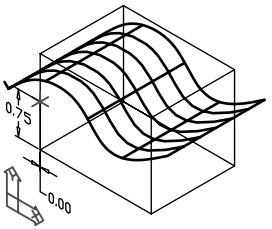
Save your file.

Creating Surface Cut Features

Surface cut features give you the flexibility of combining a parametric part and a surface. While the surface is not parametric, its position on the part is controlled by a work point which you can move parametrically.

Surface cut features may be used to add and remove material from a part.

Activate SURFCUT_1 and zoom in to it. Turn off the visibility of SHELL_1.



SURFCUT contains a simple rectangular extrusion, a work point, and a surface. The work point is constrained to the part. You use the work point to control the position of the surface cut.

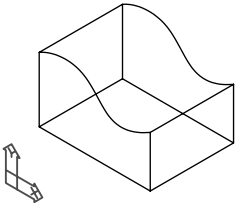
To create a surface cut

1 Use AMSURFCUT to create a surface cut, responding to the prompts.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Surface Cut.

Type=Cut
Select surface or [Type]: *Specify the surface*
Select work point: *Specify the work point*
Specify portion to remove: [Flip/Accept] <Accept>:
Enter **f** *to flip the arrow away from the part, or press* ENTER

The portion of the part above the surface is cut away, leaving the curved face of the surface.



The Browser contains a surface cut icon at the bottom of the feature hierarchy for SURFCUT_1.

Save your file.

Next, you edit the position of the surface to modify the surface cut feature.

Editing Surface Cut Features

You can modify surface cut features in one of two ways:

- Parametrically change its position.
- Manually change the shape of the surface.

In this section, you change the position of the feature by modifying the parametric dimensions controlling the work point associated with the surface.

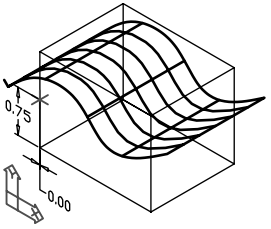
To reposition a surface cut feature

1 Use AMEDITFEAT to edit the feature, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Enter **c**
Select surfcut feature: *Specify the surface*

The surface is recovered.



In this state, you can modify the actual shape of the surface by editing its grips, or change the location of the work point that controls the position of the surface on the part.

2 Use AMMODDIM to change the vertical dimension controlling the work point, responding to the prompts.

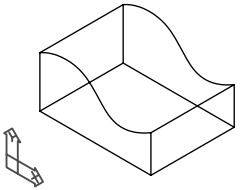
Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Specify* **.75**
New value for dimension <.75>: *Enter* **.5**
Select dimension to change: *Press* ENTER

3 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

The part is updated to reflect the new location for the surface cut feature.



4 Save your file.

Experiment with the surface by editing its control points. Use AMEDITFEAT to recover the surface. Then select a grip to activate it. When you move the grip to another location you will see the surface deform. Update your part to examine the effect of your changes.

Creating Pattern Features

A pattern is a collection of duplicate features. You can create patterns with rectangular, nonorthogonal rectangular, polar, and axial configurations, and patterns of other pattern features.

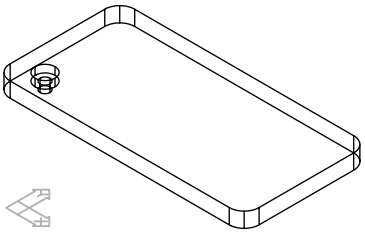
By default, a pattern feature uses the active sketch plane as the distribution plane for pattern instances.

While selecting a feature set for a pattern, you select each graphically dependent feature individually. You can select multiple independent features.

Single instances in a pattern can be made independent of an existing pattern feature. Once a feature is independent, it can be altered while its position remains intact.

In this tutorial, you create several different types of patterns, using both incremental and included spacing. In the polar pattern exercise, you make one instance independent and alter it.

Activate R-PATTERN_1, and zoom to it. Turn off the visibility of SURFCUT_1.



R-PATTERN contains a filleted plate and one counterbore hole. You create a rectangular pattern of the hole with incremental spacing and alignment to an edge.

To create a rectangular pattern

1 Use AMPATTERN to create a rectangular pattern, responding to the prompts.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Rectangle Pattern.

Select features to pattern: *Specify the hole*
Select features to pattern or [liSt/Remove] <Accept>: *Press* ENTER

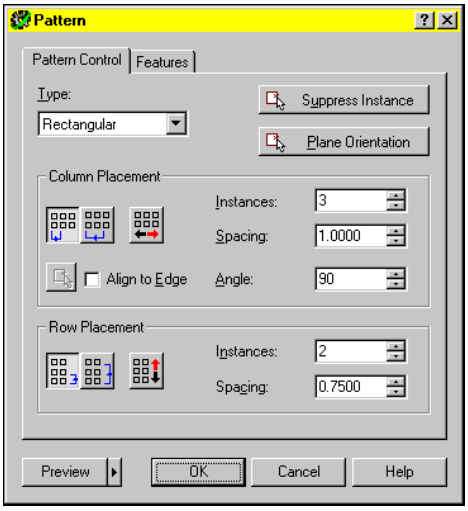
If you use multiple features to create a pattern, you select each one individually, regardless of feature dependencies.

2 In the Pattern dialog box, specify:

Type: Rectangular
Column Placement: *Choose* Incremental Spacing, *the leftmost button*
Row Placement: *Choose* Incremental Spacing, *the leftmost button*

NOTE Hold the cursor over an icon for a tooltip to identify the icon.

Enter the values shown for column and row instances and spacing.



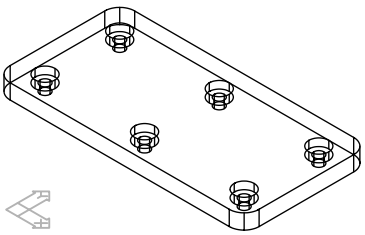
3 Choose Preview, and view your pattern on the screen.

At this point, you can redefine the pattern by changing your selections in the Pattern dialog box, and then preview the changes. Preview becomes unavailable once the parameters in the dialog box match the display on the screen.

Using the preview image, you can suppress instances of features in patterns.

4 Choose OK to create the pattern and exit the dialog box.

Your drawing should look like this.



Use R-PATTERN again to create a nonorthogonal rectangular pattern with included spacing and a value entered for the angle.

In the Browser, right-click the icon for the pattern you just created, and choose delete. Verify that the R-PATTERN part is activated.

To create a nonorthogonal rectangular pattern

1 Use AMPATTERN to create a nonorthogonal rectangular pattern, responding to the prompts.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Rectangular Pattern.

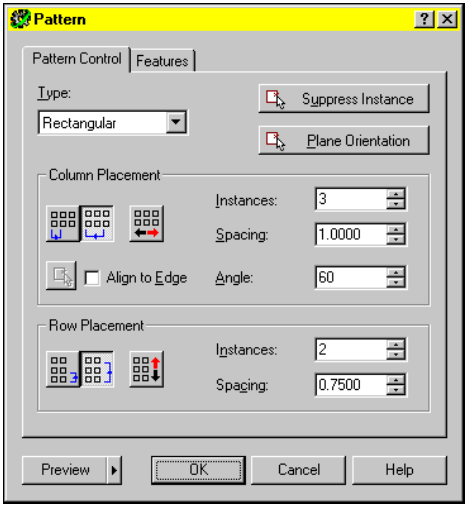
Select features to pattern: *Specify the hole*
Select features to pattern or [liSt/Remove] <Accept>: *Press* ENTER

2 In the Pattern dialog box, in Column Placement, select Included, the second button from the left. Specify:

Instances: *Enter* **3**
Angle: *Enter* **60**
Spacing: Angle: *Enter* **1**

In Row Placement, select Included, and specify:

Instances: *Enter* **2**
Spacing: *Enter* **.75**

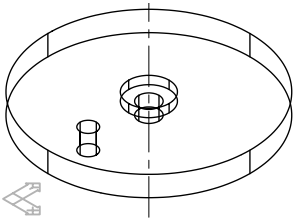


Choose OK.

The hole pattern is created at a 60-degree angle from the side of the part.

Next, create a full circle polar pattern using a work axis as the center and a specified number of instances. When you choose a different pattern type, the appropriate options are displayed in the Pattern dialog box.

Activate P-PATTERN_1 and zoom to the part.



The part is constructed with a circular plate and two holes.

To create a polar pattern

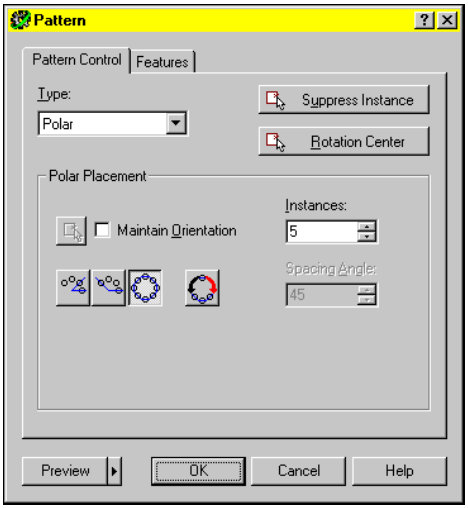
1 Use AMPATTERN create a polar pattern, responding to the prompts.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Polar Pattern.

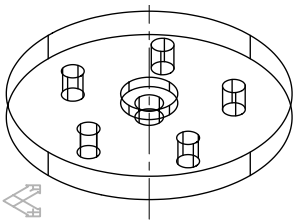
Select features to pattern: *Specify* Hole2
Enter an option [Next/Accept] <Accept>: *Press* ENTER
Select features to pattern or [liSt/Remove] <Accept>: *Press* ENTER
Valid selections: work point, work axis, linear edge, cylindrical edge/face
Select rotational center: *Specify the work axis*

2 In the Pattern dialog box, specify:

Polar Placement: *Choose* Full Circle
Instances: *Enter* **5**



Choose Preview and view the pattern. Then choose OK.



Next, make one instance of the pattern independent and then alter it.

To make a pattern instance independent

1 Select the pattern instance to make independent.

Browser Right-click PolarPattern, and choose Independent Instance.

Respond to the prompts.

Select feature pattern or array instance: *Select hole instance #4*

An independent hole based on a work point is copied from the selected hole instance. Dependent features are maintained and copied with the pattern instance.

Icons for the work point and independent Hole3 are displayed in the Browser.

The previous hole instance is suppressed. It can be reclaimed using the Pattern dialog box.

2 Use AMEDITFEAT to resize the independent pattern instance.

Browser Right-click the independent Hole4, and choose Edit.

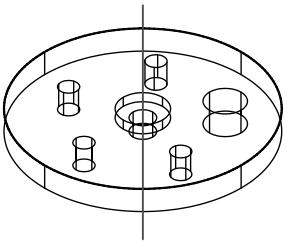
The Hole dialog box is displayed.

3 In the Hole dialog box, change the diameter to **.4**, and choose OK.

4 Use AMUPDATE to update the part, responding to the prompt.

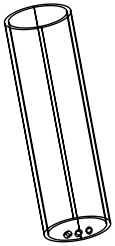
Context Menu In the graphics area, right-click and choose Update Part.

The Hole3 is resized, while it maintains its position in the pattern.



You can create axial patterns, and you can create a pattern from another pattern.

In the Browser, right-click A-PATTERN_1 and choose Activate Part. Right-click A-PATTERN_1 again, and choose Zoom to. Turn off the visibility of P-PATTERN_1.



A-PATTERN_1 contains a cylinder with a polar pattern of three holes. In this exercise, you use this polar pattern to create an axial pattern, specifying a work axis as the rotation center. You specify the number of instances, and incremental column and row placement.

After you create the axial pattern, you use it to create another polar pattern.

In the Browser, expand A-PATTERN_1.

To create an axial pattern

1 In the Browser, under A-PATTERN_1, right-click WorkAxis1 and choose Visible.

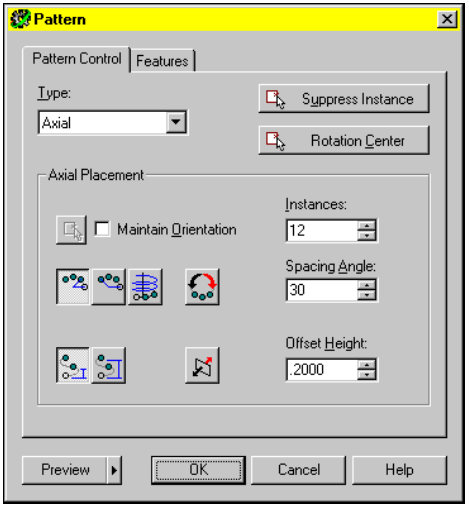
2 Use AMPATTERN to create an axial pattern, responding to the prompts.

Browser In the Browser, right-click Polar Pattern1 and choose Pattern ➤ Axial.

Valid selections: work point, work axis, linear edge, cylindrical edge/face
Select rotational center: *Specify the work axis*

In the Pattern dialog box, in Axial Placement, specify:

Instances: *Enter* **12**
Column Placement: *Select* Incremental Angle, *the button on the left*
Spacing Angle: *Enter* **30**
Row Placement: *Select* Incremental Offset, *the button on the left*
Offset Height: *Enter* **.2**



3 In the Pattern Dialog box, press Preview to view the pattern, then press OK.

The axial pattern is created on the surface of the cylinder. Hide the hidden lines to see your part better. Because the part is cylindrical, to display silhouette edges, you set the DISPSILH system variable to 1 first.

4 Change the setting for DISPSILH.

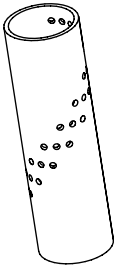
Command DISPSILH

New value for DISPSILH <0>: *Enter* **1**

5 Use HIDE to hide the hidden lines.

Desktop Menu View ➤ Hide

Your part should look like this.



6 Finish the part by using the new axial pattern to create another polar pattern.

Browser In the Browser, right-click Axial Pattern1 and choose Pattern ➤ Polar.

Select Rotational Center: *Select the work axis*

7 In the Pattern dialog box, specify:

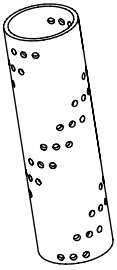
Polar Placement: *Select* Incremental Angle
Instances: *Enter* **2**
Spacing Angle: *Enter* **180**

Choose OK.

8 Use HIDE to hide the hidden lines.

Desktop Menu View ➤ Hide

Your finished part should look like this.



Editing Pattern Features

You edit pattern features in the Pattern dialog box. In the Pattern Control tab, you modify the instancing controls. In the Features tab, you redefine the features in the pattern. Once a pattern is created, you cannot change the pattern type.

When you delete a feature from a pattern set, you also remove other graphically dependent features that are children of that feature, such as fillets. If you want to add a feature to the set, a feature rollback is required.

A pattern is associative to the original feature that was patterned. When you modify the sketch of a patterned feature, you also modify the entire pattern.

Use the Pattern dialog box to preview and redefine the orientation of the distribution plane at any time. If you want to change the distribution plane to a different plane, a feature rollback is required.

Try editing the rectangular and polar patterns you created in this section.

Editing Array Features

Although you cannot create a new array, you can edit a previously-created array by editing the dimensions and instance constraints of the array using command line prompts. There is no dialog box available for editing arrays.

Pre-existing array features cannot be migrated to pattern features.

The following procedure is available only when you open a drawing file that contains a previously-created array.

To edit a previously-created array

9 Use AMEDITFEAT to edit a previously created array, responding to the prompts.

Context Menu Right-click the graphics area and choose Edit Features ➤ Edit.

Enter an option [Independent array instance/Sketch/surfCut/Toolbody/select Feature] <select Feature>: *Enter* **I**

Follow the command line prompts to edit the dimensions and instance constraints for your particular array.

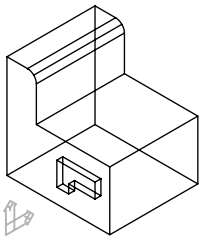
Creating Copied Features

You can copy a feature from any part, and place it on your active part on the current sketch plane. If the feature you select is on the active part, you can specify that the copy is independent. That way, you can modify either feature without affecting the other. If you do not specify that the copy is independent, or you copy a feature from an inactive part, any changes made to either the feature or the copy are reflected in both features.

Once copied, you can constrain the copied feature to the part by editing the sketch.

NOTE You cannot copy base features.

Activate CFEAT_1 and zoom in on the part. Turn off the visibility of P-PATTERN_1.



The part has a blind slot on the left front face. The current sketch plane lies on the right front face. You copy the feature to the current sketch plane and then constrain it to the part.

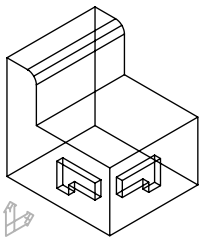
To copy a feature

1 Use AMCOPYFEAT to create a copy of the slot, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Copy.

Select feature to be copied (from any part): *Specify the blind extrusion*
Parameters=Independent
Specify location on the active part [Parameters]:
Specify a location on the current sketch plane
Parameters=Independent
Specify location on the active part [Parameters/Rotate/Flip]: *Enter* **f**
Parameters=Independent
Specify location on the active part [Parameters/Rotate/Flip]: *Enter* **r**
Parameters=Independent
Specify location on the active part [Parameters/Rotate/Flip]: *Enter* **r**
Parameters=Independent
Specify location on the active part [Parameters/Rotate/Flip]:
Respecify the location or press ENTER

Your drawing should look like this.



Next, you constrain the copied feature to the part by editing the feature's sketch. Three dimensions constrain the original feature to the part. You create three identical dimensions to constrain the new feature to the part.

To constrain a copied feature

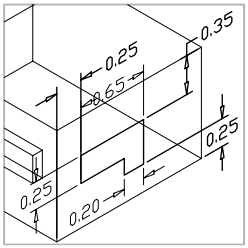
1 Use the Browser to edit the sketch.

Browser Right-click ExtrusionBlind3 and choose Edit Sketch.

2 Use AMPARDIM to add three parametric dimensions to constrain the sketch to the part.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

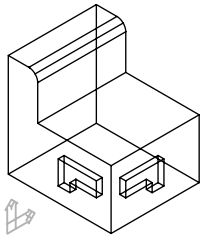
3 Place a 0.25 horizontal dimension, a 0.35 vertical dimension, and a 0.25 vertical dimension as illustrated below.



4 Use AMUPDATE to update your part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Enter an option [active Part/aLl parts] <active Part>: *Press* ENTER



Save your file.

Editing Copied Features

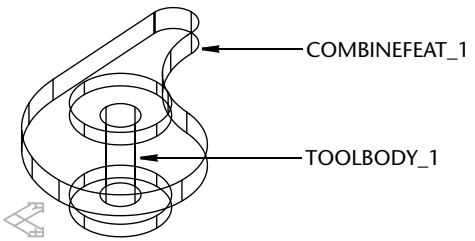
You can edit a copied feature by modifying the feature itself, or by modifying its location on the part. If the copy is dependent on the original feature, or if it was created from a feature on an inactive part, any changes to either feature are reflected in both features.

The copied feature you created is independent from the original feature. Try modifying the shape of the copied feature, using what you learned earlier in this tutorial about editing extrusions.

Creating Combined Features

You create combined features by combining two parts, using Boolean operations. The part that is combined is called the toolbody. You position the toolbody on the base part using assembly constraints, and then combine the parts.

Activate COMBINEFEAT_1 and zoom in so you can see it and TOOLBODY_1. Turn off the visibility of CFEAT_1.



The parts have already been constrained with assembly constraints. You learn to use assembly constraints in chapter 16, “Assembling Parts.”

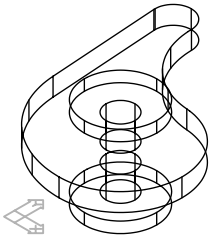
To create a combined feature

1 Use AMCOMBINE to create a combined feature, responding to the prompts.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Combine.

Enter a parametric Boolean operation [Cut/Intersect/Join] <Cut>: *Enter* **j**
Select a part (toolbody) to be joined: *Specify* TOOLBODY_1

The parts are combined into one part.



2 Look at the Browser. Expand the Combine1 icon nested under COMBINEFEAT_1. TOOLBODY_1 has become a combined feature and is no longer a separate part definition.

Save your file.

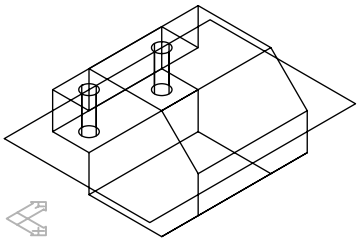
Editing Combined Features

Combined features can be modified by changing the assembly constraints controlling the base part and the toolbody, by editing the base part, or by making changes to the toolbody. You'll learn more about combining parts and editing toolbodies in chapter 17, "Combining Parts."

Creating Part Splits

You can split parts by creating a planar or a nonplanar split feature. A planar split uses a work plane, existing part face, or a split line. A nonplanar split uses a constrained sketch and a Boolean operation.

Activate P-SPLIT_1 and zoom in on the part. Turn off the visibility of COMBINEFEAT_1.



The part is a simple extrusion with two holes and a work plane located at the midplane of the part. You split the part into two distinct part definitions with the work plane.

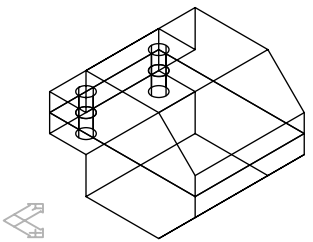
To create a part split

1 Use AMPARTSPLIT to split the part, responding to the prompts.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Part Split.

Select planar face, work plane, surface, or split line for split:
Specify the work plane
Define side for new part: [Accept/Flip] <Accept>: *Press* ENTER
Enter name of the new part <PART2>: *Press* ENTER

The part is split along the work plane and a new part definition is created.



NOTE For clarity, the work plane is not shown.

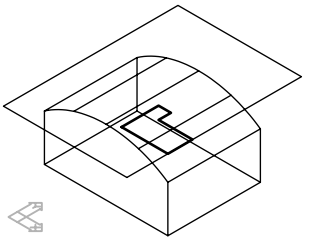
2 Expand PART2_1 in the Browser and compare its features with P-SPLIT_1.

Both parts contain a Part Split feature, two holes, and a work plane.

Save your file.

You can also create planar splits with an existing part face, or a split line constrained to the part. Next, create a nonplanar split.

Activate N-SPLIT_1 and zoom in on the part. Turn off the visibility of P-SPLIT_1 and PART2_1.



NOTE For clarity, the profile's dimensions are not shown.

You create a nonplanar split by extruding the profile into the part. A new part definition is created from the volume shared by the part and the extrusion.

To create a non-planar part split

1 Use AMEXTRUDE to create the part split.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion Feature dialog box, specify:

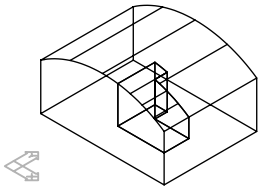
Operation: Split
Termination: Blind
Distance: *Enter* **.7**
Flip: *Make sure the direction arrow is flipped into the part*

Choose OK to exit the dialog box.

2 Continue on the command line.

Enter name of the new part <PART3>: *Press* ENTER

The part is split and a new part definition is created.



Save your file.

Editing Part Splits

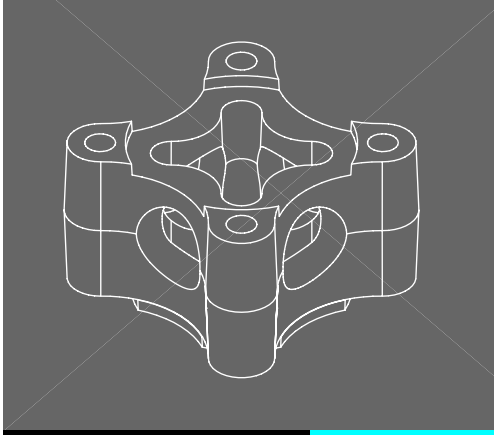
Parts created by a part split can be edited in the same way as the parts they were created from. The new parts contain identical work geometry, and if any feature was split, each part contains a version of that feature.

Nonplanar splits are used to create parts that fit together. Face drafts can be applied to the faces of both parts to make them fit together easier.

Try editing the sketched and placed features that make up the parts you have split in this section.

You are now ready to create a complex part.

Using Design Variables



In This Chapter

11

- Creating active part design variables
- Assigning variables to an active part
- Modifying design variables
- Creating global design variables

You can assign variables to the parametric dimensions that control a part. Variables can be assigned to the active part, or they can be global.

Active part design variables control only the features of the part they are assigned to. Global design variables control the features of any number of parts.

Autodesk® Mechanical Desktop® automatically reevaluates parts, and updates them when design variables have been modified.

Key Terms

Term	Definition
active part variable	A parametric variable used in the dimensions that control features of the active part.
global variable	A parametric variable that can be used by any number of parametric features and parts. Also used for single parts and to constrain parts.
helical sweep	A geometric feature defined by the volume from moving a profile along a 3D path about a work axis.
pitch	The measured distance parallel to the axis of a helical path, from one point on the path to the corresponding point on the adjacent revolution.
profile plane	A work plane at the start point of a helical path, placed normal to the start of the path or at the center of the axis/path.
start angle	The angle at which a helical path begins from the *X* axis of the active sketch plane.
table driven variable	A global or active part design variable controlled by values in a linked external spreadsheet.
taper angle	The angle where a helical sweep is tapered as it is created.

Basic Concepts of Design Variables

Parts and features are controlled by dimensions and other parameters that define their shapes. By creating design variables and assigning them to these parameters, you gain greater control over these values. There are two types of design variables:

- Global
- Active part

You use global design variables when you want to control parameters that belong to more than one part. When you want to control only a specific part, you use active part design variables.

You can create design variables using the Design Variables dialog box, or you can use the Equation Assistant dialog box to create design variables on the fly as you are creating a part.

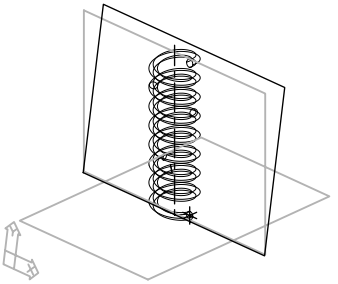
Design variables are also used in tables to control versions of a part. You learn to create these tables in chapter 15, "Creating Table Driven Parts."

This tutorial introduces design variables for controlling features. The tutorial drawing file contains a helical sweep. For clarity, the sweep is represented by four wires. The work features used to create the sweep are visible to help you understand how it was created. Before you begin the tutorial, turn off the visibility of the work features, and set the number of wires to a lower value; this increases the speed of recalculation and regeneration of the part.

Open the file *helix1.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing file contains a spring created from a helical sweep.



Three work planes are associated with the part. Two were used to create the sketched work axis for the sweep. The third, also called a *profile plane*, was created normal to the start of the path when it was defined. It was used to sketch the profile for the sweep. The profile is constrained to a work point at the beginning of the path.

Preparing The Drawing File

Before you begin, turn off the visibility of the work features. Leave the work axis visible because it will be helpful in keeping you oriented when you change the variables that control the 3D path.

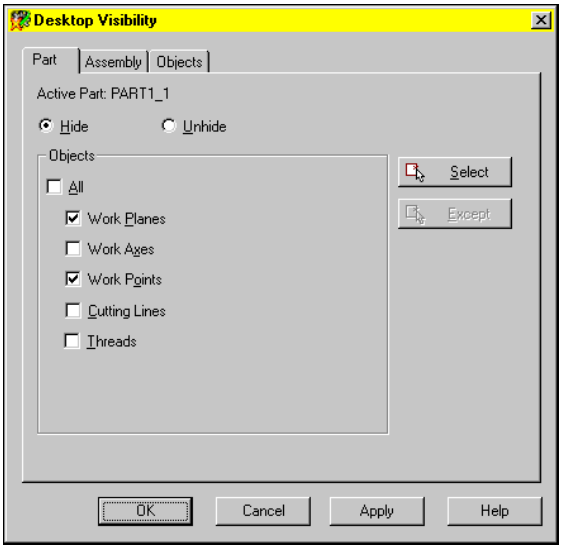
To hide a work feature

1 Use AMVISIBLE to turn off the visibility of the first work plane.

Desktop Menu Part ➤ Part Visibility

2 In the Desktop Visibility dialog box, with Hide turned on, choose Work Planes and Work Points.

Choose Apply.



Choose OK to exit the dialog box.

To speed up recalculations and regenerations of the helical sweep, set the ISOLINES variable to its default value. This will display the sweep using only one wire. Currently it is set to display the sweep as a helical tube.

To set isolines

1 Change the setting for ISOLINES, responding to the prompt.

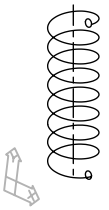
Command ISOLINES

New value for ISOLINES <8>: *Enter* **4**

2 Use REGEN to regenerate your drawing.

Desktop Menu View ➤ Regen

The helix should look like this.



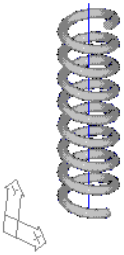
To see your model better, use the shade button on the Desktop View toolbar to toggle shading on and off. If you prefer, leave the shade option on.

To toggle shading of a part

1 Use SHADE to shade your part.

Desktop Menu View ➤ Shade ➤ Gouraud Shaded

Your part should now look like this.



NOTE Shading is turned off throughout this tutorial.

The Desktop View toolbar also contains commands to dynamically rotate your design and control views.

To dynamically rotate a part

1 Use 3DORBIT to rotate the view of your part.

Context Menu In the graphics area, right-click and choose 3D Orbit.

2 Select a point near the center of the part. This point acts as the central point for the rotation. Press the mouse button as you move your cursor around the screen. The part dynamically rotates as the cursor moves.

3 Release the mouse button when the display is to your liking.

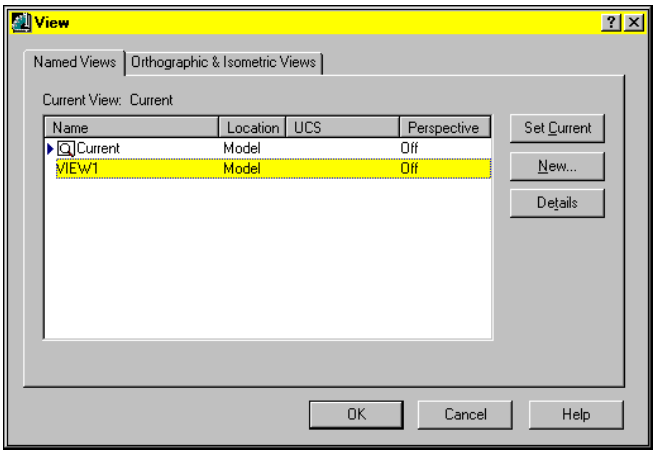
In the next procedure, you restore the view to its original display. The *helix1.dwg* file has one saved view, View1.

To restore a saved view

1 Use VIEW to restore the original drawing view.

Desktop Menu View ➤ Named Views

2 In the View dialog box, highlight View1, and choose Set Current.



Choose OK.

Your drawing is returned to the original view.

Next, you define active part design variables and then assign them to the existing helical part.

Using Design Variables

Design variables provide a tool for controlling dimensions, and using equations and relationships between dimensions. Changing one or more variables affects the entire part.

Design variables can be either active or global.

Active Part Design Variables

Active part design variables control only the part they are assigned to.

Global Design Variables

Global design variables allow you to use the same variables for multiple features across multiple parts. If you are designing multiple parts in the same file, you may use global design variables to control some or all of the parts with the same variables.

Creating Active Part Design Variables

The helical sweep is governed by the following parameters:

- Type of sweep
- Number of revolutions
- Pitch
- Height
- Shape
- Diameter of the sweep
- Taper angle
- Orientation
- Start angle
- Radius of the swept profile

In addition to the method used in the following exercises, you can create design variables in the Equation Assistant dialog box while you are in the modeling process. In the Equation Assistant dialog box, you right-click in the variables list area and choose New. A space for the new variable is provided at the end of the list, and your cursor is positioned in the Name column. There you enter a name for the new variable, and then you define it in the Equation column.

In this lesson, you create variables and parametric equations to control the number of revolutions, height, and diameter of the sweep. You also assign a variable to control the radius of the profile that is swept along the helical path. Because you are working with a single part, you create these variables as active part design variables.

To create an active part design variable

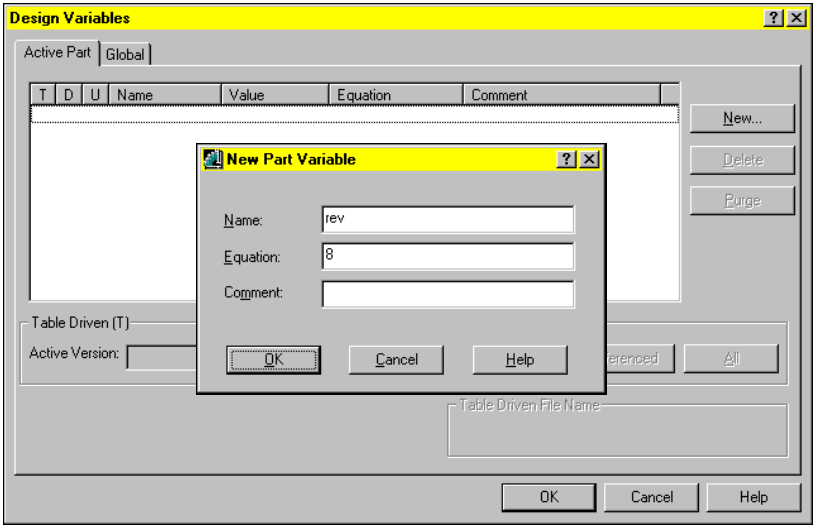
1 Use AMVARS to create a design variable.

Desktop Menu Part ➤ Design Variables

2 In the Design Variables dialog box, make sure the Active Part tab is selected and choose New.

3 In the New Part Variable dialog box, specify:

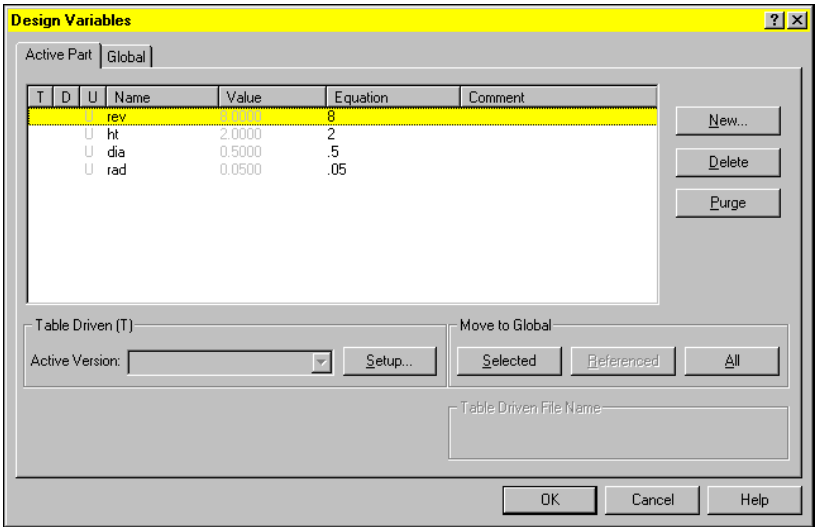
Name: *Enter* **rev**
Equation: *Enter* **8**



Press ENTER.

4 Repeat step 3 to enter the following variables:

Ht 2
Dia .5
Rad .05



Choose OK.

The next step is to edit the existing part by replacing its dimensions with the design variables you have just created.

Assigning Design Variables to Active Parts

Before the spring can be table driven, you need to assign the design variables you have defined. You edit the sweep feature and the profile used to create the sweep. You change the values controlling the feature with the design variables you have just created.

To edit a sweep feature

1 Use AMDIMDSP to set dimensions to display as equations.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Dimensions as Equations.

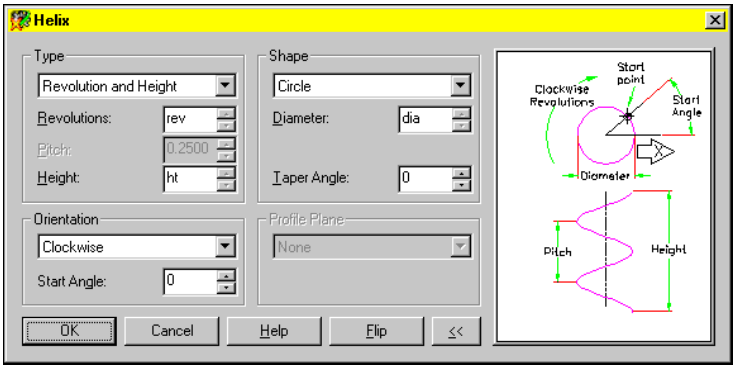
2 Use AMEDITFEAT to define the sweep feature to edit.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Highlight the helical sweep.

3 In the Helix dialog box, enter the following:

Revolutions: **rev**
Height: **ht**
Diameter: **dia**

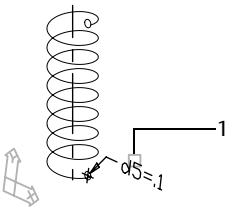


Choose OK. Then choose OK to exit the Sweep dialog box.

You have assigned design variables to the parameters controlling the sweep.

4 Assign the last variable to the radius of the profile, responding to the prompts as follows:

Select object: *Select the dimension (1)*
Enter dimension value <.1>: *Enter* **rad**
Select object: *Press* ENTER



NOTE For clarity, shading has been turned off in these illustrations. You may prefer to keep it on throughout the tutorial.

5 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

The spring is updated. The part changes because the value for the rad design variable you assigned is not the same as the original value used to create the sweep.

Modifying Design Variables

Design variables can be added and modified anytime during the design process. When the part is updated, changes to the design variables are automatically applied.

In this exercise, you add a design variable for a taper angle on the active part.

To add a design variable to an active part

1 Use AMVARS to add a design variable.

Desktop Menu Part ➤ Design Variables

2 In the Design Variables dialog box, with the Active Part tab selected, choose New.

3 In the New Part Variable dialog box, specify:

Name: **taper**
Equation: *Enter* **0**

Choose OK.

4 Choose OK to exit the Design Variables dialog box.

Next, you edit the sweep feature by adding the new variable to its design parameters.

To edit a sweep feature

1 Use AMEDITFEAT to add the new design variable to the sweep feature.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

2 Highlight the helical sweep.

3 In the Helix dialog box, specify the following:

Taper Angle: **taper**

Choose OK. Then choose OK to exit the Sweep dialog box.

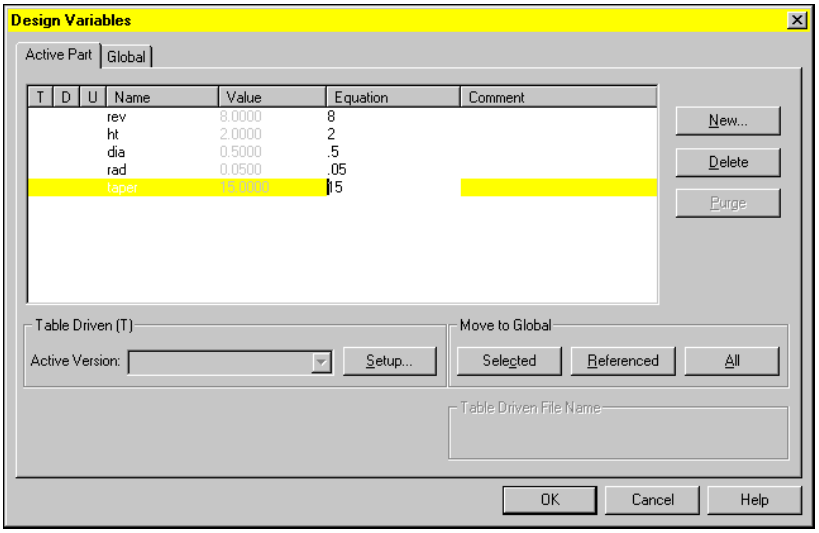
4 Press ENTER to end the command.

To modify a design variable

1 Use AMVARS to modify the design variable.

Desktop Menu Part ➤ Design Variables

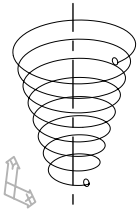
2 In the Design Variables dialog box, with the Active Part tab selected, highlight the taper variable. In the highlighted line, double-click the Equation field and enter **15**.



Choose OK to exit the Design Variables dialog box.

3 Use AMUPDATE to update the part.

Context Menu In the graphics area, right-click and choose Update Part.



Save your file.

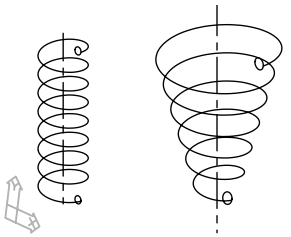
Working with Global Design Variables

You can assign global variables to more than one part to control similar features. In this lesson, you move some of the active part design variables to global variables and assign them to two parts.

Open the file *helix2.dwg* in the *desktop\tutorial* folder.

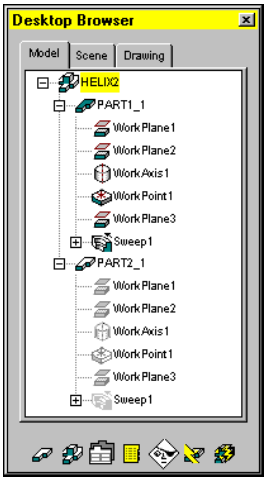
NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing contains two springs created from helical sweeps.



Both helical sweeps have active part design variables already assigned to them. In this lesson, you create global design variables to control identical features of each helix.

First, expand the Browser hierarchy by clicking the plus sign in front of HELIX2. Then click the plus sign in front of PART1_1 and PART2_1. The Browser should look like this. Notice PART1_1 is the active part.



Next, examine the active part design variables for both parts.

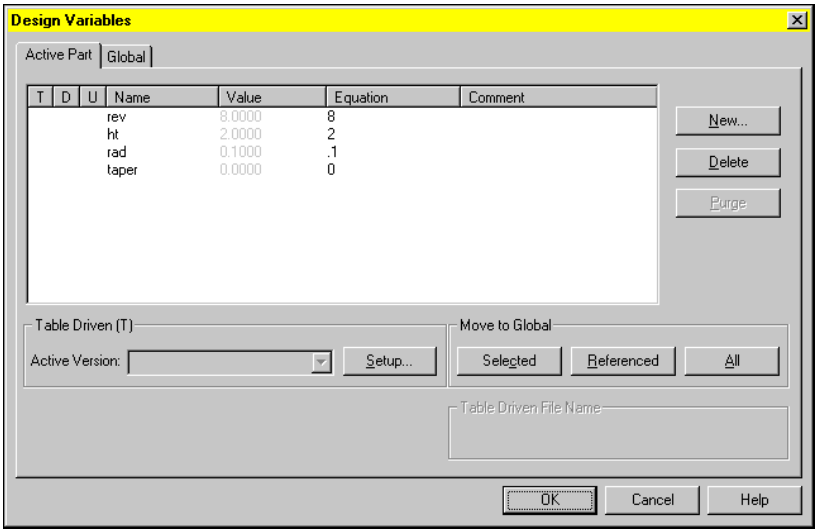
To examine a design variable for an active part

1 Use AMVARS to open the Design Variables dialog box.

Desktop Menu Part ➤ Design Variables

2 In the Design Variables dialog box, with the Active Part tab selected, examine the values for the design variables assigned to PART1_1.

You should see four variables controlling the number of revolutions, height, radius of the swept profile, and the taper angle of the active part. There is no variable defined for the diameter of the helical sweep.



Choose OK to exit the dialog box.

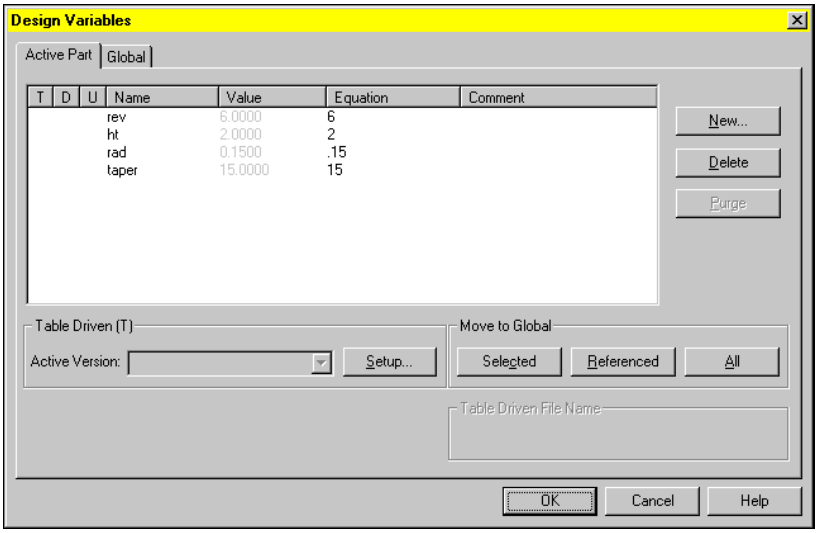
3 Use AMACTIVATE to activate PART2_1.

Browser In the Browser, right-click PART2_1 and choose Activate Part.

4 Examine the design variables assigned to PART2_1.

Desktop Menu Part ➤ Design Variables

5 In the Design Variables dialog box, make sure the Active Part tab is selected.



Choose OK.

Both parts have active part design variables controlling the same features. The variable controlling the height of both helical sweeps contains the same value.

Next, you move this active part design variable to a global design variable so that one variable controls both parts.

To move an active part variable to a global design variable

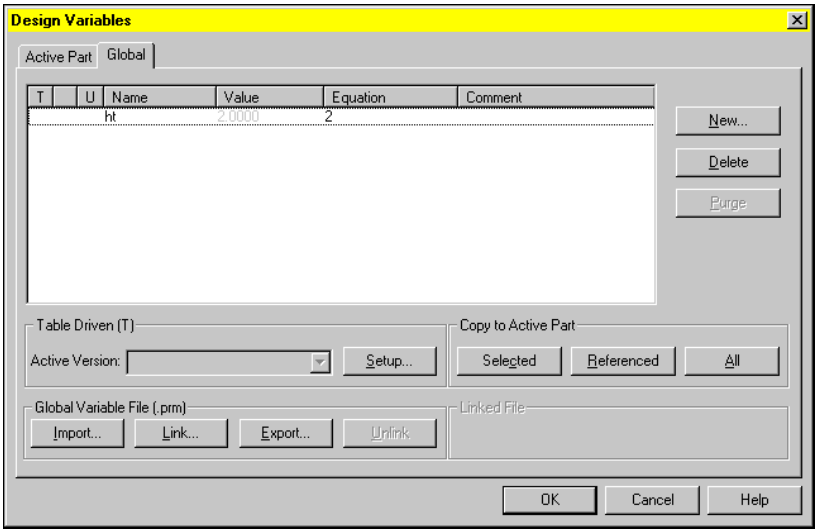
1 Open the Design Variables dialog box.

Desktop Menu Part ➤ Design Variables

2 In the Design Variables dialog box, with the Active Part tab selected, highlight the ht variable. Under Move to Global, choose Selected.

The variable is removed from the list of active part design variables.

3 Click the Global tab and examine the list of global variables.



Choose OK to exit the dialog box.

Mechanical Desktop re-evaluates the features of the part and updates the part. Because the value of the variable has not changed, the part does not change.

Although the ht variable for PART2_1 has been moved to global, the same variable for PART1_1 is still an active part design variable. Because one global variable will drive both parts, you remove the ht variable from the PART1_1 list of active part design variables.

To delete an active part design variable

1 Use AMACTIVATE to activate PART1_1.

Browser In the Browser, right-click PART1_1 and choose Activate Part.

2 Open the Design Variables dialog box.

Desktop Menu Part ➤ Design Variables

3 In the Design Variables dialog box, with the Active Part tab selected, highlight the ht variable and choose Delete.

The variable is removed from the list of active part design variables.

Choose OK to exit the dialog box.

PART1_1 is re-evaluated and updated. The global design variable is now controlling the height of both helical sweeps.

Next, you create a new global design variable to control the diameter of the springs and assign it to both parts. Then you modify the value of the global design variable controlling their height.

To create a global design variable

1 Open the Design Variables dialog box.

Desktop Menu Part ➤ Design Variables

2 In the Design Variables dialog box, select the Global tab and choose New.

3 In the New Part Variable dialog box, specify:

Name: *Enter* **dia**
Equation: *Enter* **.75**

Choose OK. The Global tab now contains two variables, ht and dia.

Choose OK to exit the dialog box.

The parts do not change because the variable has not yet been assigned to them.

To assign a global design variable to a part

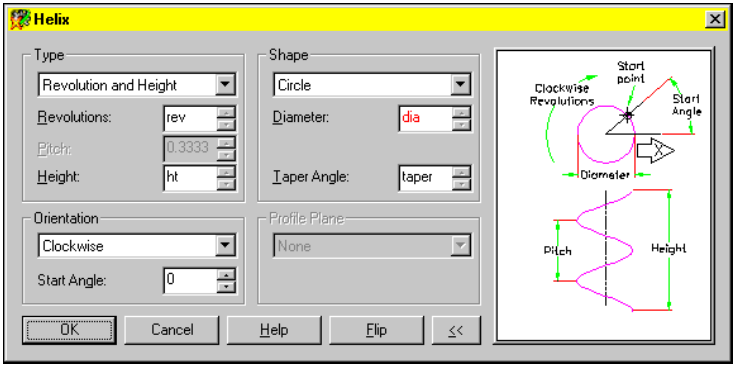
1 Edit the sweep feature for PART1_1:

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

2 Highlight the helical sweep.

3 In the Helix dialog box, specify the following:

Diameter: **dia**



Choose OK. Then choose OK to exit the Sweep dialog box.

4 Continue on the command line.

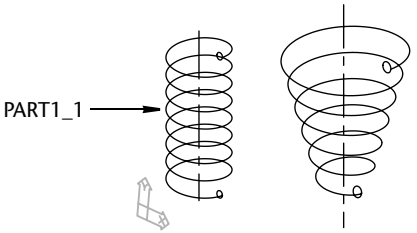
Select object: *Press* ENTER

5 Update the part.

Context Menu In the graphics area, right-click and choose Update Part.

Enter an option [active Part/Assembly/aLl parts/linKs] <active Part>:
Press ENTER

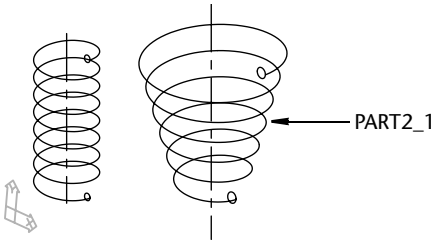
Mechanical Desktop updates PART1_1 using the new global design variable to control the diameter of the sweep.



6 Activate PART2_1.

7 Repeat steps 1 through 5 for PART2_1.

Your drawing should look like this.



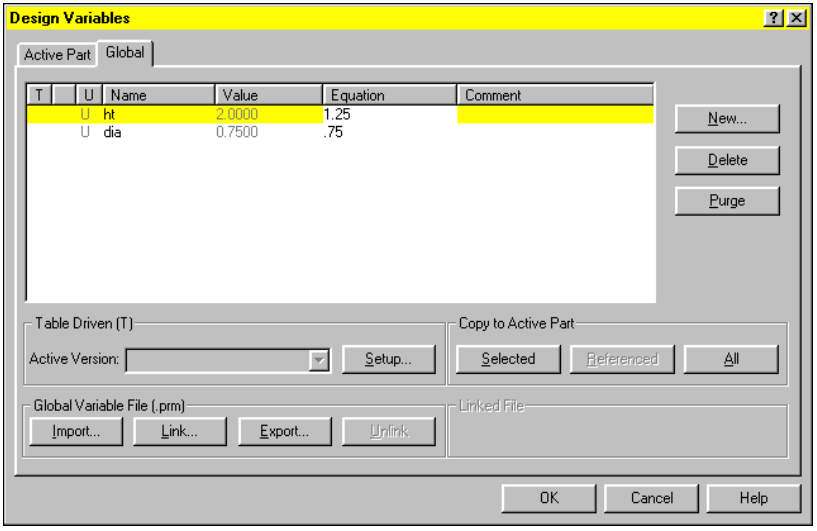
Next, you modify the global design variable controlling the height of the parts.

To modify a global design variable

1 Open the Design Variables dialog box.

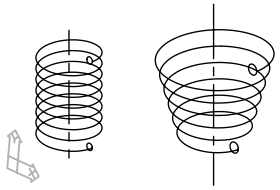
Desktop Menu Part ➤ Design Variables

2 In the Design Variables dialog box, with the Global tab selected, highlight the ht variable. In the highlighted line, double-click the Equation field and enter **1.25**.



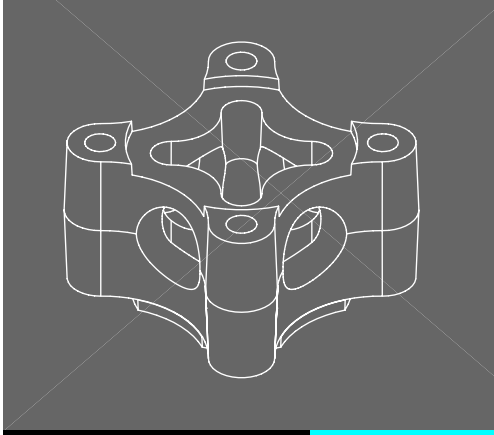
Choose OK to exit the dialog box.

Mechanical Desktop re-evaluates the features and updates both parts.



Design variables are a powerful way to control the features of a part. Both active part and global design variables may be table driven. To create a table driven part, you use Microsoft® Excel software to create a spreadsheet containing values for different versions of a part. You learn more about table driven parts in chapter 15, “Creating Table Driven Parts.”

Creating Parts



In This Chapter

12

- Analyzing design ideas to simplify sketching
- Selecting the base feature
- Planning the order in which to add features
- Stabilizing features with constraints and dimensions
- Creating features that remain fixed relative to work planes and work axes
- Refining features
- Adjusting features according to design changes

This tutorial continues with techniques you learned in previous lessons. You use sketches to create features. You position standard features, such as holes, and then combine them to create a part. You analyze your design and build a model so that you can easily incorporate changes. This is a problem-solving process that you can apply to any parts you create using Autodesk® Mechanical Desktop®.

In this tutorial, you create a saddle bracket in two phases. First, you create all the features of the part in rough form. Then, you refine those features to complete the part.

Key Terms

Term	Definition
base feature	The first feature you create. As the basic element of your part, it represents its simplest shape. All geometry you create for a part depends on the base feature.
consumed sketch	A sketch used in a feature, for example, an extruded profile sketch. The sketch is consumed when the feature is created.
Desktop Browser	A graphical representation of the features that make up your model. You can work in the Browser to create and restructure parts and assemblies, define scenes, create drawing views, and control overall preferences.
placed feature	A mechanical shape that does not require sketches, such as a hole, chamfer, or fillet. Placed features are constrained to the feature on which they are placed and are geometrically dependent.
sketch plane	A temporary drawing surface that corresponds to a real plane on a feature. It is an infinite plane with both *X* and *Y* axes, where you sketch or place a feature.
sketched feature	A three-dimensional solid whose shape is defined by constrained sketches and located parametrically on a part. Sketched features are extrudes, lofts, revolves, sweeps or face splits.
work axis	A parametric construction line created along the centerline of a cylindrical feature, or sketched on the current sketch plane. A work axis can be used as the axis of revolution for a revolved or swept feature, an array of features, to place a work plane, and to locate new sketch geometry. It can be included in dimensions.
work feature	A work axis, work point, or work plane used to construct and position a feature where there is no face on which to sketch or place the feature. You constrain or dimension work features to maintain symmetry throughout updates.
work plane	An infinite plane attached to a part. Can be designated as a sketch plane and can be included in a constraint or dimension scheme. Work planes can be either parametric, or non-parametric.
work point	A parametric work feature used to position a hole, the center of an array, or any other point for which there is no other geometric reference.

Basic Concepts of Creating Parts

You construct a model bit by bit, fashioning shapes to add to it and using tools to cut away the portions of the shapes you do not need. In Mechanical Desktop®, these shapes are the features of the part you are creating.

Analyzing Rough Sketches

You may be accustomed to jotting down design ideas on paper, starting with a rough outline for a part and adding details as you go. Working with Mechanical Desktop is similar: you put some thought into your idea, planning the best way to implement your concept.

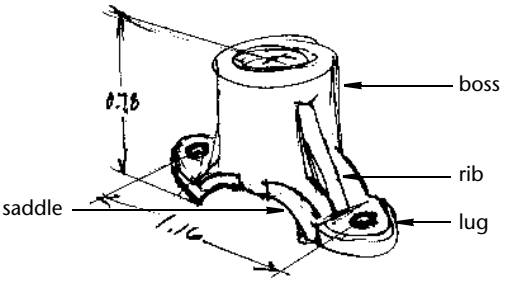
In general, you follow this process to develop a part design:

- Look at the whole part and decide how to break it down into simple shapes.
- Identify the simplest element to use as your base feature.
- Decide the order for creating additional features.
- Determine the methods for creating the features.
- As you build individual features, review and adjust your ideas about how the features work together.
- As you adjust your design strategy, you can revise the features you created earlier.

With early planning, you can express your design in modular, simple terms. When changes occur, as they often do in design work, you can easily accommodate them because of the parametric capabilities in Mechanical Desktop. Any changes you make to your design are quickly recalculated.

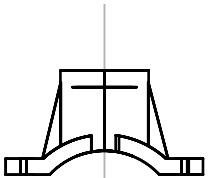
As you study the part to determine the features you need and the order in which to create them, also notice the relationships and patterns of the shapes. Some features may be symmetrical, but others may be built most easily from simple shapes combined to form compound shapes.

The saddle bracket in this rough sketch has four distinct features: the saddle, the mounting lugs, a boss, and strengthening ribs.

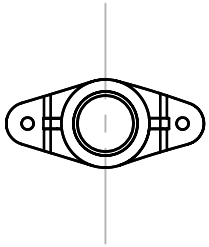


The part is symmetrical. Visualize two perpendicular centerlines—one along the axis of the boss and another intersecting both lugs. As you create this part, consider this symmetry as you constrain features.

As you build the saddle bracket, you learn to create features according to the relationships among them. In this case, the base feature of the part is the saddle and lugs. Because the remaining features attach to the saddle and lugs, you create the main shape first. The next feature you create is the boss because it rests directly on the saddle. Finally, you create the ribs because they attach to both the saddle and the boss.



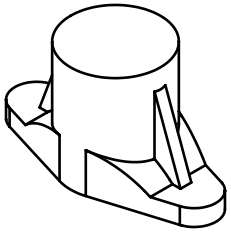
front view



top view

Creating Rough Parts

In the saddle bracket, features are present but lack details such as the arch of the saddle, the mounting holes for the lugs, and the pipe hole for the boss. Despite the missing details, the shape of the part and the placement of features are symmetrical. Working from this basic part, you will add those details.



Dimensioning and Constraining Parts

You apply dimensions and constraints to control the size and shape of a part, and the position of part features. Dimensions can be expressed as numbers, parameters, or equations.

You can use the Design Variables dialog box to create equations and control the relationships between the dimensions on your model. Then you apply the variables to your model and the model is updated to reflect the changes.

If you want to assign design variables as you are defining part sketches and creating features, use the Equation Assistant. You can activate the Equation Assistant in two ways:

- When you are prompted for a dimension value, right-click the graphics area.
- While you are creating sketched and placed features, in the feature dialog box, right-click a value field.

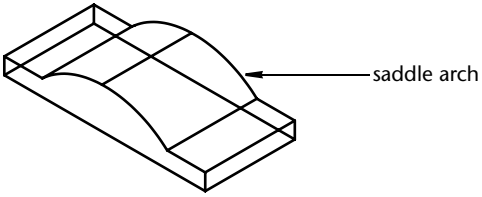
For more information about working with design variables, see "Using Design Variables" on page 239.

To begin this lesson, open the file *saddle.dwg* in the *desktop\tutorial* folder. The drawing is blank but contains the settings you need for this tutorial.

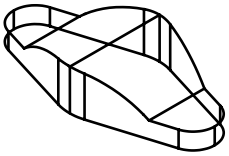
NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

Creating Base Features

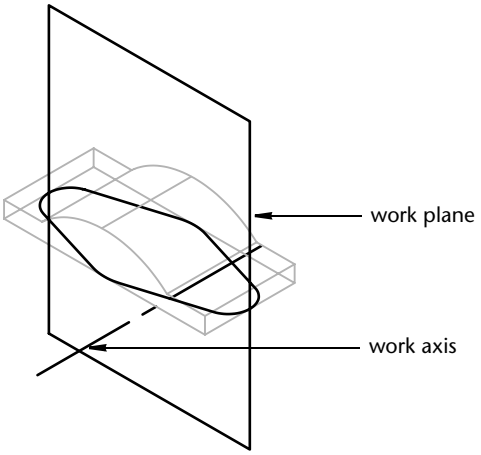
The overall shape of the saddle bracket is simple. First, you sketch a shape to represent the saddle and lugs.



Next, you convert the sketch to a base feature and modify its shape by intersecting it with a second feature. Intersecting the base feature is like cutting away material you don't need.



When you create these features, you position them symmetrically using a work axis and a work plane. Like other features, you include work features in your constraint scheme to maintain symmetry throughout future updates to the part.



Sketching Base Features

After you have a strategy, you are ready to sketch, constrain, and extrude the base feature of the part. Begin by creating a sketch of the block and then converting it to a profile sketch.

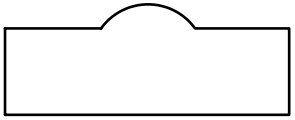
To make it easier to sketch the shape, turn off Polar, Osnap, and Otrack at the bottom of your screen.

To create a profile sketch

1 Use PLINE to sketch this shape. Draw the shape starting at the lower left of the sketch.

You can use the cursor crosshairs to align the top horizontal lines (that is, make them collinear). Use the Direction option of PLINE to control the direction of the arc.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.



2 Use AMPROFILE to profile your sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Mechanical Desktop analyzes the sketch and displays a message on the command line:

Solved underconstrained sketch requiring 5 dimensions or constraints.

NOTE Throughout this tutorial, the number of constraints your sketch needs may differ from the example, depending on how precisely you draw the sketch. You learn how to modify constraints so that your sketch solves correctly.

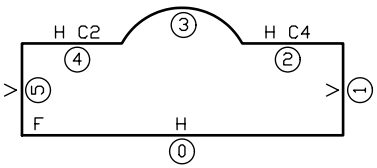
Look at the assumed constraints and determine which constraints you need.

3 Use AMSHOWCON to display all of the existing constraints.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

Respond to the prompt to show all constraints.

Your sketch should look like this. However, the constraint numbering may differ, depending on the order in which you drew the geometry.



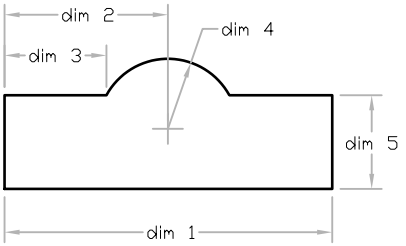
In the example, all sketch elements have constraints except the arc. The lines show vertical (V) or horizontal (H) constraints and the top two horizontal lines show a collinear (C) constraint. A fix constraint is located at the start point of line 0.

NOTE If the fix constraint in your sketch does not appear in the same location as the illustration above, redraw the sketch starting at the lower left.

Now that the basic sketch shape is defined, you need to add dimensions to stabilize its size. Start with its longest lengths to minimize the risk of distorting the shape as it is resized.

For this exercise, add dimensions in the order shown, starting with the dimension for the bottom line.

Depending on your sketch, your default dimension values may differ from those in this exercise.

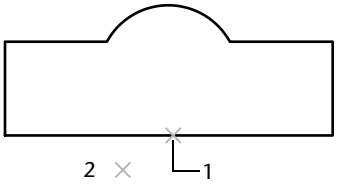


To constrain a sketch

1 Use AMPARDIM to add parametric dimensions to fully constrain the sketch, following the prompts.

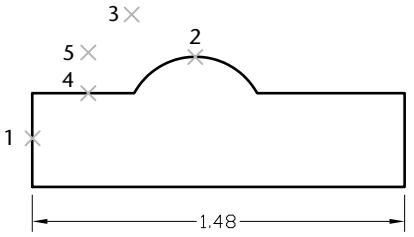
Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify the line (1)*
Select second object or place dimension: *Place the dimension (2)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <1.2297>: *Enter* **1.48**
Solved underconstrained sketch requiring 4 dimensions or constraints.



2 To center the arc, create a horizontal dimension from the center of the arc to the left edge of the sketch.

Select first object: *Specify the left edge (1)*
Select second object or place dimension: *Specify the arc (2)*
Specify dimension placement: *Place the dimension (3)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.5944>: *Enter* **.74**
Solved underconstrained sketch requiring 3 dimensions or constraints.



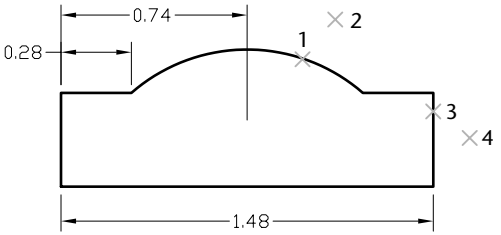
3 Create the dimension for the top left horizontal line. Continue to follow the selection points.

Select first object: *Specify the left horizontal line (4)*
Select second object or place dimension: *Place the dimension (5)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.3951>: *Enter* **.28**
Solved underconstrained sketch requiring 2 dimensions or constraints.

NOTE You may get a message stating that adding a dimension will overconstrain the sketch. This can occur if your sketch does not closely resemble this exercise. Try adding the dimensions in a different order, or re-create your sketch.

4 Finish dimensioning the sketch.

Select first object: *Specify the arc (1)*
Select second object or place dimension: *Place the dimension (2)*
Enter dimension value or [Undo/Diameter/Ordinate/Placement point] <0.4600>: *Enter* **.68**
Solved underconstrained sketch requiring 1 dimensions or constraints.
Select first object: *Specify the line on the right (3)*
Select second object or place dimension: *Place the dimension (4)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.2176>: *Enter* **.20**
Solved fully constrained sketch.
Select first object: *Press* ENTER



Now that your profile sketch is fully constrained, create a solid feature.

To extrude a feature

1 Change to an isometric view of your part.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

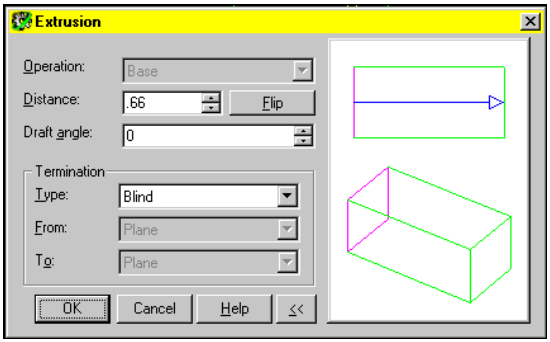
You need to specify the type of extrusion operation, how to terminate the extrusion, and its size.

2 Use AMEXTRUDE to extrude the profile.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

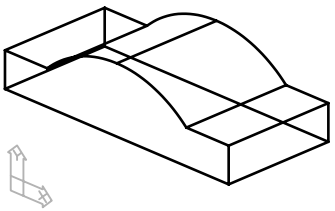
3 In the Extrusion dialog box, specify:

Distance: **.66**
Termination: Blind



Choose OK to create the feature.

The base feature should look like this.



4 Refer to the Desktop Browser, which shows that you have added an extrusion feature to the base feature and that the extrusion was blind (a specific depth).

Click the plus sign beside the extrusion feature to display a profile icon. This display tells you that the extrusion feature originated with the profiled sketch. If you complete a feature and then need to change its size or shape, you can edit it and update the part to reflect the change.

To edit a consumed sketch in the Browser, double-click the profile icon to display the original sketch, or right-click to show the menu, and choose Edit Sketch. Make any changes and choose Part ➤ Update to resize the part with the changed values.

To edit a base feature

1 Select the sketch to edit, responding to the prompts.

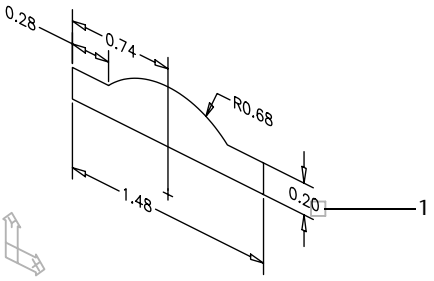
Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Independent array instance/Sketch/surfCut/Toolbody/select Feature] <select Feature>: *Enter* **s**
Select sketched feature: *Specify the extrusion*

2 Modify the height of the sketch, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

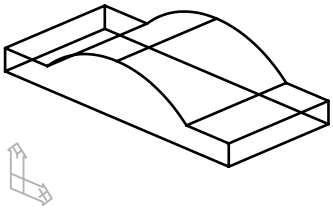
Select dimension to change: *Select the 0.20 dimension (1)*
New value for dimension <.20>: *Enter* **.12**
Solved fully constrained sketch.
Select dimension to change: *Press* ENTER



3 Use AMUPDATE to update the model, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Your part is updated according to the changed dimension and looks like this.



Save your file.

Creating Work Features

Now that you have created the base feature, add the feature that defines the rough shape of the bracket. First, create work features to maintain symmetry. Then, use them to draw, constrain, and extrude the sketch.

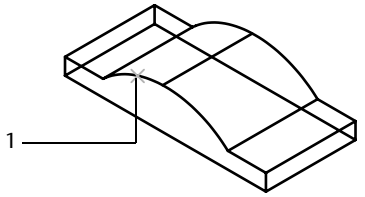
The first work feature is a work axis along the centerline of the arc on the base feature. This work axis anchors your next sketch to the base feature.

To create a work axis

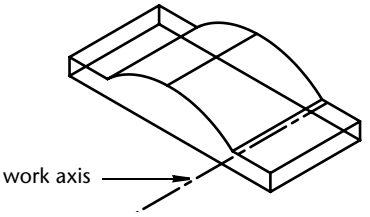
1 Use AMWORKAXIS to create the work axis, responding to the prompt.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Axis.

Select cylinder, cone or torus [Sketch]: *Specify the cylindrical face (1)*



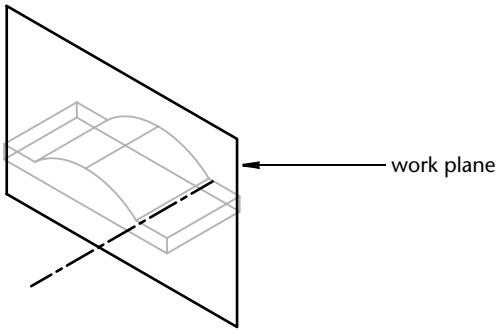
The work axis is displayed as a line along the center of the arc.



If the work axis is not visible, the work axis display is probably turned off.

2 To turn on the display, in the Browser right-click Work Axis1. Choose Visible.

The next work feature, the work plane, forms the second axis of symmetry. This plane is parallel to the front face and intersects both lugs. You specify the work plane position as parallel to the selected face and offset one-half the depth of the part.



To locate the work plane parametrically, specify the offset depth as an equation. By using an equation, the work plane tracks changes in the bracket width and always remains centered. To use an equation, you must determine the dimension parameter before you define the work plane.

To create a work plane

1 Use AMDIMDSP to set dimensions as equations.

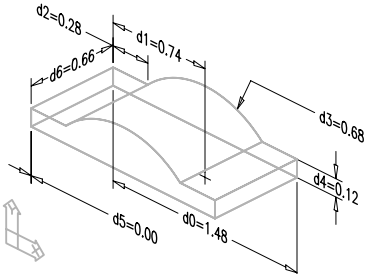
Context Menu In the graphics area, right-click and choose Dimensioning ➤ Dimensions as Equations.

2 Redisplay the sketch dimensions, following the prompt.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Specify any point on the part

3 Choose OK to exit the Extrusion dialog box.



Parameter d6 is the dimension that specifies the width of the feature. Because the dimension parameters for your sketch may differ, make note of the parameter for your part.

4 Press ENTER to exit the command.

5 Create a parametric work plane in the center of the part, parallel to the front surface, and offset one-half the width of the part.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Plane.

6 In the Work Plane Feature dialog box, specify:

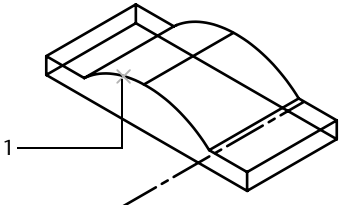
1st Modifier: Planar Parallel
2nd Modifier: Offset
Offset: **d6/2** *(substitute your parameter value for d6)*
Create Sketch Plane: *Clear the check box*

NOTE By default, the Create Sketch Plane option in the Work Plane Feature dialog box is selected. This setting automatically places the sketch plane (the location on which the next feature will be sketched or placed) on the work plane. For this exercise, you specify a sketch plane on a surface of the feature, not on the work plane.

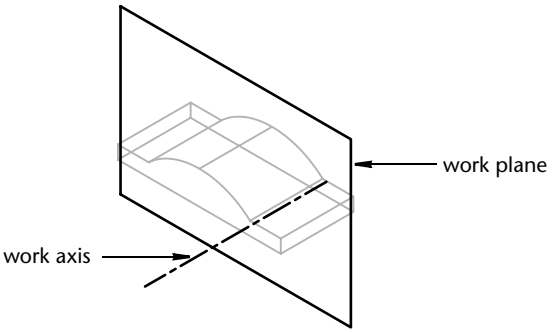
Choose OK.

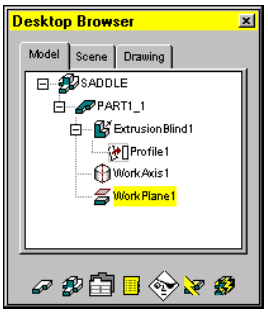
7 Identify the part face to which the work plane is parallel, responding to the prompts.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the curved edge on the front face (1)
Enter an option [Next/Accept] <Accept>: *Press* ENTER
Enter an option [Flip/Accept] <Accept>: *Enter* **f** *to flip the direction into the part*
Enter an option [Flip/Accept] <Accept>: *Press* ENTER



The work plane is displayed as a planar rectangle. The Desktop Browser displays both a work axis and a work plane icon.





Save your file.

Defining Sketch Planes

Before you can sketch the next feature, you must define a new *sketch plane*, an infinite *XY* plane that locates a 2D sketching surface in 3D space.

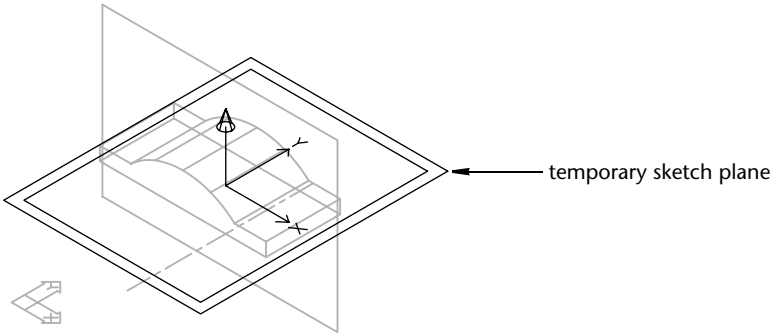
When you create sketched features, you determine the placement and orientation of the sketch plane on a 2D plane. A 2D plane is

- A flat part surface
- The *XY*, *YZ*, or *ZX* axes of the World Coordinate System (WCS)
- A previously defined work plane
- The *XY* plane of the current user coordinate system (UCS)

Unlike a work feature, a sketch plane is a temporary object. Only one sketch plane can exist at the same time.

NOTE Except for base features, you must specify a sketch plane before you can draw a sketch. With base features, the sketch plane is automatically placed on the current UCS.

As you move your mouse over a part, Mechanical Desktop highlights the faces that can be used to define a new sketch plane. Faces that cannot be used are not highlighted. When you select a face, a temporary sketch plane appears on that face.



You can choose the *Z* direction and orientation of the *XY* axes for the new sketch plane.

After you have selected the options, the temporary sketch plane disappears from the screen. You are ready to create the sketch geometry.

In the next exercise, the bottom face of the base feature is the sketch plane. On this face, you sketch a profile to extrude through the part. Once placed, the sketch and subsequent features remain attached to the base feature, regardless of changes you make later.

To create a sketch plane

1 Use MCAD2 to change your display to two viewports.

Desktop Menu View ➤ Viewports ➤ 2Viewports

The left viewport is a top view of the part; the right viewport is an isometric view.

Before you create the sketch plane, check the system variable that controls the UCS settings for your viewports. By default, each viewport has its own UCS.

2 If necessary, change the UCS setting so that each viewport uses the same UCS, responding to the prompt.

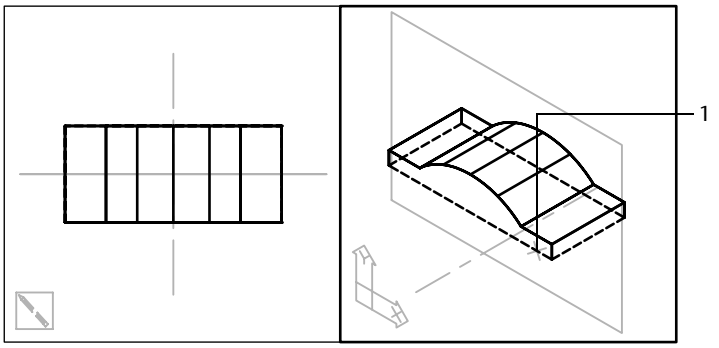
Command UCSVP

Enter new value for UCSVP <1>: *Enter* **0**

3 Use AMSKPLN to create a new sketch plane for the profile to be extruded, responding to the prompts.

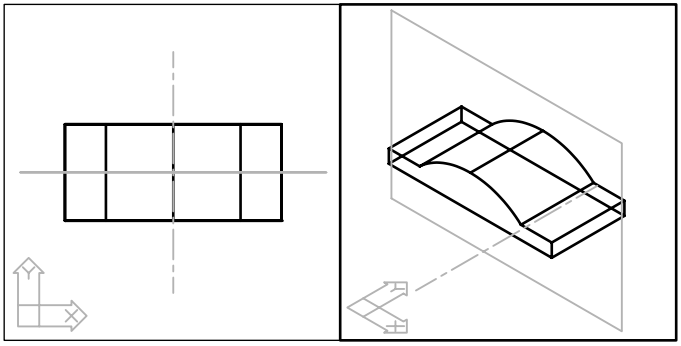
Context Menu In the graphics area, right-click and choose New Sketch Plane.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the bottom face when it is highlighted (1)
Enter an option [Accept/Next] <Accept>:
Choose **n** *to cycle to the bottom face, or press* ENTER
Select edge to align X axis or [Z-flip/Rotate]:
Enter **z** *to flip the Z axis up through the part*
Plane = Parametric
Select edge to align X axis or [Z-flip/Rotate] <accept>:
Verify that the X axis is pointing to the right, and press ENTER



You can pick the *Z* axis arrow to flip the *Z* axis orientation. You can also pick part and work feature edges to orient the *XY* plane.

The UCS icon in the viewports is updated to reflect changes in the sketch plane orientation. The sketch plane is always coincident with the UCS *XY* plane.



Creating Extruded Features

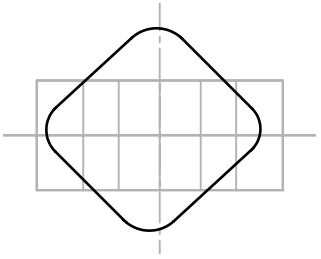
To define the rough shape of the saddle bracket, you sketch a diamond shape with filleted corners and add constraints to stabilize its shape. When the feature is stabilized with geometric constraints, you add dimensions to fully define its size. Finally, you extrude the sketch, creating a solid feature from the combined volume of the original base feature and the extruded feature.

To create a profile sketch

1 Use PLINE to sketch this shape in the left viewport. With PLINE, you may need to use the Direction option to control the direction of arcs.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.

NOTE To make it easier to sketch the shape, make sure POLAR, OSNAP, and OTRACK are turned off at the bottom of your screen.



2 Use AMPROFILE to create a profile from the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Mechanical Desktop analyzes the sketch, redraws it, and displays this message:

Solved underconstrained sketch requiring 10 dimensions or constraints.

NOTE If your sketch needs more than 10 dimensions or constraints to solve the sketch, you probably need some tangency and constraints. Look for sharp discontinuities between the fillets and the lines they join. You make these corrections when you constrain the sketch to the base feature.

---

3 Look at the Desktop Browser. The profile you just created is represented as Profile2.

Because you have not extruded the profile, it is not consumed by a feature. Therefore, the Browser shows that Profile2 is aligned at the same level in the hierarchy as ExtrusionBlind1.

Because you added this feature to the base feature, you need to constrain its shape and size and then constrain it to the existing part.

Constraining Sketches

To constrain a sketch, first you add and change geometric constraints to create the shape of the bracket and to define its symmetry about the two centerlines formed by the work plane and the work axis. Then you dimension the sketch to maintain the proper length and width.

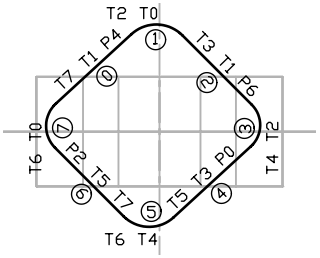
NOTE Don't be concerned if your sketch appears to be misshapen compared to the illustrations. Constraining the sketch to the base feature will correct its shape.

To geometrically constrain a sketch

1 Use AMADDCON to add tangent constraints to the arcs and lines, following the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Tangent.

Valid selection(s): line, circle, arc, ellipse or spline segment
Select object to be reoriented: *Specify an arc segment*
Valid selection(s): line, circle, arc, ellipse or spline segment
Select object to be made tangent to: *Specify an adjoining line segment*
Solved underconstrained sketch requiring n dimensions or constraints.
Valid selection(s): line, circle, arc, ellipse or spline segment
Select object to be reoriented:
Continue adding constraints, or press ENTER *twice to end the command*



NOTE If the constraint display is too small, choose Part ➤ Part Options and adjust the constraint size in the Desktop Options dialog box. Redisplay the constraints.

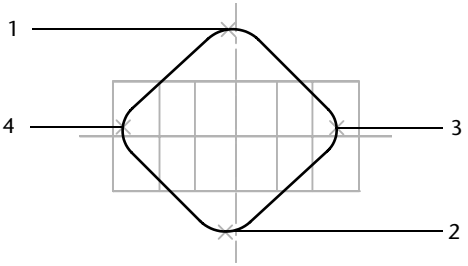
You need to add radial constraints so that opposing arcs have equal radii. Radial constraints make the arcs the same size and maintain the symmetry needed between the sides of the bracket.

Fewer dimensions are needed because one parametric dimension solves 2 degrees of freedom by specifying the size of 2 arcs.

2 Select the arcs to constrain, following the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Radius.

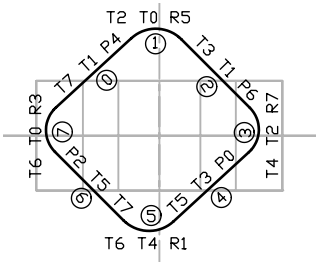
Valid selection(s): arc or circle
Select object to be resized: *Select the arc at the top of the sketch (1)*
Valid selection(s): arc or circle
Select object radius is based on: *Select the arc at the bottom of the sketch (2)*
Solved underconstrained sketch requiring 9 dimensions or constraints.



3 Add radial constraints to the left and right arcs to make them equal in size.

Valid selection(s): arc or circle
Select object to be resized: *Select the arc at the right of the sketch (3)*
Valid selection(s): arc or circle
Select object radius is based on: *Select the arc at the left of the sketch (4)*
Solved underconstrained sketch requiring 8 dimensions or constraints.

Your left viewport should look like this.



If you sketched in a different order, your arcs and lines may be numbered differently.

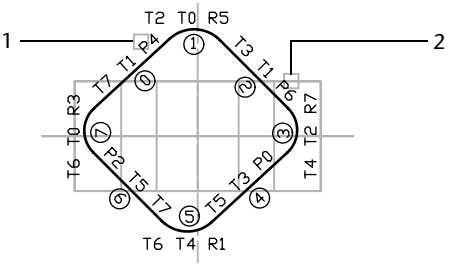
Valid selection(s): arc or circle
Select object to be resized: *Press* ENTER
Enter an option
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix/eXit] <eXit>: *Press* ENTER

4 Delete any parallel constraints, responding to the prompts

If your sketch doesn't contain parallel constraints, skip this procedure.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Delete Constraints.

Select or [Size/All]: *Specify the constraint with the P symbol (1)*
Select or [Size/All]: *Specify the constraint with the P symbol (2)*
Select or [Size/All]: *Press* ENTER

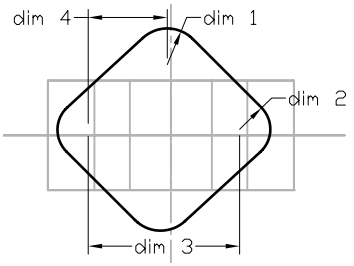


These parallel constraints, although valid, conflict with adding dimensions between arc centers. You need to remove the parallel constraints to prevent overconstraining the sketch.

Save your file.

Dimensioning Sketches

Now that the feature is stabilized with geometric constraints, you can dimension the distance between the arc centers and specify the arc radius. You need four dimensions: a radius dimension for each arc, a dimension between the left and right arc centers, and a dimension between the center of the sketch and the center of either the left or right arc.



To dimension a sketch

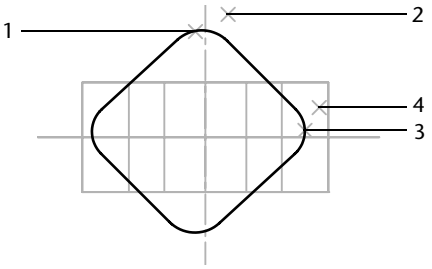
1 Use AMDIMDSP to change the dimension display back to numbers.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Dimensions as Numbers.

2 Use AMPARDIM to dimension the radius for the top and right arcs, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

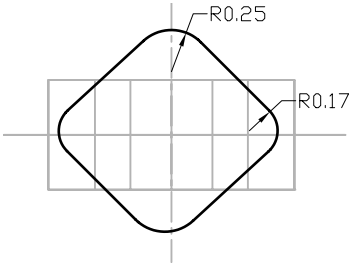
Select first object: *Specify the arc (1)*
Select second object or place dimension: *Place the dimension (2)*
Enter dimension value or [Undo/Diameter/Ordinate/Placement point] <0.1986>: *Enter* **.25**



3 Continue on the command line.

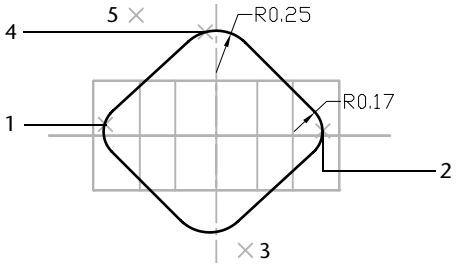
Select first object: *Specify the arc (3)*
Select second object or place dimension: *Place the dimension (4)*
Enter dimension value or [Undo/Diameter/Ordinate/Placement point] <0.1676>: *Enter* **.17**

Your sketch should look like this.



4 Create a horizontal dimension between the centers of the left and right arcs.

Select first object: *Specify the left arc center (1)*
Select second object or place dimension: *Specify the right arc center (2)*
Specify dimension placement: *Place the dimension (3)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.9797>: *Press* ENTER



5 Dimension the distance between the centers of the top and left arcs.

Select first object: *Specify the left arc center (1)*
Select second object or place dimension: *Specify the top arc center (4)*
Specify dimension placement: *Create a horizontal dimension (5)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.5135>: *Press* ENTER

6 Press ENTER to exit the command.

In this case, you do not change the values while you create the dimensions. While a sketch is underconstrained, dimension changes can cause it to distort, and you may not be able to recover its correct shape.

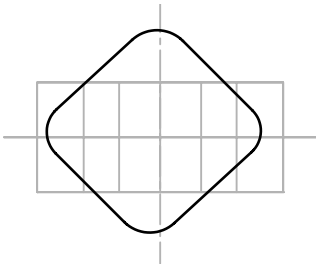
Creating Constraints Between Features

The sketch geometry is now completely defined. However, to position the sketch symmetrically on the base feature, you need to constrain the sketch to the work plane and the work axis because they serve as centerlines for the part.

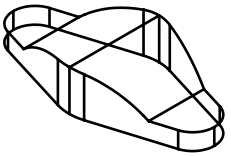
You use the project (PR) constraint to project points onto objects (similar to the NEA object snap) and the concentric (C) constraint to force two arc or circle centers to be coincident.

As you determined when you first analyzed the part,

- The left and right arcs of the sketch form the lugs for the saddle bracket. The arc centers must lie on the work plane.
- The top and bottom arcs of the sketch form the base for the boss, in the exact center of the part. The centers of both top and bottom arcs are coincident with the intersection of the work plane and the work axis.



rough shape



rough shape as a part

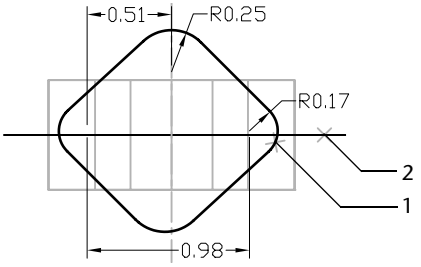
To constrain a sketch to a base feature

1 Use AMADDCON to make the center of the right arc lie on the work plane, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Project.

Valid selection(s): line, circle, arc, ellipse or spline segment
Specify a point to project: *Enter* **cen**
of: *Specify the arc (1)*
Valid selection(s): line, circle, arc, ellipse, work point or spline segment
Select object to be projected to: *Specify the work plane (2)*

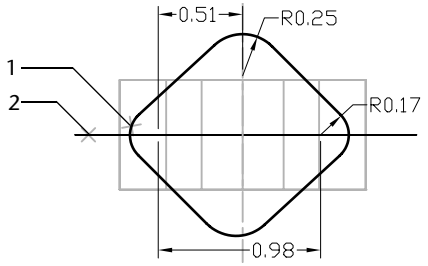
To make selecting lines and arcs easier, use transparent ZOOM. You can zoom in or out while using an active command. At the Command prompt, enter **'z,** and select the area of the sketch you want to magnify. Then continue with the active command.



NOTE If you do not use the **cen** object snap to specify the arc centers, you will not be able to create the project constraints.

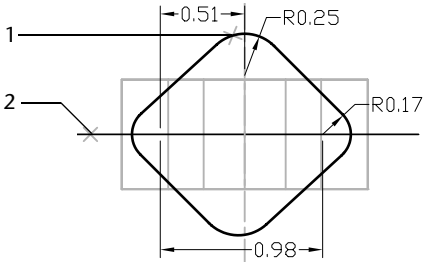
2 Make the center of the left arc lie on the work plane.

Valid selection(s): line, circle, arc, ellipse or spline segment
Specify a point to project: *Enter* **cen**
of: *Specify the arc (1)*
Valid selection(s): line, circle, arc, ellipse, work point or spline segment
Select object to be projected to: *Specify the work plane (2)*



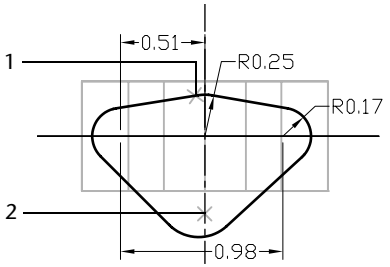
3 Position the center of the top arc on the work plane.

Valid selection(s): line, circle, arc, ellipse or spline segment
Specify a point to project: *Enter* **cen**
of: *Specify the arc (1)*
Valid selection(s): line, circle, arc, ellipse, work point or spline segment
Select object to be projected to: *Specify the work plane (2)*



4 Position the center of the top arc on the work axis.

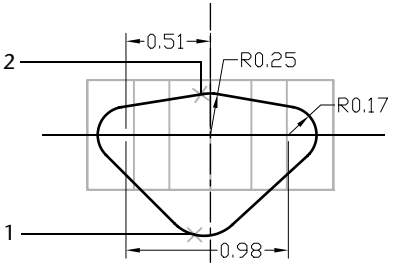
Valid selection(s): line, circle, arc, ellipse or spline segment
Specify a point to project: *Enter* **cen**
of: *Specify the arc (1)*
Valid selection(s): line, circle, arc, ellipse, work point or spline segment
Select object to be projected to: *Specify the work axis (2)*



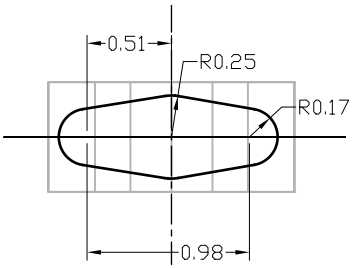
5 Use AMADDCON to make the center of the bottom arc concentric with the center of the top arc, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Concentric.

Valid selection(s): arc, circle, or ellipse
Select object to be reoriented: *Specify the bottom arc (1)*
Valid selection(s): arc, circle, ellipse, or work point
Select object to be made concentric to: *Specify the top arc (2)*
Valid selection(s): arc, circle, or ellipse
Select object to be reoriented: *Press* ENTER
Enter an option
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix/eXit] <eXit>: *Press* ENTER



Your sketch should be fully solved and look like this.



Save your file.

Editing Sketches

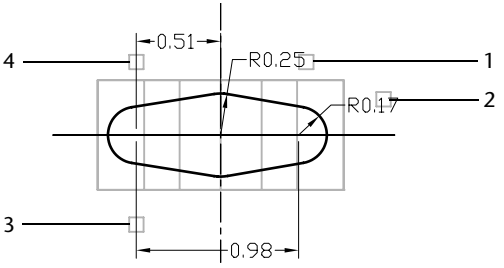
Now that the sketch is fully constrained, you can change the sketch dimensions to position the sketch on your part. Modify the distances between the center of the left arc and the center of the sketch and between the centers of the left and right arcs.

To change a sketch dimension

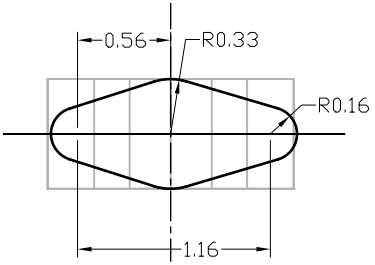
1 Use AMMODDIM to modify the values of the dimensions, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Specify the dimension (1)*
New value for dimension: <0.25>: *Enter* **.33**
Select dimension to change: *Specify the dimension (2)*
New value for dimension: <0.17>: *Enter* **.16**
Select dimension to change: *Specify the dimension (3)*
New value for dimension: <0.98>: *Enter* **1.16**
Select dimension to change: *Specify the dimension (4)*
New value for dimension: <0.51>: *Enter* **.56**
Select dimension to change: *Press* ENTER



Your part should look like this.



Now, you need to create an equation between the overall dimension and the dimension that centers the feature on the part and maintains symmetry relative to the work axis. Display the dimensions as parameters, and then use them as variables in the parametric equation.

2 Use AMDIMDSP to display the dimensions as parameters.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Dimensions As Parameters.

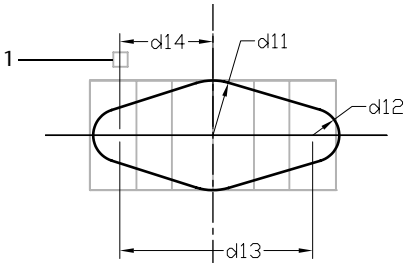
NOTE Your dimension parameter numbers may differ from those shown in the illustration.

---

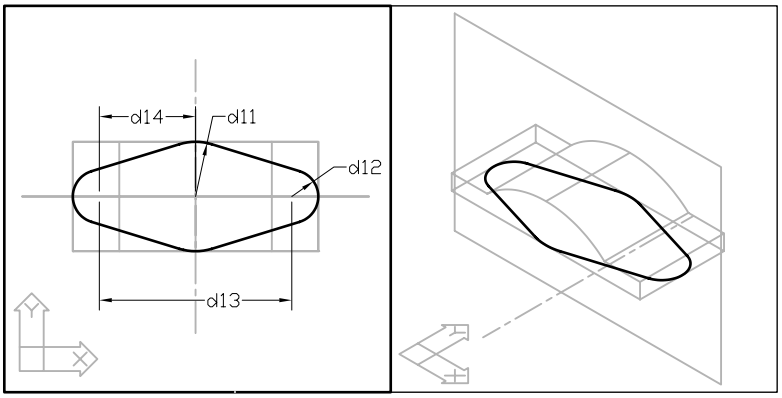
3 Make the dimension between the top and left arcs one-half the horizontal distance between the left and right arcs, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Specify the dimension (1)*
Enter new value for dimension <.56>:
Enter **=dx/2**, *where x is the dimension that corresponds to d13 in the illustration*
Select dimension to change: *Press* ENTER

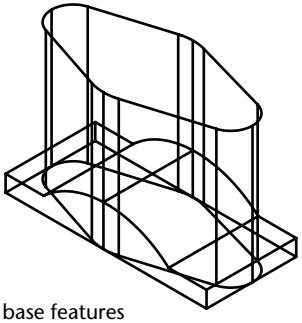


Now that the profile sketch is completely constrained and dimensioned, you can use it to change the shape of the base feature.

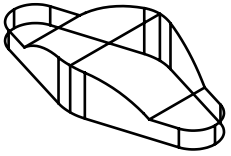


Extruding Profiles

You create a solid feature by extruding the profile through to the boundary of the base feature, retaining the common volume. To create the rough shape of the saddle bracket, you extrude the profile sketch up and completely through the base feature. Because the sketch you extrude changes the shape of the base feature, the intersection shares the volume of both.



base features



resulting intersection

To extrude a profile through a base feature

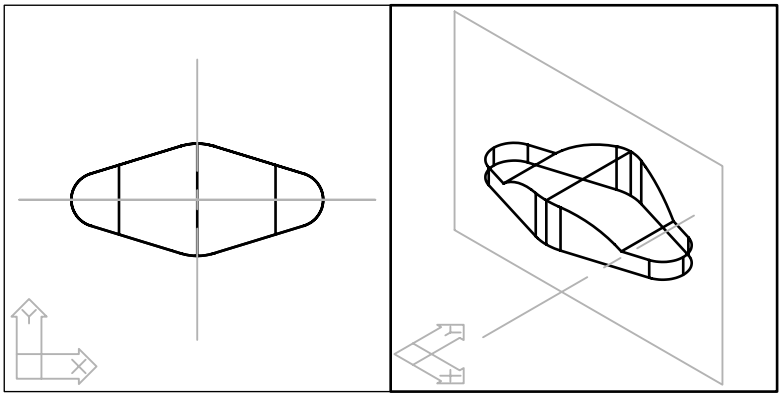
1 Use AMEXTRUDE to create the extrusion.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, accept the default size and specify:

Operation: Intersect
Flip: *Verify that the direction arrow is pointed up through the part*
Termination: Through

Choose OK to exit the dialog box and create the extrusion.

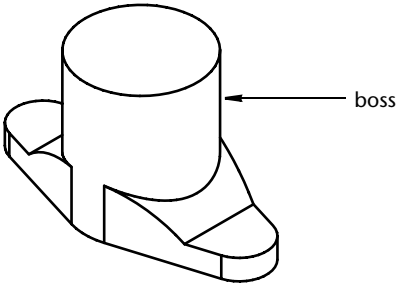


Save your file.

Creating Revolved Features

With the rough shape of the saddle bracket defined, you can create the next dependent feature, the boss, which is a cylinder. The fastest and most efficient method to model the cylindrical boss is to extrude a circle. Alternatively, you can revolve a rectangle about a central axis. This method is used here to teach you the revolving method.

When you finish the exercise, your model will look like this.



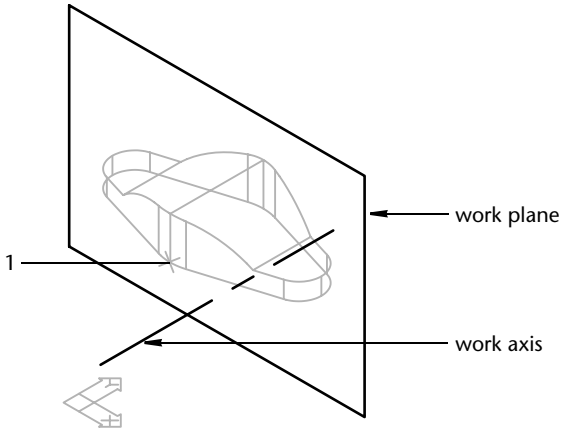
Before you can sketch the profile for the revolved feature, you need to create a work axis to serve as the centerline for the revolved feature. Work in the right viewport, the isometric view.

To sketch a profile for a revolved feature

1 Use AMWORKAXIS to create a work axis, responding to the prompt.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Axis.

Select cylinder, cone or torus [Sketch]: *Specify the face (1)*



2 A work axis passes vertically through the part. If the work axis is not displayed, use AMVISIBLE to display it.

Desktop Menu Part ➤ Part Visibility

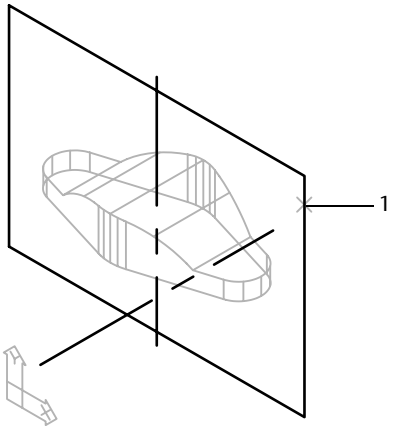
In the Desktop Visibility dialog box, choose the Part tab and check Work Axes. Select Unhide and choose OK.

Next, you need to create a new sketch plane. Because the cylinder is vertical, you place the sketch plane on the previously defined work plane.

3 Create a new sketch plane, responding to the prompts.

Context Menu In the graphics area, right-click and choose New Sketch Plane.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Specify the work plane (1)
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>: *Press* ENTER



The sketch plane assumes the *Z* direction and *XY* orientation of the work plane.

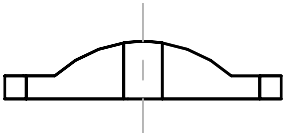
4 Hide the work plane. This time use the Browser method.

Browser Right-click WorkPlane1 and choose Visible

The work plane is no longer visible.

5 Make the left viewport active and change the view so that you see a front view of the part as you look at the sketch plane.

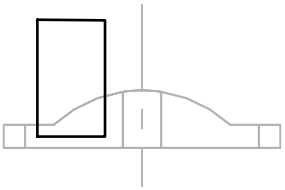
Desktop Menu View ➤ 3D Views ➤ Front



6 Sketch a rectangular outline of the cylinder, following the prompts.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Rectangle.

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
Specify a point
Specify other corner point: *Specify a second point*



7 Use AMPROFILE to convert the sketch to a profile for the feature.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Mechanical Desktop selects the sketch you just drew, and converts it to a profile. The sketch still needs four dimensions or constraints.

To constrain a profile sketch to revolve

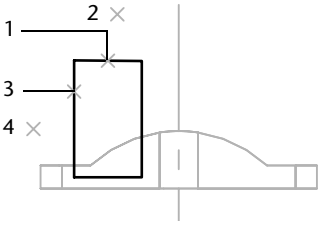
1 Use AMDIMDSP to change the dimension display to numbers.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Dimensions as Numbers.

2 Use AMPARDIM to dimension the length and width of the sketch, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify the line (1)*
Select second object or place dimension: *Place the dimension (2)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.3629>: *Enter* **.33**
Solved underconstrained sketch requiring 3 dimensions or constraints.
Select first object: *Specify the line (3)*
Select second object or place dimension: *Place the dimension (4)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.6687>: *Enter* **.78**
Solved underconstrained sketch requiring 2 dimensions or constraints.
Select first object: *Press* ENTER



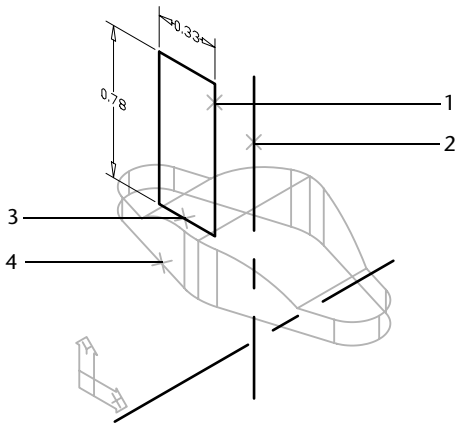
In the right viewport, constrain the sketch to the part as follows:

- Make the bottom line of the sketch collinear with the bottom of the part.
- Make the right side of the rectangle collinear with the vertical work axis so that it serves as the axis of revolution of the feature.

3 Use AMADDCON to add collinear constraints, following the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Collinear.

Valid selections: line or spline segment
Select object to be reoriented: *Specify the line (1)*
Valid selections: line or spline segment
Select object to be made collinear to: *Specify the vertical work axis (2)*
Solved underconstrained sketch requiring 1 dimensions or constraints.
Valid selections: line or spline segment
Select object to be reoriented: *Specify the line (3)*
Valid selections: line or spline segment
Select object to be made clinger to: *Specify the part edge (4)*
Solved fully constrained sketch.
Valid selections: line or spline segment
Select object to be reoriented: *Press* ENTER
Enter an option [Hor/Ver/PErp/PAr/Tan/CL/CN/POroj/Join/XValue/YValue/Radius/Length/Mir/Fix/eXit] <eXit>: *Press* ENTER



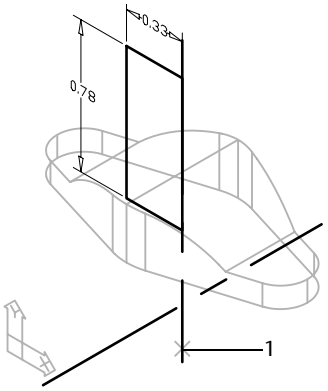
In the next procedure, you create the cylinder by revolving the sketch about the work axis. You can also revolve a sketch about a part edge or about a line in the profile sketch.

To revolve a feature about a work axis

1 Use AMREVOLVE to revolve the sketch about the work axis, responding to the prompt.

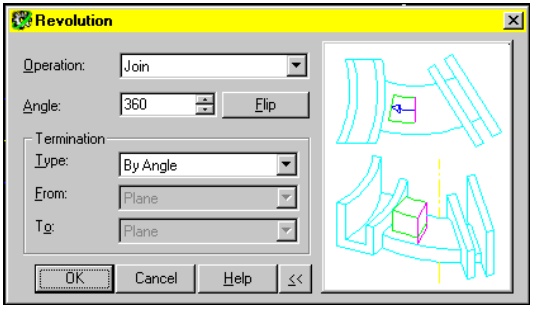
Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Revolve.

Select revolution axis: *Specify the axis (1)*



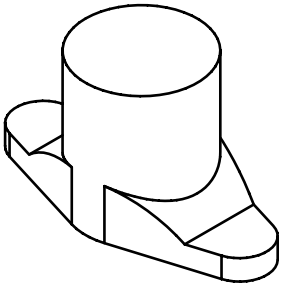
2 In the Revolution dialog box, specify the operation, termination, and angle of revolution. Because the cylinder attaches to the part, define the revolution to be a full (360 degrees) termination that joins to the part.

Operation: Join
Angle: *Enter* **360**
Termination: By Angle



Choose OK.

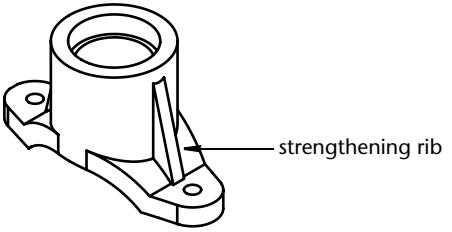
After specifying the type of revolution and the axis of rotation, the cylinder is created on your model.



Save your file.

Creating Symmetrical Features

The final features are the strengthening ribs, located on each side of the saddle just above the lugs.



The ribs can be created simultaneously from a single open profile sketch. You sketch an outline of the ribs, and add dimensions and constraints to make the ribs symmetrical. Then you extrude the ribs automatically with the Rib feature.

The sketch you create lies on the same plane as the revolution feature, so it is not necessary to create a new sketch plane.

Before you begin, change to the front view, and one viewport.

To sketch a feature on a part

1 Use PLINE to sketch the ribs.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.

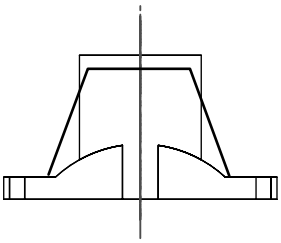
Sketch a three-segment polyline in the approximate outline of the ribs. The lines don't have to touch the saddle.

2 Use AMPROFILE to create an open profile from the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Respond to the prompt.

Select part edge to close the profile <open profile>: *Press* ENTER



Next, constrain the sketch.

Constraining Sketches

When you solved the sketch, a parallel constraint was applied between the top horizontal line of the part and the horizontal segment of the sketch. Six additional dimension or constraints are needed to fully constrain the sketch.

Use dimensions to adjust the size of the ribs and to center them on the part.

To constrain a sketch

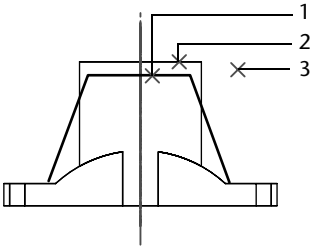
1 Use AMPARDIM to dimension the distance between the top of the sketch and the top of the part.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Respond to the prompts as follows:

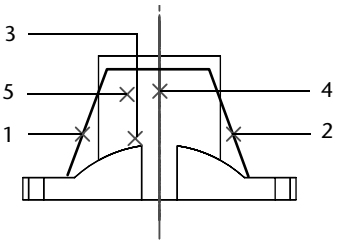
Select first object: *Specify the line (1)*
Select second object or place dimension: *Specify the line (2)*
Specify dimension placement: *Place the vertical dimension (3)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.1683>: *Enter* **.08**

Solved under constrained sketch requiring 6 dimensions or constraints.



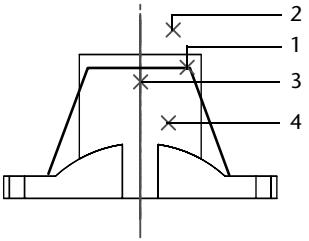
2 Add dimensions for the angle between the two ribs, and the angle between the work axis and one rib.

Select first object: *Specify the line (1)*
Select second object or place dimension: *Specify the line (2)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.1683>: *Enter* **n**
Specify dimension placement: *Place the dimension (3)*
Enter dimension value or [Undo/Placement point] <47>: *Enter* **40**
Select first object: *Specify the vertical work axis (4)*
Select second object or place dimension: *Specify the line (1)*
Specify dimension placement: *Place the angular dimension (5)*
Enter dimension value or [Undo/Placement point] <20>: *Press* ENTER



3 Add horizontal dimensions for the top line of the sketch, and from the work axis to the outer edge of the top line.

Select first object: *Specify the line (1)*
Select second object or place dimension: *Place the horizontal dimension (2)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.9806>: *Enter* **.58**
Select first object: *Specify the outer end of line (1)*
Select second object or place dimension: *Specify the work axis(3)*
Specify dimension placement: *Place the horizontal dimension (4)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.9806>: *Enter* **.29**



4 Repeat step three to add a horizontal dimension of **.98** between the two lower endpoints of the sketch, and **.49** between the work axis and one lower endpoint of the sketch.

Solved fully constrained sketch.

To verify that the ribs are symmetrical, express the dimensions as equations. Set the distance and the angle between the axis and the rib to one-half the distance and angle between both ribs.

To display the dimensions as parameters

1 Use AMDIMDSP to change the display of the dimensions from numeric to parametric.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Dimensions As Parameters.

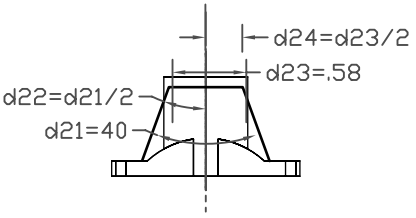
Display the dimensions as equations.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Dimensions As Equations.

2 Use AMMODDIM to edit the dimensions. Use the work axis as the centerline of the part.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Specify the horizontal dimension from the work axis to either endpoint of the top line of the sketch*
New value for dimension <*current*>:
Enter **dx/2**, *where x is the horizontal dimension for the top line of the sketch*
Solved fully constrained sketch.
Select dimension to change:
Specify the dimension for the angle between the work axis and a side of the sketch
New value for dimension <*current*>:
Enter **dy/2**, *where y is the dimension for the angle between the sides of the sketch*
Solved fully constrained sketch.
Select dimension to change: *Press* ENTER



3 Use AMUPDATE to apply any changes to the rib sketch.

Context Menu Right-click the graphics area and choose Update Part.

You are ready to extrude the sketch to form symmetrical ribs.

4 Use 3DOrbit to adjust the view so you can see the rib feature preview before you create the ribs.

Desktop Menu Choose View ➤ 3D Orbit. Rotate the view slightly to the left, and tilt it slightly downward.

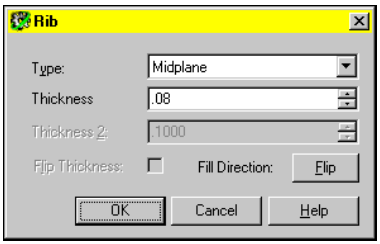
5 Use AMRIB to extrude the ribs.

Browser In the Browser, right-click the open profile icon, and choose Rib.

In the Rib dialog box, specify:

Type: Midplane
Thickness: Enter **.08**

Verify the direction arrow points into the part, and choose OK.



The two symmetrical ribs are extruded to the face of the cylinder.

Next, suppress the hidden lines so that you can see your model more clearly.

To suppress silhouette edges from Mechanical Desktop parts

1 Set the DISPSILH system variable to **1**, responding to the prompts.

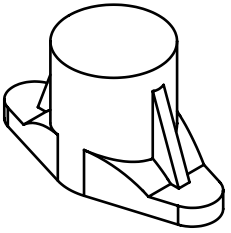
Command DISPSILH

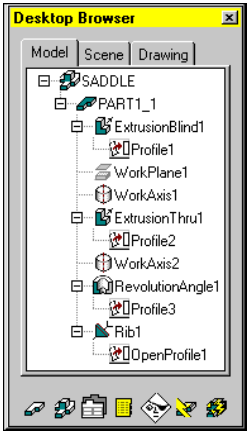
Enter new value for DISPSILH <0>: *Enter* **1**

2 Use HIDE to remove the hidden lines from your display.

Desktop Menu View ➤ Hide

Your part should now look like this. The Desktop Browser shows the hierarchy of the part features.





3 Return to wireframe display.

Desktop Menu View ➤ Shade ➤ 3D Wireframe

Save your file.

Refining Parts

Now, you complete the part by modifying its features in the same order as you created them: the saddle and lugs, the boss, and the ribs.

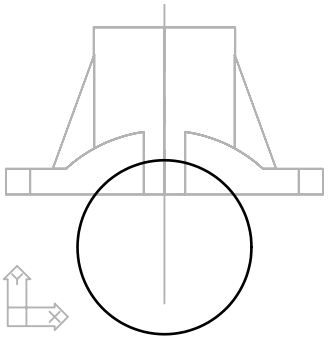
To finish the body of the saddle bracket, you need to cut the pipe saddle, adjust the length of the lugs, and create mounting holes. To create the saddle, you cut an arc through the front of the saddle body. To cut the arc, you create a circle and extrude it through the part, along the horizontal work axis. For this feature, you use the previously defined sketch plane.

To sketch and constrain the circle to be extruded

1 Use CIRCLE to draw the circle to extrude, following the prompts. Work in the left viewport.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Circle.

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify a center point
Specify radius of circle or [Diameter]: *Specify a point to define the radius*



2 Use AMPROFILE to solve the sketch to convert it to a profile sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

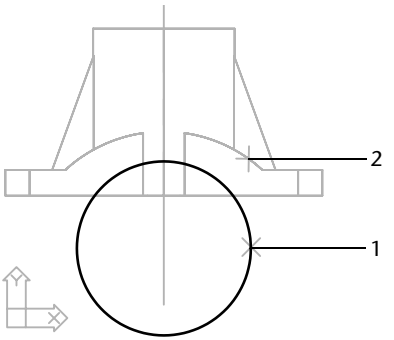
You need to constrain the circle to the part. The sketch also needs two more dimensions: the location of the center and the diameter of the circle.

The work axis is the center of the saddle arcs on the front and back of the bracket. By making the circle concentric with the arcs, you satisfy two constraints, the location of the center and the relationship of the circle to the part.

3 Use AMADDCON to constrain the circle to be concentric with the saddle arcs, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Concentric.

Valid selection(s): arc, circle, or ellipse
Select object to be reoriented: *Specify the circle (1)*
Valid selection(s): arc, circle, ellipse, or work point
Select object to be made concentric to: *Specify the arc (2)*
Valid selection(s): arc, circle, or ellipse
Select object to be reoriented: *Press* ENTER
Enter an option
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix/eXit] <eXit>: *Press* ENTER



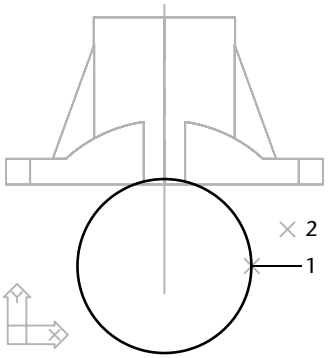
4 Use AMDIMDSP to return the dimension display to numeric, responding to the prompt.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Dimensions As Numbers.

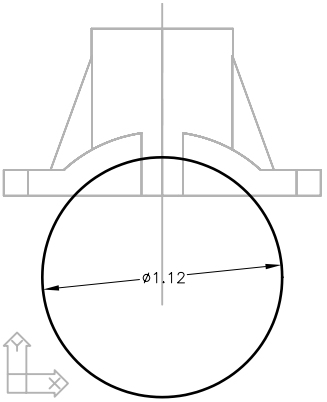
5 Use AMPARDIM to dimension the diameter of the circle, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Specify the circle (1)*
Select second object or place dimension: *Place the dimension (2)*
Enter dimension value or [Undo/Placement point] <1.0976>: *Enter* **1.12**
Select first object: *Press* ENTER



The sketch is now fully constrained and looks like this.



To extrude a feature

1 Extrude the feature, specifying a cut operation with a midplane termination.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

2 In the Extrusion dialog box, specify:

Operation: Cut
Distance: *Enter* **.66**
Termination: Type: Mid Plane

Choose OK. The arc shape cuts through the saddle bracket.

To complete the body of the bracket, you need a placed feature on each of the lugs for mounting holes.

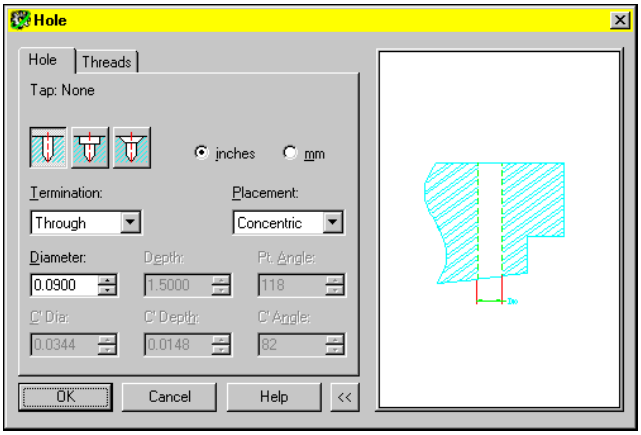
To create a drilled hole

1 Use AMHOLE to place the mounting holes. Work in the isometric view.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Hole.

2 In the Hole dialog box, select the Drilled hole type icon and specify:

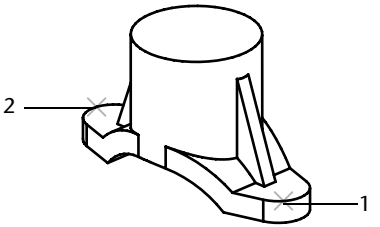
Termination: Through
Placement: Concentric
Diameter: *Enter* **.09**



Choose OK.

3 Respond to the prompts as follows:

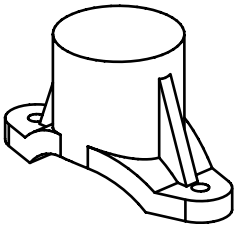
Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Specify face (1)
Enter an option [Next/Accept]<Accept>: *Press* ENTER
Select the concentric edge: *Specify edges (1) for the first hole*



4 Continue on the command line to place the second hole.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Specify face (2)
Select the concentric edge: *Specify edges (2) for the second hole*
Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]: *Press* ENTER

Your part should look like this.



To complete the boss, you create a counterbored hole through the cylinder. You create the hole as a placed feature on the same vertical work axis as the cylinder.

Keep the right viewport active, and specify a counterbored hole drilled through the part, concentric with the cylinder.

To create a counterbored hole

1 Use AMHOLE to place the counterbored hole.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Hole.

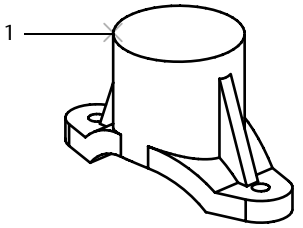
In the Hole dialog box, select the Counterbore hole type icon and specify:

Termination: Through
Placement: Concentric
Hole Parameters: Size: *Enter* **.42**
C'Bore/Sunk Size: C' Dia: *Enter* **.48**
C'Bore/Sunk Size: C' Depth: *Enter* **.125**

Choose OK.

2 Respond to the prompts as follows:

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Specify face (1)
Select the concentric edge: *Specify edge (1)*
Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]: *Press* ENTER



The ribs currently extend too far onto the lug area, leaving little room for the mounting holes. To adjust the design, you need to reduce the width and angle of the ribs.

Work in the left viewport. Modify the ribs by changing a few sketch dimensions. The previously-defined equations keep the ribs symmetrical.

Use the Browser to select the rib feature and redisplay its sketch dimensions. After you change the dimension values, use the Update icon in the Browser to incorporate the changes.

To edit a feature

1 Use AMEDITFEAT to edit the rib sketch.

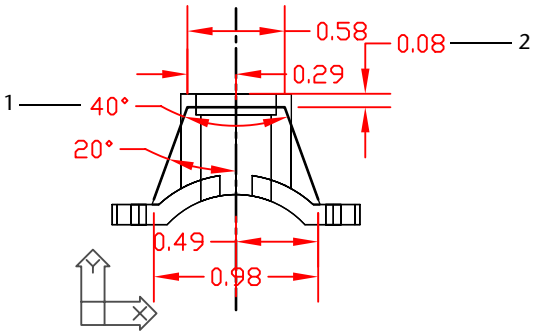
Browser Right-click OpenProfile1 and choose Edit Sketch.

The rib sketch and its dimensions become visible on the screen.

2 Change two of the dimensions in the sketch, following the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

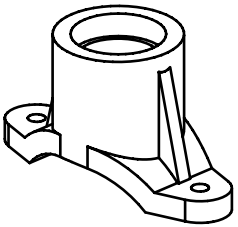
Select object: *Specify the dimension (1)*
Enter new value for dimension <40>: *Enter* **28**
Select object: *Specify the dimension (2)*
Enter new value for dimension <.o8>: *Enter .06*
Select object: *Press* ENTER



3 Update the part to reflect the new dimension values in the sketch.

Context Menu In the graphics area, right-click and choose Update Part.

The ribs are updated to reflect your dimensional changes.



In the Browser, each feature is placed in the order it was created.

Shading and Lighting Models

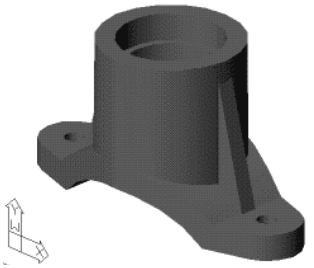
To see your model better, use the shade button on the Desktop View toolbar to toggle shading on. Then adjust the lighting of your shaded model.

To toggle shading of a part

1 Use SHADE to shade your part.

Desktop Menu View ➤ Shade ➤ Gouraud Shaded

Your part should now look like this.



The Desktop View toolbar also contains commands to dynamically rotate your design and control views.

Now adjust the ambient and direct lighting of your shaded part.

Ambient light provides constant illumination in the drawing environment. It has no particular source or direction. You can adjust the intensity of ambient light. Keep ambient light low to prevent washing out your image.

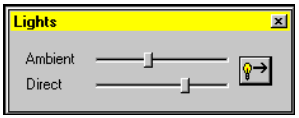
Direct light illuminates your image from a specified direction. You can adjust the intensity and direction of direct light.

To control the lighting of a shaded part

1 Use AMLIGHT to adjust the intensity of ambient and direct light.

Toolbutton Lighting Control

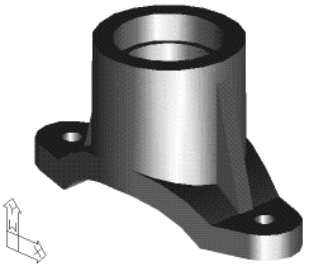
In the Lights dialog box, use the slider bars to adjust the intensity of the ambient light and the direct light as follows.



2 Use AMLIGHTDIR to specify a direction for direct light.

In the Lights dialog box, click the Light Direction button. Respond to the prompt as follows:

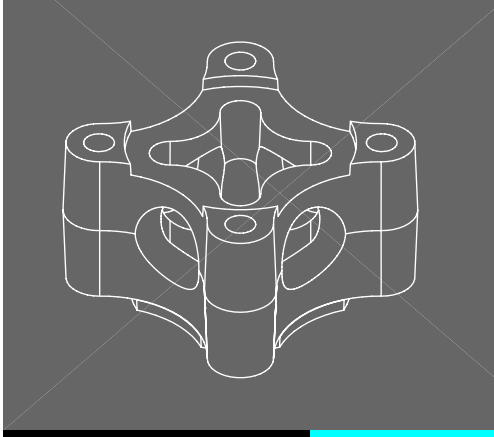
Select a point that will be used with the current target point for light direction:
Specify a point in the upper left of the graphics area



The light adjustments are reflected in your drawing. Experiment with other light settings.

Save your file.

Creating Drawing Views



In This Chapter

13

Autodesk® Mechanical Desktop® simplifies both the drawing and the documentation of your design. Drawing views are associated with a part and with one another. You lay out drawing views in any position on a screen. You can move them and make changes easily. Most dimensions are automatically placed on the views when you create them, but you can easily add missing reference dimensions and other annotations. In this tutorial, you will learn to use the Drawing tab in the Desktop Browser to manage and edit drawing views.

- Planning and setting up your drawing
- Multiple document layouts
- Creating drawing views
- Hiding extraneous dimensions
- Moving dimensions to a different view
- Customizing annotations
- Relocating and editing views

Key Terms

Term	Definition
balloon	A circular annotation tag that ties components in an assembly into a bill of material.
base view	The first view you create. Other views are derived from this view.
Desktop Browser	A graphical representation of the features that make up your model. You can work in the Browser to create and restructure parts and assemblies, define scenes, create drawing views, and control overall preferences.
Drawing mode	Establishes the settings for paper space so that you can create a drawing of your model. When Drawing mode is off, you are in model space.
hidden line	A line that is not visible in a specified view. For example, in a front view, lines behind the front plane are not visible.
Model mode	Creates 3D models on which drawing views are based.
parametric dimension	A dimension created during the sketch phase of feature creation. Parametric dimensions control size and update a part when you change its values.
parent view	A view on which to base another drawing view. For example, the base view is the parent view for auxiliary and orthographic views. Any view can be the parent view for a detail view.
reference dimension	An annotation dimension placed on a Mechanical Desktop drawing. These measurements do not control the size of the object. Instead, reference dimensions are required for manufacturing. They are updated when the geometry changes.
view scale	The scale of a base drawing relative to the model scale. Also the scale of dependent views relative to the base view.

Basic Concepts of Creating Drawing Views

Drawings and documentation are often the true products of design because they guide the manufacture of a mechanical device.

Mechanical Desktop adds an important dimension to drawing creation by doing most of the work for you. Traditional 2D orthographic, isometric, auxiliary, section, and detail views of parts and assemblies can be automatically created.

Mechanical Desktop creates these views complete with dimensions derived from the models. You can then add annotations or more dimensions. Because the views are derived from the models, they are updated as you make changes to your design.

In Mechanical Desktop, you can set up multiple layouts for complex models that require more than one drawing sheet to document.

Planning and Setting Up Drawings

Before you create drawing views, plan the views you need. Set up dimensioning and text styles and the drafting standard for your dimensions and other annotations.

To customize your drawing, use system parameters to specify drawing characteristics. You can define parameters in a prototype drawing so that they are automatically set before you begin a new project.

For this exercise, open the file *partview.dwg* in the *desktop\tutorial* folder. The file contains the saddle bracket model from the previous tutorial. It has been created in a prototype drawing that contains a predefined drawing style and title block.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

Creating Drawing Views

The first view you create is a base view. In Model mode, you specify the orientation of the view, and then change to Drawing mode to position it on the page. A title block and drawing border have been placed on the TITLE_BLK layer.

When you place the base view, hidden lines are removed. Parametric dimensions are shown according to the currently-defined dimension style.

By default, a dimension is shown in one view only—the first view you create displays the object that the dimension references. You can specify that parametric dimensions be displayed in other views as they are created, or you can move a dimension, if you prefer to show it on a different view.

When drawing views are created, by default their size is determined by the size of the data set. However, you can manually grip edit a viewport border if Parametric Border Sizing is turned off in the Edit Drawing View dialog box. If you turn Parametric Border Sizing back on, parametric sizing is restored.

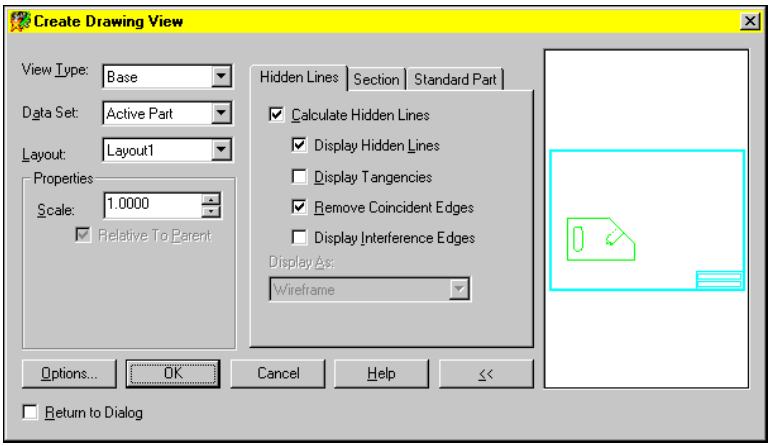
First, define a front view as a base view for the drawing. Before you begin this exercise, select the Drawing tab to change to Drawing mode.

To create a base view

1 Use AMDWGVIEW to define the base view. Verify that the Drawing tab is selected in the Browser.

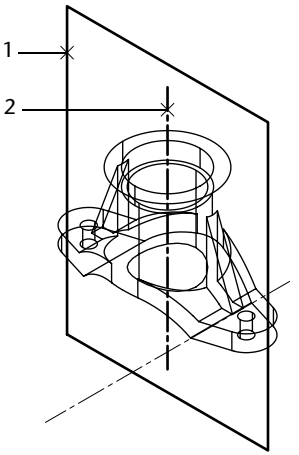
Context Menu In the graphics area, right-click and choose New View.

In the Create Drawing View dialog box, choose OK to accept the default options.



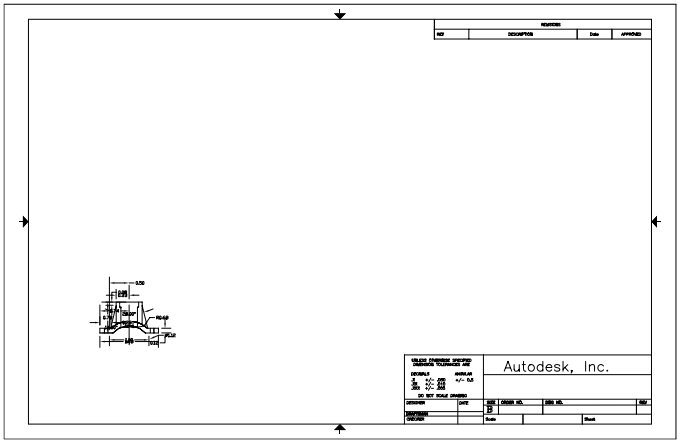
2 Respond to the prompts as follows:

Select a planar face, work plane, or [Ucs/View/worldXy/worldYz/worldZx]:
Specify the work plane (1)
Define X axis direction:
Select work axis, straight edge or [worldX/worldY/worldZ]:
Specify the axis of revolution (2)
Adjust orientation [Flip/Rotate] <Accept>:
Enter **r** *until the UCS icon is upright, or press* ENTER



3 On the command line, define a location on the drawing for the base view.

Specify location of base view:
Specify a point in the lower-left corner, inside the drawing border
Specify location of base view: *Press* ENTER



The base view is placed at the location you selected.

The Desktop Browser displays a hierarchy of the views you create. Because you have only the base view, it is listed below the part. As you create views from the base view, they are nested beneath the base view in the Browser.

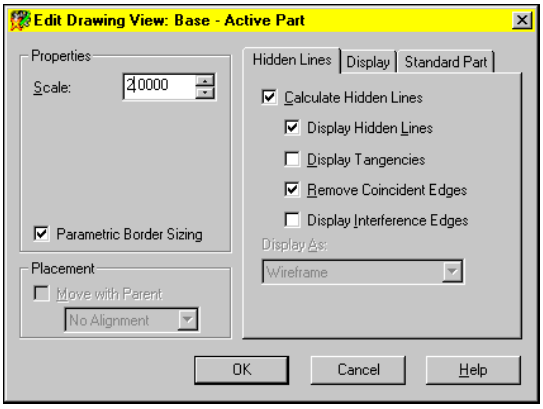
Because the base view is too small to be easily read, enlarge it by changing the view scale. Subsequent views will use the enlarged view scale until you specify a different one.

4 Use AMEDITVIEW to edit the scale of the base view.

Context Menu In the graphics area, right-click and choose Edit View.

Select the view you created.

In the Edit Drawing View dialog box, specify a scale of **2**, and choose OK.



Next, create an orthographic top view for the part.

When projected orthographically from the front view, the top view is aligned horizontally or vertically with the base view and maintains the same scale.

To create a top and detail view

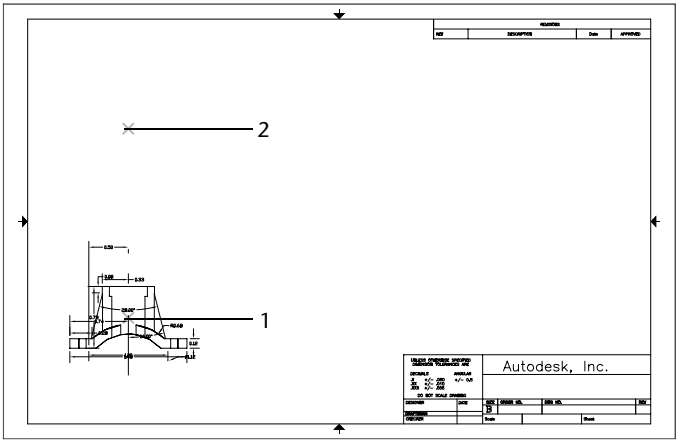
1 Create the orthographic view.

Context Menu In the graphics area, right-click and choose New View.

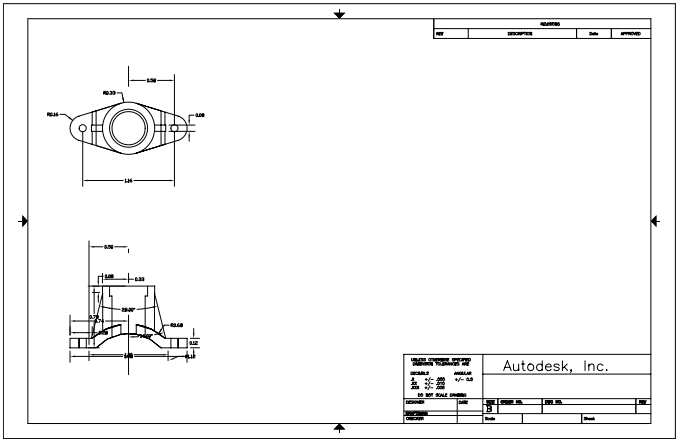
In the Create Drawing View dialog box, specify the Ortho view type and Choose OK.

2 Define a location for the orthographic view, responding to the prompts.

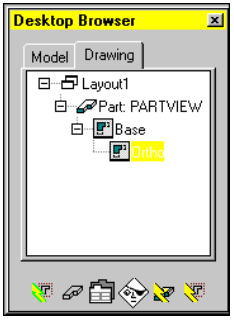
Select parent view: *Specify the front view (1)*
Specify location for orthogonal view:
Specify any location above the front view, within the drawing border (2)
Specify location for orthogonal view: *Press* ENTER



Your drawing should now look like this.



Because the orthogonal view is created from the base view, it is nested below the Base icon in the Desktop Browser.



Next, create an independent detail view of one of the lugs. Properties of independent detail views can be changed without affecting the properties of the parent view.

To create a detail view, choose the parent view and the area in the parent to show in detail. In this case, create a detail view of the rightmost mounting lug. For detail views, you always define the viewport border. The border is not controlled parametrically by the size of the part or geometry.

To create an independent detail view

1 Use AMDWGVIEW to create the detail view.

Context Menu In the graphics area, right-click and choose New View.

In the Create Drawing View dialog box, specify:

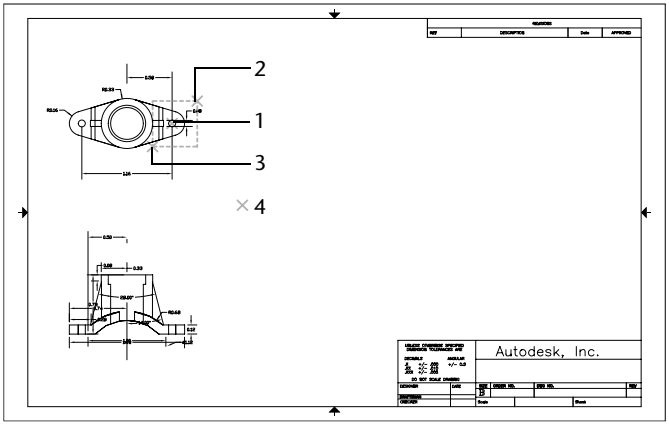
View Type: Detail
Scale: *Enter* **1.75**
Relative to Parent: *Select the check box*
Detail Symbol: *Enter* **A**
Label Pattern: *Enter* **VIEW**
Independent View Display: *Select the check box*

Choose OK.

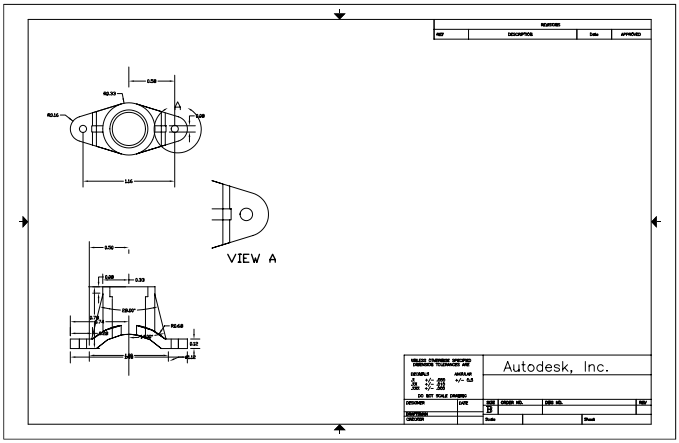
2 Define the detail view, responding to the prompts.

Select vertex in parent view to attach detail:
Specify the center of the rightmost lug (1)
Specify center point for circular area or [Ellipse/Polygon/Rect/Select]: *Enter* **r**
Specify first corner of rectangular area:
Specify the first point of the selection rectangle (2)
Specify opposite corner: *Specify the second point of the selection rectangle (3)*
Specify location for detail view:
Specify a point to the lower right of the top view (4)
Specify location for detail view: *Press* ENTER

NOTE To facilitate selection, turn off Object Tracking and Object Snaps by clicking the buttons at the bottom of your screen. You may need to zoom in to select the circle.



Your drawing should look like this.



The Browser displays a Detail icon nested below the Ortho icon.

3 Use AMEDITVIEW to edit the edge properties of the detail view.

Context Menu In the graphics area, right-click and choose Edit View.

Select the detail view you created.

In the Edit Drawing View dialog box, select the Display tab, and select Edge properties.

4 Edit the detail view edge properties, responding to the prompts.

Enter an option (edge properties) [Remove all/Select/Unhide all] <Select>:
Press ENTER
Select Edges: *Specify the circular lug*
Select Edges: *Press* ENTER

In the Edge Properties dialog box, choose Color.

In the Select Color dialog box, select red, and press ENTER.

Choose OK to close the Edge Properties dialog box.

Choose OK to close the Edit Drawing View dialog box.

The lug color in the detail view changes to red. However, the lug color remains unchanged in the parent view.

For practice, create the same detail view using a circle for selection. Notice how the command line prompts change according to the selection type you use.

Next, you create a cross section—a view that cuts through a point on the part along a work plane, or if the part is an offset section, through a sketch. Work planes are often easier to visualize and select than cutting planes.

If you choose not to create a work plane, you will find it easier to select only the endpoints of edges and the centers of circles or arcs to specify a cutting plane.

In this tutorial, you will create a work plane for the cross-section view, using an axis and an existing work plane.

To create a cross-section and isometric view

1 Return to Model mode.

Browser Select the Model tab.

2 Use AMWORKPLN to create a work plane for the cross-section view.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Plane.

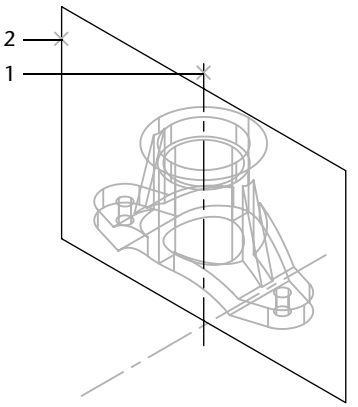
In the Work Plane dialog box, specify:

1st Modifier: On Edge/Axis
2nd Modifier: Planar Normal
Create Sketch Plane: *Clear the check box*

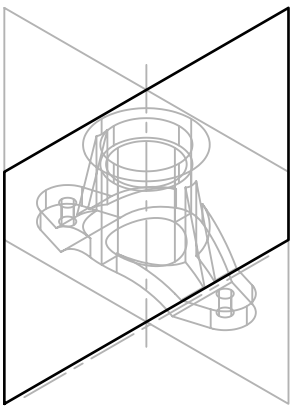
Choose OK.

3 On the command line, respond to the prompts as follows.

Select work axis, straight edge or [worldX/worldY/worldZ]:
Specify the work axis (1)
Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Specify the work plane (2)



Your model should now look like this.



Next, you create a full cross-section view of the part that is an orthographic projection of the front view.

4 Return to Drawing mode.

Browser Select the Drawing tab.

5 Create a new drawing view.

Context Menu In the graphics area, right-click and choose New View.

In the Create Drawing View dialog box, specify:

View Type: Ortho

Choose the Section tab and specify:

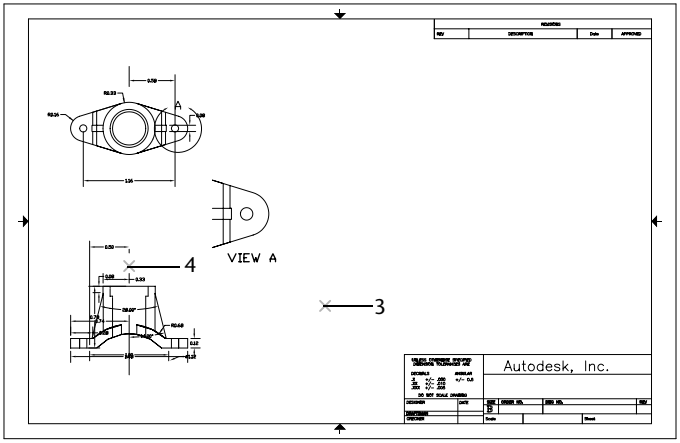
Type: Full
Label: *Enter* **A**
Label Pattern: **Section A-A**
Hatch: *Select the check box, and press Pattern*

Use the Hatch Pattern dialog box to define the hatch pattern, and choose OK.

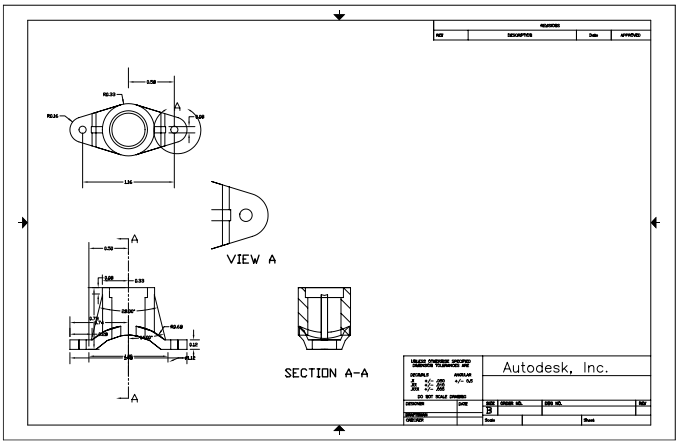
Choose OK to close the Create Drawing View dialog box.

6 Define the orthogonal view, responding to the prompts.

Select parent view: *Specify a point anywhere inside the front view*
Specify location for orthogonal view:
Specify a point to the right of the front view (3)
Specify location for orthogonal view: *Press* ENTER
Enter section through type [Point/Work plane] <Work plane>:
Press ENTER
Select work plane in parent view for the section:
Select the edge of the second work plane at a point inside the view box (4)



Your display should now look like this.



7 Create an isometric view, using the base view as the parent view.

Context Menu In the graphics area, right-click and choose New View.

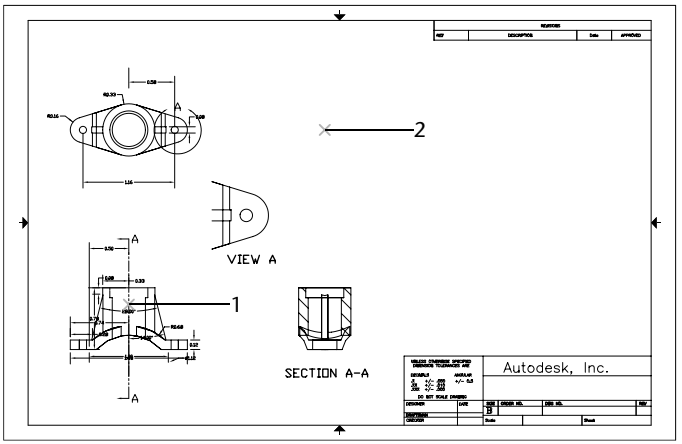
In the Create Drawing View dialog box, specify:

View Type: Iso
Scale: *Enter* **1**
Relative to Parent: *Select the check box*

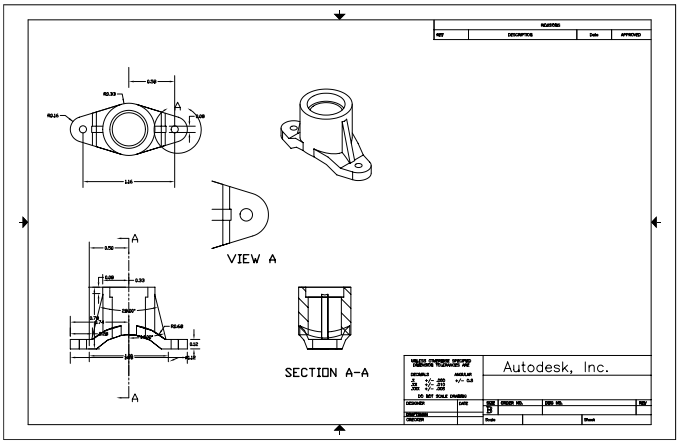
Choose OK.

8 Define the isometric view, responding to the prompts.

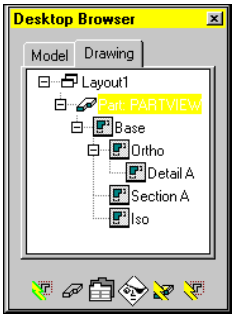
Select parent view: *Specify the base view (1)*
Location for isometric view: *Specify a point to the right of the top view (2)*
Location for isometric view: *Press* ENTER



Your drawing should look like this.



Each drawing view is represented as it relates to other views. For example, the ortho, section, and iso views are derived from the base view. Also, it is clear that the detail view is based on the ortho view. Detail and section views are named according to the labels you specified.



Save your file.

Cleaning Up Drawings

After creating the drawing views, you need to clean up the parametric dimensions and some extraneous lines.

Parametric dimensions are automatically placed on the AM_PARDIM layer.

Hiding Extraneous Dimensions

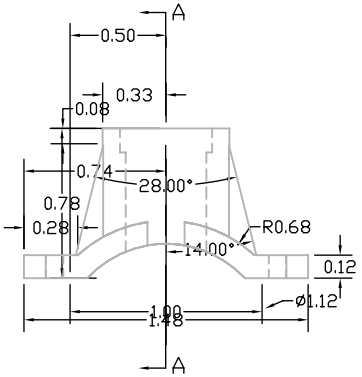
Because dimensions originate on the model, some might be redundant or conflict with others. For example, because the saddle bracket is symmetrical, one dimension states the overall length of a feature while another states the length of one side. Only one of these dimensions is necessary because the other can be derived. Decide which dimensions to show, and then selectively hide the others. Hiding dimensions does not delete them. They can be redisplayed from the Desktop Visibility dialog box.

Other dimensions may be redundant because you created the model by constructing individual features. For example, when you sketched the arc that represents the rough saddle form, you specified a radius of .33. This dimension appears in the top view of the drawing. When you created the boss, you specified a dimension of .33 to revolve the boss. This dimension appears in the front view of the saddle bracket. Only one of the .33 dimensions is needed.

To hide extraneous dimensions in a front view

1 Zoom to the base view.

Context Menu In the graphics area, right-click and choose Zoom.



2 Activate the Desktop Visibility dialog box.

Desktop Menu Drawing ➤ Drawing Visibility

In the Desktop Visibility dialog box, verify that the Hide option is selected. Then choose Select.

3 On the command line, respond to the prompts to select the redundant .33 and .74 dimensions to hide.

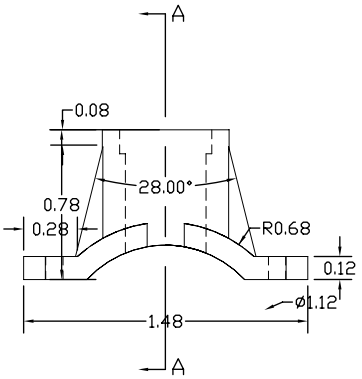
Select drawing objects to hide: *Specify the 0.33 dimension*
Select drawing objects to hide: 1 found
Select drawing objects to hide: *Specify the 0.74 dimension*
Select drawing objects to hide: 1 found, 2 total

The view also contains a number of dimensions associated with the rib sketch. The ribs were created from a trapezoid shape, where only two of the sides are used by the part. The other sides are not visible, so their dimensions should not appear in the drawing.

Select drawing objects to hide: *Specify the 1.00 dimension*
Select drawing objects to hide: 1 found, 3 total
Select drawing objects to hide: *Specify the 0.50 dimension*
Select drawing objects to hide: 1 found, 4 total
Select drawing objects to hide: *Specify the 14° dimension*
Select drawing objects to hide: 1 found, 5 total
Select drawing objects to hide: *Press* ENTER

Choose OK to exit the dialog box.

4 Your display should look like this.

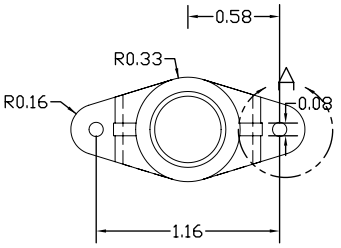


In the top view, the 1.16 dimension specifies the distance between arc centers. You can hide the extraneous .58 and 0.08 dimensions.

To hide extraneous dimensions in a top view

1 Zoom to the top view.

Browser Right-click Ortho and choose Zoom to.



2 Activate the Desktop Visibility dialog box.

Desktop Menu Drawing ➤ Drawing Visibility

3 In the Desktop Visibility dialog box, verify that the Hide check box is selected and choose Select.

4 Respond to the prompts as follows.

Select drawing objects to hide: *Specify the 0.58 dimension*
Select drawing objects to hide: *Specify the 0.08 dimension*
Select drawing objects to hide: *Press* ENTER

Choose OK to exit the dialog box.

The dimensions should be hidden on the view.

Moving Dimensions

Mechanical Desktop places dimensions on the drawing according to the way they were created during sketching. Usually, some cleanup is required, to comply with drafting standards.

In the following exercises, you will move dimensions within and between views until all the dimensions needed to define the part are visible on the drawing.

All the dimensions for the drawing currently exist in the front and top views. Originally these views were cluttered with extraneous dimensions. Now that those dimensions are gone, it is much easier to move the remaining dimensions to other views.

To move a dimension within a view

1 Zoom to the base view.

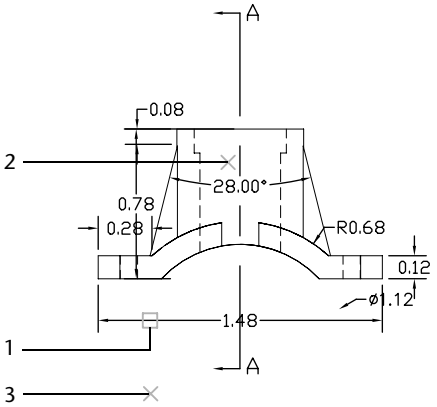
Browser Right-click Base and choose Zoom to.

2 Use AMMOVEDIM to move some dimensions to clean up your view, following the prompts.

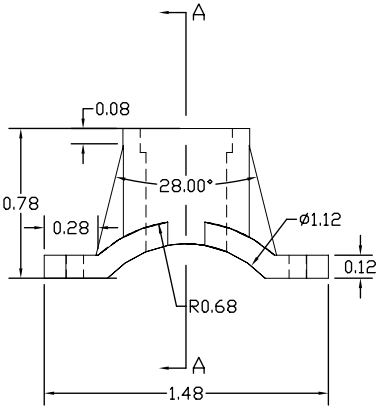
Context Menu In the graphics area, right-click and choose Annotate Menu ➤ Edit Dimensions ➤ Move Dimension.

Enter an option [Flip/Move/move mUltiple/Reattach] <Move>: *Press* ENTER
Select dimension to move: *Specify the 1.48 dimension (1)*
Select destination view: *Specify a point near the center of the front view (2)*
Select location: *Specify a point slightly below the A for the section cut (3)*
Select location: *Press* ENTER

Press ENTER to repeat the command.



3 Continue moving dimensions until the front view looks like this.



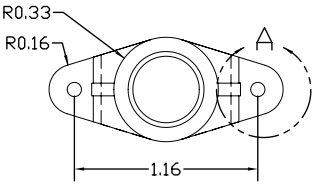
4 Zoom to the top view.

Browser Right-click Ortho and choose Zoom to.

5 Use AMMOVEDIM to move some of the dimensions in the top view.

Context Menu In the graphics area, right-click and choose Annotate Menu ➤ Edit Dimensions ➤ Move Dimension.

Follow the command line prompts to move dimensions until your view looks like this.



Because dimensions are placed on the first true-size view of the part, most dimensions clutter the first few views you create. In this exercise, you move a dimension from the front view to its cross-section view.

6 Zoom to return to the drawing layout.

Context Menu In the graphics area, right-click and choose Zoom. Right-click again, and choose Zoom Extents. Right-click again and choose Exit to close the command.

To move a dimension to a different view

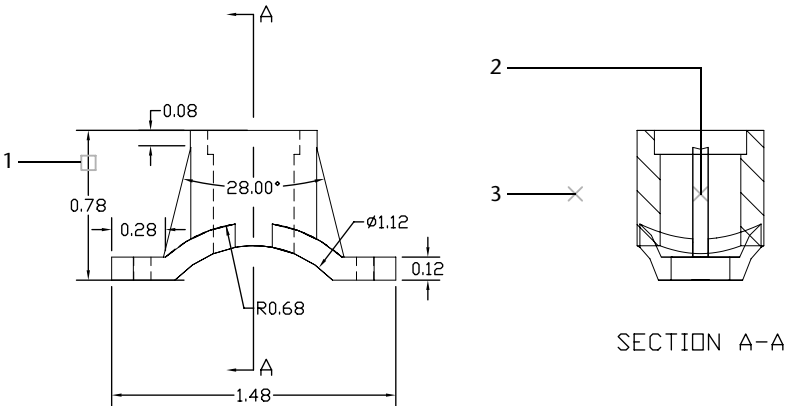
1 Zoom in to view the front and cross-section views.

Context Menu In the graphics area, right-click and choose Zoom.

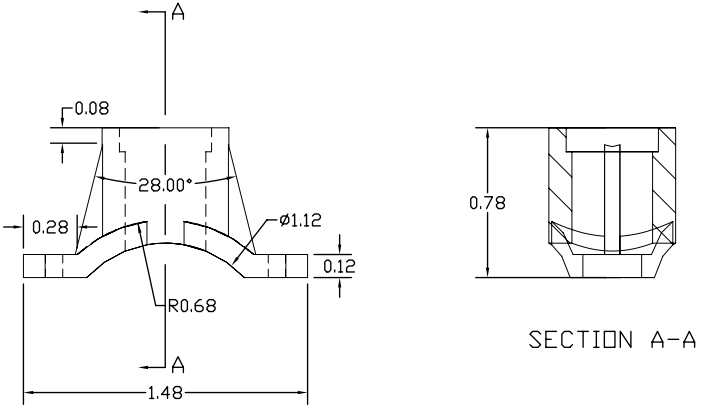
2 Use AMMOVEDIM to move a dimension from the front view to the cross-section view, following the prompts.

Context Menu In the graphics area, right-click and choose Annotate Menu ➤ Edit Dimensions ➤ Move Dimension.

Enter an option [Flip/Move/move mUltiple/Reattach] <Move>: *Press* ENTER
Select dimension to move: *Specify the 0.78 dimension (1)*
Select destination view: *Specify the cross-section view (2)*
Select location: *Place the dimension to the left of the cross-section view (3)*
Select location: *Press* ENTER



Your drawing views should look like this.



Hiding Extraneous Lines

Although Mechanical Desktop eliminates lines when it creates views, you may want to edit the views to remove additional, unwanted lines.

To hide an extraneous line

1 Zoom to the isometric view.

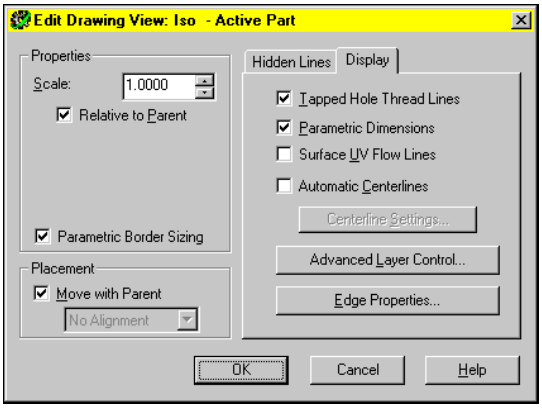
Browser Right-click Iso and choose Zoom to.

2 Use AMEDITVIEW to edit the Iso view.

Context Menu In the graphics area, right-click and choose Drawing Menu ➤ Edit View.

3 Specify the isometric view.

4 In the Edit Drawing View dialog box, choose the Display tab and then choose Edge Properties.

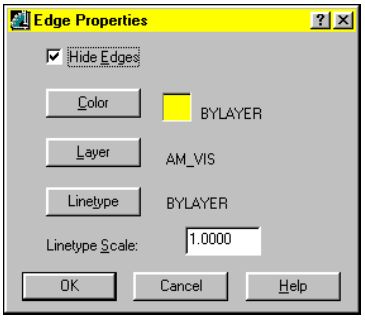


5 On the command line, respond to the prompts as follows:

Enter an option (edge properties) [Remove all/Select/Unhide all] <Select>:
Press ENTER
Select Edges: *Specify the vertical line on the lug (1)*
Select Edges: *Specify the vertical line on the lug (2)*
Select Edges: *Press* ENTER

6 In the Edge Properties dialog box, specify:

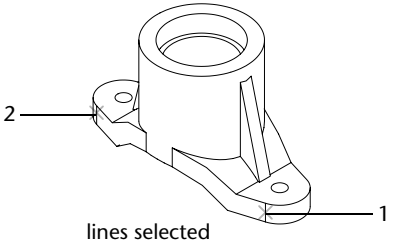
Hide Edges: *Select the check box*



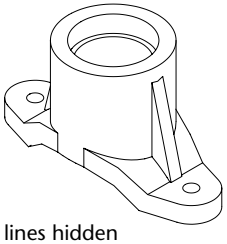
Choose OK.

7 Choose OK to exit the Edit Drawing View dialog box.

The selected lines are removed from the view.



lines selected



lines hidden

Enhancing Drawings

When you are satisfied with the drawing views, you can modify and enhance them. Enhancements include:

- Adding more dimensions
- Adding annotations such as callouts, hole notes, and centerlines
- Relocating views
- Modifying the part from the drawing view

Changing Dimension Attributes

Even though you set up the dimension style before creating the dimensions, some dimensions may need to be displayed in a particular way.

To edit a dimension attribute

1 Zoom to the front and cross-section views.

Context Menu In the graphics area, right-click and choose Zoom.

2 Use AMPOWEREDIT to edit a dimension on the front view to show a tolerance range, responding to the prompts.

Context Menu In the graphics area, right-click and choose Annotate Menu ➤ Edit Dimensions ➤ Power Edit.

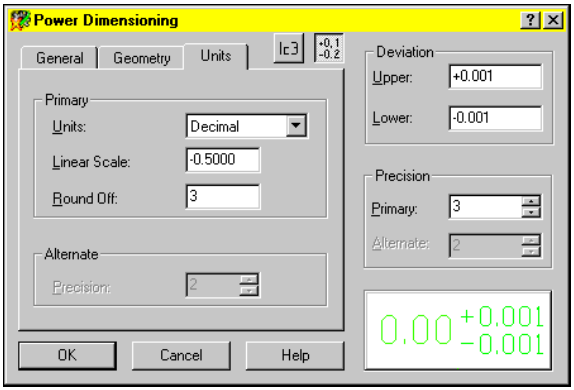
Select object: *Specify the .12 dimension in the front view*

3 In the Power Dimensioning dialog box, choose the Units tab and specify:

Units: Decimal
Round Off: *Enter* **3**

Select the Add Tolerance button in the upper right of the dialog box and specify:

Upper: *Enter* **+0.001**
Lower: *Enter* **-0.001**



4 Choose the General tab.

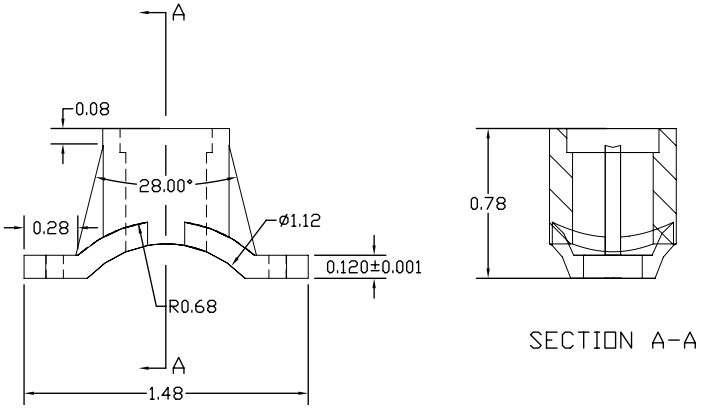
The .12 dimension should now be expressed as 0.120 +/- .001. Now that the dimension is longer, it may overlap the drawing view.

Choose OK.

5 Move the dimension so that it does not overlap any geometry.

Context Menu In the graphics area, right-click and choose Edit Dimensions ➤ Move Dimension.

Move the dimension so that it looks like this.



Creating Reference Dimensions

You can supplement parametric dimensions with reference dimensions. The reference dimensions do not control the size of the model; however, if you change the model, the reference dimensions are updated to reflect the new size. Reference dimensions reside on the AM_REFDIM layer.

In the next exercise, you add a reference dimension to the front view.

To add a reference dimension

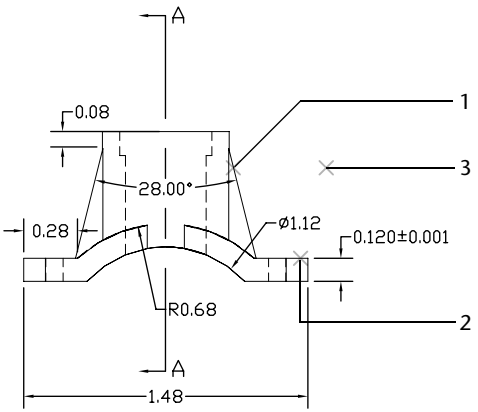
1 Zoom to the front view.

Browser Right-click Base and choose Zoom to.

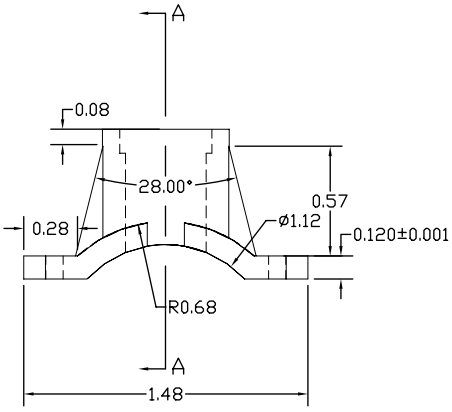
2 Use AMREFDIM to dimension the vertical distance from the top of the rightmost rib to the top of the rightmost lug, responding to the prompts.

Context Menu In the graphics area, right-click and choose Annotate Menu ➤ Reference Dimension.

Select first object: *Specify a point at the top of rib (1)*
Select second object or place dimension: *Specify a point at the top of lug (2)*
Specify dimension placement: *Specify a point to place the dimension (3)*
Specify placement point or [Undo/Hor/Ver/Align/Par/aNgle/Ord/reF/Basic]:
Enter **v** *to force a vertical dimension, or press* ENTER
Select first object: *Press* ENTER



points selected



reference dimension added

NOTE You can move some of the dimensions to avoid a cluttered view.

Creating Hole Notes

Mechanical Desktop does not automatically display hole dimensions on the drawing, but you can add this information.

First, you add a hole note to the boss in the top view, and tapped hole information to the mounting hole in the detail view.

To create a hole note

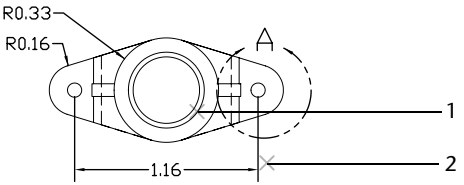
1 Zoom to the top view.

Browser Right-click Ortho and choose Zoom to.

2 Use AMNOTE to create a hole note for the hole through the boss, responding to the prompts.

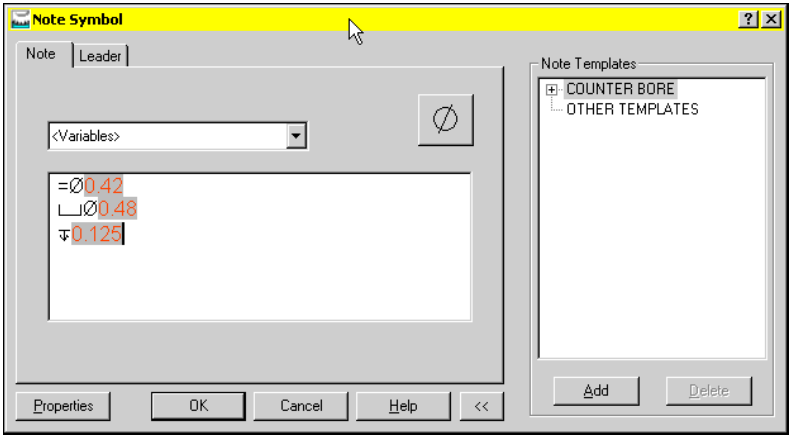
Context Menu In the graphics area, right-click and choose Annotate Menu ➤ Annotation ➤ Hole Note.

Select object to attach [rEorganize]: *Specify one of the two inner circles (1)*
Next Point <Symbol>: *Specify the location (2), and press* ENTER



3 In the Note Symbol dialog box, choose the more button to display the Note Templates section.

In Note Templates, choose the COUNTER BORE template.



4 Select the Leader tab, and set the leader justification to Middle of All Text.

Choose OK.

A hole note with the hole diameter, the counterbore diameter, and the hole depth is displayed on your drawing.

Next, add hole information to the mounting hole in the detail view. This procedure is similar to adding standard hole note information, except that you include additional information when you create the hole. You edit the text in the hole note template, but it applies to that hole note only and does not alter the template.

To create a modified hole note

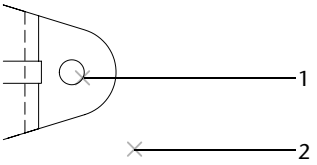
1 Zoom to the detail view.

Browser Right-click Detail and choose Zoom to.

2 Create the hole note, responding to the prompts.

Context Menu In the graphics area, right-click and choose Annotation ➤ Hole Note.

Select object to attach [rEorganize]: *Specify the hole in detail view (1)*
Next Point <Symbol>: *Specify the location (2), and press* ENTER.



3 In the Note Symbol dialog box, choose the more button to display the Note Templates section.

In Note Templates, choose the THRU HOLE template.

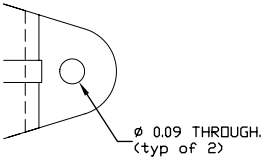
The Multiline Text Editor is displayed.

4 In the Multiline Text Editor, place the cursor at the end of the existing text and press ENTER. Add **(typ of 2)** on the second line.

Select the Leader tab, and set the leader justification to Middle of All Text.

Choose OK.

Your drawing should look like this.



Creating Centerlines

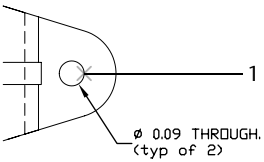
In this exercise, you create a parametric centerline for the top view and a center mark for the detail view. Centerlines and center marks are attached to the view and move with the view as the model changes.

To create a centerline

1 Add a center mark, responding to the prompts.

Context Menu In the graphics area, right-click and choose Annotation ➤ Centerline.

Select Edge: *Specify the hole in the detail view (1)*
Select mirrored edge or <ENTER>: *Press* ENTER



The size and characteristics of the center mark are defined by settings in the Centerline Properties dialog box. To change the settings, choose Drawing ➤ Drawing Options. On the Annotation tab, choose Centerline Settings, and choose the Center Line Properties button.

In the Center Line Properties dialog box, you can choose to use standard settings, or you can enter new values for overshoot and center mark size that will apply to this drawing only.

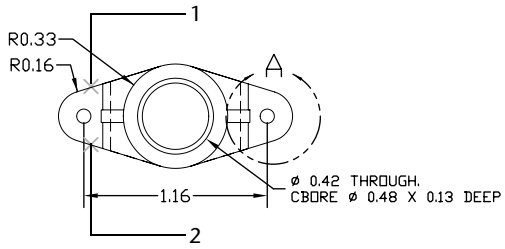
2 Zoom to the top view.

Browser Right-click Ortho and choose Zoom to.

3 Create another centerline, responding to the prompts.

Context Menu In the graphics area, right-click and choose Annotation ➤ Centerline.

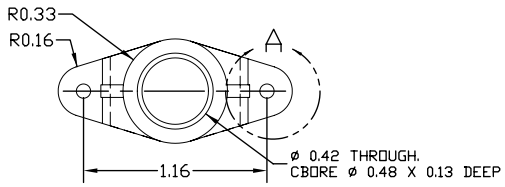
Select Edge: *Specify the top edge of the left lug (1)*
Select mirrored edge or <RETURN>: *Specify the lower edge of the left lug (2)*



A centerline is placed through the view. Now, specify where to trim the centerline endpoints.

Select first trim point: *Specify a point to the right of the part*
Select second trim point: *Specify a point to the left of the part*

Your display should look like this.



Creating Other Annotation Items

When you make changes to a model, the geometry and dimensions are updated automatically. Special commands create drawing annotations such as reference dimensions, hole notes, and centerlines.

You can create other annotations such as callout bubbles, surface finish symbols, and Geometric Distancing and Tolerancing (GD&T) symbols. The annotation items do not change when you make changes to the model. To make these annotation objects parametric, you convert them after you create them.

In the next exercise, you convert a callout bubble into a Mechanical Desktop annotation item. The callout bubble is already created and placed on the AM_ANNOTE layer of your drawing.

To convert a callout bubble

1 Zoom out to view the entire drawing.

Context Menu In the graphics area, right-click and choose Zoom. Right-click again and choose Zoom Extents. Right-click again and choose Exit.

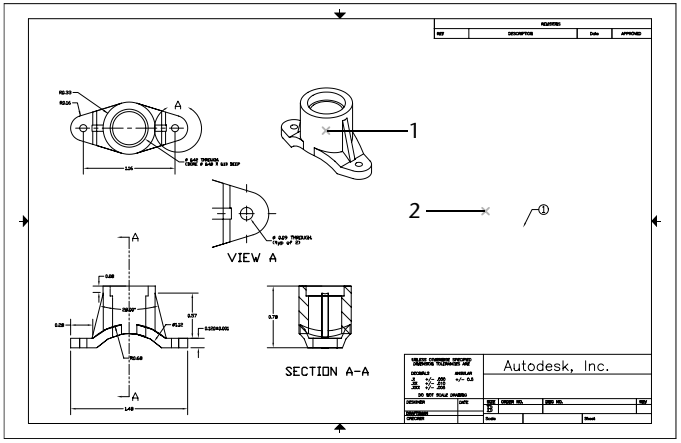
2 Use LAYER to turn on the AM_ANNOTE layer. You should see a callout bubble containing the number 1 and a leader.

3 Use AMMOVEVIEW to position the isometric view near the callout, responding to the prompts.

Context Menu In the graphics area, right-click and choose Drawing Menu ➤ Move View.

Select view to move: *Select the isometric view (1)*
Specify new view location: *Specify a point (2)*
Specify new view location: *Press* ENTER

If you use the Browser to move a view, you will not be asked to select it.



The callout does not move. Convert it to an annotation so that it is associated with the isometric view.

4 Convert the callout bubble to an annotation, responding to the prompts.

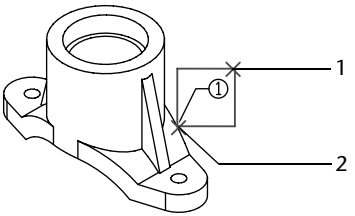
Context Menu In the graphics area, right-click and choose Annotation Menu ➤ Annotation ➤ Create Annotation.

Select objects to associate with view:

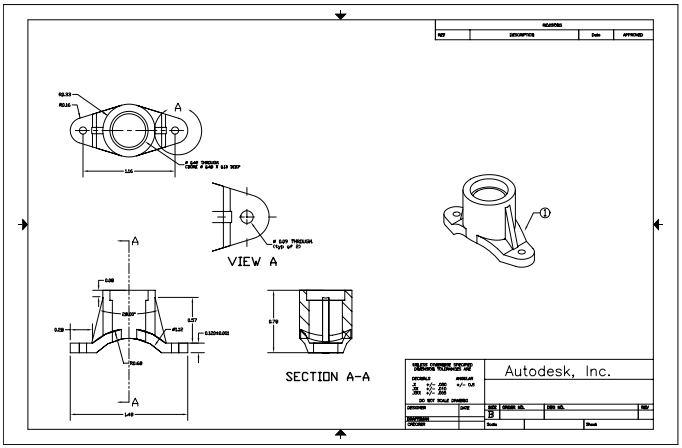
Draw a selection rectangle around the callout bubble, numeral, and leader (1, 2)

Select objects: *Press* ENTER

Select point in view to attach annotation: *Specify a point (2)*



Your drawing should look like this.



Modifying Drawing Views

You can relocate views or change the model from a drawing view. The drawing and, if appropriate, the model, are updated to reflect the changes you made.

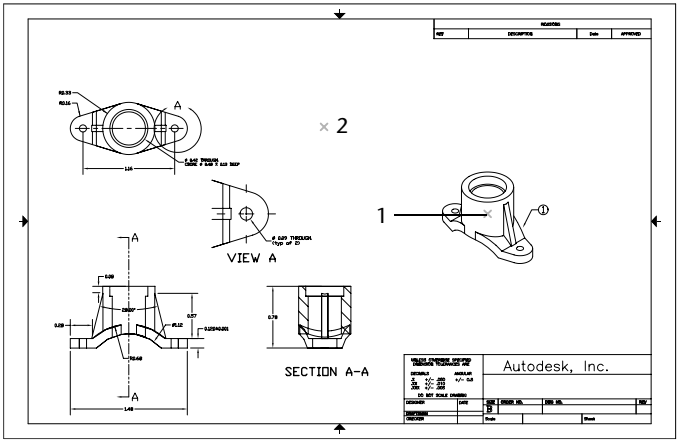
Move the isometric view. The callout bubble moves with the view because it is associated with the part. Then, relocate the isometric and detail views, and change the detail view. The model and all drawing views are updated.

To relocate a drawing view

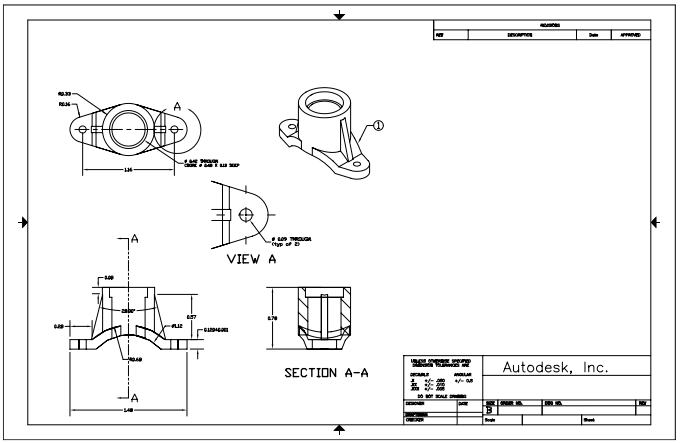
1 Move the Iso view back to its former location, responding to the prompts.

Context Menu In the graphics area, right-click and choose Move View.

Select view to move: *Specify center of isometric view (1)*
Specify new view location: *Specify new location (2) and press* ENTER
Specify new view location: *Press* ENTER



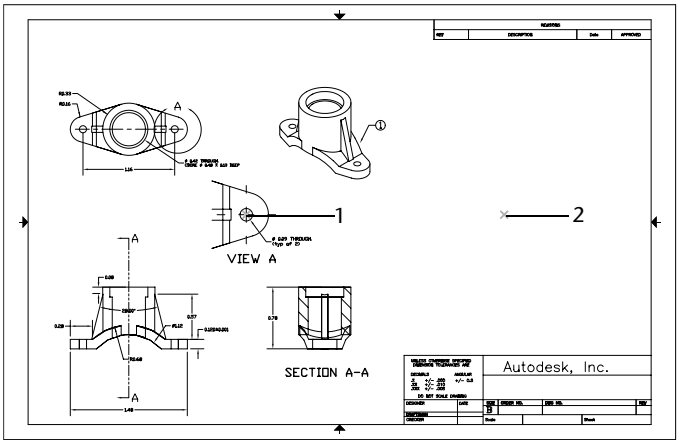
The callout bubble moves with the view.



2 Relocate the detail view to the right of and below the isometric view, responding to the prompts.

Context Menu In the graphics area, right-click and choose Move View.

Select view to move: *Specify center of the detail view (1)*
Specify new view location: *Specify new location (2) and press* ENTER
Specify new view location: *Press* ENTER



All annotations associated with the view move with it and keep their positions relative to the view. You can move views from layer to layer.

Now, change one of the parametric dimensions within a view and watch the resulting changes.

To modify a drawing view

1 Zoom to the top view.

Context Menu In the graphics area, right-click and choose Zoom.

2 Use AMMODDIM to change the radius of the lug, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Dimensions ➤ Edit Dimension.

Select dimension to change: *Select the .16 value of the lug radius*
New value for dimension <.16>: *Enter* **.13**
Select dimension to change: *Press* ENTER

You must update the part to show the changes.

3 Zoom out to display all the drawing views.

Context Menu In the graphics area, right-click and choose Zoom. Right-click again, and choose Zoom Extents. Right-click again and choose Exit.

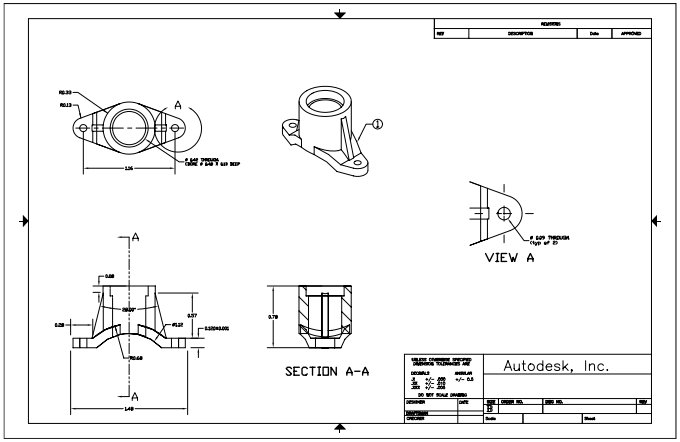
4 Use AMUPDATE to update your part.

Context Menu In the graphics area, right-click and choose Update Part.

Update part now? [Yes/No]: *Press* ENTER

In the Update Dependent Part dialog box, choose Yes.

Mechanical Desktop updates the part, it also updates each drawing view. After it completes the updating, your drawing should look like this.



Save your file.

Exporting Drawing Views

You can save your 2D drawing views directly to Mechanical Desktop versions other than Release 6 as DWG, DWT, or DXF files. You can export the entire current layout, including all views and geometry, or you can select views and entities to export.

The Export Drawing Views dialog box provides options to:

- Convert views at true scale (1:1)
- Convert circular and linear splines that project to 2D arcs, circles and lines
- Flatten all source data, including any 3D autoCAD entities, or flatten only MDT objects

The export options in the Export Drawing Views dialog box enable you to export drawing views to:

- A new layout in the current file
- An external file in either model space or in layout
- Past and present versions of AutoCAD and DXF

Usually, only one base view is shown in a drawing view. With the True Scale option turned on, a warning is displayed if two or more base views are shown with different scales. You export base views separately, whether or not they are scaled by the same factor. In the exporting process, isometric and detail views are scaled by the same factor as the base view.

In this exercise, you save your drawing file to AutoCAD 2000.

To export Mechanical Desktop drawing views

1 Use AMVIEWOUT to save your drawing view file to AutoCAD 2000.

Browser In the Browser, right-click Base and choose Export View.

2 In the Export Drawing Views dialog box, specify the following:

Source: *Specify Current Layout*
Data Handling:
Export Views True Scale (1:1): *Select the check box*
Flatten All Selected Objects: *Select the check box*
Convert Circular/Linear Splines to Circles/Lines: *Select the check box*
Generate Preview Image: *Select the check box*
File Name: *Enter or browse to a file name for the new version*

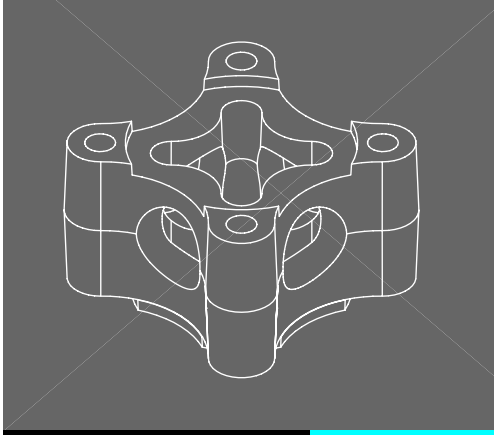
Press the More button (<<), and in Export Options, specify:

To New External File: *Select the check box, and select In Layout*
Save as type: *Specify AutoCAD 2000 Drawing (*.dwg)*

Choose OK.

The drawing view is saved into the first layout in the new external file.

Creating Shells



In This Chapter

14

This tutorial teaches you how to create and edit a shelled part in Autodesk® Mechanical Desktop®. With the shell feature, you can create complex parts with walls of varying thickness. In the tutorial, you add a shell feature to an existing die cast engine part, and then edit the shell. The procedures you learn here can be applied to a variety of shelled parts.

- Adding a shell feature to an existing part
- Modifying wall thickness
- Adding multiple thickness overrides
- Managing thickness overrides

Key Terms

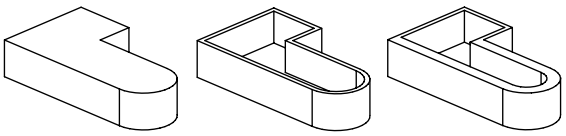
Term	Definition
converging radial shapes	A sharp corner where cylindrical faces, such as fillets, are offset and converge to form a zero radius. Parts with more complex shapes, such as variable radius fillets and surfcuts, need a shell thickness large enough so that the offset face does not converge.
default thickness	The offset value initially applied to all faces of your part.
excluded face	Face on a shelled part you select that will not be offset.
reclaimed face	Face on a shelled part that was previously excluded can be selected, or reclaimed. Reclaimed faces are offset by the default thickness.
shell	A Mechanical Desktop® feature that cuts portions of the active part by offsetting its faces.
thickness override	An offset value that takes precedence over the default thickness of a shell feature. You can define a list of thickness override values and apply them to any face on the part.

Basic Concepts of Creating Shells

Unlike other sketched or placed features, a shell feature is initially applied to all the faces of your active part, instead of only those you select. It doesn't need parametric dimensions to control placement. A part can have only one shell feature.

When you add a shell feature to a part, Mechanical Desktop creates new faces by offsetting existing ones inside or outside of their original positions. You can also choose the midplane option, which offsets faces by half the entered value to one side and half to the other side.

Mechanical Desktop treats continuously tangent faces as a single face when offsetting. This illustration shows the progression for adding a shell feature to a part and then a thickness override to the cylindrical face. The faces tangent to the cylindrical face are also offset in the operation.



You can change a shell feature in different ways. You can change the offset type and offset values in the Shell Feature dialog box. If you choose to exclude or reclaim faces, or change thickness overrides, you select faces on your part. In both cases, to apply your modifications you must update the part.

Adding Shell Features to Models

In this tutorial, you add a shell feature to the existing model of a clutch housing for a 250cc two-stroke engine. The shell feature requires an excluded face and multiple thickness overrides.

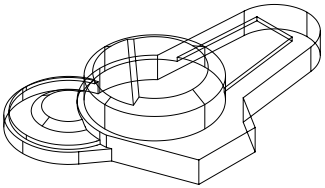
You apply what you've learned in previous chapters about constrained sketches and features to examine an existing part. You then add a shell feature to the part, later modifying it to suit your design requirements.

Using Replay to Examine Designs

First, review the clutch housing design.

Open the file *clutch.dwg* in the *desktop\tutorial* folder. The clutch housing has been modeled to a point in the design where it is ready for you to add the shell feature. It contains six extruded features.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.



For the constraint system in this model, construction lines were used to align its cylindrical features. You can change the angle of the features with respect to one another to alter the design to fit engines of different sizes.

To replay the design of a part

1 Use AMREPLAY to review the process used to build this model, responding to the prompts.

Context Menu In the graphics area, right-click and choose Part ➤ Replay.

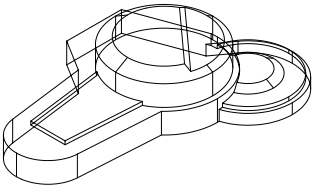
```
Closed profile
Enter an option [Display/Exit/Next/Size/Truncate] <Next>:
   Press ENTER to review each design step
Enter an option [Exit/Next/suPpress/Truncate] <Next>:
   Continue to press ENTER until you see the following message
Part replay complete
```

The construction of the model is also displayed in the Desktop Browser. Each feature is located in the order it was created. Expand the browser and examine the feature hierarchy.

2 Change your view to Back Right Isometric.

Desktop Menu View ➤ 3D Views ➤ Back Right Isometric

This displays the model in another isometric view.



Next, you remove the silhouette edges from your model so you can visualize it better. Silhouette edges are similar to hidden lines, but to remove them you need to modify a system variable.

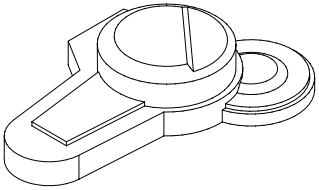
3 Change the system variable controlling the visibility of silhouette edges.

Command DISPSILH

New value for DISPSILH <0>: *Enter* **1**

4 Use HIDE to remove hidden lines from your display. This improves your view of the features of the housing.

Desktop Menu View ➤ Hide

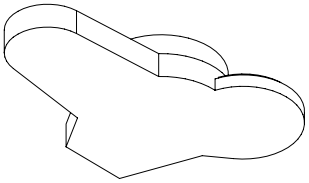


5 Use 3DORBIT to dynamically rotate the model, and view the underside of the housing.

Context Menu In the graphics area, right-click and choose 3D Orbit.

Click the bottom control point on the 3D orbit icon in the graphics area, and rotate the part upward.

6 Remove the hidden lines again. Your drawing should resemble the following illustration.



Cutting Models to Create Shells

Now that you have examined the model, the next step is to cut it, removing the interior area, so that all that remains is a shell of the part. This part is a magnesium alloy casting which requires a wall thickness of about 4 mm in most areas, but some walls must be thicker to withstand forces applied to them.

To cut a model

1 Return to the back right isometric view.

Desktop Menu View ➤ 3D Views ➤ Back Right Isometric

2 Return to a wireframe display of your model.

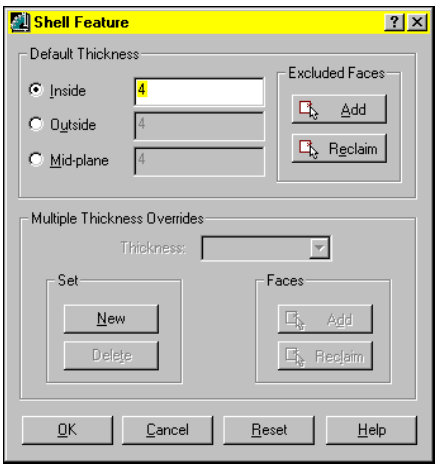
Desktop Menu View ➤ Shade ➤ 3D Wireframe

3 Use AMSHELL to create a shell feature.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Shell.

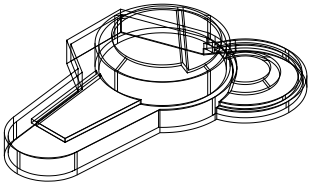
4 In the Shell Feature dialog box, specify:

Default Thickness: Inside: *Enter* **4**



Choose OK.

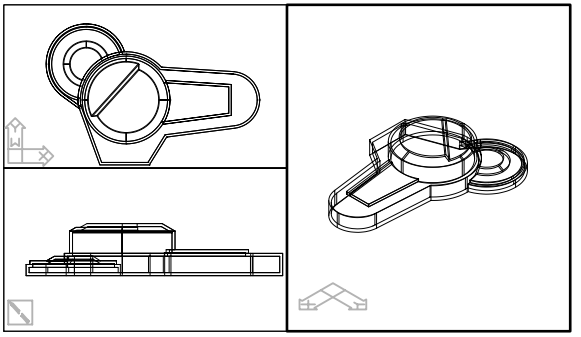
The shell feature is calculated and the model is updated.



5 Change your display to three viewports.

Command 3

This gives you a better view of the thickness of the walls in the model.



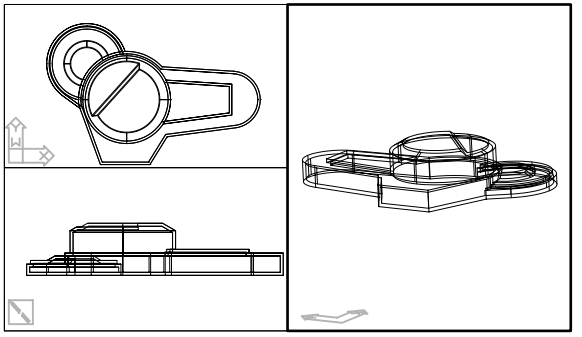
An isometric view of the bottom of the housing has been previously saved.

6 Click in the right viewport to make it current, and restore the saved view.

Desktop Menu View ➤ Named Views

7 In the View dialog box, highlight BOTTOM_ISOMETRIC and choose Set Current.

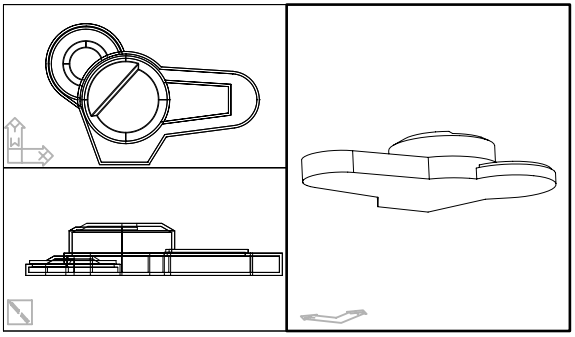
Choose OK. The right viewport changes to display the selected view.



When the shell feature was added, all the faces in the model were offset. The result is a hollow model. For a better view, suppress the hidden lines.

8 Use HIDE to remove the hidden lines.

Desktop Menu View ➤ Hide



Save your file.

Editing Shell Features

When you create a shell, your wireframe display becomes more complex because Mechanical Desktop offsets each face in your model, doubling the number of faces. One way to edit a shell feature is to use AMEDITFEAT and select an offset face edge. However, choosing the shell feature icon from the Desktop Browser is easier.

To remove the bottom face from the part, you need to edit the shell feature and exclude the bottom face.

To exclude a face on a shell feature

1 Use AMEDITFEAT to edit the shell feature.

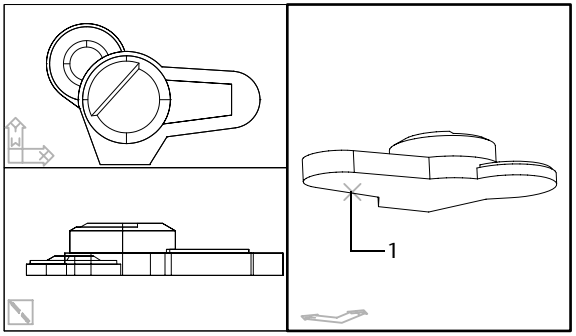
Browser In the Browser, right-click Shell1 and choose Edit.

If you choose a method other than the Browser, you must select the shell feature first. The Shell Feature dialog box is displayed.

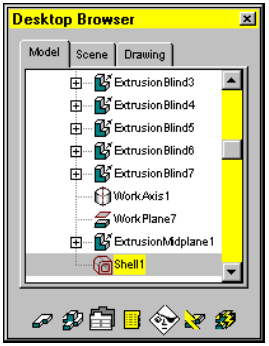
2 In the Shell Feature dialog box, choose Add and then respond to the prompts as follows:

Select faces to exclude: *Specify a point (1)*
Enter an option [Accept/Next] <Accept>:
Enter **n** *until the bottom face is selected or press* ENTER
Select faces to exclude: *Press* ENTER

Choose OK to exit the dialog box.



In the Browser, the shell icon has a yellow background indicating that it needs to be updated.

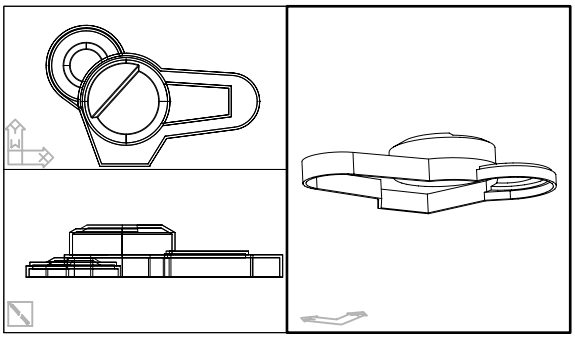


3 Use AMUPDATE to update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Enter an option [active Part/aLl parts] <active Part>: *Press* ENTER

The model is updated to reflect the modified shell feature.



Save your file.

Adding Multiple Wall Thicknesses

When the clutch housing is attached to an operating engine, the stresses are higher on some casting walls than on others. The walls surrounding the water pump shaft and the clutch support the most force. You need to thicken these walls to conform to design requirements.

Wall thickness overrides are applied to only those faces you select. A chain of tangent faces is treated as a single face.

You can have as many wall thickness overrides as you like in a shell feature, but most applications require only a few. Mechanical Desktop keeps track of which faces have thickness overrides.

To edit a shell through the Browser

1 Return to the back right isometric view.

Desktop Menu View ➤ 3D Views ➤ Back Right Isometric

The face surrounding the water pump can now be selected easily.

2 Return to a wireframe display of your model.

Desktop Menu View ➤ Shade ➤ 3D Wireframe

3 Edit the shell feature again.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Select the shell feature. The Shell Feature dialog box is displayed.

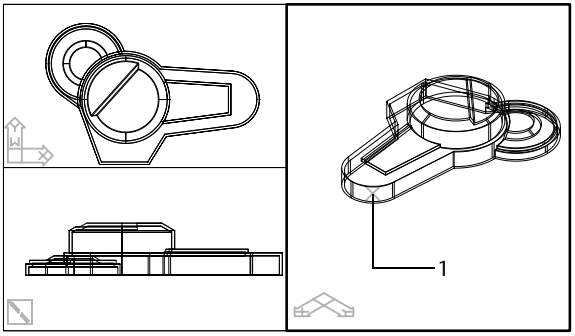
4 In the Shell Feature dialog box, in Multiple Thickness Overrides, specify:

Set: New
Thickness: *Enter* **8**
Faces: Add

On the command line, respond to the prompts as follows:

Select faces to add: *Specify a point (1)*
Select faces to add: *Press* ENTER

Choose OK to exit the dialog box.

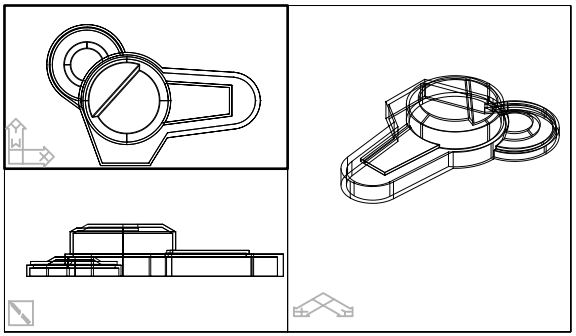


5 Update the part, responding to the prompt.

Context Menu In the graphics area, right-click and choose Update Part.

Enter an option [active Part/aLl parts] <active Part>: *Press* ENTER

Refer to the top view in the upper left viewport to see the results. When you selected the cylindrical face, tangent faces were automatically selected. The wall thickness surrounding the water pump should look twice as thick as the rest of the walls.



To create a multiple thickness override

1 In the right viewport, change to the left isometric view of the model so that the faces surrounding the clutch can be easily selected.

Desktop Menu View ➤ 3D Views ➤ Front Left Isometric

2 Use AMEDITFEAT to create a multiple thickness override for the walls surrounding the water pump.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Select any one of the edges of the inside shell walls. The Shell Feature dialog box is displayed.

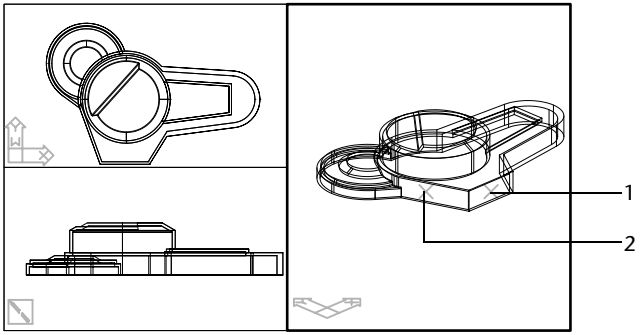
3 In the Shell Feature dialog box, specify:

Multiple Thickness Overrides: New
Thickness: *Enter* **6**

Choose Add, and respond to the prompts as follows:

Select faces to add: *Specify a point on the model (1)*
Select faces to add: *Specify a second point (2)*
Select faces to add: *Press* ENTER

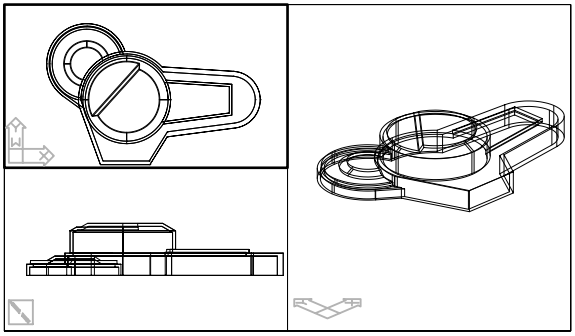
Choose OK to exit the dialog box.



4 Update the part.

Context Menu In the graphics area, right-click and choose Update Part.

In the top view, the two faces you selected are thicker.



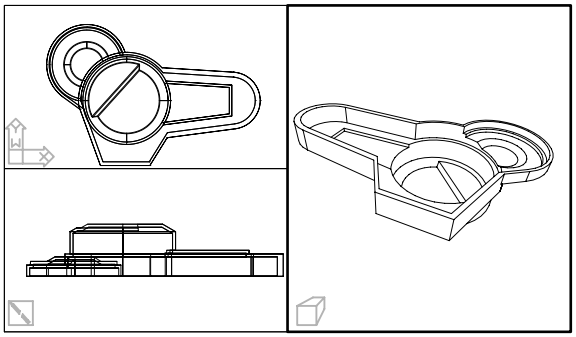
5 Make the right viewport active and restore another saved view.

Desktop Menu View ➤ Named Views

6 In the View dialog box, highlight BOTTOM_PERSPECTIVE, choose Set Current, and then choose OK.

7 Use HIDE to remove the hidden lines.

Desktop Menu View ➤ Hide



Save your file.

Managing Multiple Thickness Overrides

Occasionally, you may create complex parts that require more thickness override values. As you apply the override values to faces on your part, it is easy to lose track of which faces are using different overrides, especially if you are viewing a part that was designed by someone else.

However, Mechanical Desktop makes it easy to manage overrides and identify the faces with specific override values. You may have noticed that when you chose an override from the list in the Shell Feature dialog box, all faces with that override were highlighted.

Mechanical Desktop also audits the override list each time you edit the shell feature. Override values that are no longer used are removed from the list. This eliminates the need to manually delete obsolete overrides, which ensures that the list always reflects the wall thicknesses of your current part.

Design changes are also easy to implement. When you change an override thickness value in the Shell Feature dialog box, you change the wall thicknesses of all the faces that reference that override value.

For the clutch assembly, the wall thickness around the water pump can be reduced from 6 to 4. Because the default wall thickness of the shell is 4, you remove the override of 6 from the list. When you delete an override value from the list, faces that once referenced that value revert to the default thickness.

To change a wall thickness

1 Use AMEDITFEAT to edit the shell feature again to change the wall thickness around the water pump.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Select the shell feature. The Shell Feature dialog box is displayed.

2 In the Shell Feature dialog box, specify:

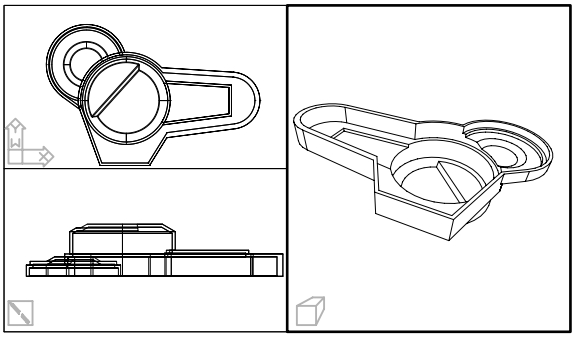
Thickness: *Enter* **6**
Set: Delete

Choose OK and then press ENTER on the command line.

3 Use AMUPDATE to update the part.

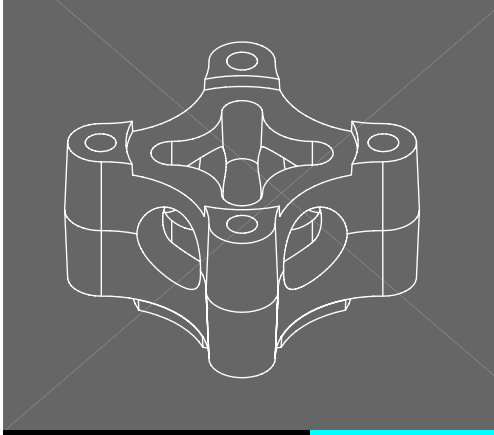
Context Menu In the graphics area, right-click and choose Update Part.

The model is updated to reflect the modified shell feature.



Save your file.

Creating Table Driven Parts



In This Chapter

15

- Creating a table
- Displaying part versions
- Editing a table
- Solving common problems with table driven parts
- Suppressing features in table driven parts
- Setting up a generic drawing to define the table driven part

You can assign variables to the parametric dimensions that control a generic part and then use a table (an external spreadsheet) to control the size and shape of the part. The spreadsheet can contain several versions of the part. Each version uses different values for the variables you define. Autodesk® Mechanical Desktop® redraws the part using the variables linked to that version.

Documentation for all versions can be simplified by plotting one drawing representing the generic part. You can paste the spreadsheet into your drawing to list the values for each version.

For this tutorial, you need Microsoft® Excel on your system.

Key Terms

Term	Definition
active part variable	A parametric variable used in the dimensions that control features of the active part.
feature suppression	Temporarily removing features from the calculation of a part. Features can be suppressed manually through the Desktop Browser, or through a linked external table.
global variable	A parametric variable that can be used by any number of parametric features and parts. Also used for single parts and to constrain parts.
linked spreadsheet	An external spreadsheet linked to the current drawing file. This spreadsheet can be used to change parts in the drawing file, if the variables in the spreadsheet are used in the drawing file.
manual suppression	Suppressing features through the browser or command line. These features will remain suppressed unless manually unsuppressed or a version of a linked table that unsuppresses it becomes active.
part version	The version of variables that the active part is currently using from the external table. Part versions can be changed through the Desktop Browser or the Design Variables dialog box.
table	An external spreadsheet that drives versions of a part.
table driven suppression	Features suppressed in an external spreadsheet. A table driven suppressed feature is suppressed only in the part version specified in the spreadsheet.
table driven variable	A global or active part design variable controlled by values in a linked external spreadsheet.

Basic Concepts of Table Driven Parts

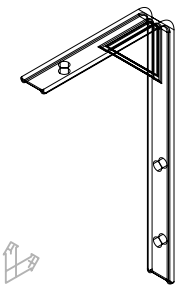
In the manufacturing industry, you often have parts that are similar to each other except for size or a particular feature. Some examples are springs, brackets, plates, nuts, and bolts. By driving part versions from an external spreadsheet, you can document a number of similar parts using one drawing.

The external spreadsheet, or table, is where you make modifications to your design specifications once your drawing is set up. By controlling a part from a table, you avoid errors due to design changes that have not been implemented across a number of drawings. All design data is contained in the table, with one drawing representing many parts.

Open the file *tdpart1.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing is a simple bracket that you link to a spreadsheet.



The part is controlled by its relationships to nine active part design variables. Changing the value of one variable affects every dimension that has a relationship to it. For information on creating and modifying design variables, see chapter 11, "Using Design Variables."

Setting Up Tables

A table is a spreadsheet that contains the various part versions and the values of the design variables for each version. You use Microsoft Excel to create table driven parts.

First, examine the part, using the Desktop Browser. Expand the Browser by clicking the plus sign in front of TDPART1 and PART1_1. The bracket is constructed from three extrusions, three holes, and five fillets.

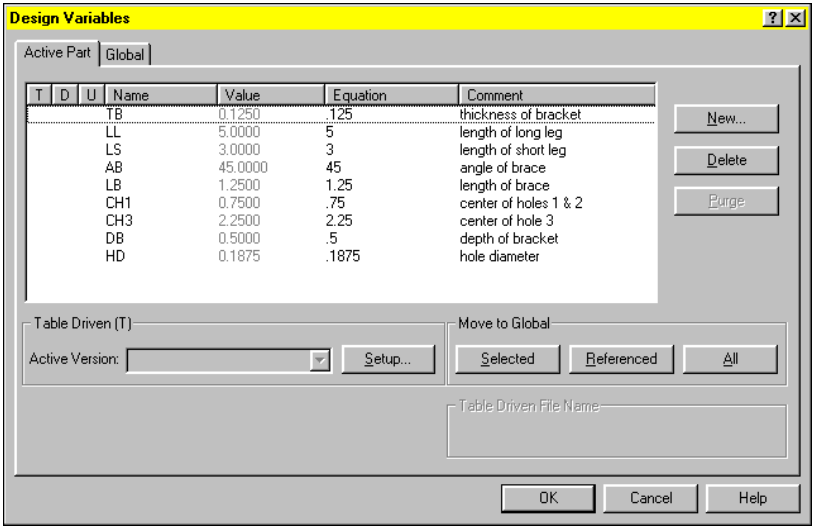
Each feature is controlled by a design variable. In this exercise, you create a table containing the design variables, and define four different part versions.

To set up a table

1 Use AMVARS to open the Design Variables dialog box and set up a table.

Desktop Menu Part ➤ Design Variables

2 In the Design Variables dialog box, verify that the Active Part tab is selected. Examine the active part design variables.



3 Under Table Driven, choose Setup.

4 In the Table Driven Setup dialog box, verify that the Active Part tab is selected. Under Layout, specify:

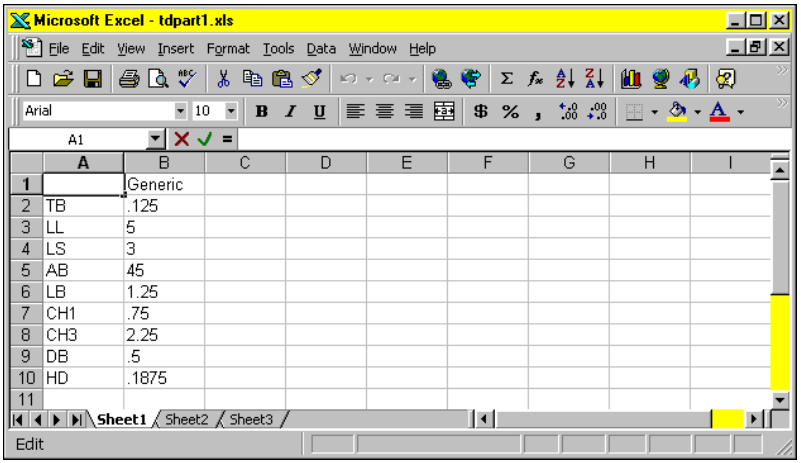
Version Names: Across

Choose Create.

5 In the Create Table dialog box, select the *desktop\tutorial* folder and specify *tdpart1.xls* as the name of your table.

Choose Save to exit the dialog box.

Microsoft Excel opens a new spreadsheet containing a generic part and a value for each design variable assigned to it.

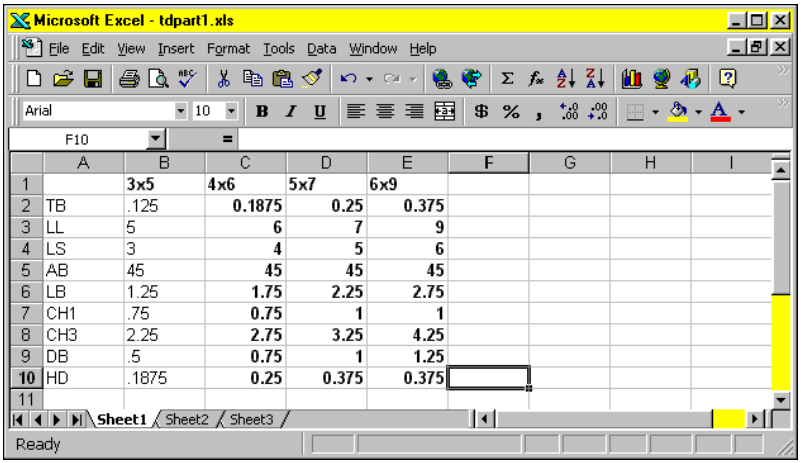


	A	B	C	D	E	F	G	H	I
1		Generic							
2	TB	.125							
3	LL	5							
4	LS	3							
5	AB	45							
6	LB	1.25							
7	CH1	.75							
8	CH3	2.25							
9	DB	.5							
10	HD	.1875							
11									

Next, change the name of the generic part to 3×5 and add three more part versions.

To define additional part versions

1 Fill in the value of the design variable for each new part definition, as shown in the following illustration:



	A	B	C	D	E	F	G	H	I
1		3x5	4x6	5x7	6x9				
2	TB	.125	0.1875	0.25	0.375				
3	LL	5	6	7	9				
4	LS	3	4	5	6				
5	AB	45	45	45	45				
6	LB	1.25	1.75	2.25	2.75				
7	CH1	.75	0.75	1	1				
8	CH3	2.25	2.75	3.25	4.25				
9	DB	.5	0.75	1	1.25				
10	HD	.1875	0.25	0.375	0.375				
11									

2 Save the file and exit Microsoft Excel.

3 In the Table Driven Setup dialog box, select Update Link and choose OK.

Choose OK to exit the Design Variables dialog box.

The Desktop Browser now contains a Table icon nested below PART1_1. The four part versions you created in the spreadsheet are listed below the Table icon.

The PART1_1 (3×5) icon displays the active version in parentheses. The Table (*tdpart1.xls*) icon indicates the name of the spreadsheet that is linked to your drawing.

Next, display each version of the table driven part. When you select a version, Mechanical Desktop recalculates the part, using the values for the variables that correspond to the selected version.

Displaying Part Versions

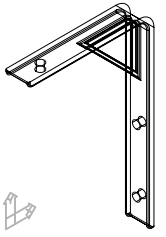
You display part versions by selecting them from the Desktop Browser.

To display a part version

1 Display the 4×6 part version.

Browser Double-click the 4x6 icon.

The 4x6 version is calculated and displayed.



2 Repeat step 1 for the other two versions.

The part versions do not differ, except in size. In the next lesson, you add another part version by editing the spreadsheet.

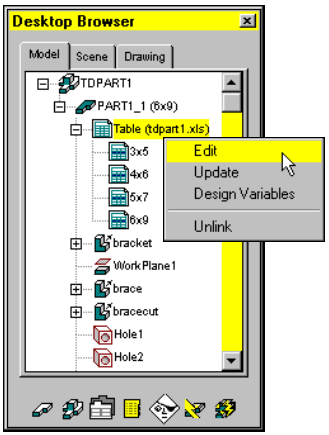
Editing Tables

Using a table to drive multiple versions of a part gives you flexibility in designing the part. You can quickly create variations of the part, and easily edit the design parameters by changing values in the cells of the spreadsheet.

To add a part version to an existing table

1 Open the spreadsheet to edit.

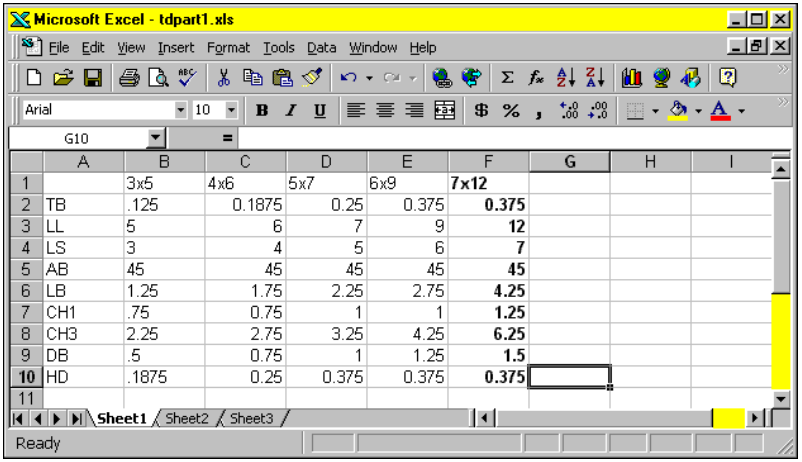
Browser Right-click Table (*tdpart1.xls*) and choose Edit.



NOTE The part definition in the Browser displays the current version of the part for easy reference.

Microsoft Excel opens, displaying your spreadsheet.

2 Add a new part version in the spreadsheet, using the values in the following illustration:



	A	B	C	D	E	F	G	H	I
1		3x5	4x6	5x7	6x9	**7x12**			
2	TB	.125	0.1875	0.25	0.375	**0.375**			
3	LL	5	6	7	9	**12**			
4	LS	3	4	5	6	**7**			
5	AB	45	45	45	45	**45**			
6	LB	1.25	1.75	2.25	2.75	**4.25**			
7	CH1	.75	0.75	1	1	**1.25**			
8	CH3	2.25	2.75	3.25	4.25	**6.25**			
9	DB	.5	0.75	1	1.25	**1.5**			
10	HD	.1875	0.25	0.375	0.375	**0.375**			
11									

3 Save the spreadsheet and exit Microsoft Excel.

4 Update the link to the spreadsheet.

Browser Right-click Table (*tdpart1.xls*) and choose Update.

Examine the Browser. It now displays five part versions under the Table (*tdpart1.xls*) icon.

Occasionally, errors occur when you link a spreadsheet to Mechanical Desktop. In the next section, you learn how to fix errors such as

- Spreadsheets that cannot be found
- Spreadsheets that have been modified since the drawing was saved
- Incorrect or missing data in a spreadsheet

Resolving Common Table Errors

Open the file *tdpart2.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

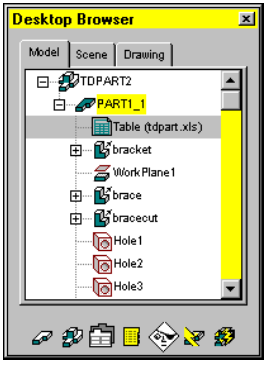
The drawing contains a version of the simple bracket used in the previous example, but it is linked to a different spreadsheet.

Before you can look for errors in the link between the drawing and the spreadsheet, you need to expand the part hierarchy.

To expand a part hierarchy

1 Expand the part hierarchy.

Browser Click the plus sign in front of TDPART2 and then click PART1_1.



Notice the red background behind the Table icon. This indicates that the spreadsheet cannot be found, it has been modified since the drawing was last saved, or it contains incorrect data in one or more cells. Before the spreadsheet can be used to drive the part, you must resolve the conflict.

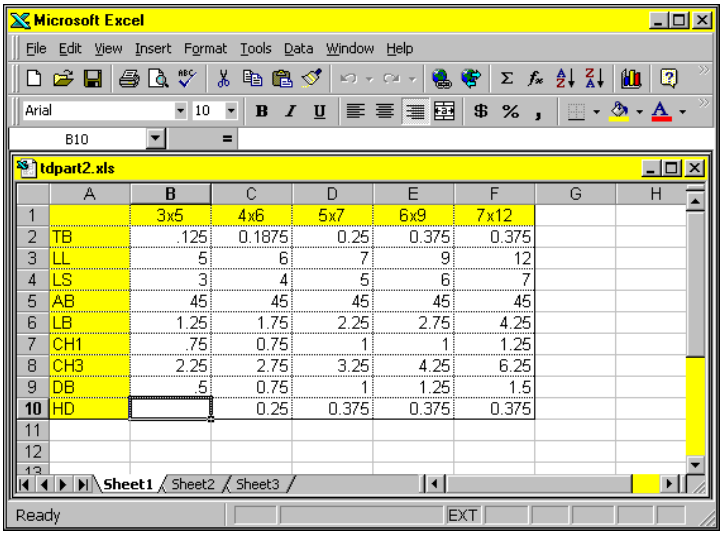
To resolve a conflict with a linked spreadsheet

1 Resolve the conflict between the drawing and the linked spreadsheet.

Browser Right-click Table (*tdpart.xls*) and choose Resolve Conflict.

An AutoCAD message dialog box is displayed asking if you would like to update the table.

2 In the message dialog box, choose Yes.



	A	B	C	D	E	F	G	H
1		3x5	4x6	5x7	6x9	7x12		
2	TB	.125	0.1875	0.25	0.375	0.375		
3	LL	5	6	7	9	12		
4	LS	3	4	5	6	7		
5	AB	45	45	45	45	45		
6	LB	1.25	1.75	2.25	2.75	4.25		
7	CH1	.75	0.75	1	1	1.25		
8	CH3	2.25	2.75	3.25	4.25	6.25		
9	DB	.5	0.75	1	1.25	1.5		
10	HD		0.25	0.375	0.375	0.375		
11								
12								

The HD variable controls the diameters of the holes for the bracket. The first part version is missing an entry in cell B10.

3 In cell B10, enter **.1875**. Then save the spreadsheet and exit Microsoft Excel.

4 Update the link to the table.

Browser Right-click Table (*tdpart2.xls*) and choose Update.

Mechanical Desktop updates the link.

5 Use AMUPDATE to update the part.

Context Menu In the graphics area, right-click and choose Update Part.

Next, display the part versions.

To display the first part version

1 Display the first part version.

Browser Double-click the 3×5 icon nested under PART1_1.

The 3×5 angle bracket is recalculated from the values in the table and then displayed.

2 Repeat step 1 for the remaining four part versions.

Save your file.

Next, you suppress features for some of the part versions in your table. The smaller brackets do not require the brace, so you will suppress the features associated with it for those versions.

Suppressing Features

Suppressing features can be done in the Browser, in a linked table, or in the Suppress By Type dialog box. Suppressing a feature manually in the Browser affects all part versions. Suppressing a table driven feature gives you control over each version.

Use the Suppress By Type dialog box to suppress and unsuppress features by specific type.

With an existing table, you append the features you want suppressed to the table. First, you manually suppress the brace feature and the bracecut feature in the Browser. It is not important which version of the part is active when the features are suppressed.

To suppress a feature

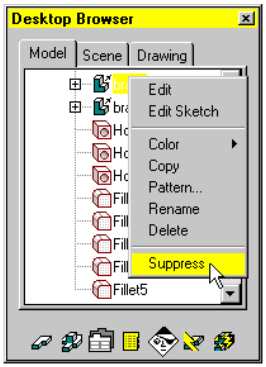
1 Display the first part version.

Browser Double-click the 3×5 icon nested under PART1_1.

2 Use AMSUPPRESSFEAT to suppress the brace feature, responding to the prompt.

Browser Right-click Brace and choose Suppress.

The highlighted features will be suppressed.
Continue? [Yes/No] <Yes>: *Press* ENTER



The brace feature is suppressed and is no longer visible on your screen. The bracecut feature and four fillets are dependent on the brace feature, so they are also suppressed.

Manually suppressed features are represented in the Browser by a circle and a dashed line preceding the grayed-out feature name. These features can be unsuppressed at any time.

Next, append the suppressed features to the table.

To append a suppressed feature to a table

1 Use AMVARS to append the suppressed features to the spreadsheet.

Desktop Menu Part ➤ Design Variables

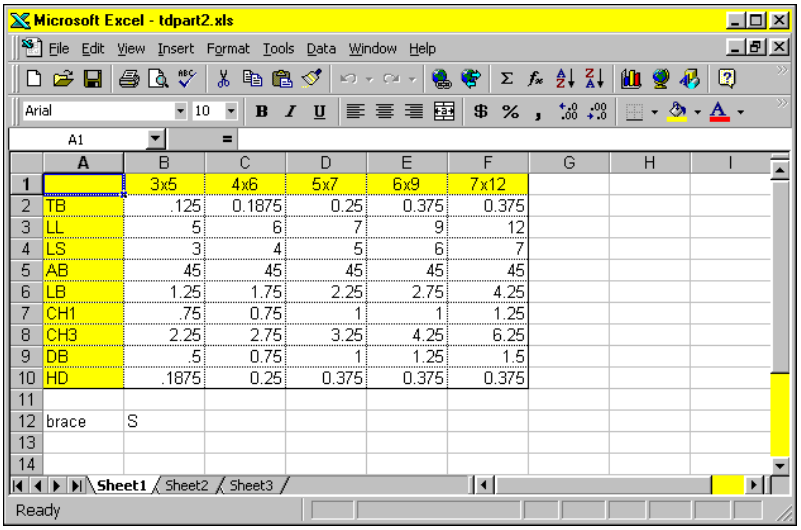
2 In the Design Variables dialog box, under Table Driven, choose Setup.

3 In the Table Driven Setup dialog box, specify:

Type: Both
Format: Concatenate Tables

Choose Append.

Microsoft Excel spreadsheet *tdpart2.xls* is displayed. A new entry, brace, has been added under the existing design variables. The letter "S" in the 3×5 column indicates that this feature is suppressed.



	A	B	C	D	E	F	G	H	I
1		3x5	4x6	5x7	6x9	7x12			
2	TB	.125	0.1875	0.25	0.375	0.375			
3	LL	5	6	7	9	12			
4	LS	3	4	5	6	7			
5	AB	45	45	45	45	45			
6	LB	1.25	1.75	2.25	2.75	4.25			
7	CH1	.75	0.75	1	1	1.25			
8	CH3	2.25	2.75	3.25	4.25	6.25			
9	DB	.5	0.75	1	1.25	1.5			
10	HD	.1875	0.25	0.375	0.375	0.375			
11									
12	brace	S							
13									
14									

Next, suppress the brace and its dependent features for the 4×6 version.

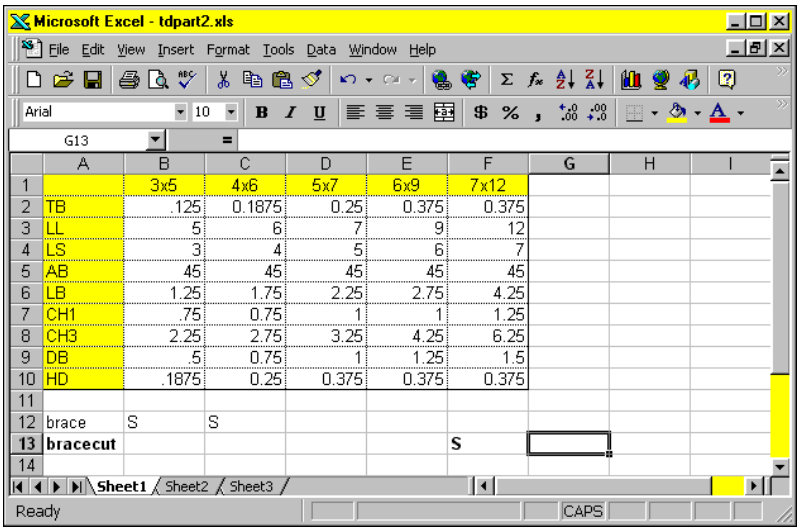
4 In the spreadsheet, enter **S** in cell C12.

For the largest bracket, you suppress the bracecut feature while retaining the brace feature for strength.

To create a suppressed feature in a table

1 In the spreadsheet, in cell A13 enter **bracecut**.

2 In cell F13 enter **S**.



	A	B	C	D	E	F	G	H	I
1		3x5	4x6	5x7	6x9	7x12			
2	TB	.125	0.1875	0.25	0.375	0.375			
3	LL	5	6	7	9	12			
4	LS	3	4	5	6	7			
5	AB	45	45	45	45	45			
6	LB	1.25	1.75	2.25	2.75	4.25			
7	CH1	.75	0.75	1	1	1.25			
8	CH3	2.25	2.75	3.25	4.25	6.25			
9	DB	.5	0.75	1	1.25	1.5			
10	HD	.1875	0.25	0.375	0.375	0.375			
11									
12	brace	S	S						
13	**bracecut**					**S**			
14									

3 Save the spreadsheet and exit Microsoft Excel.

4 Choose OK to exit the Table Driven Setup dialog box without updating the link.

5 Choose OK to exit the Design Variables dialog box.

You have created two table driven suppressed features. To activate the table driven suppression of these features, you update the link to the table. After the table has been linked to the drawing, these features cannot be manually unsuppressed. To unsuppress these features, you edit the spreadsheet.

Manually suppressed features affect all part versions so you must manually unsuppress the features in the Browser before you update the link to the table.

To manually unsuppress a feature

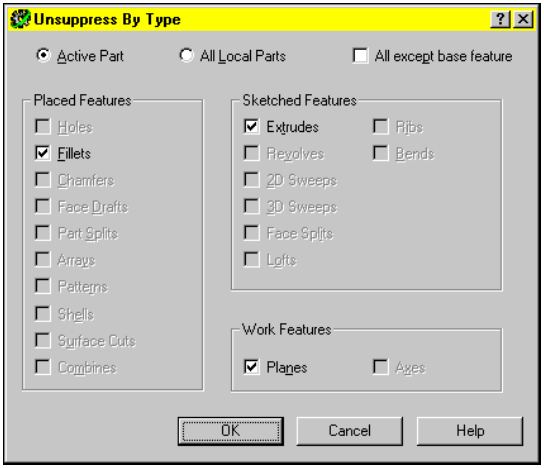
1 Use AMUNSUPPRESSFEAT to unsuppress the brace feature.

Browser Right-click Brace and choose Unsuppress+.

NOTE By using Unsuppress+ in the Browser, you unsuppress the feature you select and all dependent features. If you use the Browser, the dialog box is not displayed.

---

2 If you use the Unsuppress By Type dialog box, verify that Fillets, which are the dependent features, and Extrudes are selected.



Choose OK.

The brace and its dependent features are unsuppressed. Look at the Browser. There should be no suppressed features visible.

3 Update the link to the table.

Browser Right-click Table (*tdpart2.xls*) and choose Update.

The suppressed features for the active version are now represented in the Browser by a different symbol. A circle with a diagonal line through it indicates that the feature is suppressed by the table and can be unsuppressed only by editing the table.

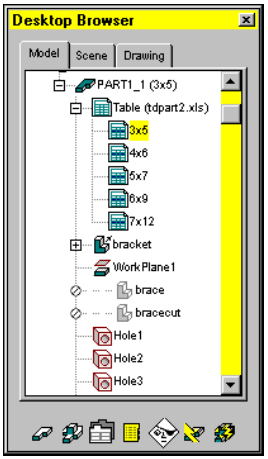
Next, display the 3×5 version of the bracket.

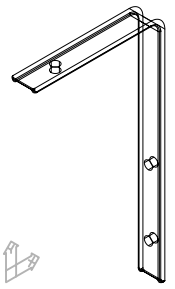
To display a part version

1 Display the part.

Browser Double-click the 3×5 icon nested below Table (*tdpart2.xls*).

The part is recalculated using the values for the 3×5 version in the linked table. In the Browser, the suppressed features are grayed out.



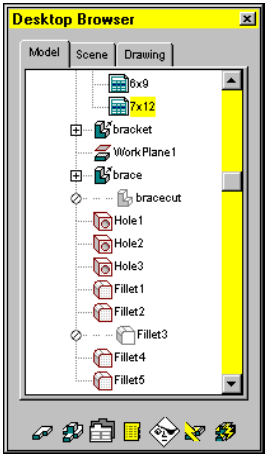


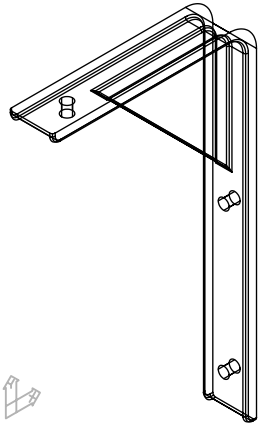
3×5 version

2 Repeat step 1 for the 4×6 version. The brace and its dependent features in this version are suppressed.

3 Repeat step 1 for the 5×7 and 6×9 versions. Note that the features are not suppressed in these versions.

4 Repeat step 1 for the 7×12 version. The brace feature remains in this version, but the bracecut and its dependent features are suppressed.





7×12 version

Save your file.

Working with Two Part Versions

You can create copies of a part and work with different versions to create assemblies. In this lesson, you copy a part and display two versions simultaneously.

To copy a part definition

1 Display the 3×5 version of the bracket.

Browser Double-click the 3×5 icon nested below Table (*tdpart2.xls*).

2 Use UCS to return to the World Coordinate System so that the copy of the part is oriented the same as the original.

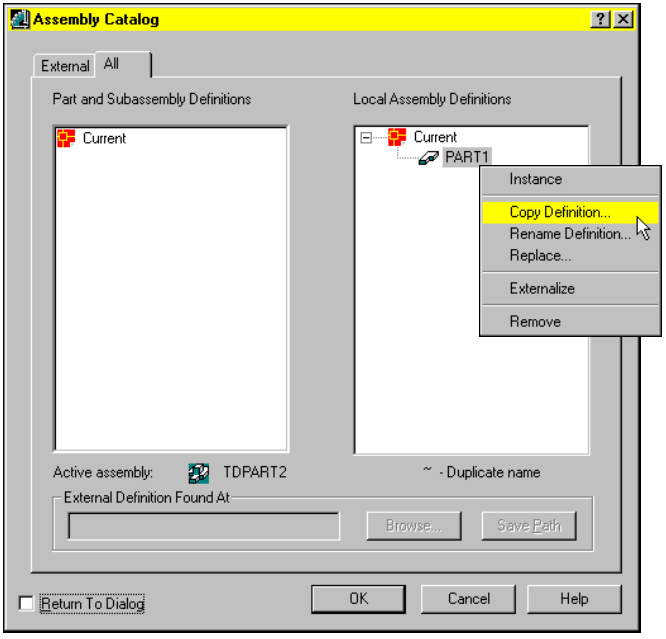
Desktop Menu Assist ➤ New UCS ➤ World

3 Use AMCATALOG to make a copy of PART1_1.

Context Menu In the graphics area, right-click and choose Catalog.

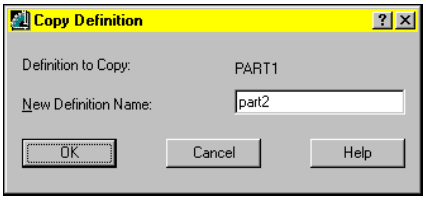
4 In the Assembly Catalog, clear the Return to Dialog check box and select the All tab.

5 In Local Assembly Definitions, right-click PART1_1 and choose Copy Definition.



6 In the Copy Definition dialog box, specify:

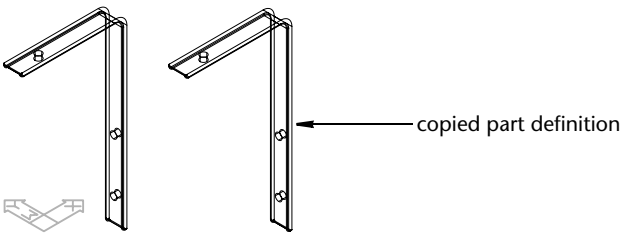
New Definition Name: *Enter* **part2**



Choose OK.

7 Continue on the command line.

Specify new insertion point: *Specify a point to the right of the existing part*
Specify insertion point for another instance or <continue>: *Press* ENTER



Choose OK to close the Assembly Catalog dialog box.

The Browser now contains a PART2_1 definition. Because you copied the original part definition, its relationship to the spreadsheet, *tdpart2.xls*, has been duplicated in the new part.

Next, change the version of the copied part in your drawing.

To display a different version

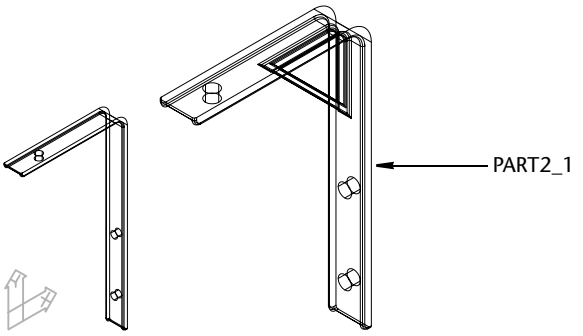
1 Use AMACTIVATE to activate PART2_1.

Browser In the Browser, right-click PART2_1 and choose Activate Part.

2 Display the 5×7 version for PART2_1.

Browser Double-click the 5×7 icon nested under PART2_1.

The 5×7 version of PART2_1 is displayed.



Save your file.

Next, you set up the drawing for plotting. Instead of using numeric dimensions to annotate five separate parts, you define views for one part and annotate the drawing, using active part design variables and equations. You paste into the drawing a copy of your spreadsheet that lists the values of those variables for each part version.

First, create drawing views to display the part.

Creating Drawing Views

The first view you create is the base view. You define the plane to orient your view using a part face or work feature, and then position the view in Drawing mode.

Because you are creating a generic view to represent all versions of the part, you use the 5×7 version, which contains the brace. When you import the table, it will indicate the versions for which the brace is suppressed.

To create a base view

1 Use the Browser to turn off the visibility of PART1_1.

Browser Right-click PART1_1 and choose Visible.

2 Use AMMODE to switch to Drawing mode.

Browser Select the Drawing tab.

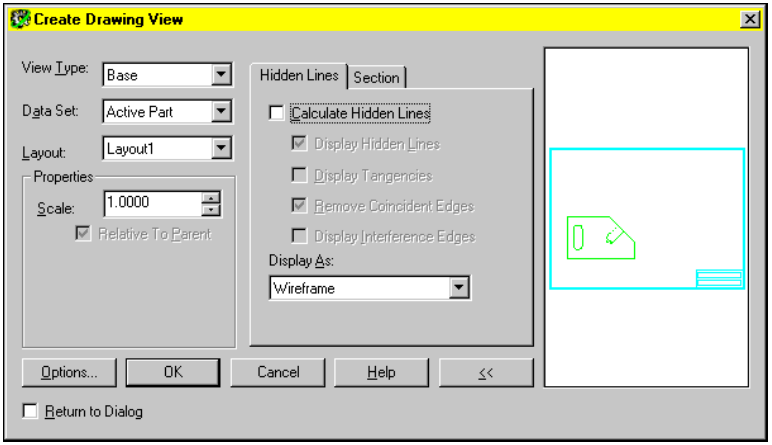
Mechanical Desktop switches to Drawing mode. A title block has been inserted into the drawing.

3 Use AMDWGVIEW to create the base view.

Context Menu In the graphics area, right-click and choose New View.

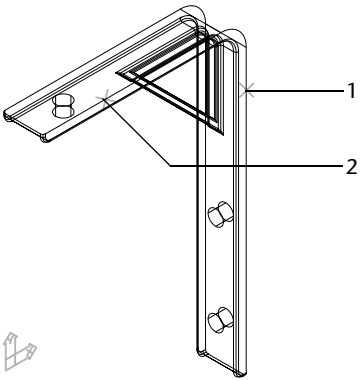
In the Create Drawing View dialog box, select the Hidden Lines tab and specify:

Calculate Hidden Lines: *Clear the check box*
Display As: Wireframe



Choose OK, and continue on the command line.

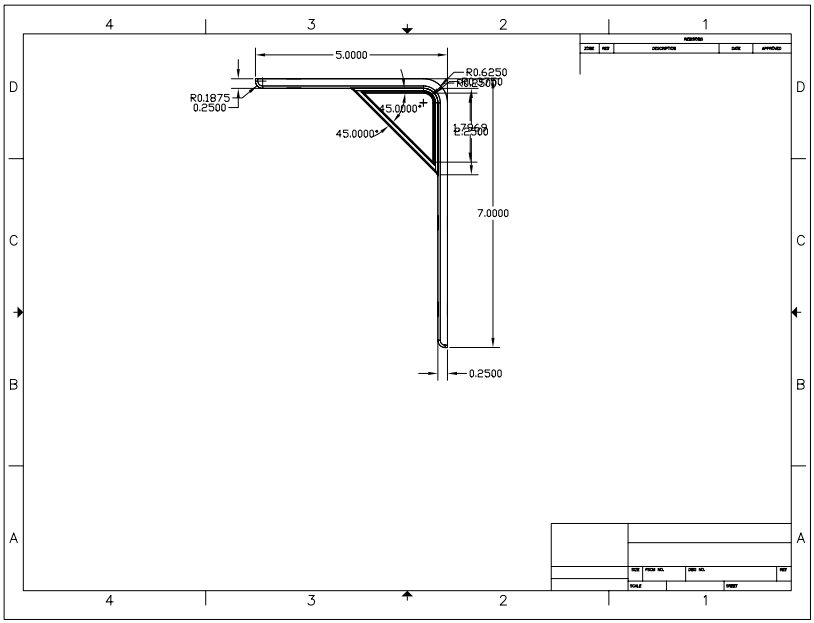
Select planar face, work plane or [Ucs/View/worldXy/worldYz/worldZx]:
Specify the front edge (1)
Enter an option [Accept/Next] <Accept>:
Enter **n** *to highlight the front face, or press* ENTER
Select work axis, straight edge or [worldX/worldY/worldZ]:
Specify the top edge (2)
Adjust orientation [Flip/Rotate] <Accept>:
Verify that the UCS is upright, and press ENTER



4 Continue on the command line to place the base view.

Specify location of base view: *Specify a point in the top center of the title block*
Specify location of base view: *Press* ENTER

Your drawing should look like this.



Next, create side and bottom orthographic views of the part.

To create an orthographic view

1 Create the orthographic view.

Context Menu In the graphics area, right-click and choose New View.

2 In the Create Drawing View dialog box, select the view type Ortho. On the Hidden Lines tab, specify:

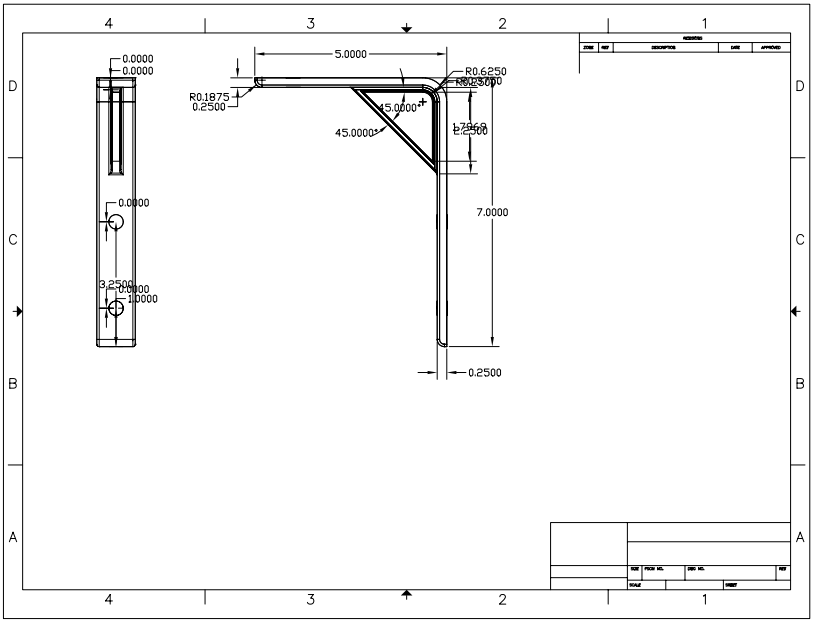
Display As: Wireframe

Choose OK.

3 On the command line, respond to the prompts as follows:

Select parent view: *Pick a point inside the base view*
Specify location for orthogonal view: *Specify a point to the left of the base view*
Specify location for orthogonal view: *Press* ENTER

Your drawing should look like this.



Next, create a bottom ortho view.

4 Create the orthographic view.

Context Menu In the graphics area, right-click and choose New View.

5 In the Create Drawing View dialog box, select the view type Ortho. On the Hidden Lines tab, specify:

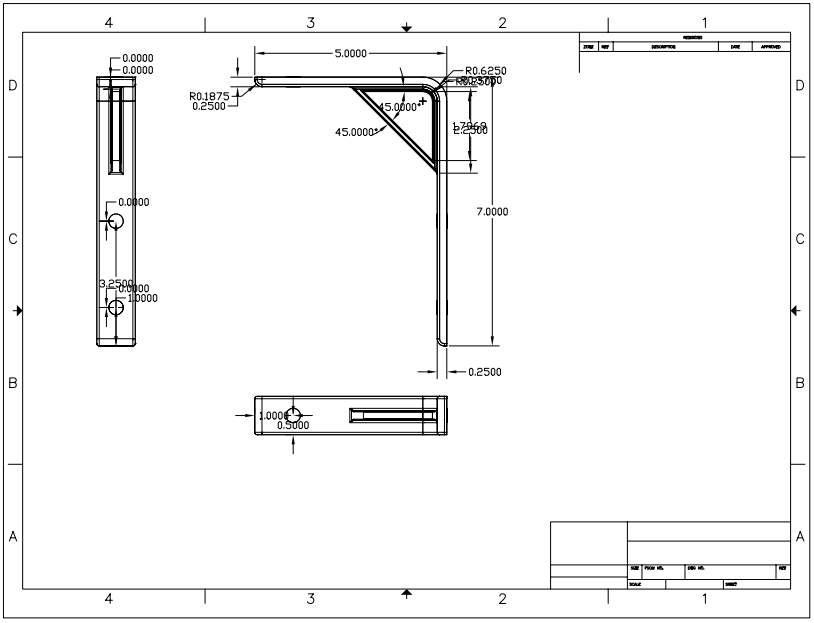
Display As: Wireframe

Choose OK.

6 On the command line, respond to the prompts as follows:

Select parent view: *Pick a point inside the base view*
Specify location for orthogonal view: *Specify a point below the base view*
Specify location for orthogonal view: *Press* ENTER

Your drawing should look like this.



Save your file.

Look at the Browser. There are two ortho views and a base view. Because the ortho views are dependent on the base view, they are nested below the Base icon.

The parametric dimensions that define the part are displayed in each view. Because Mechanical Desktop displays the parametric dimensions in the order in which they were created, dimensions may conflict with each other, overlap, or be redundant in a drawing view.

Next, clean up the display of parametric dimensions in each of the views.

Cleaning Up the Drawing

To clean up the drawing, you change the parametric dimensions to be displayed as parameters, hide extraneous dimensions, and move dimensions for clarity.

NOTE For detailed cleanup instructions see "Cleaning Up Drawings" on page 322 in chapter 13.

Displaying Dimensions as Parameters

The parametric dimensions used to define a part are displayed with the values for the active version of the part. Because you are creating a drawing of the generic part and linking a table to describe the values for the parameters controlling each part version, you change dimensions to be displayed as parameters.

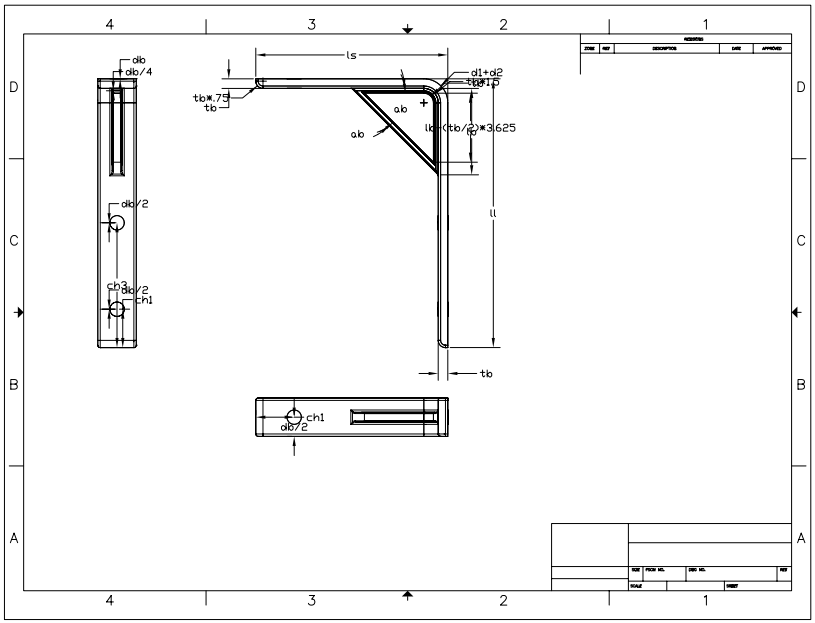
To change a dimension to be displayed as a parameter

1 Display the dimensions as parameters, responding to the prompt.

Desktop Menu Drawing ➤ Parametric Dim Display ➤ Dimensions as Parameters

Select dimension or [All/View/Select dimensions] <Select dimensions>: *Enter* **a**

Your drawing should look like this.



Next, hide the extraneous dimensions.

Hiding Extraneous Dimensions

When drawing views are created, Mechanical Desktop displays all the parametric dimensions that are related to the part display in each view. Usually, some cleanup is required because of overlapping or redundant dimensions.

To hide an extraneous dimension in a base view

1 Use AMVISIBLE to hide extraneous dimensions.

Desktop Menu Drawing ➤ Drawing Visibility

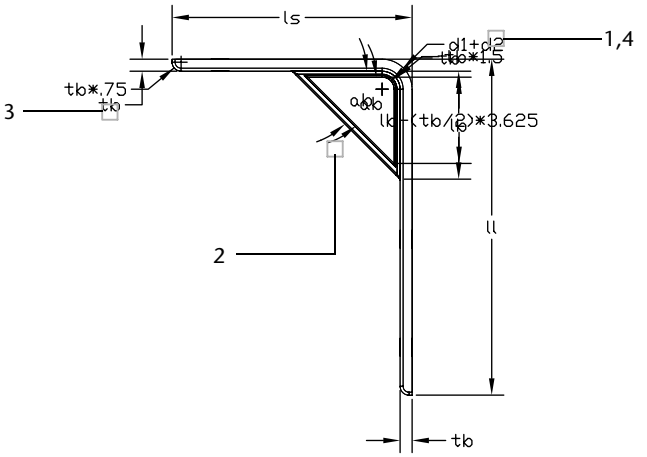
In the Desktop Visibility dialog box, choose Select.

NOTE If you choose the toolbutton method to hide the dimensions, the Desktop Visibility dialog box is not displayed. Select the dimensions to hide. Use Zoom Realtime while selecting the dimensions.

---

2 Respond to the prompts as follows:

Select drawing objects to hide: *Specify the* **d1+d2** *dimension (1)*
Select drawing objects to hide: *Specify the* **ab** *dimension (2)*
Select drawing objects to hide: *Specify the* **tb** *dimension (3)*
Select drawing objects to hide: *Specify the* **tb** *dimension (4)*
Select drawing objects to hide: *Press* ENTER



In the Desktop Visibility dialog box, choose OK.

To hide a dimension in an orthographic view

1 Use AMVISIBLE to hide dimensions.

Desktop Menu Drawing ➤ Drawing Visibility

2 In the Desktop Visibility dialog box, choose Select.

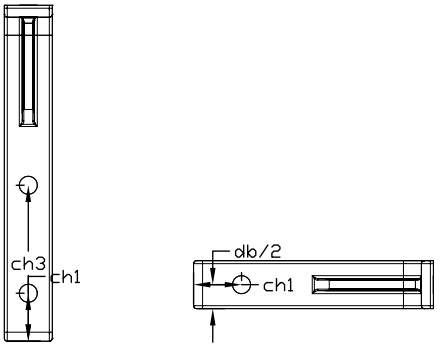
NOTE If you choose the toolbutton method to hide the dimensions, the Desktop Visibility dialog box is not displayed. Select the dimensions to hide.

---

3 In the side ortho view, hide the **db** dimension, the two **db/2** dimensions, and the **db/4** dimension.

You do not need to hide dimensions in the bottom ortho view.

The ortho views should look like this.



side view bottom view

Now that the extraneous dimensions are hidden, it is easier to move the remaining dimensions for clarity.

Moving Dimensions

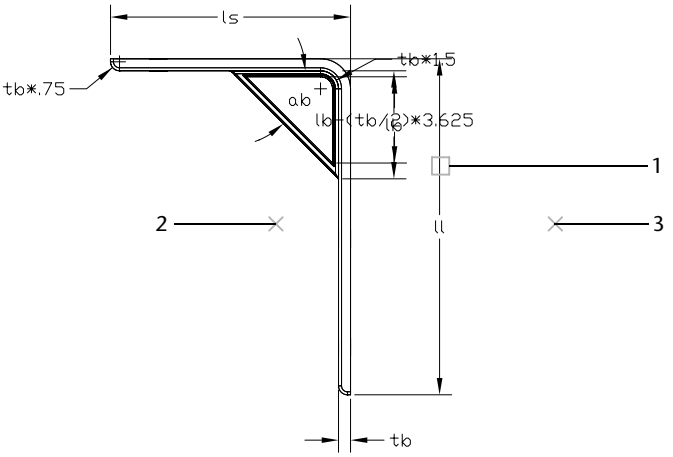
Because several of the remaining dimensions overlap, you need to rearrange them so that they are easy to view.

To move a dimension

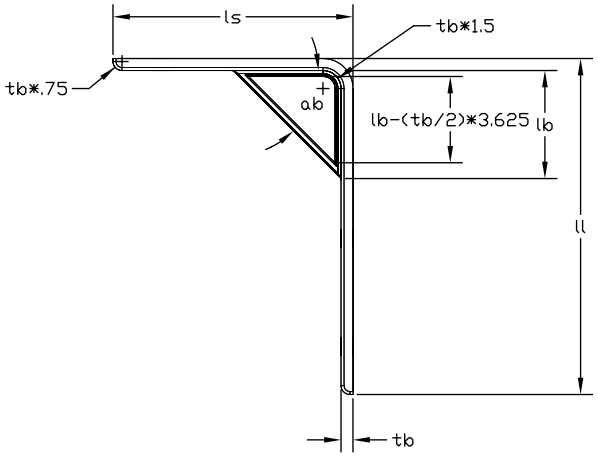
1 Use AMMOVEDIM to move the parametric dimensions in the base view, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Dimensions ➤ Move Dimension.

Enter an option [Flip/Move/move mUltiple/Reattach] <Move>: *Press* ENTER
Select dimension to move: *Select the dimension for the long leg of the bracket (1)*
Select destination view: *Specify a point in the base view (2)*
Select location: *Specify a point to the right (3)*
Select location: *Press* ENTER

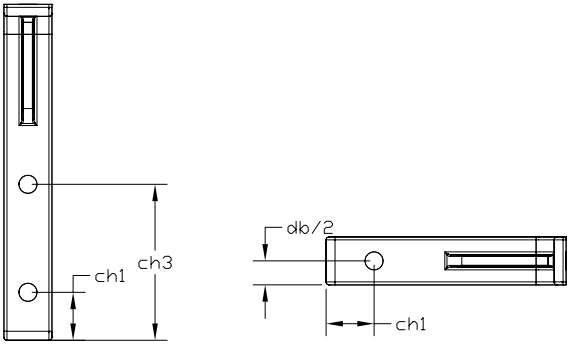


2 Continue moving dimensions until your base view looks like this.



3 Move the dimensions in the ortho views.

The ortho views should look like this.



side view bottom view

Next, add reference dimensions, to fully define the part.

Enhancing Drawings

To finalize the presentation of the drawing, you add power dimensions, displayed as parameters and create a hole note to describe the three holes in the bracket.

Creating Power Dimensions

The drawing views are intended to display the generic part. When you display parametric dimensions using design variables, and you power dimension your drawing views, the views represent the generic part.

Because reference dimensions are not displayed as parameters, you use power dimensioning to create dimensions represented as parameters. Power dimensioning allows you to specify tolerance and fit information for your parts as you dimension and to modify the default value of a dimension as you create it.

Later, when you paste the spreadsheet into the drawing, you will have a list of values for the variables in each version that cross-references the dimensions in each view.

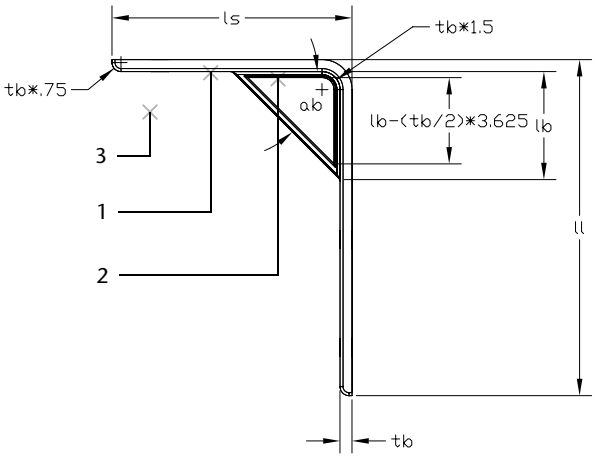
To add a reference dimension

Before you begin this procedure, enable Osnaps. If Osnaps are set to off, you will be unable to create the dimension. Work in the base view.

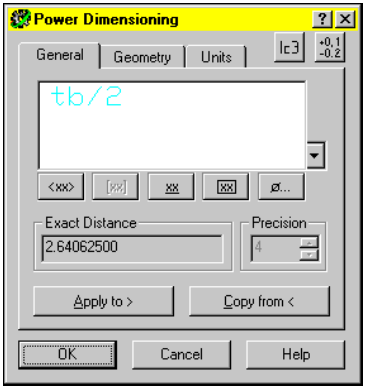
1 Use AMPOWERDIM to create power dimensions, responding to the prompts.

Context Menu In the graphics area, right-click and choose Power Dimensioning.

(Single) Specify first extension line origin or [Angular/Options/Baseline/Chain/Update] <Select>: *Specify a point at the bottom of the short leg (1)*
Specify second extension line origin:
Specify a point at the inside top of the bracecut (2)
Place dimension line [Options/Pickobj] <Options>: *Specify a location (3)*



2 In the Power Dimensioning dialog box, change the default text to **tb/2**.

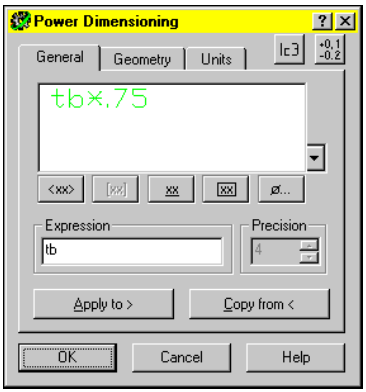


Choose OK.

3 Continue on the command line to add a power dimension to the fillet at the bottom of the long leg:

(Single) Specify first extension line origin or [Angular/Options/Baseline/Chain/Update] <Select>: *Press* ENTER
Select arc, line, circle, or dimension: *Specify the arc*
Enter an option [Next/Accept} <Accept>: *Press* ENTER
Specify dimension line location or [Linear/Diameter/Options]: *Specify a location for the dimension*
(Single) Specify first extension line origin or [Angular/Options/Baseline/Chain/Update] <Select>: *Press* ENTER
Select arc, line, circle, or dimension <Exit>: *Specify the dimension*

4 In the Power Dimensioning dialog box, specify **tb*.75** for the dimension text.

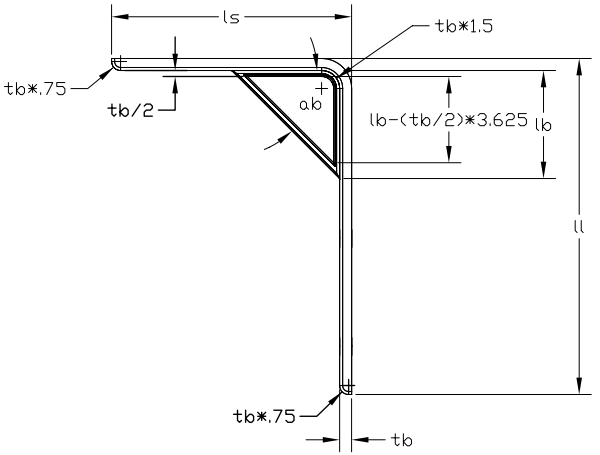


Choose OK.

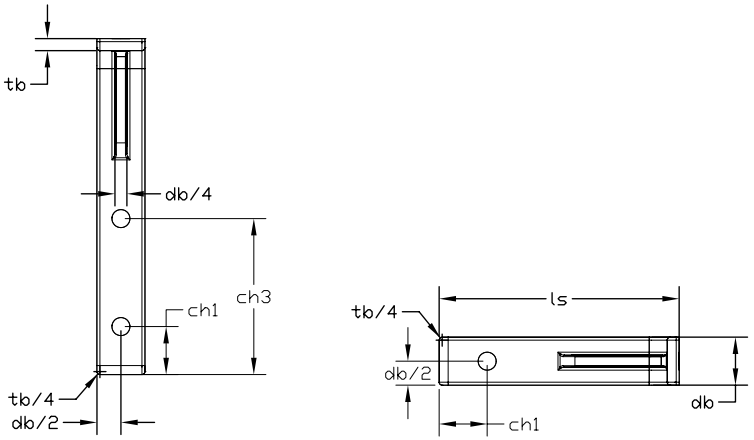
5 Continue on the command line.

Select arc, line, circle, or dimension <Exit>: *Press* ENTER

Your drawing should resemble the following illustration.



6 Place power dimensions so that the orthographic views look like this.



Next, create a hole note that describes the three holes in the bracket.

Creating Hole Notes

Mechanical Desktop provides a tool for creating hole notes, which saves you time when annotating your drawing.

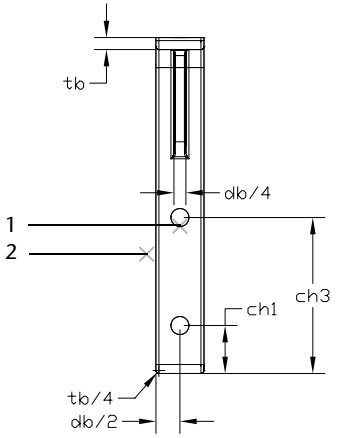
In the side view, create a hole note for one of the holes in the long leg of the bracket.

To create a hole note

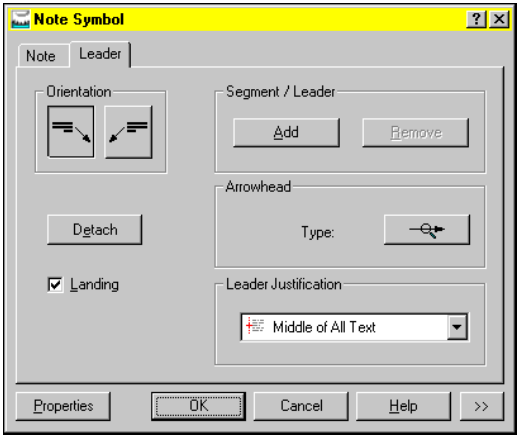
1 Use AMNOTE to create the hole note, responding to the prompts.

Context Menu In the graphics area, right-click and choose Annotation ➤ Hole Note.

Select object to attach [rEorganize]:
Specify the upper hole in the long leg of the bracket (1)
Next Point <Symbol>: *Specify the location (2) and press* ENTER.



2 In the Note Symbol dialog box, select the Leader tab. In Leader Justification, specify Middle of All Text.



Choose OK.

The hole note is displayed in your drawing.

Edit the hole note so that it is typical for all three holes.

3 Use AMPOWEREDIT to edit the hole note, responding to the prompt.

Context Menu Select the hole note, then right-click the note and choose Edit.

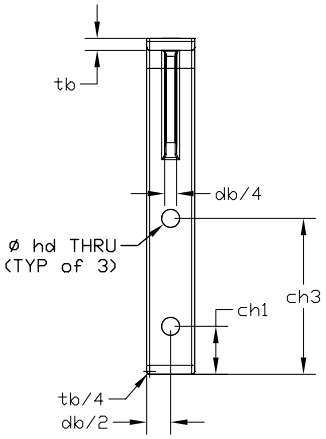
4 In the Note Symbol dialog box, select the Note tab, and change the text to read as follows:

%%c hd THRU

(typ of 3)

Choose OK.

The side view should now look like this.



Save your file.

Now that the power dimensions and annotations are in place, paste the linked spreadsheet into the drawing.

Pasting Linked Spreadsheets

Pasting a linked spreadsheet into a drawing provides more flexibility for working with the table that defines the values for the part versions. The external spreadsheet can be opened and modified while you are working, and the results are reflected in the drawing.

To paste a linked spreadsheet into a drawing

1 Use AMMODE to return to the Part/Assembly environment.

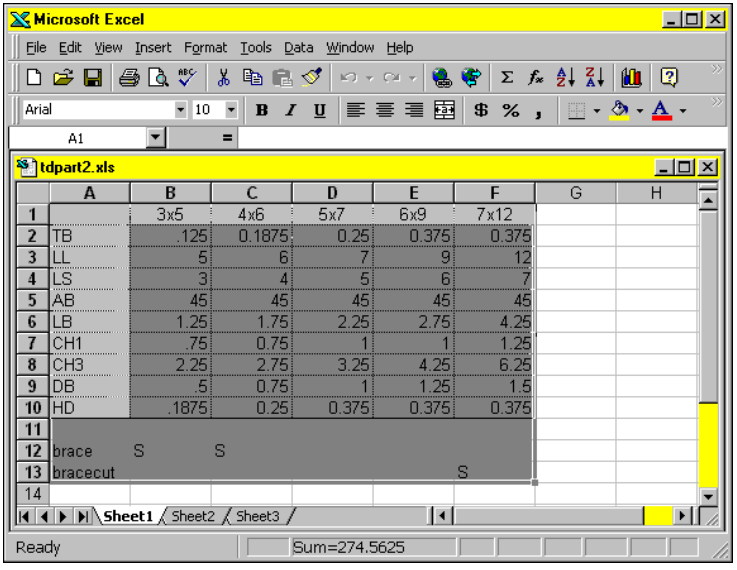
Browser Select the Model tab.

2 Open the spreadsheet.

Browser Right-click Table (*tdpart2.xls*) and choose Edit.

The Microsoft Excel spreadsheet is displayed.

3 Select the cells to be copied.



	A	B	C	D	E	F	G	H
1		3x5	4x6	5x7	6x9	7x12		
2	TB	.125	0.1875	0.25	0.375	0.375		
3	LL	5	6	7	9	12		
4	LS	3	4	5	6	7		
5	AB	45	45	45	45	45		
6	LB	1.25	1.75	2.25	2.75	4.25		
7	CH1	.75	0.75	1	1	1.25		
8	CH3	2.25	2.75	3.25	4.25	6.25		
9	DB	.5	0.75	1	1.25	1.5		
10	HD	.1875	0.25	0.375	0.375	0.375		
11								
12	brace	S	S					
13	bracecut					S		
14								

4 In Microsoft Excel, choose Edit ➤ Copy.

5 In the drawing, return to Drawing mode.

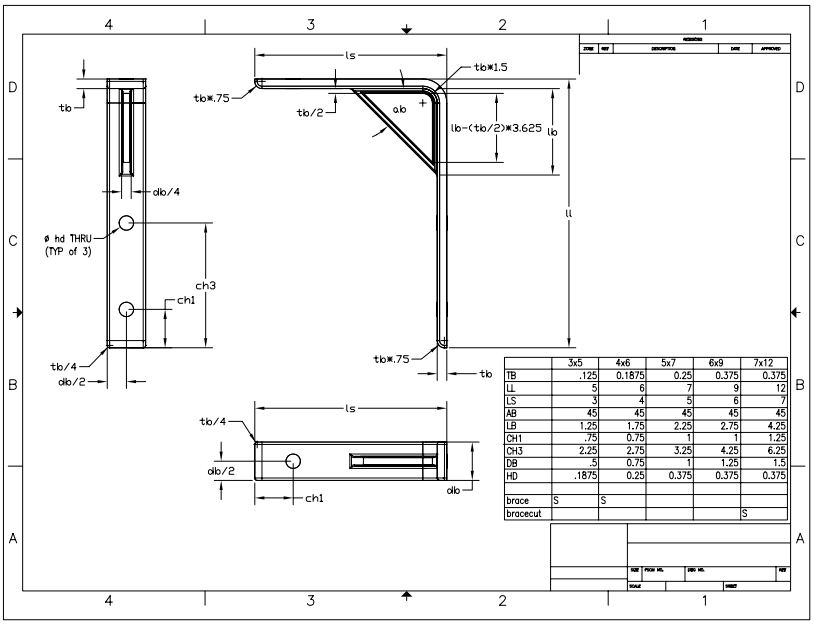
Browser Select the Drawing tab.

6 Paste the selected area of the spreadsheet into the drawing.

Desktop Menu Edit ➤ Paste Special

7 In the Paste Special dialog box, choose Paste Link.

8 Place the selected cells in the drawing. Your drawing should look like this.



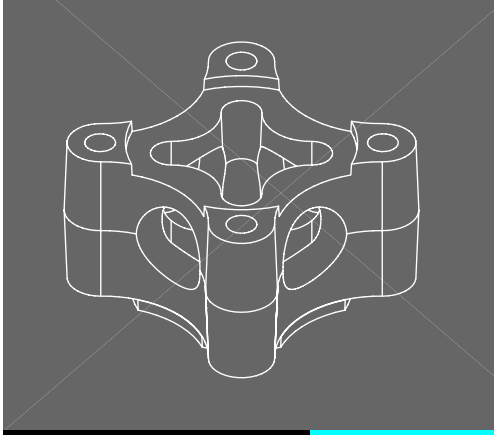
	3x5	4x6	5x7	6x9	7x12
TB	.125	0.1875	0.25	0.375	0.375
LL	5	6	7	9	12
LS	3	4	5	6	7
AB	45	45	45	45	45
LB	1.25	1.75	2.25	2.75	4.25
CH1	.75	0.75	1	1	1.25
CH3	2.25	2.75	3.25	4.25	6.25
DB	.5	0.75	1	1.25	1.5
HD	.1875	0.25	0.375	0.375	0.375
brace	S	S			
bracecut					S

NOTE Depending on the zoom factor of your display at the time you paste the image, you may have to resize it. Select the image, and use a corner grip to resize it to fit in the drawing.

Save your file.

Now that the linked spreadsheet has been pasted into the drawing, any changes made in the spreadsheet will automatically be reflected in the drawing. Experiment with modifications to the spreadsheet to see how your part and the table are automatically updated.

Assembling Parts



In This Chapter

16

- Using external parts in an assembly
- Using assembly constraints
- Analyzing the assembly
- Creating assembly scenes
- Using assembly tweaks and trails
- Creating assembly drawing views
- Editing an external reference and updating an assembly

You can build part assembly models from two or more parts, or parts grouped in subassemblies. Like part features, parts and subassemblies act as building blocks. Autodesk® Mechanical Desktop® builds individual parts and subassemblies into an assembly.

In this tutorial, you create an assembly model of a pair of slip pliers from four parts. Three of the parts originate in files that are externally referenced to the assembly file.

Using externally referenced parts creates a truly parametric assembly design. Changes to an external reference can be made from within the assembly or in the original file.

Key Terms

Term	Definition
3D constraint	In assembly modeling, an associative link between two or more parts that controls their locations relative to each other and to their placement within the assembly.
Assembly Catalog	The means of attaching and cataloging local and external parts and subassemblies in the Assembly Modeling environment. Use the All and External tabs to specify contents, which can be instanced, copied, renamed, replaced, externalized, removed, localized, sorted, unloaded, and reloaded.
assembly tree	A graphical hierarchy that illustrates the order in which parts and subassemblies are combined in the current assembly. The assembly tree is managed in the Desktop Browser.
attach	The act of connecting a reference file to the current assembly file. The attachment remains with the current file after the file is saved.
definition	All information about a part or subassembly, including its name, location, and attributes.
detach	Permanently removing a file as an external reference in an assembly.
external reference	A part or assembly that resides in a file other than the current part or assembly file.
insert constraint	Aligns center points and planes of two circles in a specified direction. Solves translation degrees of freedom. Used to constrain a bolt in a hole, for example.
localized part	Changes the definition of a part from external to local, severing the link to the external file. Changes made to a localized part affect only the current part or assembly file; other part or assembly files that reference the part are not affected.
mate constraint	Causes a plane or axis on one part to be coincident with a plane, point, or axis on another part in a specified direction. Removes a translational degree of freedom.
rename definition	In the Part or Assembly Catalog, an attached external part or subassembly may be renamed. The alias name is displayed beside the drawing name in parentheses.
scene	A 3D orientation of an assembly that you can use to create a 2D view in Drawing mode. You use scenes to provide exploded or assembled views of your assembly without destroying the constraints.
trail	In an exploded view, a line that shows how parts in an assembly are assembled.
tweak	Adjusts the position of parts in an assembly scene to avoid overlap in some views or to make some parts more visible.

Basic Concepts of Assembling Parts

You create assemblies from parts, either combined individually or grouped in subassemblies. Mechanical Desktop builds these individual parts and subassemblies into an assembly in a hierarchical manner according to relationships defined by constraints. Using the Desktop Browser, you can restructure the hierarchy of an assembly as needed, while retaining the design constraints. See "Using the Desktop Browser" on page 414.

As in part modeling, the parametric relationships allow you to quickly update an entire assembly based on a change in one of its parts.

You can build 3D solid assembly models from two or more parts or subassemblies. Like part features, parts and subassemblies act as building blocks.

The following process for building assemblies and subassemblies is similar to that for building parts:

- Lay out the assembly.
- Create the base part.
- Create the remaining parts.
- Create the assembly and subassemblies.
- Analyze the assembly.
- Modify the assembly as necessary.

When you create an assembly file, you can create your parts in the assembly drawing, or you can reference external files.

Using externally referenced parts gives you more flexibility over the control of your assembly. If you need to make modifications to any of your parts, you can open the individual part file and make changes to it. Because more than one drawing can be open in the same Mechanical Desktop session, you can immediately see the effects of your changes in the assembly file. You can also edit external references from within the assembly file. This is particularly useful in smaller assemblies. Depending on your system resources, you can edit external files individually if they are part of a large assembly.

After you have assembled your parts, you need to check the assembly for interferences. You may also want to perform mass property calculations on your parts to ensure that they are structurally sound.

Finally, you need to document your design. To make it easier to visualize your design, you may want to adjust, or tweak, your assembly and add trails to indicate how your parts fit together. Then, you set up your drawing views and add information, such as reference dimensions and annotations, before finalizing the drawing for plotting.

Starting Assembly Designs

An assembly design might begin as an overall conceptual design. You may know how the parts are assembled but you may not know all the details about each part.

Before you begin, decide how you want to lay out your assembly.

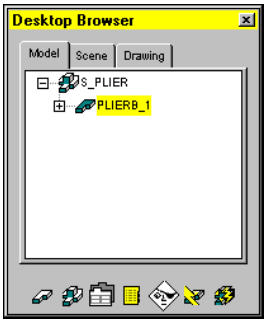
- Start with a design idea.
- Decide whether you need to create new parts, or if you can use existing parts.
- Start drawing the parts.

Although you can begin designing immediately and reorganize the hierarchy of your assembly later, for this lesson you assemble parts in a logical order, starting at the top level in the assembly tree. One part has already been created in the tutorial file for this lesson. The other three parts reside in separate files. You attach the files and then instance the parts into the assembly.

To begin, open the file *s_plier.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

In the Browser, the PLIERB part is nested under the assembly icon.



Next, you reference three other parts to create the assembly.

Using External Parts in Assemblies

The parts that make up the assembly model can be created and maintained in other files. These parts are called externally referenced parts. When a part changes, all instances of that part in other files are automatically updated.

Using parts from other files as external references is similar to external referencing (xref) in AutoCAD. Because they are external, you can reuse referenced parts in future assemblies. You can start a library of commonly used parts in a directory where you can easily locate them. Externally referenced parts can be instanced in the current file at any time.

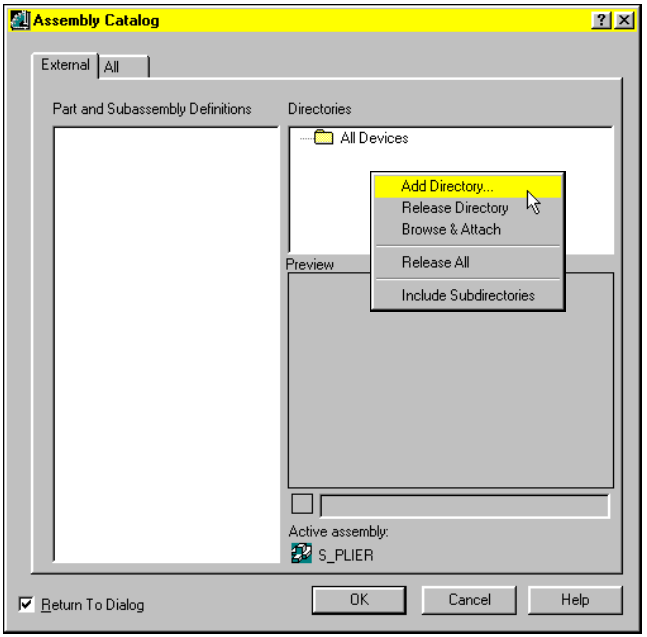
To attach and instance an external part

1 Use AMCATALOG to attach the external parts.

Context Menu In the graphics area, right-click and choose Assembly Menu ➤ Catalog.

2 In the Assembly Catalog, verify that the External tab is selected and Return to Dialog is checked.

In the Directories window, right-click and choose Add Directory.



A Browse for Folder dialog box lists your network connections and directories.

3 In the Browse for Folder dialog box, specify the *desktop\tutorial* folder to be your working directory.

Choose OK.

NOTE If you have installed Mechanical Desktop in a different location, browse through your directories to locate the correct folder.

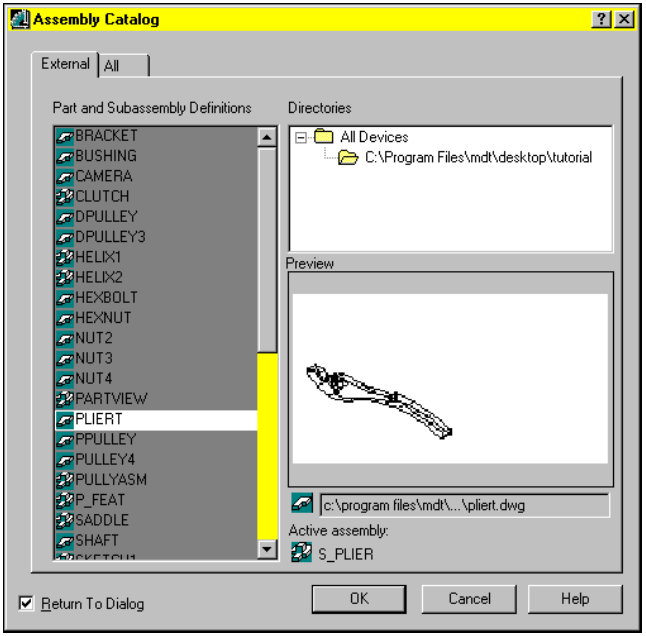
All drawing files in the *desktop\tutorial* folder are listed under Part and Subassembly Definitions on the External tab.

4 In the *desktop\tutorial* folder, double-click the PLIERT definition to insert an instance into the drawing, responding to the prompts.

Specify new insertion point: *Specify a point below* PLIERB
Specify insertion point for another instance or <continue>: *Press* ENTER

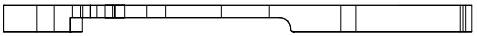
You can also right-click the definition in the Catalog and choose Attach.

The *pliert.dwg* file is attached to the current assembly, and an instance is inserted into the current assembly. In the Assembly Catalog, PLIERT is displayed on a white background.

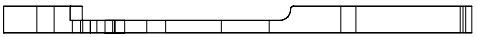


Notice that each of the external files is preceded by an icon, indicating whether it is a part or assembly file.

5 Attach and instance the HEXNUT and HEXBOLT parts to the current assembly.



plierb



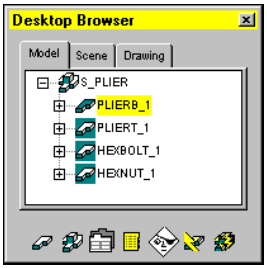
pliert

hexbolt

hexnut

As you instance each part, you return to the Assembly Catalog. In the Part and Subassembly Definitions list, each attached part is displayed on a white background. When all parts are instanced into the assembly, choose OK.

Examine the Browser. Four parts are nested under the assembly definition.



Each part is followed by a number. As you create instances of a part, each instance is numbered to indicate the order in which you added them to the assembly.

Notice that PLIERT, HEXBOLT, and HEXNUT have teal colored icons. This indicates that the parts are externally referenced.

6 Change to an isometric view of your drawing.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

Now you are ready to start the assembly process.

NOTE For information about using the shortcut method in the Desktop Browser to localize and externalize assembly component definitions, see "To localize an external part with the Browser" on page 417.

Assembling Parts

After parts or subassemblies have been created, you apply constraints to position them relative to one another. Each time you apply a constraint to a part, you eliminate some degrees of freedom (DOF). The number of degrees of freedom determines the movement of a part in any direction; the more constraints applied, the less the part can move.

A degrees of freedom symbol illustrates the instance order of the parts and how the parts can move. The DOF symbol shows how many degrees of freedom are not solved and help you visualize and apply constraints to parts.

Apply multiple assembly constraints on two parts to fully position them relative to each other. A bolt might still turn (rotational degree of freedom is not solved), but as long as the bolt and hole are aligned on their axes, and the bolt face is flush with the hole, no other constraints are needed.

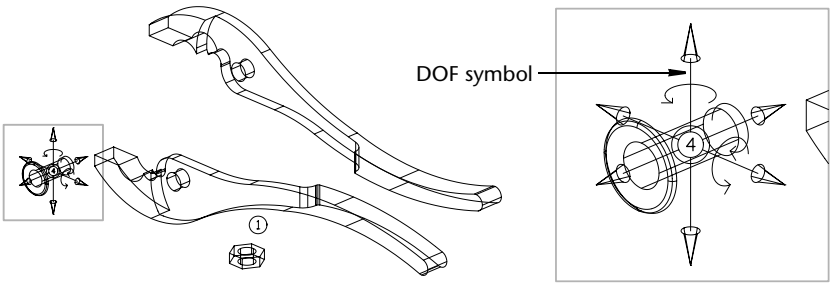
Because the parts and their assembly constraints are parametric, they can be edited. The assembly constraints applied on each part are permanently stored with the assembly to allow parametric updating if the parts change.

To display a DOF symbol

1 Use the Browser to display the DOF symbol for HEXBOLT_1.

Browser Right-click HEXBOLT_1, and choose DOF Symbol.

The DOF symbol is displayed in the center of HEXBOLT. It indicates that the bolt can move in any direction.



Constraining Parts

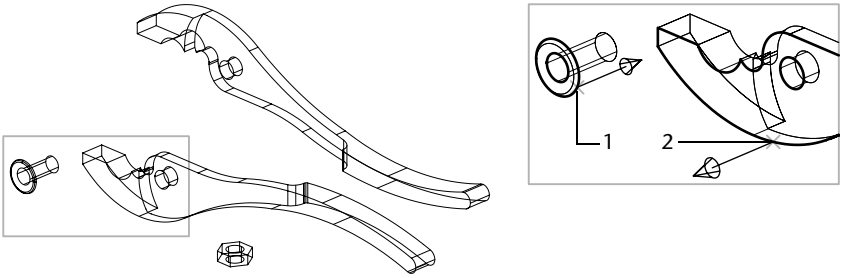
Use a mate constraint to join the PLIERT and HEXBOLT parts. Zoom in as needed to make the selection easier.

To add a mate constraint between parts

1 Use AMMATE to select the first set of geometry, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

Select first set of geometry: *Select* HEXBOLT *(1)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **a**
First set = Plane
Enter an option [Clear/aXis/Point/Flip/cYcle] <accEpt>:
Enter **y** *to cycle to face, or press* ENTER

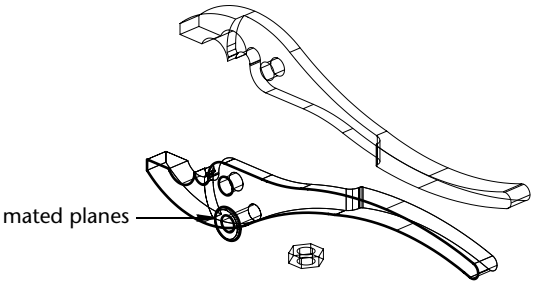


2 Select the second set of geometry to complete the mate constraint.

Select second set of geometry: *Select* PLIERT *(2)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>:
Enter **a** *to highlight the corresponding face of* PLIERT
Second set = Plane
Enter an option [Clear/aXis/Point/Next/Flip/cYcle] <accEpt>:
Enter **y** *to cycle to the next face, or press* ENTER
Enter offset <0.0000>: *Press* ENTER

To select geometry, enter responses on the command line, or use the animated cursor to cycle through options. Click the left (red) mouse button to cycle. Then, press the right (green) mouse button to accept. You may also use the animated cursor to select a face by picking in the area defined by the face, then cycling to other logical faces before accepting.

The corresponding faces of the HEXBOLT and PLIERT parts are mate constrained.

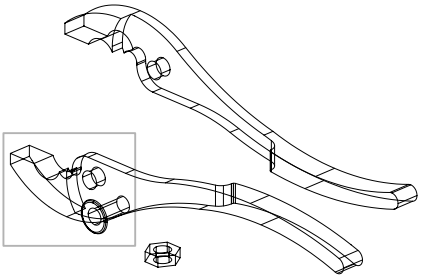


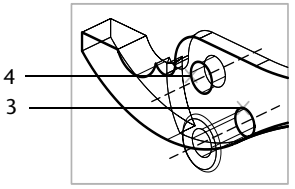
NOTE The DOF symbol is useful during constraining but has been turned off for clarity in the illustrations.

3 Use AMMATE to constrain HEXBOLT to the bolt hole on PLIERT along their axes, following the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

Select first set of geometry: *Select the end of the* HEXBOLT *(3)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Press* ENTER
Select second set of geometry: *Select the bolt hole of* PLIERT *(4)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>: *Press* ENTER
Enter offset <0.0000>: *Press* ENTER

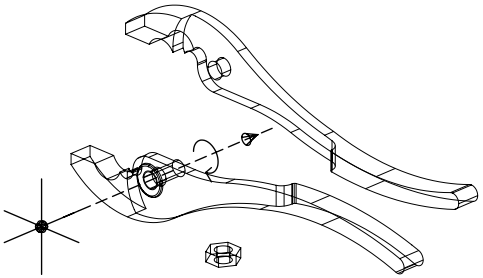




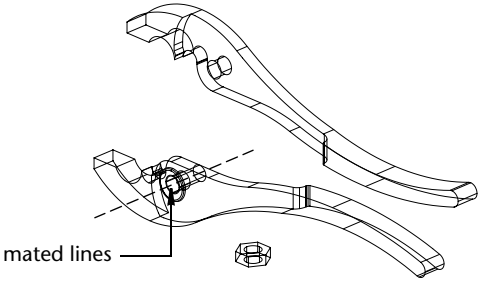
In addition to a mate constraint on the faces of the bolt head and pliers, the HEXBOLT and the PLIERT bolt hole are constrained along their axes.

4 Examine the DOF symbol.

It should show that only the rotational degree of freedom remains unsolved. In this case, it does not need to be solved.



You could use the insert constraint to solve the same degrees of freedom as the two mate constraints. By choosing the faces of HEXBOLT and PLIERT, the parts would be aligned on their axes and mate constrained on their corresponding faces.



NOTE To make part selections easier, use MOVE to reposition a constrained part. The part will automatically move back to its constrained position once you have added a new assembly constraint.

To mate to a grounded part

1 Turn on the DOF symbol for PLIERB.

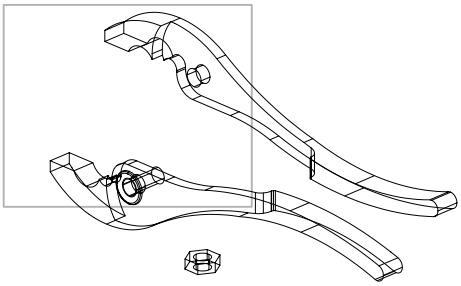
Browser Right-click PLIERB_1 and choose DOF Symbol.

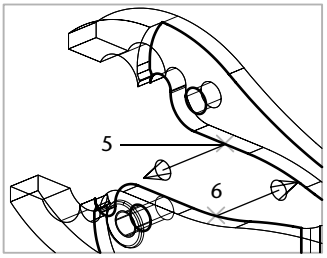
The DOF symbol is represented by a number within a circle, indicating that it has no degrees of freedom. Because it is the first part created in the assembly drawing, PLIERB becomes the *grounded part*. As you apply assembly constraints, the grounded part remains stationary. If you move the grounded part, all parts constrained to it also move.

2 Mate the PLIERB and PLIERT parts, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

Select first set of geometry: *Select the inside face of* PLIERB *(5)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **a** *to highlight the face*
First set = Plane
Enter an option [Clear/aXis/Point/Flip/cYcle] <accEpt>:
Enter **y** *to cycle to the next face, or press* ENTER

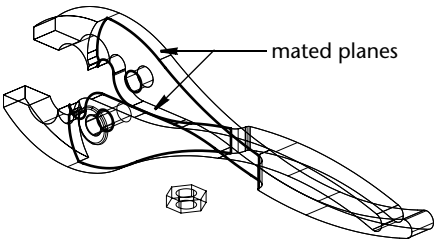




3 Select the second set of geometry to complete the mate constraint.

Select second set of geometry: *Select the inside face of* PLIERT *(6)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>:
Enter **a** *to highlight the face*
Second set = Plane
Enter an option [Clear/aXis/Point/Next/Flip/cYcle] <accEpt>:
Enter **y** *to cycle to the inside face, or press* ENTER
Enter offset <0.0000>: *Press* ENTER

The two parts of the pliers body are now plane mated on corresponding faces.

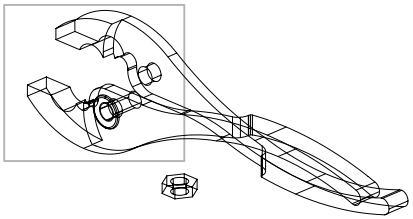


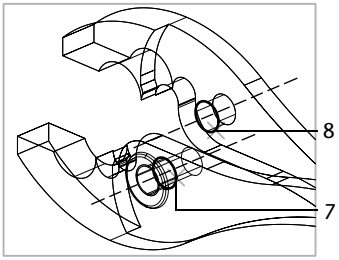
To mate parts on their axes

1 Mate the HEXBOLT part to the PLIERB part along their axes, responding to the prompts.

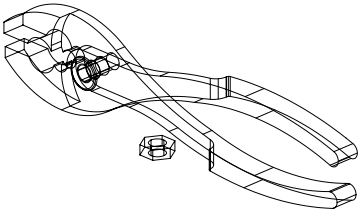
Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

Select first set of geometry: *Select the bolt hole of* PLIERT *(7)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Press* ENTER
Select second set of geometry: *Select the bolt hole of* PLIERB *(8)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>: *Press* ENTER
Enter offset <0.0000>: *Press* ENTER





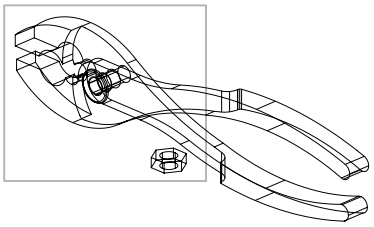
The PLIERB and PLIERT parts are line mated along the axes of the holes, and the bolt passes through both parts.

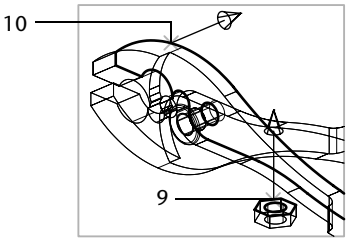


2 Mate the HEXNUT to the face of the pliers, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

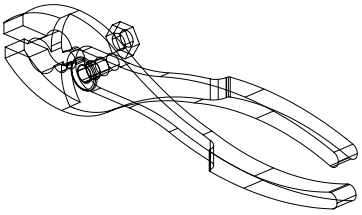
Select first set of geometry: *Select the top face of* HEXNUT *(9)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **a** *to highlight the face*
First set = Plane
Enter an option [Clear/aXis/Point/Next/Flip/cYcle] <accEpt>:
Enter **n** *to highlight the top face, if necessary, and then press* ENTER





3 Select the second set of geometry to complete the mate constraint.

Select second set of geometry: *Select the outside face of* PLIERB *(10)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>:
Enter **a** *to highlight the face*
Second set = Plane
Enter an option [Clear/aXis/Point/Next/Flip/cYcle] <accEpt>:
Enter **f** *to flip the direction arrow away from* PLIERB*, and then press* ENTER
Enter offset <0.0000>: *Press* ENTER

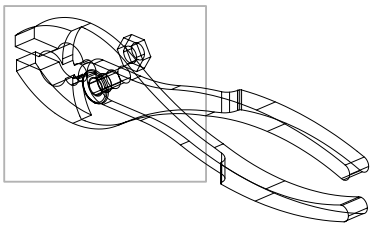


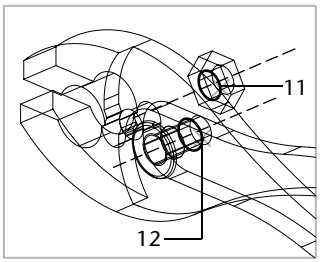
HEXNUT is plane mated to the face of PLIERB.

4 Mate the nut to the bolt along their axes to pass the bolt through the hole.

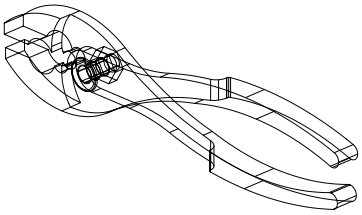
Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

Select first set of geometry: *Select the bolt hole on* HEXNUT *(11)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Press* ENTER
Select second set of geometry: *Select the bolt hole on* PLIERB *(12)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>: *Press* ENTER
Enter offset <0.0000>: *Press* ENTER





The parts are now assembled.



5 Check that the parts are assembled correctly.

Desktop Menu View ➤ 3D Views ➤ Top

6 Return to the isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

Save your file.

Using the Desktop Browser

As you assemble parts, a graphical hierarchy of the assembly is illustrated in the Desktop Browser. Each 3D constraint applied to an assembly component is listed below the component. You can tell at a glance which constraints exist between which components, because the other component to which the constraint applies is shown in the hierarchy. When you hold the cursor over a constraint, feature, or tweak in the Browser, a tooltip displays pertinent data. For instance, the offset value is displayed for a mate constraint, type and size data is displayed for a hole feature, and a distance value is displayed for a tweak. In scenes, explosion factors are displayed in a tooltip.

Using the Browser, you can both edit the hierarchy of an assembly and move components into an empty assembly file. To restructure the hierarchy, you either drag components, or use the cut and paste options in cases where the Browser display is long and detailed. The design constraints for restructured assemblies remain unchanged.

For more information about using in the Browser to restructure assemblies, see "Restructuring Assemblies" on page 504 in chapter 18.

You can use the Browser to add, change, delete, and copy, and to localize and externalize component definitions. To see how this works, edit the mate constraint on the hexbolt. Then change the color of a part, make copies, and externalize a component definition.

To edit an assembly constraint with the Browser

1 Change your display to two viewports.

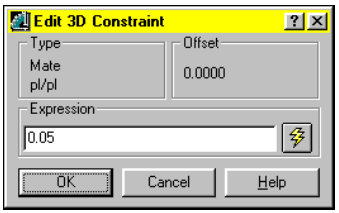
Command 2

2 Zoom to enlarge the hexbolt.

Context Menu In the graphics area, right-click and choose Zoom.

3 In the Browser, right-click the Mate pl/pl constraint under HEXBOLT_1, and choose Edit.

4 In the Edit 3D Constraint dialog box, change the offset to **0.05**. Click the Update icon.



You can see the hexbolt offset the new distance. Experiment with changing the offset again, using the Update icon to see the change.

Choose Cancel. The original offset distance of 0.00 remains in effect.

You can easily edit values and see them take effect without permanently changing your assembly. Use the Update icon in the Edit 3D Constraint dialog box to see the change, and then choose Cancel to return to the original values.

To copy a part with the Browser

1 Collapse the assembly tree to better visualize the changes you make.

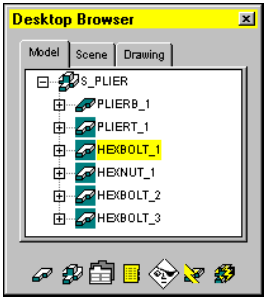
Browser Right-click S_PLIER and choose Browser Node ➤ Collapse.

2 Make a copy of HEXBOLT_1.

Browser Right-click HEXBOLT_1 and choose Copy.

3 Place one or more copies near the pliers, and then press ENTER.

You can see the new hexbolts listed in the Browser hierarchy, each numbered in the sequence you added them.



To change the color of a part with the Browser

1 Activate HEXBOLT_1.

Browser Right-click HEXBOLT_1 and choose Activate Part.

2 Open the Select Color dialog box to change the color of HEXBOLT_1.

Browser Right-click HEXBOLT_1 and choose Properties ➤ Color.

3 In the Select Color dialog box, select Red, and then choose OK.

This color applies to all instances of HEXBOLT.

4 Activate the assembly to view the changed color

Browser Right-click S_PLIER and choose Activate Assembly.

Next, change the bolt color to the original and remove the experimental copies.

5 Activate HEXBOLT_1

Browser Right-click HEXBOLT_1 and choose Activate Part.

6 Open the Select Color dialog box to change the color of HEXBOLT_1 back to white.

Browser Right-click HEXBOLT_1 and choose Properties ➤ Color.

7 In the Select Color dialog box, select White, and then choose OK.

8 Activate PLIERB_1 to return the bolts back to their original color.

Browser Right-click PLIERB_1 and choose Activate Part.

To delete part copies with the Browser

1 Delete the HEXBOLT copy.

Browser Right-click HEXBOLT_2 and choose Delete.

2 Repeat for the other copies if needed.

Your experimental copies are no longer visible in the Browser assembly hierarchy.

To localize an external part with the Browser

1 Localize the external parts HEXBOLT_1 and HEXBOLT_3.

Browser Press CTRL and select HEXBOLT_1 and HEXBOLT_3. Right-click the selected parts and choose All Instances ➤ Localize.

This shortcut method enables you to localize and externalize parts without opening the Assembly Catalog. When you select a local part in the Browser, the externalize option is available.

For more information about local and external assembly components, see "Creating Local and External Parts" on page 481.

You are now ready to check the assembly for mass properties and interference.

Getting Information from Assemblies

Now that all the parts are constrained, you can check for interference and obtain mass property information for some of the parts or the entire assembly.

Checking for interference between two or more parts in the assembly can detect design problems before manufacturing starts. Mass property information can tell you whether the dimensions or materials of a part need to be changed to achieve the desired results. This analysis is critical for making decisions early in the design phase that can prevent manufacturing problems later.

Checking for Interference

An interference analysis detects interference between two parts or all parts. You can move up or down the assembly tree, selecting parts as you proceed. Next, you will perform an interference analysis on the entire pliers assembly.

To check for interference

1 Change to one viewport.

Command 1

2 Change to the right isometric view.

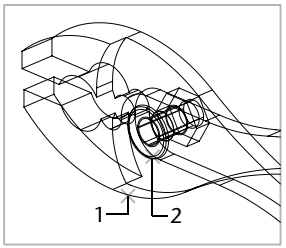
Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

3 Use AMINTERFERE to check for interference, responding to the prompts.

Context Menu In the graphics area, right-click and choose Analysis ➤ Check Interference.

Nested part or subassembly selection? [Yes/No] <No>: *Press* ENTER
Select first set of parts or subassemblies: *Select the* PLIERT *part (1)*
Select first set of parts or subassemblies: *Press* ENTER
Select second set of parts or subassemblies: *Select the* HEXBOLT *part (2)*
Select second set of parts or subassemblies: *Press* ENTER
Parts/subassemblies do not interfere.

If you have trouble selecting a part, use the Browser to change its color.



4 Check for interference between the PLIERB and HEXBOLT parts.

The parts should show no interference.

Next, you will calculate the mass properties for the assembly.

Calculating Mass Properties

Once an assembly is completed, you can calculate its mass properties. Tolerance values control the accuracy of the mass property information. The material types can be different for each part, resulting in accurate mass property values no matter how complex the assembly.

To calculate mass properties

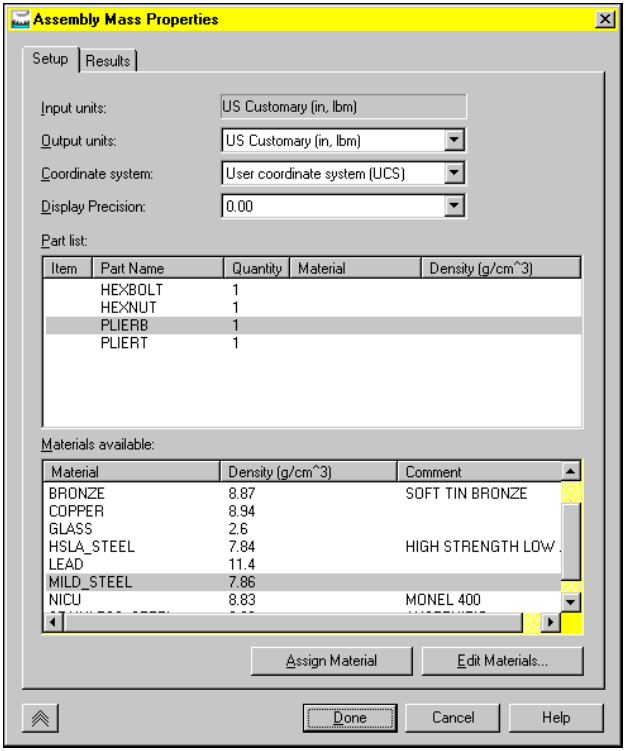
1 Use AMASSMPROP to select the parts for the mass properties calculation, following the prompts, and open the Assembly Mass Properties dialog box.

Context Menu In the graphics area, right-click and choose Analysis ➤ Mass Properties.

Enter an option (parts/subassemblies) [Name/Select] <Select>: *Press* ENTER
Select part and subassembly instances: *Select* PLIERB
Select part and subassembly instances: *Press* ENTER

2 In the Assembly Mass Properties dialog box, select the Setup tab and specify:

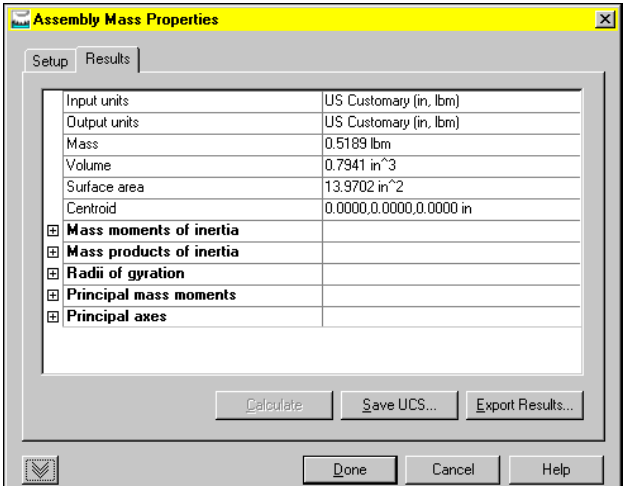
Input units: US Customary (in, lbm)
Output units: US Customary (in, lbm)
Coordinate system: User coordinate system (UCS)
Display Precision: *Enter* **2.0**
Part List: *Select* PLIERB
Materials available: *Select* Mild_Steel.



3 Select the Results tab. The results window for PLIERB remains empty until you calculate the results.

4 Choose Calculate. Message dialogs appear warning that density is not specified for the other parts in the assembly. Choose OK to proceed.

Mass properties are calculated according to the values you set.



Choose Done.

NOTE You can save mass properties calculations to a file to use in design analysis, and you can export the results.

Next, you create scenes of the assembly.

Creating Assembly Scenes

Now that you have instanced the parts and applied the assembly constraints, you can create and lay out assembly scenes. A scene is an exploded view that separates the parts of an assembly or subassembly to show how they fit together. You can create a scene quickly; it updates automatically every time you change the assembly. The separation of the parts is based on an explosion factor you set and the assembly constraints you used to position the parts. You can create multiple exploded scenes of the same assembly and save them for later use.

Assembly trails indicate the path of the assembly explosion. With the exception of the grounded part, assembly trails can be created for all parts. When you create assembly trails, a new layer is automatically created for them. You can automatically create trails when you create tweaks.

First, you set an explosion factor and then create an exploded assembly scene. Then you add trails to show how parts are assembled. From a scene, you create drawing views.

To create an exploded view

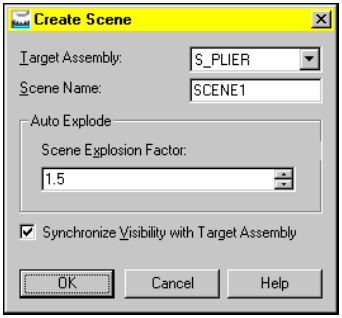
1 Use AMNEW to create a new scene.

Context Menu In the graphics area, right-click and choose New Scene.

The Create Scene dialog box is displayed.

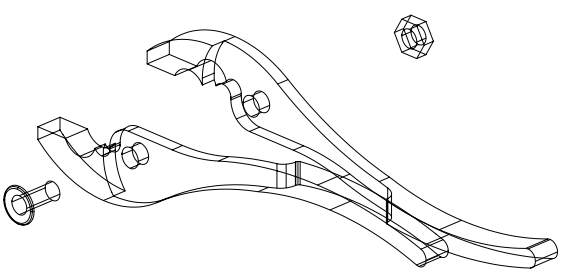
2 In the Create Scene dialog box, specify:

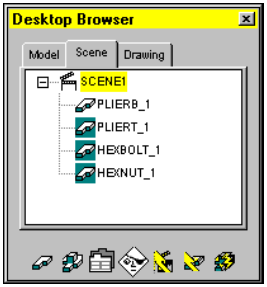
Target Assembly: S_PLIER
Scene Name: SCENE1
Auto Explode: Scene Explosion Factor: **1.5**
Synchronize Visibility with Target Assembly: *Select the check box*



Choose OK.

The exploded assembly scene is displayed. Its name is shown below the command line. The Browser shows all parts in the scene.





Next, align the exploded parts in the assembly scene. You can tweak the position and orientation of individual parts or rotate them for better visibility.

In the Browser, verify that the Scene tab is selected. Unless the scene is activated, you cannot tweak parts to adjust their position or add trails to show how they are assembled.

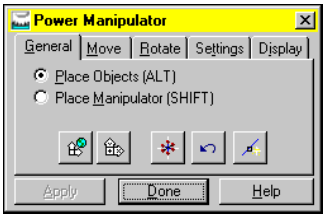
To align scene parts

1 Use AMTWEAK to move the HEXNUT part closer to the other parts, responding to the prompt.

Context Menu In the graphics area, right-click and choose New Tweak.

Select part or subassembly to tweak: *Select* HEXNUT *(1)*

2 In the Power Manipulator dialog box, on the General tab, verify that Place Objects (ALT) is selected.

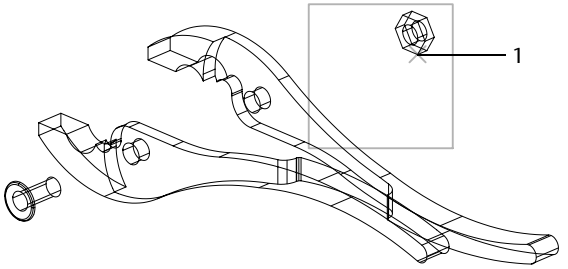


Choose Done.

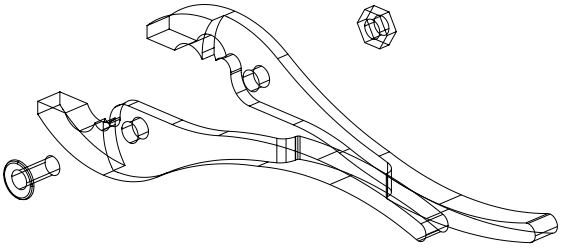
The Power Manipulator dialog box is displayed automatically only the first time you create a new tweak. After that, to access the Power Manipulator dialog box, right-click the Power Manipulator symbol on your screen, and select Options.

3 Continue on the command line.

Select handle or Geometry
[undo/UCS/WCS/Select/Options/Pancenter/Type/tRails/X/Y/Z] <Accept>:
Enter **z**
Enter tweak distance [Rotate]<1.0000>: *Enter* **.25**
Select handle or Geometry
[undo/UCS/WCS/Select/Options/Pancenter/Type/tRails/X/Y/Z] <Accept>:
Press ENTER.



The HEXNUT position is tweaked by the specified distance.

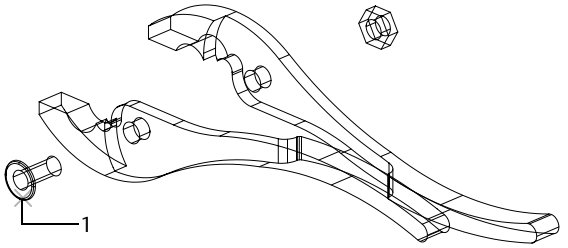


NOTE The grounded part of an assembly or subassembly cannot be tweaked. Its position is fixed.

4 Use AMTRAIL to show the direction of the explosion and tweak paths, responding to the prompt.

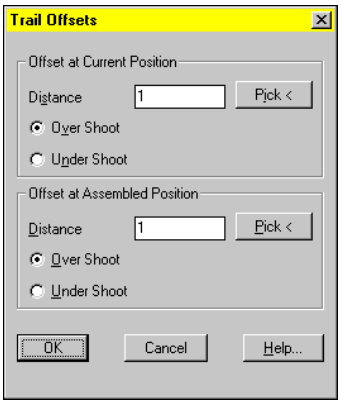
Context Menu In the graphics area, right-click and choose New Trail.

Select reference point on part or subassembly: *Select the end of* HEXBOLT *(1)*



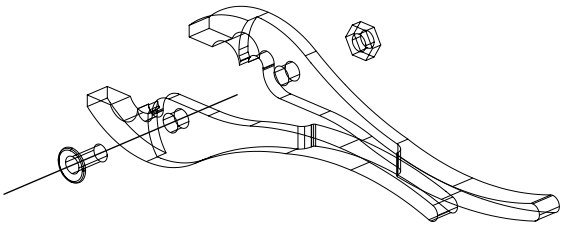
5 In the Trail Offsets dialog box, specify:

Offset at Current Position: Distance: *Enter* **1**
Over Shoot: *Select the option*
Offset at Assembled Position: Distance: *Enter* **1**
Over Shoot: *Select the option*



Choose OK.

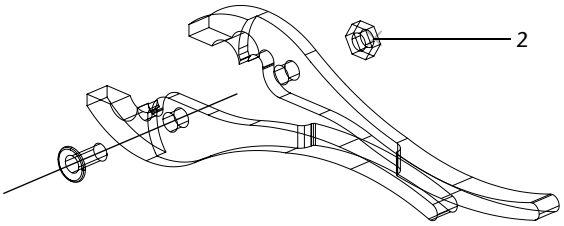
The assembly trail for HEXBOLT is displayed.



6 Apply assembly trails to the PLIERB and HEXNUT parts, responding to the prompt.

Context Menu In the graphics area, right-click and choose New Trail.

Select reference point on part or subassembly:
Select the outside hole of HEXNUT *(2)*

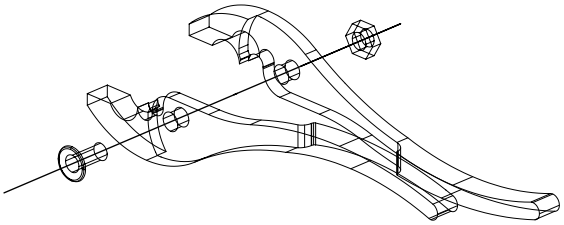


7 In the Trail Offsets dialog box, specify:

Offset at Current Position: Distance: *Enter* **1**
Over Shoot: *Select the option*
Offset at Assembled Position: Distance: *Enter* **1**
Over Shoot: *Select the option*

Choose OK.

The assembly drawing automatically updates the current scene to reflect the tweaks and assembly trails.



Save your file. Choose OK in the External Part Save dialog box to bring all parts up to date.

Next, you create drawing views of the assembly scene.

Creating Assembly Drawing Views

After you complete the assembly model and the scene, you can create 2D orthogonal views or 3D isometric and exploded views of the entire assembly. Then you add reference dimensions.

You can set up as many drawing layouts as you need to document your design. Because this assembly is small, you create only one layout.

Before creating the base view of the pliers assembly, modify a plotter setup to include a custom paper size. Then set up the drawing layout to arrange the drawing views on paper.

To add a custom paper size to a plotter setup

1 Use AMMODE to switch to Drawing mode.

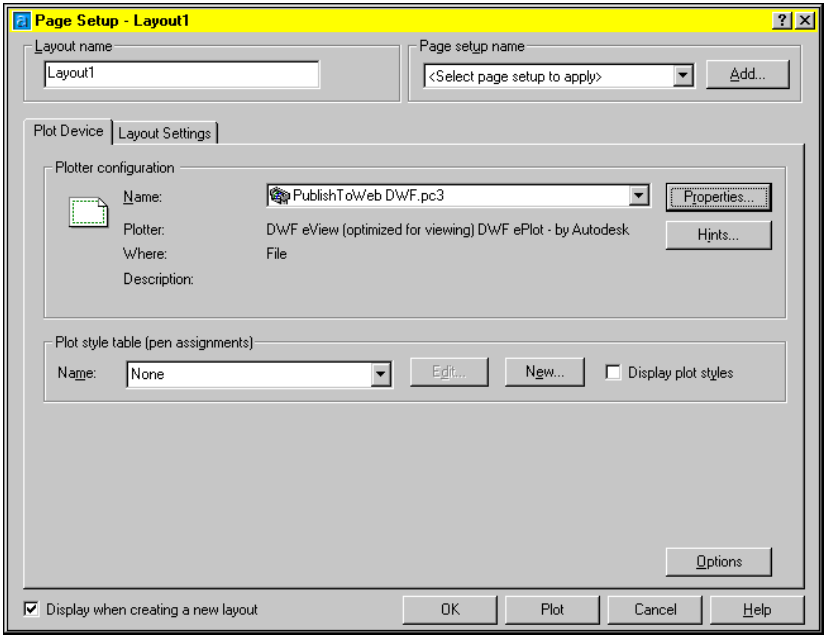
Browser Choose the Drawing tab.

2 Add a custom paper size to an existing plotter.

Browser Right-click Layout1 and choose Page Setup.

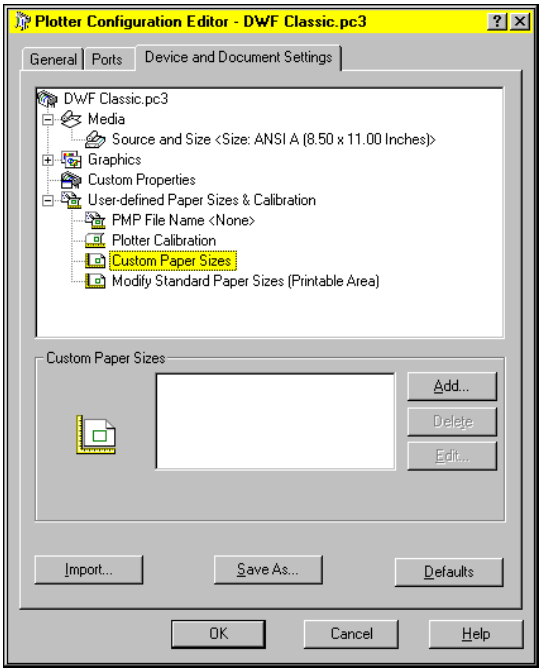
3 In the Page Setup dialog box, select the Plot Device tab and specify:

Name: DWF Classic.pc3



4 Choose Properties.

5 In the Plotter Configuration Editor dialog box, expand User-Defined Paper Sizes and Calibration. Select Custom Paper Sizes and choose Add.



6 Use the Custom Paper Size Wizard to define a paper size of 18 x 12 inches with no indents. Choose Next until the setup is finished.

7 Save your changes to the *DWF Classic.pc3* file.

Next, you set up the drawing layout and insert a title block.

To set up a drawing layout

1 In the Page Setup dialog box, select the Layout Settings tab and specify:

Paper Size: User 1 (18.00 x 12.00 inches)
Plot Scale: 1:1

Choose OK.

2 Use MVSETUP to insert a title block.

Browser Right-click Layout1 and choose Insert Title Block.

In the AutoCAD text window, respond to the prompt as follows:

Enter number of title block to load or [Add/Delete/Redisplay]: *Enter* **8**

3 Continue on the command line.

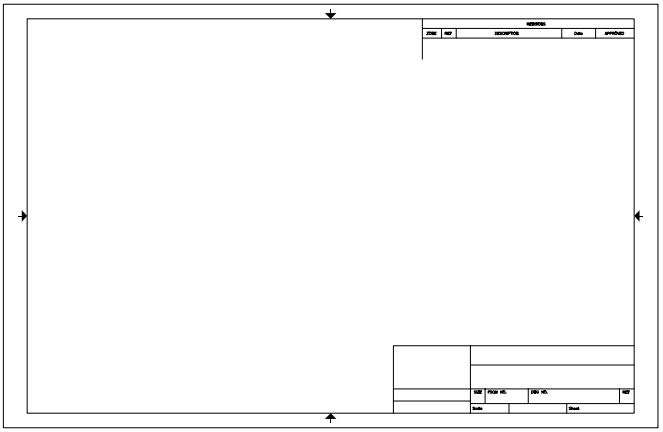
Create a drawing named ansi_b.dwg? <Y>: *Enter* **n**
Enter an option [Align/Create/Scale viewports/Options/Title block/Undo]:
Press ENTER

The title block is inserted into the drawing.

4 Use MOVE to center the title block in the layout, responding to the prompts.

Desktop Menu Modify ➤ Move

Select objects: *Select the title block*
Select objects: *Press* ENTER
Specify base point or displacement: *Specify a point*
Specify second point of displacement or <use first point as displacement>:
Specify another point



Next, define drawing views to display the assembly.

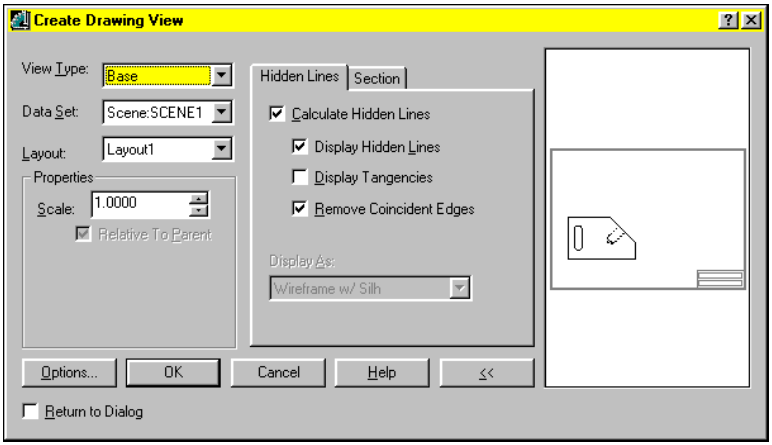
To create a base assembly drawing view

1 Use AMDWGVIEW to create a base view.

Context Menu In the graphics area, right-click and choose New View.

In the Create Drawing View dialog box, specify:

Type: Base
Data Set: Scene: SCENE1

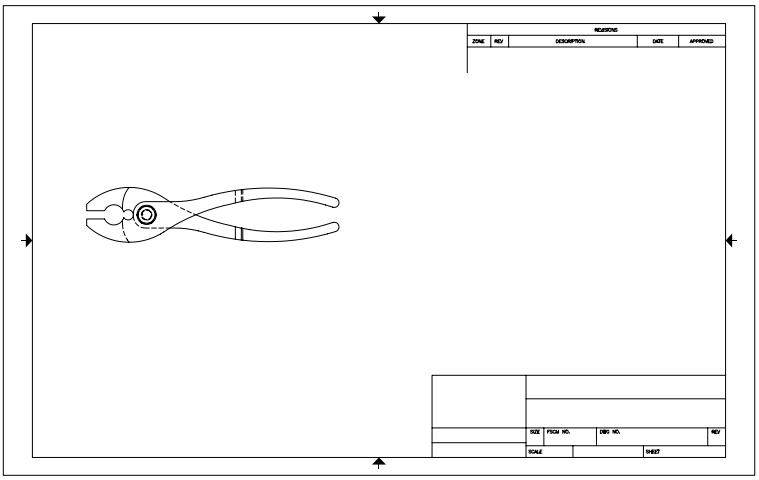


Choose OK.

2 Respond to the prompts as follows:

Select a planar face, work plane, or [Ucs/View/worldXy/worldYz/worldZx]:
Enter **z**
Select work axis or straight edge [worldX/worldY/worldZ]: *Enter* **x**
Adjust orientation [Flip/Rotate] <accEpt>: *Press* ENTER
Specify location of base view: *Specify a point*
Specify location of base view: *Specify another point or press* ENTER

The base drawing view is displayed.



Next, create an isometric view of the assembly.

To create an isometric assembly drawing view

1 Create an isometric view.

Context Menu In the graphics area, right-click and choose New View.

In the Create Drawing View dialog box, specify:

View Type: Iso

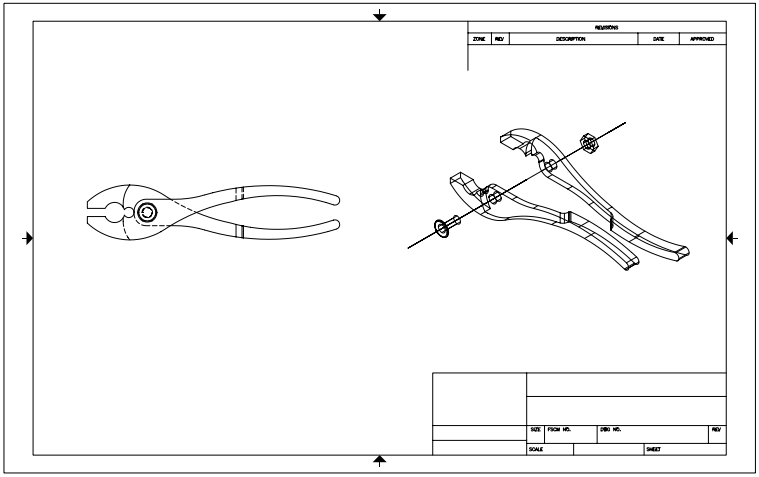
In the Hidden Lines tab, specify:

Calculate Hidden Lines: *Clear the check box*

Choose OK.

2 Respond to the prompts as follows:

Select parent view: *Specify the base view*
Specify location for isometric view: *Specify a point to place the isometric view*
Specify location for isometric view: *Specify another point or press* ENTER



Examine the Browser. The views are nested under a Scene icon which is nested under Layout1.

Save your file.

Now you can add reference dimensions, which can be moved, frozen, and thawed in each drawing view. Reference dimensions are not parametric, but they update when the model changes.

To add a reference dimension

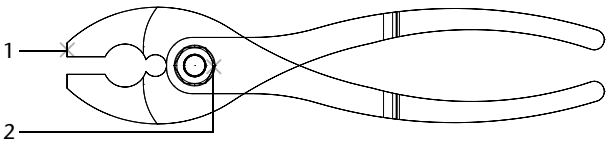
1 Zoom in to enlarge the area you want to dimension.

Context Menu In the graphics area, right-click and choose Zoom.

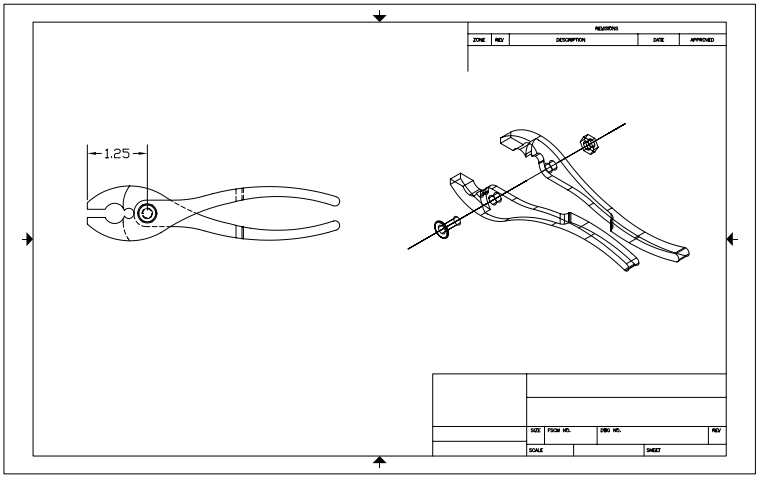
2 Use AMREFDIM to add a reference dimension, following the prompts.

Context Menu In the graphics area, right-click and choose Reference Dimension.

Select first object: *Select the endpoint of the pliers (1)*
Select second object or place dimension: *Select the bolt (2)*
Specify dimension placement: *Specify a point above the pliers*
Specify placement point or [Undo/Hor/Ver/Align/Par/aNgle/Ord/reF/Basic]:
Press ENTER
Select first object: *Press* ENTER



The reference dimension is added to the current view.



Save your file.

Now that you have created drawing views, you edit a part and automatically update the drawing views.

Editing Assemblies

Design or specification changes require most assembly designs to be documented and edited frequently. You modify parts, rearrange parts and features in the hierarchy of the assembly tree, and change or delete constraints. Because the parts and assembly are parametric, changes are fast and updates are immediate.

Editing an external part definition automatically changes the assembly model wherever the part is instanced.

Editing External Subassemblies

The process for editing external subassemblies and combined parts is much like the one for editing external parts. To begin, you activate the subassembly by double-clicking it in the Browser or entering information on the command line. When external files are active in the ref-edit state, they are not available for simultaneous use by others.

The active subassembly retains its color on the screen, while other geometry is dimmed. Non-active instances of the active subassembly dim one-half as much. This indicates which instance is active and reflects all of the instances that will be updated as a result of the change.

When active, you can edit the subassembly. The editing operations take place as if the active subassembly was an open document. You alter subassemblies by adding or removing components, changing constraints, adding new features, or restructuring the assembly. Newly created non-part based geometry such as surfaces and wires are created in the master assembly until made part of a component.

You can save your changes to an external subassembly file in several ways.

- Use an option in AMUPDATE to commit the changes to the external file.
- Activate a part or subassembly that is contained in another file and, in the resulting dialog box, commit to changes.
- Activate another subassembly that is external to the active subassembly.
- Save the assembly and choose either to not save changes or to save the pending changes to any external files.

Editing External Parts

To update all instances of a part in assemblies, you need to edit the original part. You alter part features by changing dimensions, changing the constraints, or adding new features. The changes take effect in the assembly.

In this tutorial, you edit the external PLIERT part from within your assembly drawing. This is called *editing in place*. In the following steps, you add another hole to the PLIERT part, and modify assembly constraints.

To edit an external part in place

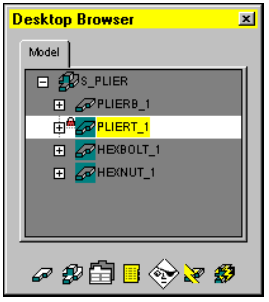
1 Use AMMODE to return to Model mode.

Browser Select the Model tab.

2 Use AMACTIVATE to activate the PLIERT part.

Context Menu In the graphics area, right-click and choose Part ➤ Activate Part.

The inactive parts are grayed out in the Desktop Browser, and dimmed on screen.



By dimming the inactive parts, it is easier for you to work on the active external part without moving other parts.

3 Notice the red lock preceding the PLIERT icon in the browser. This indicates that the file has been locked and cannot be modified by another user.

NOTE For clarity, the visibility of the other parts has been turned off in this section. If you prefer, turn off the visibility of PLIERB, HEXBOLT, and HEXNUT before you activate PLIERT.

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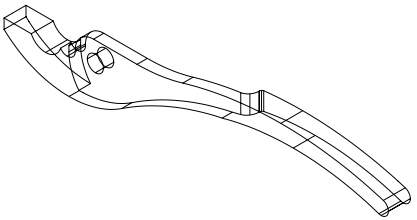
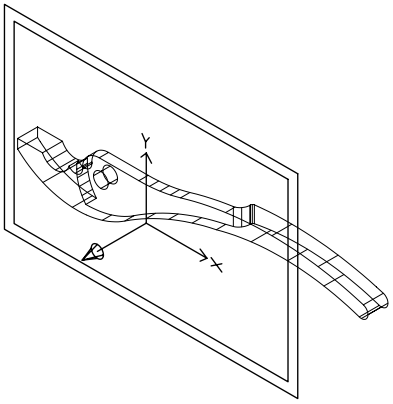
4 Zoom in on PLIERT.



5 Use AMSKPLN to create a new sketch plane, responding to the prompts.

Context Menu In the graphics area, right-click and choose New Sketch Plane.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Make sure the front face of the part is highlighted, and press ENTER
Plane=Parametric
Select edge to align X axis or [Z-flip/Rotate] <accept>:
Make sure the UCS icon is upright and the Z axis points away from the part, and then press ENTER



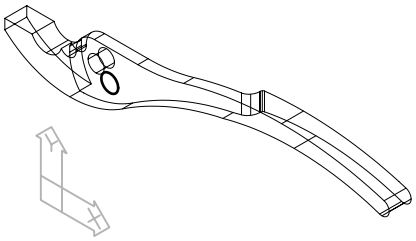
The sketch plane is defined on the front face of the pliers. Now, create the new slip hole to increase movement of the slip pliers.

To create a new feature on an external part

1 Use CIRCLE to place a new hole, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Circle.

CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Select a point near the existing hole
Specify radius of circle or [Diameter]:
Draw a circle approximately the same size as the other hole



2 Use AMPROFILE to solve the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

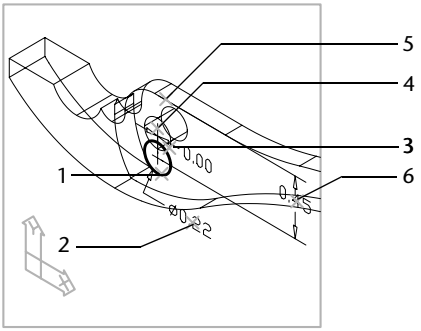
Mechanical Desktop indicates that three dimensions are required to fully constrain the sketch.

3 Use AMPARDIM to add the parametric dimensions, responding to the prompts. Zoom in as needed to magnify the holes.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Select the circle (1)*
Select second object or place dimension:
Specify a point for the location of the dimension (2)
Enter dimension value or [Undo/Radius/Ordinate/Placement point]
<0.2003>: *Enter* **.22**
Solved underconstrained sketch requiring 2 dimensions or constraints.
Select first object: *Select the circle (1)*
Select second object or place dimension: *Select the existing extruded hole (3)*
Specify dimension placement: *Specify a point for the location of dimension (4)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace]
<0.0454>: *Enter* **0**
Solved underconstrained sketch requiring 1 dimensions or constraints.
Select first object: *Select the circle (1)*
Select second object or place dimension: *Select the top line on* PLIERT *(5)*
Specify dimension placement: *Specify a point for the location of dimension (6)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace]
<0.5093>: *Enter* **.45**
Solved fully constrained sketch.
Select first object: *Press* ENTER

The circle is now constrained.



4 Use AMEXTRUDE to extrude the new hole through the pliers.

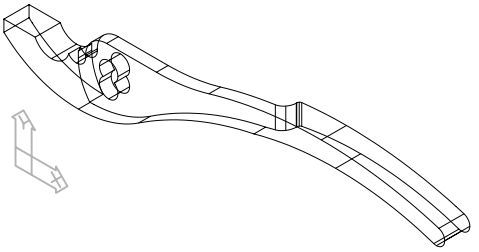
Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Termination: Through
Operation: Cut

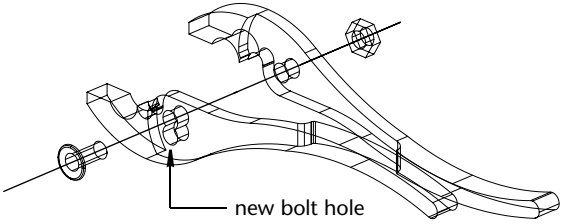
Choose OK to exit the dialog box.

The new hole is extruded, and its position is constrained to the original hole.



Save your file.

5 The External Part Save dialog box indicates a change in the PLIERT drawing. Choose OK to save the changes you have made.



The Browser returns to normal. The inactive parts are no longer dimmed, and the assembly reflects the new PLIERT part.

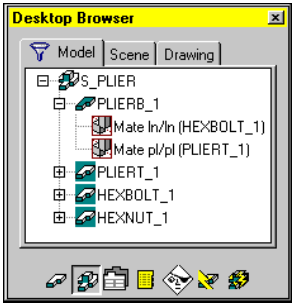
Editing Assembly Constraints

Using the Browser, you can selectively delete, edit, and add constraints to realign or change relationships of parts.

Use MOVE to reposition the parts as you edit and add constraints. If you do not want the parts to reassemble throughout editing, set AMAUTOASSEMBLE to **0** (off). To reassemble parts after you add a constraint, select the Update icon on the Desktop Browser.

To delete an assembly constraint

1 In the Desktop Browser, click the plus sign on PLIERB_1 to expand the hierarchy. Select the Assembly filter at the bottom of the Browser to filter out all information except the assembly constraints.



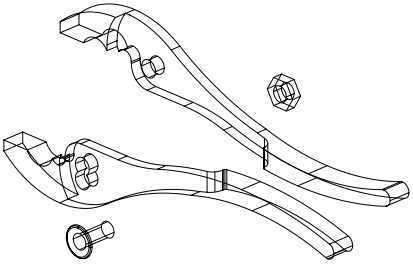
2 In the Browser, click the plus sign on HEXBOLT_1 to expand the hierarchy. Right-click the Mate ln/ln constraint of HEXBOLT_1, and choose Delete.

3 Delete the Mate pl/pl constraints of HEXBOLT_1.

4 Delete constraints for PLIERB, PLIERT, and HEXNUT.

5 Use MOVE to separate the parts to apply the new constraints.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Move.



Next, you apply new assembly constraints. You use an insert constraint to align HEXBOLT to the new hole along their axes while mating the face of PLIERT and the corresponding face of the bolt head.

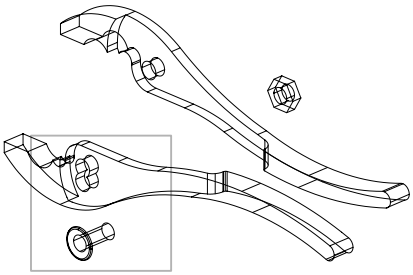
Use the DOF symbol to illustrate how many rigid body degrees of freedom are eliminated for each part.

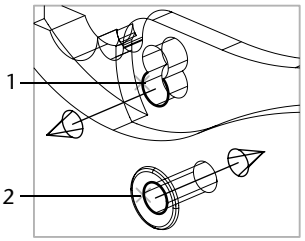
To apply an assembly constraint

1 Use AMINSERT to constrain PLIERT_1 and HEXBOLT_1, responding to the prompts.

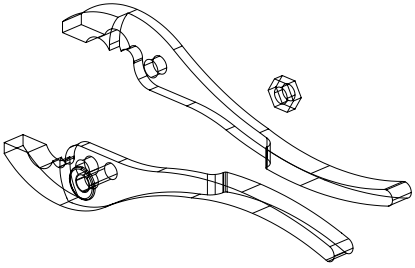
Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

Select first circular edge: *Select the new bolt hole of* PLIERT *(1)*
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>:
Flip the direction arrow toward HEXBOLT, *and press* ENTER
Select second circular edge: *Select the bolt shaft near the head (2)*
Second set = Plane/Axis
Enter an option [Clear/Flip] <accept>:
Flip the direction arrow toward PLIERT, *and press* ENTER
Enter offset <0.0000>: *Press* ENTER





The bolt is now inserted through the new hole.

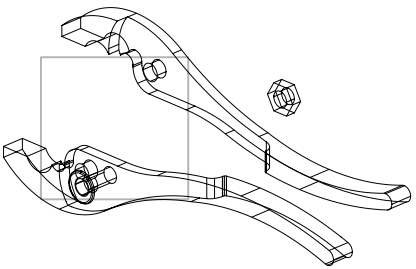


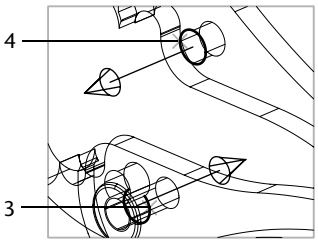
Now you can align the new hole in PLIERT and the corresponding hole in PLIERB along their axes.

2 Use AMINSERT to constrain PLIERB and PLIERT, responding to the prompts.

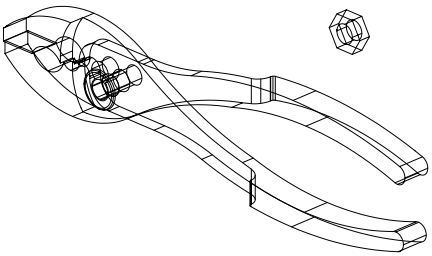
Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

Select first circular edge: *Select the new bolt hole of* PLIERT *(3)*
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>:
Flip the direction arrow toward PLIERB *and press* ENTER
Select second circular edge: *Select the hole on the inner face of* PLIERB (4)
Second set = Plane/Axis
Enter an option [Clear/Flip] <accept>:
Flip the direction arrow toward PLIERT *and press* ENTER
Enter offset <0.0000>: *Press* ENTER





The PLIERB and PLIERT parts are constrained to each other on the inside face and along the axes of the lower hole of PLIERT and the single hole of PLIERB. Transactional degrees of freedom are solved. In this case, you want to allow the pliers and hexbolt to rotate on the axes, so you leave the rotational degree of freedom unsolved.

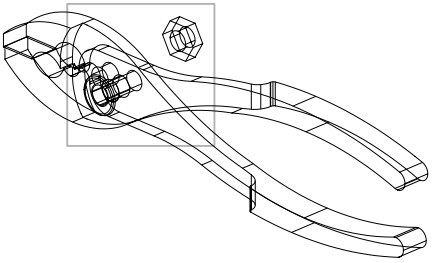


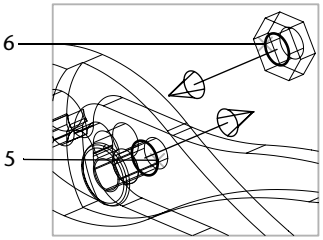
3 Use AMINSERT to constrain the facing planes of HEXNUT and PLIERB at the bolt holes, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

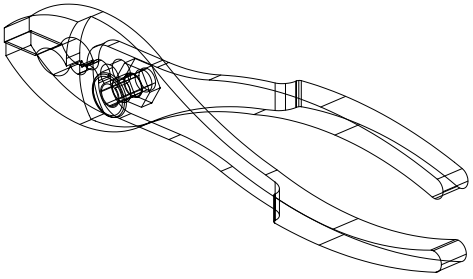
Move parts so you can easily see selection points.

Select first circular edge: *Select the bolt hole on* PLIERB (5)
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>:
Flip the direction arrow toward HEXNUT *and press* ENTER
Select second circular edge: *Select the hole on* HEXNUT *(6)*
Second set = Plane/Axis
Enter an option [Clear/Flip] <accept>:
Flip the direction arrow toward PLIERB *and press* ENTER
Enter offset <0.0000>: *Press* ENTER





The PLIERB and PLIERT parts are constrained on their facing planes. The bolt passes through both parts, and the holes and bolt shaft are aligned along their axes.



4 Check that the parts are assembled correctly.

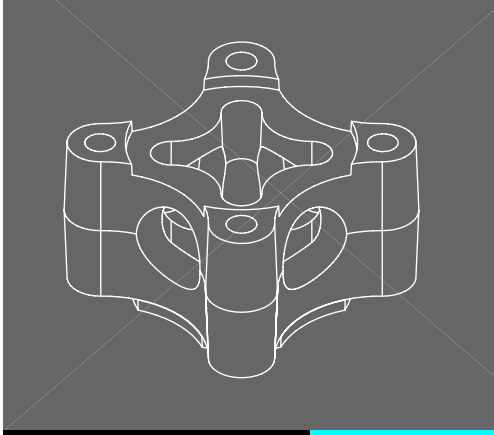
Desktop Menu View ➤ 3D Views ➤ Top

5 Return to the isometric view.

Desktop Menu View ➤ 3DViews ➤ Front Right Isometric

Save your file.

Combining Parts



In This Chapter

17

- Working in Single Part mode
- Changing part definitions
- Combining and intersecting parts
- Creating toolbody and nested toolbody parts
- Reducing weight parametrically

This Autodesk® Mechanical Desktop® tutorial builds on the part and assembly modeling techniques that you learned in previous chapters. In this chapter, you create a part and combine toolbodies with it, using parametric Boolean operations such as cut, join, and intersect, to construct a single part. You also learn how the display of complex parts is organized in the Desktop Browser.

In this tutorial, you work in Single Part mode to create a complex part to be used as a component for an off-road vehicle. You build the part by combining several toolbodies with a base part.

Key Terms

Term	Definition
base part	The active part where toolbody parts are aligned and subsequently combined.
Boolean modeling	A solid modeling technique in which two solids are combined to form one resulting solid. Boolean operations include cut, join, and intersect. Cut subtracts the volume of one solid from the other. Join unites two solid volumes. Intersect leaves only the volume shared by the two solids.
combine feature	A parametric feature resulting from the union, subtraction, or intersection of a base part with a toolbody part.
complex part	A parametric part containing one or more parametric parts as features.
Part Catalog	The means of attaching and cataloging local and external parts in the Part Modeling environment. Use the All and External tabs to specify contents, which can be instanced, copied, renamed, replaced, externalized, removed, localized and sorted.
part definition	Contains information about a part, including its name, geometric data, specifications, and parameters. If you instance a part multiple times, the assembly contains only one definition of the part.
part instance	A copy of the part definition. The part instance is inserted into the drawing and is visible as a solid object on the graphics screen. When a part definition is changed, so are all of its instances. Part instance names are displayed in the Desktop Browser.
toolbody	A part that is aligned with the base part and then used to join, intersect, or cut volume from the base part. In the Part Modeling environment, a part created after a base part, that automatically becomes an unconsumed toolbody.
toolbody consumption	When a toolbody part is combined with a base part, the toolbody part instance disappears from the graphics screen and appears as a new combine feature of the base part in the Desktop Browser.
toolbody rollback	A special option of the AMEDITFEAT command that enables you to change a toolbody part after it has been consumed as a combine feature.

Basic Concepts of Combining Parts

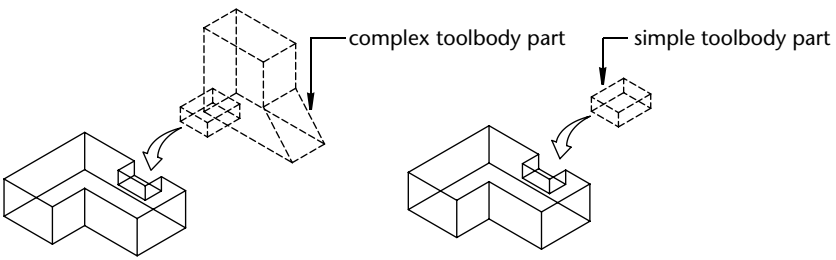
In Mechanical Desktop® the parametric Boolean capabilities for combining parts provide a combination of modeling flexibility and convenience. To combine two parts, you identify which part you want to use as the base part and make it active. Then, you position the toolbody part on the base part, using the MOVE or ROTATE command or assembly constraints. You use AMCOMBINE to cut, join, or intersect the toolbody part with the base part.

You can combine as many toolbodies with a base part as you like, but the base part and the toolbody must be instances of different parts. In other words, you cannot combine a part with an instance of itself.

Because the end result is a single part, you can create combined parts in Single Part mode. If you place more than one part in a part file, the additional parts automatically become unconsumed toolbodies.

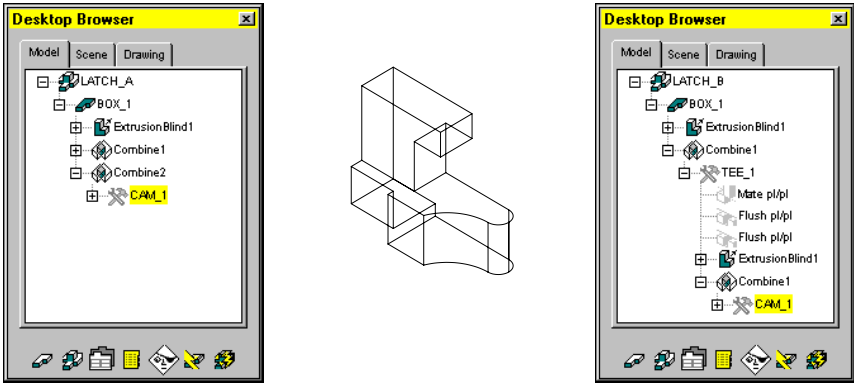
To combine a toolbody with a base part in an Assembly file, both parts must exist in the same active assembly.

When you create a complex part, the complete definitions of the toolbodies are stored in the assembly model file. To avoid creating files that are unnecessarily complex, use simple parts as toolbodies. In the following illustration, the highlighted parts are used to cut a slot. The resultant parts look identical, but the one created with the complex toolbody part consumes more disk space. Feature editing operations, such as cutting a slot, take longer.



With Mechanical Desktop, you can create toolbody parts that contain other toolbody parts. These are called *nested toolbodies*. However, you may be able to achieve the same result without nesting toolbodies.

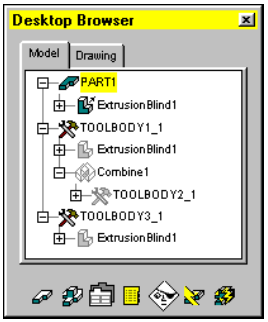
In the following illustration, the appearance of the part is the same, whether or not you nest the toolbodies, but the part displayed in the Desktop Browser on the left is easier to manage and has a less cumbersome display than the one in the Browser on the right.



To edit CAM_1, on the left, you need to expose only one toolbody. Nested toolbody parts, like those in the example on the right, usually have more complex constraint systems and require multiple part updates after modification.

Working in Single Part Mode

If you are creating combined parts, you can work in Single Part mode. In a single part file, you can only have one part definition, but you can work with more than one part. If you create or externally reference more than one part, the additional parts become unconsumed toolbodies that you can use to combine with the first part created in the drawing.



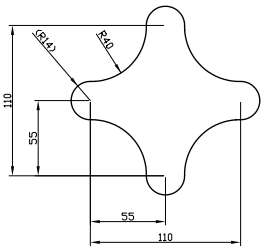
In the Browser above, TOOLBODY1 and TOOLBODY3 are unconsumed. TOOLBODY2 is consumed, since it has been combined with TOOLBODY1.

Creating Parts

In this tutorial, you create a chassis suspension component for an off-road recreational vehicle. The part is an axle spacer. You create most of the features of this part by first creating the basic shape. Then, you create separate parts that you use as tools to add additional features to the basic shape.

Open the file *spacer.dwg* in the *desktop\tutorial* folder. This drawing contains a fully constrained profile sketch of the basic shape of the axle spacer.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.



To create an axle spacer, you begin by extruding the part. First, review the constraint system for this sketch.

To extrude a part

1 Use AMSHOWCON to check the existing constraints.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.

2 Choose All.

Each arc uses the geometric constraints tangent and radius. The upper and lower outside arcs are aligned using the X Value constraint, and the left and right outside arcs use the Y Value constraint.

3 Press ENTER.

Because this part is cast aluminum, you must extrude it with a draft angle. Expand the part hierarchy by clicking the plus icon next to the part name in the Desktop Browser. The Browser shows an existing part, SPACER, that contains an unconsumed profile.

4 In the Desktop Browser, expand SPACER. Under SPACER, select the Profile1 icon. The sketch is highlighted.

5 Use VIEW to change your viewpoint to a previously saved view.

Desktop Menu View ➤ Named Views

In the View dialog box, select SPACER_VIEW, and choose Set Current.

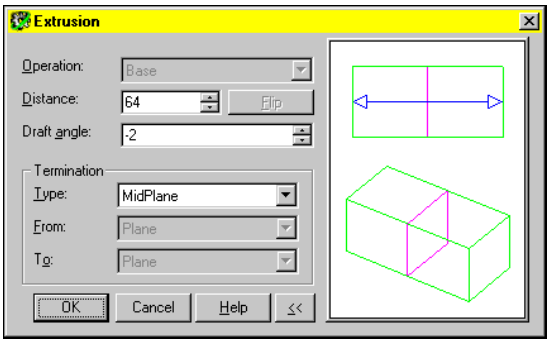
Choose OK.

6 Use AMEXTRUDE to extrude the profile.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Distance: *Enter* **64**
Draft Angle: *Enter* **-2**
Termination: Type: MidPlane



Choose OK.

Next, adjust the system settings so that you can hide the silhouette edges of your part.

To hide silhouette edges

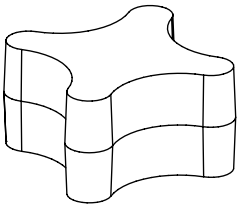
1 Set the AutoCAD system variable that controls the display of silhouette edges, responding to the prompt.

Command DISPSILH

Enter new value for DISPSILH <0>: *Enter* **1**

2 Use HIDE to hide the silhouette edges.

Desktop Menu View ➤ Hide



The spacer has a boss at the bottom and a relief at the top. Next, you use two part definitions to construct the toolbody parts. You combine those toolbody parts with the spacer to create the boss and relief.

Creating Toolbody Part Definitions

The shapes of the new toolbody parts are similar to the shape of the spacer profile. The easiest way to create the toolbodies is to use copies of the spacer to construct the new toolbody parts. Because you cannot copy a base part definition in the Part Modeling environment, you use the Part Catalog to attach a copy of the part to the current drawing as a toolbody definition.

To externally reference a toolbody definition

1 Change the display back to wireframe.

Desktop Menu View ➤ Shade ➤ 2D Wireframe, and then View ➤ Regen

2 Use AMCATALOG to attach the *boss.dwg* file as a toolbody. This drawing is a duplicate of the spacer.

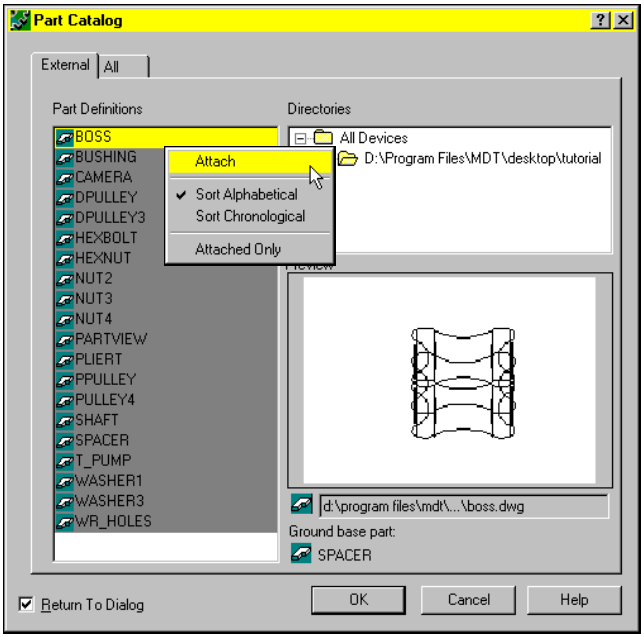
Context Menu In the graphics area, right-click and choose Toolbody Menu ➤ Catalog.

In the Part Catalog, choose the External tab and select Return to Dialog. Right-click in Directories, and choose Add Directory.

3 In the Browse for Folder dialog box, select the folder containing your tutorial drawings. Choose OK.

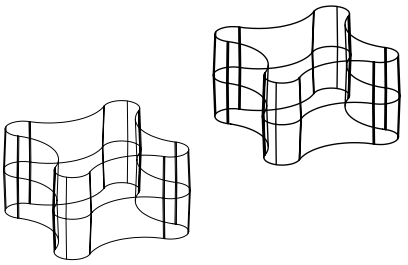
Because you are working in the Part Modeling environment, Mechanical Desktop filters the part and assembly drawings in your working directory and lists only the part files. A thumbnail preview of the part icon precedes the drawing name. If a part file does not contain features, it is preceded by a red AutoCAD icon.

4 In the Part Catalog, right-click BOSS and choose Attach.



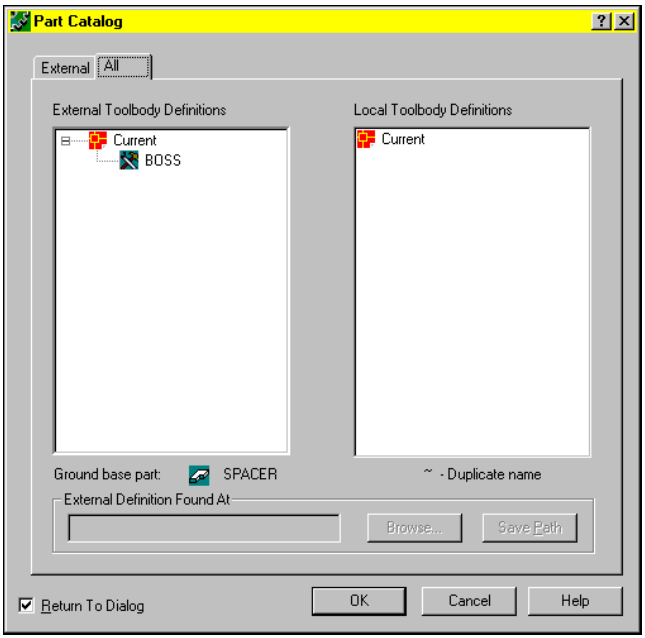
5 Respond to the prompts as follows:

Specify new insertion point: *Specify a point above and to the right of the spacer*
Specify insertion point for another instance or <continue>: *Press* ENTER



The Part Catalog is displayed.

6 Choose the All tab. The boss toolbody is listed in External Toolbody Definitions.



Choose OK.

Next, localize and make a copy of the boss toolbody, to create a definition for the relief toolbody using the Browser shortcut methods.

To localize an external toolbody and copy its definition

1 Localize external toolbody BOSS_1.

Browser Right-click BOSS_1, and choose All Instances ➤ Localize.

The boss toolbody is localized.

Next, copy the boss toolbody definition to create a relief toolbody.

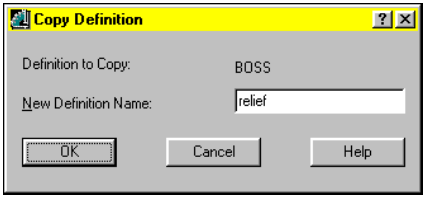
2 Copy the boss toolbody

Browser Right-click BOSS_1, and choose Show Definition.

3 In the Part Catalog, choose the All tab. The boss toolbody is listed in Local Toolbody Definitions.

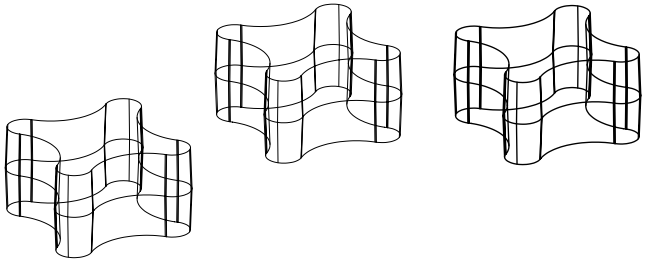
Right-click BOSS, and choose Copy Definition.

4 The Copy Definition dialog box is displayed. In New Definition Name , enter **relief**.



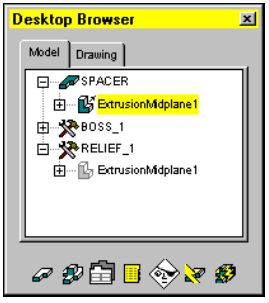
Choose OK.

5 Position the instance of the relief toolbody definition to the *right* of the boss toolbody, and press ENTER.



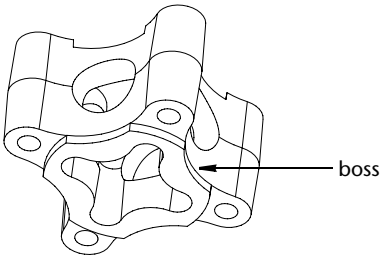
The new relief toolbody definition is listed under Local Toolbody Definitions in the Part Catalog. Choose OK.

Examine the Browser. It contains one part and two unconsumed toolbodies.



Save your file.

The boss toolbody on the completed spacer follows the profile of the spacer, but its corners are rounded. The next step is to combine a cylinder with the boss toolbody.



In the Browser, right-click BOSS_1 and choose Activate Toolbody. Right-click BOSS_1 again, and choose Zoom To.

To create a cylinder toolbody to combine with the boss toolbody

1 Use AMNEW to create a new toolbody definition, responding to the prompts.

Context Menu In the graphics area, right-click and choose Toolbody ➤ New Toolbody.

Enter an option [Instance/Part} <Part>: *Press* ENTER
Select an object or enter a new part name <TOOLBODY1>:
Enter **boss_cylinder** *and press* ENTER

The new toolbody is created, and the toolbody name is added to the Browser.

In the graphics area, right-click and choose Part Menu.

2 Use CIRCLE to create a circle close to the boss toolbody.

Context Menu In the graphics area, right-click and choose Part Menu ➤ 2D Sketching ➤ Circle.

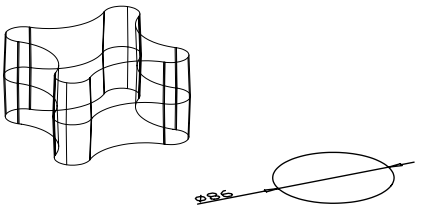
3 Use AMPROFILE to create a profile from the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

4 Use AMPARDIM to constrain the profile.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

5 Select the circle, and enter a dimension of **86**.



6 Use AMEXTRUDE to extrude the profile.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Distance: *Enter* **5**
Draft Angle: *Enter* **2**
Termination: Type: Blind

Choose OK.

Next, you use assembly constraints to position the cylinder at the bottom of the BOSS_1 toolbody. Then you use a Boolean intersect operation to combine the two parts.

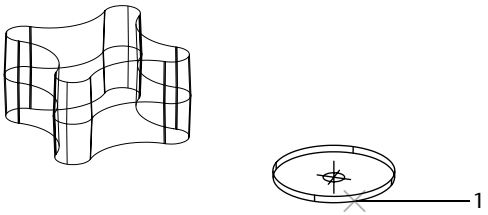
To align the cylinder with the boss toolbody, you create two mate-line constraints. Follow the prompts carefully, using the illustrations as your guide to selecting the correct part edges.

To align a part with a relief toolbody

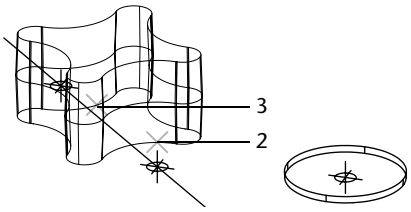
1 Use AMMATE to create a mate constraint, responding to the prompts.

Context Menu In the graphics area, right-click and choose Toolbody Menu ➤ 3D Constraints ➤ Mate.

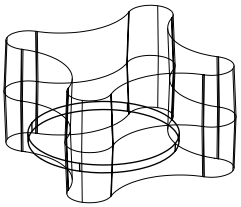
Select first set of geometry: *Select the bottom edge of the cylinder (1)*
First set = Axis, (arc)
Select first set or [Clear/Face/Point/cYcle] <accEpt>: *Enter* **p**
First set = Point, (arc)
Select first set or [Clear/aXis/fAce/cYcle] <accEpt>: *Press* ENTER



Select second set of geometry: *Select the arc (2)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **p**
Second set = Point, (arc)
Select second set or [Clear/aXis/fAce/cYcle] <accEpt>: *Select the arc (3)*
Second set = Plane, (arc)
Enter an option [Clear/aXis/Flip/cYcle] <accEpt>: *Enter* **x**
Second set = Axis, (arc)
Select first set or [Clear/fAce/Midpoint/cYcle] <accEpt>: *Enter* **m**
Second set = Axis, (arc)
Select first set or [Clear/fAce/Midpoint/cYcle] <accEpt>: *Press* ENTER
Enter offset <0>: *Press* ENTER



The center of the cylinder is aligned with the line between the two spacer arc centers.



2 Use MOVE to move the cylinder for easier selection, responding to the prompts.

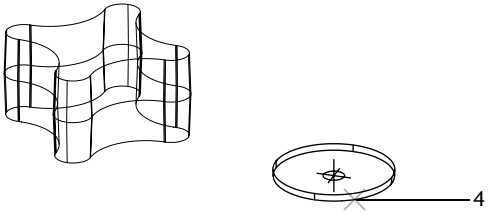
Context Menu In the graphics area, right-click and choose Part Menu ➤ 2D Sketching ➤ Move.

Select objects: *Specify the cylinder*
Select objects: *Press* ENTER
Base point or displacement: *Specify a point*
Second point of displacement: *Specify a second point and press* ENTER

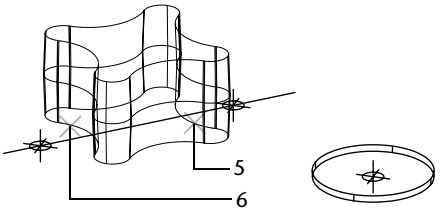
3 Create the second mate-line constraint, responding to the prompts.

Context Menu In the graphics area, right-click and choose Toolbody Menu ➤ 3D Constraints ➤ Mate.

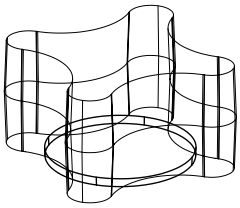
Select first set of geometry: *Select the bottom edge of the cylinder (4)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **p**
First set = Point, (arc)
Select first set or [Clear/aXis/fAce/cYcle] <accEpt>: *Press* ENTER



Select second set of geometry: *Select the arc (5)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **p**
Second set = Point, (arc)
Select second set or [Clear/aXis/fAce/cYcle] <accEpt>: *Select the arc (6)*
Second set = Plane, (arc)
Enter an option [Clear/aXis/Flip/cYcle] <accEpt>: *Enter* **x**
Second set = Axis, (arc)
Select first set or [Clear/fAce/Midpoint/cYcle] <accEpt>: *Press* ENTER
Enter offset <0>: *Press* ENTER



The center of the cylinder is aligned with the line between the two boss arc centers. Together, the two mate constraints position the cylinder at the bottom of the boss. The center of the cylinder is coincident with the center of the boss.



Now, you are ready to combine the boss toolbody with the cylinder. Because the boss toolbody will be the *base* part in the Boolean operation, you need to make it active.

To create a combine feature

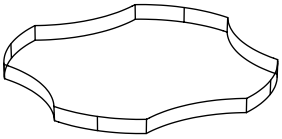
1 Use AMACTIVATE to activate BOSS_1.

Browser In the Browser, right-click BOSS_1 and choose Activate Toolbody.

2 Use AMCOMBINE to combine the toolbody and the cylinder, responding to the prompts.

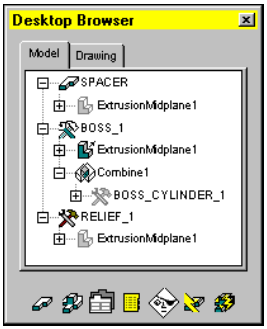
Context Menu In the graphics area, right-click and choose Part Menu ➤ Placed Features ➤ Combine.

Enter parametric boolean operation [Cut/Intersect/Join] <Cut>: *Enter* **i**
Select part (toolbody) to use for intersecting: *Select the cylinder*



Save your file.

The Desktop Browser now shows that the boss toolbody has a combine feature. The boss cylinder is a toolbody in the combine feature.



The next step is to constrain and combine the boss toolbody with the spacer.

To constrain and combine a toolbody to the base part

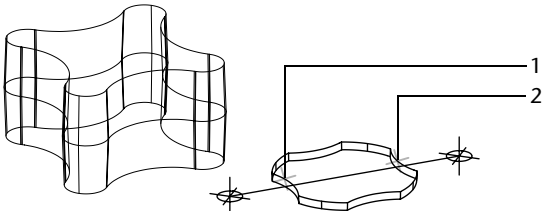
1 Use ACTIVATE to activate the SPACER.

Browser In the Browser, right-click SPACER and choose Activate Part.

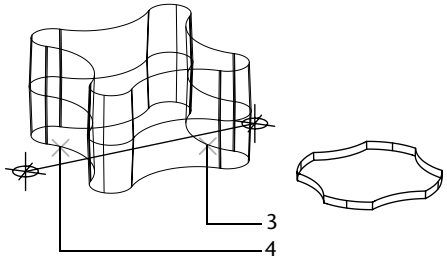
2 Use AMMATE to apply a mate constraint to the boss toolbody and the spacer, responding to the prompts.

Desktop Menu Toolbody ➤ 3D Constraints ➤ Mate.

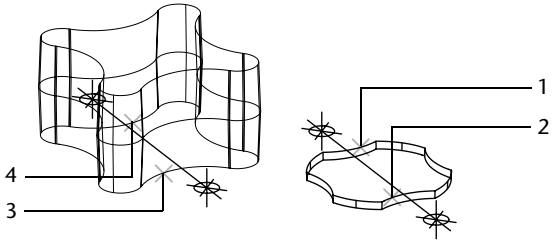
Select first set of geometry: *Select the top edge of the boss toolbody (1)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **p**
First set = Point, (arc)
Select first set or [Clear/aXis/fAce/cYcle] <accEpt>:
Select the opposite edge of the boss toolbody (2)
First set = Plane, (arc)
Enter an option [Clear/aXis/Flip/cYcle] <accEpt>: *Enter* **x**
First set = Axis, (arc)
Select first set or [Clear/fAce/Midpoint/cYcle] <accEpt>: *Press* ENTER



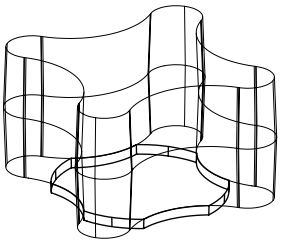
Select second set of geometry: *Select the bottom right edge of the spacer (3)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **p**
Second set = Point, (arc)
Select second set or [Clear/aXis/fAce/cYcle] <accEpt>:
Select the opposite edge of the spacer (4)
Second set = Plane, (arc)
Enter an option [Clear/aXis/Flip/cYcle] <accEpt>: *Enter* **x**
Second set = Axis, (arc)
Select second set or [Clear/fAce/Midpoint/cYcle] <accEpt>: *Press* ENTER
Enter offset <0>: *Press* ENTER



3 Move the boss toolbody, and repeat step 2 for the second constraint. Be sure to select the top edges of the boss toolbody.



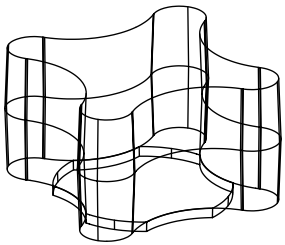
The boss toolbody is now aligned with the spacer.



4 Use AMCOMBINE to combine the spacer and the boss toolbody, responding to the prompts.

Context Menu In the graphics area, right-click and choose Part Menu ➤ Placed Features ➤ Combine.

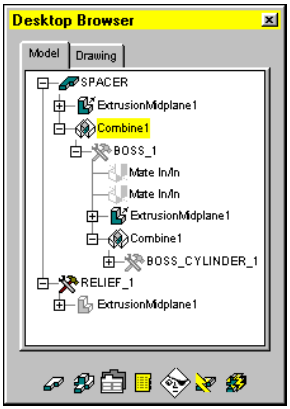
Enter parametric boolean operation [Cut/Intersect/Join] <Cut>: *Enter* **j**
Select part (toolbody) to be joined: *Select the boss toolbody*



Save your file.

Creating Relief Toolbodies

The Desktop Browser now shows a nested toolbody construction. The boss cylinder toolbody is a combine feature of the boss toolbody, and the boss toolbody is a combine feature of the spacer.



Next, you create the relief toolbody, to cut material from the spacer.

In the Browser, right-click RELIEF_1 and choose Zoom To.

To add a new toolbody name in the Browser

1 Use AMNEW to create a new toolbody called RELIEF_CYLINDER, responding to the prompts.

Context Menu In the graphics area, right-click and choose Toolbody Menu ➤ Toolbody ➤ New Toolbody.

Enter an option [Instance/Part} <Part>: *Press* ENTER
Select an object or enter new part name <TOOLBODY1>:
Enter **relief_cylinder** *and press* ENTER

The new part name is added to the Desktop Browser.

To create a new part

1 Use CIRCLE to draw a circle near RELIEF_1.

Context Menu In the graphics area, right-click and choose Part Menu ➤ 2D Sketching ➤ Circle.

2 Use AMPROFILE to create a profile from the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

3 Use AMPARDIM to constrain the profile.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select the circle, and enter a dimension of **90**.

4 Use AMEXTRUDE to extrude the profile.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion Feature dialog box specify:

Termination: Blind
Distance: *Enter* **10**
Draft Angle: *Enter* **2**

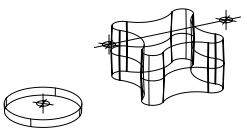
Choose OK.

Next, you position the cylinder at the top of RELIEF_1, using assembly constraints just as you did for the boss cylinder. As you select geometry for the constraints, be sure to select the top edges of both the relief cylinder and the relief toolbody.

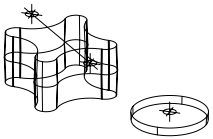
To constrain the toolbodies

1 Use AMMATE for two mate constraints to align the toolbodies.

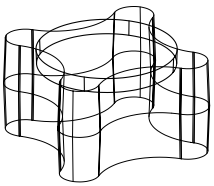
Desktop Menu In the Desktop Menu, choose Toolbody Menu ➤ Toolbody ➤3D Constraints ➤ Mate.



first mate constraint



second mate constraint



result

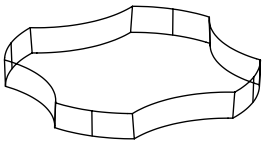
2 After adding the constraints, use AMACTIVATE to activate RELIEF_1.

Browser In the Browser, right-click RELIEF_1 and choose Activate Toolbody.

3 Combine the relief cylinder and the relief toolbody.

Context Menu In the graphics area, right-click and choose Part Menu ➤ Placed Features ➤ Combine.

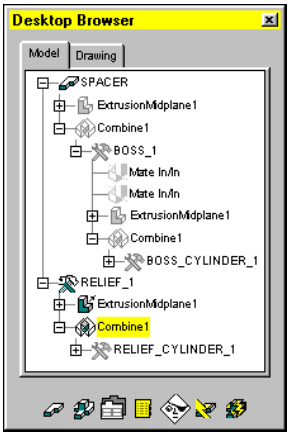
4 Choose Intersect, and select the relief cylinder as the toolbody.



Save your file.

Combining Toolbodies with Spacers

In the Desktop Browser, make sure that the relief toolbody has a combine feature and that it contains the relief cylinder toolbody.



In the Browser, right-click SPACER and choose Activate Part.

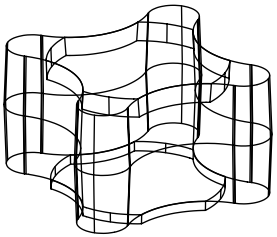
To combine a relief toolbody with a spacer

1 Use AMMATE for assembly constraints just as you did to align the relief toolbody with the spacer.

Desktop Menu Toolbody ➤ 3D Constraints ➤ Mate.

When you combine the spacer and the relief toolbody in step 3, you will cut the spacer with the toolbody. Therefore, be sure to align the top of the toolbody with the top of the spacer.

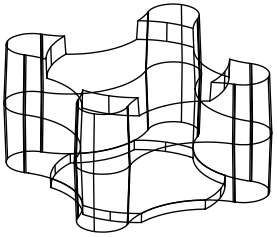
After you constrain the relief toolbody, your model should look like this:



2 Use AMCOMBINE to combine the spacer and the relief toolbody.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Combine.

3 Choose Cut, and select the relief toolbody.



Save your file.

Adding Weight Reduction Holes

The axle spacer is a high-performance chassis component, so its weight must be kept to a minimum. To achieve this, you cut weight reduction holes into the part. The manufacturer of the part offers several size spacers with different size weight reduction holes. The use of parametric Boolean operations is an ideal way to model the part, because it is easy to replace one combine feature with another.

The file *spacer.dwg* already contains the geometry you need to create a weight reduction extrusion that cuts material from the middle of the spacer. An external file contains the part that you will use to remove material from each of the spacer's four sides.

First, you attach the external file.

To minimize the weight of a part, using an external toolbody

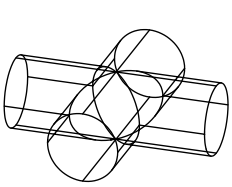
1 Use AMCATALOG to attach the weight reduction holes toolbody.

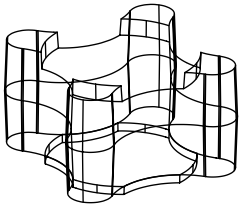
Context Menu In the graphics area, right-click and choose Catalog.

In the Part Catalog, choose the External tab. Clear the Return to Dialog check box. Right-click WR_HOLES, and choose Attach.

2 Respond to the prompts as follows:

Specify new insertion point: *Specify a point to the left of the spacer*
Specify insertion point for another instance or <continue> *Press* ENTER



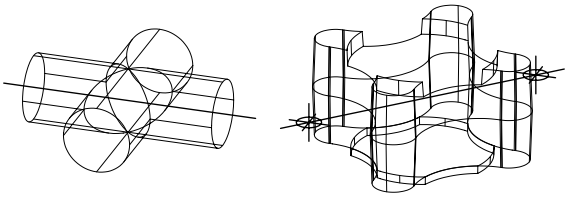


The spacer is created as a midplane extrusion. Therefore, the *parting line* appears as a profile that encircles the part at its midsection. When you constrain the weight reduction extrusion to the spacer, you select the parting-line geometry.

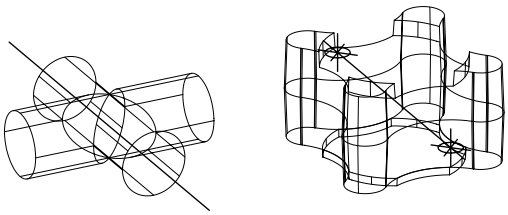
3 Use AMMATE to constrain the two parts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

4 Align the axis of one of the reduction extrusion cylinders with a line that runs through the center points of the spacer arcs. Use the point option when you define the axis, as you did with previous mate constraints.



5 Use another mate constraint to align the axis of the adjacent weight reduction extrusion cylinder with a line that runs through the center points of the spacer arcs.



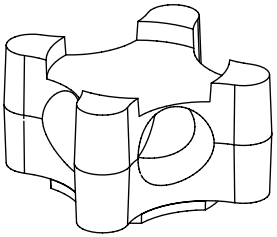
6 Make sure that the spacer is the active part, and use AMCOMBINE to combine the two parts.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Combine.

7 To cut the weight reduction extrusion from the spacer, choose Cut, and select the weight reduction extrusion as the toolbody.

8 Remove the hidden lines.

Desktop Menu View ➤ Hide



Save your file.

The weight reduction holes are very close to the relief cut. For balance, the holes must remain centered in the spacer. To provide enough material between the holes and the relief, you need to reduce the depth of the relief and the diameter of the holes.

To make the change, you edit the nested relief cylinder toolbody and reduce its extrusion distance.

NOTE When you edit more complex parts, it is sometimes easier to select commands from menus or toolbars instead of searching for the feature in the Browser and using the Browser menus.

To center the weight reduction holes

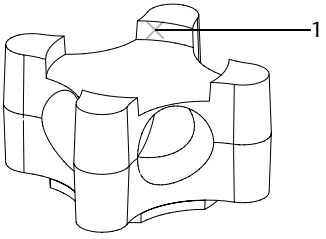
1 Return to wireframe display.

Desktop Menu View ➤ Shade ➤ 2D Wireframe, and then View ➤ Regen

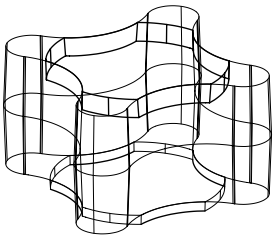
2 Use AMEDITFEAT to recover the relief toolbody, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Independent array instance/Sketch/surfCut/Toolbody/select Feature] <select Feature>: *Enter* **t**
Select parametric boolean to edit: *Select the edge of the relief toolbody (1)*
Enter an option [Accept/Next] <Accept>:
When the relief toolbody is highlighted, press ENTER



Mechanical Desktop recovers the toolbody and displays it in its constrained position on the spacer. The relief toolbody is active, and it contains the relief cylinder toolbody.

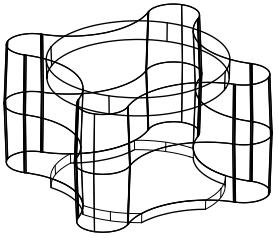


3 Use AMEDITFEAT to recover the relief cylinder, responding to the prompt.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Independent array instance/Sketch/surfCut/Toolbody/select Feature] <select Feature>: *Enter* **t**

Mechanical Desktop recovers the relief cylinder toolbody and displays it in its constrained position on the relief toolbody.



4 Change the thickness of the relief cylinder, responding to the prompt.

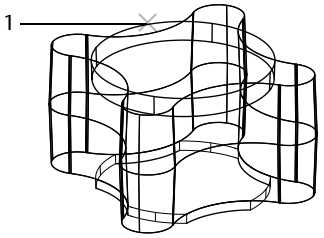
Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Independent array instance/Sketch/surfCut/Toolbody/select Feature] <select Feature>: *Select the cylinder (1)*

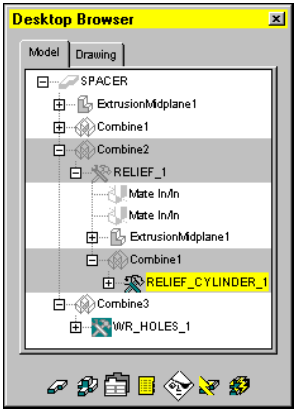
5 In the Extrusion dialog box, change the distance to **5**. Then choose OK.

6 Continue on the command line.

Select object: *Press* ENTER



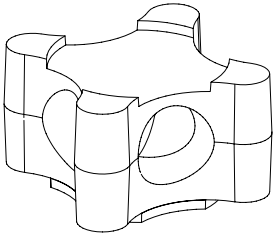
In the Browser, note that the relief toolbody and the relief cylinder toolbody have yellow backgrounds. This indicates that they need to be updated.



7 Use AMUPDATE to update the parts, responding to the prompts.

Context Menu In the graphics area, right-click and choose Update Full.

Toolbody Updates Pending: 2
Enter an option [Full/stEp/posiTioning] <Full>: *Press* ENTER *to update both parts*



Next, you change the diameter of the weight reduction holes. Because the toolbody is an external reference, you activate it first. Then you change the diameters of the cylinders.

To edit the weight reduction cylinders

1. In the Browser, right-click WR_HOLES_1 and choose Open to Edit.

   Mechanical Desktop opens the external file containing the weight reduction holes.
2. Expand WR_HOLES in the Browser.
3. Right-click ExtrusionMidplane1 and choose Edit.
4. Choose OK to exit the Extrusion dialog box.
5. Continue on the command line.

   Select object: *Specify the diameter dimension*
   Enter dimension value <42>: *Enter* **35**
   Solved fully constrained sketch.
   Select object: *Press* ENTER
6. Repeat steps 3 through 5 for the adjacent cylinder.

Next, commit your changes to the external file, and then update your combined part.

To commit changes to an external file

1. Use AMUPDATE to update the external part, responding to the prompts.

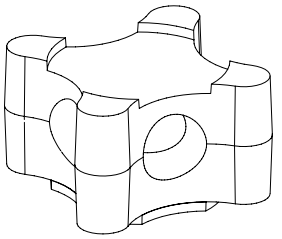
Context Menu In the graphics area, right-click and choose Part ➤ Update Part.

2. Save and close *wr_holes.dwg*.
3. Reload the external file.

Browser Right-click and choose Show Definition.

4. In the Part Catalog, under the All tab right-click WR_HOLES, and choose Reload.
5. Choose OK to exit the Part Catalog.
6. Use HIDE to remove the hidden lines to verify the design changes.

Desktop Menu View ➤ Hide



Save your file.

Adding Weight Reduction Extrusions

One more weight reduction extrusion remains. The geometry for the sketch is stored on the WEIGHT_REDUCTION_EXTRUSION layer.

To copy a sketch to create a new sketch

1 Return to wireframe display.

Desktop Menu View ➤ Shade ➤ 2D Wireframe, and then View ➤ Regen

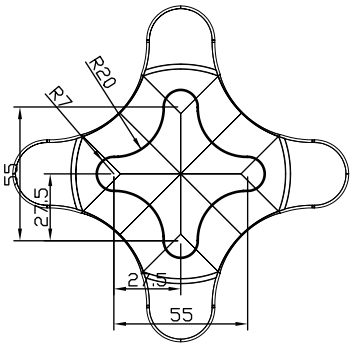
2 Use LAYER to turn on the WEIGHT_REDUCTION_EXTRUSION layer and make it current.

Desktop Menu Assist ➤ Format ➤ Layers

This sketch was easily constructed by creating a copy of the spacer profile sketch before it was consumed. Its scale was then reduced by 50 percent, using a base point at the center of the sketch.

3 Switch to a top view of your part.

Desktop Menu View ➤ 3D Views ➤ Top



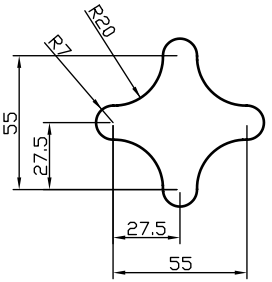
4 Use AMNEW to create a new toolbody.

Context Menu In the graphics area, right-click and choose Toolbody Menu ➤ Toolbody ➤ New Toolbody.

5 Enter the name **wt_reduction_extrusion**.

6 Turn off LAYER 0, which contains the spacer.

Desktop Menu Assist ➤ Format ➤ Layer



7 Use AMPROFILE to profile the sketch.

Context Menu In the graphics area, right-click and choose Part Menu ➤ Sketch Solving ➤ Profile.

8 Select the sketch and all of its existing dimensions.

Mechanical Desktop converts the standard dimensions to parametric dimensions and solves the sketch.

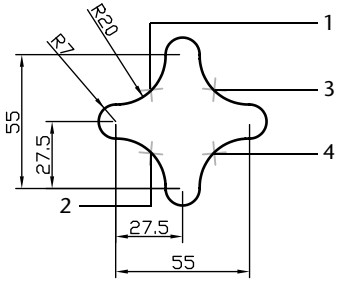
Solved underconstrained sketch requiring 2 dimensions or constraints.

To constrain and extrude sketches

1 Use AMADDCON to add two X Value constraints to the profile, responding to the prompts to fully constrain the sketch.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ X Value.

Valid selections: line, arc, circle or spline segment
Select object to be reoriented: *Select the arc (1)*
Valid selections: line, arc, circle or spline segment
Select object x value is based on: *Select the arc (2)*
Solved underconstrained sketch requiring 1 dimensions or constraints.



Valid selections: line, arc, circle or spline segment
Select object to be reoriented: *Select the arc (3)*
Valid selections: line, arc, circle or spline segment
Select object x value is based on: *Select the arc (4)*
Solved fully constrained sketch.
Valid selections: line, arc, circle or spline segment
Select object to be reoriented: *Press* ENTER
Enter an option
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix/eXit] <eXit>: *Press* ENTER

2 Use VIEW to restore the saved view.

Desktop Menu View ➤ Named Views

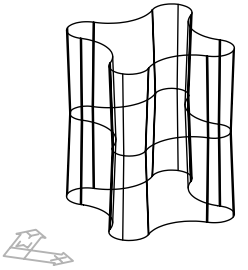
In the View dialog box, make SPACER_VIEW current, and choose OK.

3 Use AMEXTRUDE to extrude the profile.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion Feature dialog box, specify:

Termination: MidPlane
Distance: *Enter* **75**
Draft Angle: *Enter* **2**



Choose OK.

Next, combine the new toolbody with SPACER_1.

To combine a weight reduction extrusion with a spacer

1 Turn on LAYER 0, and make it current.

Desktop Menu Assist ➤ Format ➤ Layer

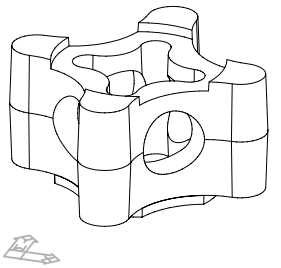
2 Activate the spacer, and then combine the weight reduction extrusion and the spacer.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Combine.

3 Choose Cut, to cut the weight reduction extrusion from the spacer, and then select the weight reduction extrusion as the toolbody.

4 Remove the hidden lines.

Desktop Menu View ➤ Hide



Save your file.

Adding Mounting Holes

The final step in your model is to add the mounting holes.

To add a mounting hole

1 Return to wireframe display.

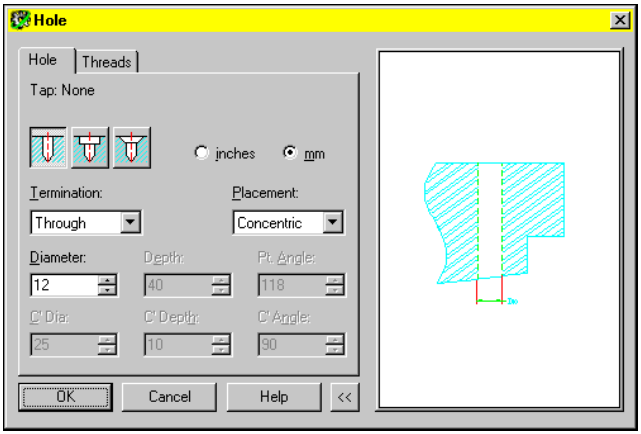
Desktop Menu View ➤ Shade ➤ 2D Wireframe

2 Use AMHOLE to create the mounting holes.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Hole.

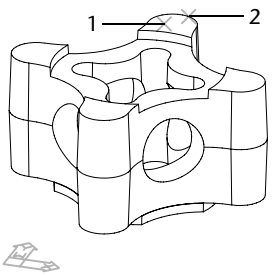
In the Hole dialog box, specify:

Operation: Drilled
Termination: Through
Placement: Concentric
Hole Parameter: Size: 12



3 Respond to the prompts as follows:

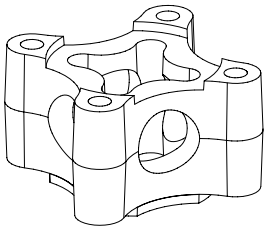
Select work plane, planar face, or [worldXy/worldYz/worldZx/Ucs]:
Select the top face (1)
Select concentric edge: *Select the cylindrical edge (2)*



4 Repeat steps 2 and 3 to create three more holes, and then press ENTER.

5 Use HIDE to remove the hidden lines.

Desktop Menu View ➤ Hide

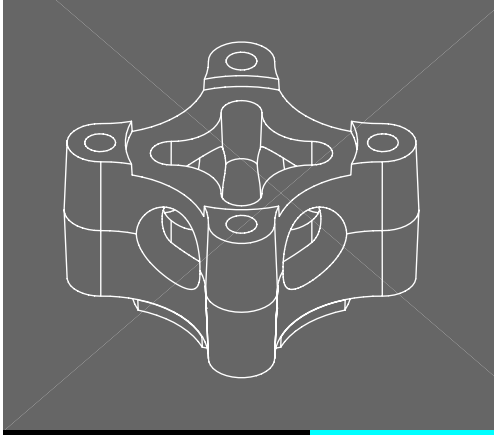


The spacer contains one extrusion, four combine features, and four holes.

Save your file.

You have now created and edited a combined part in the Part Modeling environment.

Assembling Complex Models



In This Chapter

18

In a previous Autodesk® Mechanical Desktop® tutorial, you created a simple assembly. In this tutorial, you create a more complex assembly that includes a subassembly. You work with contraints, external parts, and part instances. Then you examine your model for interference and edit an external part.

When the model is complete, you create assembly scenes with tweaks, drawing views, and annotations in preparation for plotting.

- Creating local and external parts
- Constraining parts to create an assembly
- Creating a new part
- Creating a subassembly and constraining it to the base assembly
- Creating and annotating assembly scenes
- Editing a part in a completed assembly
- Adding annotations
- Checking for interference
- Updating an assembly and its scenes
- Finishing a drawing for plotting

Key Terms

Term	Definition
base view	The first drawing view you create. Other drawing views are derived from this view.
bill of material database	A dynamic database containing a list of all the parts in an assembly. Used to generate parts lists that contain associated attributes such as part number, manufacturer, and quantity.
exploded view	Separates parts and subassemblies to show how they fit together. Automatically updated if the assembly or one of its parts changes.
explosion factor	Defines separation of parts in an assembly exploded view by a set value, plus offset, if applicable. Value is based on model units.
external part	A part reference that resides in a file other than the current part or assembly file.
flush constraint	Makes two planes coplanar, with their faces aligned in the same direction.
local part	A part created in the current assembly file.
parent view	A drawing view on which other views are based. For example, the base view is the parent view for an isometric or orthographic view.
part reference	An attributed object associated with a part. Used to provide information about the part when generating a parts list.
parts list	A dynamic list of parts and associated attributes generated from a bill of material database and placed in the drawing. The parts list automatically reflects additions and subtractions of parts from an assembly.
structure	The hierarchical tree of component to component relationships that define the assembly model. Assembly models can be restructured in the Browser.
subassembly	A group of parts constrained together. Can be used as a single object in a larger assembly. A subassembly may be created in the current assembly or referenced from an external file.
trail	In an exploded view in a scene, a line that shows how parts in an assembly are assembled.
tweak	Adjusts the position of parts in an assembly scene to avoid overlap in some views or to make some parts more visible.
zero explosion factor	Allows individually specified movements of parts in an exploded view.

Basic Concepts of Complex Assemblies

Assemblies can consist of any number of externally referenced and local parts. You can also have any number of subassemblies, both local and externally referenced. The advantage to having externally referenced parts and subassemblies is that you can use the files in any number of assembly files.

In complex assemblies, the same part is often used in multiple locations. Each part definition defines a unique part. By instancing a part definition, you can create multiple copies of a part while maintaining only one definition in your drawing. Any change to the part definition affects all instances of the part.

After building the assembly, you check for interference between the parts, and perform mass properties calculations to ensure that the parts are designed correctly. If a design change affects a part used in more than one assembly, make the change in the external file.

In a previous tutorial, you made changes to an external part from within an assembly file by editing in place. In this tutorial, you open an external file and modify it directly. Then you update the assembly file to reflect the changes, set up scenes to illustrate the assembly, in both exploded and non-exploded views, create a parts list, and finalize the drawing for plotting.

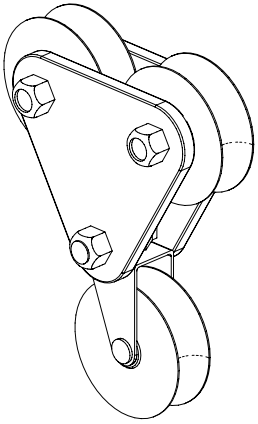
Starting the Assembly Process

There are two ways to approach the design process; start by thinking about how the assembly is organized and decide the order in which to instance the local and external parts into the assembly, or start designing immediately and then reorganize the hierarchy of the assembly as needed, using assembly restructure. Constraints are maintained with parts and subassemblies that are moved in the restructure process.

The assembly restructure feature provides more flexibility in the design process. You can place or create an assembly of components at a single level and subsequently adjust the assembly structure, grouping components into subassemblies for manufacturing, inventory, and other uses.

For more information about using assembly restructure, see “Restructuring Assemblies” on page 504.

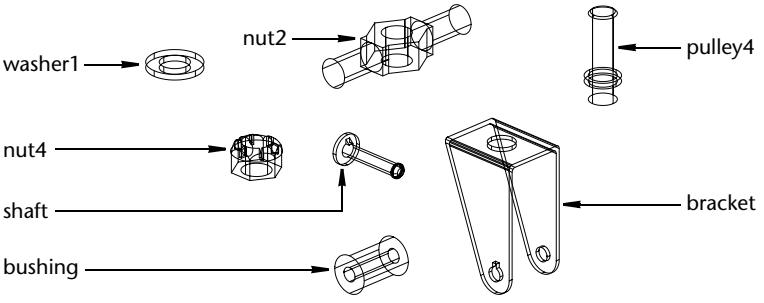
In this lesson, the model is organized and assembled in a particular order. The assembly contains existing parts and referenced external files. A new plate design is also referenced into the current file as part of a subassembly.



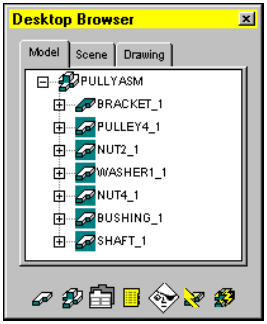
Open the file *pullyasm.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

This drawing contains one local part, a bracket, and six external parts—a pulley, a washer, a shaft, a bushing, and two nuts.



In the Desktop Browser, click the plus sign next to PULLYASM. The assembly tree expands to reveal the part files in the order in which they were added, or referenced, into the file *pullyasm.dwg*.



In the Browser, each part is followed by a number that indicates the order in which it was instanced. In this case, each part has only one instance. As you add more instances, each one will be numbered incrementally. Icons with a teal background are externally referenced parts.

Creating Local and External Parts

An assembly consists of a combination of existing or new parts and local and external parts. You create local parts in the current assembly file.

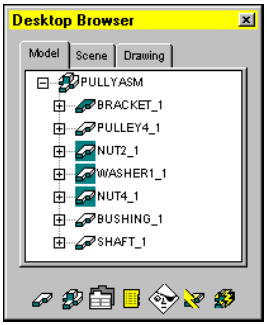
You use the Assembly Catalog to manage existing local parts and to create new external part references and new local parts. You can use the Browser for quick access to the localize and externalize functions without opening the Assembly Catalog.

To localize a part

1 Localize the PULLEY4, BUSHING, and SHAFT parts.

Browser Press CRTL and select PULLEY4, BUSHING, and SHAFT. Right-click PULLEY4 and choose All Instances ➤ Localize.

In the Browser, note that the PULLEY4, BUSHING, and SHAFT icons no longer have a teal background, which indicates that the externally referenced Mechanical Desktop® parts are now local parts.



Links to the external files are severed. Any changes made to these parts affect only the instances in the current assembly.

2 Check the Assembly Catalog to see how the localized parts are displayed.

Context Menu In the graphics area, right-click and choose Catalog.

Choose the All tab. The PULLEY4, BUSHING, SHAFT and BRACKET are listed under Local Assembly Definitions.

Choose OK.

When you have a library of existing parts, bring the files containing those parts into the assembly drawing, and localize them. Parts that may change should be externally referenced so that if the original part is changed, the change will be reflected in the assembly drawing.

To reference an external part

1 Use AMCATALOG to reference an external part.

Context Menu In the graphics area, right-click and choose Catalog.

In the Assembly Catalog, choose the External tab.

2 In Directories, right-click and choose Add Directory.

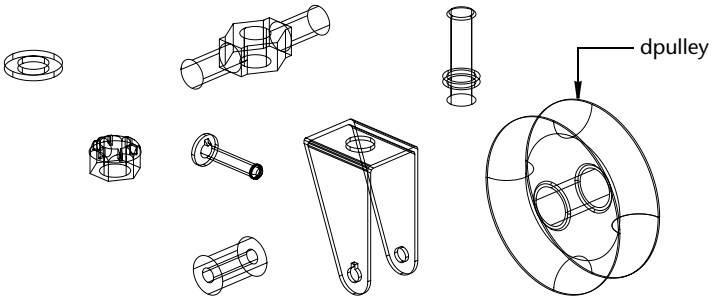
Select the *desktop**tutorial* folder, and choose OK.

All the part and assembly files in the folder are displayed; the icon in front of each file indicates whether it is a part file or an assembly file.

3 Clear the Return to Dialog check box.

4 In Part and Subassembly Definitions, right-click DPULLEY and choose Attach.

5 In the graphics area on your screen, specify an insertion point and press ENTER.



DPULLEY is now externally referenced and instanced into your copy of the *pullyasm.dwg* file.

Applying Assembly Constraints

Now that the parts are referenced or localized into the assembly drawing, you build the assembly by applying assembly constraints. You apply them to one part at a time, removing degrees of freedom. The more constraints a part has, the less freedom of movement it has.

You link the parts, one by one, like a chain, by constraining each part to another part. Use the DOF symbol to illustrate how many degrees of freedom are removed. Usually, you need to solve at least 2 degrees of freedom to sufficiently constrain a part.

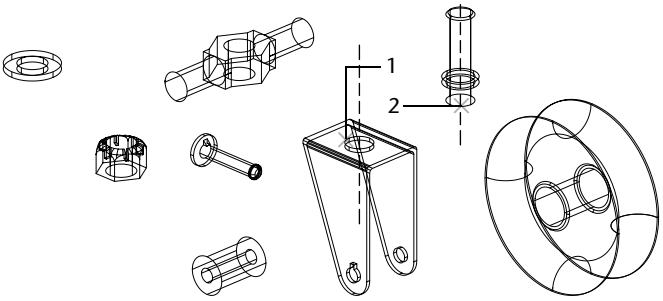
Zoom in as needed to make selection points easier to see. First, you constrain the pulley to the bracket along the common axes. Then, you add another constraint, to mate the planes of the two parts.

To constrain part faces along their axes using mate

1 Use AMMATE to constrain the part faces, responding to the prompts.

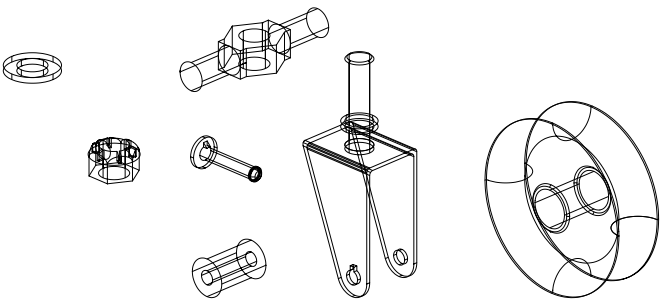
Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

Select first set of geometry: *Select a cylindrical face on* BRACKET *(1)*
First set = Axis, (arc)
Select first set or [Clear/fAce/Point/cYcle] <accEpt>: *Press* ENTER
Select second set of geometry: *Select a cylindrical face on* PULLEY4 *(2)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>: *Press* ENTER
Enter offset <0.0000>: *Press* ENTER



To select the geometry to be constrained, enter responses on the command line, or use the animated mouse. Click the left (red) mouse button to cycle through the options. Click the right (green) mouse button to select an option.

The PULLEY4 part is partially constrained to the BRACKET part along the common axes.

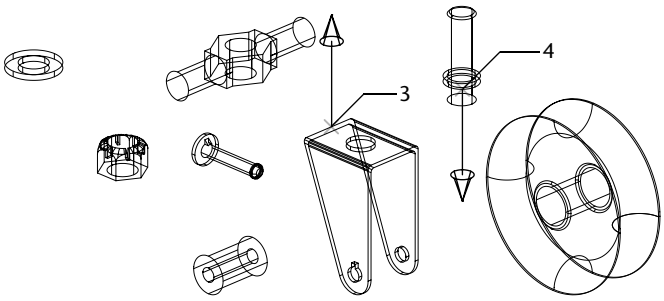


You can move parts to make selection easier. The parts are automatically reassembled when you add a constraint. Or, to manually reassemble parts, set AMAUTOASSEMBLE to **0** (off). Use the Assembly Update icon in the Browser to reassemble the parts after you finish adding constraints.

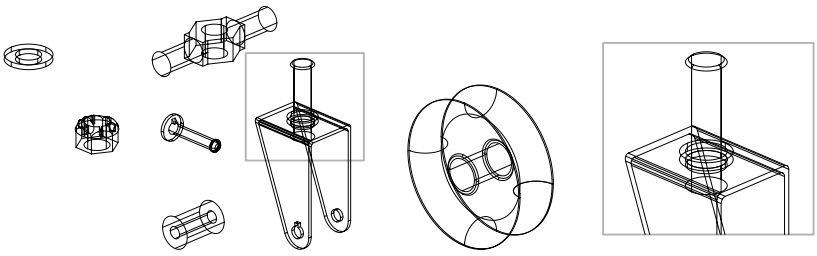
2 Mate the parts on facing planes, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.

Select first set of geometry: *Specify the top plane on* BRACKET *(3)*
Select first set or [Clear/aXis/fAce/Next/cYcle] <accEpt>: *Enter* **a**
First set = Plane
Enter an option [Clear/Next/Flip/cYcle] <accEpt>:
Cycle to point arrow away from the part, then press ENTER
Select second set of geometry: *Specify the flange on* PULLEY4 *(4)*
Second set = Axis, (arc)
Select second set or [Clear/fAce/Point/cYcle] <accEpt>: *Enter* **a**
Second set = Plane
Enter an option [Clear/aXis/Point/Next/Flip/cYcle] <accEpt>: *Press* ENTER
Enter offset <0.0000>: *Press* ENTER

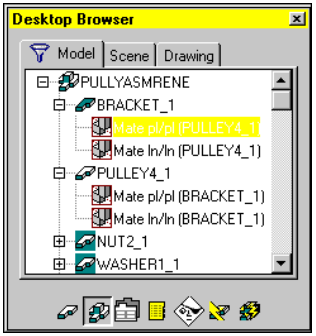


You have constrained BRACKET and PULLEY4 along the common axes and mating planes.



The constraints are illustrated in the Desktop Browser as you create them, showing the other part the constraint is applied to.

To make it easier to see the constraints you have applied, select the Assembly filter in the Browser. Then, the features are hidden, and only the assembly constraints are displayed.

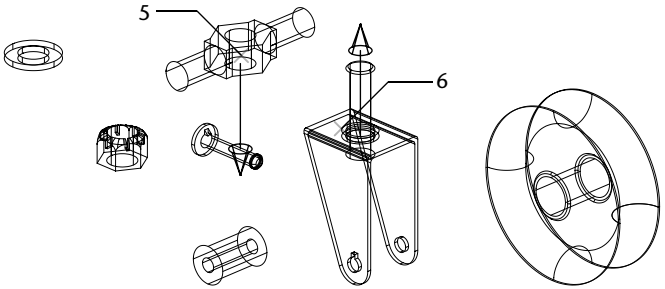


To constrain part faces along their axes using Insert

1 Use AMINSERT to constrain the part faces along the common axes and corresponding planes, responding to the prompts.

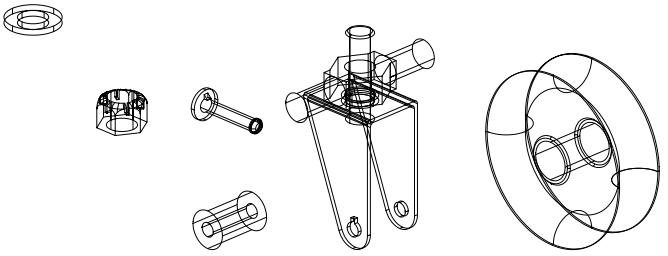
Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

Select first circular edge: *Specify the hole on* NUT2 *(5)*
First set = Plane/Axis
Enter an option [Clear/Flip] <accEpt>: *Press* ENTER
Select second circular edge: *Specify the hole on the top flange of* PULLEY4 *(6)*
Second set = Plane/Axis
Enter an option [Clear/Flip] <accEpt>: *Press* ENTER
Enter offset <0.0000>: *Press* ENTER



Using the insert constraint removes the same degrees of freedom as constraining planes and axes separately. This is particularly useful for bolt-in-hole type constraints.

The NUT2 part is constrained to BRACKET and PULLEY4 along the common axes and mating planes.

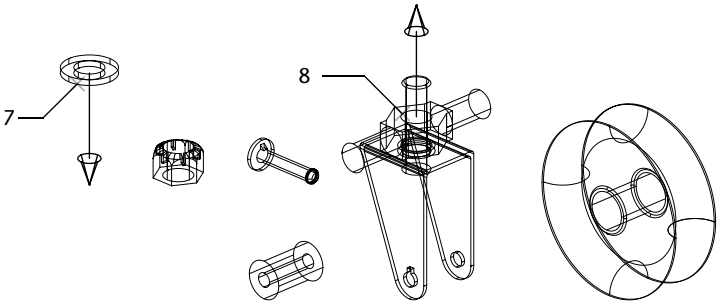


2 Constrain WASHER1 to NUT2, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

Zoom in as needed to see the arrows that indicate the direction of insertion.

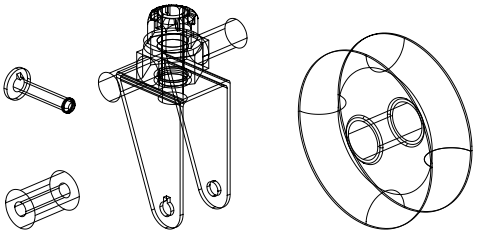
Select first circular edge: *Specify a point on* WASHER1 *(7)*
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Press* ENTER
Select second circular edge: *Specify the hole on* NUT2 *(8)*
Second set = Plane/Axis
Enter an option [Clear/Flip] <accept>:
Flip the direction arrow as needed, and press ENTER
Enter offset <0.0000>: *Press* ENTER



WASHER1 is constrained to NUT2. Next, constrain NUT4 to WASHER1 along the common axes and mating planes.

3 Constrain NUT4 to WASHER1.

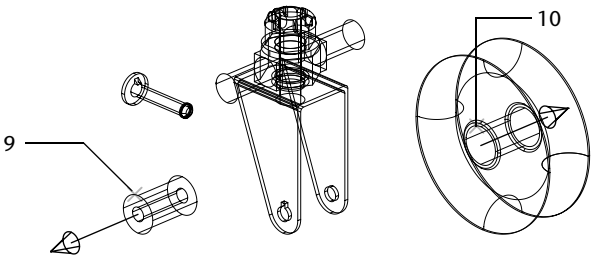
Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.



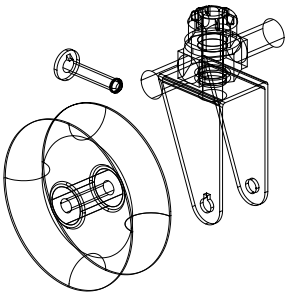
4 Constrain BUSHING and DPULLEY along the common axes, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

Select first circular edge: *Specify a point on* BUSHING *(9)*
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Press* ENTER
Select second circular edge: *Specify the hole on* DPULLEY *(10)*
Second set = Plane/Axis
Enter an option [Clear/Flip] <accept>:
Enter **f** *to flip the direction arrow away from* BUSHING
Second set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Press* ENTER
Enter offset <0.0000>: *Press* ENTER



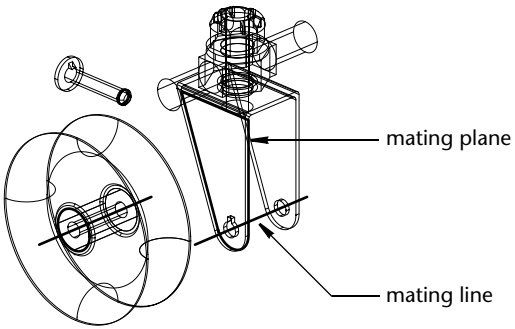
DPULLEY is constrained to BUSHING.



To apply final mate constraints

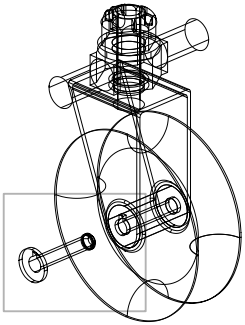
1 Use AMMATE to constrain DPULLEY and BUSHING to BRACKET with two constraints: one to mate planes and one to mate along their axes. If you need to, refer to "To constrain part faces along their axes using mate" on page 484.

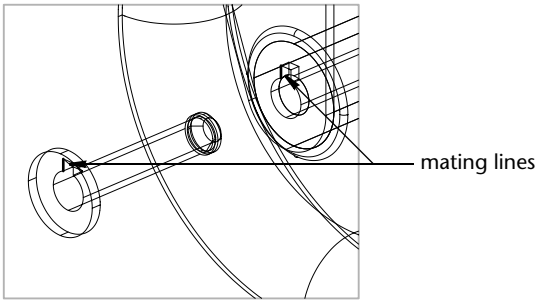
Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.



2 Use AMMATE to constrain SHAFT to BRACKET along the left vertical line of the notch.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Mate.



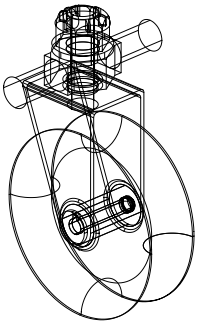


The rotational degree of freedom is removed.

Save your file.

The parts are assembled, and all required degrees of freedom are solved by the constraints.

3 Change to a top view and then to a right view, to verify that the parts are positioned correctly.



Use the Browser to delete unwanted assembly constraints. Right-click the Constraint icon, and choose Delete. You can then apply new constraints.

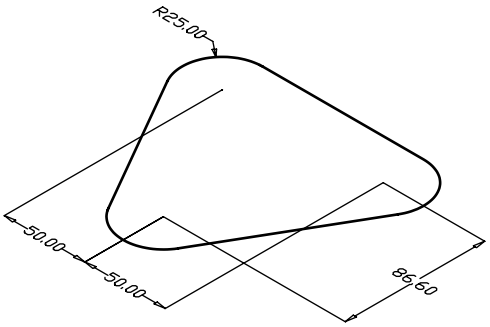
Next, create a new part to use as part of a subassembly.

Creating New Parts

Before you build the subassembly, you create a pulley plate part. First, you open a part file containing a constrained profile. Because you can have more than one drawing open at a time in Mechanical Desktop, you do not need to close your assembly file. In the part file, you add thickness to the profile and create additional features so that you can use it as a subassembly in your assembly.

Open the file *ppulley.dwg*.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.



The sketch of the pulley plate has already been profiled and constrained, so you only need to extrude it. Extrusion adds thickness to a constrained profile.

To extrude a sketch

1 Expand the feature hierarchy in the Browser by clicking the plus sign in front of PPULLEY_1.

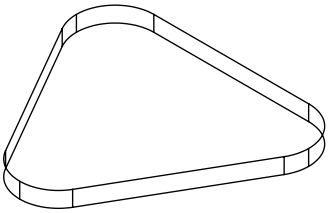
2 Use AMEXTRUDE to extrude the profile.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

3 In the Extrusion dialog box, specify:

Distance: *Enter* **10**

Choose OK to create the extrusion.



To add a placed feature to a part

1 Use AMHOLE to add three drilled bolt holes to the pulley plate.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Hole.

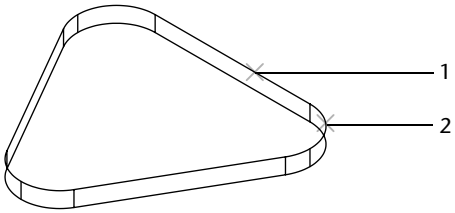
In the Hole dialog box, select the Drilled hole type icon, and specify:

Termination: Through
Placement: Concentric
Diameter: *Enter* **10**

Choose OK.

2 Continue on the command line.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the pulley plate edge (1)
Enter an option (Next/accEpt) <accEpt>: *Press* ENTER
Select concentric edge: *Select the arc (2)*



3 Continue placing two more holes concentric to the remaining arcs. Then press ENTER.

The drilled holes are added and cut through the pulley plate. Later, you check for interference and modify the diameter of the holes.

4 Use AMFILLET to fillet the pulley plate edges.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Fillet.

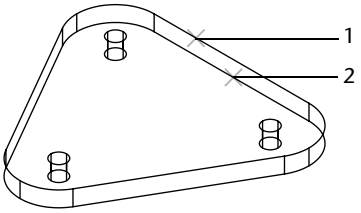
In the Fillet dialog box, select the Constant check box and specify:

Radius: *Enter* **1**
Return to dialog: *Clear the check box*

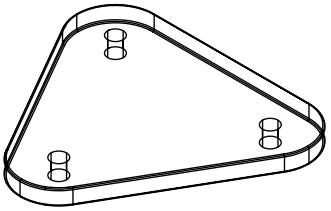
Choose OK.

5 Respond to the prompts as follows:

Select edges or faces to fillet: *Select the pulley plate top edge (1)*
Select edges or faces to fillet: *Select the pulley plate bottom edge (2)*
Select edges or faces to fillet: *Press* ENTER



The pulley plate edges are filleted.



Now that the pulley plate is complete, you have all the parts you need to build the subassembly.

Save your file, but do not close it. You will make some changes to it later.

Creating Subassemblies

You created an assembly. Now, you will create a subassembly. Each subassembly contains one or more parts or subassemblies. In the Browser, subassemblies and parts are nested under the assembly.

You create, instance, and constrain parts into a subassembly just as you do into an assembly. Once the subassembly is created, it is constrained to the assembly, completing the assembly model.

Defining and Activating Subassemblies

Before you can instance any parts into the subassembly, you must create and activate a subassembly definition and then load and instance external part drawings as the parts for the subassembly.

To create and activate a new subassembly

1 Switch to the window containing your assembly.

2 Activate the assembly, PULLYASM.

Browser Right-click PULLYASM and choose Activate.

3 In the Browser, collapse the feature hierarchy.

Browser Right-click PULLYASM and ➤ Collapse. Then click the plus sign in front of PULLYASM.

4 Create a new subassembly definition, responding to the prompt.

Context Menu In the graphics area, right-click and choose Assembly ➤ New Subassembly.

Enter new subassembly name <SUB1>: *Enter* **subpully**

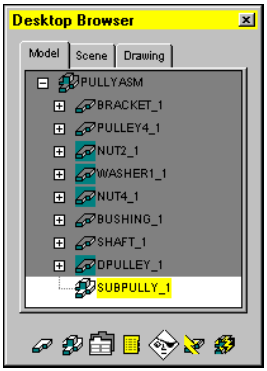
The Desktop Browser adds a subassembly called SUBPULLY_1 to the assembly tree. Because there are no parts instanced or files attached, the subassembly location is empty.

5 Activate the new subassembly, responding to the prompt.

Context Menu In the graphics area, right-click and choose Assembly ➤ Activate Assembly.

Enter assembly name to activate or [?] <PULLYASM>: *Enter* **subpully_1**

The inactive PULLYASM assembly is shaded in the Desktop Browser, and the SUBPULLY subassembly is active.



The active subassembly name is displayed below the command line (Target: SUBPULLY).

Using External Parts

Now that you have activated the new subassembly, you use external parts to create the subassembly.

To attach an external part drawing as an external part

1 Use AMCATALOG to attach an external part.

Context Menu In the graphics area, right-click and choose Catalog.

In the Assembly Catalog, choose the External tab. Verify that *tutorial* is listed in Directories and that Return to Dialog is selected.

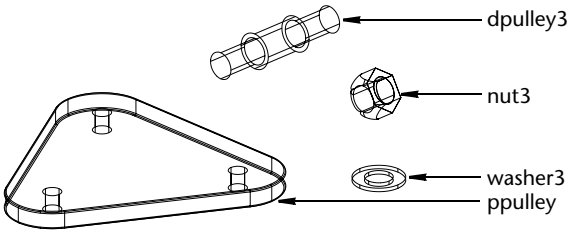
2 Double-click the file *ppulley.dwg*.

3 Position the pulley plate on the screen, and press ENTER.

The pulley plate is attached to the SUBPULLY subassembly.

4 In the Assembly Catalog, right-click DPULLEY3 and choose Attach.

5 Attach WASHER3 and NUT3.



Notice that an attached part is indicated by a white background in the Assembly Catalog.

Choose OK.

Examine the Browser. The referenced parts are nested under the new sub-assembly hierarchy.

Instancing Parts

You have already attached parts to the subassembly. Each part definition is listed in the Assembly Catalog. When you instance a part in the current assembly, it refers to its part definition in the Catalog. Once a part is instanced into the subassembly, you can copy it from the Browser.

To instance a part

1 Use AMCATALOG to instance a part.

Context Menu In the graphics area, right-click and choose Catalog.

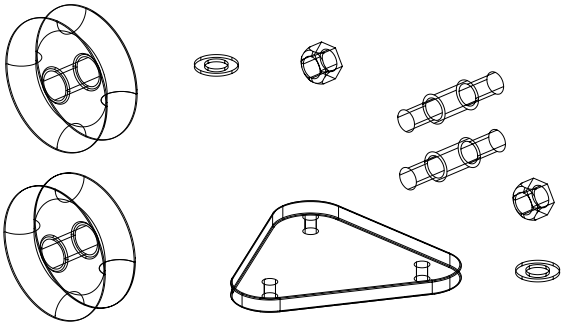
In the Assembly Catalog, choose the All tab. In External Assembly Definitions, select DPULLEY and verify that Return to Dialog is selected.

2 Insert a copy of DPULLEY, and choose OK to exit the Assembly Catalog.

3 Use the Browser to create another instance of DPULLEY.

Browser Right-click DPULLEY and choose Copy. In the graphics area, click a location for the copy and press ENTER.

4 Make one copy each of DPULLEY3, WASHER3, and NUT3.



The Browser is updated to include the new instances.

Save your file.

Next, apply assembly constraints to build the subassembly.

Completing Assemblies

Now that the parts are instanced into the subassembly, you can complete the subassembly and constrain it to the base assembly. When the entire assembly is complete, check it for interference among parts, and obtain mass property information.

NOTE The Desktop Browser shows the order in which parts and subassemblies are assembled. You can drag a part or subassembly to a different position in the Desktop Browser. Always save your file before you reorder the hierarchy. Reordering may affect offsets for explosion factors in scenes.

Applying Assembly Constraints

First, constrain parts in the subassembly, and then constrain the subassembly to the base assembly.

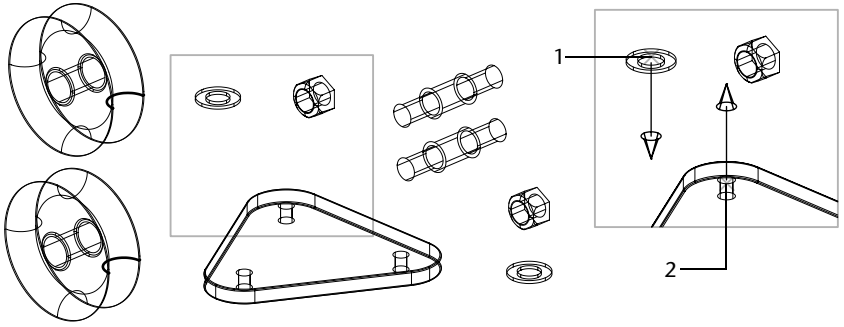
The instructions specify responses on the command line for constraint application, but if you prefer, you can use the animated cursor. After you select the object to constrain, click the left mouse button (red) to cycle through the options. When the object you want is highlighted, click the right mouse button (green) to accept the selection.

To constrain a subassembly part

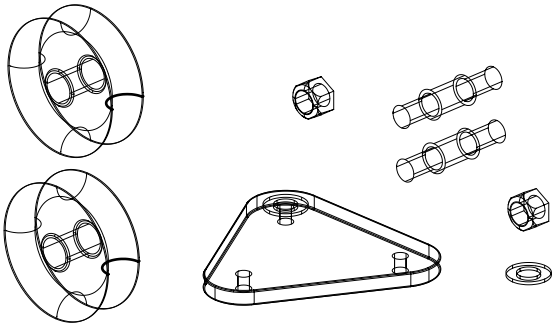
1 Use AMINSERT to constrain WASHER3 to PPULLEY, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

Select first circular edge: *Specify the hole on the bottom of* WASHER3 *(1)*
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Enter* **f**, *if needed*
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Press* ENTER
Select second circular edge:
Specify the top cylindrical edge of the hole in PPULLEY *(2)*
Second set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Press* ENTER
Offset <0.0000>: *Press* ENTER



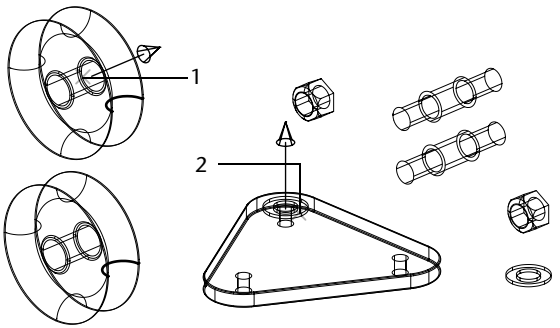
WASHER3 is constrained to PPULLEY along common axes and mating planes.



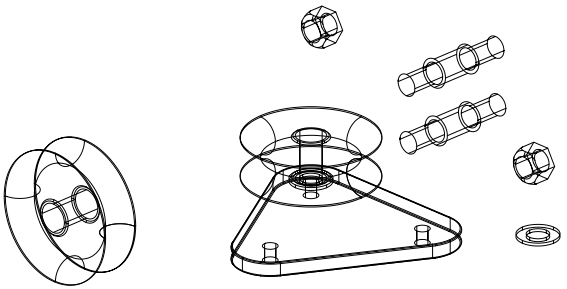
2 Constrain DPULLEY to WASHER3, responding to the prompts.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

Select first circular edge: *Specify inner edge of hole on* DPULLEY *(1)*
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Enter* **f**
First set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Press* ENTER
Select second circular edge: *Specify the top cylindrical edge of* WASHER3 *(2)*
Second set = Plane/Axis
Enter an option [Clear/Flip] <accept>: *Press* ENTER
Offset <0.0000>: *Press* ENTER

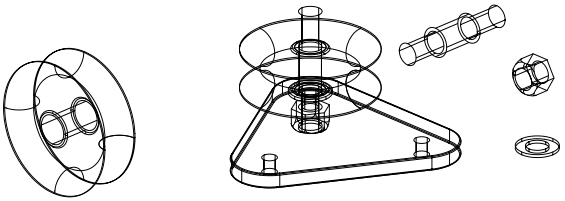


The DPULLEY part is constrained to WASHER3 along the common axes and mating planes.



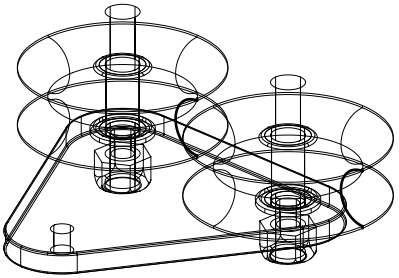
3 Constrain DPULLEY3 and NUT3 to the subassembly.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.



4 Constrain the remaining parts to finish the subassembly.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.



Save your file.

Next, activate the top level of the assembly and constrain the subassembly to the top assembly.

To constrain a subassembly to a top-level assembly

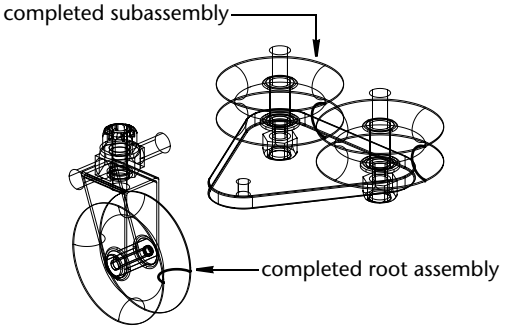
1 Use AMACTIVATE to activate the PULLYASM assembly, responding to the prompt.

Context Menu In the graphics area, right-click and choose Assembly ➤ Activate Assembly.

Enter assembly name to activate or [?] <PULLYASM>: *Press* ENTER

In the Desktop Browser, the top-level assembly is no longer shaded.

The active assembly name is displayed below the command line (Target: PULLYASM).

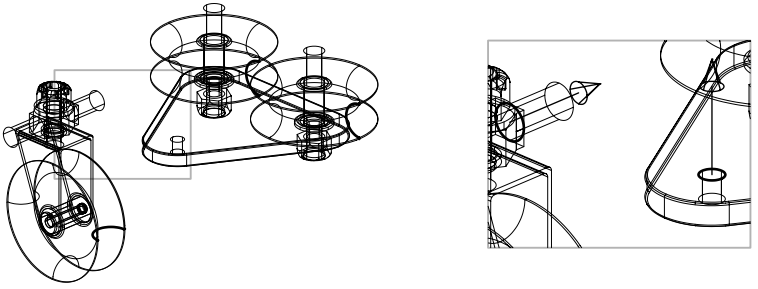


2 Move the subassembly, to make viewing easier. Then zoom in as needed to magnify selection points.

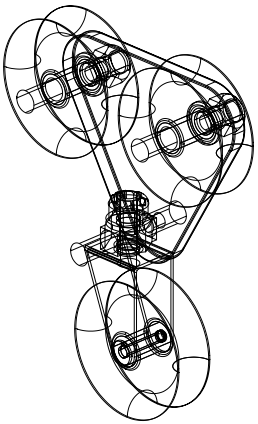
Next, constrain the pulley plate subassembly to the root assembly.

3 Choose Insert to constrain the shaft of NUT2 to the pulley plate hole on the mating planes and along the common axes.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.



The subassembly is now constrained to the root assembly.



Copy another instance of NUT3 in your drawing and constrain it to the shaft of NUT2.

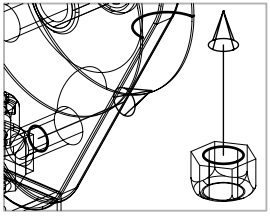
To add and constrain a part instance

1 Use the Browser to copy another instance of the NUT3 external part into the assembly.

Browser Right-click NUT3, and choose Copy. In the graphics area, click a location for the copy, and press ENTER.

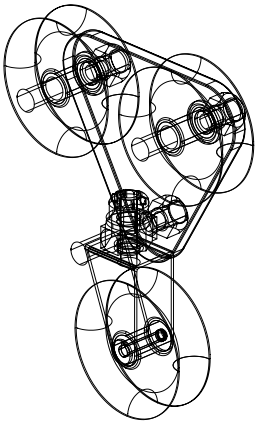
2 Constrain NUT3 to the pulley plate and the shaft of NUT2.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.



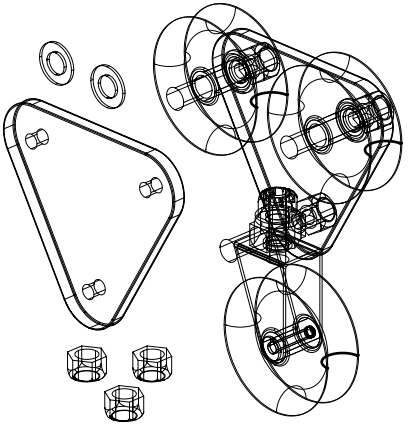
It is a good idea to check the position of a constrained part in another view. If the part is constrained incorrectly, use the Browser to delete the constraint. Right-click to display the menu, and choose Delete. Then move the parts as needed, and reapply the constraints.

One side of the pulley assembly is complete.



3 Repeat steps 1 and 2 to copy and apply assembly constraints to WASHER3, PPULLEY, and NUT3, to complete the pulley assembly.

4 Add one instance of your copy of the PPULLEY external part, two copies of WASHER3, and three copies of NUT3 into the assembly drawing.



NOTE Depending on the current UCS of your drawing, the parts may not be oriented the same as they are in the above illustration. This does not affect your ability to place the constraints.

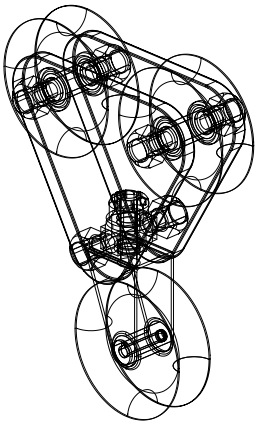
5 Constrain both WASHER3 parts to the DPULLEY3 parts on the pulley plate.

Context Menu In the graphics area, right-click and choose 3D Constraints ➤ Insert.

6 Use Insert to constrain PPULLEY_2 to the WASHER3 parts.

Constrain PPULLEY to any two instances of WASHER3, to remove the rotational degree of freedom from the pulley plate.

7 Use Insert again to constrain the NUT3 parts to the pulley plate.



Save your file. The pulley assembly is complete.

Restructuring Assemblies

The pulley assembly model in the previous exercise was planned, and then assembled in a particular order. In Mechanical Desktop, you can use another design process called assembly restructure. You can design and constrain parts and create assemblies in any order, and subsequently adjust the assembly structure within the Desktop Browser.

Using the assembly restructure feature in the Browser, you can

- Use either the drag or cut and paste method to move components
- Move components from the master assembly into a local or external subassembly
- Select multiple parts and subassemblies and move them within the Browser to restructure the assembly
- Move components while in the context of an active external subassembly

In most cases, constraints are maintained with the parts and subassemblies that you move. In the event that a constraint cannot be maintained, a warning message is displayed.

NOTE Instances can be lost if you restructure them up the assembly hierarchy where multiple instances of the same definition exist.

Open the assembly file *pullyasm.dwg* from the *desktop\tutorial* folder, and practice using assembly restructure.

To restructure an assembly hierarchy

1 Expand the hierarchy in the Browser so you can see all of the components.

Browser Right-click PULLYASM and choose Expand.

2 Select a part name to activate the part.

To select more than one part, hold down CTRL as you select.

To select all parts within a range, hold down SHIFT as you select the first and last part in the range. All components in between are automatically selected.

3 Drag the selected parts up or down in the assembly structure.

Use a left-click to drag your selection to a location and release the mouse.

Use a right-click to drag your selection to a location and release the mouse to display a context menu with options Move Here, Copy Here, Create Subassembly Here, and Cancel.

If the assembly tree is too long to scroll conveniently, right-click the component and choose Cut, scroll to the new location, and Choose paste.

If a part cannot be moved logically, an error icon is displayed.

When restructure is activated, the cursor changes to look like a shadow of the original part name.

4 Drop the part in a valid destination.

Valid destinations are highlighted as you move the cursor over them.

The part is restructured within the assembly, and the assembly is updated.

Analyzing Assemblies

Next, check for interference between parts. This analysis is useful for detecting problems that may arise during the final design stages. Check each part for interference. If any interference is detected between parts, interference solids can be created to illustrate where the interference occurs.

To check for interference

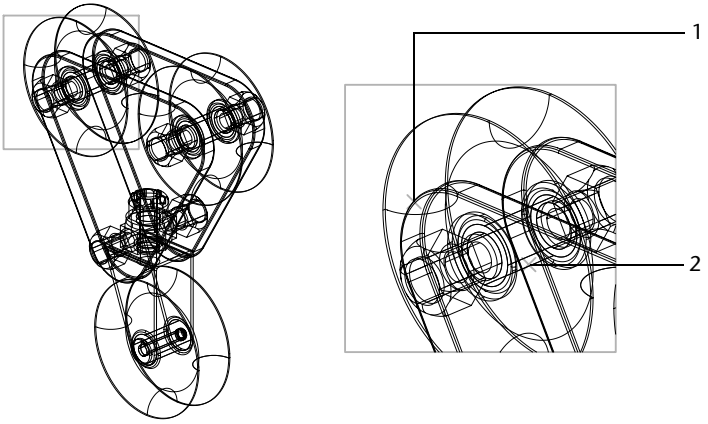
1 Use AMINTERFERE to check for interference.

Context Menu In the graphics area, right-click and choose Analysis ➤ Check Interference.

Respond to the prompts.

Nested part or subassembly selection? [Yes/No] <No>: *Enter* **y**
Specify first set of parts or subassemblies or [?]: *Select* PPULLEY *(1)*
Instance = PPULLEY_2
Enter an option [Down/Next/Accept] <Accept>: *Press* ENTER
Specify first set of parts or subassemblies: *Press* ENTER
Specify second set of parts or subassemblies: *Select* DPULLEY3 *(2)*
Instance = SUBPULLY_1
Enter an option [Down/Next/Accept] <Accept>: *Enter* **d**
Instance = DPULLEY3_1
Enter an option [Up/Down/Next/Accept] <Accept>: *Press* ENTER
Specify second set of parts or subassemblies: *Press* ENTER
Comparing 1 parts/subassemblies against 1 parts/subassemblies.
Interference 1:
PPULLEY_2
SUBPULLY_1
DPULLEY3_1
Create interference solids? [Yes/No] <No>: *Enter* **n**
Highlight pairs of interfering part/subassemblies?[Yes/No] <No>: *Enter* **y**
Enter an option [eXit/Next pair] <Next pair>: *Enter* **x**

NOTE Resize the text window above the command line to read the results of interference checking.

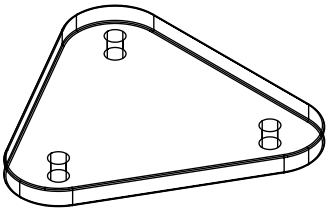


You should detect two examples of interference. Both DPULLEY and WASHER3 parts interfere with the holes on PPULLEY. The drilled holes in the pulley plate are too small.

The pulley plate is an external part, but it can be edited from within the assembly file. If you are working in a very large assembly, it may be easier to open the external file and make changes.

Editing Mechanical Desktop Parts

Because interference was detected between the pulley plate and the DPULLEY3 parts, you need to enlarge the drilled holes on the pulley plate. Editing an external part updates all instances of the parts in the assembly drawing. The modified part retains the applied assembly constraints.



In this tutorial, you return to the open *ppulley.dwg* file and make the holes larger, to remove the interference.

To edit an external part

1. Switch to the window containing the PPULLEY part you created earlier.
2. In the Browser, click the plus sign in front of PPULLEY to display the part features.
3. Use AMEDITFEAT to edit the holes, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Independent array instance/Sketch/surfCut/Toolbody/select Feature] <select Feature>: *Select one of the drilled holes*
Enter an option [Accept/Next] <Accept>: *Enter* **n** *or press* ENTER

The Hole dialog box is displayed.

4. In the Hole dialog box, enter **15** in the Drill Size field. Then choose OK.
5. Repeat steps 3 and 4 for Hole2 and Hole3.
6. Use AMUPDATE to update the edited part.

Context Menu In the graphics area, right-click and choose Update Part.

The pulley plate reflects the new design.

Save your file.

Reloading External References

To update the assembly to reflect the changes you made to the external part, reload the external definition.

To reload an external definition

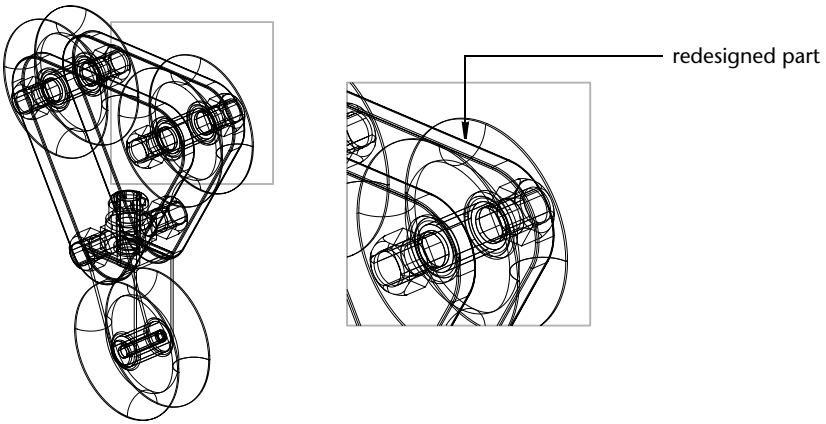
1. Switch to the window containing the assembly.
2. Use AMCATALOG to reload the PPULLEY definition.

Context Menu In the graphics area, right-click and choose Catalog.

In the Assembly Catalog, in External Assembly Definitions, right-click PPULLEY and choose Reload.

Choose OK.

The pulley plate reflects the new design.



If the assembly looks incorrect, choose Assembly ➤ Assembly Update to update the assembly constraints.

Save your file.

Check the assembly again for interference.

To check for interference

1 Use AMACTIVATE to activate the SUBPULLY subassembly.

Context Menu In the graphics area, right-click and choose Assembly ➤ Activate Assembly.

Specify the SUBPULLY subassembly.

2 Use AMINTERFERE to check for interference.

Context Menu In the graphics area, right-click and choose Analysis ➤ Check Interference.

3 Specify DPULLEY3. No interference should be detected.

Assigning Mass Properties

Next, calculate mass property information. You can analyze parts and assemblies during the course of designing. You may need to optimize weight, maximize stiffness, balance loads, ease assembly, or meet particular requirements.

One mass properties dialog box contains both Setup and Results tabs that function the same for both parts and assemblies. In this case, the parts list in the dialog box displays all parts and part properties in the activated assembly to be analyzed. Item numbers are also displayed if a BOM exists. When you select an item in the parts list, it is graphically highlighted on the screen.

In these steps, you analyze the bracket and change tolerance values and material types.

To set up mass properties

1 Activate the PULLYASM assembly.

Context Menu In the graphics area, right-click and choose Assembly ➤ Activate Assembly.

2 Use AMMASSPROP to analyze the mass properties of the assembly, responding to the prompt.

Context Menu In the graphics area, right-click and choose Analysis ➤ Mass Properties.

Select parts or subassemblies: *Select the bracket, and press* ENTER

The Assembly Mass Properties dialog box is displayed with the Setup tab active. In the Materials Available window, the materials listed are all of those defined in the active assembly.

3 In the Assembly Mass Properties dialog box Setup tab, specify:

Output Units: Metric (mm, g)
Coordinate System: User coordinate system (UCS)
Display Precision: *Select* 0.00000
Part Name: *Select* BRACKET
Materials Available: Material: *Select* Stainless_Steel

Choose Assign Material.

The material information is transferred to the part material attribute and BOM, and is updated in the part name list.

Next, change the material definition for a part in the assembly.

4 In the Part Name list, select BRACKET, and then select Edit Materials.

5 In the Physical Materials List dialog box Material List, select Stainless_Steel.

In the Properties Window, specify

Density: **8.5**

Choose OK.

The new material definition information is transferred to the part, BOM, and is updated in the Assembly Mass Properties dialog box Part List view.

Choose Done to exit the Assembly Mass Properties dialog box.

You are ready to calculate the mass properties based on the new information you entered.

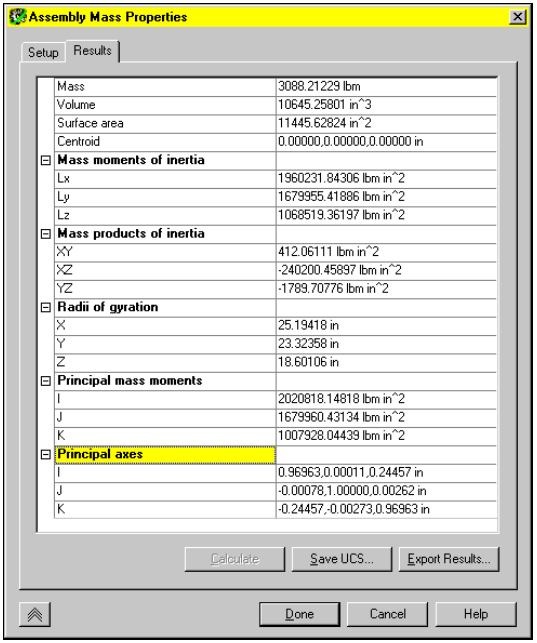
Calculating Mass Properties

In the Mass Properties dialog box the Results tab is blank until you use the Calculate button to retrieve the results of your input in the Setup tab.

To calculate mass properties

1 In the Mass Properties Dialog Box, select the Results tab. Then select the Calculate button.

The results are calculated and displayed.



The Calculate button is no longer available because the Setup and Results fields are in sync. If you change an item on the Setup tab, the results are cleared and the Calculate button becomes available.

You can use the Insert UCS button to create and insert a user coordinate system (UCS) based on a parts or assemblies center of gravity (CG).

2 Choose Export Results.

In the File dialog box, define a file name and save the file.

This report file can be imported by many external programs.

3 Choose Done to close the Assembly Mass Properties dialog box.

Save your file.

Now, you create an exploded view of the assembly.

Reviewing Assembly Models

Assembly scenes and drawing views are essential for reviewing the assembly model. For this lesson, you first create an exploded assembly scene, and then tweak the positions of parts and add assembly trails and annotations.

Creating Exploded Assembly Scenes

After assembly constraints have been applied to each part, you can create a scene (an exploded view of the entire assembly). Multiple scenes can be created and named. You set an explosion factor to determine the separation of parts in the scene. If you do not want an exploded view in a scene, set the explosion factor to **0**.

Before you begin, select the Scene tab to switch to Scene mode.

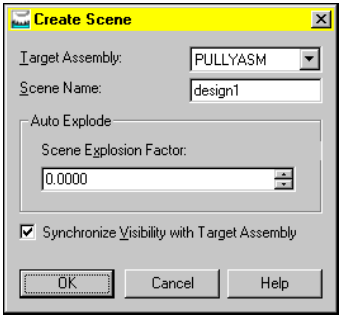
To create an exploded assembly scene

1 Use AMNEW to create a new scene, responding to the prompts.

Context Menu In the graphics area, right-click and choose New Scene.

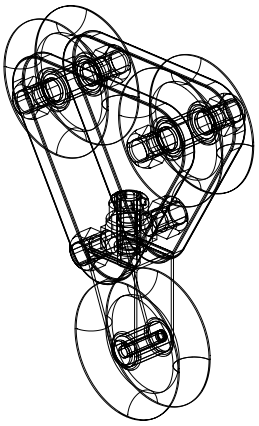
In the Create Scene dialog box, specify:

Target Assembly: PULLYASM
Scene Name: *Enter* **design1**
Scene explosion factor <.0000>: *Specify* 0.0000
Synchronize Visibility with Target Assembly: *Select the check box*

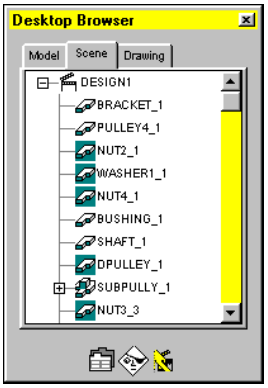


Choose OK.

The scene, design1, is displayed.



2 Look at the Browser. The names of all parts in the *design1* scene are listed.



Next, align the exploded parts in the assembly scene.

Using Tweaks and Trails in Scenes

In an exploded scene, sometimes parts obscure other parts. You can use tweaks to change the positions of individual parts and then adjust the positions of the parts in the *design1* scene. Zoom in to magnify the parts to be tweaked.

In the Browser, tweaks are nested under the respective parts. You can select and multi-select tweaks in the Browser to delete them. When you pause the cursor over a tweak in the Browser, a tooltip displays the distance factor for the tweak.

NOTE The grounded part of an assembly or a subassembly cannot be tweaked. Its position is fixed.

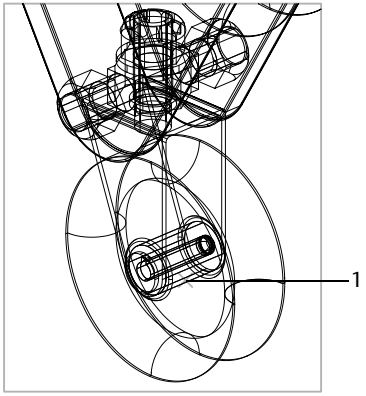
Assembly trails use a continuous or other defined linetype to indicate the path of the explosion. These trails use assembly constraint information to visually demonstrate how the assembly design fits together.

To tweak an exploded assembly scene

1 Use AMTWEAK to open the Power Manipulator dialog box.

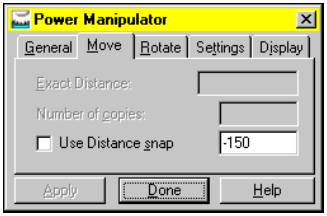
Context Menu In the graphics area, right-click and choose New Tweak.

Select part/subassembly to tweak: *Specify a point on* BUSHING_1 *(1)*
Enter an option [Next/Accept] <Accept>: *Press* ENTER



The Power Manipulator dialog box is displayed only the first time you create a new tweak.

In the Power Manipulator dialog box, on the Move tab, verify that Place Objects (ALT) is selected.



Choose Done.

NOTE To access the Power Manipulator dialog box later, right-click the Power Manipulator symbol on your screen, and select Options.

2 Use AMTWEAK to tweak the BUSHING part, responding to the prompts.

Context Menu In the graphics area, right-click and choose New Tweak.

Select part/subassembly to tweak: *Specify a point on* DPULLEY_1 *(1)*
Enter an option [Next/Accept] <Accept>: *Press* ENTER

This time, the Power Manipulator symbol is displayed on the BUSHING part you selected.

3 Click the *-Z* axis on the Power Manipulator symbol, drag away from the assembly, and click at a distance of -150.

4 Continue on the command line.

Select handle or Geometry
[Undo/UCS/WCS/Select/Options/Pancenter/Type/tRails/X/Y/Z] <Accept>:
Press ENTER

The BUSHING is moved away from the assembly. In the Browser, a Tweak icon is displayed.

1 Use AMTWEAK to tweak the DPULLEY part, responding to the prompts.

Context Menu In the graphics area, right-click and choose New Tweak.

Select part/subassembly to tweak: *Specify a point on* DPULLEY_1 *(1)*
Enter an option [Next/Accept] <Accept>: *Press* ENTER

2 Click the *-Z* axis on the Power Manipulator symbol, drag away from the assembly, and click at a distance of -70.

3 Continue on the command line.

Select handle or Geometry
[Undo/UCS/WCS/Select/Options/Pancenter/Type/tRails/X/Y/Z] <Accept>:
Press ENTER.

4 Use AMTWEAK to tweak the SHAFT part, responding to the prompts.

Context Menu In the graphics area, right-click and choose New Tweak.

Select part/subassembly to tweak: *Specify a point on* SHAFT_1 *(1)*
Enter an option [Next/Accept] <Accept>: *Press* ENTER

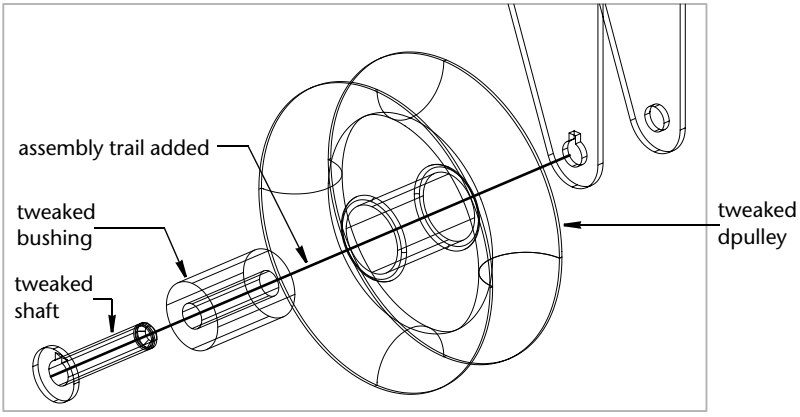
The Power Manipulator symbol is displayed on the SHAFT part.

5 Click the *-Z* axis on the Power Manipulator symbol, drag away from the assembly, and click at a distance of -200.

6 Continue on the command line.

Select handle or Geometry
[Undo/UCS/WCS/Select/Options/Pancenter/Type/tRails/X/Y/Z] <Accept>: *Enter* **r**
Create Trails? [Yes/No] <No>: *Enter* **y**
[Undo/UCS/WCS/Select/Options/Pancenter/Type/tRails/X/Y/Z] <Accept>:
Press ENTER

The BUSHING, DPULLEY, and SHAFT part tweaks are displayed with trails.



You can adjust assembly trails.

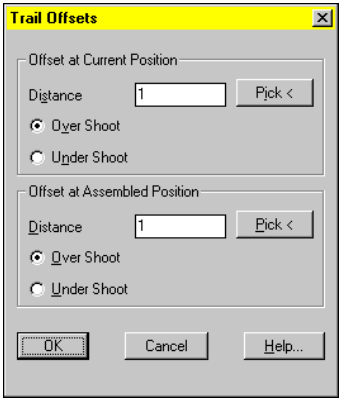
To adjust assembly trails

1 Use AMTRAIL to adjust your assembly trails, responding to the prompt.

Command AMTRAIL

The Trail Offsets dialog box is displayed.

2 Use the options in the Trail Offsets dialog box, to adjust over shoots and undershoots for your trails.



Choose OK to apply your selections.

Next, create an assembly drawing view.

Creating Assembly Drawing Views

An assembly drawing view shows a 2D representation of the 3D assembly. You use the base part for a base view. Then you create an isometric view of the entire assembly model. Drawing views are automatically updated when you change a part or subassembly.

Before you begin, select the Drawing tab to switch to Drawing mode.

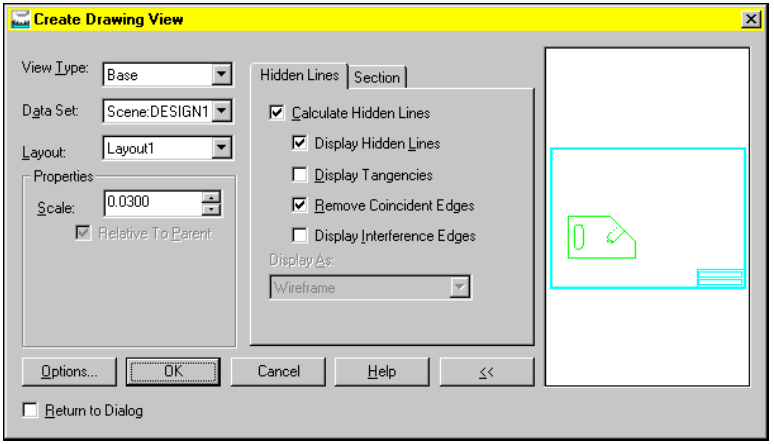
To create a drawing view

1 Use AMDWGVIEW to create a new drawing view.

Context Menu In the graphics area, right-click and choose New View.

2 In the Create Drawing View dialog box, specify:

Type: Base
Data Set: Scene: DESIGN1
Scale: *Enter* **.03** *(or* **.75** *mm)*

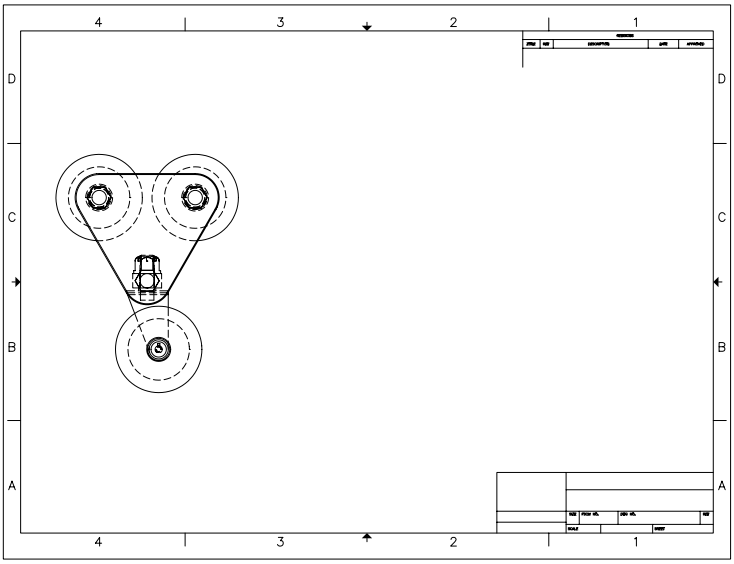


Choose OK.

3 Respond to the prompts as follows:

Select planar face, work plane or [Ucs/View/worldXy/worldYz/worldZx]:
Enter **z**
Select work axis, straight edge or [worldX/worldY/worldZ]:
Enter **x**
Adjust orientation [Flip/Rotate] <accept>
Enter **r** *until the UCS icon is upright*
Adjust orientation [Flip/Rotate] <accept>: *Press* ENTER
Specify location of base view: *Specify a point in the left of the title block*
Specify location of base view: *Press* ENTER

The base assembly drawing view is displayed.



4 Create an isometric view of the assembly.

Context Menu In the graphics area, right-click and choose New View.

In the Create Drawing View dialog box, specify:

View Type: Iso
Scale: *Enter* **1**
Relative to Parent: *Select the check box*

Choose OK.

5 Respond to the prompts as follows:

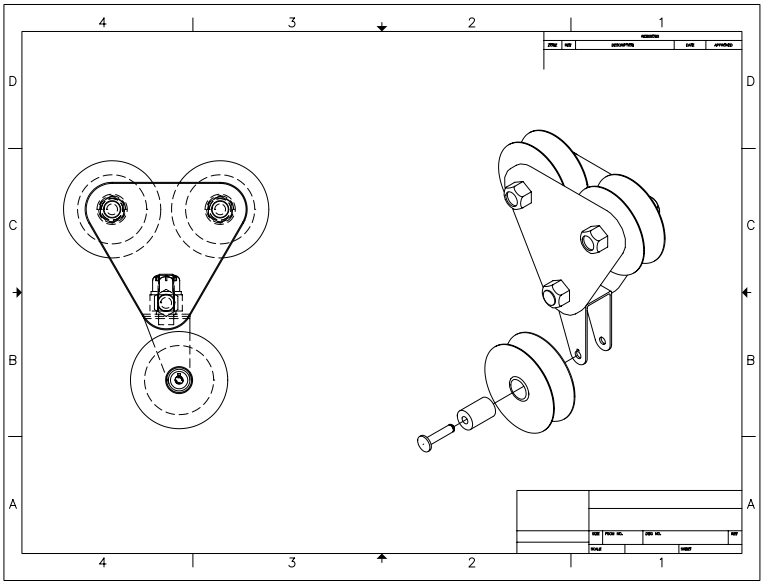
Select parent view: *Select the base view*
Specify location for isometric view: *Specify a point to the right of the base view*
Specify location for isometric view: *Press* ENTER

An isometric view of the assembly is displayed.

6 Use AMMOVEVIEW to align the views, responding to the prompts.

Context Menu In the graphics area, right-click and choose Move View.

Select view to move: *Select the isometric view*
Specify new view location or [Layout]: *Align the views and press* ENTER



Examine the Browser. The isometric view is listed under the base view.

Now, create a BOM database, and add a parts list and associative balloon callouts.

Creating Bills of Material

After you have assembled your parts, you can create a bill of material (BOM) database. This database contains a list of attributes assigned to each part. The attributes store information such as manufacturer, description, and vendor part number.

The attributes are contained in part references that are assigned to each part. Part references can also be created to reference other geometry in your drawing, such as surfaces. The geometry can then be included in a parts list.

The BOM database counts the number of instances in an assembly and tallies instances for each part. In the BOM table, defined columns can be edited, added, deleted, moved, and sorted.

By accessing the information in the BOM database, you can add balloons and insert parts lists into your drawing. You can edit part references, balloons, and parts lists. The BOM database is automatically updated.

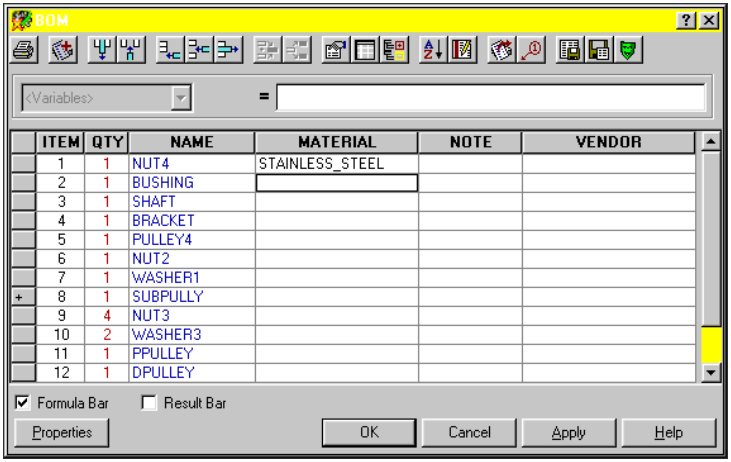
Because the BOM database is fully parametric, any changes to it update the information stored in balloons, part references, and parts lists.

To create a BOM database

1 Use AMBOM to create the BOM database.

Command AMBOM

In the BOM dialog box, review the parts in the database.



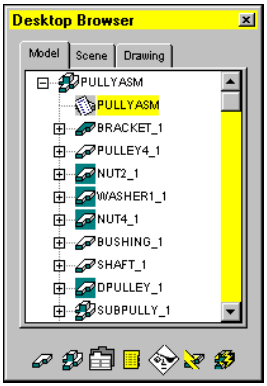
Notice the SUBPULLY definition in the list of parts. The plus sign in front of it indicates that it is a subassembly.

2 Click the plus sign in front of SUBPULLY.

The BOM database now lists all the parts in the assembly, including those in the subassembly.

Choose OK to exit the BOM dialog box.

3 Select the Model tab in the browser. A BOM icon is located under PULLYASM.



By default, the BOM takes the same name as the assembly file. You can change this setting by editing the name of the BOM directly in the Browser.

To edit the name of a BOM

1 Using the Browser, rename the BOM database.

Browser Right-click the BOM icon and choose Rename.

2 Change the name to **BOM_1**.

Customizing BOM Databases

Mechanical Desktop provides symbol standards for major drafting conventions, including ANSI, BSI, CSN, DIN, GB, ISO, and JIS. By modifying the symbol standards, you can control the way symbols, balloons, and parts lists are displayed in the drawing, when you create them.

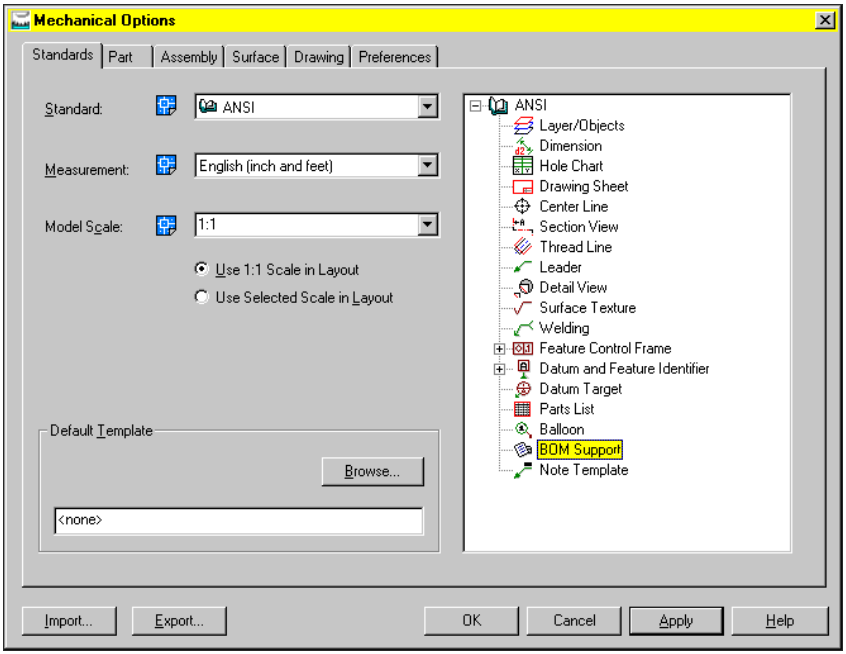
Next, modify the symbol standards to control the number of columns used in the parts list and the name of one of the columns.

To modify symbol standards

1 Use AMOPTIONS to access the Mechanical Desktop symbol standards.

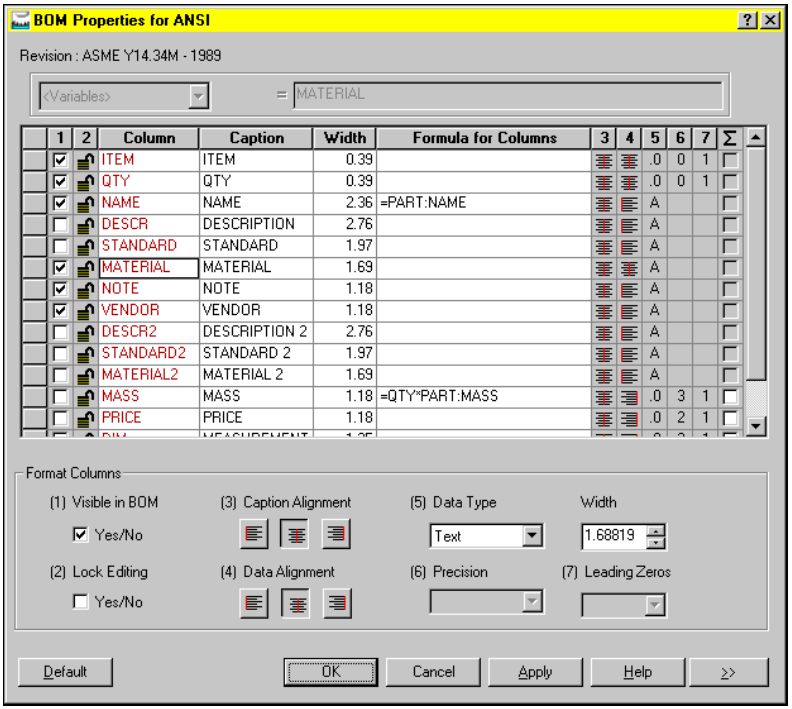
Command AMOPTIONS

In the Mechanical Options dialog box, expand the hierarchy of ANSI, and double-click the icon in front of BOM Support.



2 In the BOM Properties for ANSI dialog box, in Columns select Material and specify:

Caption Alignment: *Select the* Center Align Text *icon*
Data Alignment: *Select the* Center Align Text *icon*



Choose Apply, then OK.

The Mechanical Options dialog box is still open.

3 Choose OK to close the Symbol Standards dialog box.

Save your file.

Working with Part References

When you create a BOM database, each part in the assembly is assigned a part reference. A part reference is an attributed block that can be modified to include any information you want to attach to the part. That information is used by the BOM database and included in the parts lists you generate.

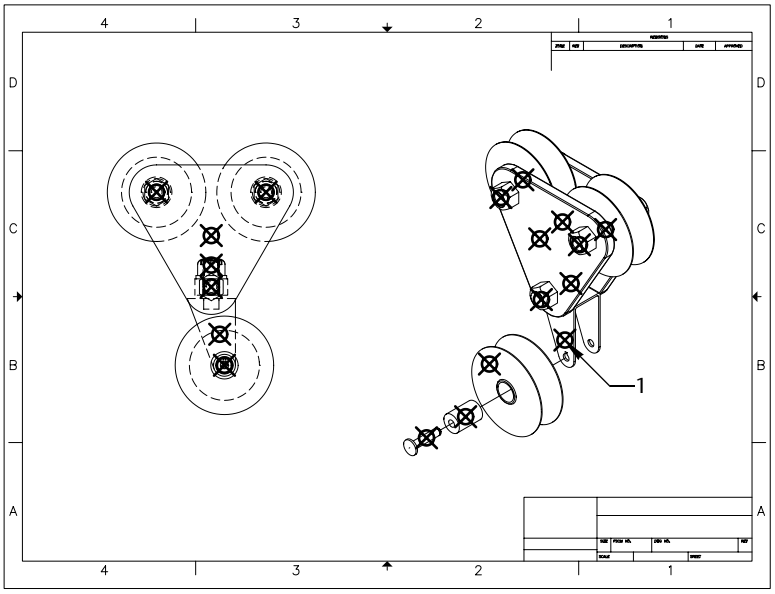
Before you begin, in the Browser, select the Drawing tab to switch to Drawing mode.

To edit a part reference

1 Use AMPARTREFEDIT to edit a part reference, responding to the prompt.

Context Menu In the graphics area, right-click and choose Annotate Menu ➤ Parts List ➤ Part Reference Edit.

Select pick object: *Select the part reference for* BRACKET *(1)*



2 In the Part Ref Attributes dialog box, double-click in the Name field and enter **Pulley Bracket.**

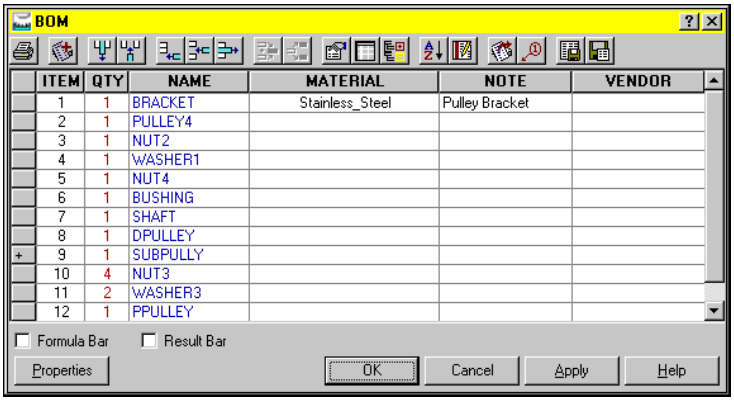
Choose OK.

3 Use AMBOM to display the BOM database.

Context Menu In the graphics area, right-click and choose Parts List ➤ BOM Database.

Bom table [Delete/Edit] <Edit>: *Press* ENTER

Notice that Pulley Bracket is now listed under Note for BRACKET.



Choose OK to exit the BOM dialog box.

Next, add balloon callouts to the isometric view.

Adding Balloons

Balloons are used to reference parts in your drawing to a parts list. They contain the same information as the part reference they are attached to. You can edit that data by selecting balloons. Changes made to the data associated with a balloon are reflected in the BOM database and the parts list.

You can control the size and appearance of balloons, using the Symbol Standards dialog box.

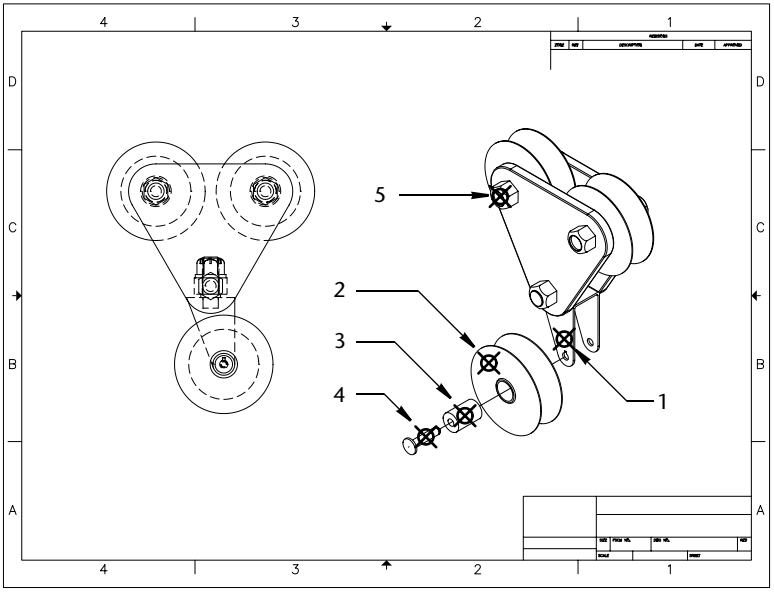
To place a balloon callout

1 Use AMBALLOON to create balloon callouts for BRACKET, DPULLEY, BUSHING, SHAFT, and NUT3, responding to the prompts.

Desktop Menu Choose Annotate ➤ Parts List ➤ Balloon.

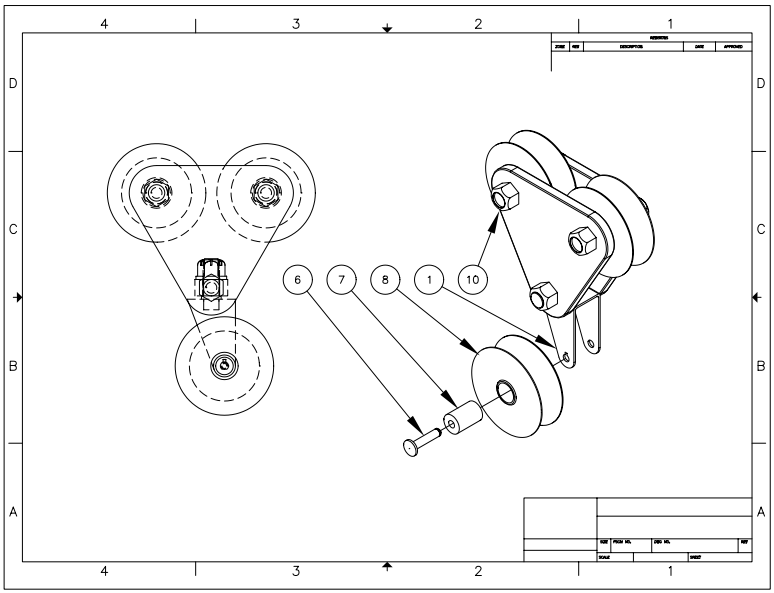
Select part/assembly or [auTo/autoAll/Collect/Manual/One/ Renumber/ rEorganize]: *Enter* **T**
Align [Angle/Standalone/Horizontal/Vertical] <Vertical>: *Press* ENTER
Select pick object: *Select the part reference for* BRACKET *(1)*
Select pick object: *Select the part reference for* DPULLEY *(2)*
Select pick object: *Select the part reference for* BUSHING *(3)*
Select pick object: *Select the part reference for* SHAFT *(4)*
Select pick object: *Select the part reference for* NUT3 *(5)*

The objects are aligned after you pick them.



Continue on the command line:

Select pick object: *Press* ENTER
Align Standalone/Horizontal/<Vertical>:
Specify a point between the two drawing views



Next, place the parts list in the drawing.

Placing Parts Lists

A parts list is an associative block of information about your assembly. It displays information about the parts, according to the settings specified in the BOM database. Because it is parametric, any changes you make to the BOM database or symbol standards are automatically reflected in the parts list.

To place a parts list

1 Use AMPARTLIST to place the parts list under the base view of your assembly, responding to the prompts.

Context Menu In the graphics area, right-click and choose Parts List ➤ Part List.

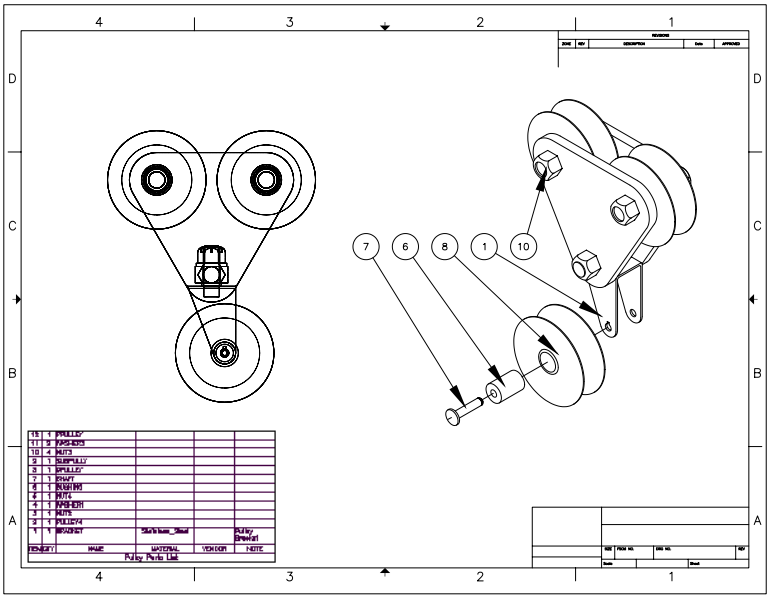
The Parts List dialog box is displayed.

2 In the Parts List dialog box, in Title, enter the new name Pulley Parts List.

3 Choose Apply, then choose OK.

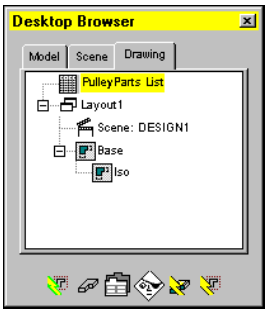
Specify location: *Specify a point under the base view*

The parts list is placed in the drawing.



NOTE You may need to move the drawing views to make room for the parts list. Place the parts list first, and then move the views as needed. Because the balloons are associated with the views, they will also move.

When the parts list is created, a parts list icon is displayed in the Browser.



Save your file.

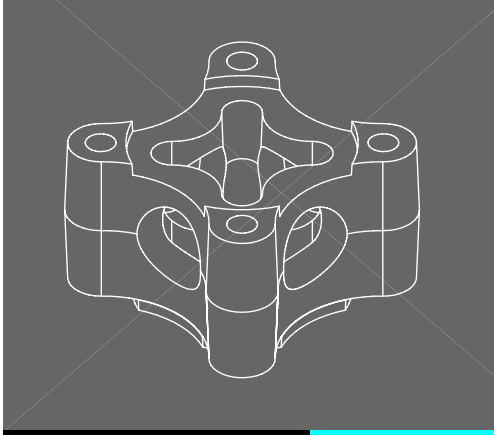
Finishing Drawings for Plotting

Now that the assembly has been documented, plot the drawing.

Paper drawings are useful for reviewing the entire assembly to make sure that the design is feasible and can be manufactured. The Parts List on the assembly drawing provides information about the parts needed to manufacture the assembly. If you wish, add more reference dimensions, fill in the title block, and add some notes to your drawing before you plot.

This tutorial has demonstrated the flexibility of using subassemblies to create complex models. Even a complex assembly can be easily modified and documented, if it is truly a parametric assembly.

Creating and Editing Surfaces



In This Chapter

19

- Creating surfaces—motion based, skin, and derived
- Joining and trimming surfaces
- Editing surfaces

This Autodesk® Mechanical Desktop®tutorial introduces surfaces grouped by function, and provides instructions for creating the different types of surfaces. You learn surface types, practice surface modeling, and work with modeling some typical surfaces.

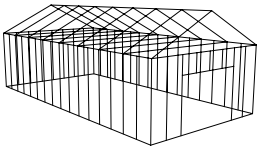
Once you have completed this lesson, you will understand the techniques required to surface a more complex part.

Key Terms

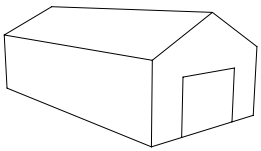
Term	Definition
augmented line	A 3D polyline with vector information at each vertex. An augmented line is a surface creation tool that allows you to control the curvature and the tangency of a surface.
base surface	A basic underlying surface that carries a shape across a larger area. Can be trimmed to precise shapes as needed, but the base surface remains intact and may be displayed.
derived surface	A surface that gets some or all of its attributes from one or more base surfaces.
motion-based surface	A surface created by moving wires through space.
rail	One or more curved lines along which a surface is swept. Rails form the curvature of a swept surface.
skin surface	The surface draped over a wireframe.
surface normal	A short line perpendicular to a surface that shows where the surface starts and which direction is out.
surface primitive	Surface created by values you specify. It does not require a wireframe model.
U or V display lines	Lines that correspond to rails and wires.
wire	Generic term for lines, arcs, circles, ellipses, 2D and 3D polylines, augmented lines, and splines.
wireframe modeling	Wires and surface parts intermixed to construct the basic framework of a 3D model. The initial step in creating a surfaced model.
wireframe surfacing	Covering a wireframe model with surfaces.

Basic Concepts of Creating Surfaces

Three-dimensional surface modeling can be compared to constructing a building. You start by establishing the initial shape. Then you cover the rough framing with siding and roofing.

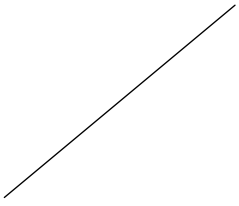


rough frame

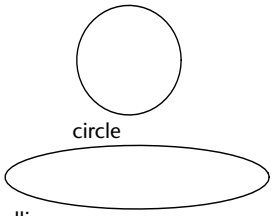


siding applied

One approach to surface modeling is to create a 3D framework of wires. *Wire* is a generic term for lines, arcs, circles, ellipses, 2D and 3D polylines, augmented lines, and splines, including splines created from existing part edges. This framework is called a *wireframe*.

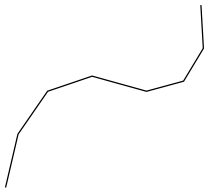


line

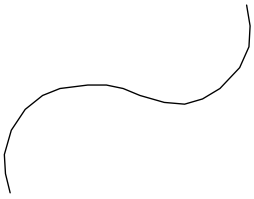


circle

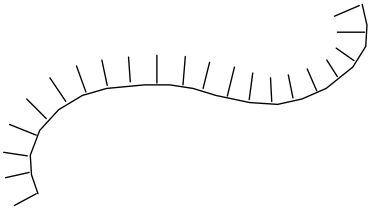
ellipse



polyline



spline



augmented line

Wires and surface parts can be intermixed to construct the framework of your 3D model. This initial step in creating a surfaced model is called *wireframe modeling*.

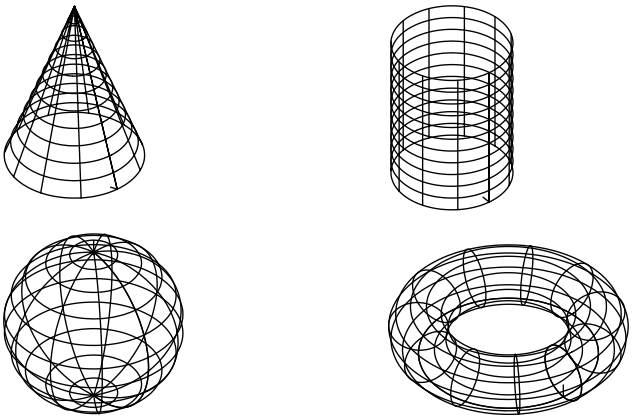
Once the 3D wireframe model is created, the next step is to cover the framework with a surface. This task is called *wireframe surfacing*. This technique is one approach to creating surfaces.

Working with Surfaces

In this tutorial you'll learn about these types of surfaces:

- Primitive, created by specifying values
- Motion-based, created by moving wires through space
- Skin, applied over a wireframe
- Derived, generated from existing surfaces

Primitive surfaces (cone, cylinder, sphere, and torus) do not require wireframes for their construction. To create a sphere surface, for example, you determine the center of the sphere and then enter a value for its radius. Primitive surfaces are most often used for conceptual design.



You can practice creating primitive surfaces on your own by choosing Surface ➤ Create Primitives.

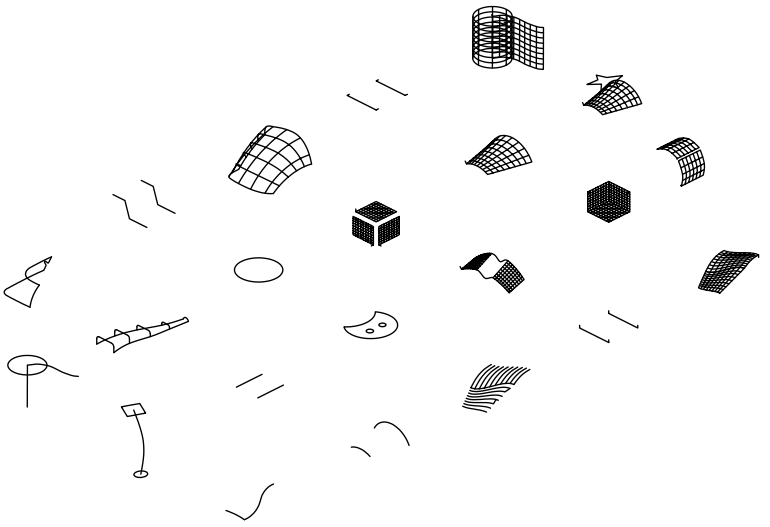
In this tutorial, you'll practice working with motion-based, skin, and derived surfaces you create from wireframes. You'll also work with simple wireframe geometry to learn surfacing techniques.

After you create each surface, use the Extents option of the ZOOM command to redisplay all of the wireframes provided in the drawing file. Then zoom to enlarge the object you need for the next exercise.

To set up your file

1 Open the file *t_surfs.dwg* in the *desktop\tutorial* folder. The wireframe objects you need for this lesson are included in this file.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.



Surface lines, called *U* and *V* lines, indicate the direction of the surface. Increasing the number of lines increases the density of the surface image.

2 Use the Mechanical Options dialog box to change the number of surface lines used to display surfaces.

Desktop Menu Surface ➤ Surface Options

Choose the Surface tab, and in Surface Properties specify:

U Display Wires: *Enter* **7**
V Display Wires: *Enter* **7**

Choose OK.

NOTE If you shade the surfaces you create to better view them, adjust the AutoCAD setting that controls back faces. Go to Assist ➤ Options and select the System tab. Choose Properties, then clear the check box for Discard Back Faces. Choose Apply & Close, then OK.

You are ready to create your first surfaces.

Creating Motion-Based Surfaces

Some surfaces are created by "moving" wires through space. These motion-based surfaces are revolved, extruded, and swept.

Revolved Surfaces

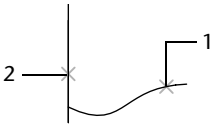
A revolved surface uses two wires: one establishes the constant shape of the surface, and the other is the axis about which to spin the shape. The revolved surface is created by the motion of a wire shape through space.

To revolve a surface

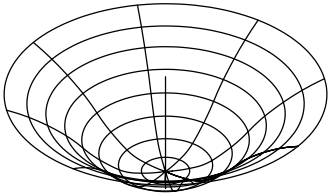
1 Use AMREVOLVESF to revolve a spline curve about an axis, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Revolve

Select path curves to revolve: *Select spline (1)*
Select path curves to revolve: *Press* ENTER
Specify axis start point or [Wire]: *Enter* **w**
Select wire to define axis: *Select line (2)*
Enter start angle <0>: *Press* ENTER
Enter included angle (+=ccw, -=cw) <Full circle>: *Press* ENTER



The curve revolves about the vertical line.



2 Use the Zoom Extents option of ZOOM to redisplay all the wireframes and select the object for the next exercise.

Extruded Surfaces

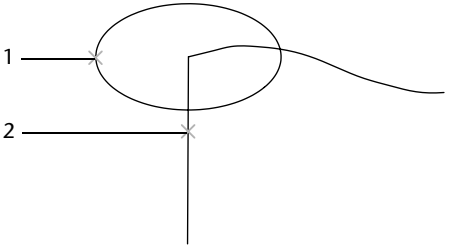
An extruded surface is created by a 3D wire shape moved along a straight line. You select a line, polyline, arc, or spline to extrude, and you specify the direction and magnitude of the extrusion.

To extrude a surface

1 Use AMEXTRUDESF to extrude a circle into a cylinder, responding to the prompts.

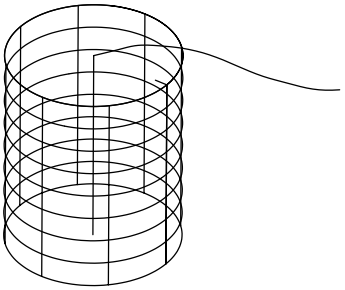
Desktop Menu Surface ➤ Create Surface ➤ Extrude

Select wires to extrude: *Select circle (1)*
Select wires to extrude: *Press* ENTER
Define direction and length.
Specify start point or [Viewdir/Wire/X/Y/Z]: *Enter* **w**
Select wire to define direction: *Select line (2) above its midpoint*
Enter an option [Accept/Flip] <Accept>:
Press ENTER *to accept the extrude direction*
Enter taper angle <0>: *Press* ENTER *to accept the default*

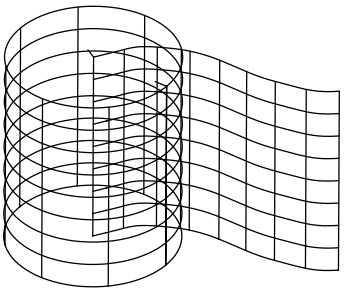


The extrusion direction is determined by your selection point on the wire. You selected the upper half of the line to extrude the circle along the length of the line. To extrude the circle in the opposite direction, select a location below the midpoint of the line.

The extruded surface should look like this.



2 For more practice, choose Extrude again. Select the spline and then a location on the line to determine the length and direction.



Because the line is obscured by the first surface you created, you may have difficulty selecting it. Press CTRL as you select, to cycle through the objects. Press ENTER when the line is highlighted.

3 Use the Zoom Extents option of ZOOM to redisplay all the wireframes and select the object for the next exercise.

Swept Surfaces

A swept surface is a wire cross section moved along a curved line called a *rail*. You can use multiple cross sections and one or two rails. The lines on the swept surface in the illustration are U and V display lines. Cross sections can be dissimilar but you need to select them in order.

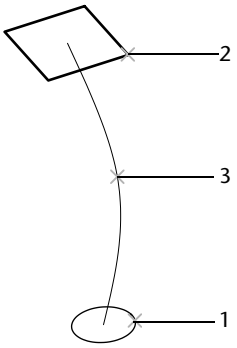
In this exercise, you use different combinations of cross sections and rails to create four swept surfaces.

To create a swept surface

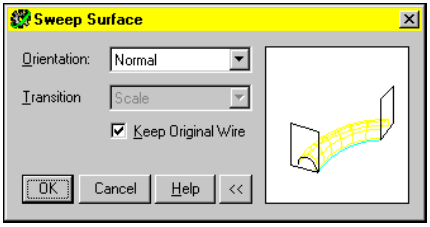
1 Use AMSWEEPSF to sweep two cross sections along a rail, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

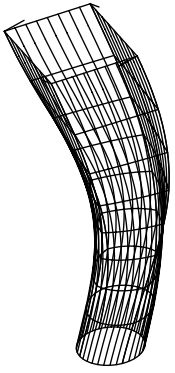
Select cross sections: *Select the first cross section (1)*
Select cross sections: *Select the second cross section (2) and press* ENTER
Select rails: *Select line (3) and press* ENTER



2 In the Sweep Surface dialog box, choose OK to accept default settings.



A message tells you that four surfaces will be created. Choose Continue. The surfaces should look like this.



You can select the individual surfaces to see the shape of each one.

Because one of the cross sections has sharp corners, a single surface cannot be created. Instead, four separate surfaces are created, each corresponding to one side of the rectangular cross section. For more information, see "To surface polylines with sharp corners" on page 550.

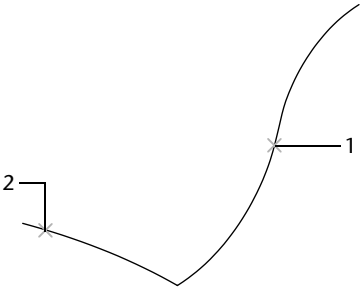
3 Use the Extents option of ZOOM to redisplay all the wireframes and select the object for the next exercise.

To sweep a spline along a rail

1 Sweep one spline along a rail, responding to the prompts.

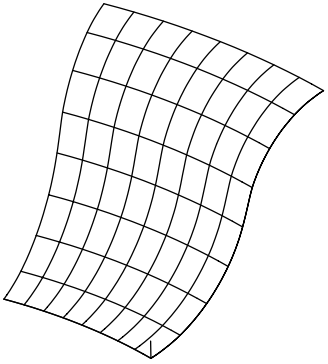
Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *Select spline (1) and press* ENTER
Select rails: *Select line (2) and press* ENTER



2 In the Sweep Surface dialog box, choose OK to accept the default settings.

Your surface should look like this.



You created a nonuniform rational B-spline (NURBS) surface from a spline.

You can convert a NURBS surface into a part by adding thickness.

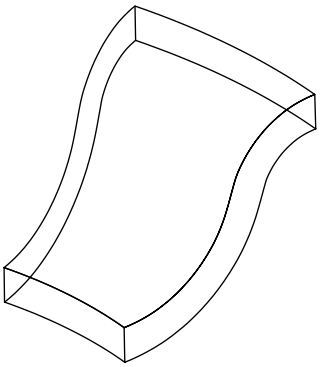
NOTE Save a copy of your NURBS surface if you will need it later. When you thicken a surface, the original surface is consumed and disappears.

To convert a NURBS surface to a solid

1 Use AMTHICKEN to convert a NURBS surface to a thin solid, responding to the prompts.

Desktop Menu Surface ➤ Surface Thicken

Select surfaces to thicken: *Select the surface*
Select surfaces to thicken: *Press* ENTER
Select direction to thicken [Flip/Accept] <Accept>:
Flip to point the direction arrow down, and press ENTER
Thickness <1.0000>: *Press* ENTER



A solid is created, and a new part icon is displayed in the Browser.

2 Use the Extents option of ZOOM to redisplay all the wireframes and select the object for the next exercise.

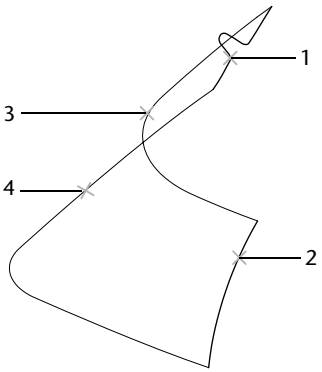
A swept surface is useful when your design has different shapes at either end.

To sweep dissimilar shapes

1 Sweep two dissimilar shapes along two nonparallel rails, responding to the prompts.

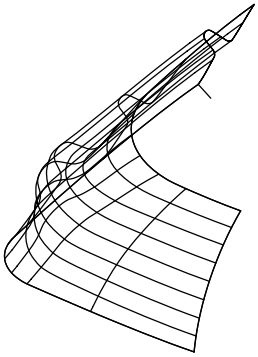
Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *Select first shape (1)*
Select cross sections: *Select second shape (2) and press* ENTER
Select rails: *Select first rail (3)*
Select rails: *Select second rail (4)*



2 In the Sweep Surface dialog box, choose OK to accept the default settings.

The swept surface should look like this.



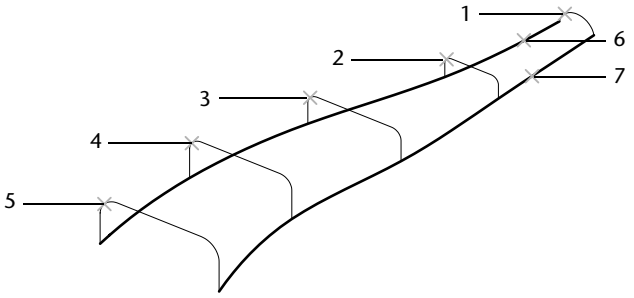
3 Use the Extents option of ZOOM to redisplay all the wireframes and select the object for the next exercise.

To sweep multiple cross sections along two rails

1 Sweep multiple cross sections along two rails, responding to the prompts.

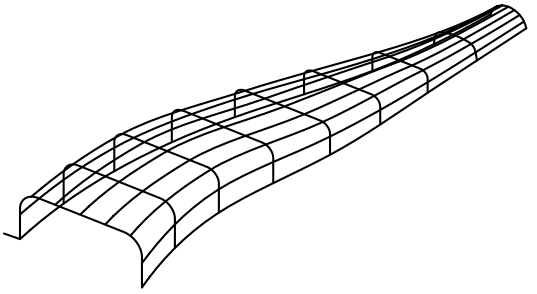
Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections:
Select cross sections (1) through (5) in consecutive order and press ENTER
Select rails: *Select first rail (6)*
Select rails: *Select second rail (7)*



2 In the Sweep Surface dialog box, choose OK to accept the default scale.

Because the cross sections have rounded corners, a single surface results. If the cross section had square corners, three surfaces would have been created.



3 Use the Extents option of ZOOM to redisplay all the wireframes and select the object for the next exercise.

Creating Skin Surfaces

A skin surface drapes over a wireframe model. After the wireframe is removed, the surface retains the shape of the wireframe. Skin surfaces are ruled, planar, lofted U, and lofted UV.

Ruled Surfaces

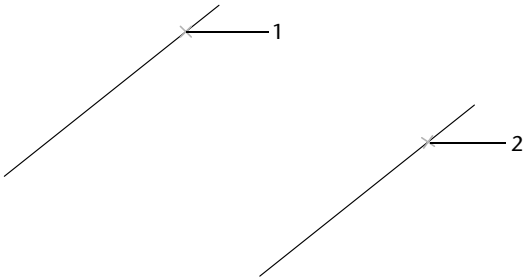
A ruled surface is a straight, flat shape stretched between two wires of any 3D shape. You can create a ruled surface between any two nonintersecting wires that can represent the top and the bottom. The top and bottom can be open or closed wires. You can create a ruled surface from two augmented lines as well, and from a single augmented line by adding thickness to it.

To surface two wires

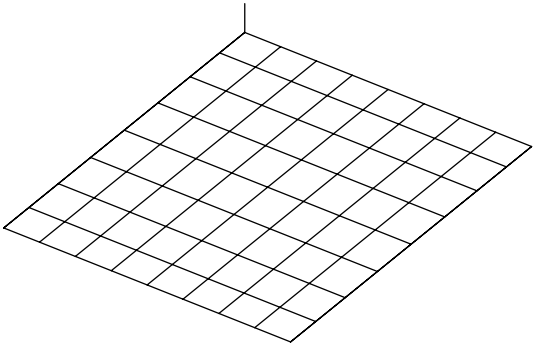
1 Use AMRULE to create the ruled surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select line (1)*
Select second wire: *Select line (2)*



The ruled surface shows the surface *normal*, a short vertical line in one corner. A surface normal shows where the surface starts and which direction is out.



If surface normal indicators are too small, use the DISPSF system variable to change the size. In the Individual Surface Display dialog box, change the value in the Normal Length field. Adjust the setting as needed.

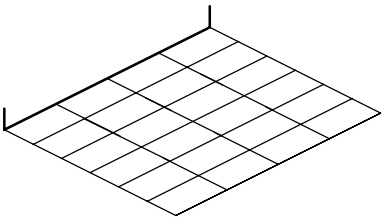
You can create a ruled surface from two augment lines, or by adding width to a single augmented line. Besides using the command method, you can access AMAUGMENT in the Desktop menu by choosing Surface ➤ Create Wireframe ➤ Augmented Lines.

To create an augmented line on a surface

1 Use AMAUGMENT to create an augmented line on the surface.

Command AMAUGMENT

Select surface wire or [Angle/Distance/Spacing]:
Select the leftmost wire of the surface
Select surface wire or [Angle/Distance/Spacing]: *Press* ENTER



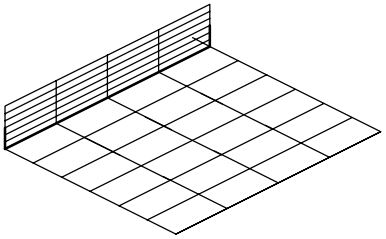
When you use an augmented wire to create a surface, the vectors are ignored. You specify a value for the width of the surface.

To surface an augmented line

1 Use AMRULE to create a surface from the augmented wire, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select the augmented line*
Enter an option [Next/Accept] <Accept>: *Press* ENTER
Enter width (or) [Select second wire] <1.0000>: *Press* ENTER



The surface extends beyond the vectors to the specified width.

You can use this technique to create a ruled surface normal to any existing ruled surface. First you create a parting line on the surface and project it to get an augmented line, and then create the ruled surface from the augmented line.

2 Undo the surfaces you created so that you can use the two original augmented lines in subsequent exercises.

Desktop Menu Edit ➤ Undo

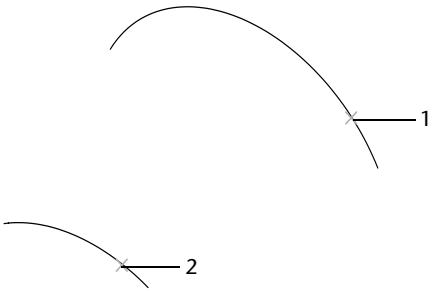
3 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

To surface two arcs with different radii

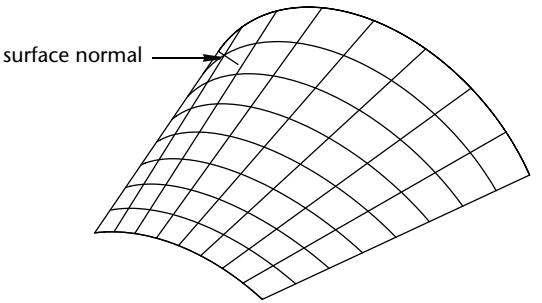
1 Create a surface from two arcs of different radii, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select arc (1)*
Select second wire: *Select arc (2)*



The ruled surface should look like this.



2 To experiment, use the previous example and select the arcs in a different order.

3 Erase the surface you just created.

Desktop Menu Modify ➤ Erase

Select the surface and press ENTER.

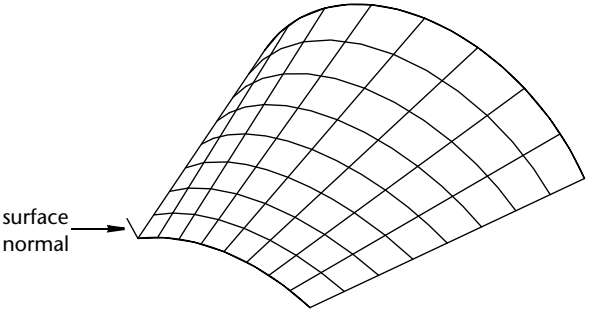
If creating the surface was the last command, you can use UNDO.

4 Create a surface again, selecting the arcs in reverse order.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select arc (2)*
Select second wire: *Select arc (1)*

Selecting the arcs in different order changes the surface normal.



5 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

In the next exercise, you surface polylines. A surface follows a spline exactly but approximates the polyline by a curve. The Polyline Fit default setting of 150 maintains all corners less than 150 degrees as sharp corners. In such cases, multiple surfaces are created.

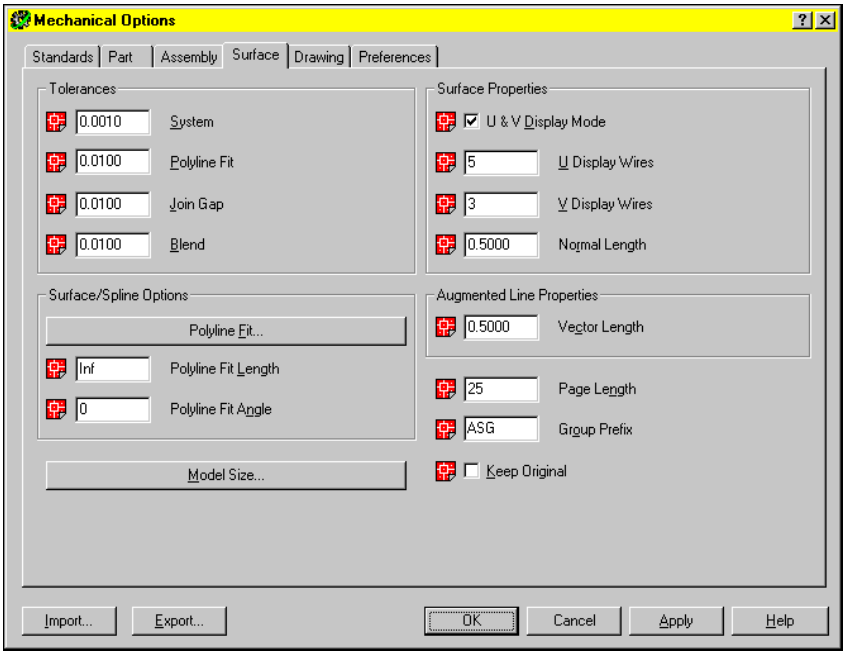
To surface polylines with sharp corners

1 Use the Mechanical Options dialog box to adjust the Polyline Fit settings:

Desktop Menu Surface ➤ Surface Options

Choose the Surfaces tab and then choose Polyline Fit.

2 In the Polyline Fit dialog box, change the angle setting to **0** and change the length setting to **inf** (infinity) or a specific length longer than the longest line segment.



Choose OK.

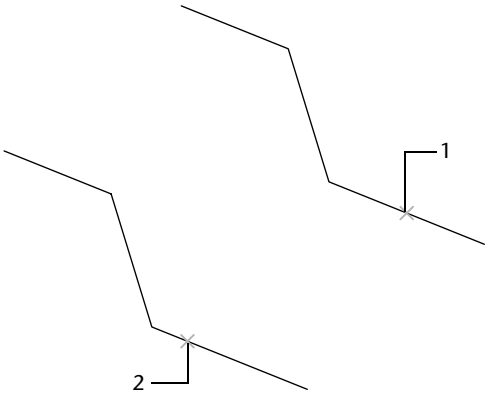
These settings force sharp corners to convert to a smooth curved surface.

3 Choose OK to exit the Mechanical Options dialog box.

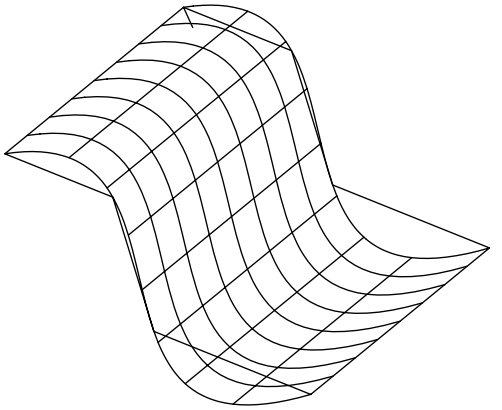
4 Use AMRULE to create a curved surface from polylines drawn with sharp corners, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select polyline (1)*
Select second wire: *Select polyline (2)*



A continuous smooth curved surface is created.



5 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

Now use the same polylines, but choose a fit angle to recognize sharp corners.

Use the Polyline Fit dialog box to change the angle setting until the image tile resembles the angles you want to recognize as corners. Choose a setting between 150 and 165 to recognize most corners.

To create surfaces with sharp angles

1 Use AMOPTIONS to reset the fit angle to 150 and re-create the ruled surface.

Desktop Menu Surface ➤ Surface Options

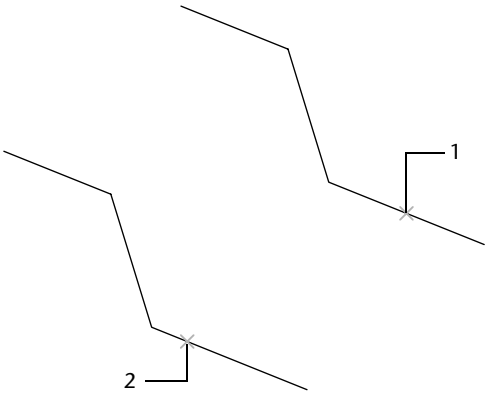
2 Erase the surface you just created.

Desktop Menu Modify ➤ Erase

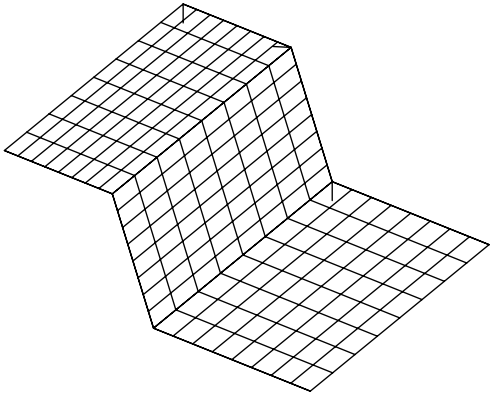
3 Use AMRULE to create three ruled surfaces from polylines drawn with sharp corners, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select polyline (1)*
Select second wire: *Select polyline (2)*



4 A message tells you that three surfaces will be created. Choose Continue.



The three surfaces follow the sharp corners of the polylines.

5 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

Trimmed Planar Surfaces

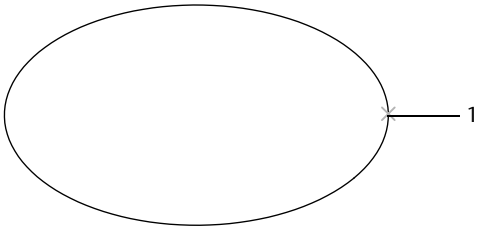
A planar surface may be constructed from lines, arcs, splines, polylines, or simply two locations, if the selected objects are closed and on the same plane. The exterior shape of the 2D wire shape becomes the trimmed edge of the surface. In this exercise, you create trimmed planar surfaces.

To create a planar trimmed surface using a closed polyline

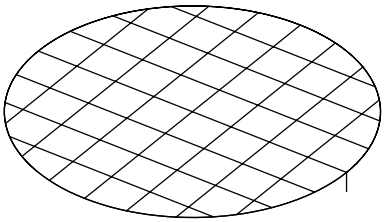
1 Use AMPLANE to create a planar trim surface from a closed polyline, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Planar Trim

Specify first corner or [Plane/Wires]: _wire
Select wires: *Select object (1)*
Select wires: *Press* ENTER



The planar surface is trimmed, using the closed polyline as the border. The *U* and *V* lines appear inside the border.



2 Use the Extents option of ZOOM to redisplay all of the wireframes, and select the object for the next exercise.

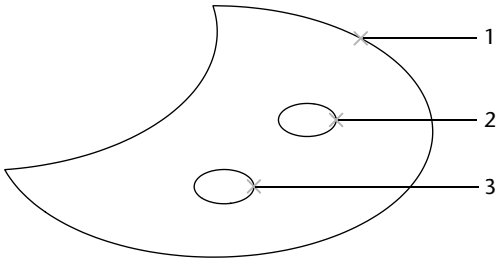
The closed polyline for the next exercise contains two interior circles that are coplanar. You create a trimmed surface using the polyline and circles.

To create a trimmed planar surface using three closed polylines

1 Create a planar trim surface from three polylines, responding to the prompts.

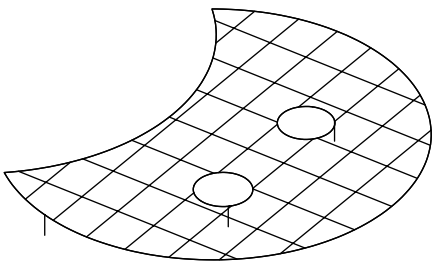
Desktop Menu Surface ➤ Create Surface ➤ Planar Trim

Specify first corner or [Plane/Wires]: _wire
Select wires: *Select object (1)*
Select wires: *Select interior circle (2)*
Select wires: *Select circle (3)*
Select wires: *Press* ENTER



Instead of selecting the objects individually, you could also drag a crossing window around all of them and press ENTER.

A trimmed planar surface is automatically trimmed to the largest boundary created by the wire. As in this example, when closed areas exist inside the boundary, these holes are trimmed out.



2 Use the Extents option of ZOOM to redisplay all of the wireframes, and select the object for the next exercise.

Lofted Surfaces

You can create a lofted surface from one or two sets of wires, each with similar attributes, such as having approximately the same direction. You must select the wires in consecutive order.

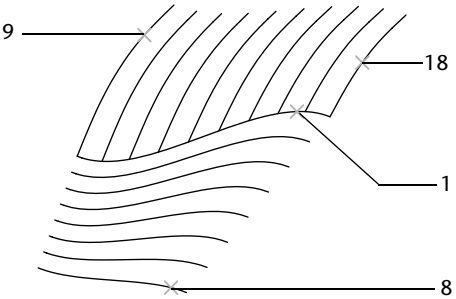
A lofted U surface is stretched between any number of wires that share similar characteristics. The example contains two sets of wires from which you create two surfaces. The light blue polylines are approximately horizontal, and the green lines are approximately vertical. First, you create a surface from the horizontal polylines.

To create a lofted surface using a set of wires

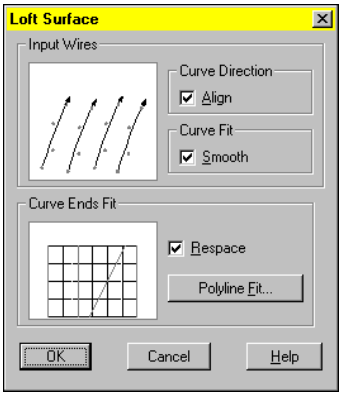
1 Use AMLOFTU to surface a set of vertical wires, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ LoftU

Select U wires: *Select lines (1) through (8) in consecutive order and press* ENTER



2 In the Loft Surface dialog box, choose OK to accept the default settings.



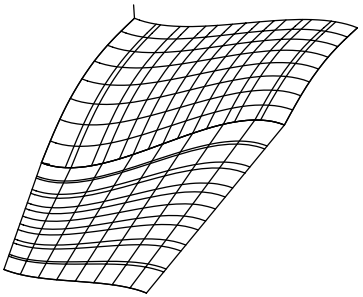
3 Surface a set of horizontal wires, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ LoftU

Select U wires: *Select lines (9) through (18) in consecutive order and press* ENTER

4 In the Loft Surface dialog box, choose OK to accept the default settings.

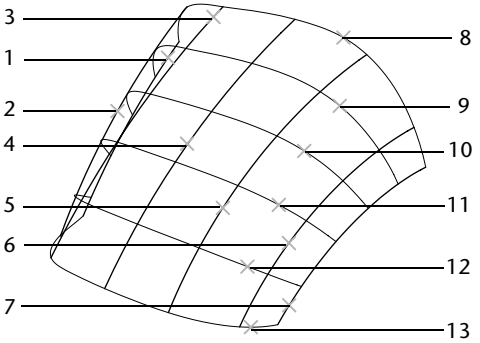
The two surfaces, one horizontal and one vertical, should look like these.



5 Use the Extents option of ZOOM to redisplay all the wireframes and select the object for the next exercise.

A lofted UV surface is stretched over two sets of wires. Each wire in one set crosses every wire in the other set. Two sets of wires can accurately describe a complex surface.

You can select wires directly, as you did in the previous exercise, or you can select two *groups* of wires. In this exercise, you create two groups of wires and then create a lofted surface from the two groups. The wires in the *U* direction are magenta (purple), and those in the *V* direction are cyan (light blue).



To create a single lofted surface from two groups of wires

1 Group the magenta *U* lines.

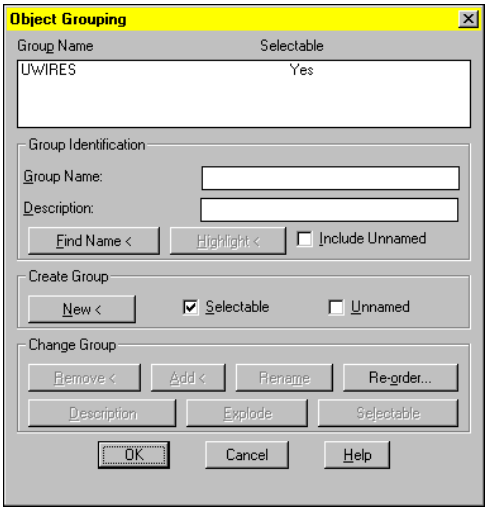
Command GROUP

In the Object Grouping dialog box specify:

Group Name: *Enter* **uwires**
Create Group: *Choose New to close the dialog box*

Respond to the prompt as follows:

Select objects: *Select lines (1) through (7) in consecutive order and press* ENTER



In the Object Grouping dialog box, choose OK.

2 Press ENTER to repeat the GROUP command.

In the Object Grouping dialog box specify:

Group name: *Enter* **vwires**
Create Group: *Choose New to close the dialog box*

Respond to the prompt as follows:

Select objects: *Select lines (8) through (13) in consecutive order and press* ENTER

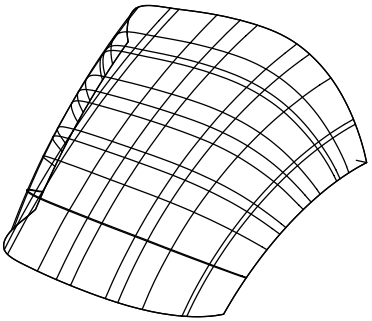
In the dialog box, choose OK.

3 Use AMLOFTU to loft a surface from two groups of wires, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ LoftUV

Select U wires: *Enter* **g**
Enter group name: *Enter* **uwires**
7 found
Select U wires: *Press* ENTER
Select V wires: *Enter* **g**
Enter group name: *Enter* **vwires**
6 found
Select V wires: *Press* ENTER
Enter an option [eXit/Loft/Node check] <Loft>: *Press* ENTER

You have created a surface from the two groups of wires.



4 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

Creating Derived Surfaces

Derived surfaces are generated from existing surfaces: blended, offset, fillet, and corner fillet. Derived surfaces can be trimmed.

Blended Surfaces

You can create a blended surface between two, three, or four wires or surfaces. The blended surface is tangent to the surfaces or wires from which it is created.

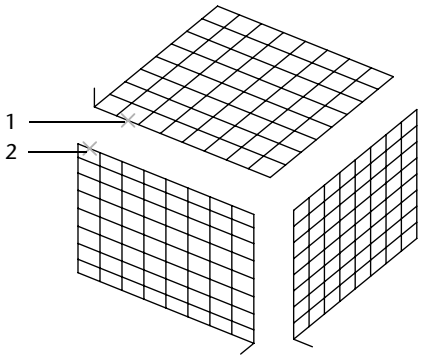
NOTE To create surfaces correctly, select lines at points shown in the illustrations.

To create a blended surface

1 Use AMBLEND to create a blended surface from two wires, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Blend

Select first wire: *Select surface (1)*
Select second wire: *Select surface (2)*
Select third wire [Weights]: *Press* ENTER



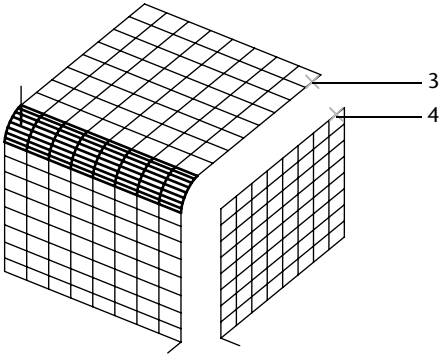
You have created the first blended surface.

The type of object you select affects the blended surface. When you select surfaces or augmented lines, you are prompted for the weight of the surface edge.

2 Create the second blended surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Blend

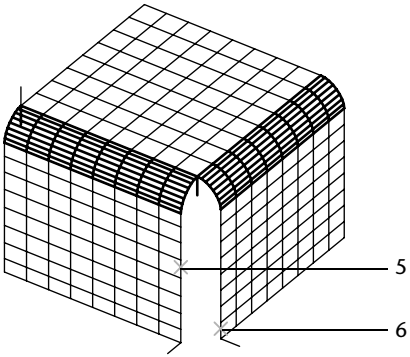
Select first wire: *Select surface (3)*
Select second wire: *Select surface (4)*
Select third wire [Weights]: *Press* ENTER



3 Create the third blended surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Blend

Select first wire: *Select surface (5)*
Select second wire: *Select surface (6)*
Select third wire [Weights]: *Press* ENTER

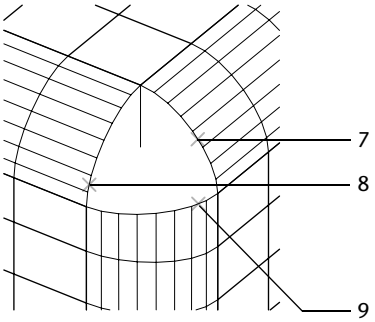


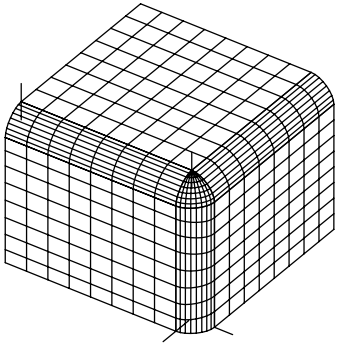
4 Use ZOOM to enlarge the corner area created by the blended surfaces.

5 Use AMBLEND to create a corner blended surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Blend.

Select first wire: *Select surface (7)*
Enter an option [next/Accept] <Accept>: *Press* ENTER
Select second wire: *Select surface (8)*
Enter an option [next/Accept] <Accept>: *Press* ENTER
Select third wire [Weights]: *Select surface (9)*
Select fourth wire: *Press* ENTER





The order in which you select the objects to blend determines the orientation of the corner created by the blend.

Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

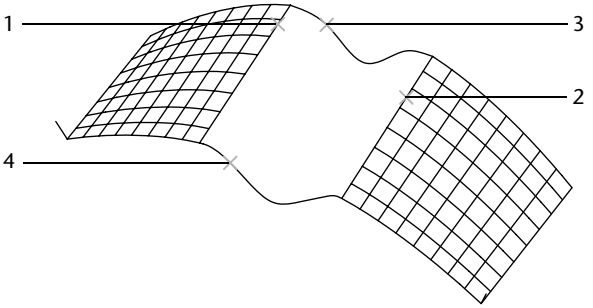
Next, you create a blended surface from two surfaces and lines. To create a blended surface from four objects, make the selections in the order shown; the objects cannot be selected in consecutive order.

To blend surfaces and lines

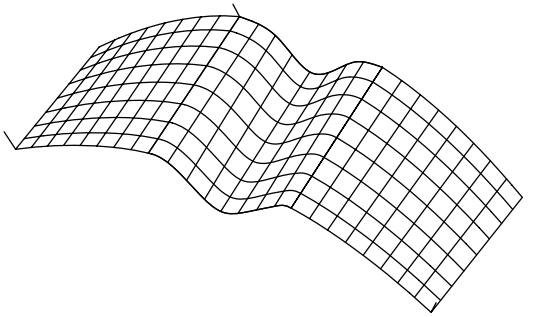
1 Use AMBLEND to create a surface from two surfaces and two lines, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Blend

Select first wire: *Select surface (1)*
Select second wire: *Select surface (2)*
Enter an option [Next/Accept] <Accept>: *Press* ENTER
Select third wire [Weights]: *Select wire (3)*
Select fourth wire: *Select wire (4)*



The new surface is created between the original surfaces and lines.



2 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

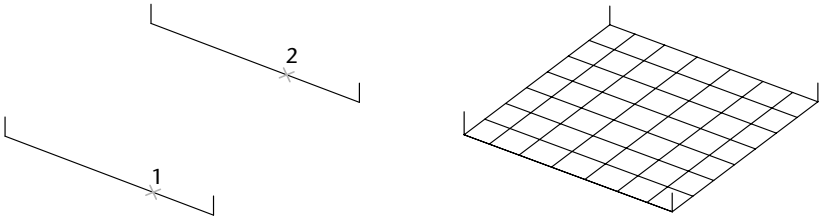
Next, you create two surfaces, each blended from two augmented lines.

To blend augmented lines

1 Use AMBLEND to create a surface from two augmented lines, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Blend

Select first wire: *Select line (1)*
Select second wire: *Select line (2)*
Select third wire [Weights]: *Press* ENTER

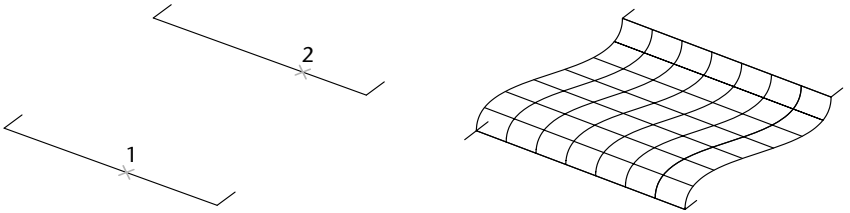


2 Use ZOOM to redisplay wireframes. Then select the second set of lines.

3 Create a surface from the second set of augmented lines, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Blend

Select first wire: *Select augmented line (1)*
Select second wire: *Select augmented line (2)*
Select third wire [Weights]: *Press* ENTER



Compare these two surfaces to see how the direction of the augmented line vectors affects the resulting surface.

4 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

Offset Surfaces

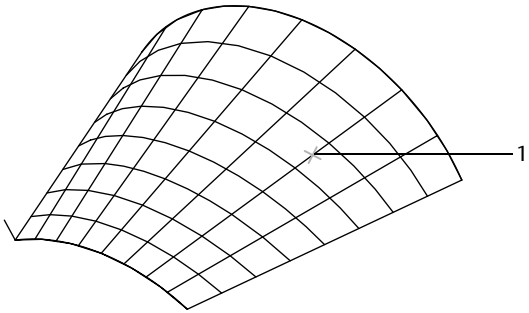
An offset surface is a duplicate of an existing surface, offset by a specific distance. When you create an offset surface, you can keep the original or remove it, as needed.

To create an offset surface

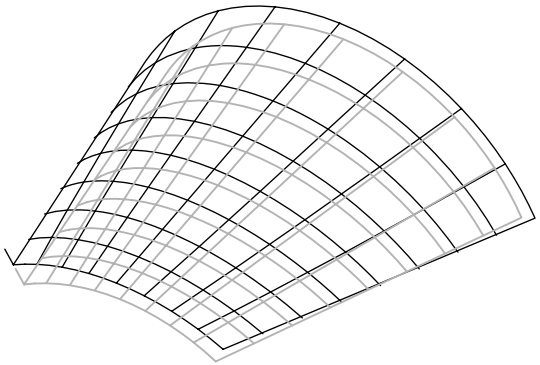
1 Use AMOFFSETSF to create an offset surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Offset

Select surfaces to offset: *Select surface (1)*
Select surfaces to offset: *Press* ENTER
Distance=1.0000 Keep=Yes
Enter offset distance or [Keep] <1.0000>: *Enter* **k**
Keep original surface(s) [Yes/No] <No>: *Enter* **y**
Distance=1.0000 Keep=Yes
Enter offset distance or [Keep] <1.0000>: *Enter* **0.5**



The new surface is offset from the original surface, which is normal at all locations.



2 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

Fillet and Corner Surfaces

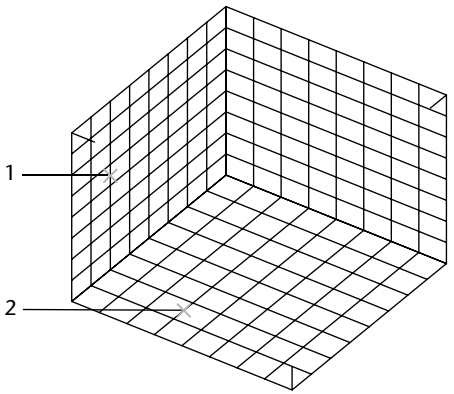
In this exercise, you create fillet surfaces between two selected surfaces, and a corner fillet where three fillet surfaces intersect. You trim the original surfaces back to the fillet surfaces.

To create fillet and corner surfaces

1 Use AMFILLETSF to create a fillet between two surfaces, responding to the prompts.

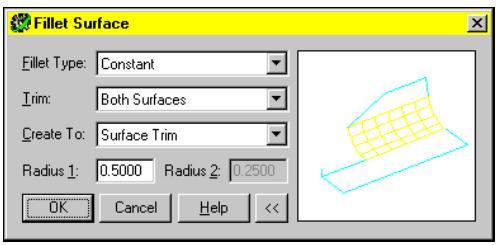
Desktop Menu Surface ➤ Create Surface ➤ Fillet

Select first surface or quilt interior edge: *Select surface (1)*
Select second surface: *Select surface (2)*



2 In the Fillet Surface dialog box, specify:

Fillet Type: Constant
Trim: Both Surfaces
Create To: Surface Trim
Radius: *Enter* **0.5**

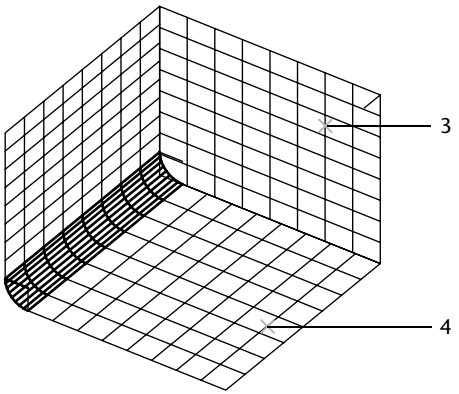


Choose OK.

3 Create a fillet between another two surfaces, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Fillet

Select first surface or quilt interior edge: *Select surface (3)*
Select second surface: *Select surface (4)*



4 In the Fillet Surface dialog box, specify:

Fillet Type: Constant
Trim: Both Surfaces
Create To: Base Surface
Radius: *Enter* **0.75**

Choose OK.

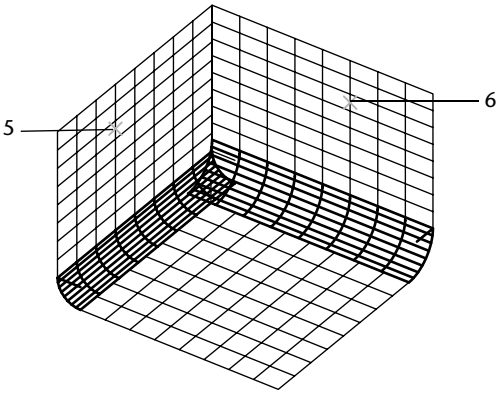
The Base Surface option was not available when you created the first fillet surface. It is available now because the first surface was trimmed and now differs from the base surface. Next, trim to the base surface, not the trimmed surface.

The next illustration shows both fillets, which you will trim later.

5 Create a fillet between another two surfaces, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Fillet

Select first surface or quilt interior edge: *Select surface (5)*
Select second surface: *Select surface (6)*



6 In the Fillet Surface dialog box, specify:

Fillet Type: Constant
Trim: Both Surfaces
Create To: Base Surface
Radius: *Enter* **0.6**

Choose OK.

The fillets overlap at the corner. You need to trim the overlap with a corner fillet.

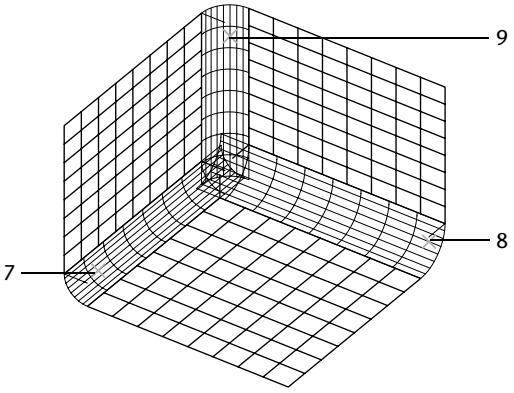
To create a corner fillet

1 Use AMCORNER to trim the overlapping fillets, responding to the prompts.

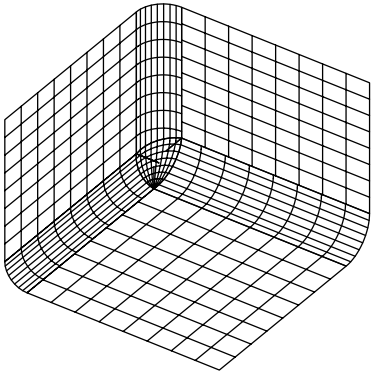
Desktop Menu Surface ➤ Create Surface ➤ Corner Fillet

NOTE The default is set to trim the corner fillet to the three fillet surfaces. If you do not want to trim, enter **T** at the first prompt and change the setting to No.

Trim=Yes
Select first fillet surface or [Trim]: *Select surface (7)*
Select first fillet surface or [Trim]: *Select surface (8)*
Select first fillet surface or [Trim]: *Select surface (9)*



The trimmed corner fillet looks like this.



2 Use the Extents option of ZOOM and select the object for the next exercise.

Editing Surfaces

As you create models, you need to combine surfaces and trim them where they overlap. You will learn four surface editing techniques: adjusting surfaces, joining surfaces, trimming surfaces at intersections, and trimming surfaces by projection.

Adjusting Adjacent Surfaces

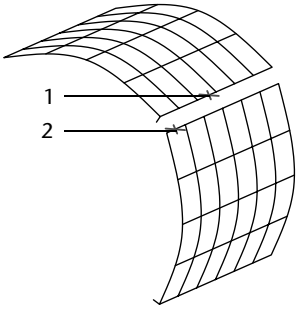
You can control the tangency of two adjacent surfaces by adjusting them to create one continuous surface. When you select the edges of two adjacent surfaces to adjust, the first edge you select is the control surface. You can use a quilt for the control surface. By default, 20 per cent of the total area of each surface is adjusted. You can change the percentage for either or both of the the adjustment areas, depending on which surface shape you need to retain.

To adjust adjacent surfaces

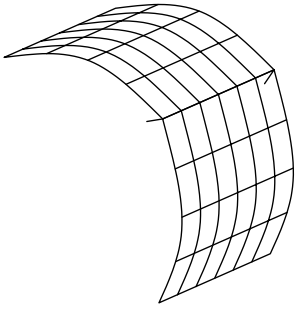
1 Use AMADJUST to convert two surfaces into one surface, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Adjust

Select surface edge to adjust: *Select the control surface edge (1)*
Select surface edge to adjust: *Select the adjacent surface edge (2)*



First surface=20.0000% Second surface=20.0000% cOntinuity=Smooth
Keep=No
Enter an option [First surface/Second surface/cOntinuity/Keep] <Continue>:
Enter **s**
Enter adjustment for the second surface <20.0000%>: *Enter* **40**
First surface=20.0000% Second surface=40.0000% cOntinuity=Smooth
Keep=No
Enter an option [First surface/Second surface/cOntinuity/Keep] <Continue>:
Enter **o**
Continuity [Coincident/Smooth] <Coincident>: *Press* ENTER
First surface=20.0000% Second surface=40.0000% cOntinuity=Coincident
Keep=No
Enter an option [First surface/Second surface/cOntinuity/Keep] <Continue>:
Press ENTER



The two surfaces are adjusted to meet.

Joining Surfaces

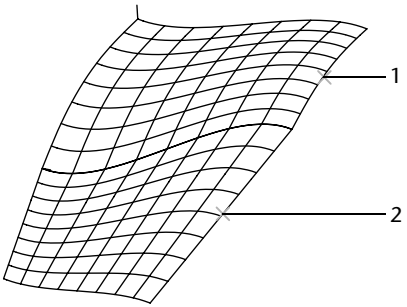
You can join multiple surfaces into a single surface. Each surface is indicated by two surface normal indicators.

To join surfaces

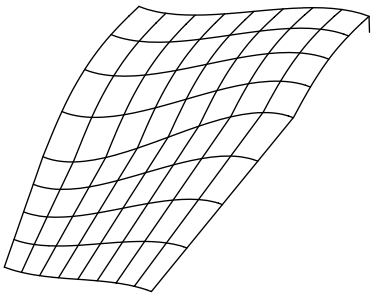
1 Use AMJOINSF to join the surfaces you select, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Join

Select surfaces to join: *Select surface (1)*
Select surfaces to join: *Select surface (2)*
Select surfaces to join: *Press* ENTER



The two surfaces are joined. Only one surface normal indicator is shown.



To force two surfaces to join, choose Surface ➤ Surface Options. In the Mechanical Options dialog box, choose the Surface tab and adjust the Join Gap Tolerance. Two untrimmed surfaces are joined automatically if they are within twice the gap tolerance.

2 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

Trimming Intersecting Surfaces

You can trim intersecting surfaces, including quilted surfaces, and create a 3D polyline at the intersection. The 3D polyline can then be used to create another surface.

A 3D point is created at the intersection of a wire curve and a surface or quilt. If the wire curve intersects the surface or quilt more than once, points are created everywhere an intersection takes place.

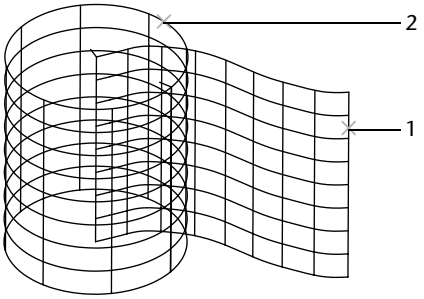
When you trim, always make your selections on the portion of the surface you want to keep.

To trim intersecting surfaces

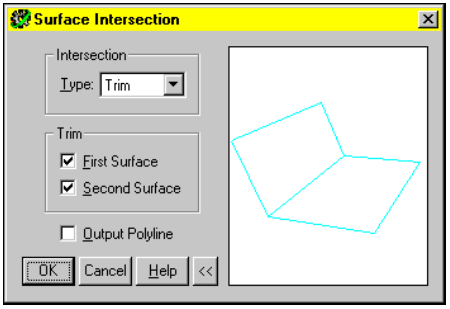
1 Use AMINTERSF to trim the surface you want to keep, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Intersect Trim

Select first surface/quilt or wire: *Select surface (1)*
Select second surface: *Select surface (2)*

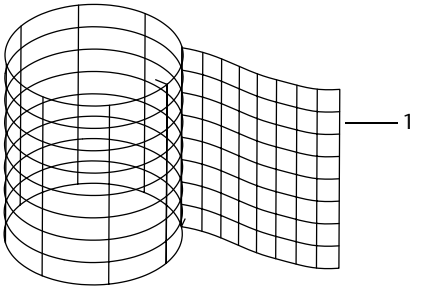


2 In the Surface Intersection dialog box, check the options indicated in the following illustration and choose OK.



The first surface you selected is trimmed at its intersection with the second surface.

Using the Trim options, you can trim neither, one, or both surfaces at their intersection. As you change these options, the dynamic image in the dialog box displays the results.



3 Use UNDO to erase the trim. Then try selecting surfaces at different locations.

4 Use the Extents option of ZOOM to redisplay all of the wireframes and select the object for the next exercise.

Trimming Surfaces by Projection

You can project a wire onto a surface to trim a shape that corresponds to the wire shape. You select the portion of the surface you want to keep.

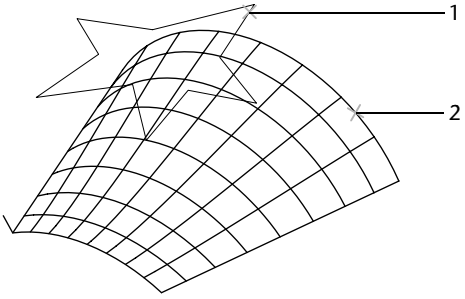
In this exercise, you trim a curved surface with a star-shaped polyline.

To trim a surface by a projected wire

1 Use AMPROJECT to project the wire to trim the surface, responding to the prompts.

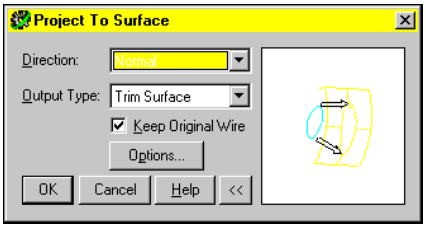
Desktop Menu Surface ➤ Edit Surface ➤ Project Trim

Select wires to project: *Select polyline (1) and press* ENTER
Select target surfaces/quilts: *Select surface (2) on the side you want to keep*
Select target surfaces/quilts: *Press* ENTER



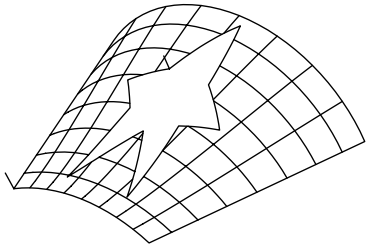
2 In the Project to Surface dialog box, specify:

Direction: *Choose* Normal
Output Type: *Choose* Trim Surface
Keep Original Wire: *Check the check box*



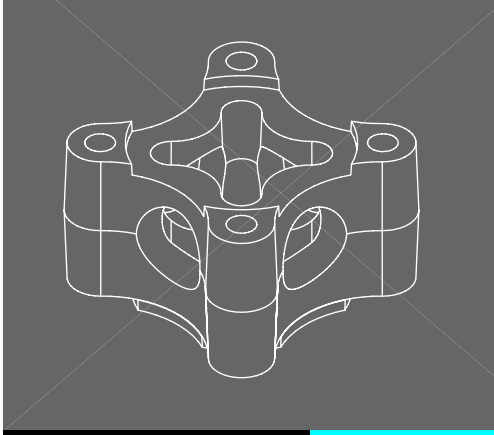
Choose OK.

The polyline is projected onto the curved surface, trimming out the surface inside the polyline.



3 Use UNDO and try different selection points and output types for the projection.

Combining Parts and Surfaces



In This Chapter

20

In Autodesk® Mechanical Desktop®, surfaces are valuable features because they can represent complex curved shapes. When joined to a parametric part, they cut away an angular surface and replace it with a sculpted face. A surface may also add material to a part as a protrusion. In this tutorial, you combine parametric and surface modeling by creating a camera body with a sculpted face.

- Creating a part with multiple features
- Creating a simple surface
- Attaching a surface parametrically to a part
- Cutting out features
- Creating mounting holes
- Sketching on work planes
- Revising and finishing a design

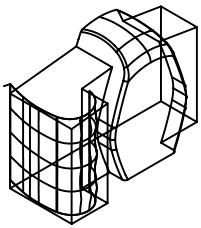
Key Terms

Term	Definition
base surface	A basic underlying surface that carries a shape across a larger area. Can be trimmed to precise shapes as needed, but the base surface remains intact and may be displayed.
model view	Changes orientation of the viewer so that the object is viewed from a different position. Individual views can be displayed in multiple viewports. For example, enter **3** at the Command prompt to create three viewports with default views: top, front, and right isometric.
NURBS	Acronym for nonuniform rational B-spline. The SPLINE command creates a true NURBS curve and can be used to create a surface.
rail	One or more curved lines along which surfaces are swept. They form the curvature of a swept surface.
spline	A curved line defined by specified control points that assumes a unique shape. Used to create curved surfaces. The radius of a spline curve is constantly changing. Splines are used as the basis of free-form surfaces.
surface cut	A feature created when a surface is joined to a part. Where the surface cuts the part or protrudes, the part face assumes the curved shape of the surface. The surface, like other features, is parametric; both the surface and the part retain their parametric relationship whenever either is modified.
wire	A generic term referring to lines, arcs, circles, ellipses, 2D and 3D polylines, augmented lines, and splines.
work plane	An infinite plane attached to a part. Can be designated as a sketch plane and can be included in a constraint or dimension scheme. Work planes can be either parametric, or non-parametric.
work point	A parametric work feature used to position a hole, the center of a pattern, or any other point for which there is no other geometric reference.

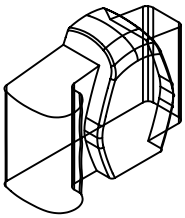
Basic Concepts of Combining Parts and Surfaces

You can use Mechanical Desktop® to create angular-shaped parts. You can apply 3D surfaces to those parts to create hybrid parts consisting of a mixture of angular and curved shapes. With Mechanical Desktop you can create model designs with shapes of varying types.

You can apply surfaces to Mechanical Desktop parts and use those surfaces to cut material from a parametric part, to create any hybrid shapes that your design requires.



before surface cut



after surface cut

You can also use surfaces to add material to angular parts.

Using Surface Features

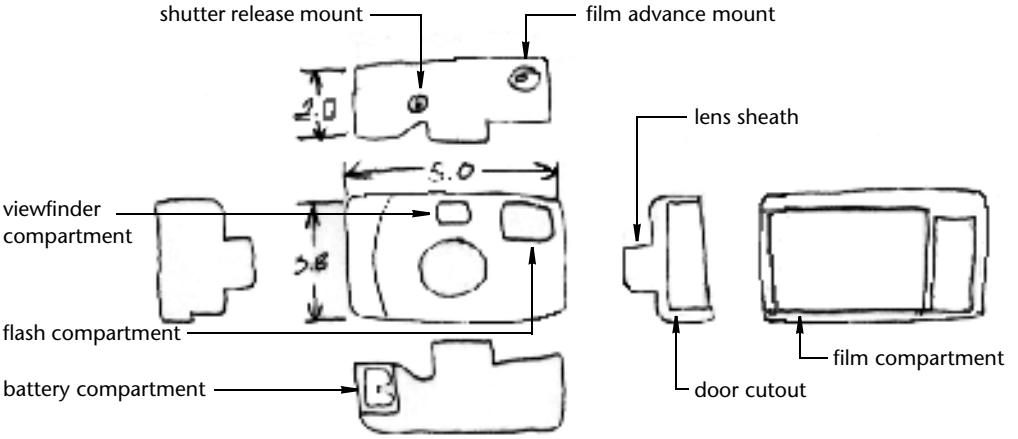
A feature created from surfaces has the shape of a contoured surface. You either cut away material or add material as a protrusion to join it to a part.

In this tutorial, you cut away an angular face and replace it with a sculpted surface.

Surfaces must have these characteristics to be used as features on models:

- A contoured surface must have four logical boundaries.
- The curved shape must be a single surface. If you need multiple surfaces to represent the shape, you must join them into a single surface.
- The surface must be a nontrimmed base surface. Join only base surfaces, not interior trim edges of trimmed surfaces.
- The surface must extend past the part on all four sides.
- A surface cannot contain sharp corners.
- Surfaces should have a minimum number of internal patches. These surfaces work better and faster than complex ones.

First, sketch the camera from all sides (top, side, front, and isometric views). With a complete idea, you can decide where to place features on the camera body.



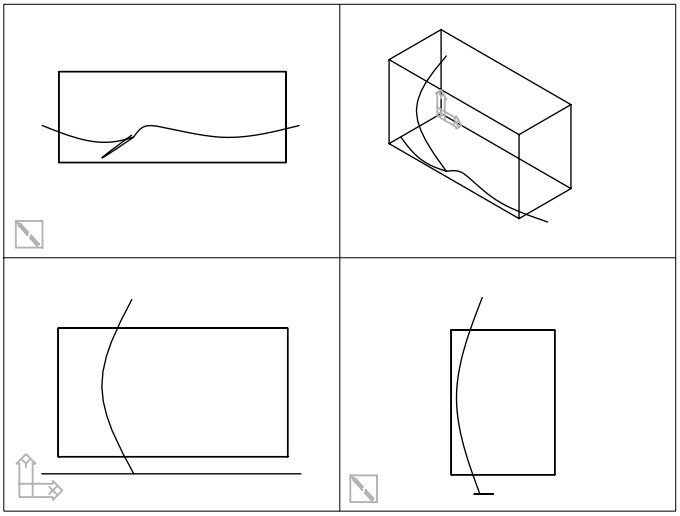
The camera body, which is common to all other features, is the base feature. The camera design has many other features, some of which are cutaways from the body: battery and film compartments and cutouts for their doors; mounting holes for the film advance and shutter release; and compartments for the viewfinder and the flash. The lens sheath feature, a sculpted surface, is joined to the camera face.

Creating Surface Features

Open the file *camera.dwg* in the *desktop\tutorial* folder. The file contains the settings you need, and the geometry to create the camera body—an extruded feature and some NURBS curves. You use NURBS to create the surface for the sculpted camera face.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

You can work on your model in any viewport, moving among views as you create features.



First, you create two surfaces by sweeping wires along a rail. Then, you join them into a single surface and extend the surface so that it covers the camera body.

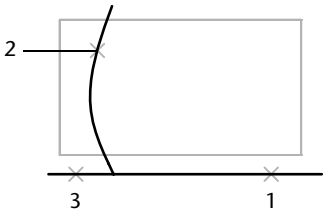
NOTE If you prefer to use toolbuttons to access commands, choose Surface ➤ Launch Toolbar.

To create a swept surface

1 Use AMSWEEPSF to sweep a spline along a rail, responding to the prompts.

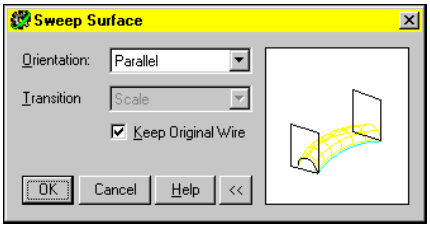
Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *In the front view, choose the right horizontal spline (1)*
Select cross sections: *Press* ENTER
Select rails: *Select the vertical spline (2)*
Select rails: *Press* ENTER



2 In the Sweep Surface dialog box, specify:

Orientation: Parallel



Choose OK. The first half of the swept surface is created.

3 Create the second half of the swept surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *Select the left horizontal spline (3)*
Select cross sections: *Press* ENTER
Select rails: *Select the vertical spline (2)*
Enter an option [Next/Accept] <Accept>: *Press* ENTER
Select rails: *Press* ENTER

In the Sweep Surface dialog box, specify:

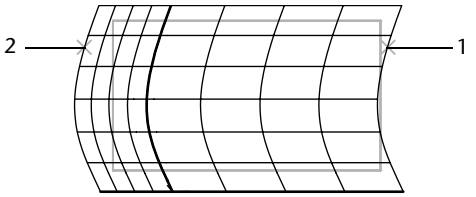
Orientation: Parallel

Choose OK.

4 Use AMJOINSF to join the two surfaces, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Join

Select surfaces to join: *Select the right surface (1)*
Select surfaces to join: *Select the left surface (2) and press* ENTER

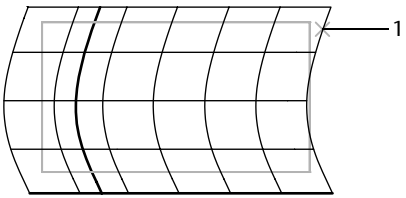


The two surfaces create a single surface. The resulting surface probably does not extend beyond the part on all sides, so you need to lengthen the surface.

5 Use AMLENGTHEN to lengthen the surface, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Lengthen

Base edge=Single Extension=Percent Method=Parabolic Value=110.0000%
Select surface edge or spline [eDge/Extend/Keep/Mode/Value]: *Enter* **v**
Enter percent <110.0000%>: *Enter* **105**
Base edge=Single Extension=Percent Method=Parabolic Value=105.0000%
Select surface edge or spline [eDge/Extend/Keep/Mode/Value]:
Select the rightmost vertical edge of the surface (1)
Base edge=Single Extension=Percent Method=Parabolic Value=105.0000%
Select surface edge or spline [eDge/Extend/Keep/Mode/Value]: *Press* ENTER



The surface now extends past the cube representing the camera body.

Attaching Surfaces Parametrically

Next, you create a work plane and work point and then dimension the work point to the part. This dimension establishes a parametric relationship between the surface and the part. The position of the surface is controlled by the work point, and its orientation is controlled by the work plane associated with the work point. Later, if you modify the position of the work point, the surface location moves accordingly.

To position the work plane and work point easily, work in the isometric view.

To attach a surface to a part

1 Use AMWORKPLN to create a work plane.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Plane.

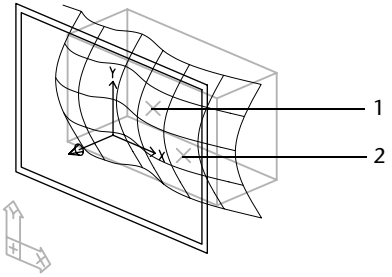
In the Work Plane Feature dialog box, specify:

1st Modifier: Planar Parallel
2nd Modifier: Offset
Offset: *Enter* **1**
Create Sketch Plane: *Select the check box*

Choose OK.

2 Position the offset work plane on the part, responding to the prompts.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the front face of the camera (1)
Enter an option [Next/Accept] <Accept>:
Choose **n** *to cycle to the front face, or press* ENTER
Enter an option [Flip/Accept] <Accept>:
Choose **f** *to flip the direction arrow away from the camera body, or press* ENTER
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>:
Verify that the UCS icon is upright and press ENTER



You have created a parallel work plane offset from the front face of the part.

3 Use AMWORKPT to place a work point on the work plane, responding to the prompts.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Point.

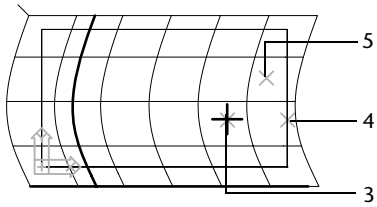
Workpoint will be placed on the current sketch plane.
Specify the location of the workpoint: *Specify a location (2)*

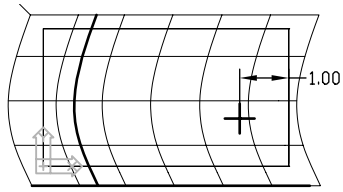
You have created a work point on the sketch plane.

4 Use AMPARDIM to constrain the work point to the camera body, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

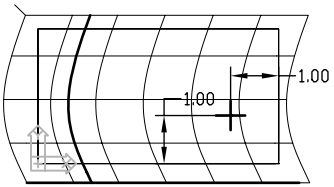
Select first object: *Select the work point (3)*
Select second object or place dimension:
Select the right edge of the camera body (4)
Specify dimension placement: *Place the horizontal dimension (5)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <1.3316>: *Enter* **1**
Solved underconstrained sketch requiring 1 dimensions or constraints.





5 Continue on the command line.

Select first object: *Select the work point*
Select second object or place dimension:
Select the bottom edge of the camera body
Specify dimension placement: *Place the vertical dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.8898>: *Enter* **1**
Solved fully constrained sketch.
Select first object: *Press* ENTER



The work point is fully constrained.

Save your file.

Cutting Parts with Surfaces

Now that the surface is positioned relative to the camera body, you can use it to cut away material from the planar camera face.

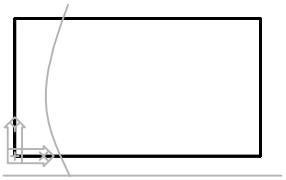
To cut away from a part

1 Use AMSURFCUT to create a surface cut, responding to the prompts.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Surface Cut.

(Type: Cut)
Select surface or [Type]: *Select the surface*
Select work point: *Select the work point*
Specify portion to remove: [Flip/Accept] <Accept>:
Verify the direction arrow points away from the camera body and press ENTER

One side of the part is cut away, leaving the curved face of the surface. Your model shows the modified block and the splines used to create the surface.



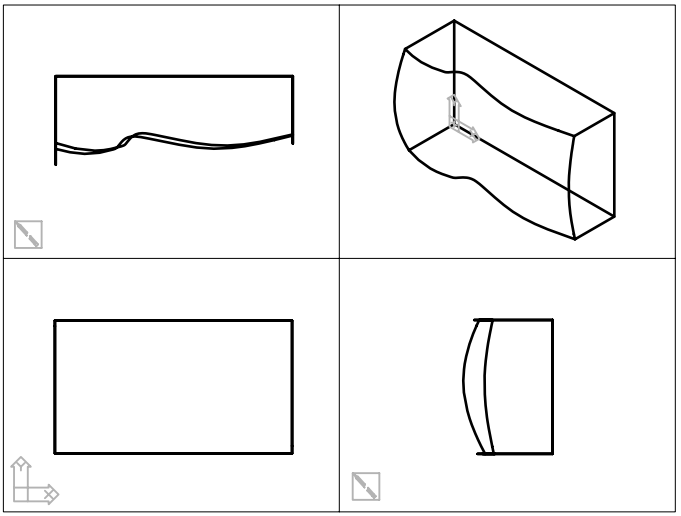
2 Use REGENALL to regenerate the drawing views.

Desktop Menu View ➤ Regen All

3 Remove the three splines used to create the surface.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Erase.

4 Select the three splines and press ENTER.



Save your file.

Creating Extruded Features

The film compartment at the back of the base feature has two features—the compartment and the door.

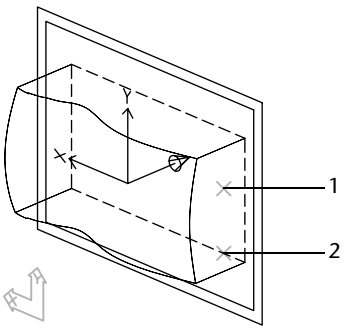
The camera back is a flat plane. You specify it as the sketch plane, sketch the profile, and extrude it directly into the camera body.

To sketch the film compartment

1 Use AMSKPLN to create a new sketch plane, responding to the prompts. Work in the isometric view.

Context Menu In the graphics area, right-click and choose New Sketch Plane.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the back face of the camera (1)
Enter an option [Accept/Next] <Accept>:
Choose **n** *to cycle to the back face, or press* ENTER
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>:
Select the bottom edge of the camera (2)
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>:
Verify that the X axis is pointing to the left and press ENTER



The back of the camera has been specified as the sketch plane. In the isometric view, the UCS icon is displayed with the *X* axis pointing left.

Next, change the view to see the back of the camera. If needed, zoom out to see the entire back of the camera.

2 Change the front view to a back view.

Desktop Menu View ➤ 3D Views ➤ Back

3 Use RECTANG to sketch a rectangle to the left on the camera back.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Rectangle.

4 Use AMPROFILE to create a profile from the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

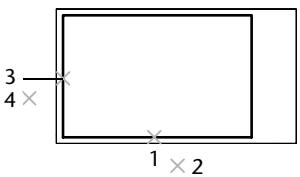
You need to place four dimensions or constraints: two to define the sketch size and two to specify the sketch location on the camera body.

To add dimensions and constraints to the film compartment sketch

1 Use AMPARDIM to dimension the width of the rectangle, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Select the bottom horizontal line of the sketch (1)*
Select second object or place dimension: *Specify a location (2)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <3.7546>: *Enter* **4**
Solved underconstrained sketch requiring 3 dimensions or constraints.



2 Define the height of the rectangle.

Select first object: *Select the left vertical line of the sketch (3)*
Select second object or place dimension: *Specify a location (4)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <2.6011>: *Enter* **2.5**
Solved underconstrained sketch requiring 2 dimensions or constraints.
Select first object: *Press* ENTER

3 Make the isometric view active.

To see the dimensions and the profile sketch more clearly, rotate the isometric view until the back of the camera faces you.

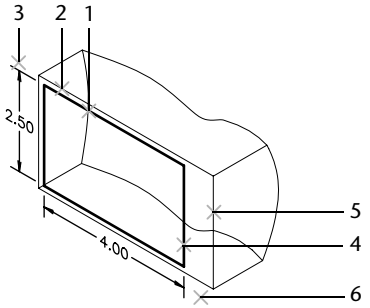
Desktop Menu View ➤ 3D Views ➤ Back Left Isometric

Define the distance between the top of the sketch and the top of the camera back.

4 Use AMPARDIM to constrain the rectangle to the camera body, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Select line (1)*
Select second object or place dimension: *Select line (2)*
Specify dimension placement: *Specify a location (3)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.2355>: *Enter* **.1626**
Solved underconstrained sketch requiring 1 dimensions or constraints.



5 Define the distance between the right side of the sketch and the right edge of the camera back.

Select first object: *Select line (4)*
Select second object or place dimension: *Select line (5)*
Specify dimension placement: *Specify a location (6)*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.8583>: *Enter* **.8426**
Solved fully constrained sketch.
Select first object: *Press* ENTER

Next, cut the film compartment from the camera body.

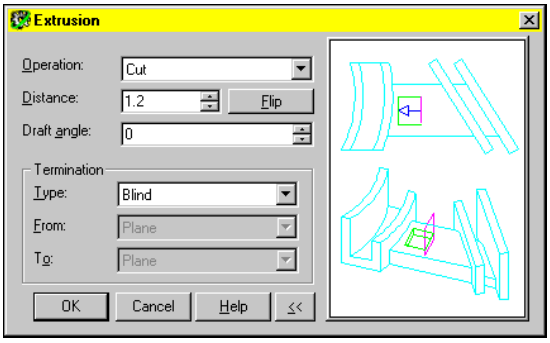
To cut the film compartment

1 Use AMEXTRUDE to cut the film compartment from the camera body.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

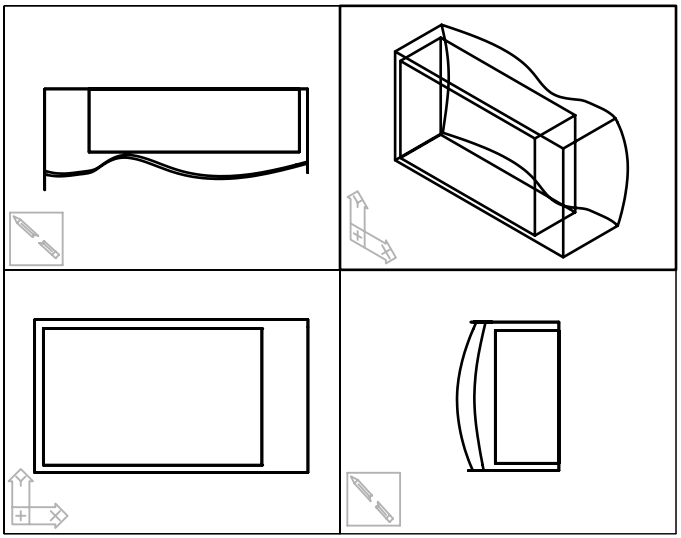
In the Extrusion dialog box, specify:

Operation: Cut
Termination: Blind
Distance: *Enter* **1.2**
Flip: *Point the direction arrow into the camera body*



Choose OK.

The cut-out compartment is displayed in all four views.



2 Activate and then restore the viewports to the original orientation.

Upper right viewport: Front Right Isometric View
Lower left viewport: Front View

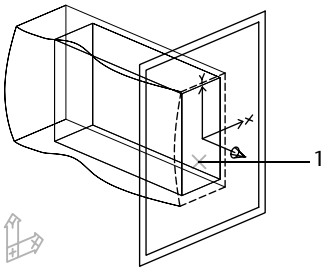
Cutting the door is similar to cutting the film compartment. You sketch a rectangle on the right side of the camera and blindly extrude it as a cut into the camera body.

To sketch the film compartment door

1 Use AMSKPLN to create a new sketch plane, responding to the prompts. Work in the isometric view.

Context Menu In the graphics area, right-click and choose New Sketch Plane.

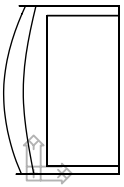
Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the side face of the camera (1)
Enter an option [Accept/Next] <Accept>:
Verify that the side face is highlighted and press ENTER
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>:
Verify that the Z axis points away from the camera and press ENTER



2 Set the UCS origin to the lower-left corner of the right side of the camera, responding to the prompts.

Desktop Menu Assist ➤ New UCS ➤ Origin

Specify new origin point <0,0,0>: *Enter* **end**
of: *Specify a point near the lower-left corner of the side view*



NOTE If the UCS icon does not snap to the lower-left corner of the camera, set the AutoCAD system variable UCSICON to On.

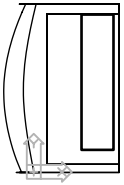
3 In the side view, zoom in on the camera face.

Context Menu In the graphics area, right-click and choose Zoom.

4 Hold down the left mouse button to size the view, and then press ENTER to end the command.

5 Sketch a rectangular shape for the door cutout.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Rectangle.



6 Create the profile sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

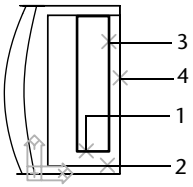
You need four dimensions or constraints to solve the sketch. Add constraints that define the location of the profile on the camera side.

To constrain the film compartment door

1 Use AMADDCON to make the bottom edge of the profile sketch collinear with the bottom line of the film compartment, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Collinear.

Valid selection(s): line or spline segment
Select object to be reoriented: *Select line (1)*
Valid selection(s): line or spline segment
Select object to be made collinear to: *Select line (2)*
Solved underconstrained sketch requiring 3 dimensions or constraints.



2 Make the right side of the profile sketch collinear with the right edge of the camera body.

Valid selection(s): line or spline segment
Select object to be reoriented: *Select line (3)*
Valid selection(s): line or spline segment
Select object to be made collinear to: *Select line (4)*
Solved underconstrained sketch requiring 2 dimensions or constraints.
Valid selection(s): line or spline segment
Select object to be reoriented: *Press* ENTER
Enter an option
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix]
<eXit>: *Press* ENTER

3 Use AMPARDIM to dimension the width and height of the profile sketch, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Select a horizontal profile edge*
Select second object or place dimension: *Place the horizontal dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.6840>: *Enter* **.6**
Solved underconstrained sketch requiring 1 dimensions or constraints.
Select first object: *Select a vertical profile edge*
Select second object or place dimension: *Place the vertical dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <2.3218>: *Enter* **2.5**
Solved fully constrained sketch.
Select first object: *Press* ENTER

The horizontal dimension makes the width of the profile equal to half the depth of the film compartment and the height of the profile equal to the height of the compartment.

For practice, express the width and height of the profile as equations.

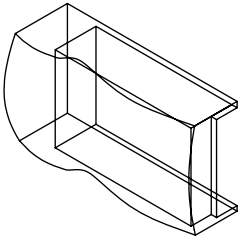
To cut the film compartment door

1 Use AMEXTRUDE to cut the profile from the camera body.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Operation: Cut
Termination: Blind
Distance: *Enter* **.1574**
Flip: *Point the direction arrow into the camera*



2 Choose OK to create the extrusion.

Save your file.

The battery compartment also has a cutout for a door. The order in which you create these features does not matter, but the natural order would be to create the film compartment first.

The cutout for the battery compartment is more complicated because of its shape. The key to creating this feature is to locate the sketch plane properly on the bottom left side of the camera body.

NOTE Watch the UCS icon in the isometric view and make sure it is positioned on the bottom of the camera.

To sketch the battery compartment

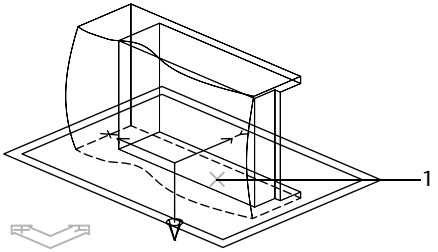
1 Change the top view to a bottom view.

Desktop Menu View ➤ 3D Views ➤ Bottom

2 Use AMSKPLN to create a new sketch plane, responding to the prompts. Work in the isometric view.

Context Menu In the graphics area, right-click and choose New Sketch Plane.

Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Select the bottom face of the camera (1)
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>:
Verify that the Z axis arrow points down, away from the camera body
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>:
Verify that the X axis points to the left and press ENTER



3 Use PLINE to sketch the profile of the battery compartment on the bottom of the camera body. Work in the bottom view.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Polyline.

4 Use AMPROFILE to create the profile sketch.

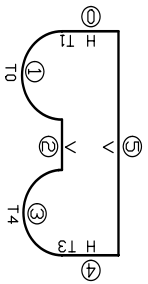
Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

The sketch needs seven to nine dimensions or constraints, depending on how precisely you drew the sketch. If you need more than seven constraints, you need to add some missing geometric constraints.

To constrain the battery compartment

1 Use AMSHOWCON to view the current geometric constraints.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Show Constraints.



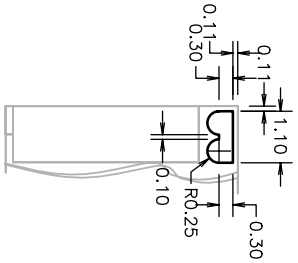
2 Add any missing geometric constraints.

Typically, the radial (R) constraints and one of the tangent (T) constraints are missing from the arcs.

NOTE When you add constraints, the sketch shape might become distorted, but you can restore it when you complete the dimensions. Dimension the largest vertical dimension and the arcs before you dimension smaller objects.

3 Use AMPARDIM to add the following dimensions.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.



The sketch is fully constrained.

To cut the battery compartment

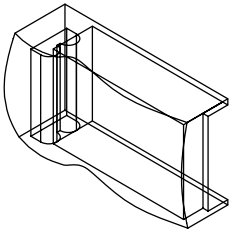
1 Use AMEXTRUDE to cut the profile from the camera body.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Operation: Cut
Termination: Blind
Distance: *Enter* **2.4**
Flip: *Point the direction arrow into the camera body*

Choose OK.



The door opening of the battery compartment is located on the same plane as the battery compartment. Therefore, you need only to sketch and constrain a rectangle, cutting it into the camera body to the proper depth.

To sketch and constrain the battery compartment door

1 Use RECTANG to sketch the profile of the cutout. Work in the bottom view.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Rectangle.

2 Use AMPROFILE to create the profile sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

The sketch requires four dimensions or constraints.

3 Use AMADDCON to constrain the sketch to the bottom of the camera body.

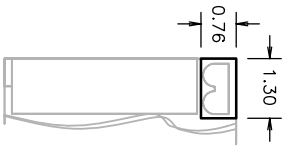
Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Collinear.

Select lines that make the outside edges of the sketch collinear with the outside edges of the camera body.

4 Use AMPARDIM to dimension the length and width of the profile sketch, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.

Select first object: *Select the narrow side of the rectangle*
Select second object or place dimension: *Place the horizontal dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <0.7463>: *Enter* **.76**
Solved underconstrained sketch requiring 1 dimensions or constraints.
Select first object: *Select the long side of the rectangle*
Select second object or place dimension: *Place the vertical dimension*
Enter dimension value or [Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace] <1.1274>: *Enter* **1.3**
Solved fully constrained sketch.
Select first object: *Press* ENTER



The sketch is fully constrained.

To cut the battery compartment door

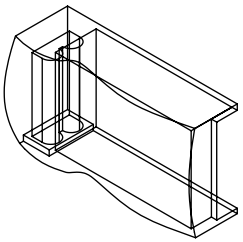
1 Use Extrude to cut the door opening from the camera body. Make sure the direction of the cut is into the camera body.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Operation: Cut
Termination: Blind
Distance: *Enter* **.1574**
Flip: *Point the direction arrow into the camera body*

Choose OK.



Save your file.

Creating Holes

Both the shutter release and the film advance mounts are counterbored holes that you can create as placed features.

To create shutter release and film advance mount holes

1 Change the upper-left viewport to a top view.

Desktop Menu View ➤ 3D Views ➤ Top

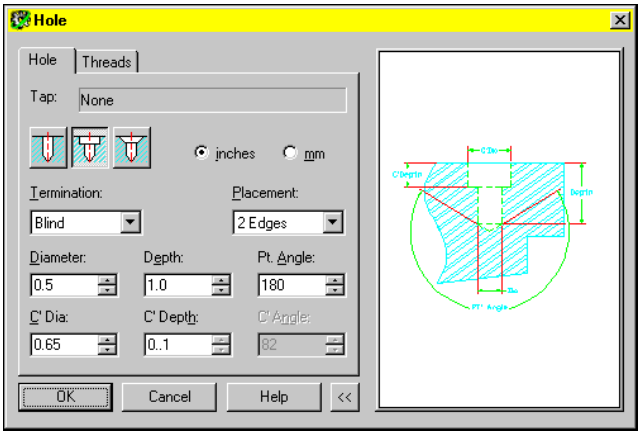
2 Zoom in to enlarge the view as needed, and then activate the isometric view.

3 Use AMHOLE to place the holes for the shutter release and the film advance.

Context Menu In the graphics area, right-click and choose Placed Features ➤ Hole.

In the Hole dialog box, select the Counterbore hole type icon and specify:

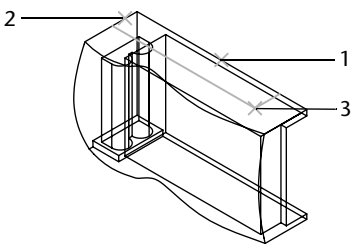
Termination: Blind
Placement: 2 Edges
Dia: *Enter* **.5**
Depth: **1.0**
Pt. Angle: *Enter* **180**
C'Dia: *Enter* **.65**
C'Depth: *Enter* **1**



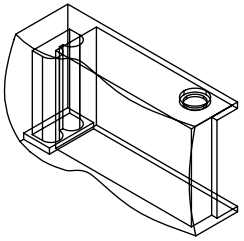
Choose OK.

4 Respond to the prompts as follows:

Select the first edge: *Select the top, back edge in the isometric view (1)*
Select the second edge: *Select the top, left edge in the isometric view (2)*
Specify the hole location: *Specify a location (3)*
Enter the distance from first edge (highlighted) <0.4146>: *Enter* **.5**
Enter the distance from second edge (highlighted) <4.3456>: *Enter* **4.25**
Select the first edge: *Press* ENTER



A hole is created for the film advance component.



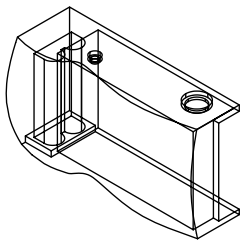
5 Press ENTER to redisplay the Hole Feature dialog box. Specify:

Operation: C'Bore
Termination: Blind
Placement 2 Edges
Drill Size: Custom, *enter* **.2** *diameter,* **1.0** *depth, and* **180** *degrees point angle*
C'bore/Sunk Size: *Enter* **.3** *diameter and* **.1** *depth and choose OK*

6 Respond to the prompts as follows:

Select the first edge: *Select the top, back edge in the isometric view*
Select the second edge: *Select the top, left edge in the isometric view*
Specify the hole location: *Specify a location*
Enter the distance from first edge (highlighted) <0.6946>: *Enter* **.85**
Enter the distance from second edge (highlighted) <1.7487>: *Enter* **1.6**
Select the first edge: *Press* ENTER

A hole is created for the shutter release mount.



Save your file.

Creating Features on a Work Plane

The camera body is complete except for features on the camera face. Unlike the previous features, you sketch these features on a work plane parallel to the front of the camera. You extrude the features from the work plane and into the camera body to the correct depth.

You sketch on the work plane because 2D sketches cannot be drawn and profiled on a NURBS surface.

The lens sheath, a hollow cylinder joined to the face of the camera, has two features: a solid cylinder and a circle used to cut out the center of the cylinder.

To extrude the lens sheath on a work plane

1 Use AMWORKPLN to create a new work plane on which to locate the sketch plane.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Work Plane.

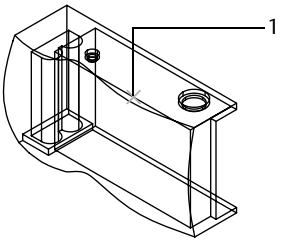
In the Work Plane Feature dialog box, specify:

1st Modifier: Planar Parallel
2nd Modifier: Offset
Offset: *Enter* **1.25**
Create Sketch Plane: *Select the check box*

Choose OK.

2 Respond to the prompts as follows:

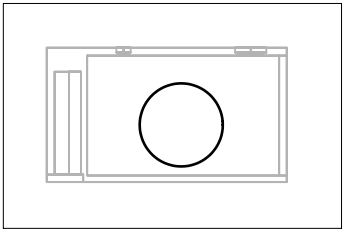
Select work plane, planar face or [worldXy/worldYz/worldZx/Ucs]:
Specify a point (1)
Enter an option [Next/Accept] <Accept>:
Press ENTER *when the front of the camera is selected*
Enter an option [Flip/Accept] <Accept>:
Verify that the work plane is offset from the camera front and press ENTER
Plane = Parametric
Select edge to align X axis or [Flip/Rotate/Origin] <Accept>:
Point the Z axis away from the camera front and press ENTER



The work plane is created in front of the camera face. Because the sketch plane is specified on the work plane, the UCS icon is also displayed on the work plane.

3 Use CIRCLE to sketch a circle in front of the camera face. Work in the front view.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Circle.



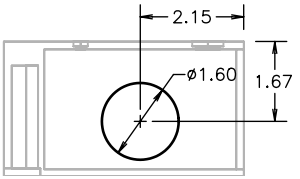
4 Use AMPROFILE to create the profile sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

To position the circle, you need three dimensions or constraints: a diameter and two dimensions to locate the circle on the sketch plane relative to the camera body.

5 Use AMPARDIM to dimension the sketch with the following values.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.



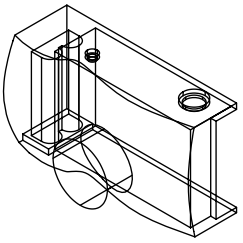
6 Use EXTRUDE to extrude the profile to create the outer cover of the lens sheath. Work in the isometric view.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Operation: Join
Termination: Blind
Distance: *Enter* **1.25**
Flip: *Point the direction arrow into the camera body*

Choose OK.

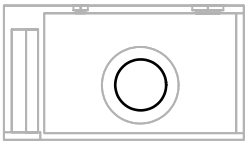


The lens sheath is complete. Now, cut a smaller cylinder to hollow out the sheath.

To hollow out the lens sheath

1 Activate the front view, and sketch a circle on the work plane.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Circle.



2 Profile the sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

Three dimensions or constraints are needed to solve the sketch.

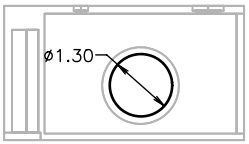
3 Use AMADDCON to constrain the sketch to be concentric with the lens sheath, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Concentric.

Valid selection(s): arc, circle, or ellipse
Select object to be reoriented: *Select the small circle*
Valid selection(s): arc, circle, ellipse, or work point
Select object to be made concentric to: *Select the large circle*
Solved underconstrained sketch requiring 1 dimensions or constraints.
Valid selection(s): arc, circle, or ellipse
Select object to be reoriented: *Press* ENTER
Enter an option
[Hor/Ver/PErp/PAr/Tan/CL/CN/PRoj/Join/XValue/YValue/Radius/Length/Mir/Fix]
<eXit>: *Press* ENTER

4 Use AMPARDIM to dimension the sketch to the value shown.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.



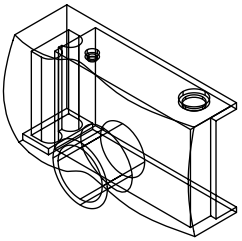
5 Make the isometric view active and use AMEXTRUDE to extrude the sketch to hollow out the lens sheath.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Operation: Cut
Termination: Through
Flip: *Point the direction arrow into the camera body*

Choose OK.



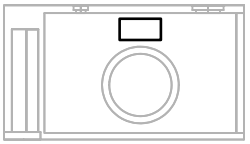
Save your file.

Next, you create the viewfinder compartment, a filleted rectangle that is cut from the camera face.

To cut the viewfinder compartment

1 Use RECTANG to sketch a rectangle on the sketch plane above the lens sheath. Work in the isometric view.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Rectangle.



2 Use FILLET to define the fillet for the corners of the rectangle, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Fillet.

Current settings: Mode = TRIM, Radius = 0.5000
Select first object or [Polyline/Radius/Trim]: *Enter* **r**
Specify fillet radius <0.5000>: *Enter* **.1**, *and choose OK.*

3 Press ENTER to restart FILLET. Apply the fillet, responding to the prompts.

Current settings: Mode = TRIM, Radius = 0.1000
Select first object or [Polyline/Radius/Trim]: *Enter* **p**
Select 2D polyline: *Specify the rectangle*

4 Use AMPROFILE to create the profile sketch.

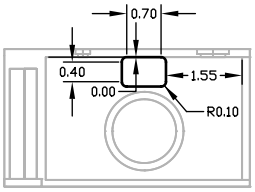
Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

You need five or more dimensions or constraints to solve the sketch. Add the dimensions for the length and width of the shape, one dimension for the fillets, and two dimensions to locate the sketch in relationship to the camera body.

5 In the front view, zoom in to enlarge the model as needed.

6 Use AMPARDIM to add the following dimensions.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.



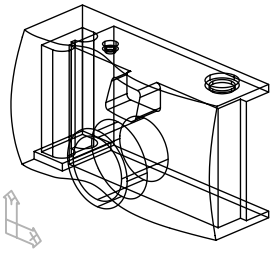
7 Use AMEXTRUDE to cut the sketch through the camera body.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Operation: Cut
Termination: Through
Flip: *Point the direction arrow into the camera body*

Choose OK.

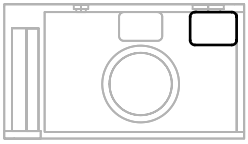


The last feature is the flash compartment. It has a shape similar to the viewfinder but is larger and located in the upper-right corner of the camera face.

To cut the flash compartment

1 Sketch a rectangle to the right of the viewfinder.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Rectangle.



2 Define a fillet for the corners of the rectangle, responding to the prompts.

Context Menu In the graphics area, right-click and choose 2D Sketching ➤ Fillet.

Current settings: Mode = TRIM, Radius = 0.5000
Select first object or [Polyline/Radius/Trim]: *Enter* **r**
Specify fillet radius <0.5000>: *Enter* **.1**

3 Press ENTER to restart FILLET. Apply the fillet, responding to the prompts.

Current settings: Mode = TRIM, Radius = 0.1000
Select first object or [Polyline/Radius/Trim]: *Enter* **p**
Select 2D polyline: *Specify the rectangle*

4 Create the profile sketch.

Context Menu In the graphics area, right-click and choose Sketch Solving ➤ Single Profile.

You need five or more dimensions or constraints to solve the sketch, just as you did when you sketched the viewfinder. Dimension the length, width, and the fillets, and locate the sketch in relationship to the camera body.

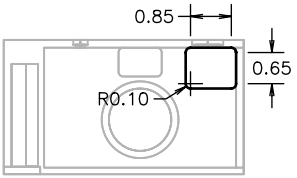
Zoom in on the front view as needed.

5 Use AMADDCON to make the top and right edges of the sketch collinear with the upper-right corner of the film compartment.

Context Menu In the graphics area, right-click and choose 2D Constraints ➤ Collinear.

6 Add the following dimensions.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ New Dimension.



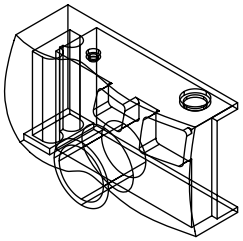
7 Extrude the sketch to cut it through the camera body.

Context Menu In the graphics area, right-click and choose Sketched & Work Features ➤ Extrude.

In the Extrusion dialog box, specify:

Operation: Cut
Termination: Through
Flip: *Point the direction arrow into the camera body*

Choose OK.



Save your file.

Modifying Designs

As with all projects, designs change during the development process. For example, you might want to scale the camera to a smaller size and change the dimension that positions the camera face on the solid model. Because you want both the surface and the camera body at the same scale, you first resize them.

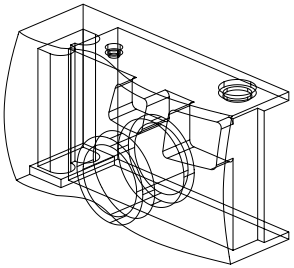
In this exercise, you specify a percentage of the camera's current size. Then, to position the surface on the camera proportionately, you modify the parametric dimension.

To scale the camera body and face

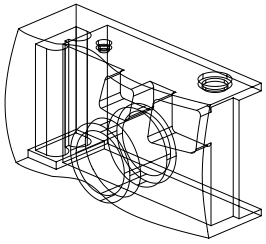
1. Zoom in to magnify the isometric view.
2. Use SCALE to reduce the scale of the part, responding to the prompts.

Desktop Menu Modify ➤ Scale

Select objects: *Select the camera and press* ENTER
Specify base point: *Select the rear corner of the camera*
Specify scale factor or [Reference] <1.0000>: *Enter* ***.9*** *and press* ENTER



full-scale camera body



scaled camera body

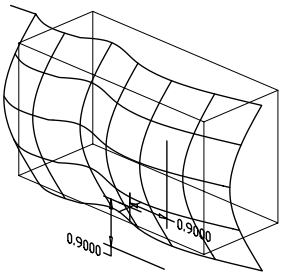
The camera and surface are resized to 90 percent of their original size. The surface and other features retain their original geometric relationships.

To reposition the camera face

1 Use AMEDITFEAT to edit the surfcut feature, responding to the prompts.

Context Menu In the graphics area, right-click and choose Edit Features ➤ Edit.

Enter an option [Sketch/surfCut/Toolbody/select Feature] <select Feature>:
Enter **c**
Select surfcut feature: *Select a curved edge of the surface*



The original sketch, work plane, and work point are displayed.

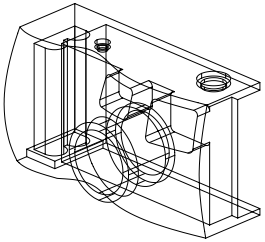
2 Use AMMODDIM to modify the dimension that positions the surface to the work point, responding to the prompts.

Context Menu In the graphics area, right-click and choose Dimensioning ➤ Edit Dimension.

Select dimension to change: *Select the horizontal dimension*
New value for dimension <0.9>: *Enter* **.75**
Select dimension to change: *Press* ENTER

3 Use AMUPDATE to update the part.

Context Menu In the graphics area, right-click and choose Update Part.



The surface is repositioned on the camera with the new value.

Finishing Touches on Models

The finishing touch for the camera body is to fillet the corners where the different sides meet.

To finish the camera body

1 Use AMVISIBLE to hide the work plane from your display.

Desktop Menu Part ➤ Part Visibility

2 In the Desktop Visibility dialog box, select the Part tab and choose Work Planes and Hide. Choose OK.

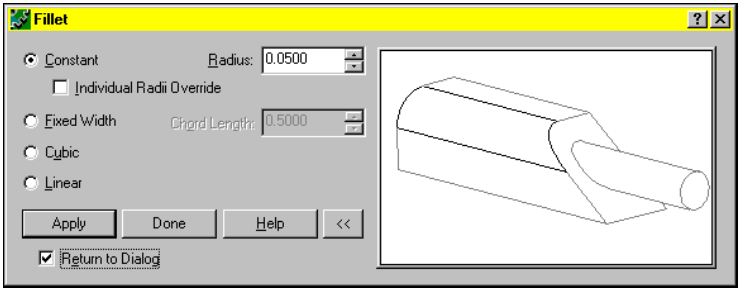
3 Use ISOLINES to increase the number of isolines. Change the value to **8** to show more detail on the model. The display will change when you edit your model.

4 Use AMFILLET to fillet the camera body.

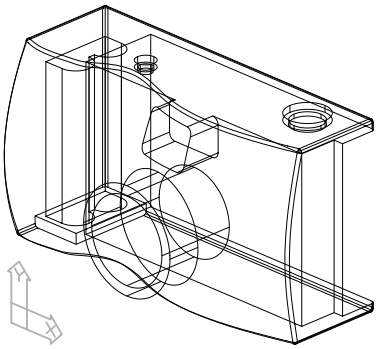
Context Menu In the graphics area, right-click and choose Placed Features ➤ Fillet.

5 In the Fillet dialog box, specify Constant using a fillet radius of **.05**. Select Return to Dialog and choose Apply.

NOTE To speed up filleting a complex model, select only a few lines at a time. Repeat the command with more lines until the filleting is finished.

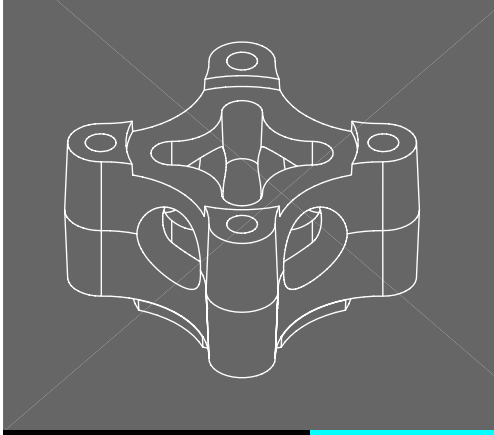


6 Fillet the outside corners and edges of the camera body. When you are finished, choose Done.



The camera body is finished.

Surfacing Wireframe Models



In This Chapter

21

- Studying the design intent and developing a strategy
- Identifying logical surface areas
- Identifying base surface areas
- Using creative techniques to surface a wireframe model
- Verifying surfacing results

This Autodesk® Mechanical Desktop® tutorial introduces wireframe surface modeling, one of the key uses for surface modeling. You learn how to develop a strategy for a surfacing project, and how to achieve the design intent. The tutorial provides instructions for surfacing a wireframe model of a pump housing. The wireframe is the outline of a pump housing, and the only data you have to work with.

You should already know how to create surfaces before you begin this tutorial. If you do not, complete the exercises in chapter 19, "Creating and Editing Surfaces."

Key Terms

Term	Definition
base surface	A basic underlying surface that carries a shape across a larger area. Can be trimmed to precise shapes as needed, but the base surface remains intact and may be displayed.
logical surface area	An area that can be described by a single surface.
projected wire	A 2D line that represents an opening on a surface and trims a hole in the surface. Can also be a 3D polyline that represents the extents of the opening in the wireframe.
watertight	Surfaces conform to the wireframe model; gaps between surfaces are within allowable tolerances.

Basic Concepts of Surfacing Wireframe Models

A completely surfaced model is a single electronic master suitable for engineering and manufacturing activities, such as:

- Generating accurate sections for engineering and packaging studies.
- Providing input for finite element modeling and analysis.
- Producing shaded renderings for marketing.
- Providing input for rapid prototyping equipment.
- Supplying rotated surfaces for tool, mold, and die design.
- Supplying surfaces for numerical control machining of models and tools.

In this tutorial, building on your knowledge of surface types, you examine the wireframe to be surfaced and determine which surface will produce the best results.

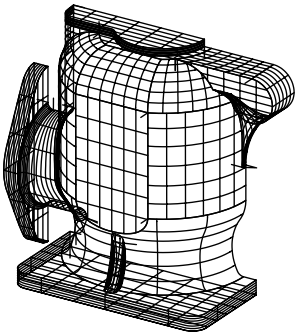
Discerning Design Intent

Because wireframe models have complex shapes, they are usually made up of many surfaces joined together. For a model like the pump housing, you will use many different individual surfaces to completely define it. When you surface a wireframe model, you complete its design.

Before you begin, analyze the design and then plan how you can achieve your design goals. This process of planning before you begin modeling can help you avoid errors. For example, you usually follow these steps for wireframe modeling:

- Study the data to understand the design intent.
- Identify the location and extent of each surface area.
- Identify the base surface area(s) that can be trimmed later to adjacent surfaces and wires.
- Determine where you can use trimmed planar (flat) surfaces.
- Decide on the best surface types and approach for combining them.
- Create additional geometry as needed to resolve problem areas.
- Verify your surfacing results.
- Add the finishing touches to a watertight model.

Review the wireframe in detail, to determine where you will have design challenges.



Consider the following:

- The complexity of the surfaces you need to create. For example, what curvature is required of surfaces? Is it sufficient to have surfaces with no curvature (such as ruled surfaces), or do you need surfaces with multiple curvatures?
- How you can simplify shapes. Surfaces created from polylines or splines with a large number of points are complex and greatly increase computation time.
- Which surfaces are continuous. Continuous surfaces are smoother and take less time to compute. You can set preferences so that lines with breaks or changes in curvature aren't converted to splines.
- Are default preference settings appropriate for the model. Allow as much tolerance as is practical to avoid converting polylines to splines. Splines take longer to compute than polylines—a factor that becomes more important with complex models.

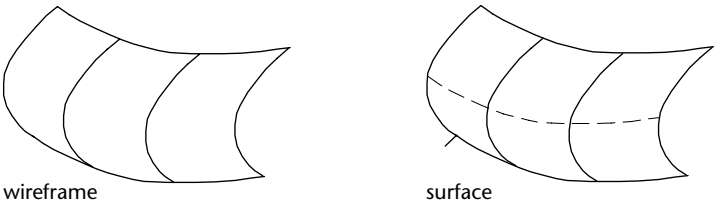
Identifying Logical Surface Areas

Once you determine the intent of the pump design, you get an idea of the requirements for creating its shape and for constructing it. Identify the location and extent of each logical surface area—an area that can be described by a single surface.

A surface must be smooth and free of sharp breaks. Often, an individual surface area is clearly-defined because it is surrounded by sharp break lines on all sides. The pump top is a surface because it is surrounded by sharp edges on three sides. The fourth edge is the end of the part.

Likewise, the side of the top part of the pump constitutes a single surface. Each of these two surface areas requires a surface because no single surface could cover both.

Surfaces can contain multiple wires.

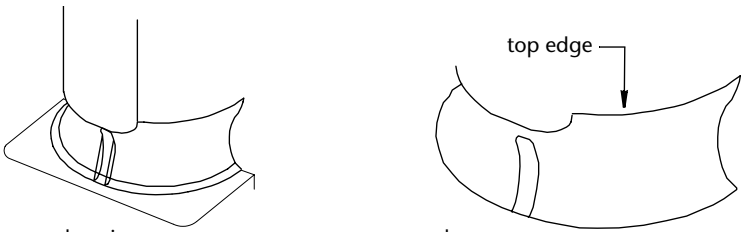


wireframe surface

All lines inside the four boundaries share the same smooth curvature as the boundary edges. There are no abrupt curvature changes, so the goal is to surface the entire area with a single surface, using the additional wires to constrain the surface shape.

Identifying Base Surface Areas

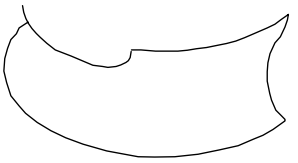
The bottom of the pump housing appears to be a smoothly-curved area. However, at the top edge, there is an almost 90-degree bend. Also, a flanged area is formed by a slot opening in the otherwise smooth area.



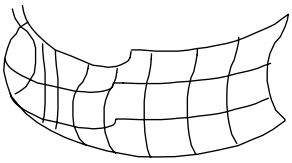
pump housing close-up

The primary guideline in wireframe surfacing is to create an acceptable surface first. Later, add a hole by trimming the surface with the shape of the hole.

In general, use only smooth wires to create surfaces. When you use a wire with sharp corners, those sharp areas do not produce an acceptable surface.

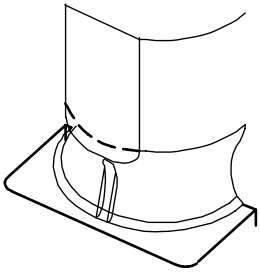


wireframe

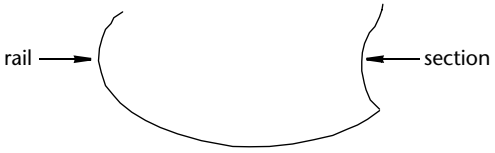


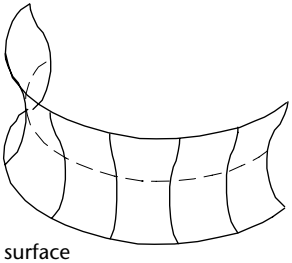
unacceptable surface

You need to find some other way to surface the area. Consider the design intent again. A second look at the area reveals a flat surface on the front of the pump housing that intersects a smoothly curved surface at the bottom.



How do you know that the front surface is flat? One way to check is to look at the top line in another view. The approach to surfacing that area is to create the smooth bottom surface and the flat surface. Then intersect one with the other and create a wire at their intersection. If the new wire is the same as the existing wire, you confirm your observation and surfacing approach.

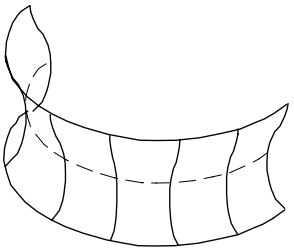




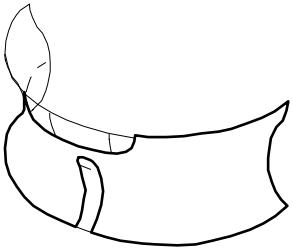
surface

Next, consider the bottom surface. You already know that you cannot use the top wire because it has an abrupt corner. A good approach is to use only the bottom wire as a rail, and the far edge as a cross section.

A surface like this one is a basic surface that carries a shape across larger areas. This surface is referred to as a *base surface*. Even after many areas of the surface are trimmed away, the underlying base surface remains intact and may be displayed at any time with the Surface Display dialog box.



base surface



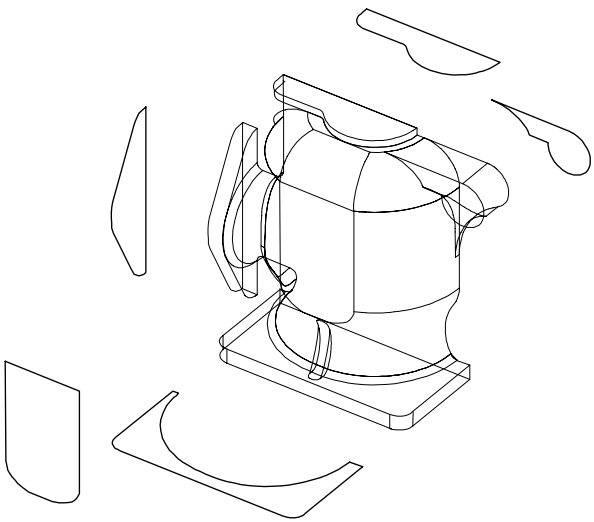
trimmed surface

Identifying base surfaces is an important part of wireframe surfacing. Another approach is to categorize surfaces by type and eliminate those you won't need.

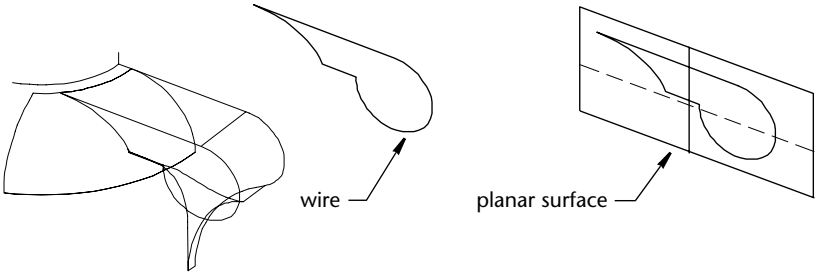
To get shapes you can use for creating surfaces, you may need to break polylines into segments. Then combine selected segments to form boundaries for individual surfaces.

Using Trimmed Planar Surfaces

Use a trimmed planar surface for an area that you know is a flat plane. By glancing at the pump model, you can see areas that appear to be flat and can be surfaced with trimmed flat planes.



If you are in doubt about whether a given area is flat, try to make a planar surface. A planar surface requires a single closed wire as its boundary.



If the wire is *not* a closed single loop, you can see the breaks in the wire when you select it.

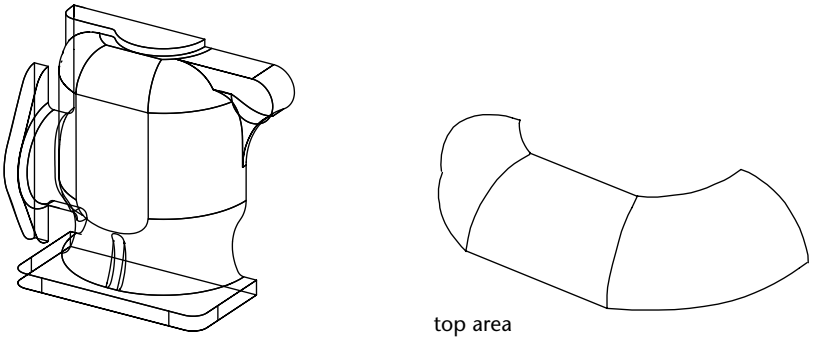
You can join line segments into a closed wire that forms the boundary of a planar surface. The surface is trimmed to the boundary shape.

NOTE When joining line segments, set tolerances to compensate for imperfect wireframe data that would otherwise cause the surface to fail.

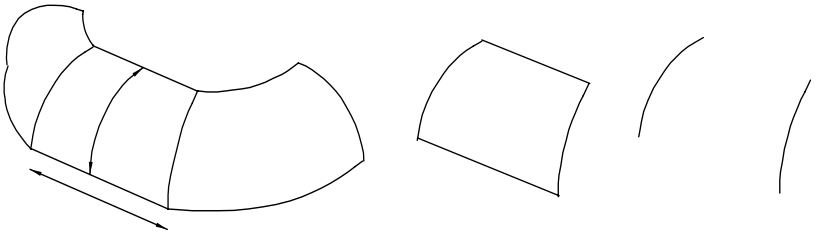
Determine if an area is meant to be flat. If it is flat within the tolerance, create a flat surface and adjust the edges.

Choosing a Surfacing Method

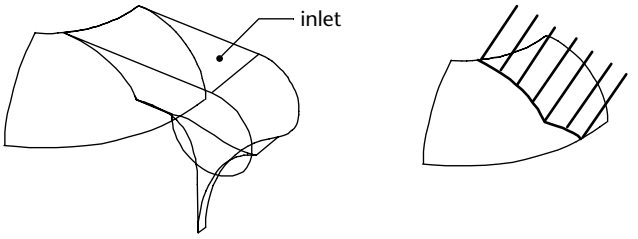
Which type of surface is best suited for a given area may not be clear.



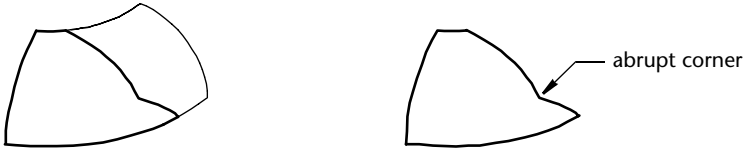
In this example, the top area of the pump is not suitable for a single surface because there are abrupt changes in its smoothness. The center area is curved in one direction but straight in the other. When you have a surface area that can be defined by a straight line between two curves, you can create a ruled surface between the two curves.



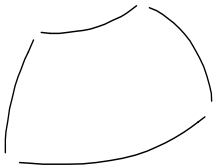
Look beyond the obvious visible surface to find a workable solution. Because the inlet at the right top area of the pump extends from the surface, consider making the base surface first and then trimming it to the correct shape.



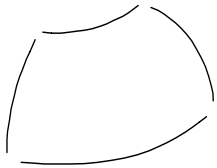
With the inlet shape removed, you can see possibilities for surfaces. The shape created by the four wires contains a sharp corner. Avoid creating a surface from these four wires because they might produce a surface that is not smooth.



You can see that each end of the area beneath the inlet is described by lines with curvatures in both directions. This offers you a choice of surfacing methods, such as a swept surface or a lofted UV surface.



swept surface

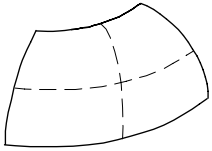


lofted UV surface

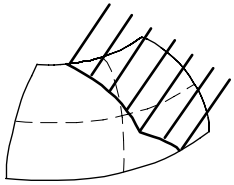
In most cases, there is more than one way to surface an area. Try both methods here, compare the results, and choose the one that produces the best result.

- Swept surfaces give you more control over the shape of the mid portion of the surface.
- Lofted UV surfaces have fewer controls but risk is minimized.

Once you create a base surface to cover an area, trim the surface back to the wire with the abrupt edge.



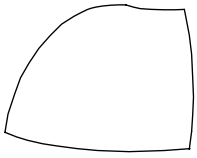
base surface

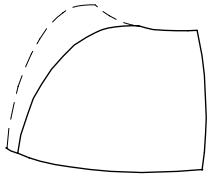


trimmed surface

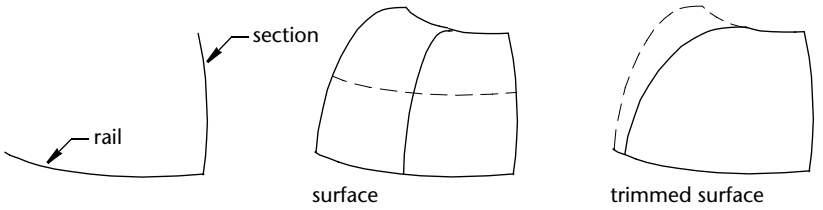
Use trimmed surfaces to create smooth underlying base surfaces that remain a permanent part of a surface definition. Trim to constrain the edges and you achieve smoothness in a base surface that contains no abrupt corners in its boundary wires, yet creates a logical surface bounded by different edges. The logical surface can contain any number of sharp corners, which have no effect on the smoothness of the base surface.

The opposite side of the pump top area may also be surfaced several ways. Again, the surface is really a larger surface cut short by an intersecting area of the pump. This time, the wire that terminates the surface has no abrupt corners, so it could be used as one of four sides of a swept or lofted UV surface.

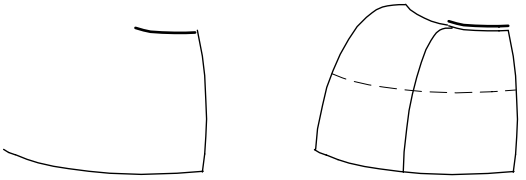




The easiest method is to use a single rail and a single section to surface the entire area, then trim the base surface to the intersecting part of the pump.



This choice might not always be correct. As you gain experience, you can predict which approach yields the most accurate results. In the previous example, verify that the surface created without the top line matches the top line within a reasonable tolerance.



Always check the fit between a newly created surface and existing wires to be sure that you are not deviating too far from the wireframe data. If the new surface is not within tolerance to the existing top line, the surface does not accurately reflect the wireframe. You can re-create it using all four wire edges.

Verifying Surfacing Results

As you gain experience, you will see the importance of learned skill and judgment. Your challenge is to produce smooth surfaces that fit the wireframe closely; the surfaces should be as simple as possible.

You can judge the smoothness of surfaces several ways:

- Analyze a surface and view its color-shaded display to detect small deviations in surface smoothness.
- Create and review flow lines in different rotated views.
- Cut sections through a complete set of surfaces, and then examine the ends to see how closely they match at the edges.
- On the surface create augmented flow lines with long vectors and examine the smoothness of the vector ends. The ends of the vectors exaggerate the smoothness of the surface; areas where it is not smooth become apparent.

Surfacing Wireframe Models

Now that you have analyzed approaches to surfacing the pump housing and practiced surfacing techniques, you are ready to surface the pump.

A surface modeling project may begin with a wireframe, whether it is a DXF or an IGES file from a client, or a 2D or 3D CAD design you created yourself. In order to describe the 3D object, most designers begin with a 2D drawing.

In this lesson, you create surfaces for an actual part, a wireframe model of a hydraulic pump. The surfaced model provides the manufacturer with information to create prototypes or to NC-machine patterns, molds, and tooling.

To set up a drawing file

1 Open the file *t_pump.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

---

2 Use AMOPTIONS to set the surfacing options.

Desktop Menu Surface ➤ Surface Options

In the Mechanical Options dialog box, select the Surfaces tab. In Surface Properties specify:

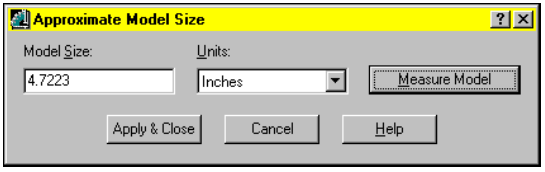
U Display Wires: *Enter* **5**
V Display Wires: *Enter* **5**

In Surface/Spline Options, specify:

Polyline Fit Length: *Enter* **1**
Polyline Fit Angle: *Enter* **150**

Choose Model Size.

3 In the Approximate Model Size dialog box, choose Measure Model.

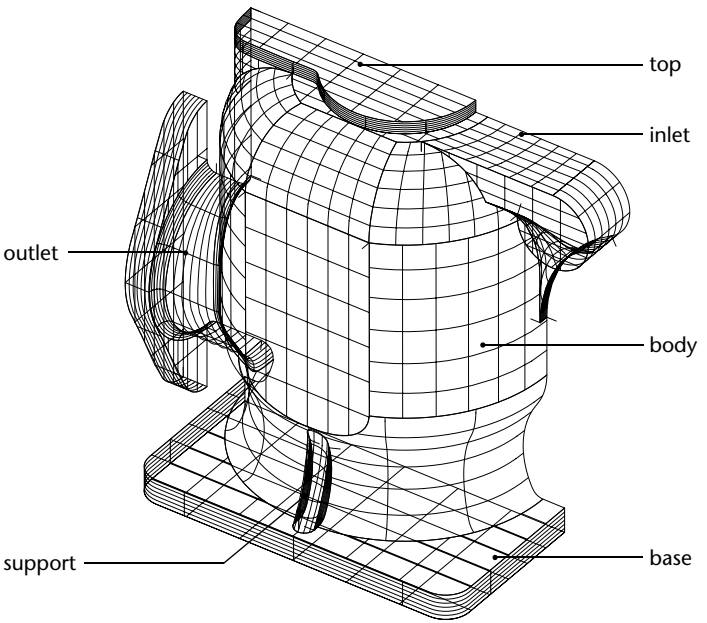


The Model Size is 4.7223. Choose Apply & Close.

Notice that the values have changed in the Mechanical Options dialog box. These settings affect the visual representation of the surfaces and the size of the surface normal. Choose OK to exit.

NOTE If you shade the surfaces you create to better view them, adjust the AutoCAD® setting that controls back faces. Go to Assist ➤ Options and select the System tab. Choose Properties and clear the check box beside Discard back faces. Choose Apply & Close, then OK.

The labeled parts of the pump are on separate layers. As you work on a part, you make its layer current and freeze the other layers to make them inactive.



4 Use LAYER to set up your current layer.

Desktop Menu Assist ➤ Format ➤ Layer

5 In the Layer Properties Manager dialog box, highlight layer 10 and click Make Current.

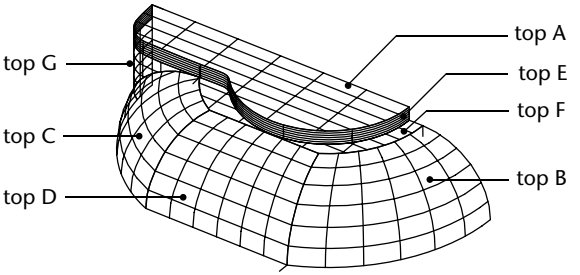
6 Select layer 20, then press SHIFT and select the last layer. In the second column, click the sun icon under Freeze to freeze all selected layers. Then freeze layer 0.

All layers except Layer 10 should be frozen (snowflake icon). Only Layer 10 is thawed (sun icon). Choose OK.

Creating Trimmed Planar Surfaces

Begin by surfacing the top section of the pump model, creating the individual surfaces. Top A is a planar surface because it is flat with sharp edges. Tops B and C are swept surfaces, bounded by curved wires. Top B uses two curves and two rails, and top C uses one curve and one rail. You trim the top C surface where it extends beyond the wireframe boundary.

As you gain experience using the menu selections that correspond to commands, you may want to use shortcuts. For a list of shortcuts that automate the selection of menu options and commands, see "Accelerator Keys" in the online Command Reference.



NOTE For a trimmed planar surface, the surface must be a single polyline that lies in a single plane. If the wireframe is composed of multiple polylines, join them into a single polyline before you create the surface.

To create a trimmed planar surface

1 Use ZOOM to enlarge the view, responding to the prompt.

Context Menu In the graphics area, right-click and choose Zoom.

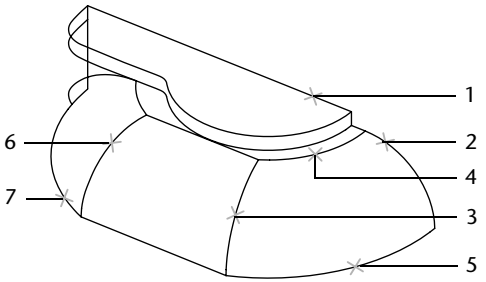
[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>:
Press ENTER

2 Use AMPLANE to create the top A surface, responding to the prompt.

Desktop Menu Surface ➤ Create Surface ➤ Planar Trim

If you use the command line method, enter **w** at the prompt before continuing to the following prompt.

Select wires: *Select wire (1) and press* ENTER



A planar surface, trimmed to the boundary of wire (1), is created on the top of the model.

To sweep a surface on two wires and two rails

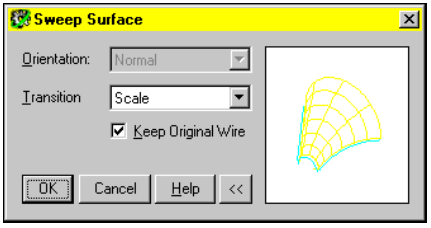
1 Use AMSWEEPSF to create the top B surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *Select wire (2)*
Select cross sections: *Select wire (3) and press* ENTER
Select rails: *Select wire (4)*
Select rails: *Select wire (5)*

In the Sweep Surface dialog box, specify:

Transition: Scale
Keep Original Wire: Check the check box



Choose OK.

To sweep a surface on one wire and one rail

1 Use AMSWEEPSF to create the top C surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *Select wire (6) and press* ENTER
Select rails: *Select wire (7) and press* ENTER

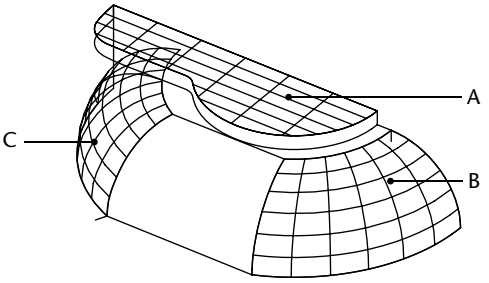
In the Sweep Surface dialog box, in Orientation select Normal. Choose OK.

The surface extends beyond the far side of the top. You will trim it later.

2 Move surfaces A, B, and C to the TOP layer, responding to the prompts.

Command CHPROP

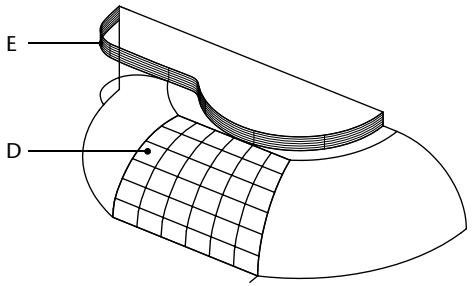
Select objects: *Select surfaces (A), (B), and (C) and press* ENTER
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness]:
Enter **La**
Enter new layer name <10>: *Enter* **top**
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness]:
Press ENTER



The surfaces are now on the TOP layer. Because the TOP layer is frozen, you cannot see the surfaces, although the wireframe is still visible.

Save the file.

The D and E surfaces are ruled surfaces.

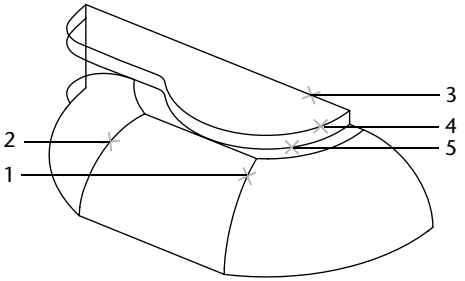


To create a ruled surface between wires

1 Use AMRULE to create the top D surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select wire (1)*
Select second wire: *Select wire (2)*



2 Use BREAK to separate line segment (4) from polyline (3), responding to the prompts.

Desktop Menu Modify ➤ Break

Select object: *Enter* **end**
of: *Select polyline (3)*
Specify second break point or [First point]: *Enter* **@**

Unless you enter **@**, the adjoining portion of the polyline is deleted. The **@** symbol breaks the polyline at the specified location and retains both segments.

3 Use AMRULE to create the top E surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select wire (4)*
Select second wire: *Select wire (5)*

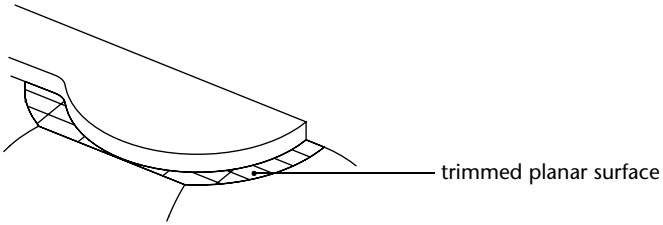
4 Move the top D and top E surfaces to the TOP layer, responding to the prompts.

Command CHPROP

Select objects: *Select surface (D)*
Select objects: *Select surface (E) and press* ENTER
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness]:
Enter **La**
Enter new layer name <10>: *Enter* **top**
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness]:
Press ENTER

Save the file.

Next, break and join lines that are needed to create the top F planar surface.

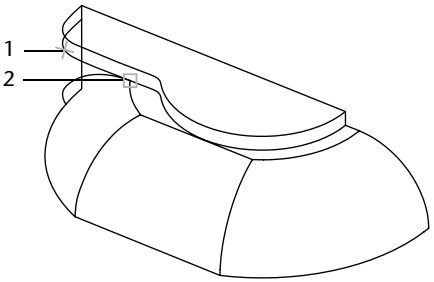


To create a planar surface with a joined polyline boundary

1 Use BREAK to break polyline (1) where it intersects polyline (2), responding to the prompts.

Desktop Menu Modify ➤ Break

Select object: *Select polyline (1)*
Specify second break point or [First point]: *Enter* **f**
Specify first break point: *Enter* **int**
of: *Select polyline (2) at the intersection*
Specify second break point: *Enter* **@**



To check that the polyline has broken correctly, select it. Grip points should appear only for the line segment you select. Press ESC to exit Grip mode.

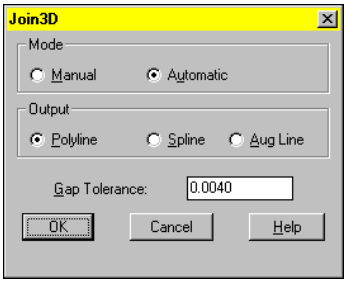
The break creates a line segment you use as part of the boundary for the next surface.

2 Use AMJOIN3D to join the polylines that form the boundary of top F.

Desktop Menu Surface ➤ Edit Wireframe ➤ Join

3 In the Join3D dialog box, specify:

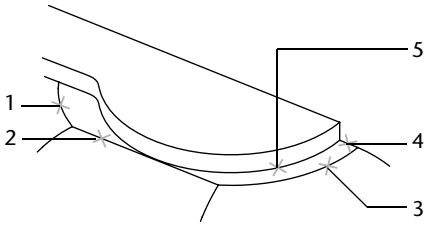
Mode: Automatic
Output: Polyline
Gap Tolerance: *Enter* **.004**



Choose OK.

4 Respond to the prompts as follows:

Select start wire or: *Select polyline (1)*
Select wires to join: *Select polylines (2) through (5)*
Select wires to join: *Press* ENTER
Reverse direction? [Yes/No] <No>:
Press ENTER *to accept the direction of the new wire*



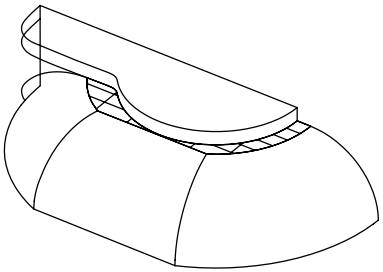
To confirm that the segments are joined, select the polyline and check the grip points.

5 Use AMPLANE to create the top F surface from the joined polyline, responding to the prompt.

Desktop Menu Surface ➤ Create Surface ➤ Planar Trim

If you use the command line method, enter **w** at the prompt before continuing to the following prompt.

Select wires: *Select the joined polyline and press* ENTER



You have created the trimmed planar surface. Save the file.

For the top G surface, extrude a polyline along a straight line, and then trim the surface to the desired shape.

To create a surface with an extruded polyline

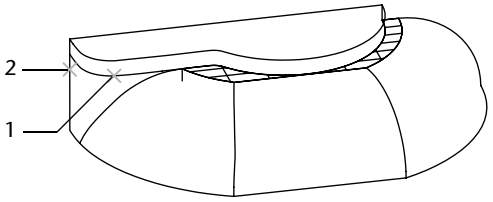
1 Change to a front left isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Left Isometric

2 Us AMEXTRUDESF to extrude the polyline (1) down the wire (2), responding to the prompts.

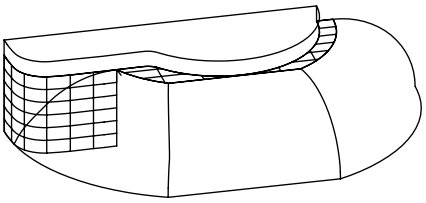
Desktop Menu Surface ➤ Create Surface ➤ Extrude

Select wires to extrude: *Select polyline (1) and press* ENTER
Define direction and length.
Specify start point or [Viewdir/Wire/X/Y/Z]: *Enter* **w**
Select wire to define direction: *Select polyline (2)*
Enter an option [Accept/Flip] <Accept>:
Enter **f** *to flip the direction arrow down, or press* ENTER
Enter taper angle <0>: *Press* ENTER *to accept the default*



Your selection point determines the extrusion direction. Select a point on polyline (2) close to polyline (1). If you select a point beyond the midpoint of polyline (2), the direction of the extrusion is reversed.

Your drawing should look like this.



3 Move top F and top G to the TOP layer, responding to the prompts.

Command CHPROP

Select objects: *Select surface (F)*
Select objects: *Select surface (G) and press* ENTER
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness]:
Enter **La**
Enter new layer name <10>: *Enter* **top**
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness]:
Press ENTER

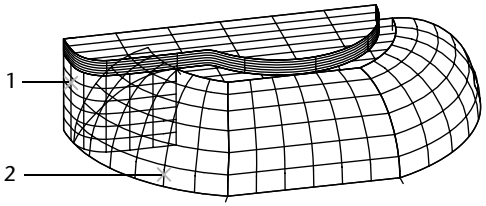
4 Choose Assist ➤ Format ➤ Layer.

Thaw the TOP layer to see all the surfaces you have created.

5 Use AMINTERSF to trim top G and top C at their intersection, responding to the prompts.

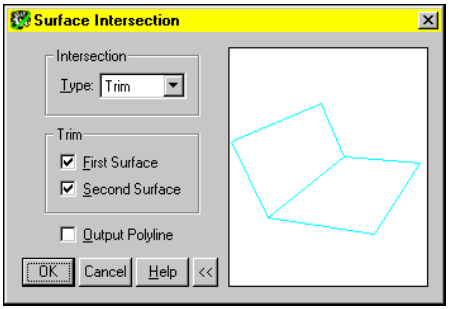
Desktop Menu Surface ➤ Edit Surface ➤ Intersect Trim

Select first surface/quilt or wire: *Select surface (1)*
Select second surface: *Select surface (2)*

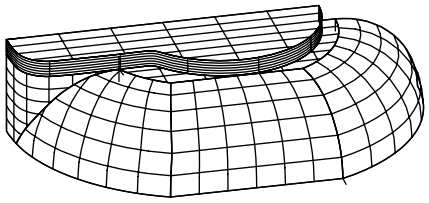


6 In the Surface Intersection dialog box, specify:

Type: Trim
Trim: *Select both* First Surface *and* Second Surface
Clear the checkbox for Output Polyline



Choose OK. The surfaces are trimmed where they intersect.

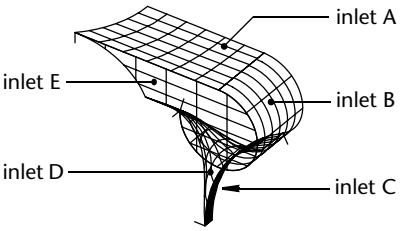


Save the file.

Joining Surfaces on Complex Shapes

Next, you surface the inlet portion of the pump. Because the inlet has a complex shape, you will need five surfaces to represent its shape.

- Inlets A and C are ruled surfaces because they follow two polylines.
- Inlet B is an extruded surface that is trimmed to its final shape.
- Inlet D is surface blended to surfaces B, C, and E.
- Inlet E is a trimmed planar surface created from joined lines that form its boundary.



NOTE Be sure to select surfaces and lines where indicated on the illustrations. To select precisely, zoom in as needed.

To create the inlet A ruled surface

1. From the Desktop menu, choose Assist ➤ Format ➤ Layer.

   In the Layer Properties Manager dialog box, thaw layer 20 and make it current. Then freeze layer 10 and TOP.

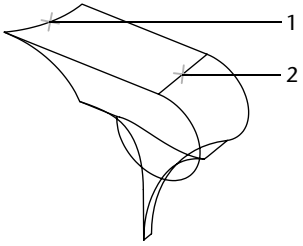
2. Change to a right isometric view.

   Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

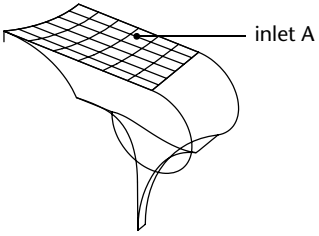
3. Use AMRULE to create the inlet A surface, responding to the prompts.

   Desktop Menu Surface ➤ Create Surface ➤ Rule

   Select first wire: *Select wire (1)*
   Select second wire: *Select wire (2)*



The ruled surface is created on the top of the inlet.



4 Move the inlet A surface to the INLET layer, responding to the prompts.

Command CHPROP

Select objects: *Select inlet A surface and press* ENTER
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness]:
Enter **La**
Enter new layer name <20>: *Enter* **inlet**
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness]:
Press ENTER

Inlet B is an extruded partial cylinder, trimmed to its final shape by a closed wire. The surface is extruded across the inlet wireframe.

The direction of the extrusion is determined by where you select the wire. You can flip the direction of the extrusion.

Next, you create a ruled surface for inlet B.

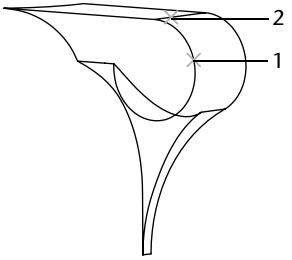
NOTE To select the wires, you might need to reorient the view. Use icons on the Mechanical View toolbar, or options from the View ➤ 3D Views menu.

To create the inlet B extruded surface

1 Use AMEXTRUDESF to extrude polyline (1) along line (2), responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Extrude

Select wires to extrude: *Select polyline (1) and press* ENTER
Define direction and length.
Specify start point or [Viewdir/Wire/X/Y/Z] *Enter* **w**
Select wire to define direction: *Select polyline (2) near (1)*
Enter an option [Accept/Flip] <Accept>:
Enter **f** *to flip direction arrow into part or press* ENTER
Enter taper angle <0>: *Press* ENTER

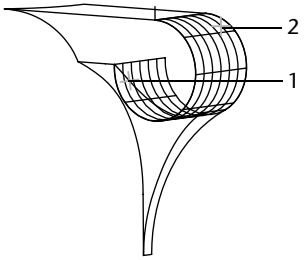


A close look at the inlet reveals that the extruded surface extends beyond the wireframe. You trim the inlet B surface to the boundary of surface D.

2 Use AMPROJECT to project the edge of inlet D to trim the inlet A surface, responding to the prompts.

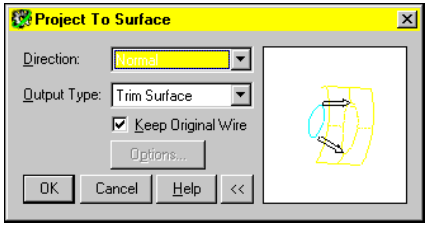
Desktop Menu Surface ➤ Edit Surface ➤ Project Trim

Select wires to project: *Select line (1) and press* ENTER
Select target surfaces/quilts: *Select surface (2) and press* ENTER



3 In the Project to Surface dialog box, specify:

Direction: Normal
Output type: Trim surface
Keep Original Wire: *Check the check box*



Choose OK.

You have trimmed inlet B by projection, keeping the wire you used to trim the inlet surface.

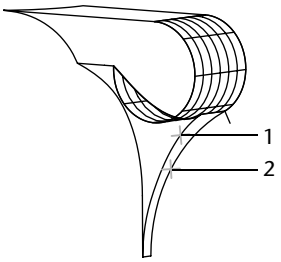
Next you create inlet C, a ruled surface between two wires.

To create the inlet C ruled surface

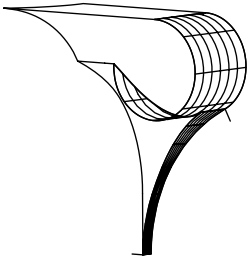
1 Use AMRULE to create the inlet C surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select wire (1)*
Select second wire: *Select wire (2)*



Your model should look like this.



2 Use CHPROP to move inlet B and C surfaces to the INLET layer.

Save the file.

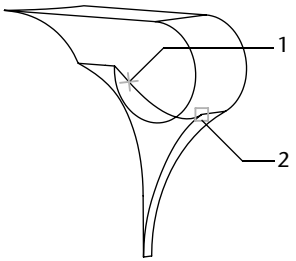
Next, you create inlet D, a surface blended from the edges of inlet B and C surfaces and the polyline that defines the edge of inlet E. You may need to rotate the model to show the intersection clearly.

To create the inlet D blended surface

1 Use BREAK to break the polyline into two line segments, responding to the prompts.

Desktop Menu Modify ➤ Break

Select object: *Select polyline (1)*
Specify second break point or [First point]: *Enter* **f**
Specify first break point: *Enter* **int**
of: *Select polyline (2)*
Specify second break point: *Enter* **@**



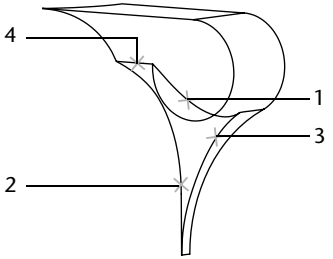
Check the grip points of the line segments after you break the polyline.

2 Use AMBLEND to create the inlet D surface, responding to the prompts.

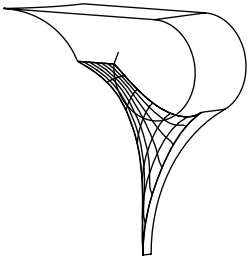
Desktop Menu Surface ➤ Create Surface ➤ Blend

Select first wire: *Select wire (1)*
Select second wire: *Select wire (2)*
Select third wire: *Select wire (3)*
Select fourth wire: *Select wire (4)*

Make selections in order, selecting opposite wires in pairs.



The blended surface should look like this.



3 Use CHPROP to move the surface to the INLET layer.

Join the lines to form the boundary of inlet E, and then create a trimmed planar surface from the joined lines. Zoom in as needed to make line selection easier.

To create the inlet E trimmed planar surface

1 Use AMJOIN3D to join selected lines to form the boundary for the inlet E surface.

Desktop Menu Surface ➤ Edit Wireframe ➤ Join

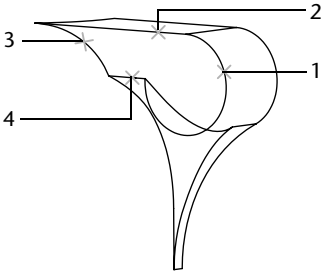
In the Join3D dialog box, specify:

Mode: Automatic
Output: Polyline
Gap Tolerance: *Enter* **.01**

Choose OK.

2 Respond to the prompts as follows:

Select start wire or: *Select polyline (1)*
Select wires to join: *Select wires (2) through (4)*
Select wires to join: *Press* ENTER
Reverse direction? [Yes/No] <No>: *Press* ENTER



This procedure joins lines regardless of their original direction and converts arcs and splines into polylines. You may need to reset the gap tolerance to correctly join the polylines.

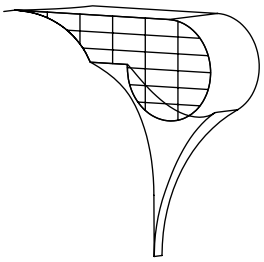
3 Use AMPLANE to create a trimmed planar surface from the joined lines, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Planar Trim

If you choose the command line method, enter **w** at the prompt before continuing to the following prompts.

Select wires: *Select polyline (1)*
Select wires: *Press* ENTER

Your surface should look like this.



4 Use CHPROP to move inlet E to the INLET layer.

Now you can trim top D by projecting the edge of the inlet. First, thaw layers to show the inlet and top sections of the pump. Then, break a polyline into segments and join one segment with other polylines. The joined polylines form the shape of the projection that cuts material where the two surfaces intersect.

To make selection easier, zoom and rotate the view as needed.

To create a shape on a surface using joined wires

1 Thaw layers 10 and 20.

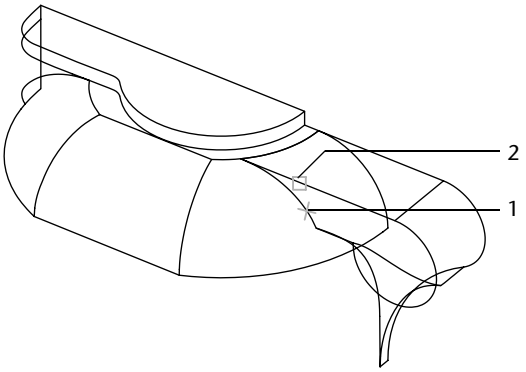
2 Change to the front right isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

3 Use BREAK to break the polyline, responding to the prompts.

Desktop Menu Modify ➤ Break

Select object: *Select polyline (1)*
Specify second break point or [First point]: *Enter* **f**
Specify first break point: *Enter* **end**
of: *Select polyline (2)*
Specify second break point: *Enter* **@**

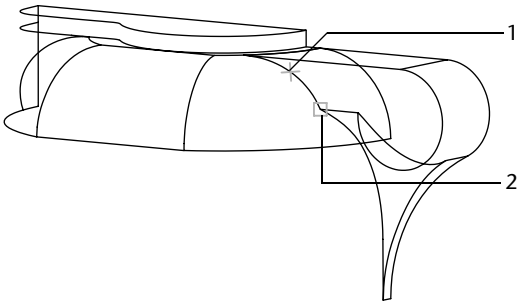


NOTE Use 3D Orbit and Zoom Realtime to rotate the view and zoom in to show the lines clearly. If you prefer, use VPOINT to set a precise viewpoint. In this case, set the coordinates **4,-6,1** to show the lines you need for the next step.

4 Break the upper part of the polyline into segments, responding to the prompts.

Desktop Menu Modify ➤ Break

Select object: *Select polyline (1)*
Specify second break point or [First point]: *Enter* **f**
Specify first break point: *Enter* **int**
of: *Select polyline (2)*
Specify second break point: *Enter* **@**



5 Use AMJOIN3D to combine three polyline segments.

Desktop Menu Surface ➤ Edit Wireframe ➤ Join

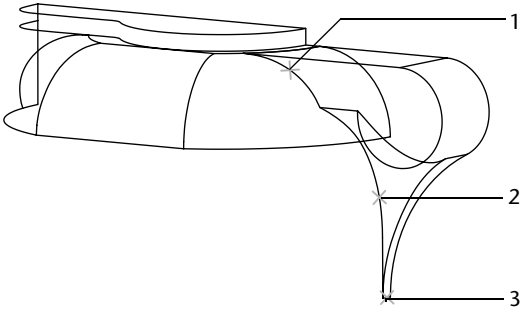
In the Join3D dialog box, specify:

Mode: Automatic
Output: Polyline
Gap Tolerance: *Enter* **.004**

Choose OK.

6 Respond to the prompts as follows:

Select start wire or: *Select polyline (1)*
Select wires to join: *Select wire (2)*
Select wires to join: *Select wire (3) and press* ENTER
Reverse direction? [Yes/No] <No>: *Press* ENTER



The segments are joined together. Later, you will project the joined line onto the top surface.

To trim a surface using a projected wire shape

1 Freeze layer 10 and thaw the TOP layer.

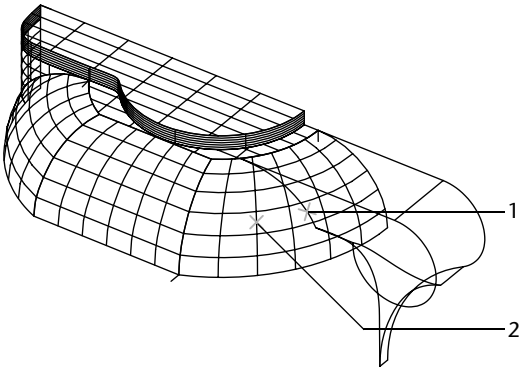
2 Return to the front right isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

3 Use AMPROJECT to cut top B where the inlet fits, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Project Trim

Select wires to project: *Select wire (1) and press* ENTER
Select target surfaces/quilts: *Select surface (2) and press* ENTER

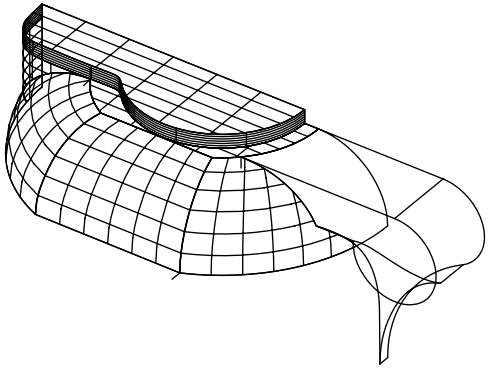


4 In the Project to Surface dialog box, specify:

Direction: Normal
Output Type: Trim Surface
Keep Original Wires: *Remove the check from the check box*

Choose OK.

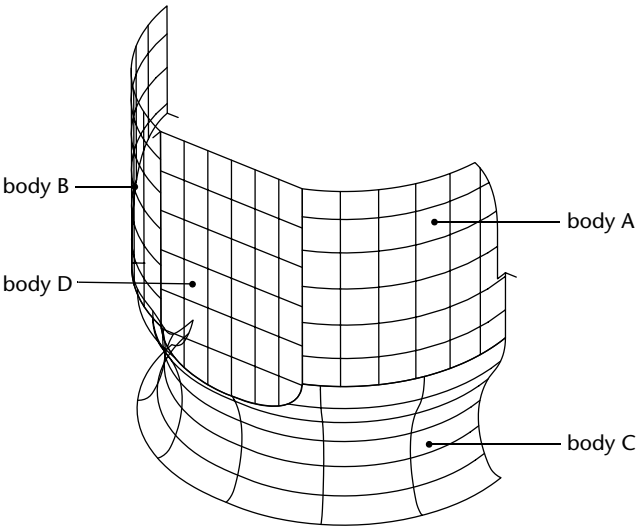
Top B is cut open for the inlet. The top and inlet are complete. Save the file.



Creating Swept and Projected Surfaces

For the main body of the pump, you continue building and trimming surfaces to their correct shapes.

- Body A, B, and C are swept surfaces created from curves and rails.
- Body D is a surface created from the boundaries of Body A, B, and C surfaces.



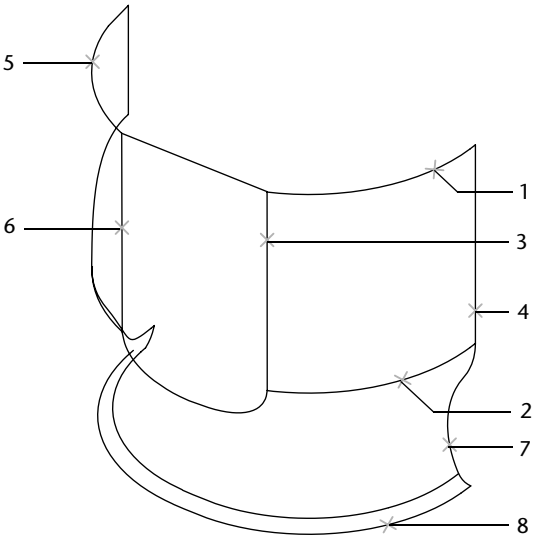
To create the body A, B, and C swept surfaces

1. Thaw layer 30 and make it current. Freeze layers 10, 20, and TOP.
2. Use AMSWEEPSF to create the body A surface on the right side of the model.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

Respond to the prompts:

Select cross sections: *Select wire (1)*
Select cross sections: *Select wire (2) and press* ENTER
Select rails: *Select wire (3)*
Select rails: *Select wire (4)*



3 In the Sweep Surface dialog box, specify:

Transition: Scale
Keep Original Wires: *Check the check box*

Choose OK

4 Use AMSWEEPSF to create the body B surface on the left side of the model, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

NOTE To repeat the previous command, press ENTER or the SPACEBAR.

Select cross sections: *Select wire (5) and press* ENTER
Select rails: *Select wire (6) and press* ENTER

5 In the Sweep Surface dialog box, under Orientation, specify Normal. Leave Keep Original Wires checked, and choose OK.

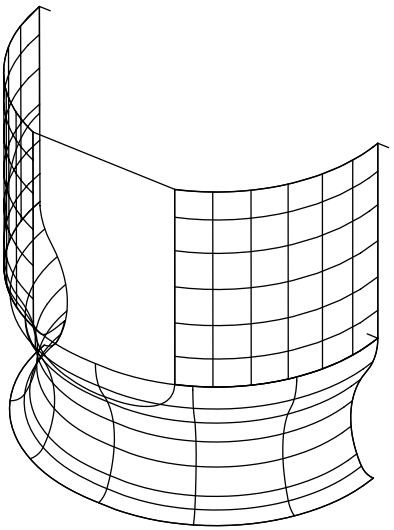
6 Create the body C surface near the bottom of the model, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *Select wire (7) and press* ENTER
Select rails: *Select the wire (8) and press* ENTER

7 In the Sweep Surface dialog box, under Orientation, specify Normal. Verify that Keep Original Wires is checked, and choose OK.

Your model should look like this.



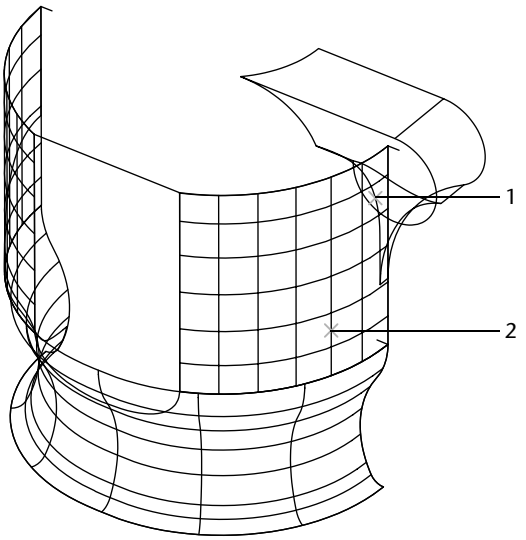
8 Thaw layer 20 to reveal the inlet wires.

To trim a surface with a projection wire

1 Use AMPROJECT to trim the body surface with the inlet edge, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Project Trim

Select wires to project: *Select wire (1)*
Select wires to project: *Press* ENTER
Select target surfaces/quilts: *Select surface (2)*
Select target surfaces/quilts: *Press* ENTER

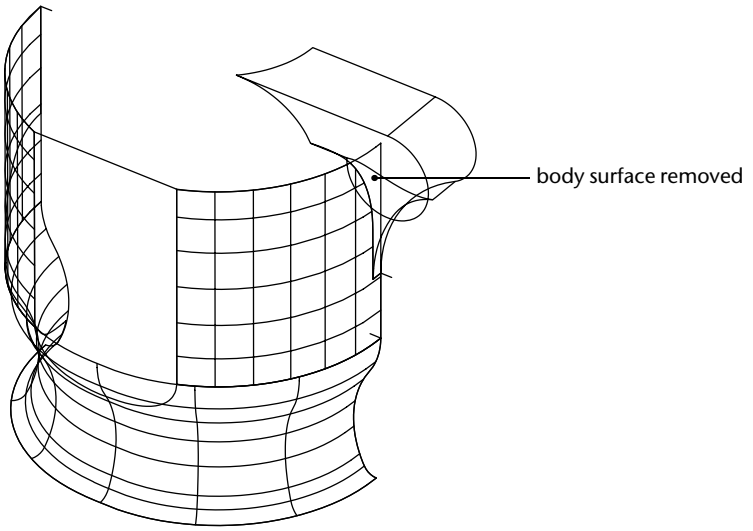


2 In the Project to Surface dialog box, specify:

Direction: Normal
Output Type: Trim Surface

Verify that Keep Original Wires is checked, and choose OK.

The projected wire cut away a portion of body A surface, but the wire was not deleted.



3 Freeze layer 20.

Cut out the surface areas on body C where body D and the outlet (to be surfaced later) extend onto body C.

To trim the body C surface with projection wires

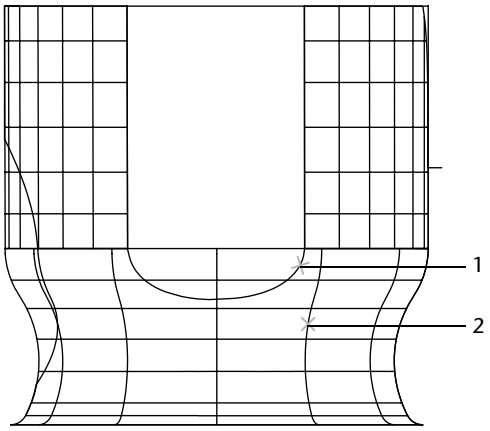
1 Change to the front view of your model.

Desktop Menu View ➤ 3D Views ➤ Front

2 Trim Body C with the lower curve of the flat surface (1), responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Project Trim

Select wires to project: *Select wire (1)*
Select wires to project: *Press* ENTER
Select target surfaces/quilts: *Select surface (2)*
Select target surfaces/quilts: *Press* ENTER

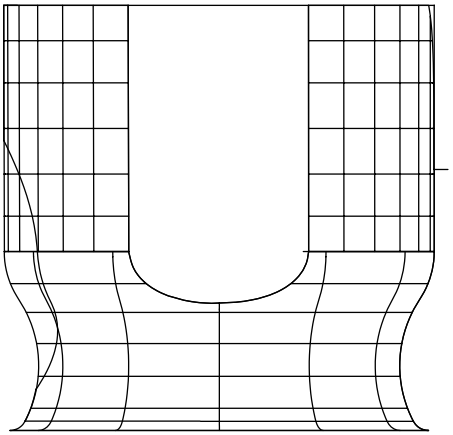


3 In the Project to Surface dialog box, specify:

Direction: Normal
Output Type: Trim Surface

Verify that Keep Original Wires is checked, and choose OK.

Your model should look like this.

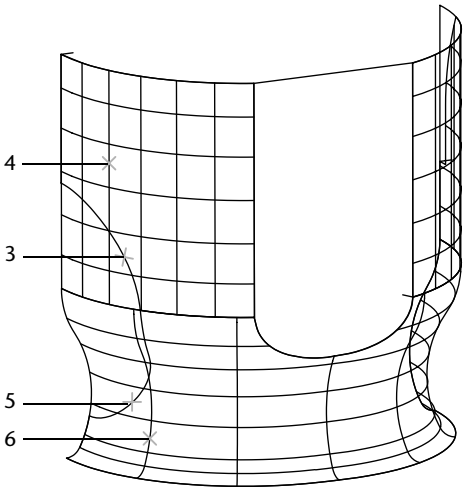


4 Change to the front left isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Left Isometric

NOTE To set the viewpoint precisely, use VPOINT to specify coordinates. For example, this viewpoint is -5,-10,3.

5 Trim the body B surface with the curve that defines the upper edge of the outlet. Repeating steps 2 and 3, project wire 3 onto surface 4.



6 Trim body C with the curve that defines the lower edge of the outlet, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Project Trim

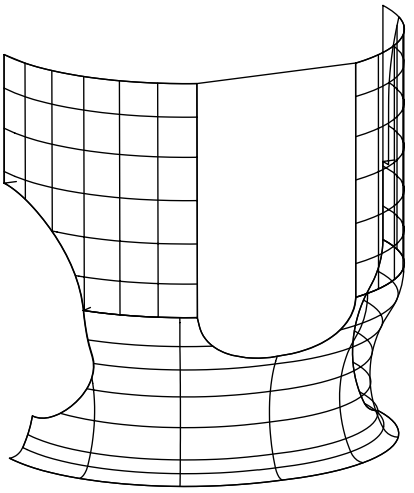
Select wires to project: *Select wire*
Select wires to project: *Press* ENTER
Select target surfaces/quilts: *Select surface (6)*
Select target surfaces/quilts: *Press* ENTER

7 In the Project to Surface dialog box, specify:

Direction: Normal
Output Type: Trim Surface

Verify that Keep Original Wires is checked, and choose OK.

Your model should look like this.



8 Use CHPROP to move surfaces A, B, and C to the BODY layer.

Next, you edit the wireframe to join the lines that form the boundary of body D. You use the polyline to create a planar surface.

Before you begin, set DELOBJ to delete original objects.

Toolbutton On the Surfacing toolbar, use the DELOBJ toolbutton to set delete original objects.

To create the body D planar surface

1 Use AMJOIN3D to join the polylines that define the boundary of body D.

Desktop Menu Surface ➤ Edit Wireframe ➤ Join

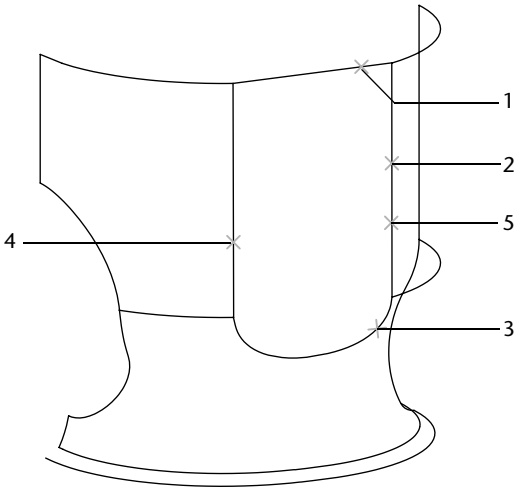
In the Join 3D dialog box, specify:

Mode: Automatic
Output: Polyline

Choose OK.

2 Respond to the prompts as follows:

Select start wire or: *Select the wire (1)*
Select wires to join: *Select wires (2), (3), and (4)*
Select wires to join: *Press* ENTER
Reverse direction? [Yes/No] <No>: *Press* ENTER

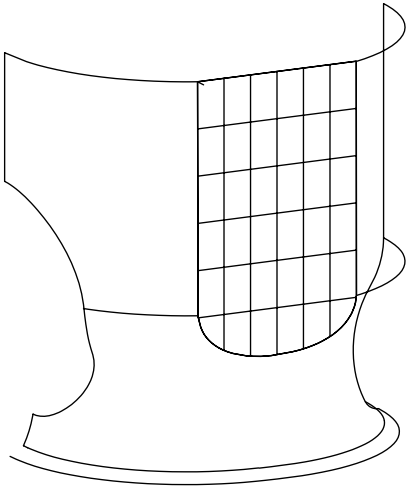


3 Use AMPLANE to create a planar surface from the joined line, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Planar

Specify first corner or [Plane/Wires]: *Enter* **w**
Select wires: *Select wire (5) and press* ENTER

Your model should look like this.

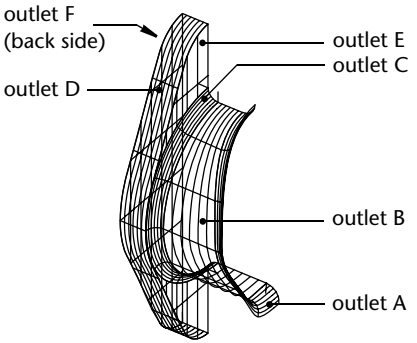


4 Use CHPROP to move body D to the BODY layer. Save the file.

The pump body surfaces are complete.

Creating Complex Swept Surfaces

Next, you create the surfaces for the outlet on the side of the pump.



Outlet A is a swept surface that blends dissimilar cross sections.

To create the outlet A swept surface

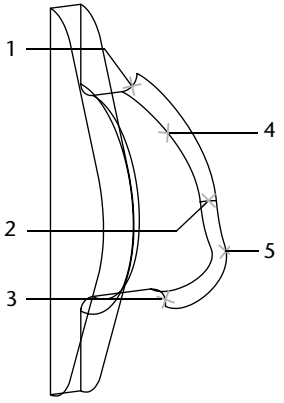
1. Thaw layer 40 and make it current, and then freeze all other layers.
2. Change to the left isometric view to make lines easier to select.

Desktop Menu View ➤ 3D Views ➤ Front Left Isometric

3. Use AMSWEEPSF to Sweep three cross sections along two rails, responding to the prompts.

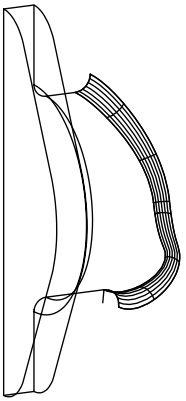
Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *Select wires (1), (2), and (3)*
Select rails: *Select wires (4) and (5)*



4 In the Sweep Surface dialog box, in Transition, specify Scale. Choose OK.

Outlet A should look like this.



5 Use CHPROP to move outlet A to the OUTLET layer.

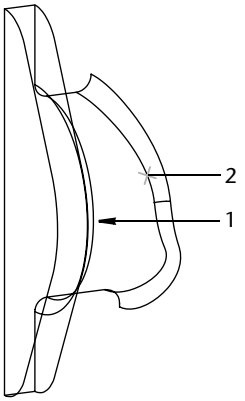
Next create a ruled surface for outlet B. The difference between this surface and the one you just completed is that outlet A is curved in two directions, and outlet B is curved in one direction and flat in the other.

To create the outlet B ruled surface

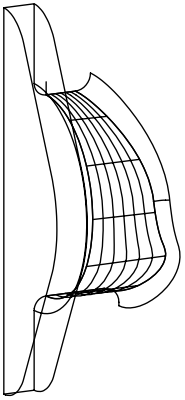
1 Use AMRULE to create the outlet B surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select wire (1)*
Select second wire: *Select wire (2)*



Outlet B should look like this.



2 Use CHPROP to move outlet B to the OUTLET layer.

Next, you create another swept surface and another ruled surface.

To create the outlet C and outlet D surfaces

1 To make selections easier, rotate the model to the left with the Desktop View icons, or set specific coordinates (**6**,**-8**,**1**) with VPOINT, responding to the prompts.

Desktop Menu View ➤ 3D Views ➤ VPOINT

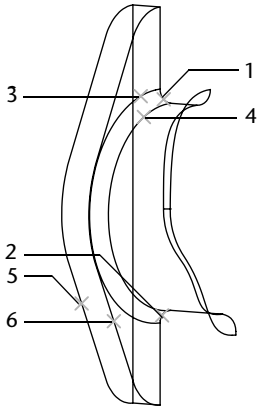
Current view direction: VIEWDIR=-1.0000,-1.0000,1.0000
Specify a view point or [Rotate] <display compass and tripod>: *Enter* **6,-8,1**

2 Use AMSWEEPSF to create outlet C swept surface.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

Respond to the prompts:

Select cross sections: *Select wire (1)*
Select cross sections: *Select wire (2) and press* ENTER
Select rails: *Select wire (3)*
Select rails: *Select wire (4)*



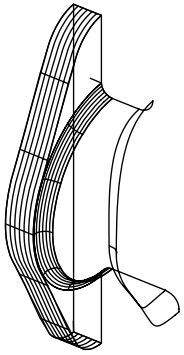
3 In the Sweep Surface dialog box, under Transition, specify Scale. Choose OK.

4 Use AMRULE to create outlet D ruled surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select wire (5)*
Select second wire: *Select wire (6)*

Your model should look like this.



5 Use CHPROP to move both of the surfaces to the OUTLET layer.

Next, you join lines to form the boundaries of outlet E and outlet F. From the newly created polyline, you create planar surfaces for the outlet.

To create the outlet E planar surface

1 Use AMJOIN3D to join the polylines.

Desktop Menu Surface ➤ Edit Wireframe ➤ Join

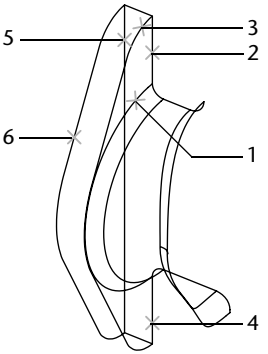
In the Join3D dialog box, specify:

Mode: Automatic
Output: Polyline

Choose OK.

2 Select the polylines, responding to the prompts.

Select start wire or: *Select wire (1)*
Select wires to join: *Select wires (2), (3), and (4), and press* ENTER
Reverse direction? [Yes/No] <No>: *Press* ENTER *to accept the join direction*



3 Use AMPLANE to create the outlet E surface from the joined polyline, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Planar

Specify first corner or [Plane/Wires]: *Enter* **w**
Select wires: *Select wire (1) and press* ENTER

To create the outlet F planar surface

1 Use AMJOIN3D to combine lines that form the boundary for outlet F.

Desktop Menu Surface ➤ Edit Wireframe ➤ Join

In the Join 3D dialog box, specify:

Mode: Manual
Output type: Polyline

Choose OK.

2 Select the polylines to join, responding to the prompts.

Select start wire or: *Select wire (5)*
Select wires to join: *Select wire (6) and press* ENTER
Reverse direction? [Yes/No] <No>: *Press* ENTER *to accept the join direction*

NOTE Use the Manual mode to join lines even if they are far apart. It joins all the lines you select in the order you choose them.

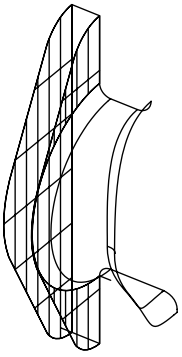
---

3 Use AMPLANE to create outlet F from the lines you just joined, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Planar

Specify first corner or [Plane/Wires]: *Enter* **w**
Select wires: *Select wire (6) and press* ENTER

Your model should look like this.

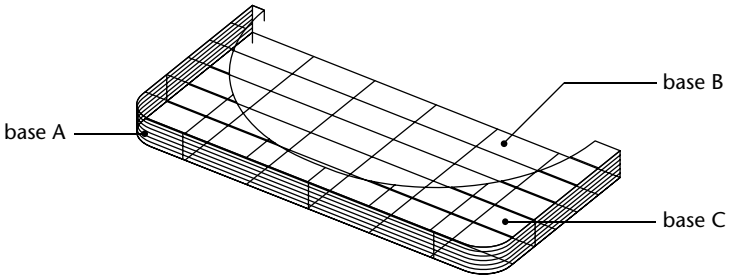


4 Use CHPROP to move the surfaces to the OUTLET layer.

Save the file.

Using Projection to Create Surfaces

Next, you use projection to create ruled and planar surfaces for the base of the pump.



To create the base A surface

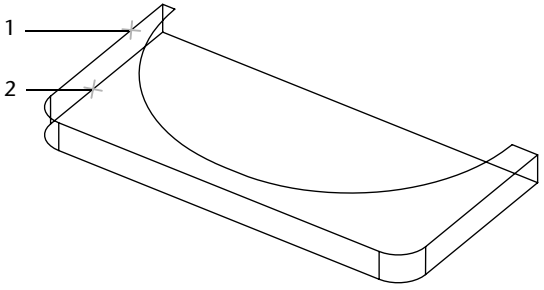
1. Thaw layer 50 and make it current. Then freeze all other layers.
2. Change to the left front isometric view.

Desktop Menu View ➤ 3D Views ➤ Front Right Isometric

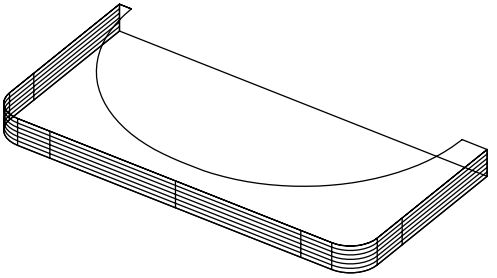
3. Use AMRULE to create the base A surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select wire (1)*
Select second wire: *Select wire (2)*



The illustration shows the ruled surface fit to the flat areas and corner curves.



4 Use CHPROP to move base A to the BASE layer.

Next, you join the lines needed to create a planar surface on the bottom of the pump. Then you copy the surface and trim it.

To create the base B and C surfaces

1 Use AMJOIN3D to create a polyline from two wires.

Desktop Menu Surface ➤ Edit Wireframe ➤ Join

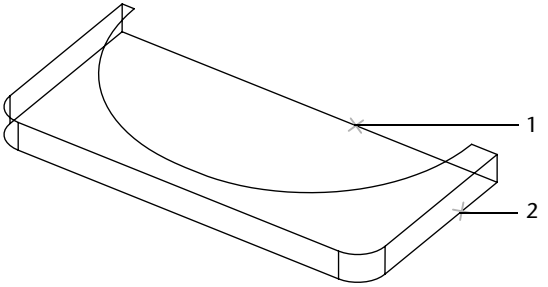
In the Join3D dialog box, specify:

Mode: Automatic
Output: Polyline

Choose OK.

2 Select the wires, responding to the prompts:

Select start wire or: *Select wire (1)*
Select wires to join: *Select wire (2) and press* ENTER
Reverse direction? [Yes/No] <No>: *Press* ENTER



3 Create a planar surface on the bottom of the base, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Planar

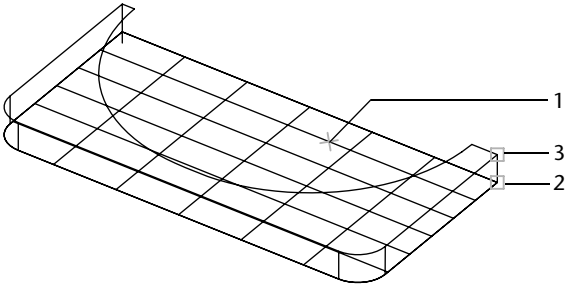
Specify first corner or [Plane/Wires]: *Enter* **w**
Select wires: *Select wire (1) and press* ENTER

The planar surface is created.

4 Use COPY to copy the last surface, responding to the prompts.

Desktop Menu Modify ➤ Copy

Select objects: *Select surface (1) and press* ENTER
Specify base point or displacement, or [Multiple]: *Enter* **end**
of: *Select point (2)*
Specify second point of displacement or <use first point as displacement>:
Enter **end**
of: *Select point (3)*



Next, project a wire onto the base C surface to trim it.

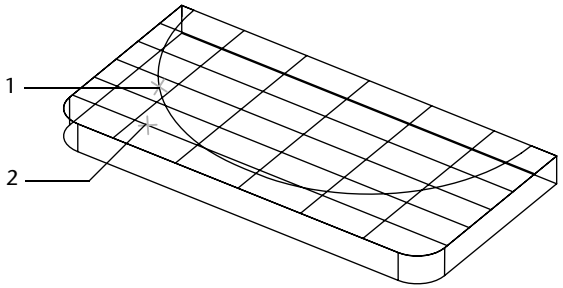
To trim the base C surface

1 Use CHPROP to move the bottom surface to the BASE layer.

2 Project the curve of the body onto the top surface of the base, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Project Trim

Select wires to project: *Select polyline (1) and press* ENTER
Select target surfaces/quilts: *Select surface (2) and press* ENTER

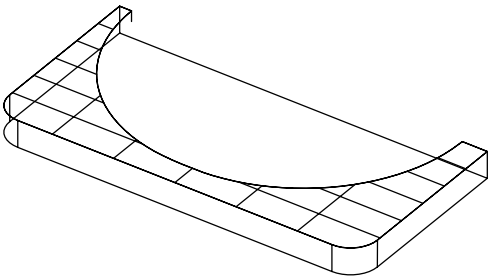


3 In the Project to Surface dialog box, specify:

Direction: Normal
Output type: Trim Surface

Choose OK.

Your model should look like this.



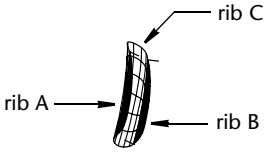
4 Use CHPROP to move the surfaces to the BASE layer.

Save the file.

Using Advanced Surfacing Techniques

Next, you create the support rib from the surfaces. Using the techniques you have already learned, surface the support rib from these general instructions.

Save a copy of your drawing before you begin working on your own.



To create the support rib

1. Thaw layer 60.
2. Create a ruled surface on the left side of the support rib (rib A).
3. Create a ruled surface on the right side of the support rib (rib B).
4. Move the surfaces to the SUPPORT_RIB layer.
5. Create a swept surface for rib C.
6. Move the surface to the SUPPORT_RIB layer.
7. Add the support rib to the body and base surfaces.

If you need to, follow these specific instructions to create the support rib.

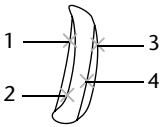
To create the rib A and rib B surfaces

1 Thaw layer 60 and make it current. Then freeze all other layers.

2 Use AMRULE to create a ruled surface on the left side of the support rib, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select wire (1)*
Select second wire: *Select wire (2)*



3 Create a ruled surface on the right side of the support rib, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Rule

Select first wire: *Select wire (3)*
Select second wire: *Select wire (4)*

The surfaces should look like this.

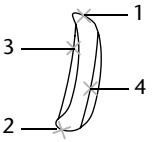
4 Use CHPROP to move the surfaces to the SUPPORT_RIB layer.

To create the rib C surface

1 Use AMSWEEPSF to create the rib C surface, responding to the prompts.

Desktop Menu Surface ➤ Create Surface ➤ Sweep

Select cross sections: *Select wire (1) and (2)*
Select cross sections: *Press* ENTER
Select rails: *Select wires (3) and (4)*



2 In the Sweep Surface dialog box, under Transition, specify Scale. Choose OK.

Your surface should look like this.

3 Move the surface to the SUPPORT_RIB layer.

4 Use AMJOIN3D to join the lines defining the boundary of the support rib.

Desktop Menu Surface ➤ Edit Wireframe ➤ Join

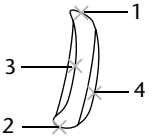
5 In the Join3D dialog box, specify:

Mode: Automatic
Output: Polyline

Choose OK.

6 Select the lines.

Select start wire or: *Select wire (1)*
Select wires to join: *Select wires (2), (3), and (4), and then press* ENTER
Reverse direction? [Yes/No] <No>: *Press* ENTER



The support rib wires are joined and ready to project onto the pump.

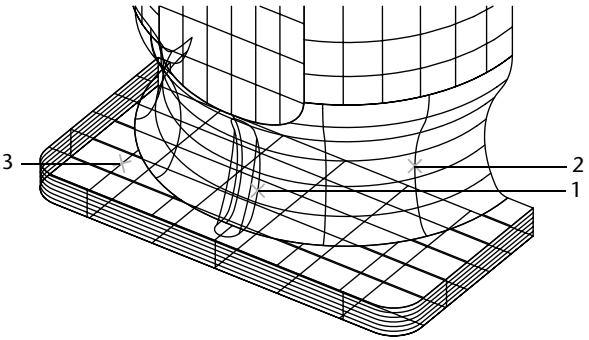
To add the support rib

1 Thaw the BODY and BASE layers.

2 Use AMPROJECT to project the support rib onto the pump, responding to the prompts.

Desktop Menu Surface ➤ Edit Surface ➤ Project Trim

Select wires to project: *Select wire (1) and press* ENTER
Select target surfaces/quilts: *Select surface (2)*
Select target surfaces/quilts: *Select surface (3) and press* ENTER

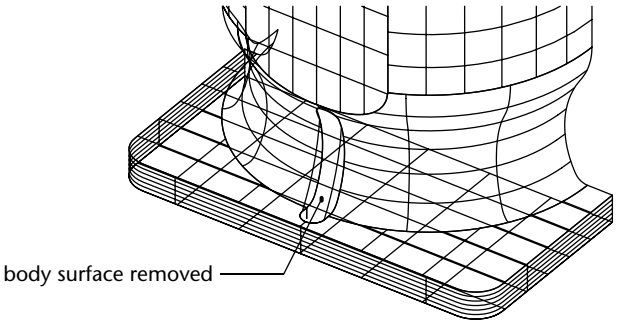


3 In the Project to Surface dialog box, specify:

Direction: Normal
Output: Trim Surface

Choose OK.

The support rib is projected onto the body and the base.

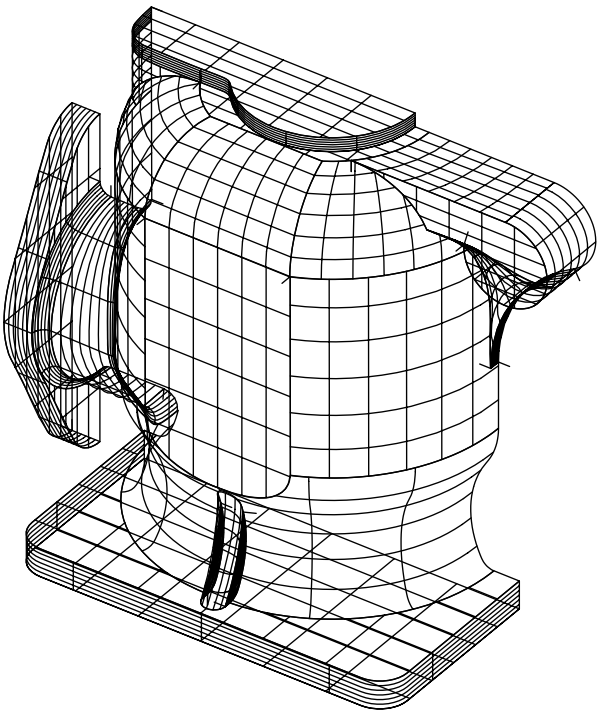


Save your file.

Viewing Completed Surfaced Models

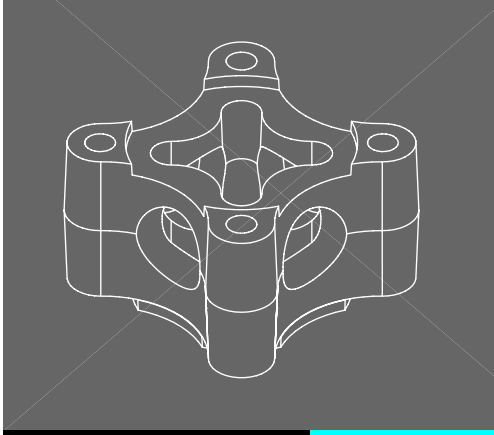
To view the completed model, freeze all layers except BASE, BODY, INLET, OUTLET, SUPPORT_RIB, and TOP.

Use the Zoom Extents option of ZOOM to view the entire wireframe model.



One half of the pump housing is complete. You can mirror the surfaces to create a complete model.

Working with Standard Parts



In This Chapter

22

- Selecting standard parts
- Inserting holes using the cylinder axial placement method
- Inserting holes using the cylinder radial placement method
- Inserting a screw connection
- Creating a drawing view with 2D standard parts

Standard parts is the term used for the vast selection of real-world reusable 2D and 3D parts, holes, features, and structural steel profiles that are available to you in Autodesk® Mechanical Desktop® 6 with the power pack. These standard parts are provided in many different base standards.

In this tutorial you select standard parts to insert through holes on a 3D part, using two different positioning methods. You also add a screw connection to a 3D assembly model using standard parts.

Run Mechanical Desktop with the power pack to perform this tutorial.

Key Terms

Term	Definition
base standard	Predefined drafting standard that conforms to International Drafting Standards ANSI, BSI, CSN, DIN, GB, ISO, and JIS.
cylinder axial	Option for placement when you insert a standard part or hole parallel to a cylinder axis.
cylinder radial	Option for placement when you insert a standard part or hole radial into a cylinder.
hole	Geometric feature with a predefined shape: drilled, counterbore, or countersink.
parallel	Being an equal distance apart at every point.
standard part	Reusable 2D and 3D parts, holes, features, and structural steel profiles available in Mechanical Desktop 6.
tangent	Touching at a single point or along a line, but not intersecting.
through	Termination method by which a feature extends from its sketch plane through the part.

Tutorial at a Glance

This tutorial is an introduction to the standard parts and calculations functionality in Mechanical Desktop 6. You will become familiar with some of the intelligence and automation built into this functionality as you perform exercises to insert

- A through hole using the cylinder axial placement method.
- Through holes using the cylinder radial placement method.
- A screw connection using the automated screw connection feature.
- A standard related 2D-Representation of standard parts.

Basic Concepts of Standard Parts

A large selection of standard parts are made available to you in Mechanical Desktop 6. These standard parts are provided in many different base standards. You make your selections from lists of standard parts as you work. For example, when you select a screw, you can choose from different types of screws and from different configurations of threads. Using the standard parts and calculations functionality, you can insert complete screw connections including screws, holes, and nuts in one automated process.

The software is intelligent, and prevents you from assembling parts that do not fit. For example, if you select a screw with a metric thread, only metric threads are added with any additional parts or features, such as threaded holes or nuts. If you select screws, holes, and nuts from varying standards, the software determines if those parts will work together.

Inserting Through Holes

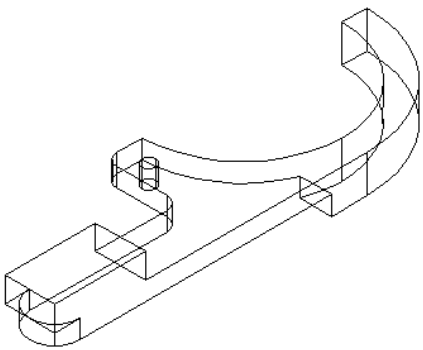
It is no longer necessary to insert a workpoint on the cylinder face, dimension it, and insert a hole on the workpoint. Instead, the standard hole function automatically defines a workpoint at the location you select, dimensions it, and places the hole you specify. You can also define the insertion point dynamically.

Using Cylinder Axial Placement

In this exercise, you insert a standard through hole using the cylinder axial placement method. Use this method to insert holes parallel to the axis of a cylinder. You enter specifications and select a size from a list of standard holes, and the hole is inserted automatically.

Open the file *md_ex01.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.



To insert a hole using cylinder axial placement

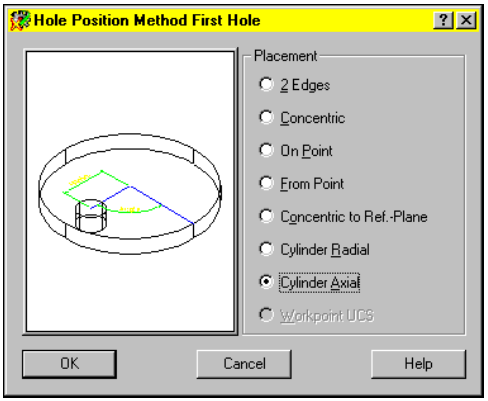
1 Use AMTHOLE3D to define the hole to insert.

Menu Content 3D ➤ Holes ➤ Through Holes

In the Select a Through Hole dialog box, select ISO 273 normal.

2 In the Hole Position Method First Hole dialog box, specify:

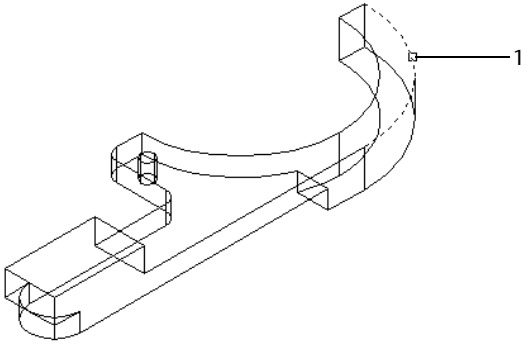
Placement: Cylinder Axial



Choose OK.

3 On the command line, respond to the prompts as follows:

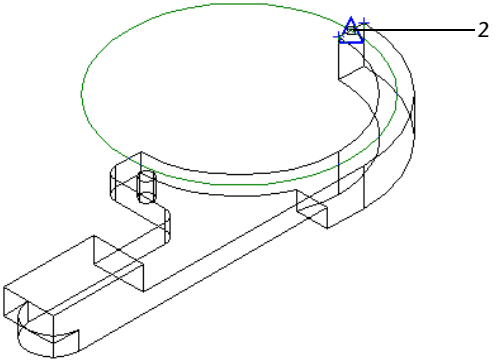
Select circular edge: *Select the upper circular edge (1)*



4 Continue on the command line.

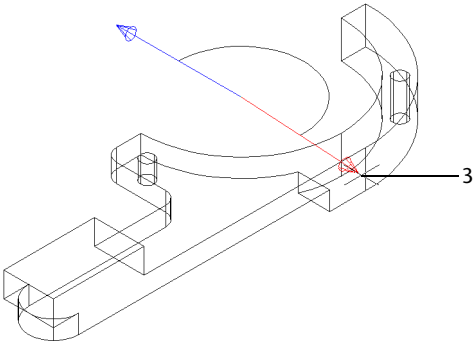
Select radius: *Press SHIFT and in the graphics area, right-click and choose Midpoint*

5 Select the midpoint of the upper horizontal edge (2).



6 Continue on the command line.

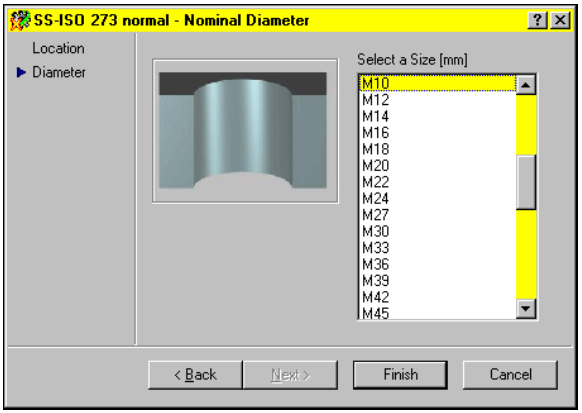
Select insertion method
[Angle to plane or edge/parallel to Line/plane Normal/plane Parallel] <plane Parallel>: *Enter* **A**
Select straight edge, work plane or planar face to add angular constraint to: *Press* ENTER
Select angle: *Select an angle of approximately 135° (3)*



Enter angle [Associate to/Equation assistant] <145>: *Enter* **135**
Radius [Associate to/Equation assistant] <90>: *Press* ENTER
Hole termination [toPlane/Thru] <Thru>: *Press* ENTER

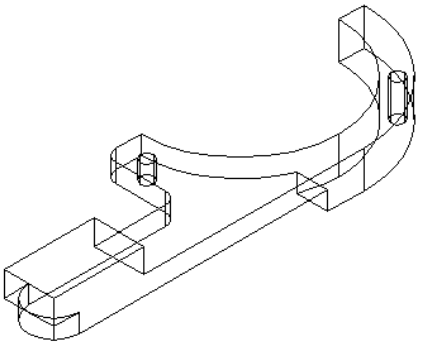
7 In the ISO 273 normal - Nominal Diameter dialog box, specify:

Select a size: M10



Choose Finish.

The through hole is inserted in the size and location you selected.



Save the file as *md2_ex01a.dwg*.

Using Cylinder Radial Placement

In this exercise, you insert a through hole using the cylinder radial method. Use this method to insert holes radial to a cylinder face.

To insert a hole using cylinder radial placement

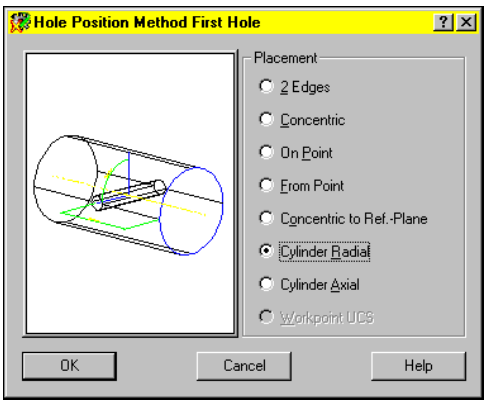
1 Use AMTHOLE3D to define the hole to insert.

Menu Content 3D ➤ Holes ➤ Through Holes

In the Select a Through Hole dialog box, select ISO 273 normal.

2 In the Hole Position Method First Hole dialog box, specify:

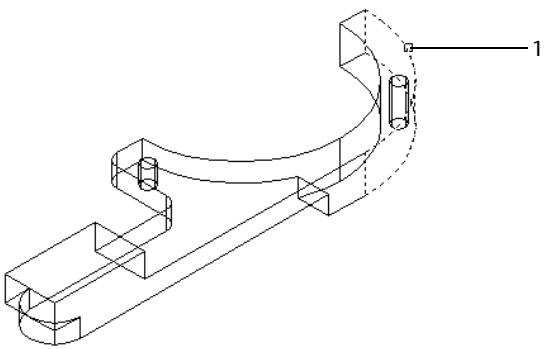
Placement: Cylinder Radial



Choose OK.

3 On the command line, respond to the prompts as follows:

Select cylindrical face: *Select the upper cylindrical face (1)*

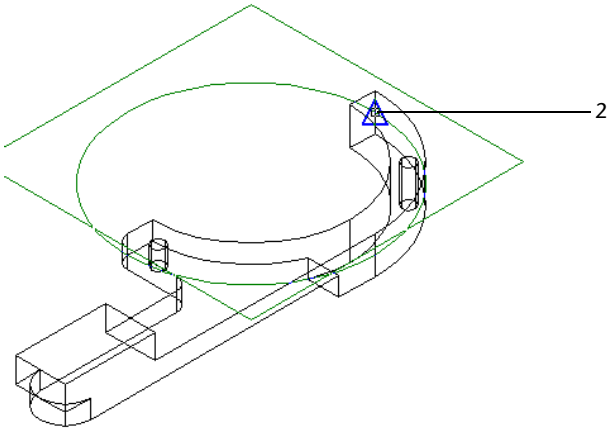


Continue on the command line.

Specify hole location [Line/Plane]:

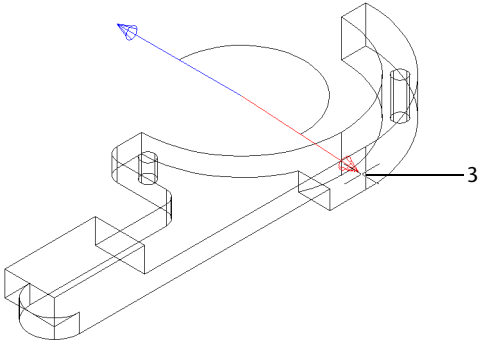
Press SHIFT and in the graphics area, right-click and choose Midpoint

4 Select the midpoint of the upper vertical edge (2).



5 Respond to the prompts as follows:

Enter distance from base plane [Associate to/Equation assistant] <15>:
Press ENTER
Select drill direction
[Angle to plane or edge/parallel to Line/plane Normal/plane Parallel]
<plane Parallel>: *Enter* **A**
Select straight edge, work plane or planar face to add angular constraint to:
Press ENTER
Select angle: *Select an angle of approximately 180° (3)*

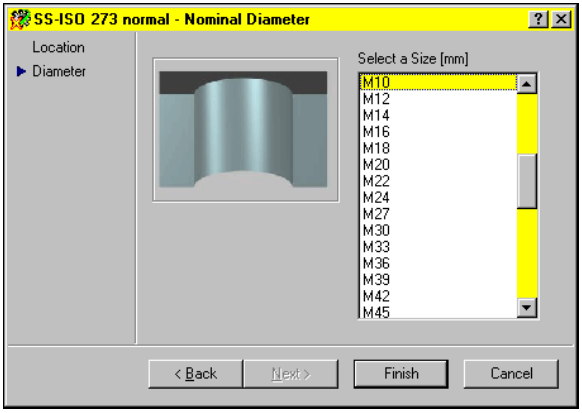


Continue on the command line:

Enter angle [Associate to/Equation assistant] <181>: *Enter* **180**
Hole termination [toPlane/Thru] <Thru>: *Press* ENTER

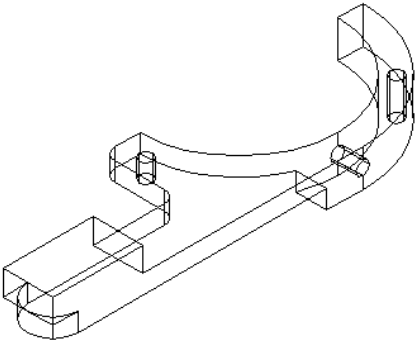
6 In the ISO 273 normal - Nominal Diameter dialog box, specify:

Select a size: M10



Choose Finish.

The through hole is inserted. Your drawing should look like this:



Save the file as *md2_ex01b.dwg*.

Inserting Screw Connections

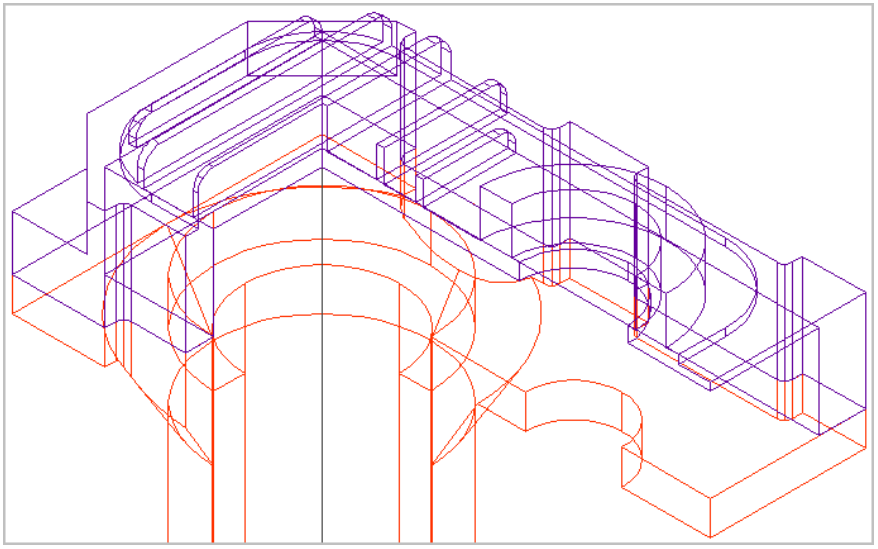
In this exercise, you begin with a drawing of two parts that need a screw connection. Using the screw connection feature, you select the screw, holes, and nut that you want to use.

You define the size of the screw in a dialog box. Then you insert the screw connection in the assembly. With this method, there is no need to create two separate holes before you insert the screw with the nut. The screw connection function does this for you automatically.

Open the file *md_ex02* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing contains two parts of a housing.

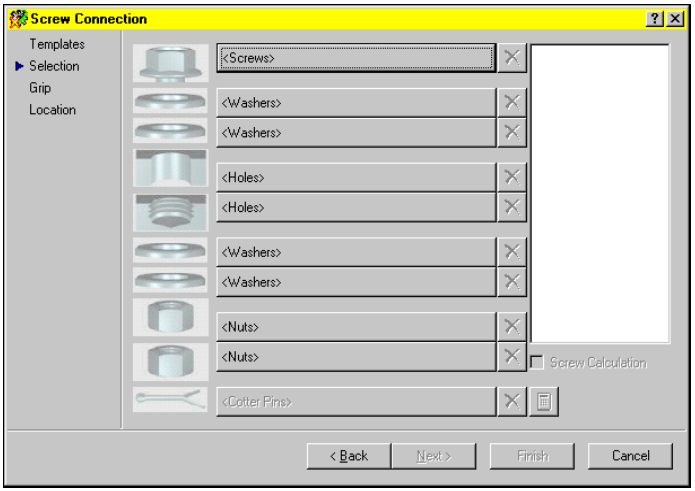


To insert a screw connection

1 Use AMSCREWCON3D to choose and define the screw to insert.

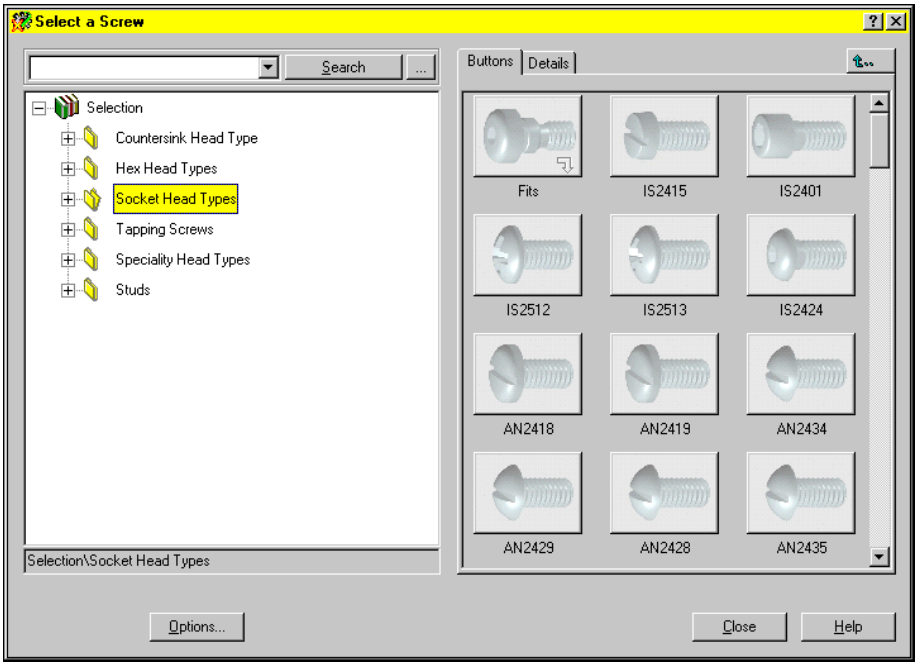
Menu Content 3D ➤ Screw Connection

In the Screw Connection dialog box, select Screws.



Next, define the type of screw.

2 In the Please select a Screw dialog box, select Socket Head Types.



Select the screw IS2401.

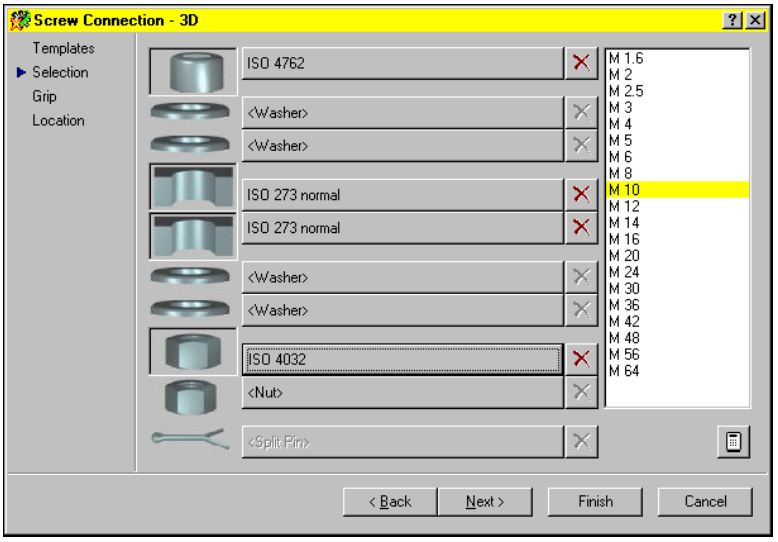
The Screw Connection dialog box is displayed again.

3 Repeat steps 1 and 2 to select and define each of the following parts:

Hole ➤ Through Cylindrical ➤ IS1602 normal
Hole ➤ Through Cylindrical ➤ IS1602 normal
Nut ➤ Hex Nuts ➤ ISO842

After you select and define the parts for the screw connection, select a size for the diameter.

4 Select M10 for the diameter.

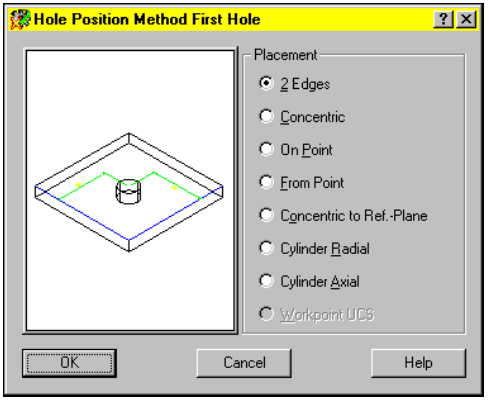


Choose Finish.

The Hole Position Method First Hole dialog box is displayed.

5 In the Hole Position Method First Hole dialog box, specify the hole positioning method as follows:

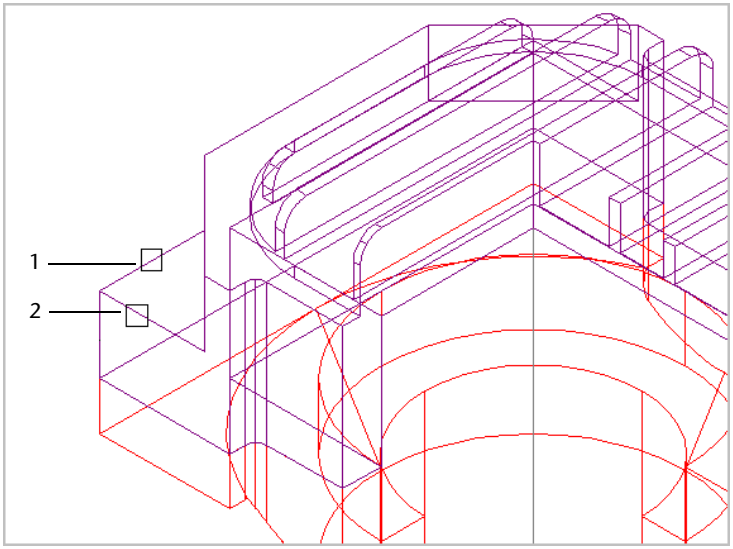
Placement: 2 Edges



Choose OK.

6 On the command line, respond to the prompts as follows:

Select first edge or planar face: *Specify the first edge (1)*
Select second edge or planar face: *Specify the second edge (2)*

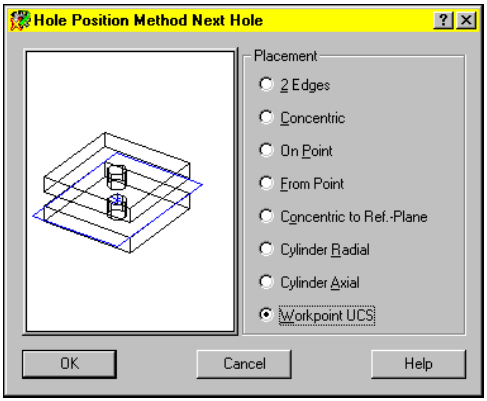


Continue on the command line.

Specify the hole location: *Specify a point on the face, offset from the two edges*
Enter distance from first geometry (highlighted) [Associate to/Equation assistant] <15.06>: *Enter* **20**
Enter distance from second geometry (highlighted) [Associate to/Equation assistant] <23.05>: *Enter* **20**
Hole termination [toPlane/Thru] <Thru>: *Press* ENTER

7 In the Hole Position Method Next Hole dialog box, specify:

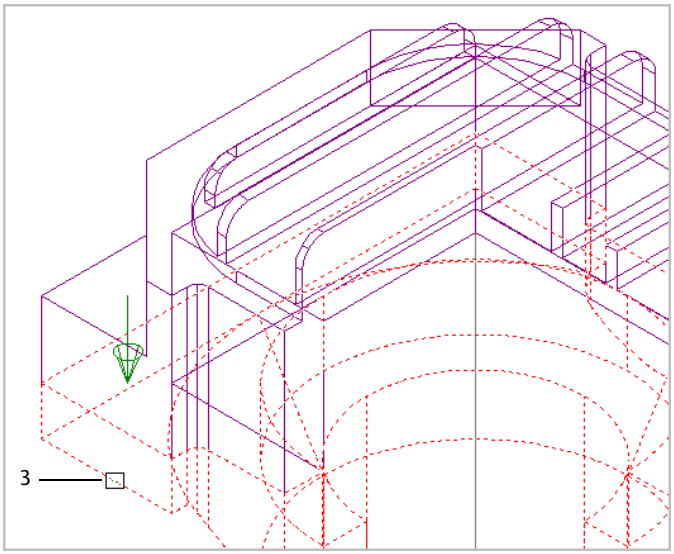
Placement: Workpoint UCS



Choose OK.

8 Respond to the prompt as follows:

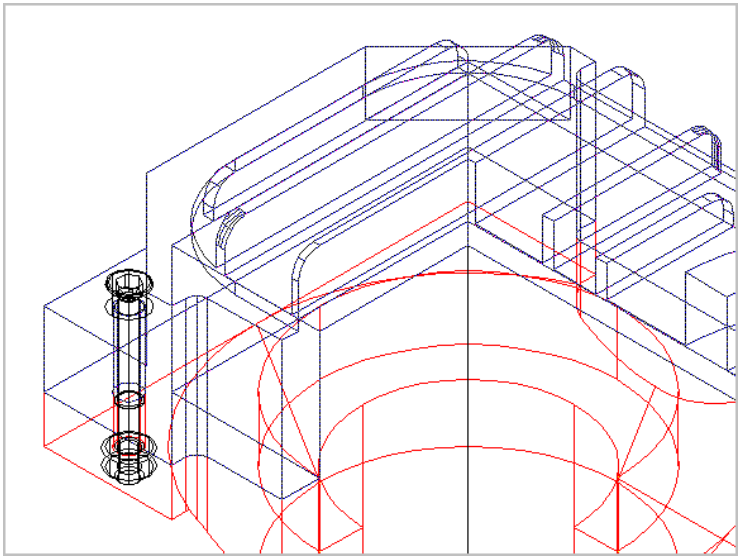
Select next part to drill through: *Select the lower part (3)*



Continue on the command line:

Hole termination [toPlane/Thru] <Thru>: *Press* ENTER

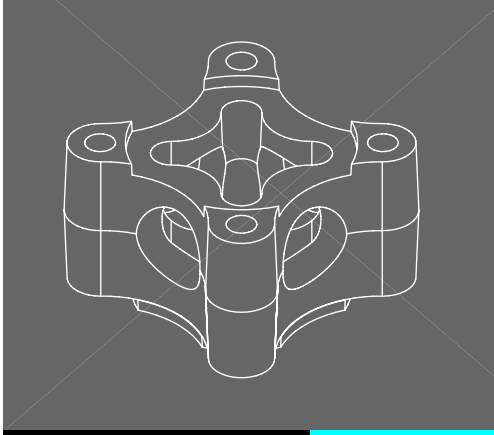
The screw connection is inserted. Your drawing should look like this:



You have completed this tutorial.

Save the file as *md6_ex17b.dwg*.

Creating Shafts



In This Chapter

23

- New in this tutorial
- Using the Shaft Generator
- Creating shaft geometry
- Adding thread and profile information to a shaft
- Editing a shaft
- Adding standard parts to a shaft
- Shading and displaying 3D views

The shaft generator in Autodesk® Mechanical Desktop® 6 is an automated feature that eliminates the need for many of the manual steps previously required to create shafts.

In this tutorial, you learn how to design shafts using the shaft generator. You create a shaft with segments of different shapes, and add threads and a profile. Then you edit the shaft and add standard parts to it. Finally, you check your design in front and isometric views with wireframe and shaded displays.

Run Mechanical Desktop with the power pack to perform this tutorial.

Key Terms

Term	Definition
bearing calculation	Calculates limiting value, dynamic and static load rating, dynamic and static equivalent load, and fatigue life of bearing in revolutions and hours.
centerline	Line in the center of a symmetrical object. When you create centerlines, you specify the start and end points.
chamfer	A beveled surface between two faces or surfaces.
dynamic calculation	Calculation required for a revolving bearing. The result is the Adjusted Rating Life. This is the life of the bearing associated with 90% reliability with contemporary, commonly used material, and under conventional operating conditions. With the number of revolutions you get the life in working hours.
dynamic dragging	The act of determining the size of a standard part with the cursor while inserting it into a side view. The standard part is displayed dynamically on the screen and can be dragged to the next possible size and length. The values (sizes) are taken from the Standard parts database.
shaft generator	Tool to draw rotationally symmetric parts.
UCS	User coordinate system. Designated by arrows that signify the XY coordinates. Establishes a construction plane and simplifies location of 3D points. Provides visual reference for positioning a surface.

Tutorial at a Glance

It is no longer necessary to create a cylinder manually and define new sketch planes and work planes in order to create a shaft. Instead, you make selections in the 3D Shaft Generator dialog box, and enter values to further define your selections.

In this tutorial you use this automated feature to

- Create a shaft with cylindrical and conical sections.
- Add threads and a profile to the shaft.
- Edit the shaft.
- Add standard parts to the shaft.
- Shade and display 3D views of the shaft.

Basic Concepts of the Shaft Generator

The shaft generator is an automated feature for creating shafts of many configurations, including outer and inner shaft contours for hollow shafts, You define the different segments of a shaft using the options in the 3D Shaft Generator dialog box.

As you select and define each segment, it is generated automatically and added to the shaft. Usually, you work from left to right as you add different types of segments, such as cylinders, cones, thread, profiles, and grooves.

Shaft segments can be inserted, deleted, or edited.

Each element of a shaft is generated from the segment preceding it. When you delete a shaft element, you also delete the shaft elements that follow.

As you create shaft segments, corresponding icons for the work features and shaft contours are displayed in the Desktop Browser.

While you are creating the shaft, it is helpful to display front and isometric views using two viewports. Toggling between wireframe and shaded displays also makes it easier to visualize your design.

Using the Shaft Generator

You use the 3D Shaft Generator dialog box to select a type of shaft segment, such as a cylinder or a cone, and then you define that segment. The shaft generator creates the segment automatically and adds it to the previous segment.

Getting Started

In order to work through this tutorial chapter, the ISO standard system has to be installed at your system.

Moreover, you need to set your measurement units to metric. This can easily be done by selecting an appropriate drawing template. Open the drawing template *acadiso.dwt*.

To open a drawing template

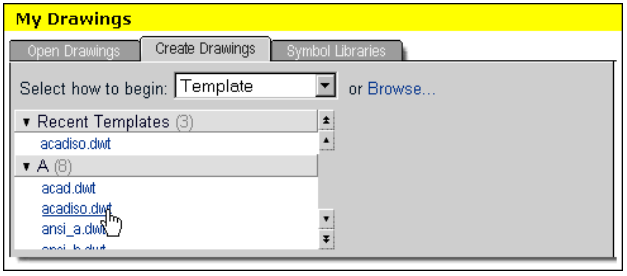
1 Open a new template.

Menu File ➤ New

Command NEW

The AutoCAD Mechanical Today window is displayed.

2 In the AutoCAD Mechanical Today window, in the section My Drawings, change to the tab Create Drawings, and select the template *acadiso.dwt*.



Now, you are working with the selected template.

Creating Shaft Geometry

You start creating the shaft by defining the segments that govern its shape. In the 3D Shaft Generator dialog box, you select the first segment type and then you define that segment. You continue adding segments until the contour of the shaft is complete.

To create a shaft using the shaft generator

1 Use AMSHAFT3D to define the shaft.

Menu Content 3D ➤ Shaft Generator

Respond to the prompts as follows:

Specify start point or [Existing shaft]: *Select a point in the drawing*
Specify centerline endpoint: *Drag a line to the right, and select a point*
Specify point for new plane <parallel to UCS>: *Press* ENTER

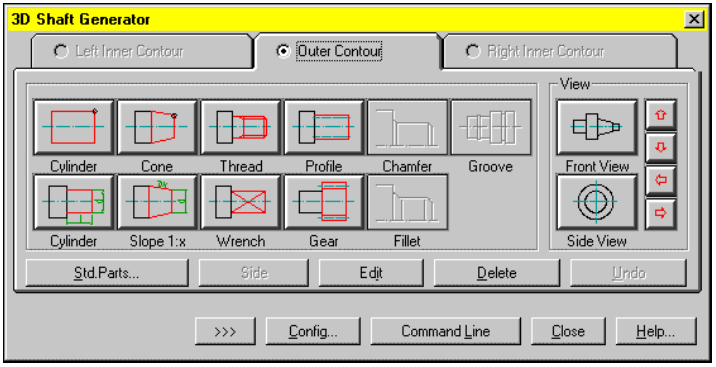
The 3D Shaft Generator dialog box is displayed.

Define the shaft contour, starting with the segment at the left, and working to the right.

2 Verify the Outer Contour tab is selected, then choose the lower Cylinder icon.

Respond to the prompts as follows:

Specify length or [Associate to/Equation assistant] <50>: *Enter* **10**
Specify diameter or [Associate to/Equation assistant] <40>: *Enter* **74**



3 Choose the Slope icon, and respond to the prompts as follows:

Specify length or [Dialog/Associate to/Equation assistant] <10>: *Enter* **7**
Specify diameter at start point or [Associate to/Equation assistant] <74>:
Press ENTER
Specify diameter at end point or [Slope/aNgle/Associate to/Equation assistant] <72>: *Enter* **48**

NOTE If the 3D Shaft Generator dialog box hides your shaft, move the dialog box to another position on the screen.

---

4 Choose the lower Cylinder icon again, and respond to the prompts as follows:

Specify length or [Associate to/Equation assistant] <7>: *Enter* **20**
Specify diameter or [Associate to/Equation assistant] <48>: *Enter* **40**

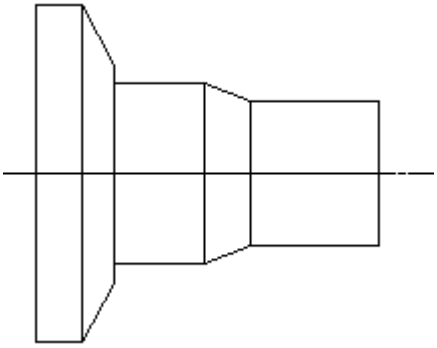
5 Choose the Slope icon again, and respond to the prompts as follows:

Specify length or [Dialog box/Associate to/Equation assistant] <20>: *Enter* **10**
Specify diameter at start point or [Associate to/Equation assistant] <40>:
Press ENTER
Specify diameter at end point or [Slope/aNgle/Associate to/Equation assistant] <36>: *Enter* **32**

6 Choose the lower Cylinder icon once more, and respond to the prompts as follows:

Specify length or [Associate to/Equation assistant] <10>: *Enter* **28**
Specify diameter or [Associate to/Equation assistant] <32>: *Press* ENTER

You have created a shaft consisting of three cylindrical and two conical segments. Your drawing should look like this.



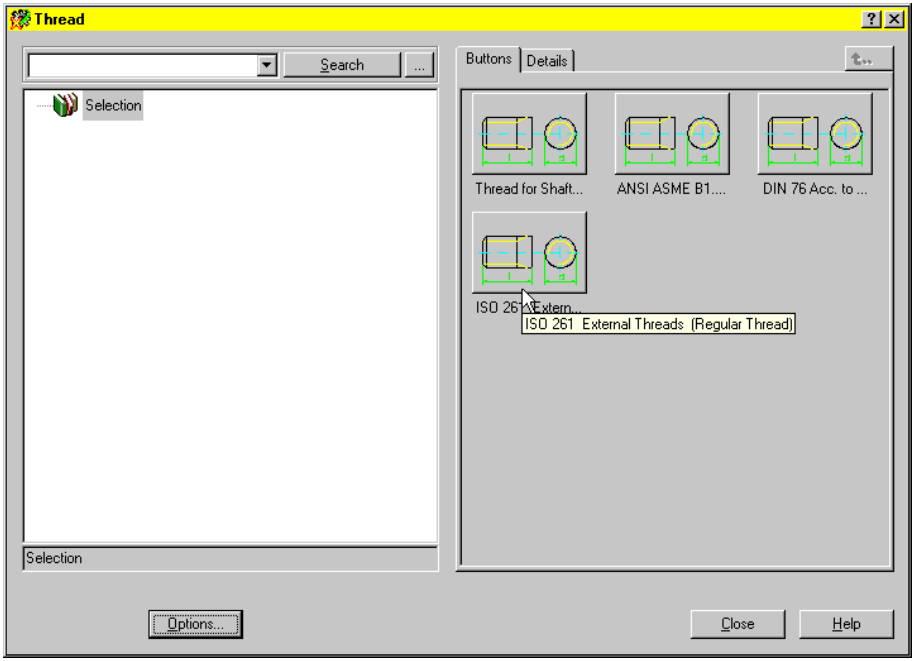
Next, you add thread information to the shaft.

Adding Threads to Shafts

The 3D Shaft Generator dialog box provides the option to add threads to a shaft. You define the thread information in the Thread dialog box, and the thread is added to the shaft automatically.

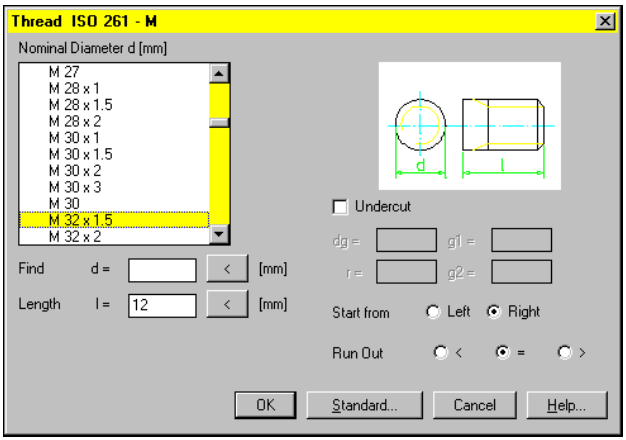
To add threads to a shaft

1. In the 3D Shaft Generator dialog box, select the Outer Contour tab and choose the Thread icon.
2. The Thread dialog box is displayed. Select ISO 261 from the available thread types.



3 The Thread ISO 261 dialog box is displayed. Specify:

Nominal Diameter d [mm]: M 32 x 1.5
Length l=: *Enter* **12**



4 Choose OK.

NOTE If you have previously chosen a thread standard, the Thread dialog box opens directly to the Nominal Diameter selection screen (*illustrated above*). There is no need to choose the same standard again. To return to the standard selection list in the Thread dialog box, choose Standard.

Mechanical Desktop Power Pack calculates the thread and adds it to the shaft, then returns you to the 3D Shaft Generator dialog box.

Following the thread segment, you need to add another cylindrical segment with a diameter smaller than the threaded section.

5 Choose the lower Cylinder icon, and respond to the prompts as follows:

Specify length or [Associate to/Equation assistant] <28>: *Enter* **2**
Specify diameter or [Associate to/Equation assistant] <32>: *Enter* **25**

The new segment is added, and you are ready to add a profile to your shaft.

Adding Profile Information to Shafts

The 3D Shaft Generator dialog box provides the option to add a profile to a shaft. You further define the profile information in the Shaft dialog box.

Add a profile segment to connect a drive to the shaft.

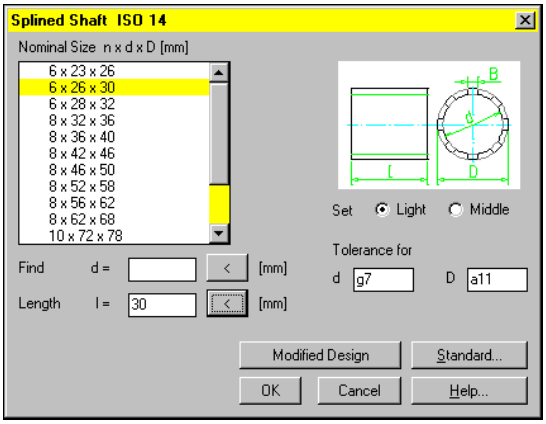
To add a profile to a shaft

1 In the 3D Shaft Generator dialog box, choose the Profile icon.

2 In the Profile dialog box, select ISO 14.

3 In the Splined Shaft dialog box, specify:

Nominal Size n x d x D [mm]: 6 x 26 x 30
Length l=: *Enter* **30**



Choose OK.

The profile is added to the shaft.

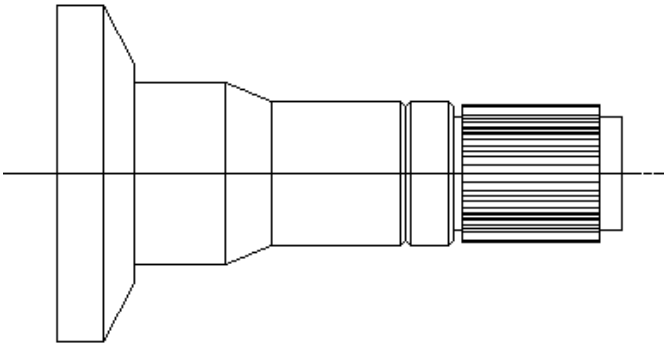
Add another small cylindrical segment to the end of the shaft.

4 In the 3D Shaft Generator dialog box, choose the lower Cylinder icon again, and respond to the prompts as follows:

Specify length or [Associate to/Equation assistant] <30>: *Enter* **5**
Specify diameter or [Associate to/Equation assistant] <30>: *Enter* **25**

5 Close the 3D Shaft Generator dialog box.

The shaft contour is complete. Your drawing should look like this.



Save your drawing as *shaft.dwg*.

Next, you edit the shaft.

Editing Shafts

You can make changes to simple shaft segments such as cylinders and cones. It is recommended that you delete a more complex segment, such as a gear, and create a new one. In this exercise, you add a chamfer to a segment and you add a groove to another segment.

First, add a chamfer to the end segment of your shaft.

To add a chamfer to a shaft segment

1 Activate the shaft generator.

Menu Content 3D ➤ Shaft Generator

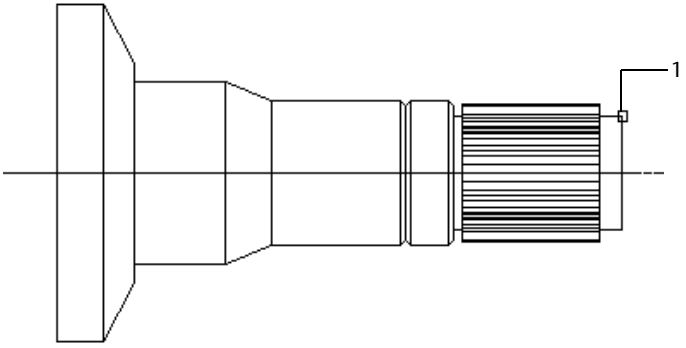
Command AMSHAFT3D

2 On the command line, respond to the prompts as follows:

Specify start point or [Existing shaft]: *Enter* **E**
Select shaft: *Select the shaft*

3 In the 3D Shaft Generator dialog box, select the Outer Contour tab. Choose the chamfer icon, and respond to the prompts as follows:

Select edge for chamfer: *Select the edge (1)*
Specify length (max. 5) or [Associate to/Equation assistant] <2.5>: *Enter* **2.5**
Specify angle (min 0.0001, max 78.69) [Distance/Associate to/Equation assistant] <45>: *Enter* **45**
Enter an option [Revolve/Chamfer] <Revolve>: *Press* ENTER



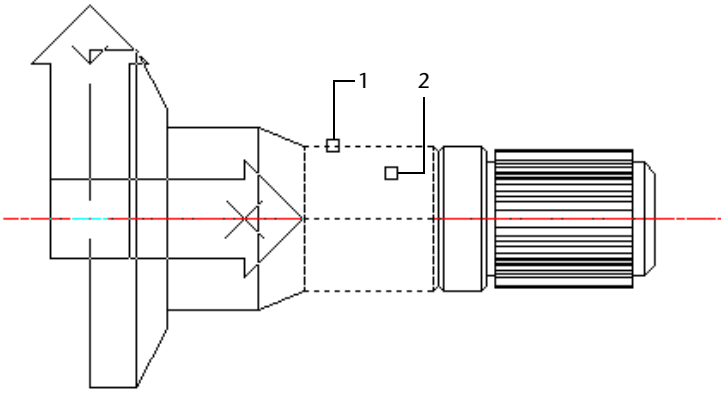
The chamfer is created on the shaft end.

In the next step, you use the 3D Shaft Generator dialog box to insert a groove for a retaining ring on a section of the shaft. This groove will be used for positioning a bearing.

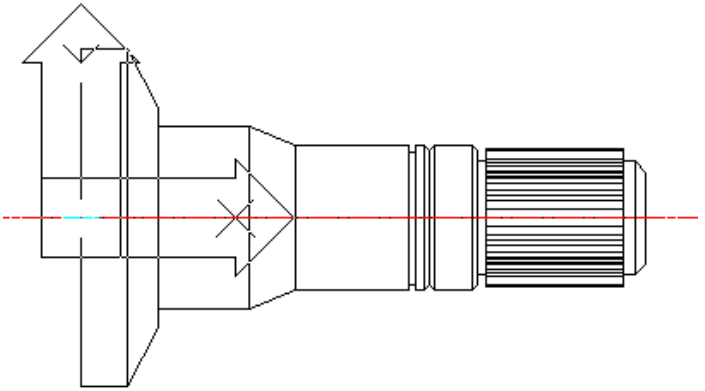
To insert a groove on a shaft segment

1 Choose the Groove icon, and respond to the prompts as follows:

Select cylinder or cone: *Select the third cylindrical section (1)*
Select position on cylinder or cone [Line/Plane]: *Specify the point (2)*
Specify direction or [Flip/Accept] <Accept>: *Press* ENTER
Enter distance from base plane [Associate to/Equation assistant] <11.4>: *Enter* **25**
Specify length or [Associate to/Equation assistant] <5>: *Enter* **1.5**
Specify diameter <50>: *Enter* **29**



The groove is inserted and displayed on the shaft.



The edits to the shaft are complete.

2 Close the 3D Shaft Generator dialog box.

Save your file.

Now that you have finished the shaft, you add standard parts to it.

Adding Standard Parts to Shafts

In Mechanical Desktop Power Pack, standard parts such as bearings, seals, circlips, keys, adjusting rings, and undercuts are available. You can select and insert these standard parts using the shaft generator.

When you insert a standard part on a shaft, the standard part is not consumed by the shaft. You are actually building an assembly. An icon for each new part is displayed in the Browser under the assembly icon.

To add a bearing to a shaft

1 Use AMSHAFT3D TO add a standard part to the shaft.

Menu Content 3D ➤ Shaft Generator

Command AMSHAFT3D

2 On the command line, respond to the prompts as follows:

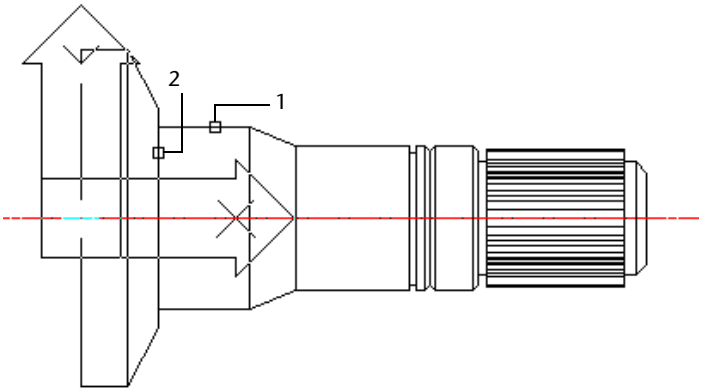
Specify start point or [Existing shaft]: *Enter* **E**
Select shaft: *Select the shaft*

3 The 3D Shaft Generator dialog box is displayed. Choose Std. Parts.

4 In the Select a Part dialog box, choose Roller Bearings ➤ Radial ➤ ISO 355.

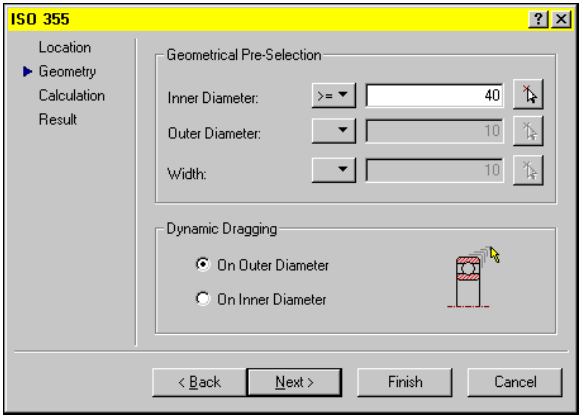
Respond to the prompts as follows:

Select cylindrical face: *Select a point on the third shaft segment (1)*
Specify location on cylindrical face [Line/Plane]:
Select the left plane of the third shaft segment (2)
Enter distance from base plane [Associate to/Equation assistant] <0>:
Press ENTER
Choose insertion direction [Flip/Accept] <Accept>: *Press* ENTER



5 In the ISO 355 dialog box, verify Geometry is selected and specify:

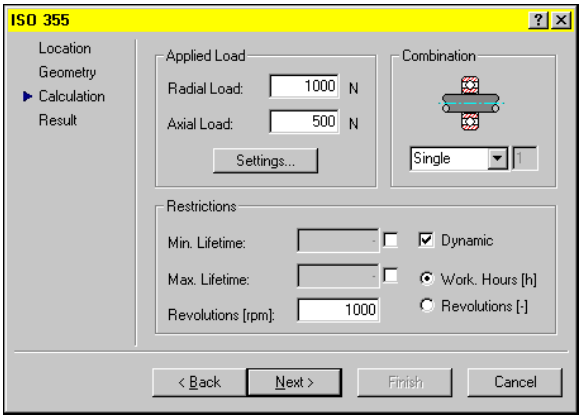
Inner Diameter: 40



Choose Next to continue.

You use the ISO 355 dialog box for the bearing calculation.

6 In the ISO 355 dialog box, verify Calculation is selected and specify the values as shown below.

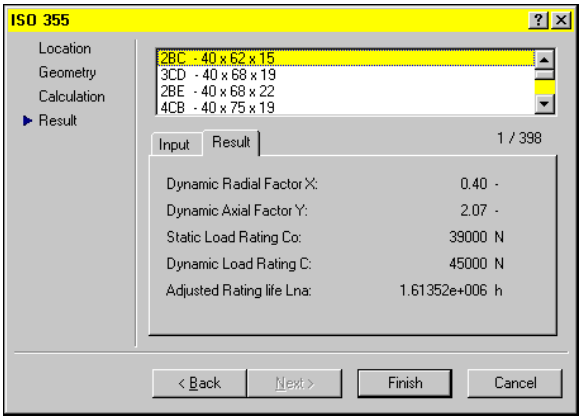


By choosing dynamic calculation, Mechanical Desktop is calculating the adjusted rating life of the bearing.

Choose Next.

The possible bearings are calculated and the results are displayed in the ISO 355 dialog box in Result. Two tabs are provided so you can check both your input and results.

7 In the ISO 355 dialog box, select the Result tab, and then select 2BC - 40 x 62 x 15.



Choose Finish.

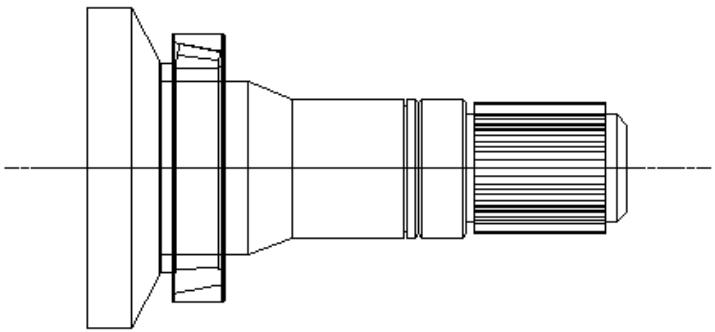
Use dynamic dragging to size the bearing on the screen.

8 Respond to the prompt as follows:

Drag size [Dialog] <10>:

Drag the bearing, and click when 2BC - 40 x 62 x 15 is displayed in the status bar

The bearing is inserted and your drawing should look like this.

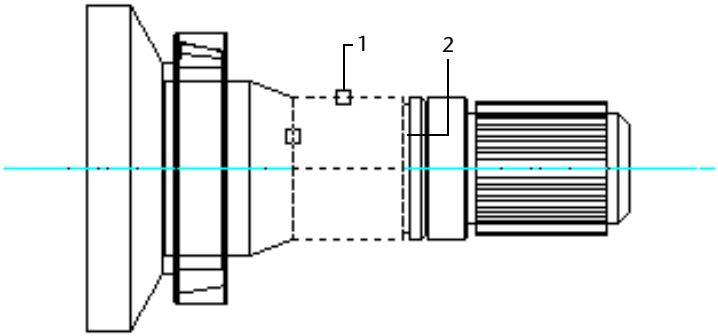


Insert a second bearing, starting from the groove at the left of the fifth segment of the shaft.

To add the second bearing

1. In the 3D Shaft Generator dialog box, choose Std. Parts.
2. In the Select a Part dialog box, choose Roller Bearings ➤ Radial ➤ ISO 355.
3. Respond to the prompts as follows:

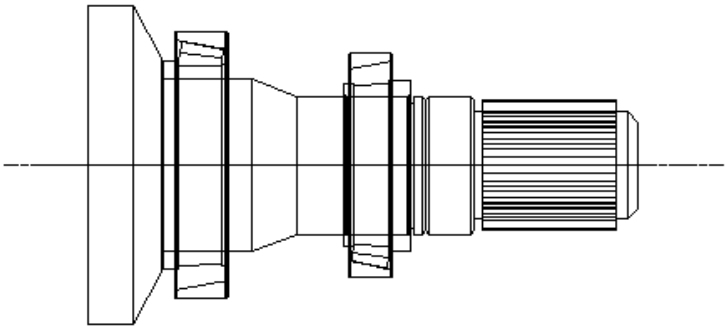
   Select cylindrical face: *Select a point on the fifth shaft segment (1)*
   Specify location on cylindrical face [Line/Plane]:
   Select the right plane of the fifth shaft segment (2)
   Enter distance from base plane [Associate to/Equation assistant] <25>:
   Press ENTER
   Choose insertion direction [Flip/Accept]: *Enter* **f**



4. Use the default values for the calculation, and select the bearing ISO 355 2BD - 32 x 52 x 14.
5. Place the bearing by responding to the prompt as follows:

   Drag size [Dialog] <10>:
   Drag the bearing, and click when 2BC - 32 x 52 x 14 is displayed in the status bar

Close the 3D Shaft Generator dialog box. Your drawing should look like this.



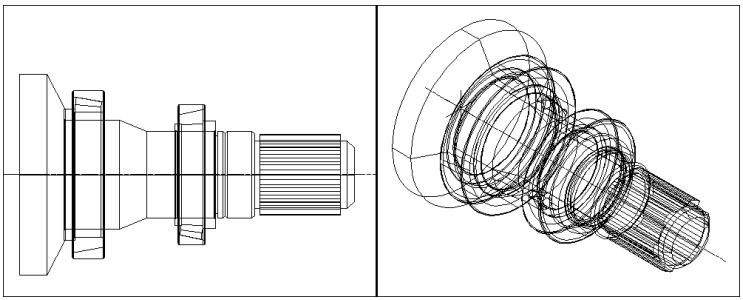
Save your file.

Displaying and Shading 3D Views

You can split your screen to display the front and isometric views of the shaft in separate viewports. You can activate a viewport by clicking inside its border.

To display front and isometric views in a split screen

1 On the command line, enter **2**, and press ENTER.



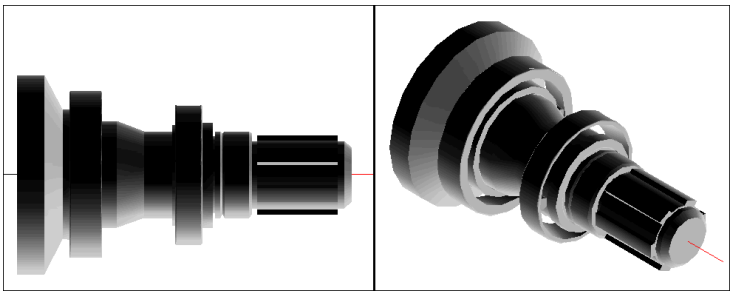
Your drawing area is split to display front and isometric views in separate viewports.

2 Click a viewport to activate it.

3 Use AMDT_TOGGLE_SHADWIREF to add shading to the shaft.

Menu View ➤ Shade ➤ Gouraud Shaded

4 Activate the second viewport and toggle to shading.



For better visualization, you can change the color properties of a single part, such as the bearing.

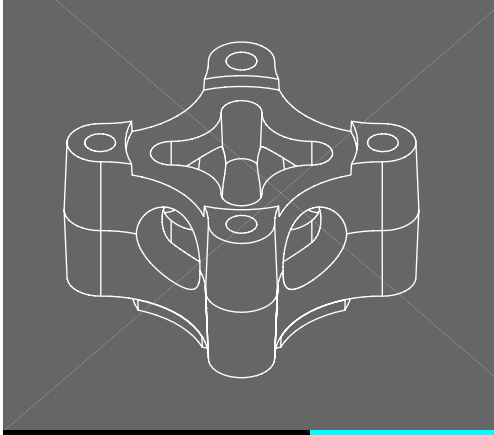
To change the color of a single part on a shaft

1. In the Desktop Browser, right-click a bearing and choose Properties ➤ Color.
2. Select a different color.

   The bearing is displayed in a different color on the shaft. Try changing the colors of the other parts on the shaft.

   Save your file.

Calculating Stress on 3D Parts



In This Chapter

24

- New in this tutorial
- Using 3D FEA calculations
- Defining loads
- Calculating and displaying the result

Autodesk® Mechanical Desktop® 6 Power Pack includes a feature called 3D finite element analysis (FEA). FEA is used to calculate deformation and stress conditions on 3D parts. The 3D FEA calculations feature is a reliable tool that helps you meet the demands of today's sophisticated mechanical engineering.

In this tutorial, you calculate stress on a 3D part using 3D FEA. You create the mesh required to display the results of the calculations, and then you define the loads and make the calculations. When you finish, the calculations are displayed in tables.

Run Mechanical Desktop with the power pack to perform this tutorial.

Key Terms

Term	Definition
distributed force	A force that is spread over an area.
FEA	Finite element analysis. A calculation routine, or method. Calculates stress and deformation in a plane for plates with a given thickness, or in a cross section with individual forces, stretching loads, and fixed and/or moveable supports. The FEA routine uses its own layer group for input and output.
fixed support	A support that is fixed to a part and cannot be moved.
load	Forces and moments that act on a part.
mesh	Graphical representation of a mesh pattern on a model, required for FEA calculation and display of the result.
result	Results of FEA calculations. Result is displayed in a table, and as surface isoareas on a model.
stress	Force or pressure on a part. Stress is force per area.
surface isoarea	Graphic representation of a stress calculation result displayed on the surface of a model.

Tutorial at a Glance

The 3D FEA calculations feature is a simple tool that eliminates the need for the complex programs and calculations previously required to perform stress and deformation calculations. In this tutorial, you learn to use 3D FEA for calculations as you

- Start a finite element analysis.
- Define stress loads.
- Generate a mesh.
- Calculate and display the result.

Basic Concepts of 3D FEA

In Mechanical Desktop Power Pack, you use the FEA Calculation 3D dialog box options to choose the

- Material of your part
- Revolution direction
- Type and quantity of supports and loads

The FEA calculations are dependent upon the definitions you enter for loads and boundary conditions.

You consider how to reproduce the reality in your model, and how to define the boundary conditions to achieve the best result. The more precise your model is, the more useful the results are.

The FEA procedure uses a separate layer group for input and output.

Using 3D FEA Calculations

To begin calculating the stress conditions on your model, you open the FEA Calculation 3D dialog box. You work in this dialog box to choose the types and quantity of loads and supports, and to define the loads and boundary conditions.

Before you perform the calculation, you create a mesh on your 3D part. This mesh is required to display the results of the calculation. The calculation is displayed on your model using isoareas that define the limits of the stresses and deformations. Tables are automatically created to define the calculations for each isoarea.

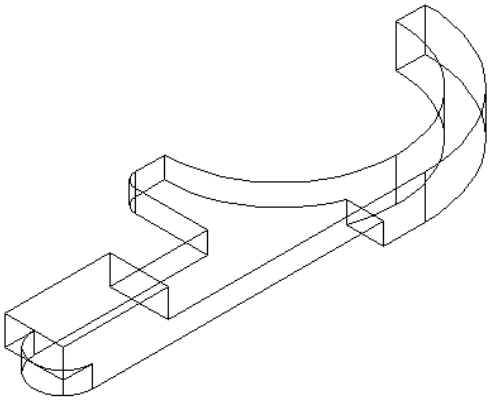
Performing Finite Element Analyses

In the following exercise, you work with a lever that is loaded with a mass and bolted at two faces. You reproduce these two points on the model as two face supports, and define the mass as a face load at the front rectangular face.

Open the file *md_ex03.dwg* in the *desktop\tutorial* folder.

NOTE Back up the tutorial drawing files so you still have the original files if you make a mistake. See "Backing up Tutorial Drawing Files" on page 40.

The drawing contains a 3D part, which is the basis for your calculations.



The first step is to open the FEA Calculations 3D dialog box.

To start a finite element analysis

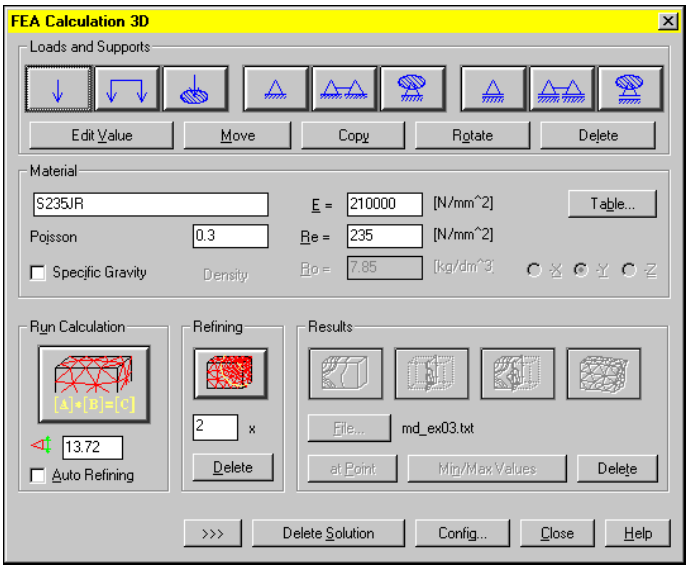
1 Use AMFEA3D to perform the FEA calculation.

Menu Content 3D ➤ Calculations ➤ FEA

Respond to the prompt as follows:

Select 3D-Body: *Select the 3D part*

The FEA Calculation 3D dialog box is displayed.



NOTE If the FEA Calculation 3D dialog box hides your drawing, move the dialog box to another position on the screen.

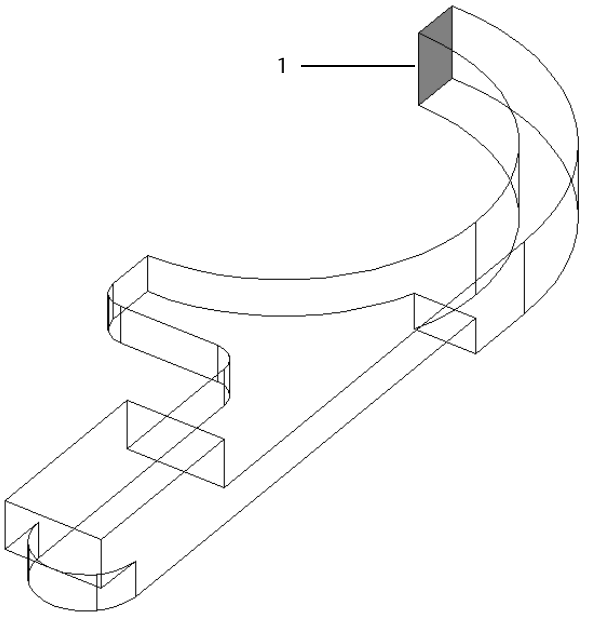
Defining Supports and Forces

Next, you define the supports and forces that act on your part. These definitions are used for the FEA calculations.

To define supports and forces on a part

1 In the FEA Calculation 3D dialog box, choose the Face Support icon and respond to the prompts as follows:

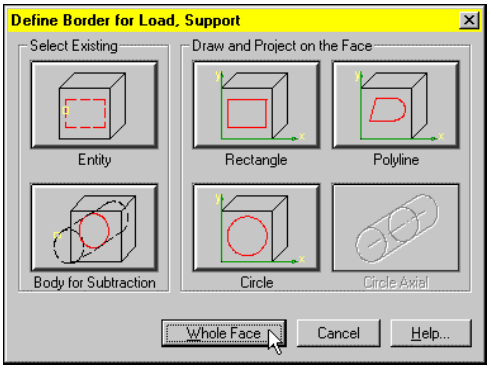
Select a surface: *Click the edge (1) of the surface*
Specify face [Accept/Next] <Accept>:
Press N *to cycle to the surface you selected, then press* ENTER



NOTE You may prefer to turn OSNAP off before you create and constrain the work point. Click the OSNAP button at the bottom of your screen.

The Define Border for Load, Support dialog box is displayed.

2 In the Define Border for Load, Support dialog box, choose Whole Face.



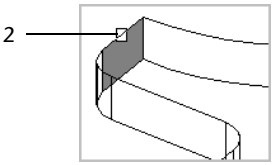
Respond to the prompts as follows:

Define insertion point for main symbol:
Specify point or [Dialog]:
Specify a point, other than an edge, on the selected face

NOTE If the support does not act on the whole face, you can define an area using the different options. See the Help for more information.

3 In the FEA Calculation dialog box, choose the Face Support icon again, and respond to the prompts as follows:

Select a surface: *Click the edge (2) of the surface*
Specify face [Accept/Next] <Accept>:
Press N *to cycle to the surface you selected, then press* ENTER



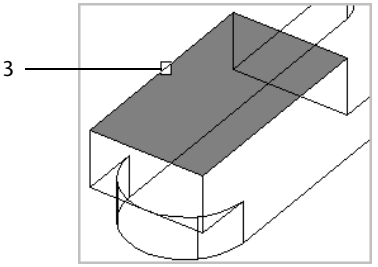
The Define Border for Load, Support dialog box is displayed.

4 In the Define Border for Load, Support dialog box, choose Whole Face, and respond to the prompts as follows:

Define insertion point for main symbol:
Specify point or [Dialog]:
Specify a point, other than an edge, on the selected face

5 In the FEA Calculation dialog box, choose the Face Force icon and respond to the prompts as follows:

Select a surface: *Click the edge (3) of the surface*
Specify face [Accept/Next] <Accept>:
Press N *to cycle to the surface you selected, then press* ENTER



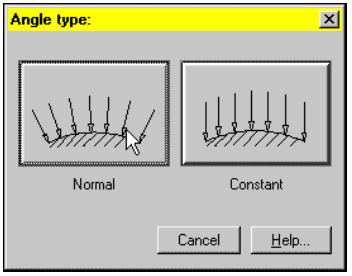
The Define Border for Load, Support dialog box is displayed.

6 In the Define Border for Load, Support dialog box, choose Whole Face, and respond to the prompts as follows:

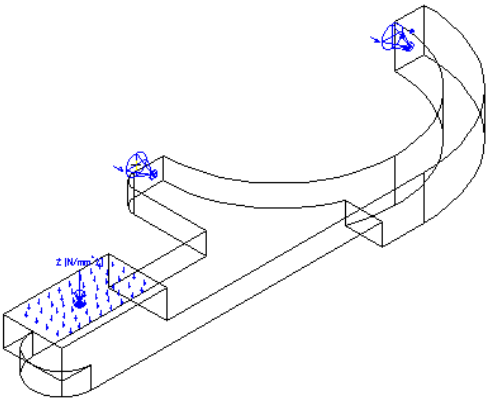
Define insertion point for main symbol:
Specify point or [Dialog]: *Specify a point, other than edge, on the selected face*

The Angle type dialog box is displayed. Choose Normal, and respond to the prompt as follows:

Specify value of load <10 (N/mm^2)>: Enter **2**



You have the defined the border conditions for two supports and one force. Your drawing should look like this:



Calculating and Displaying the Result

Continue in the FEA Calculation 3D dialog box to perform the calculations and display the results.

Before you can calculate and display the result, you need to generate a mesh.

To generate a mesh

1. In the FEA Calculation 3D dialog box, under Run Calculation, turn on Auto Refining.

2. Choose the Run Calculation icon.

   The Working dialog box is displayed during the automatic calculation of the mesh. When the calculation is finished, a mesh is displayed on your model.

   Next, define the placement of the mesh.

3. Respond to the prompt as follows:

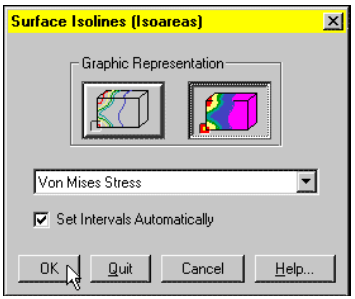
   Specify base point or displacement <in boundary>: *Press* ENTER

   This completes the mesh.

Continue in the FEA Calculation 3D dialog box to calculate surface isoareas and deformation.

4 Under Results, choose the Isolines (Isoareas) icon.

5 In the Surface Isolines (Isoareas) dialog box, choose the Isoareas button, and choose OK.



Your model with isoareas is displayed on the screen beside the model with mesh.

Respond to the prompts as follows:

Specify base point or displacement <in boundary>: *Press* ENTER
Specify insertion point or [Paper space]:
Specify a location for the table to the left of the isoareas display

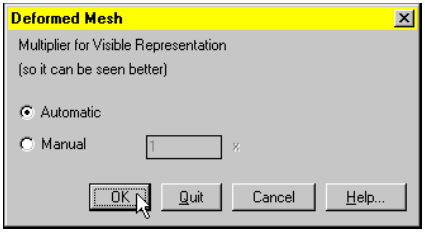
---

NOTE If necessary, zoom to fit the entire display to your screen before you place the second table.

---

6 In the Surface Isolines (Isoareas) dialog box, choose the Deformed Mesh icon.

7 In the Deformed Mesh dialog box, turn on Automatic, and choose OK.

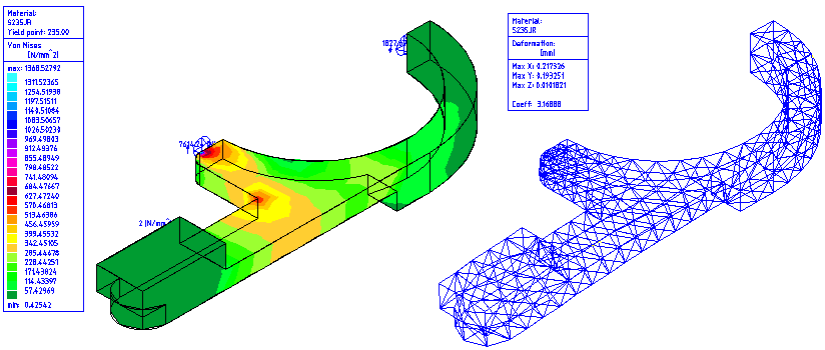


Respond to the prompts as follows:

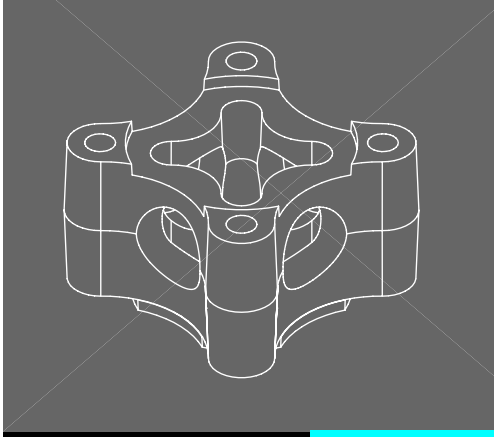
Specify base point or displacement <in boundary>: *Enter* **150,150**
Specify second point of displacement: *Press* ENTER
Specify insertion point or [Paper space]:
Specify a suitable location for the table near the mesh display

8 Close the FEA Calculation 3D dialog box.

The calculations are finished, and the results are displayed.



Toolbar Icons



In This Appendix

A

- Desktop Tools toolbar icons
- Part Modeling toolbar icons
- Toolbody Modeling toolbar icons
- Assembly Modeling toolbar icons
- Surface Modeling toolbar icons
- Scene toolbar icons
- Drawing Layout toolbar icons
- Mechanical View toolbar icons

Use this appendix as a guide to using the Autodesk® Mechanical Desktop® toolbar icons. For an overview of the toolbars and the Mechanical Desktop interface, refer to “Mechanical Desktop Interface” on page 17 in chapter 3.

Desktop Tools

If you are working in the Part Modeling environment, the Desktop Tools toolbar contains three icons that activate the Part Modeling, Toolbody Modeling, and Drawing Layout toolbars.

Part Modeling

Toolbody Modeling

Drawing Layout

If you are working in the Assembly Modeling environment, the Desktop Tools toolbar contains four icons that activate the Part Modeling, Assembly Modeling, Scene, and Drawing Layout toolbars.

Part Modeling

Assembly Modeling

Scene

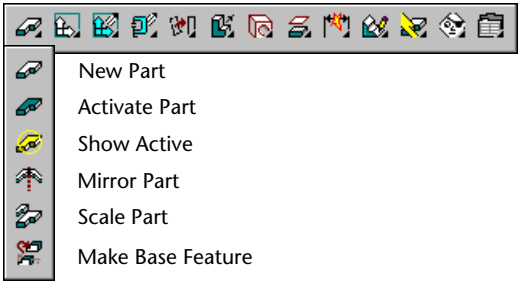
Drawing Layout

Part Modeling

The Part Modeling toolbar provides the tools you need for creating and modifying parts.

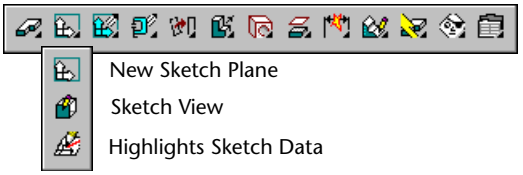
New Part Flyout
New Sketch Plane Flyout
Launches 2D Sketch Toolbar
Launches 2D Constraints Toolbar
Profile a Sketch Flyout
Sketched Features Flyout
Placed Features Flyout
Work Features Flyout
Power Dimensioning Flyout
Edit Feature Flyout
Update Part Flyout
Part Visibility Flyout
Options Flyout

Part Modeling ➤ New Part

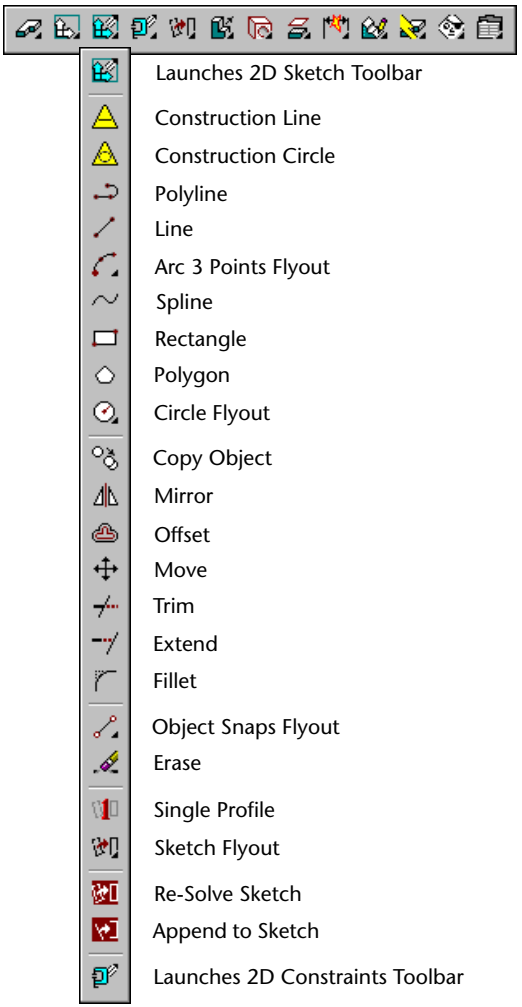


New Part
Activate Part
Show Active
Mirror Part
Scale Part
Make Base Feature

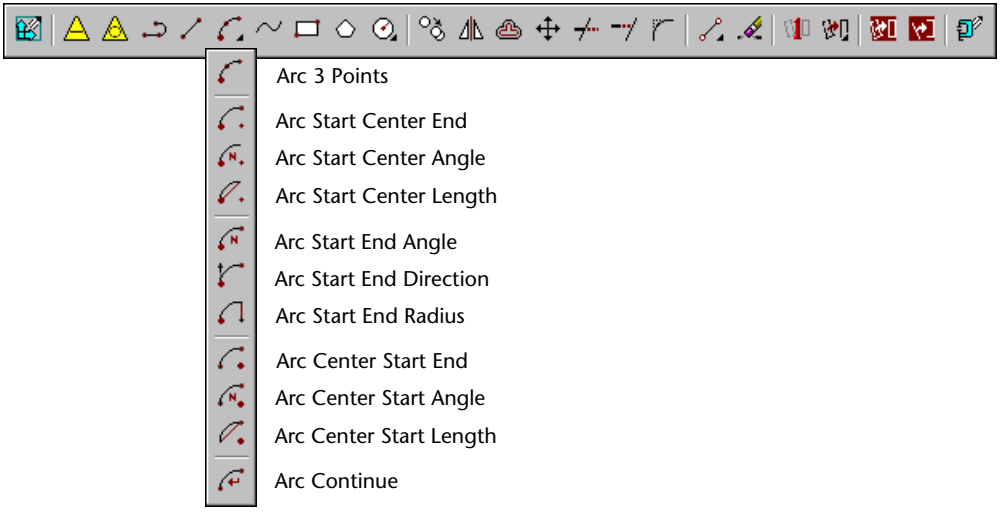
Part Modeling ➤ New Sketch Plane



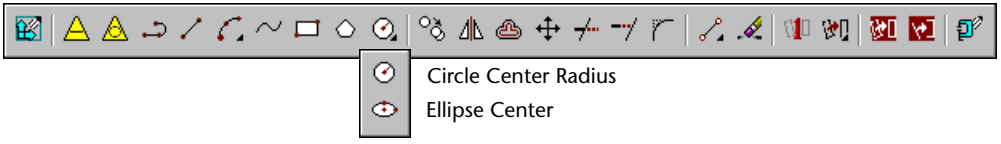
Part Modeling ➤ 2D Sketching



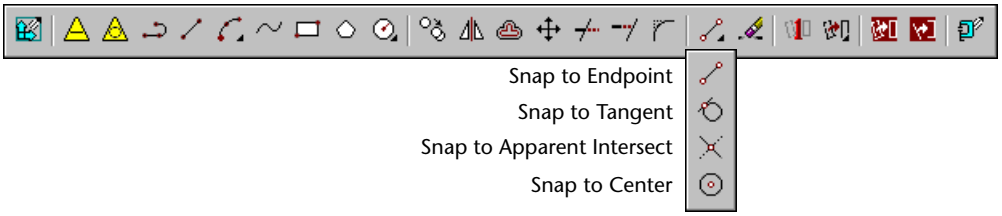
Part Modeling ➤ 2D Sketching ➤ Arc 3 Points



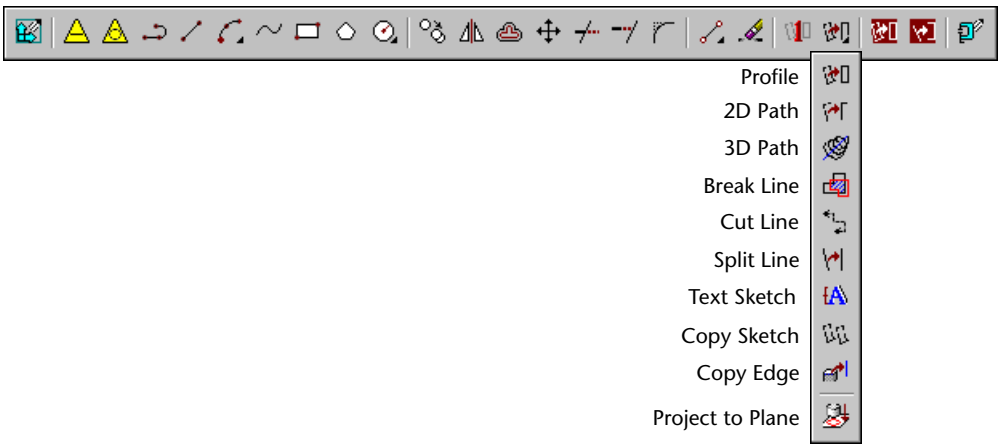
Part Modeling ➤ 2D Sketching ➤ Circle



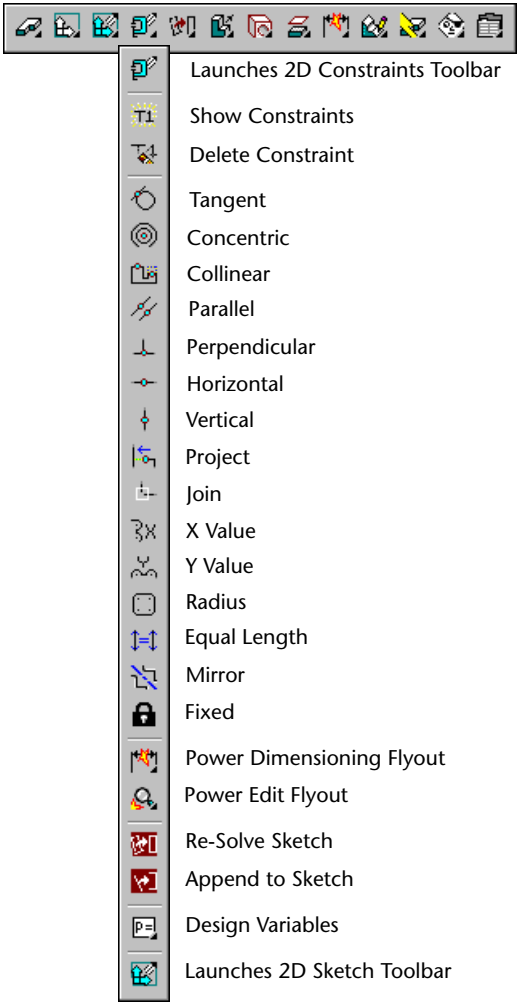
Part Modeling ➤ 2D Sketching ➤ Object Snaps



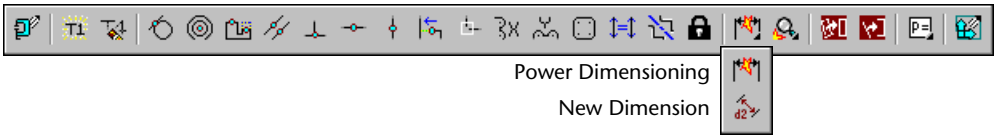
Part Modeling ➤ 2D Sketching ➤ Sketch



Part Modeling ➤ 2D Constraints

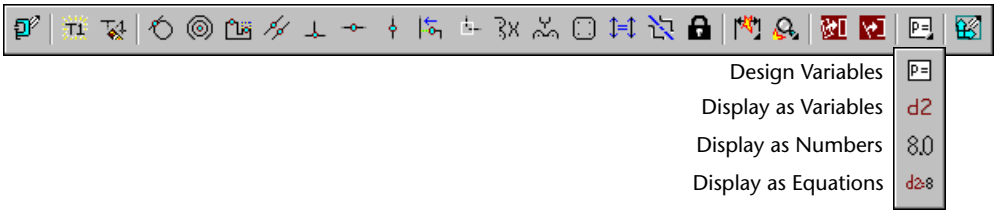


Part Modeling ➤ 2D Constraints ➤ Power Dimensioning

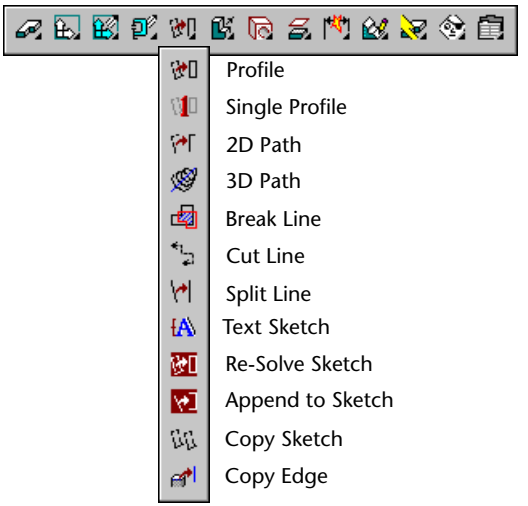


Part Modeling ➤ 2D Constraints ➤ Power Edit

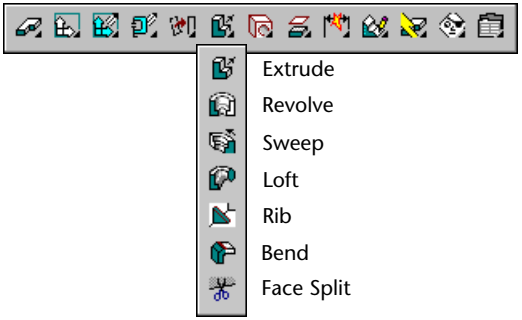
Part Modeling ➤ 2D Constraints ➤ Design Variables



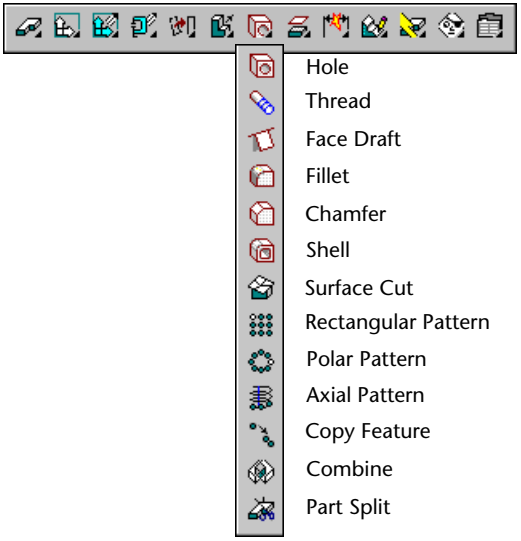
Part Modeling ➤ Profile a Sketch



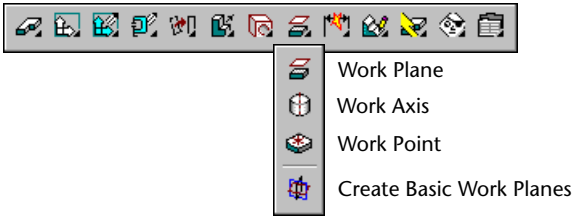
Part Modeling ➤ Sketched Features



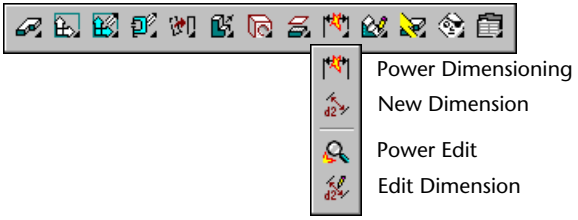
Part Modeling ➤ Placed Features



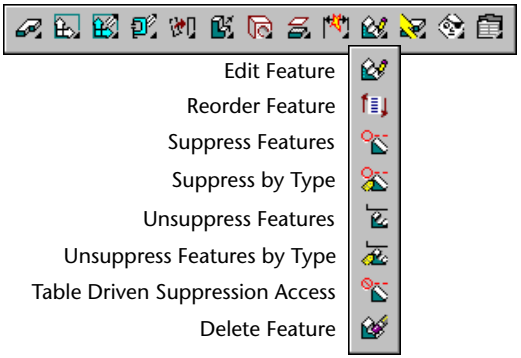
Part Modeling ➤ Work Features



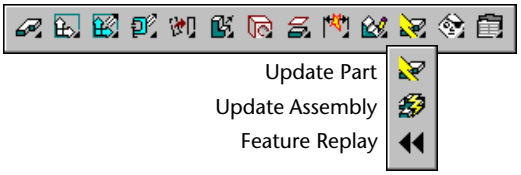
Part Modeling ➤ Power Dimensioning



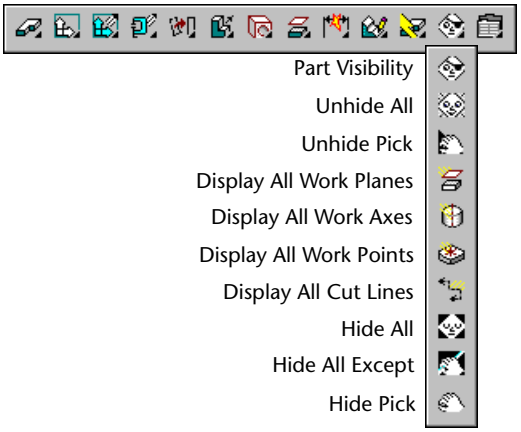
Part Modeling ➤ Edit Feature



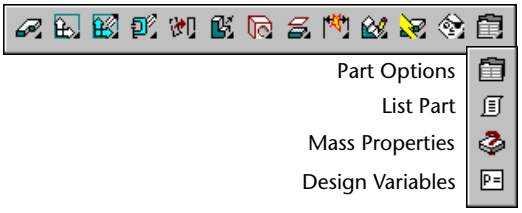
Part Modeling ➤ Update Part



Part Modeling ➤ Part Visibility



Part Modeling ➤ Options

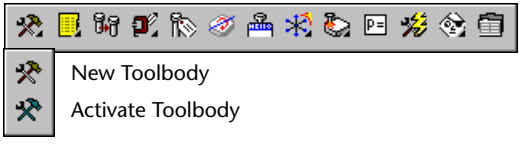


Toolbody Modeling

In the Part Modeling environment, the Toolbody Modeling toolbar contains the tools you need to create combined parts.

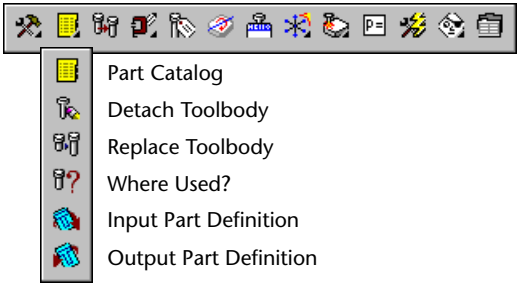
New Toolbody Flyout
Part Catalog Flyout
Instance
3D Toolbody Constraints Flyout
Assign Attributes
DOF Visibility
Rename Part
3D Manipulator Flyout
Check Interference Flyout
Design Variables
Update Positioning
Toolbody Visibility Flyout
Toolbody Options

Toolbody Modeling ➤ New Toolbody



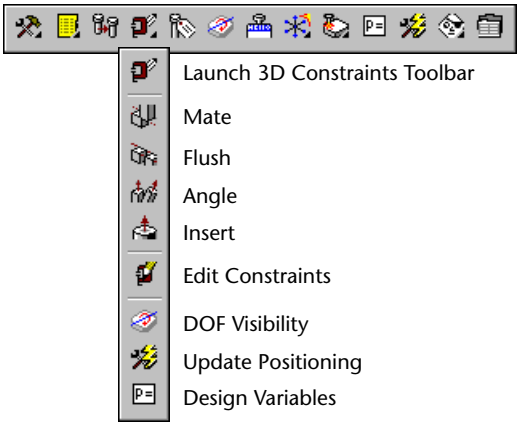
New Toolbody
Activate Toolbody

Toolbody Modeling ➤ Part Catalog

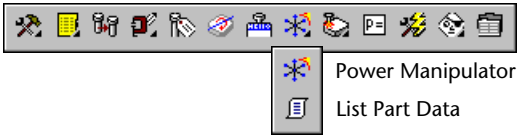


Part Catalog
Detach Toolbody
Replace Toolbody
Where Used?
Input Part Definition
Output Part Definition

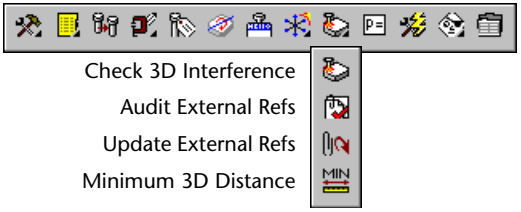
Toolbody Modeling ➤ 3D Toolbody Constraints



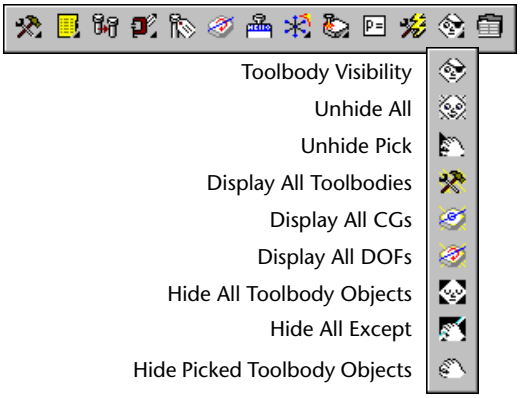
Toolbody Modeling ➤ Power Manipulator



Toolbody Modeling ➤ Check Interference

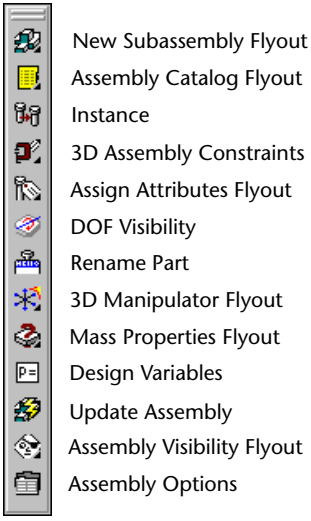


Toolbody Modeling ➤ Toolbody Visibility

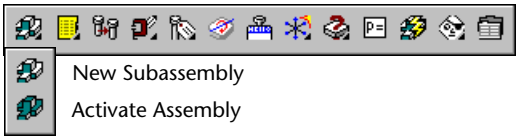


Assembly Modeling

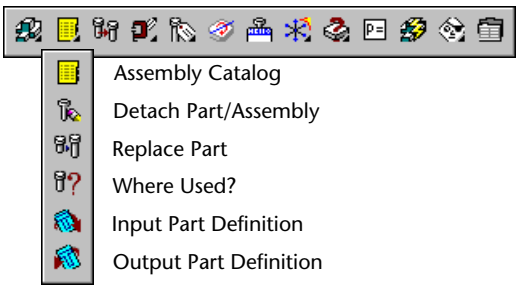
In the Assembly Modeling environment, the Assembly Modeling toolbar provides the tools you need to create and modify assemblies and subassemblies.



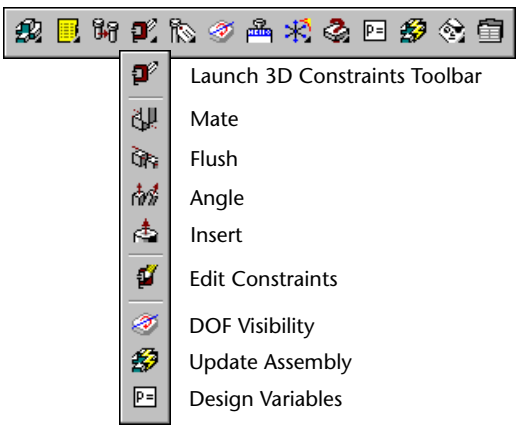
Assembly Modeling ➤ New Subassembly



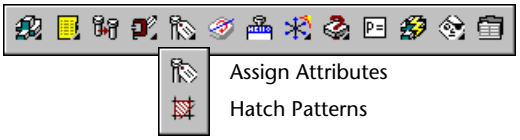
Assembly Modeling ➤ Assembly Catalog



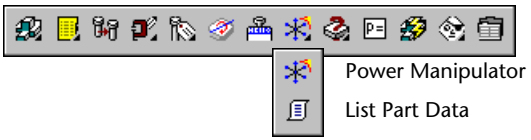
Assembly Modeling ➤ 3D Assembly Constraints



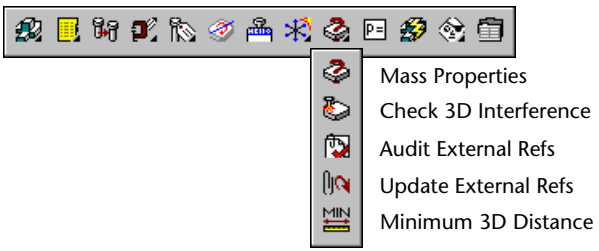
Assembly Modeling ➤ Assign Attributes



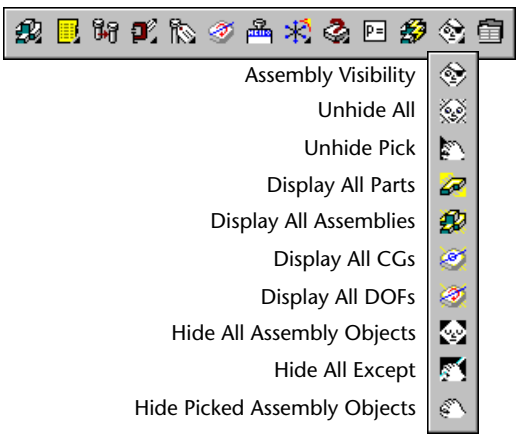
Assembly Modeling ➤ Power Manipulator



Assembly Modeling ➤ Mass Properties



Assembly Modeling ➤ Assembly Visibility

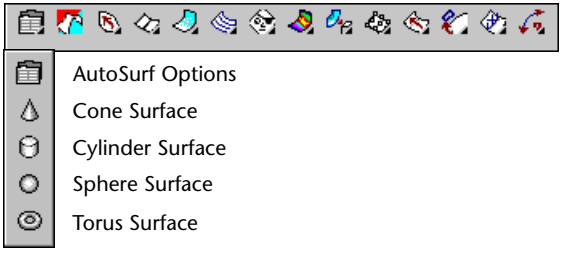


Surface Modeling

The Surface Modeling toolbar provides the tools you need to create and modify 3D wireframe surfaces. To activate the Surface Modeling toolbar, choose Surface ➤ Launch Toolbar.

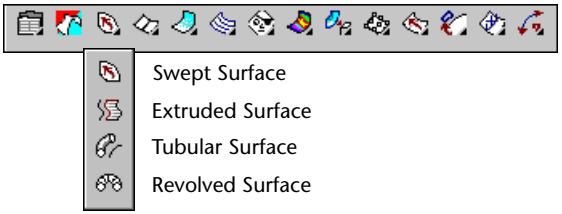
AutoSurf Options Flyout
Toggles Keep Original
Swept Surface Flyout
Loft U Surface Flyout
Blended Surface Flyout
Flow Wires Flyout
Object Visibility Flyout
Surface Analysis Flyout
Stitches Surfaces Flyout
Grip Point Placement Flyout
Lengthen Surface Flyout
Extract Surface Loop Flyout
Edit Augmented Line Flyout
Wire Direction Flyout

Surface Modeling ➤ AutoSurf Options

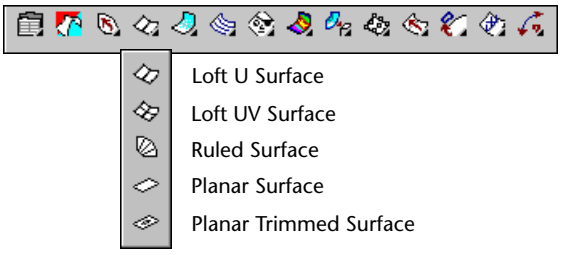


AutoSurf Options
Cone Surface
Cylinder Surface
Sphere Surface
Torus Surface

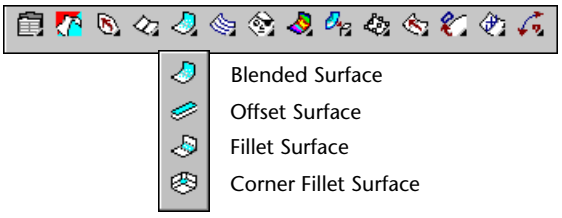
Surface Modeling ➤ Swept Surface



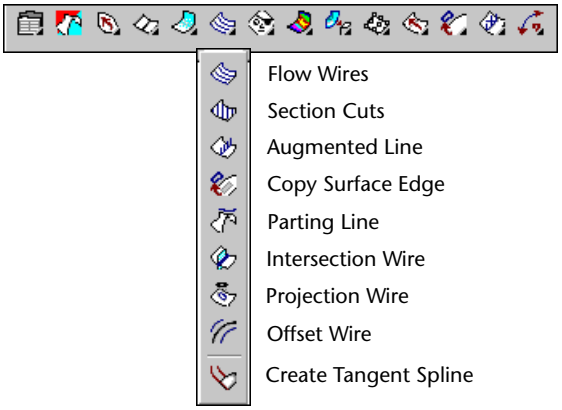
Surface Modeling ➤ Loft U Surface



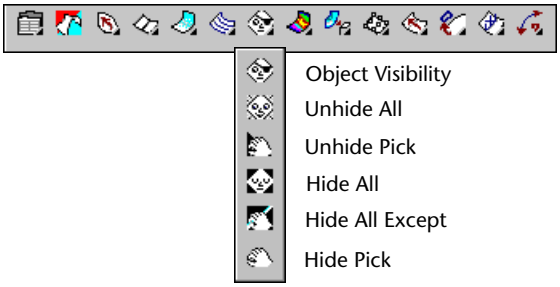
Surface Modeling ➤ Blended Surface



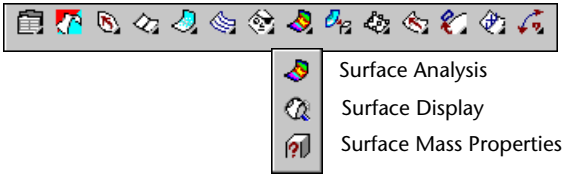
Surface Modeling ➤ Flow Wires



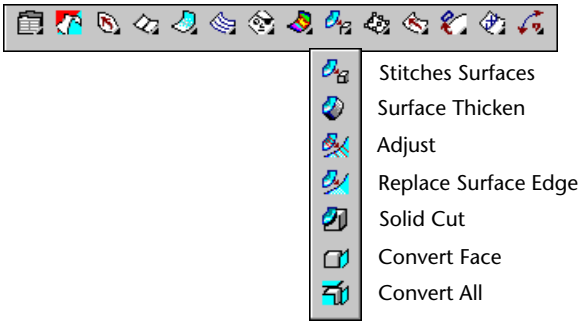
Surface Modeling ➤ Object Visibility



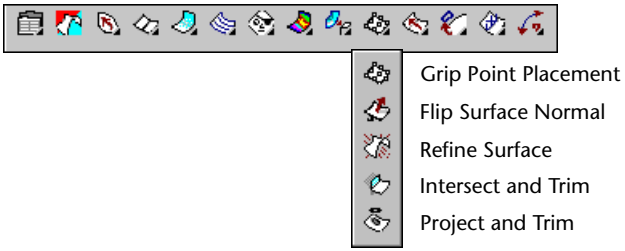
Surface Modeling ➤ Surface Display



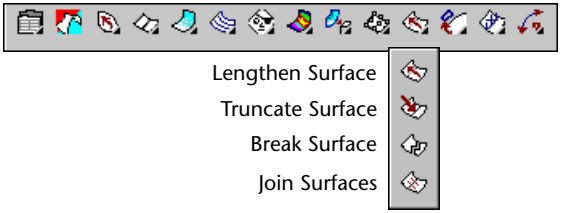
Surface Modeling ➤ Stitches Surfaces



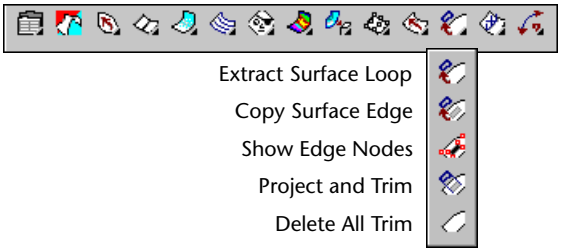
Surface Modeling ➤ Grip Point Placement



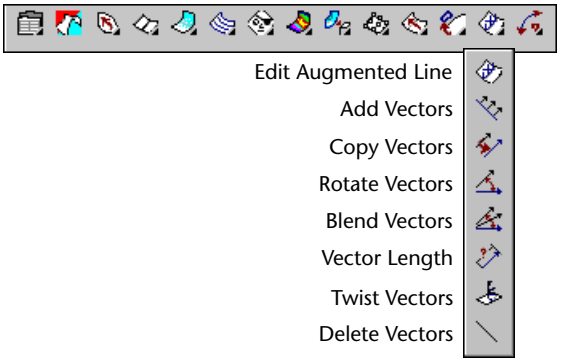
Surface Modeling ➤ Lengthen Surface



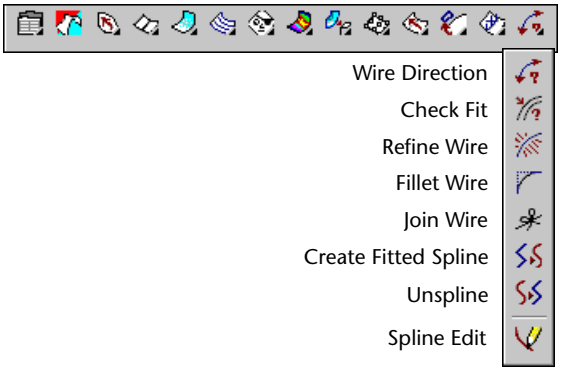
Surface Modeling ➤ Extract Surface Loop



Surface Modeling ➤ Edit Augmented Line



Surface Modeling ➤ Wire Direction

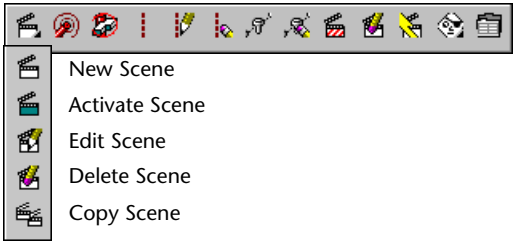


Scene

In the Assembly Modeling environment, the Scene toolbar provides the tools you need to create, modify, and manage scenes.

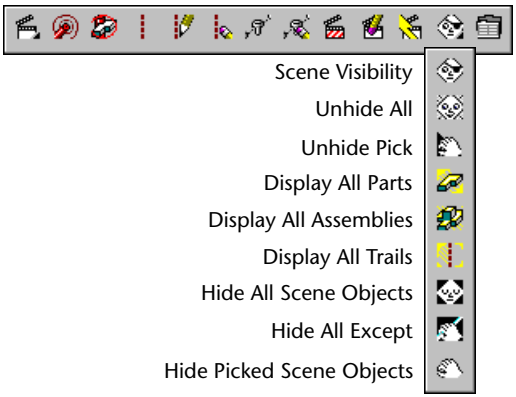
New Scene Flyout
Scene Explosion Factor
Part or Subassembly Explosion Factor
New Tweak
Edit Tweak
Delete Tweak
New Trail
Edit Trail
Delete Trail
Suppress Sectioning
Lock Scene Position
Update Scene
Scene Visibility Flyout
Scene Options

Scene ➤ New Scene



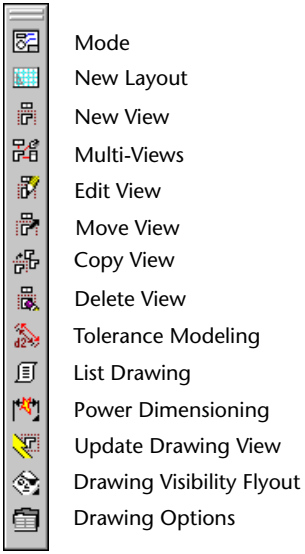
New Scene
Activate Scene
Edit Scene
Delete Scene
Copy Scene

Scene ➤ Scene Visibility

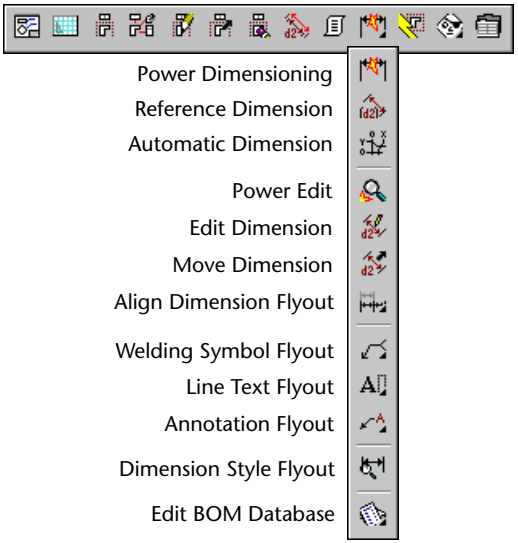


Drawing Layout

The Drawing Layout toolbar provides the tools you need to create, modify, and annotate drawing views and layouts.

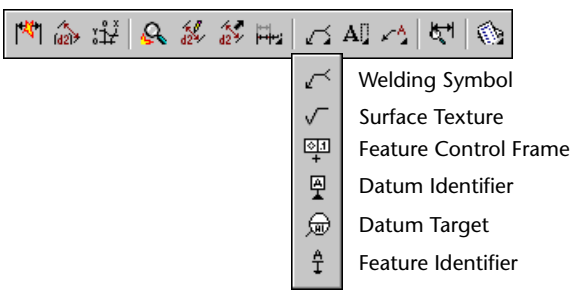


Drawing Layout ➤ Power Dimensioning

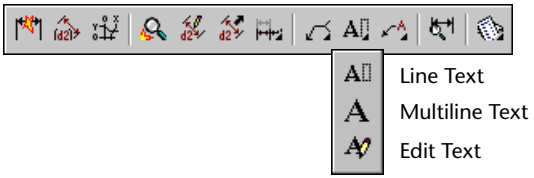


Drawing Layout ➤ Power Dimensioning ➤ Edit Format

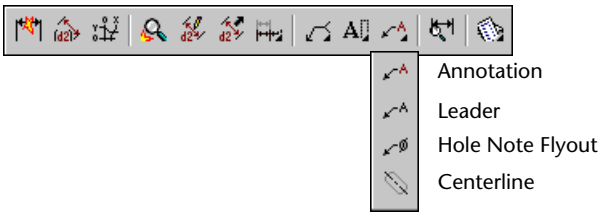
Drawing Layout ➤ Power Dimensioning ➤ Welding Symbol



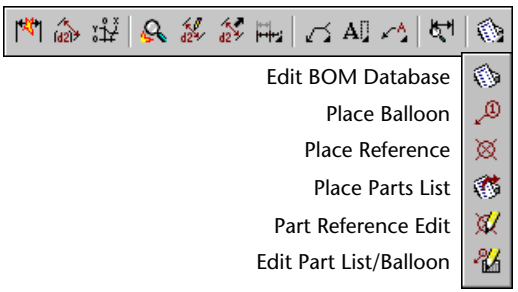
Drawing Layout ➤ Power Dimensioning ➤ Line Text



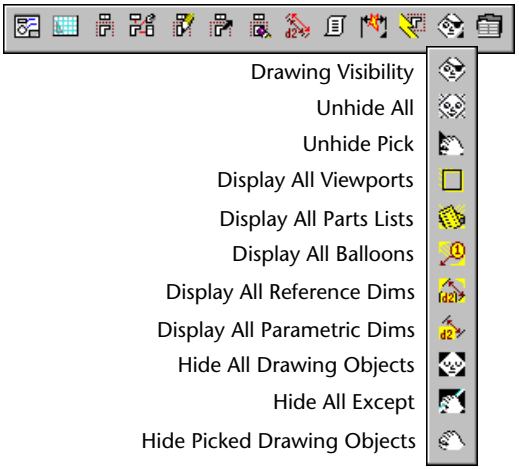
Drawing Layout ➤ Power Dimensioning ➤ Annotation



Drawing Layout ➤ Power Dimensioning ➤ Edit BOM Database



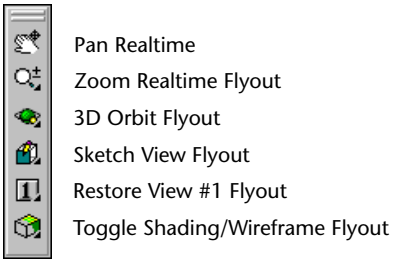
Drawing Layout ➤ Drawing Visibility



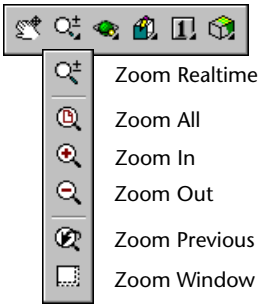
Mechanical View

The Mechanical View toolbar provides the tools you need to manage the view of your design, control 3D viewports, and create rendered views of your parts and assemblies.

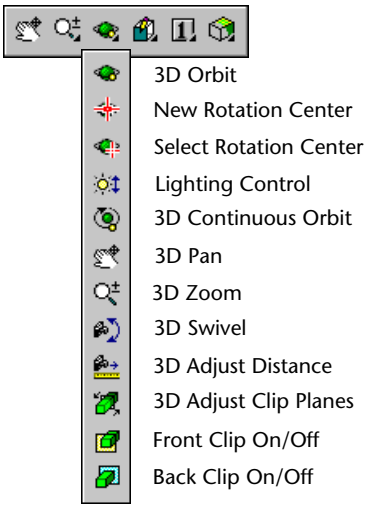
If the Mechanical View toolbar is not visible on your desktop, choose View ➤ Toolbars ➤ Mechanical View. You can also right-click any toolbar, and choose Mechanical View from the list.



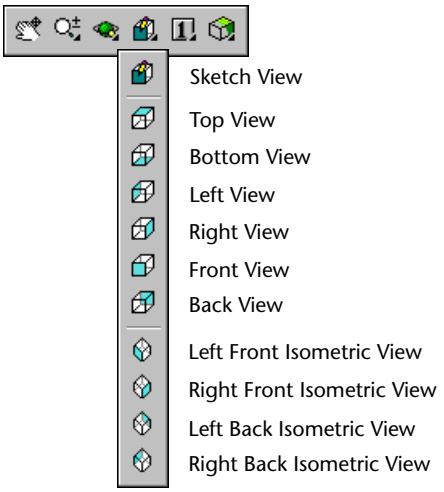
Mechanical View ➤ Zoom Realtime



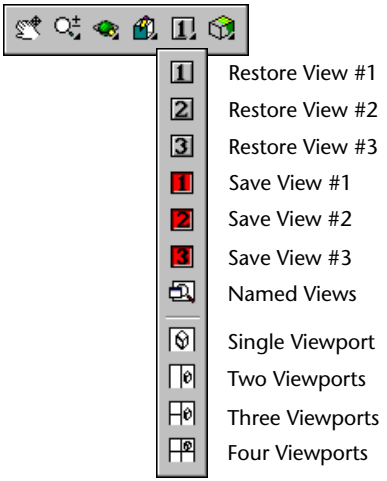
Mechanical View ➤ 3D Orbit



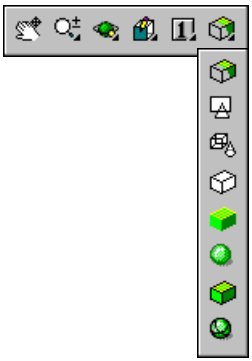
Mechanical View ➤ Sketch View



Mechanical View ➤ Restore View #1



Mechanical View ➤ Toggle Shading/Wireframe



Toggle Shading/Wireframe
2D Wireframe
3D Wireframe
Hidden
Flat Shaded
Gouraud Shaded
Flat Shaded, Edges On
Gouraud Shaded, Edges On

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