AutoCAD Mechanical 2010

User's Guide

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Contents

Overview

Part I provides information for getting started with your AutoCAD® Mechanical software.

It includes an overview of the product capabilities, a summary of commands with their toolbuttons and descriptions, and a summary of new and revised commands in this release of AutoCAD Mechanical.

In addition, Part I includes information about methods to access commands, AutoCAD Mechanical Help, and product support and training resources.

About AutoCAD Mechanical

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This chapter provides information about the AutoCAD® Mechanical software application. It describes the software package, the basic design features in the software, and the methods for accessing commands.

A brief overview of the Help, along with information about where to find resources for product learning, training, and support are included.

AutoCAD Mechanical Software Package

AutoCAD Mechanical is a 2D mechanical design and drafting solution for engineers, designers, and detailers. Its intelligent production drawing and detailing features decrease the time required to create and change 2D production designs. AutoCAD Mechanical introduces many 3D concepts in a familiar 2D environment. It is powered by AutoCAD®, with its easy-to-use palette interface and time-saving xref functionality.

The AutoCAD Mechanical design software package includes both AutoCAD Mechanical and AutoCAD. You can use one Options dialog box to customize settings for both AutoCAD Mechanical and AutoCAD.

Leveraging Legacy Data

The tools for migrating legacy data are installed automatically when you install the AutoCAD Mechanical software. A separate utility tool is available for adding structure to legacy files after they are migrated.

The integrated Autodesk® IGES Translator for transferring and sharing of CAD data between CAD/CAM/CAE systems is installed along with the AutoCAD Mechanical product.

Newly generated files in AutoCAD Mechanical can be saved to a previous version so that you can run multiple versions of AutoCAD Mechanical within the same environment.

Starting AutoCAD Mechanical

You can start AutoCAD Mechanical by using one of the following procedures:

- Click Start on the task bar, and then choose Programs. Select Autodesk ➤ AutoCAD Mechanical 2010.
- On the desktop, double-click the AutoCAD Mechanical icon:

AutoCAD Mechanical Help

The Help in AutoCAD Mechanical provides information about AutoCAD Mechanical with the power pack.

The Help is formatted for easy navigation, and includes:

- Content organized by the major functional areas of AutoCAD Mechanical, with Concept, Reference, and Procedure pages for each functional area. Procedure pages provide step by step instructions on how to execute a given task. The linked Concept page provides background information about the procedure. The linked Reference pages contain information about all the commands and dialog boxes visited while performing the procedure.
- Specific information about each of the features in the program.
- Concepts and procedures for the new features in this release.
- A keyword index and search function.
- 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.
- Click the Help button within a dialog box.

Product Support and Training Resources

Be more productive with Autodesk software. Get trained at an Autodesk Authorized Training Center (ATC®) with hands-on, instructor-led classes to help you get the most from your Autodesk products. Enhance your productivity with proven training from over 1,400 ATC sites in more than 75 countries. For more information about Autodesk Authorized Training Centers, contact *atc.program@autodesk.com* or visit the online ATC locator at *www.autodesk.com/atc*.

Sources for product support are listed on the AutoCAD Mechanical Product Information Web page. From the AutoCAD Mechanical Web site at *http://www.autodesk.com/autocadmech*, navigate to the Support Knowledge Base. You can also navigate to the Community page, which contains links to various communities, including the AutoCAD Mechanical Discussion Group.

Design Features in AutoCAD Mechanical

This section provides an overview of the functionality in the AutoCAD Mechanical software, including numerous innovative 2D design features.

Mechanical Structure

Mechanical structure comprises a suite of 2D structure tools for organizing drawings and for reusing associative data. The capabilities of reuse in blocks and accessibility in layer groups are combined in mechanical structure. When you start the AutoCAD Mechanical application, the Mechanical structure environment is enabled by default. You can also work with it disabled.

The mechanical structure tools include:

- A browser interface for structured 2D mechanical design, where parts, assemblies, views, and folders containing associated data are organized, structured, and managed. Standard parts are automatically organized and managed in the browser. All components are accessible through the browser for many functions, and filters can be set to control the type and level of detail of information displayed.
- Folders in the browser are used for capturing elements of design for reuse. These elements provide all of the associative instancing benefits of components, but do not register as items in the live BOM database. They can contain geometry.
- All geometry remains selectable and editable at all times using familiar commands in open workflows. Workflows for structure can be bottom-up (recommended), middle-out (the most flexible and common workflow), and top-down (not the primary workflow).

Associative Design and Detailing

The browser is used to manage and reuse data in both the design and detailing drafting stages. Many functions can be performed in the browser, including the following:

- You can instance components and assemblies multiple times. The live BOM database in AutoCAD Mechanical keeps track of the quantity of each part or assembly used.
- Changes made to an associative instance of a part or assembly, associative component, assembly detailing view, or a standard part or feature are automatically reflected in the other instances.
- Folders, components, and individual views of components can be reused as needed. They maintain full associativity with each other.
- Annotation views can be created for components and assemblies to fully document the design. Changes made to geometry result in associative dimensions being updated to reflect the change.

External References for Mechanical Structure

External References for mechanical structure provides for the components of a drawing to be inserted as an external reference to multiple drawings. Conversely, multiple drawings can be attached as external references to a single drawing.

The following are the key benefits of external references for mechanical structure:

- Increased efficiency by allowing insertion of structure components from many drawings as external reference associatively for concurrent design.
- Reuse of parts from existing assembly drawings very quickly.
- Those involved in multiple design projects that reference the same drawing are able to obtain the most updated design from the externally reference component.
- Ability to set up design specific reference directories as libraries for different applications.

Associative 2D Hide

The 2D hide situation tool in AutoCAD Mechanical automates the process to accurately represent parts and features which are partially or completely hidden in drawing views. The following are some of the 2D hide benefits:

- Associative hide situations are managed in the browser.
- The underlying geometry is not altered when you create an associative hide situation.
- When geometry is hidden, AutoCAD Mechanical knows it is a component in the mechanical structure, and provides a tooltip with the name and view of the component.

Autodesk Inventor link

Autodesk Inventor® link redefines the meaning of 3D to 2D interoperability. Use the functionality to link to Autodesk Inventor parts and assemblies to:

- Access and associatively document native 3D part models without the presence of Autodesk Inventor.
- Visualize part models, examine and use part properties such as material, name, and number.
- Associatively document part models using precision hidden-line removed projections, dimensions, and annotations.
- Link to the native Autodesk Inventor part models automatically notifies you of changes and enables updating of views and annotations to keep your drawing up-to-date.

2D Design Productivity

These features increase productivity and reduce the number of steps needed to complete mechanical designs:

- AutoCAD Mechanical provides an intelligent, customizable layer management system that puts objects on the appropriate layers automatically.
- Entities that are not on the current layer group, or entities that are on a locked layer group can be displayed in a different color to reduce screen clutter.
- 2D hidden-line calculations are based on defined foreground and background objects. You can choose hidden line representation types.
- Auto detailing creates detailed drawings of individual components from an assembly drawing.
- One set of power commands is used to create, update, and edit objects.
- Mechanical line objects are available for creating centerlines and center crosses, construction lines, symmetrical lines, section lines, break lines, and others.
- Linear/symmetric stretch is used to modify dimensioned geometry by changing the dimension value.

- Predefined hatch patterns are applicable in two picks from toolbars and menus.

Engineering Calculations

The automatic engineering calculations available in AutoCAD Mechanical ensure proper function in mechanical designs.

- The 2D FEA feature determines the resistance capability of an object put under a static load and analyzes design integrity under various loads.
- A number of moment of inertia and beam deflection calculations are available.
- Engineering calculations are available for shafts, bearings, and screws.

Machinery Systems Generators

Machinery systems in AutoCAD Mechanical generate the design and calculation of shafts, springs, belts and chains, and cams. These tools ensure that you get the design right the first time:

- With the shaft generator, you can create drawing views of solid and hollow shafts. Common shaft features supported include center holes, chamfers, cones, fillets, grooves, profiles, threads, undercuts, and wrench fittings. Common standard parts supported include bearings, gears, retaining rings, and seals.
- With the spring generator, you select, calculate, and insert compression, extension, and torsion springs, and Belleville spring washers in a design. You control the representation type of the spring, and create a spec form to incorporate in the drawing.
- The belt and chain generator function provides features to create chain and sprocket systems, belt and pulley systems, calculate optimal lengths for chains and belts, and insert these assemblies in your design. Chains and belts can be selected from standard libraries.
- The cam generator creates cam plates and cylindrical cams given input border conditions. You can calculate and display velocity, acceleration, and the cam curve path. You can couple driven elements to the cam and create NC data through the curve on the path.

Intelligent Production Drawing and Detailing

A number of commands are available in AutoCAD Mechanical that automate the process to create balloons and bills of material.

- You can create formatted balloons and bills of material, as well as detailed views of portions of designs.
- Multiple parts lists per drawing are supported. Grouping of a parts list provides lists of like items. Selected items can be combined to calculate total length required for stock ordering. The parts lists recognize standard parts. You can format item numbers on parts lists.
- Standard-sized drawing borders and customizable title blocks are available.
- Intelligent and associative hole tables show a total count of each type of hole along with a description of them. A second chart lists the coordinates for each of the holes selected. Any update to the holes is reflected in the charts.
- A language converter translates text on a drawing into one of seventeen different languages.
- Revision control tables in drawings track revisions and display comments.
- Fits lists chart all fits used in a drawing.

Detailing Productivity

- Smart dimensions automatically maintain the proper arrangement with each other.
- Power dimension commands provide a single command to create and edit all dimensions, apply specified formats, and add fits or tolerances.
- Dimensions are automatic for 2D geometry with either ordinate or baseline dimensions.
- One command quickly cleans up and arranges dimensions in 2D drawings. One system setting controls the scale for drawing symbols in all views.
- Commands are available for align, break, insert, and join to easily dimension a drawing.

Annotations

- Hole notes can be inserted for standard holes.
- Commands are available to create standards-based surface texture symbols, geometric dimensioning and tolerances, targets, and weld symbols.
- Fits description command creates fits descriptions for standard holes.
- Leader command creates intelligent balloons and other leaders common in mechanical drawings.

Standard Mechanical Content

Standard content includes parametrically generated, intelligent geometry that you can use to generate an object from scratch. The following are available:

- About 600,000 standard parts, including screws, nuts, washers, pins, rivets, bushings, rings, seals, bearings, keys, and others, can be quickly incorporated into any design.
- About 8,000 standard features, including center holes, undercuts, keyways, and thread ends can be quickly incorporated into any design.
- More than 20,000 standard holes, including through, blind, counterbored, countersunk, oblong, and others, can be quickly incorporated into any design.
- Thousands of structural steel shapes, including U-shape, I-shape, T-shape, L-shape, Z-shape, rectangular tube, round tube, rectangular full beam, rectangular round beam, and others, can be quickly incorporated into any design.
- An editing environment to create and publish your own content libraries.

Standard Parts Tools

Standard part tools provide for the elements that go with standard parts, such as a hole to accompany a screw. These tools include:

- Screw connection feature for selecting entire fastener assemblies at one time.

- Changeable representation of a standard part between a normal, simplified, or symbolic representation.
- Power view to automatically generate a different view of a standard part, such as a top view from a front view.

Collaboration

Enjoy the benefits of design collaboration for your 2D output through Autodesk Streamline® support. Autodesk Streamline is a hosted Web service for sharing personalized design data across the entire extended manufacturing enterprise.

Autodesk Streamline functionality includes the following:

- Members can view and interact with the 3D data set published on Autodesk Streamline, without waiting for the data to download.
- Using Streamline, many people can share design information and collaborate online. Functionality includes instant messages, email notifications, polling/voting, discussion threads, database creation, and more.
- AutoCAD Mechanical data can be written to the AutoCAD DWF file format, which is one of the file types that Autodesk Streamline leverages.
- You can export 3D CAD data in ZGL format (a compressed form of a standard Open GL file format called XGL). ZGL readily captures 3D data that can be rendered by the Open GL library. ZGL files can then be uploaded to Autodesk Streamline.

Commands in AutoCAD Mechanical

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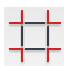
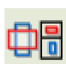
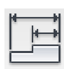
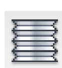
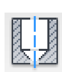
This chapter provides a list of the commands available in AutoCAD® Mechanical, along with a brief description of the function of each command and the associated tool button.

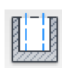
Command Summary

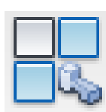
The following is a list of the AutoCAD Mechanical commands, a brief description of each, and the associated icon. The icon is used in the ribbon as well as toolbars

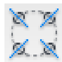
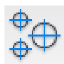
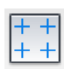
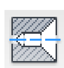
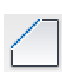
Some commands do not have an associated icon. This list does not contain AutoCAD® commands.

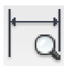
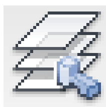
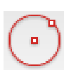
Icon	Command Name	Description
	AM2DHIDE	Draws hidden lines to represent hidden edges, when you specify what objects lie in front and what objects lie behind.
	AM2DHIDEDIT	Edits or updates hide situations created with the AM2DHIDE command.
	AMABOUT	Displays a screen containing licensing and copyright information.
	AMADJRINGS2D	Inserts an adjusting ring on a shaft.

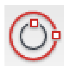
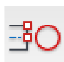
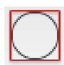
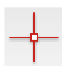
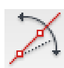
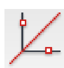
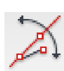
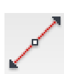
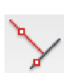
Icon	Command Name	Description
	AMANALYSEDWG	Analyzes the current drawing and writes layer information to an ALZ file.
	AMANNOTE	Creates a textual annotation that can be attached to an object in a drawing view.
	AMASSOHATCH	Creates a hatch that updates when you modify its boundaries.
	AMAUTOCLINES	Creates vertical and horizontal construction lines on all endpoints of selected objects.
	AMAUTODETAIL	Moves the selected objects to an external drawing file and creates an xref to it within the current drawing.
	AMAUTODIM	Creates multiple dimensions from a selected point to all the vertices on the contour of selected objects.
	AMBALLOON	Creates and places balloons in the drawing area.
	AMBEARCALC	Determines the limiting value, dynamic and static load rating, dynamic and static equivalent load, and fatigue life of a bearing.
	AMBELL2D	Performs a spring calculation and inserts a Bellville washer spring in the drawing area.
	AMBHOLE2D	Inserts a blind hole from the content library.
	AMBOM	Creates, edits, or deletes Bills of Materials (BOMs).
	AMBOMEXTTEMP	Creates an extraction template to be used by BOM Migration Utility.

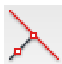
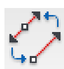
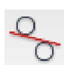
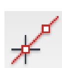
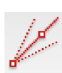
Icon	Command Name	Description
	AMBOMMIGRATE	Migrates non-AutoCAD Mechanical drawings to the latest AutoCAD Mechanical drawing format. Legacy BOM data within the drawings will be migrated to intelligent BOM and Parts list data.
	AMBREAKATPT	Breaks a line, polyline, or a spline at a specified point.
	AMBROUTLINE	Draws a special spline to show breakout borders.
	AMBROWSER	Displays, hides, and moves the mechanical browser.
	AMBSLOT2D	Inserts a blind slot from the content library.
	AMC_Break_Line (Layer Name)	Draws break out lines, as in hatched loop breaks or freehand breaks in shafts.
	AMC_Centerline (Layer Name)	Draws centerline.
	AMC_Construction (Layer Name)	Draws construction lines to display with the content.
	AMC_Construction_Invisible (Layer Name)	Draws construction lines that do not display with the content.
	AMC_Contour (Layer Name)	Draws contour edges.
	AMC_Contour_BHII (Layer Name)	Draws contours that make underlying and overlapping objects invisible in a hide situation.
	AMC_Contour_BHII_Invisible (Layer Name)	Draws contours that do not display with the content, and make underlying and overlapping objects invisible in a hide situation.

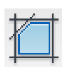
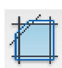
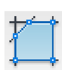
Icon	Command Name	Description
	AMC_Contour_BHIU (Layer Name)	Draws contours that make underlying objects invisible, and leave overlapping objects unchanged in a hide situation.
	AMC_Contour_BHIU_Invisible (Layer Name)	Draws contours that do not display with the content, make underlying objects invisible, and leave overlapping objects unchanged in a hide situation.
	AMC_Dimension (Layer Name)	Draws dimensions to display with the content.
	AMC_Dimension_Invisible (Layer Name)	Draws dimensions that do not display with the content.
	AMC_Hidden (Layer Name)	Draws hidden edges.
	AMC_Text_Medium (Layer Name)	Draws text that derives its color from the Text Medium object. However, this object does not control the height of the text you draw.
	AMC_Thread (Layer Name)	Draws thread lines.
	AMCAM	Displays a wizard for the design of linear, circular, or cylindrical cams.
	AMCARRAY	Creates multiple copies of content objects in a pattern.
	AMCARRAYEDIT	Edits an existing content array object.
	AMCENCRANGLE	Creates concentric pitch circles and places holes on it at specified angles.
	AMCENCRCORNER	Places a hole at a specified distance from two contour lines.

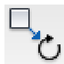
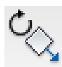
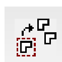
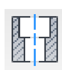
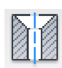
Icon	Command Name	Description
	AMCENCRFULLCIRCLE	Creates concentric pitch circles and places a specified number of holes on them, distributed uniformly.
	AMCENCRHOLE	Places a hole at a specified point.
	AMCENCRINHOLE	Places a centerline cross on a circle.
	AMCENCROSS	Places a centerline cross at a specified point.
	AMCENCRPLATE	Places holes at the corners of a closed contour that represents a plate.
	AMCENINBET	Draws a centerline in between two lines to mark them as symmetrical.
	AMCENTERHOLE2D	Inserts a centerhole from the content library.
	AMCENTLINE	Draws a centerline at a specified place in the drawing area.
	AMCHAINDRAW	Draws a chain or belt links along a polyline.
	AMCHAINLENGTHCAL	Calculates the required length of a chain or belt when you trace the path of the chain or belt.
	AMCHAM2D	Connects two non-parallel objects by extending or trimming them to intersect or to join with a beveled line.

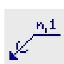
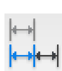
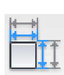
Icon	Command Name	Description
	AMCHAM2D_DIM	Creates dimensions for chamfers.
	AMCHATCH	Defines content hatch objects in templates for part or feature views.
	AMCHATCHEDIT	Edits an existing content hatch object.
	AMCHECKDIM	Highlights or edits dimensions with dimension text overrides.
	AMCLAYER	Displays the Content Editor Layers dialog box.
	AMCLEVISPIN2D	Inserts a clevis pin from the content library.
	AMCLINEL	Locks or unlocks construction line layers.
	AMCLINEL	Locks or unlocks the construction line layer.
	AMCLINEO	Freezes or thaws construction line layers.
	AMCLOSE	Closes the Content Editor.
	AMCOMP2D	Performs a spring calculation and inserts a compression spring in the drawing area.
	AMCONST_CIRCLE	Draws a construction line circle.

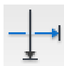
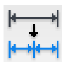
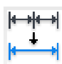
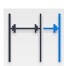
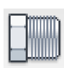
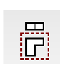
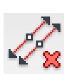
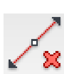
Icon	Command Name	Description
	AMCONSTC2	Draws a construction line circle that uses a specified line as a tangent.
	AMCONSTCC	Draws a construction line circle that is concentric to a specified circle.
	AMCONSTCCREA	Draws a construction line circle to represent a top view of a shaft or hole.
	AMCONSTCIRCLI	Draws a construction line rectangle around a circle.
	AMCONSTCRS	Draws a construction line cross.
	AMCONSTHB	Draws a construction line when you specify two points or a point and an angle.
	AMCONSTHM	Draws a construction line that bisects an angle.
	AMCONSTHOR	Draws a horizontal construction line.
	AMCONSTHW	Draws a construction line through a point, by specifying an angle relative to an apparent line that goes through that point.
	AMCONSTKR	Draws a construction line that is tangential to two specified circles.
	AMCONSTLINES	Draws construction lines.
	AMCONSTLOT	Draws a construction line that is perpendicular to a specified line.

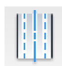
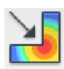
Icon	Command Name	Description
	AMCONSTLOT2	Draws a construction line through a specified point that is a perpendicular to the direction specified by another point or angle.
	AMCONSTPAR	Draws a construction line parallel to an existing line at a specified distance.
	AMCONSTPAR2	Draws a construction line parallel to an existing line and bisects the distance between the selected line and a specified point or a specified distance.
	AMCONSTSWI	Switches between construction lines that extend to infinity in both directions or lines that extend in one direction only.
	AMCONSTTAN	Draws two parallel construction lines that are tangential to a specified circle.
	AMCONSTTC	Draws two construction lines that are tangential to two specified circles.
	AMCONSTVER	Draws a vertical construction line.
	AMCONSTXLINE	Draws a construction line through a point, which extends to infinity in both directions.
	AMCONSTXRAY	Draws a construction line starting from a point and extends to infinity in one direction.
	AMCONSTZ	Draws a construction line along the z axis.
	AMCONTENTADD	Creates new content from scratch, the current drawing, or by selecting a block definition in the current drawing.

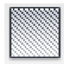
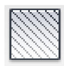
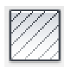
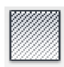
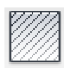
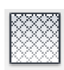
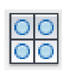
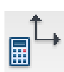
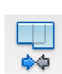
Icon	Command Name	Description
	AMCONTENTEDIT	Edits the template of the selected view when a Content Library part or feature on a drawing is selected.
	AMCONTENTLIB	Displays the Content Libraries.
	AMCONTENTLIB	Opens the Content Libraries for selection and insertion of content.
	AMCONTENTLIB-MIGRATE	Migrates Vario libraries to the current release of AutoCAD Mechanical.
	AMCONTENTMANAGER	Displays the Content Manager.
	AMCONTENTMIGRATE	Migrates legacy Vario parts and features to the current AutoCAD Mechanical format.
	AMCONTIN	Creates a closed polyline that traces the inner contour of an enclosed area.
	AMCONTOUT	Creates a closed polyline that traces the outer contour of an enclosed area.
	AMCONTRACE	Creates a closed contour that traces the contour of an area by letting you specify the boundaries segment by segment.
	AMCONVDWG	Changes layers properties (layer name, color, linetype, lineweight) to that specified in a Conversion Control file (CCF file).
	AMCOPYLG	Copies objects on one or more layer groups to a new layer group.
	AMCOPYRM	Performs copy, rotate, and move operations on specified objects, in sequence

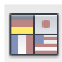
Icon	Command Name	Description
	AMCOPYRM_MR	Copies, then moves, and then rotates the specified objects.
	AMCOPYRM_R	Copies, then rotates specified objects.
	AMCOPYRM_RM	Copies, then rotates, and then moves specified objects.
	AMCOPYVIEW	Copies a drawing view to the same layout or to a different layout.
	AMCOTTERPIN2D	Inserts a cotter pin from the content library.
	AMCOUNTB2D	Inserts a counterbore from the content library.
	AMCOUNTS2D	Inserts a countersink from the content library.
	AMCPARTREF	Places one content part reference in a view.
	AMCRIVET2D	Inserts a countersunk rivet from the content library.
	AMCSAVE	Saves the current view.
	AMCSAVEAS	Saves the current view in Content Editor to a different name.
	AMCSETTINGS	Displays the Content Settings dialog box.
	AMCSWITCHVIEW	Navigates between views.
	AMCTABLE	Displays the Family Table.

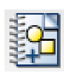
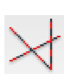
Icon	Command Name	Description
	AMCTABLECLOSE	Closes the Family Table.
	AMCTABLETOGGLE	Creates content families by attaching a Family Table to parts or features.
	AMCTESTCONTENT	Tests the view of the part or feature being authored in its current state.
	AMCYLPIN2D	Inserts a cylindrical pin from the content library.
	AMDATUMID	Creates a datum identification symbol and, attaches it to an object in the drawing area.
	AMDATUMTGT	Creates a datum target symbol and, attaches it to an object in the drawing area.
	AMDEADJOINT	Creates a dead joint symbol.
	AMDEFLINE	Calculates and draws the deflection line or moment line of a beam that is subject to various forces.
	AMDELVIEW	Deletes a drawing view.
	AMDETAIL	Create a scaled circular, rectangular, or free defined detail view of selected geometry.
	AMDIMALIGN	Lines up linear, ordinate, or angular dimensions, with a dimension you select as the baseline.
	AMDIMARRANGE	Rearranges linear and ordinate dimensions, placing them at an appropriate distance from the outer contour of an object you select.

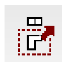
Icon	Command Name	Description
	AMDIMBREAK	Creates breaks in dimensions at specific points or at points where the dimensions cross other objects.
	AMDIMINSERT	Splits a linear or angular dimension into two individual dimensions.
	AMDIMJOIN	Combines two individual dimensions (of the same type) into a single dimension.
	AMDIMMEDIT	Edits multiple dimensions simultaneously.
	AMDIMSTRETCH	Resizes objects by stretching or shrinking a linear or symmetric dimension.
	AMDRBUSH2D	Inserts a drill bushing from the content library.
	AMDRBUSHHOLE2D	Inserts a drill bushing and the corresponding hole, from the content library.
	AMDWGVIEW	Creates a drawing view for an Inventor linked drawing.
	AMEDGESYM	Creates an edge symbol and, attaches it to an object in the drawing area.
	AMEDITVIEW	Provides the ability to modify attributes of a drawing view.
	AMERASEALLCL	Erases all construction lines.
	AMERASECL	Erases selected construction lines.

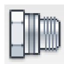
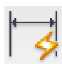
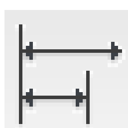
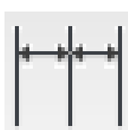
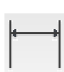
Icon	Command Name	Description
	AMEXPLODE	Converts mechanical structure objects to non-mechanical structure objects. When used on non-mechanical structure objects, breaks a compound object into its component objects.
	AMEXT2D	Performs a spring calculation and inserts a compression spring in the drawing area.
	AMEXTHREAD2D	Inserts an external thread from the content library.
	AMFCFRAME	Creates a feature control frame symbol and, attaches it to an object in the drawing.
	AMFEA2D	Performs a Finite Element Analysis on a two-dimensional object that is subject to a static load.
	AMFEATID	Creates a feature identification symbol and attaches it to an object in the drawing area.
	AMFILLET2D	Rounds and fillets the edges of objects with an arc of a specified radius.
	AMFITSLIST	Generates a fits list from the dimensions in the drawing area and enables you to place at a location of your choice.
	AMGROOVE2D	Inserts a circlips and draws the corresponding groove on a shaft.
	AMGROOVESTUD2D	Inserts a grooved drive stud from the content library.
	AMHATCH_135_11	Fills an enclosed area with a 135-degree, 11 mm/0.4" hatch.

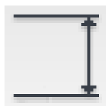
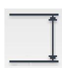
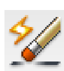
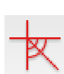
Icon	Command Name	Description
	AMHATCH_135_2	Fills an enclosed area with a 135-degree, 2.7 mm/0.12" hatch.
	AMHATCH_135_4	Fills an enclosed area with a 135-degree, 4.7 mm/0.19" hatch.
	AMHATCH_45_13	Fills an enclosed area with a 45-degree, 13 mm/0.5" hatch.
	AMHATCH_45_2	Fills an enclosed area with a 45-degree, 2.5 mm/0.1" hatch.
	AMHATCH_45_5	Fills an enclosed area with a 45-degree, 5 mm/0.22" hatch.
	AMHATCH_DBL	Fills an enclosed area with a 45 and 135 degree, 2.3 mm/0.09 cross hatch.
	AMHELP	Displays AutoCAD Mechanical online help.
	AMHOLECHART	Creates coordinate dimensions for holes in a work piece, dimensions the size of those holes, and generates a hole chart for that work piece.
	AMINERTIA	Calculates the moment of inertial of a closed contour.
	AMINERTIAPROF	Calculates the moment of inertia for cross sections of cylinders, hollow cylinders, rectangular prisms, or hollow rectangular prisms.
	AMIVCOMPONLY	Creates an Autodesk Inventor linked file.
	AMJOIN	Connects non parallel lines, polylines, arcs, and circles.

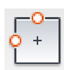
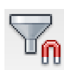
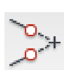
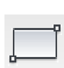
Icon	Command Name	Description
	AMLANGCONV	Translates text in the drawing to another language.
	AMLANGTEXT	Display a list of text strings for which translations are available and enables you to insert them into your drawing.
	AMLAUNCHPAD	Displays the AutoCAD Mechanical Launchpad.
	AMLAYER	Displays the Mechanical Layer Manager, which enables you to manage layers and mechanical layer definitions.
	AMLAYERGROUP	Displays the Layergroup Manager, which allows you to manage layergroups, layers, and their properties.
	AMLAYINVO	Toggles the visibility of the layer assigned to invisible lines (AM_INV).
	AMLAYMOVE	Moves objects from one layer to another.
	AMLAYMOVEPL	Move objects that lie on standard parts layers (layers AM_0N to AM_12, by default) to the corresponding working layers (layers AM_0 to AM_12, by default).
	AMLAYMOVEWL	Move objects that lie on working layers (layers AM_0 to AM_12, by default) to the corresponding standard part layers (layers AM_0N to AM_12N, by default).
	AMLAYPARTO	Toggles the visibility of the standard part layers (AM_0N to AM_12N, by default).
	AMLAYPARTREFO	Toggles the visibility of the layer assigned to part references (AM_PAREF, by default).

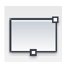
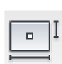
Icon	Command Name	Description
	AMLAYRESET	Resets properties of all mechanical layers to be identical to the corresponding layer definition.
	AMLAYTIBLO	Toggles the visibility of the layer assigned to title blocks and drawing borders (AM_BOR, be default)).
	AMLAYVISENH	Displays the Visibility Enhancements dialog box, which allows you to visually differentiate between the active layergroup and inactive ones.
	AMLAYVPO	Toggles the visibility of the layer assigned to viewport borders (AM_VIEWS, by default).
	AMLGMOVE	Moves the selected objects to a specified layergroup.
	AMLIBRARY	Displays the Library dialog box, which enables you to organize drawing files to retrieve and reuse them more efficiently.
	AMLISTVIEW	Lists information about a selected view.
	AMLUBRI2D	Inserts a lubricator from the content library.
	AMMANIPULATE	Displays the Power Manipulator, a tool that enables you to rotate move and copy objects by dragging.
	AMMARKSTAMP	Creates a marking symbol and attaches it to an object in the drawing area.
	AMMCONTV	Makes contours that are hidden by construction lines visible.

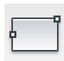
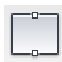
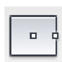
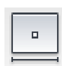
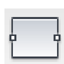
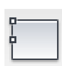
Icon	Command Name	Description
	AMMODE	Switches between model space and paper space.
	AMMOVEDIM	Moves dimensions within a view or between views, while maintaining their association to the drawing view geometry.
	AMMOVEVIEW	Moves a drawing view to another location within the same layout or to another layout.
	AMNOTE	Creates a leader note and, attaches it to an object in the drawing area.
	AMNUT2D	Inserts a nut from the content library.
	AMOFFSET	Creates concentric circles, parallel lines, and parallel curves.
	AMPARTLIST	Creates a parts list and enables you to place it in the drawing area.
	AMPARTREF	Creates a part reference and places it in the drawing area.
	AMPARTREFEDIT	Edits a part reference.
	AMPLBEAR2D	Inserts a plain bearing on a shaft.
	AMPLOTDATE	Inserts the current date in the lower right corner of all title blocks in the drawing.
	AMPLRIVET2D	Inserts a plain rivet from the content library.

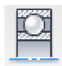
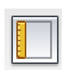
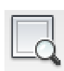
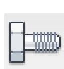
Icon	Command Name	Description
	AMPLUG2D	Inserts a plug from the content library.
	AMPOWERCOPY	Produces an identical copy of a specified object.
	AMPOWERDIM	Creates multiple types of dimensions within a single command session and provides the ability to specify tolerances or fits as appropriate.
	AMPOWERDIM_ALI	Creates aligned linear dimensions.
	AMPOWERDIM_ANG	Creates angular dimensions.
	AMPOWERDIM_ARCLEN	Creates arc length dimensions for arcs and arc segments on a polyline.
	AMPOWERDIM_BAS	Creates a linear or angular dimension from the first extension line of a selected dimension.
	AMPOWERDIM_CHAIN	Creates a linear, angular, or arc length dimension from the second extension line of a selected dimension.
	AMPOWERDIM_DIA	Creates diameter dimensions for arcs and circles.
	AMPOWERDIM_HOR	Creates horizontal linear dimensions.

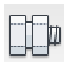
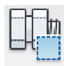
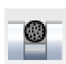
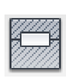
Icon	Command Name	Description
	AMPOWERDIM_JOG	Create a radius dimension with the origin of the dimension at any location you wish and a jog at a convenient location along the dimension line.
	AMPOWERDIM_RAD	Creates radius dimensions for arcs and circles.
	AMPOWERDIM_ROT	Creates rotated linear dimensions.
	AMPOWERDIM_VER	Creates vertical linear dimensions.
	AMPOWEREDIT	Recognizes any object you select and invokes the most appropriate command to edit it.
	AMPOWERERASE	Erases AutoCAD Mechanical objects cleanly and heals the surrounding area.
	AMPOWERRECALL	Recognizes the object you select and invokes the command that it was created with.
	AMPOWERSNAP	Displays the Power Snap Settings dialog box, to enable you to set the running object snap modes.
	AMPOWERVIEW	Creates a top view from a side view of a standard part and vice versa.
	AMPROJO	Creates projection lines to assist you create orthographic views.

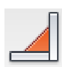
Icon	Command Name	Description
	AMPSNAP1	Sets the running object snap modes to the settings saved as Power Snap Configuration Setting 1.
	AMPSNAP2	Sets the running object snap modes to the settings saved as Power Snap Configuration Setting 2.
	AMPSNAP3	Sets the running object snap modes to the settings saved as Power Snap Configuration Setting 3.
	AMPSNAP4	Sets the running object snap modes to the settings saved as Power Snap Configuration Setting 4.
	AMPSNAPCEN	Snaps to the center of a rectangle, when invoked from within another command.
	AMPSNAPFILTERO	Switches entity filters on or off.
	AMPSNAPMID	Snaps to a point on the apparent line between two specified points, when invoked from within another command.
	AMPSNAPREL	Snaps to a point relative to a specified point, when invoked from within another command.
	AMPSNAPVINT	Snaps to the apparent intersection of two non-parallel lines, when invoked from within another command.
	AMPSNAPZO	Toggles snapping to Z axis coordinates on or off.
	AMRECTANG	Draws rectangle using a closed polyline.

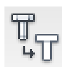
Icon	Command Name	Description
	AMRECTBWH	Creates a rectangle by using the middle of the base as the start point, and by specifying the full base and full height.
	AMRECTBWH2	Creates a rectangle by using the middle of the base as the start point, and by specifying the full base and half of the height.
	AMRECTBY	Creates a rectangle by using the middle of the base as the start point, and specifying a corner point.
	AMRECTCW2H	Creates a rectangle by selecting the center as start point, and by defining half base and full height.
	AMRECTCW2H2	Creates a rectangle by using the center of the rectangle as the start point, and specifying half of the base and half of the height.
	AMRECTCWH	Creates a rectangle by selecting the center of the rectangle as the start point, and by specifying the full base and full height.
	AMRECTCWH2	Creates a rectangle by using the center of the rectangle as the start point, and specifying the full base and half of the height.
	AMRECTCY	Creates a rectangle by using the center of the rectangle as the start point, and specifying a corner.
	AMRECTLWH	Creates a rectangle by using the midpoint of the height as the start point, and specifying the full base and full height.
	AMRECTLWH2	Creates a rectangle by selecting the height middle as start point, and by defining full base and half height.

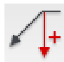
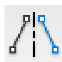
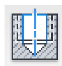
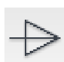
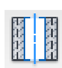
Icon	Command Name	Description
	AMRECTLY	Creates a rectangle by using the middle of the height as the start point, and specifying the opposite corner.
	AMRECTQBT	Creates a square by selecting the base middle as starting point, and by defining full base.
	AMRECTQBY	Creates a square by selecting the base middle as starting point, and by defining half base.
	AMRECTQCR	Creates a square by selecting the center as starting point, and by defining half base.
	AMRECTQCW	Creates a square by selecting the center as starting point, and by defining full base.
	AMRECTQLR	Creates a square by selecting the height middle as starting point, and by defining full base.
	AMRECTQLY	Creates a square by selecting the height middle as starting point, and by defining half base.
	AMRECTQXY	Creates a square by selecting the starting point, and by defining full base.
	AMRECTXWH	Creates a rectangle by using a corner of the rectangle as the start point, and specifying the full base and full height.
	AMREV	Turns revisions on or off.
	AMREVLINE	Adds a revision line to a revisions list.

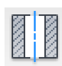
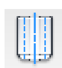
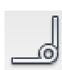
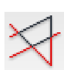
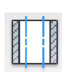
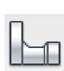
Icon	Command Name	Description
	AMREVUPDATE	Updates the revision block.
	AMROLBEAR2D	Inserts a roller bearing on a shaft.
	AMSACTIVATE	Makes the specified component view or folder the active edit target.
	AMSBASE	Changes the basepoint of a component view or folder.
	AMSCALEXY	Sets scale of X and Y axes independently of each other.
	AMSCAREA	Creates a scale area in model space.
	AMSCATALOG	Displays the Structure Catalog.
	AMSCMONITOR	Displays the scale of a scale area or viewport.
	AMSCOPYDEF	Saves a copy of the definition of the selected structure object to another name.
	AMSCREATE	Creates a new component, component view, folder or annotation view.
	AMSCREW2D	Inserts a screw or bolt from the content library.
	AMSCREWCALC	Calculates factors of safety for parts of a screw connection.

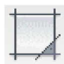
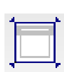
Icon	Command Name	Description
	AMSCREWCON2D	Inserts an entire fastening assembly using items from the content library.
	AMSCREWMACRO2D	Inserts an entire fastener assembly using items predefined on a template.
	AMSCRIPT	Generates Scripts to apply a specified operation on selected drawings or entire folders.
	AMSEALRING2D	Inserts a sealing ring from the content library.
	AMSEALS2D	Inserts a seal or O-ring and draws the corresponding groove on a shaft.
	AMSECTIONLINE	Draws a section line inserts the corresponding section view label in the drawing area.
	AMSEDIT	Adds, removes, or copies geometry to the currently activated folder or component view.
	AMSETUPDWG	Imports the drafting standards and the dependant settings from a template file.
	AMSHAFT2D	Starts the Shaft Generator utility.
	AMSHAFTCALC	Calculate deflection lines, bending moments, torsion moments, and factors of safety for shafts and axles put under static loads.
	AMSHAFTEND	Draws a break line on a shaft to indicate a shaft end.
	AMSHAFTKEY2D	Inserts a parallel or woodruff key on a shaft.

Icon	Command Name	Description
	AMSHAFTLNUT2D	Inserts a shaft lock nut and a lock washer on a threaded segment of a shaft.
	AMSHIDE	Draws hidden lines to represent hidden edges, when you specify what objects lie in front and what objects lie behind.
	AMSHIDEEDIT	Edits a hide situation created with the AM-SHIDE command.
	AMSHIMRING2D	Inserts a shim ring on a shaft.
	AMSIMPLEWELD	Draws seam and fillet welds on ellipses, circles, arcs, lines and polylines.
	AMSINSERT	Inserts an instance of a component, compon-ent view or folder, that is already defined in the drawing.
	AMSNAVMODE	Turns the Design Navigation Mode on or off.
	AMSNEW	Creates a new mechanical structure definition from selected objects.
	AMSPROCKET	Inserts the front view of a sprocket or pulley from the content library.
	AMSPURGE	Removes unused definitions of mechanical structure objects from the drawing.
	AMSREPLACEDEF	Replaces the definition of selected instances of a component view or folder, with another definition.

Icon	Command Name	Description
	AMSSMODE	Switches between the top-down and bottom-up geometry selection modes.
	AMSTDPLIBEDIT	Displays the Content Manager. **NOTE** This command is superseded by the AMCONTENTMANAGER command.
	AMSTDPREP	Defines how selected parts and features from the content library are represented in the drawing.
	AMSTLSHAP2D	Inserts a steel shape from the content library.
	AMSTYLEI	Imports dimension styles from another drawing.
	AMSTYLEITAL	Changes the text style to italics.
	AMSTYLEITAL	Changes the text style to italics.
	AMSTYLESIMP	Sets the current text style to SIMPLEX.
	AMSTYLESIMP	Sets the current text style to SIMPLEX.
	AMSTYLESTAND	Sets the current text style to STANDARD.
	AMSTYLETEXT	Sets the current text style to USER1.
	AMSURFSYM	Creates a surface texture symbol and attaches it to an object in the drawing area.

Icon	Command Name	Description
	AMSYMLEADER	Adds or removes leaders and leader segments to/from a symbol.
	AMSYMLINE	Draws symmetrical polylines.
	AMTAPBHOLE2D	Inserts a tapped blind hole from the content library.
	AMTAPERPIN2D	Inserts a taper pin from the content library.
	AMTAPERSYM	Creates a taper or slope symbol.
	AMTAPETHREAD2D	Inserts a tapered hole that contains external threads.
	AMTAPITHREAD2D	Inserts a tapered hole that contains internal threads.
	AMTAPTHOLE2D	Inserts a tapped through hole from the content library.
	AMTESTCONTENTCLOSE	Closes the Test Content Window and returns you to the Content Editor.
	AMTESTCONTENTINSERT	Inserts the content being tested.
	AMTEXT3	Creates a multiline text object of 3.5 mm height.
	AMTEXT5	Creates a multiline text object of 5 mm height.

Icon	Command Name	Description
	AMTEXT7	Creates a multiline text object of 7 mm height.
	AMTEXTCENT	Creates a multiline text object that is centered horizontally and vertically.
	AMTEXTHORIZ	Creates a center justified multiline text object that is aligned to the bottom of the specified area.
	AMTEXTRIGHT	Creates a right justified multiline text object that is aligned to the bottom of the specified area.
	AMTHOLE2D	Inserts a through hole from the content library.
	AMTHREADEND2D	Inserts a thread end from the content library.
	AMTITLE	Inserts a drawing border with a title block.
	AMTOR2D	Performs a spring calculation and inserts a torsion spring in the drawing area.
	AMTRCONT	Traces contours on construction lines or construction line circles.
	AMTSLOT2D	Inserts a through slot from the content library.
	AMUNDERCUT2D	Inserts an undercut on a shaft.

Icon	Command Name	Description
	AMUPDATE	Updates the specified drawing view to reflect the modifications made to the model.
	AMUSERHATCH	Fills an enclosed area with a user-defined hatch pattern.
	AMVALMIGRATE	Migrates the schema of the selected VAL file format to the current AutoCAD Mechanical format.
	AMVIEWALL	Creates the named view AM_VIEW6 and loads it.
	AMVIEWCEN	Creates the named view AM_VIEW5 and loads it.
	AMVIEWLL	Creates the named view AM_VIEW3 and loads it.
	AMVIEWLR	Creates the named view AM_VIEW4 and loads it.
	AMVIEWOUT	Exports drawing views to an AutoCAD drawing file.
	AMVIEWUL	Creates the named view AM_VIEW1 and loads it.
	AMVIEWUR	Creates the named view AM_VIEW2 and loads it.
	AMVISIBLE	Displays the Desktop Visibility dialog box to enable you to selectively display/hide objects.
	AMVPORTAUTO	Creates viewports for all scale areas that are not associated with a viewport.
	AMVPZOOMALL	Resets all viewports to their default scale factors.

Icon	Command Name	Description
	AMWASHER2D	Inserts a washer from the content library.
	AMWELDSYM	Creates a welding symbol and attaches it to the weld point.
	AMXREFSET	Displays the Xref Processing dialog box to enable you to specify how xref entities are displayed.
	AMZIGZAGLINE	Draws a zigzag line.
	AMZOOMVP	Zooms an area in a viewport.

Design and Annotation Tools

The tutorials in this section teach you how to use the tools in AutoCAD® Mechanical for design, annotation, and productivity. The lessons include step-by-step instructions and helpful illustrations. You learn how to work with templates and layers, mechanical structure, model space and layouts, dimensions, steel shapes, bills of material (BOMs) and parts lists. Instructions on how to prepare your designs for final documentation are also included.

Working with Templates

3

In this tutorial, you learn about the predefined templates and how to create your own user-defined templates in AutoCAD® Mechanical.

Key Terms

Term	Definition
base layer	A layer made up of working layers and standard parts layers. Base layers are repeated in every layer group.
layer group	A group of associated or related items in a drawing. A major advantage of working with layer groups is that you can deactivate a specific layer group and a complete component. The drawing and its overview are enhanced with a reduction in regeneration time.
part layers	A layer where the standard parts are put. All standard parts layers have the suffix AM_*N.
template	A file with predefined settings to use for new drawings. However, any drawing can be used as a template.
working layer	The layer where you are currently working.

Working with Templates

In AutoCAD Mechanical, you can use templates (**.dwt* files) to create drawings.

Predefined templates, which contain settings for various drawings, such as *am_iso.dwt* or *am_ansi.dwt*, are supplied with AutoCAD Mechanical. You can create your own templates, or use any drawing as a template. When you use a drawing as a template, the settings in that drawing are used in the new drawing.

Although you can save any drawing as a template, prepare templates to include settings and drawing elements that are consistent with your company or project standards, such as the following items:

- unit type and precision
- drawing limits
- snap, grid, and ortho settings
- layer organization
- title blocks, borders, and logos
- dimension and text styles
- linetypes and lineweights

If you start a drawing from scratch, AutoCAD Mechanical reads the system defaults from the registry. The system defaults have a predefined standard.

If you create a drawing based on an existing template and make changes to the drawing, those changes do not affect the template.

To begin working with templates immediately, you can use the predefined template files.

However, for this tutorial you create your own template.

Setting Mechanical Options

In the Options dialog box, you can specify general settings for AutoCAD Mechanical, Autodesk® Mechanical Desktop®, and AutoCAD®. Tabs that affect settings for either Mechanical Desktop or AutoCAD Mechanical, or both, have an AM prefix. Use the arrows at the right end of the tab bar to move left and right through all of the available tabs.

To set mechanical options

1 Start the Mechanical Options command.

Ribbon	None.
Menu	Assist ➤ Options.

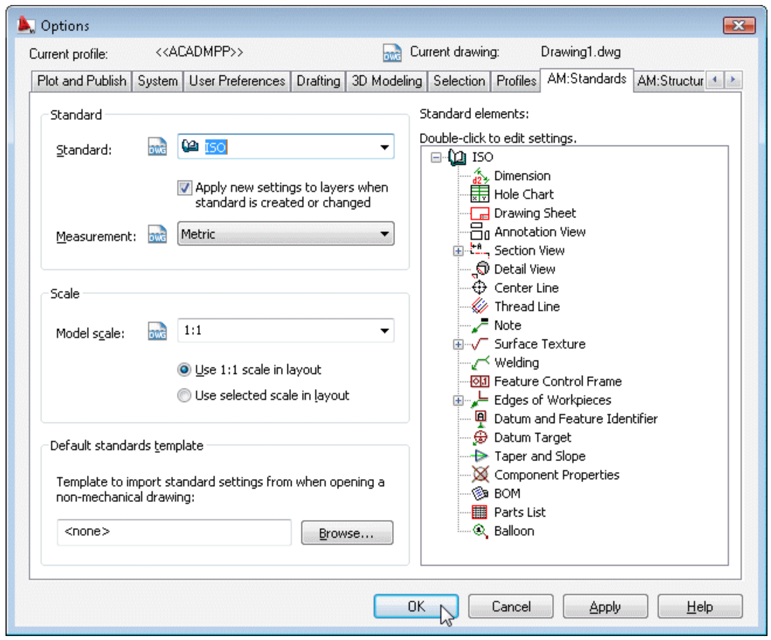
Command	OPTIONS or AMOPTIONS

2 On the AM:Standards tab, specify:

`Standard:` ISO

`Measurement:` Metric

`Model Scale:` 1:1



Click OK.

NOTE All settings in this dialog that are stored in the drawing (template) are marked with this icon:  The current standard and all related settings are listed in the right section.

Specifying Drawing Limits

Specify the drawing limits according to size A0 (840 x 1188 mm). This limits your drawing space to the specified size.

To specify the drawing limits

1 Start the Drawing Limits command.

Ribbon	None
Menu	Format ➤ Drawing Limits
Command	LIMITS

2 Respond to the prompts as follows:

`Specify lower left corner or [ON/OFF] <0.00,0.00>:` *Press* ENTER

`Specify upper right corner <420.00,297.00>:`

Enter 841, 1189, press ENTER

The limits are expanded to A0 format.

Saving Templates

Save the previously changed drawing as a template.

To save a template

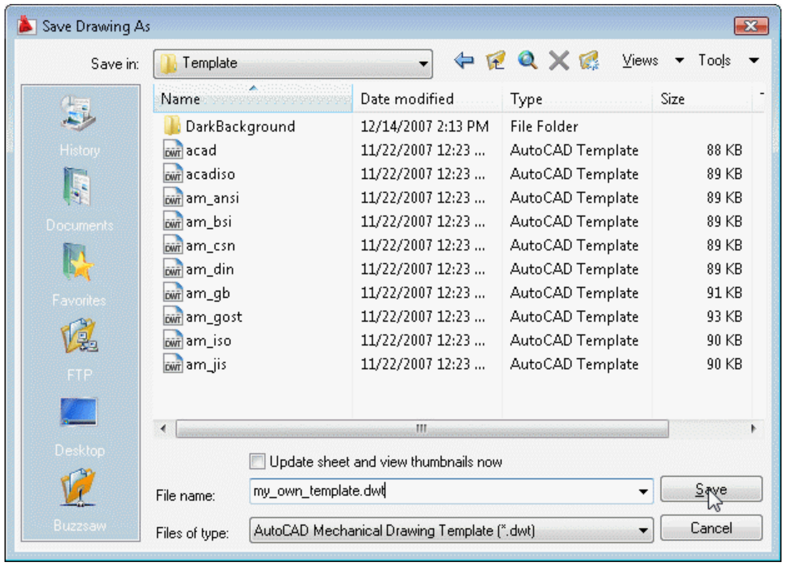
1 Start the Save As command.

Ribbon	➤ Save as ➤ AutoCAD Mechanical Drawing Template ➤
Menu	File ➤ Save As
Command	SAVEAS

2 In the Save Drawing As dialog box, specify:

`Files of type:` AutoCAD Mechanical Drawing Template (*.*dwt*)

`File name:` *my_own_template*

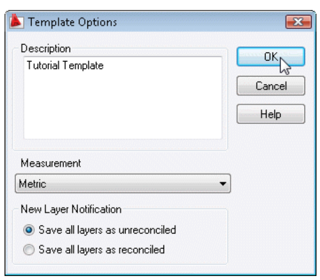


Click Save.

3 In the Template Description dialog box, specify:

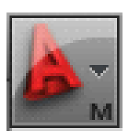
`Description:` Tutorial Template

`Measurement:` Metric



Click OK.

4 Close the drawing.

Ribbon  ➤ Close ➤ Current Drawing ➤

Menu File ➤ Window ➤ Close

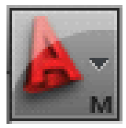
Command CLOSE

Using Templates

Use the previously created template to start a new drawing.

To open a template

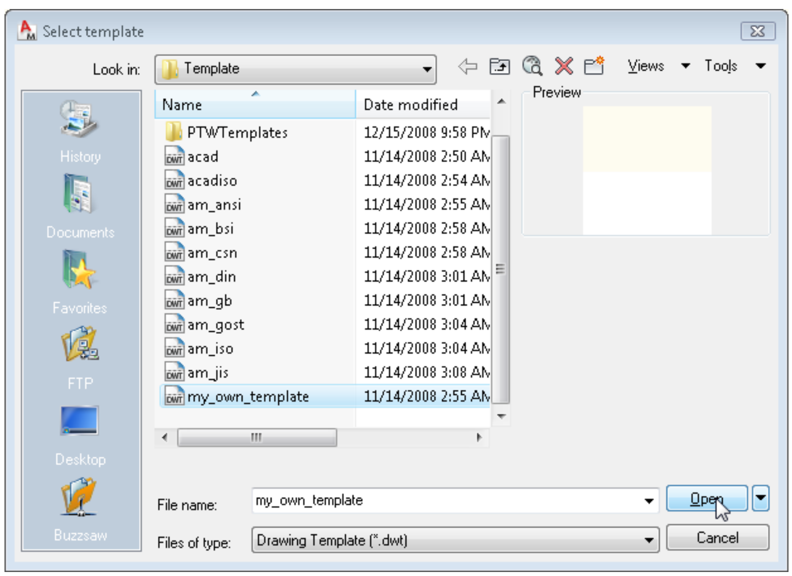
1 Start the New command.

Ribbon  ➤ New ➤ Drawing ➤

Menu File ➤ New

Command NEW

2 In the Select template dialog box, select *my_own_template.dwt*, and then choose Open.



Start the new drawing using the settings of the previously saved template.

Setting Default Standards Templates

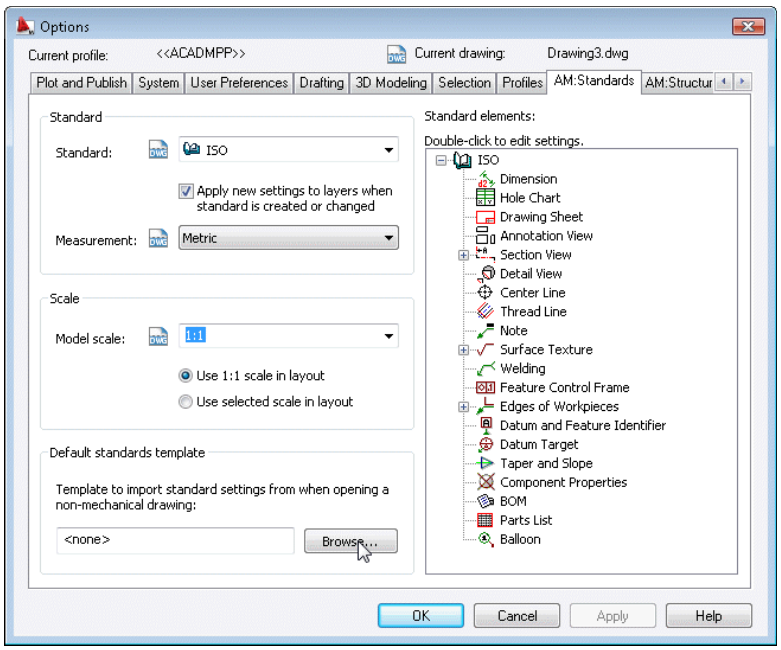
Specify your template as the default template.

To set a default template

1 Start the Mechanical Options command.

Ribbon	None.
Menu	Tools ➤ Options
Command	AMOPTIONS

2 In the Options dialog box, AM:Standards tab, choose Browse.



3 In the Open dialog box, select *my_own_template.dwt*, and then choose Open.

4 In the Options dialog box, Click OK.

The template *my_own_template* is used as the default standards template until you specify a different default template.

NOTE The default standards template is used if a drawing does not contain any AutoCAD Mechanical configuration. If a drawing already contains AutoCAD Mechanical configuration data, or a new drawing has been created using an AutoCAD Mechanical template, the default template does not affect the drawing.

This is the end of this tutorial chapter.

Using Mechanical Structure

4

In this tutorial, you learn how to use mechanical structure in AutoCAD® Mechanical. You learn how to work with folders, components and component views. You also review the bill of materials, restructure components and resolve ghost components. You learn how to insert components from external files, edit in-place, localize external components and externalize local components.

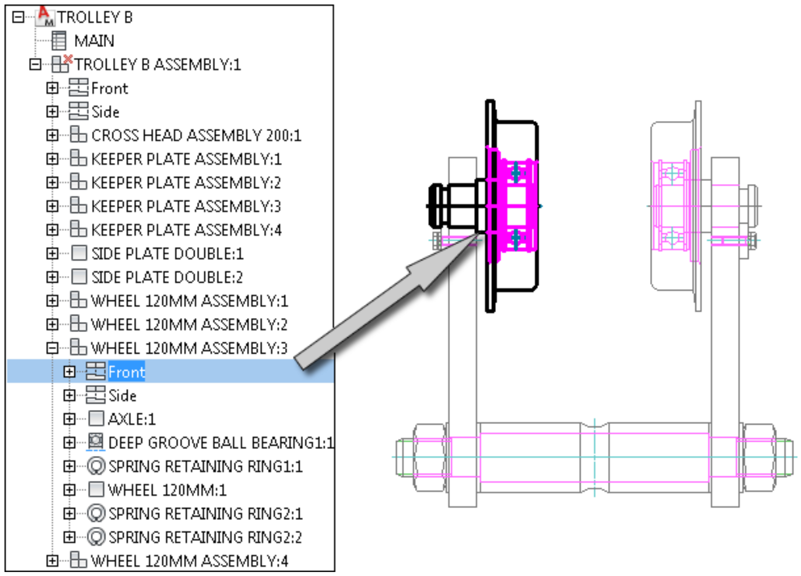
Key Terms

Term	Definition
annotation view	A folder that contains one or more component views dedicated to annotating and detailing parts and subassemblies.
associative	In mechanical structure, the implication that a change to one instance of a definition is reflected in all other instances of that definition, including the definition itself.
mechanical browser	A browser that contains the hierarchy of components, component views, annotation views, and folders of a given mechanical structure.
component	A browser placeholder and identification for the component type. A component is analogous to the manufacturing units of parts and assemblies.
component view folder	A folder nested under a component that contains the geometry for a particular view of that component.

Term	Definition
definition	A description of a folder, component, or view that AutoCAD Mechanical saves in the database, similar to a block definition.
elemental geometry	The graphical elements of a drawing that represent the shape and size of a part or assembly.
free object (as used in the Create Hide Situation dialog box)	A unit of elemental geometry.
geometry	The graphical elements of a drawing that represent the shape and size of a part or assembly.
hidden geometry	Geometry that is included in a hide situation.
instance	An iteration of a definition as it appears in mechanical structure.
object	Used variously to describe any item in mechanical structure, whether a component, folder, or geometry.
occurrence	Placement of a component, usually in multiple-level assemblies, where a component is replicated as a result of multiple placements of a single part or subassembly.

Working with Mechanical Structure

Mechanical structure is a set of tools used to organize data for reuse. Structure is graphically represented by a tree called the Mechanical Browser.



The Mechanical Browser and structure tools are not displayed by default. To display them, you must switch to the structure workspace. First, you must create a new drawing and enable mechanical structure.

To display the Mechanical Browser

1. On the command line, enter *WORKSPACE* and press ENTER.
2. Respond to the prompts as shown:

   `Enter workspace option`
   `[setCurrent/SAveas/Edit/Rename/Delete/SEttings/?]:`

   Enter C and press ENTER

   `Enter name of workspace to make current [?] <Current Workspace>:`

   Enter Structure and press ENTER

Even though you switch to the Structure Workspace, mechanical structure is not switched on automatically.

To enable mechanical structure

- Click the STRUCT status bar button and ensure that it lights up.

Folders

The basic element of mechanical structure is the *folder.* A folder is similar to a block in that it has a definition that can be instanced multiple times. Like a block, the definition is stored away in the non graphical area of the drawing. Similar to blocks, any change you make to the folder definition is reflected in all instances of that folder.

Creating Folders

1 Use the Circle tool to create a circle. The size and proportions are not important.

2 Use the Rectangle tool to draw a rectangle around the circle.

3 Right-click anywhere in the browser, and select New ➤ Folder.

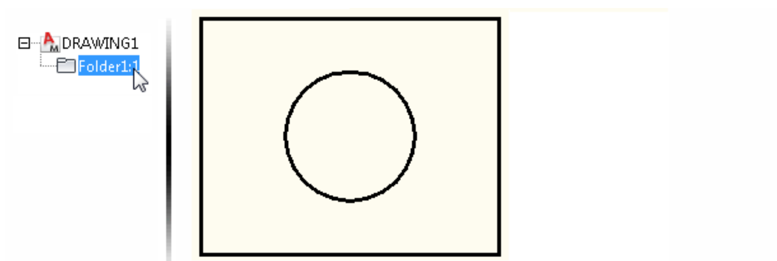
4 Respond to the prompts as shown:

`Enter folder name <Folder1>:` *Press* ENTER

`Select objects for new folder:`

Select the circle and then the rectangle and press ENTER

`Specify base point:` *Pick the lower left corner of the rectangle*



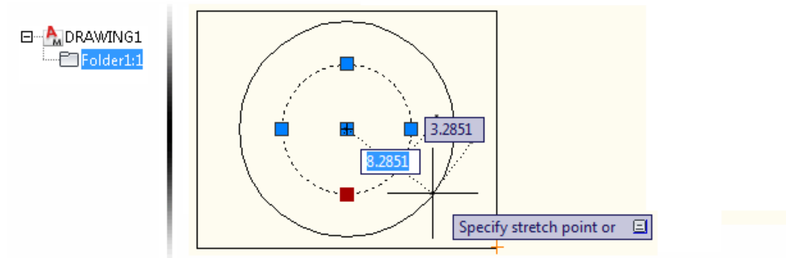
Modifying Folders

While folders are similar to blocks, there are significant differences. The most significant, is that the contents of a folder remain editable without the need for a special editing mode such as *REFEDIT*.

To grip edit the circle

1 Continue clicking the circle until you see the word CIRCLE in the tooltip window.

2 Select a grip, drag and then click.



If the contents of a folder are selectable, how do you select the folder? This is where the tooltip comes in. You select folders (and other elements of structure) by cycling through a selection, and the tooltip tells you what you are selecting. In the next exercise, you copy the folder to demonstrate structure selection.

To copy the folder

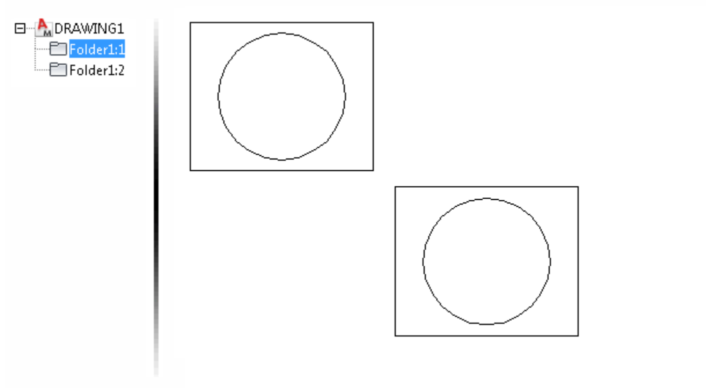
1 Press ESC to clear any preselection.

2 On the command line, enter *COPY* and press ENTER.

3 Continue clicking the circle until you see the word Folder1:1 in the tooltip window.

4 Press ENTER to complete selection, then pick points to finish the copy.

5 Press ESC to finish.



The Mechanical Browser shows a second instance of the folder (Folder1:2), implying that you copied the folder, not just the contents.

In the next exercise you modify the contents of a folder to demonstrate that modifying one instance of a folder updates both.

To edit an instance

1 Continue clicking a circle until you see the word CIRCLE in the tooltip window.

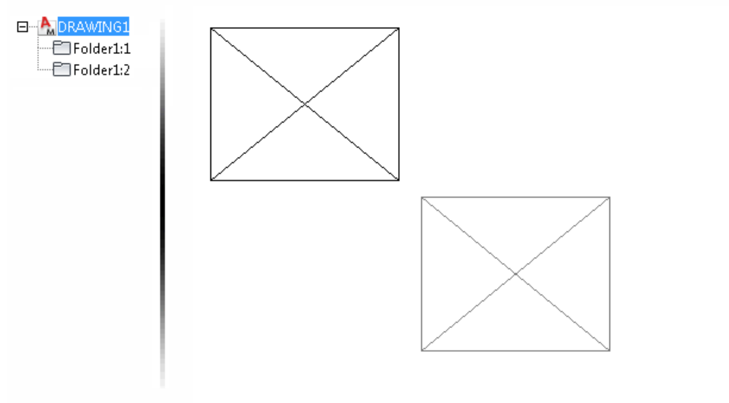
2 Press DELETE. Note how the circle is deleted from both instances.

Next, you add new geometry to a folder. Before you add geometry you must *activate* the folder to make it the active edit target. This ensures that geometry is added to the folder and not to model space.

To activate and add geometry

1 In the browser, right-click Folder1:1 and select Activate. The geometry that does not belong to this folder is dimmed out.

2 Use LINE to draw two diagonal lines from corner to corner on the rectangle. Note that the lines appear in the other instance as soon as the command is completed.



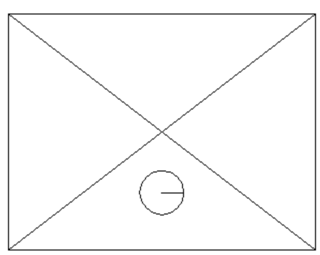
3 Double-click a vacant area in the browser to reset activation.

Nesting Folders

Like blocks, folders can be nested. However, a folder cannot be nested within itself, which is about the only restriction on folder nesting.

1 Draw a small circle in the lower triangle in the second instance of the folder.

2 Draw a line from the center of the circle to the 3 o'clock quadrant of the circle.



3 In the browser, right-click Folder1:2 and select New Folder.

4 Respond to the prompts as shown:

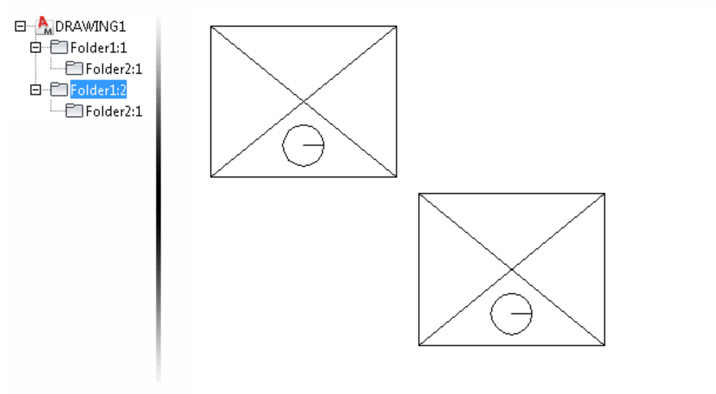
`Enter folder name <Folder2>:` *Press* ENTER

`Select objects for new folder:`

Select the circle and then the line, press ENTER

`Specify base point:` *Select the center of the circle.*

5 Expand Folder1:1 and Folder 1:2 and verify that a nested folder was created.



6 In the browser, right-click Folder1:2 again and select Insert Folder.

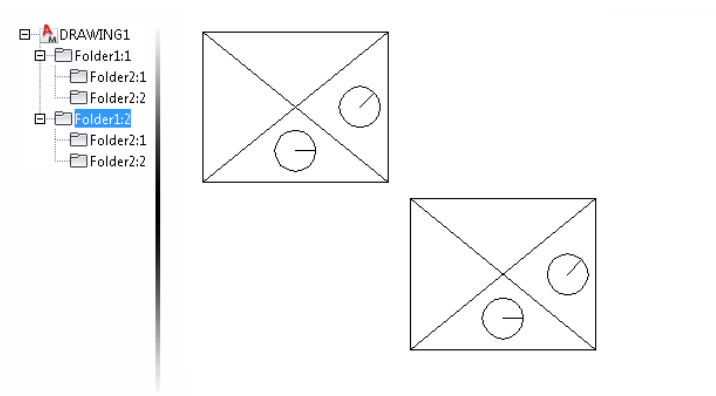
7 Respond to the prompts as below:

`Enter folder name to insert or [?] <?>:` *Enter Folder2, press* ENTER

`Specify the insertion point or [change Base point/Rotate 90]:`

Click in the triangle on the right, in the second instance of Folder1

`Specify rotation angle <0>:` *Enter 45, press* ENTER



Notice that when you added the nested folders, both instances updated, as when you added the lines. Folder2:1 was created as a child of Folder1:2 because we chose New Folder from its context menu, and Folder2:2 was inserted into Folder1:2 for the same reason. Note that as with blocks, you were able to rotate the folder instance on insertion.

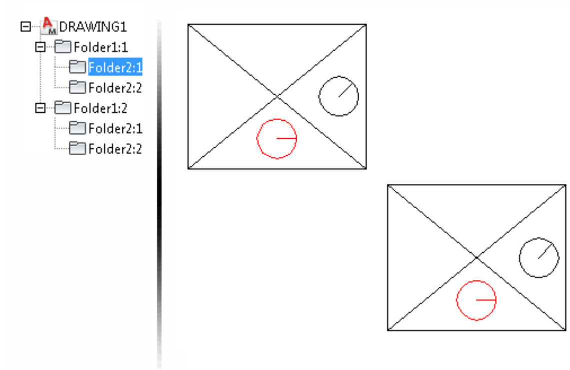
Instance vs. Occurrence

To finish with folders, you inspect a few browser functions such as visibility and property overrides. While performing these exercises you learn the difference between instances and occurrences.

To override properties

1. In the browser, right-click Folder1:1 and select Property Overrides.
2. In the Property Overrides dialog box, select the Override Properties check box.
3. Select the Color check box, The default color changes to red.
4. Click OK.

   Note how the entire instance, inclusive of the nested folders is now red. Also note how the color change did not have an effect on Folder1:2.
5. In the browser, right-click Folder1:1 again, and select Property Overrides
6. In the Property Overrides dialog box, clear the Enable overrides check box, and click OK.
7. In the browser, right click Folder1:1 ➤ Folder2:1 and select property overrides.
8. Apply a color override of red to the folder.



The subfolder you selected is now red, but the other subfolder is not. Notice that the same subfolder under Folder1:2 has changed color to red. This is because property overrides are *instance-based*. When you look at visibility you will understand why this matters.

To apply visibility overrides

1 In the browser, right-click Folder1:1 and select Visible. The entire folder is now invisible.

2 In the browser, right-click Folder1:1 and select Visible. The folder is visible again.

3 In the browser, right-click Folder1:1\Folder2:2 and select Visible. Notice that unlike the property overrides, both instances of Folder2 are visible in Folder1:2. That's because visibility is *occurrence-based*.

Selection Modes

There are three status bar buttons that control the different selection modes. These buttons are not visible by default and you must display them first.

To display the selection mode status bar buttons

1 Click the Drawing Status Bar Menu arrow at the right end of the drawing status bar.

2 Turn on the Status Toggles ➤ S-LOCK, Status Toggles ➤ R-LOCK and Status Toggles ➤ Top Down/Bottom up options.

Button	Function
BTM-UP/TOP-DN	Switches the structure selection order between bottom-up and top-down.
R-LOCK	Switches the Reference Lock on and off. When the Reference Lock is on, you cannot select entities in an external folder or view (more on this later).
S-LOCK	Switches the Selection Lock on and off. When the Selection Lock is on, selection is restricted to the active edit target and below.

The next two exercises demonstrate the behavior of the BTM-UP/TOP-DN and S-LOCK selection modes.

To select items when the selection mode is set to top-down

1 Press ESC to clear any preselection.

2 Click the BTM-UP/TOP-DN button and ensure that the text on the button reads TOP-DN.

3 Click one of the circles in Folder1:1. Note the tooltip indicates that you selected the folder, Folder1:1, and not the circle.

4 Click the circle again. Note the tooltip indicates that you selected the nested folder.

5 Click the circle again. Note the tooltip indicates that you have finally managed to select the circle.

6 Click the circle again. Selection cycles to Folder1:1 again.

When the selection mode is set to top-down, the selection sequence begins at the topmost level and ends with the elemental geometry. When the selection mode is set to bottom-up the selection begins with the elemental geometry.

You may want to set the selection mode and repeat the exercise to verify the behavior of the selection modes under the bottom-up.

To select items when S-LOCK is on

1 Press ESC to clear any preselection.

2 In the browser, double-click Folder1:1 to activate it.

3 Click the S-LOCK button and latch it down to turn on the selection lock.

4 Click one of the circles in Folder1:2. Note that the circle is no longer selectable.

5 Click one of the circles in Folder1:1. Grips appear, indicating that selection is possible.

6 Double-click the root of the Mechanical Browser tree to reset activation.

7 Close the drawing. You can save the drawing, if required.

Components and Component Views

You may notice that folders provide some useful features, but they're probably not different enough from blocks to convince you to change over to the structure paradigm. The true potential of mechanical structure becomes visible only when you start dealing with components and component views.

Component Views are basically folders with some extra rules that make them more suitable for mechanical design. You typically need more than one view to fully describe a part or assembly. Folders (and blocks before them) don't offer any mechanism other than naming to associate multiple views of the same part. Components and views solve this by allowing you to collect multiple folders (component views) under a single *Component*.

A component can be a part or assembly, based on its contents (if a component contains another component, it's an assembly). The component also gives you a place to store attributes like description and material. Components don't actually contain geometry; they group the views that contain the geometry. This will begin to make more sense when you create some components and component views.

Creating Part Components

1 Start a new drawing and draw a long thin rectangle (the edge view of a plate).

2 Draw a second rectangle, above the first, having the same width (the top view).

3 Right-click anywhere in the browser, and select New ➤ Component.

4 Respond to the prompts as shown:

`Enter new component name <COMP1>:` *Press* ENTER

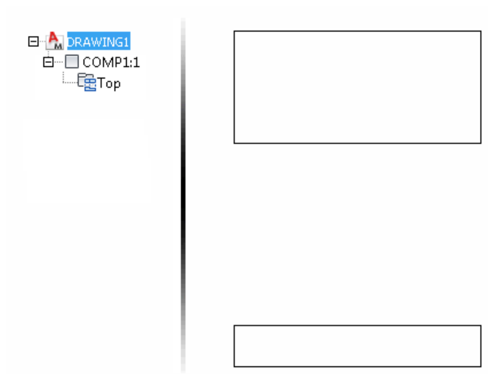
`Enter new view name <Top>:` *Press* ENTER

`Select objects for new component view:`

Select the larger rectangle and press ENTER

`Specify base point:` *Pick the lower left corner of the rectangle*

Note that the Mechanical Browser now displays the component COMP1:1 and that it contains the component view; Top, below it.



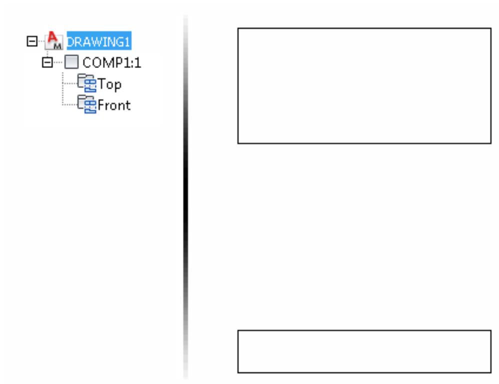
To add a new view to a component

1 In the browser, right-click COMP1:1 and select New ➤ Component View.

2 Accept the default name for the component view.

3 Select the smaller rectangle and press ENTER.

4 To specify a base point, click the lower left corner of the rectangle. Note that the new component view, Front, was added to the component COMP1:1.



Creating Assembly Components

You now have two component views; Front and Top, and they are grouped together in the browser by COMP1:1. In the next exercise, you insert another instance of COMP1 and assemble the two components (parts) in an “L” shape.

To insert a new instance of a component

1 In the browser, right-click a vacant area, and select Insert ➤ Component.

2 Respond to the prompts as shown:

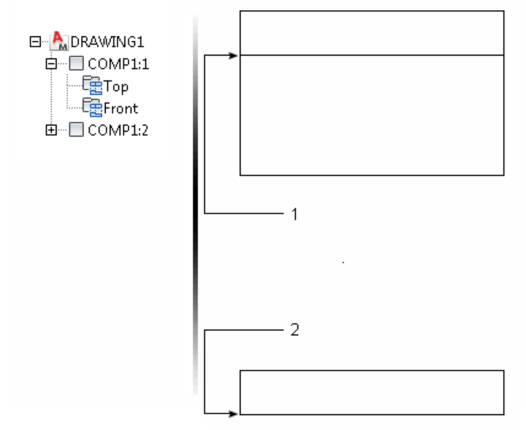
`Enter component name or [?] <?>:` *Enter COMP1 and press* ENTER

`Enter component view name or [?] <Top>:` *Enter Front and press* ENTER

`Specify the insertion point or [change Base point/Rotate 90/select next View]:`

Pick point 1, the top left corner of the larger rectangle

`Specify rotation angle <0>:` *Press* ENTER



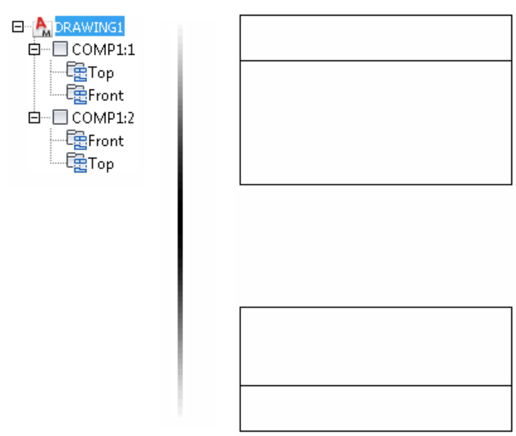
3 In the browser, right-click COMP1:2 and select Insert ➤ Component View ➤ Top.

4 Respond to the prompts as shown:

`Specify the insertion point or [change Base point/Rotate 90/select next View]:`

Pick point 2, the lower left corner of the front view of COMP1:1

`Specify rotation angle <0>:` *Press* ENTER.



To assemble components

1 Right-click anywhere In the browser, and select New ➤ Component.

2 Respond to the prompts as shown:

`Enter new component name <COMP2>:` *Enter ASSY and press* ENTER

`Enter new view name <Top>:` *Enter Front and press* ENTER

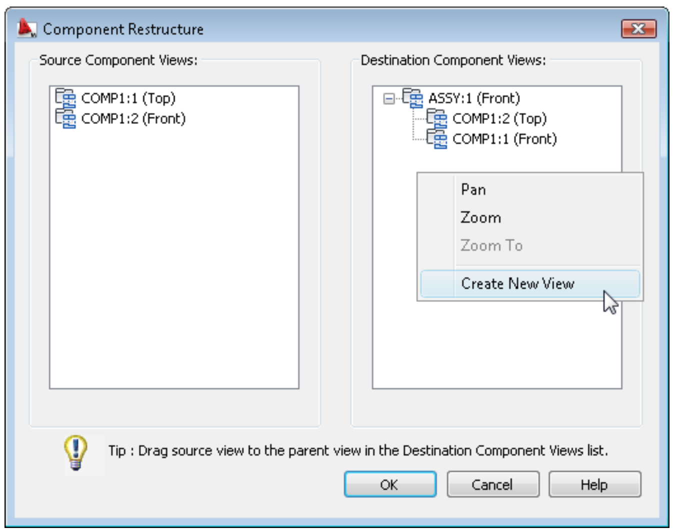
`Select objects for new component view:`

Select COMP1:1 (Front) and COMP1:2 (Top) and press ENTER

To select a component view instead of the geometry, continue clicking the geometry until you see the component view name in the tooltip window. If you accidentally select the wrong view, you can cancel the selection by selecting the view again with the SHIFT key pressed.

`Specify base point:` *Pick the lower left corner of the combined view.*

The Component Restructure dialog box is displayed.



3 In the Destination Components list, right-click a vacant area, and select Create New View.

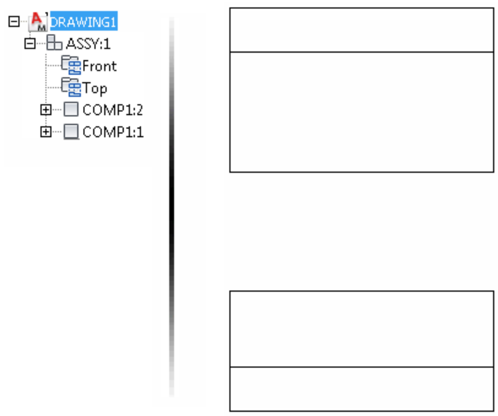
4 Respond to the prompts as shown:

`Enter new view name <Top>:` *Press* ENTER

`Select objects for new component view:`

Select COMP1:1 (Top) and COMP1:2 (Front), press ENTER

`Specify base point:` *Pick the lower left corner of the combined view*



Modifying Assembly Components

As you work, you can continue to add views as needed. To demonstrate this, in the next exercise, you add a side view of this assembly.

To add a component view

1. Draw a rectangle representing the side view of the first instance of COMP1.
2. In the browser, right-click ASSY:1 and select New ➤ Component View.
3. Respond to the prompts as follows:

   `Enter new view name <Right>:` *Enter Side and press* ENTER

   `Select objects for new component view:`

   Don't pick anything, press ENTER

   `Specify base point:` *Pick the lower left corner of the rectangle*

4. In the browser, right-click COMP1:1 and select New ➤ Component View.
5. Respond to the prompts as shown:

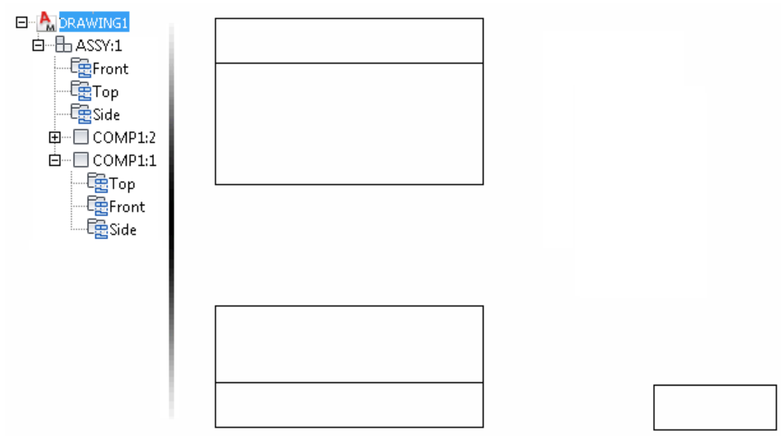
   `Enter new view name <Right>:` *Enter Side, press* ENTER

   `Specify parent view or [?] <Front>:` *Enter Side, press* ENTER

   `Select objects for new component view:`

   Pick the rectangle and press ENTER

`Specify base point:` *Pick the lower left corner of the rectangle*



6 In the browser, right-click COMP1:2 and select Insert ➤ Component View ➤ Side.

7 Respond to the prompts as shown:

`Specify parent view or [?] <Front>:` *Enter Side, press* ENTER

`Specify the insertion point or [change Base point/Rotate 90/select next View]:`

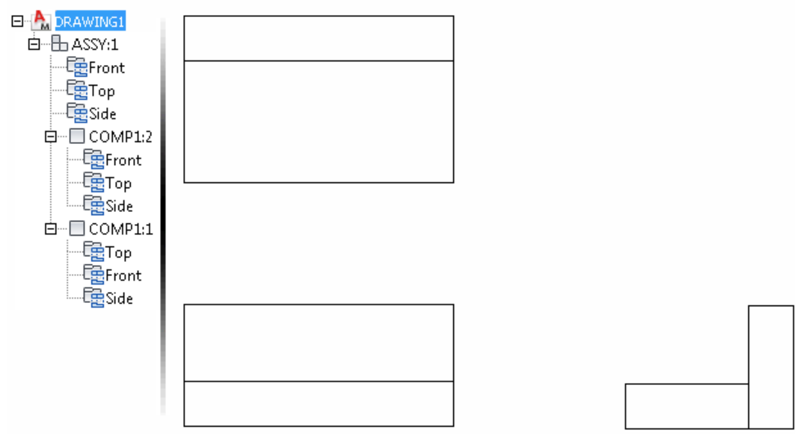
Enter R, press ENTER

`Specify the insertion point or [change Base point/Rotate 90/select next View]:`

Pick a place close to the other view

`Specify rotation angle <90>:` *Press* ENTER

8 Move the view into the correct position.



In the next exercise, you add a component to the assembly to demonstrate the ability to add a component after the assembly is created.

To add a component

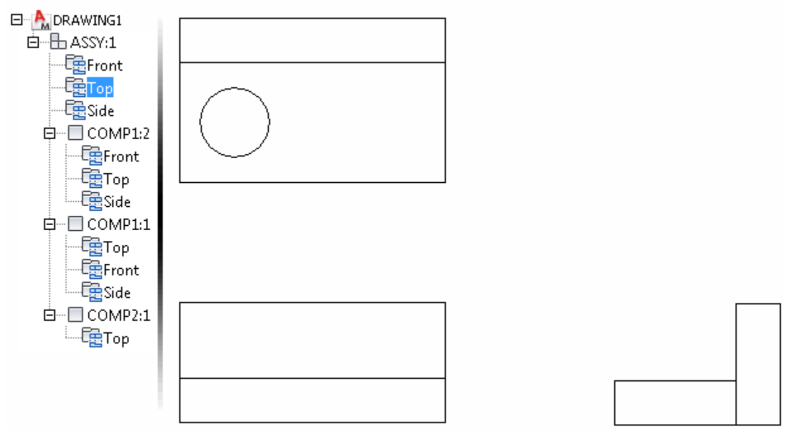
1. Draw a circle on the top view of the assembly.
2. In the browser, right-click ASSY:1(Top) and select New ➤ Component.
3. Respond to the prompts as shown:

   `Enter new component name <COMP2>:` *Press* ENTER

   `Enter new view name <Top>:` *Press* ENTER

   `Select objects for new component view:` *Select the circle, press* ENTER

   `Specify base point:` *Click the center of the circle*



4 Draw a rectangle representing the projected view in the front view of the assembly

5 Right-click COMP2:1 and select New ➤ Component View.

6 Respond to the prompts as shown:

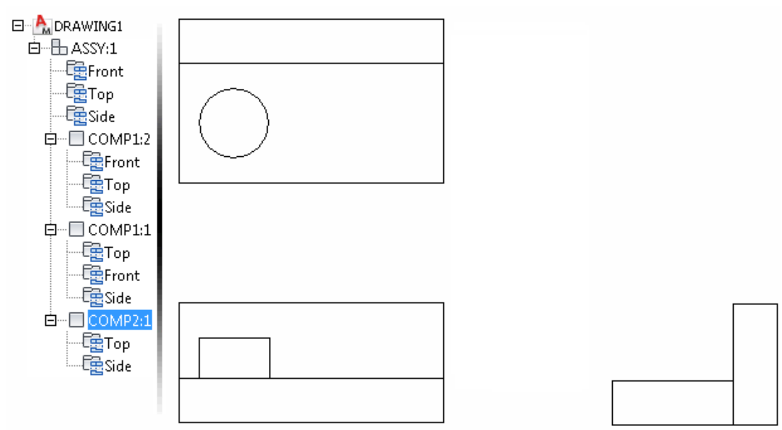
`Enter new view name <Front>:` *Enter Side, press* ENTER

`Specify parent view or [?] <Front>:` *Press* ENTER

`Select objects for new component view:`

Select the rectangle, press ENTER

`Specify base point:` *Pick the midpoint of the lower edge of the rectangle*



7 In the browser, right-click COMP2:1 and select Insert ➤ Component View ➤ Side.

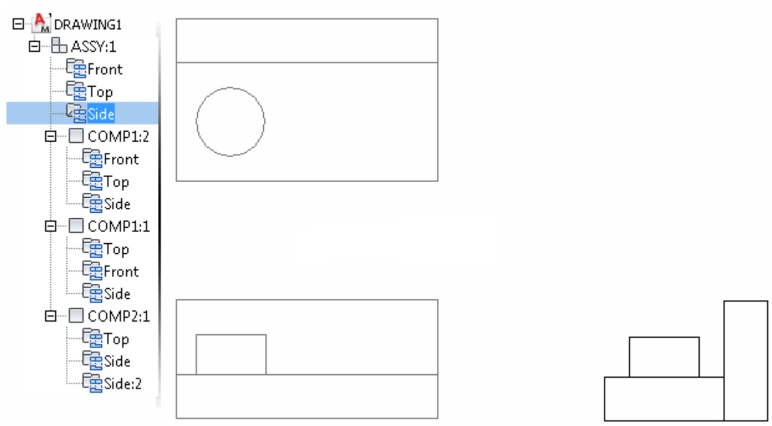
8 Respond to the prompts as shown:

`Specify parent view or [?] <Front>:` *Enter Side, press* ENTER

`Select objects for new component view:`

Select the rectangle, press ENTER

`Specify the insertion point or [change Base point/Rotate 90/select next View]:` *Pick the midpoint of the lower rectangle in the Side view of ASSY1*

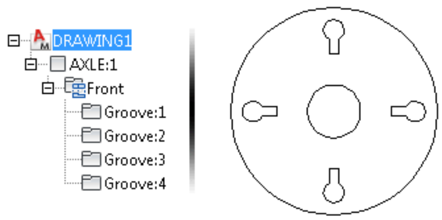


Using Folders with Component Views

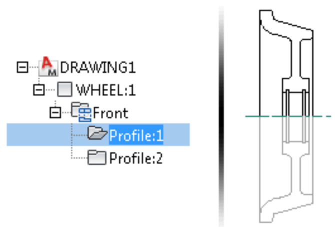
When folders are used in conjunction with component views, you can do several useful things. This section shows two examples.

You can use folders to contain drawing items that would otherwise not be accounted for with a default component view folder.

In the following example, a folder, Groove:1, was created to contain the upper groove and arrayed to create the others. Because the groove is implemented as a folder, it does not have an impact on the BOM. Modifying one of the grooves results in all grooves being updated.



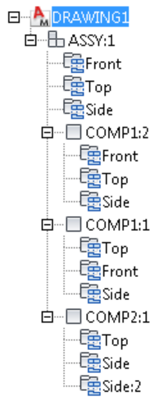
In the following example, a folder, Profile:1, was created to contain the upper-wheel profile. Profile:2 is another instance of this folder, created by mirroring Profile:1. Changing one profile automatically updates the other. The wheel component was created after the Profile folders. The design intent is captured and organized with these folders.



Mechanical Browser Display Options

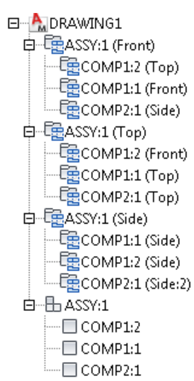
The Mechanical Browser shows the hierarchical organization of components within a drawing. In this section, you use browser options to show data in different ways to get a better understanding of components and component views.

The default view of the Mechanical Browser shows the hierarchical organization of components as well as indicates which component owns a given component view.



To show the View Tree and Component Tree

1. Right-click the root node of the Mechanical Browser and select Browser Options.
2. In the View Tree section, select the Display Tree check box.
3. In the Component Tree section, clear the Component Views check box.
4. Click OK.
5. Right-click a vacant area in the Mechanical Browser and select Expand All.



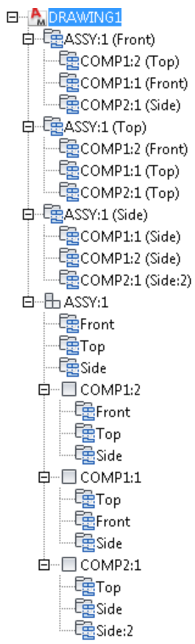
In this view, the hierarchy of components as well as views are shown.

To show both default and expandable assembly views

1 Right-click the root node of the Mechanical Browser and select Browser Options.

2 In the Component Tree section, select the Component Views check box.

3 Click OK.



In this view, the Mechanical Browser shows the hierarchy of components, component views as well as indicates which component owns a given component view. In practice, you can work with the view settings that makes most sense to you.

Mechanical Browser and BOMs

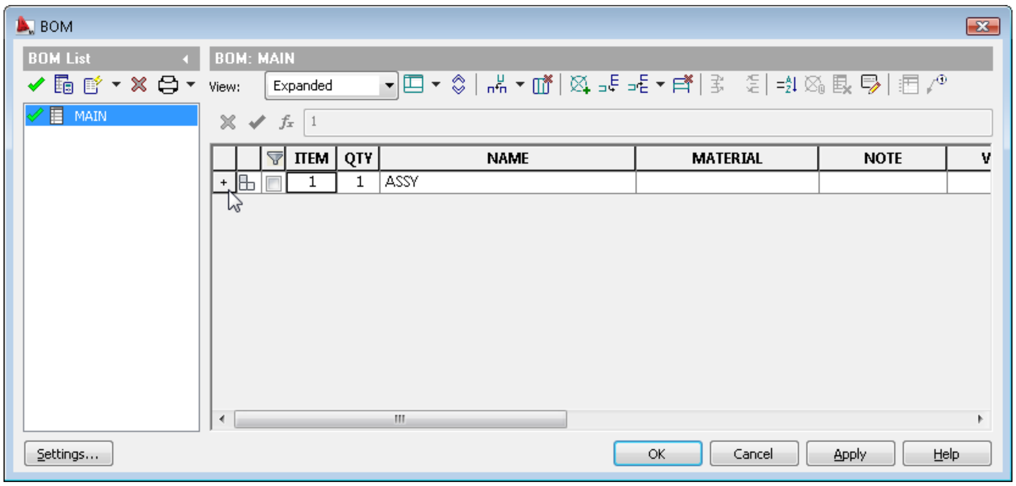
Components not only group component views, they hold bills of material (BOM) attributes as well. In the next exercise, you insert a parts list and in the process, explore the BOM of the simple assembly you created.

To insert a parts list

1 On the command line, enter *AMBOM*.

2 Respond to the prompts as shown:

`Specify BOM to create or set current [Main/?] <MAIN>:` *Press* ENTER

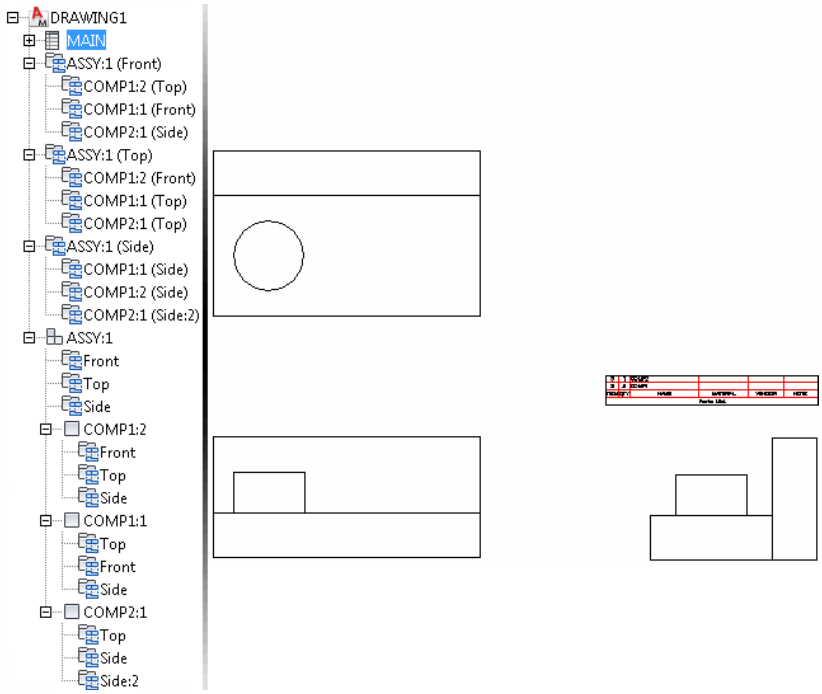


3 In the BOM dialog box, click the plus sign (+) in the first column to expand ASSY.

4 Click the Insert Parts list button on the toolbar of the BOM dialog box.

5 In the Parts List dialog box, click OK and click inside the drawing to indicate where to insert the parts list.

6 In the BOM dialog box, click OK.



By associating views through a single component, the BOM is managed accurately and semi-automatically. You can manage component attributes through the BOM editor or directly on the component from the Mechanical Browser.

Browser Restructure and Ghost Components

In the next exercise you restructure COMP1:1 and COMP2:1 to be parts of an assembly named SUB-ASSY. To do this, you must create SUB-ASSY first.

To create a component

1 Right-click a vacant area in the Mechanical Browser and choose New ➤ Component.

2 Respond to the prompts as follows:

`Enter new component name <COMP3>:` *Enter SUB-ASSY, press* ENTER

`Enter new view name <Top>:` *Press* ENTER

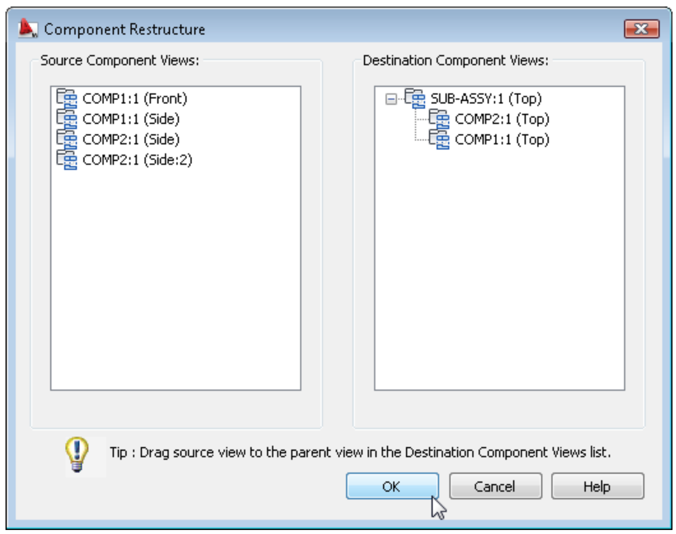
`Select objects for new component view:`

Select COMP1:1 (Top) and COMP2:1 (Top), press ENTER

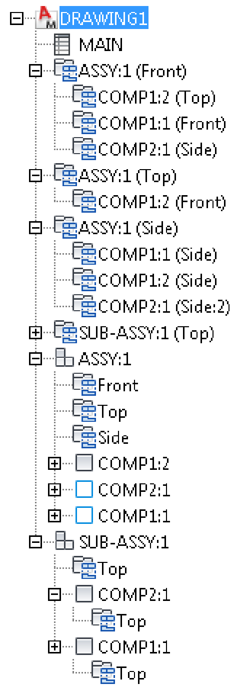
To select a component view instead of the geometry, continue clicking the geometry until you see the component view name in the tooltip window. If you accidentally select the wrong view, you can cancel the selection by selecting the view again with the SHIFT key pressed.

`Specify base point:` *Pick the lower left corner of the combined view*

The Component Restructure dialog box is displayed.



3 Observe the Mechanical Browser.



Note that the component SUB-ASSY is already created (1) and COMP1:1 and COMP2:1 are components of it. Also, the COMP1:1 and COMP2:1 continue to exist as components of ASSY1 (2), but the icon changed. This icon indicates that the component is a *Ghost Component*. Ghost components are containers of the views of components that are in an intermediate state of restructure.

To learn how to resolve ghost components, you must stop creating SUB-ASSY at this point.

4 Click OK. You now have two ghost components in the Mechanical Browser.

Before you start resolving ghost components, you must add two component views to the component SUB-ASSY.

5 In the Mechanical Browser, right-click SUB-ASSY:1 and select New ➤ Component View.

6 Respond to the prompts as follows:

`Enter new view name <Front>:` *Press* ENTER

```
Select objects for new component view:
```

Don't pick anything, press ENTER

```
Specify base point:
```

Pick the lower left corner of the large rectangle in the lower left of the drawing

7 In the Mechanical Browser, right-click SUB-ASSY:1 and select New ➤ Component View again.

8 Respond to the prompts as shown:

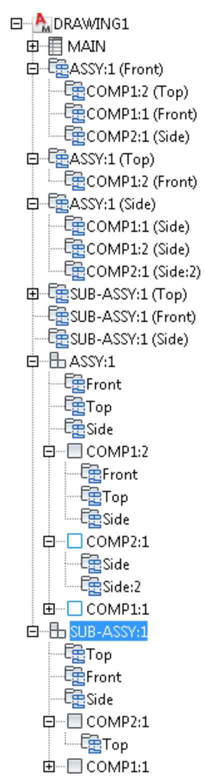
`Enter new view name <Right>:` *Enter Side, press* ENTER

```
Select objects for new component view:
```

Don't pick anything, press ENTER

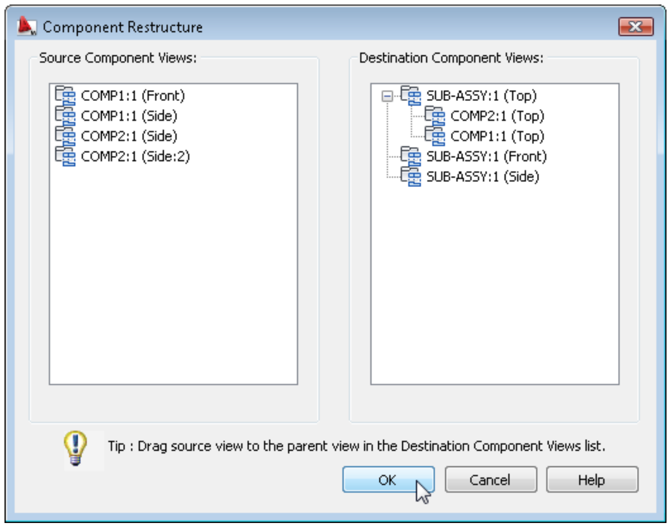
```
Specify base point:
```

Pick the lower left corner of the assembly displayed in the lower right of the drawing

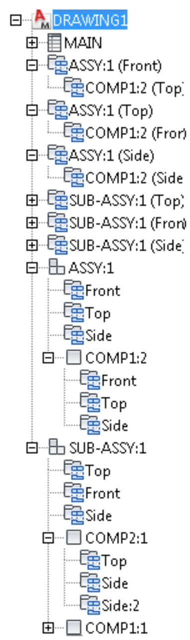


To resolve ghost components

1. In the Mechanical Browser, click the ghost component COMP1:1, press the CTRL key and click COMP2:1. Both components are selected.
2. Drag to SUB-ASSY1. The Component Restructure dialog box is displayed.



3. In the Source Component Views list, with the CTRL key pressed select COMP1:1(Front) and COMP2:1(Side).
4. Drag to SUB-ASSY1(Front). The views move from the Source Component Views list to the Destination Component Views list.
5. Drag the remaining views in the Source Component Views list to SUB-ASSY1:(Side) in the Destination Component Views list.
6. Click OK.



The ghost components disappear and COMP1:1 and COMP2:1 are now parts of SUB-ASSY1.

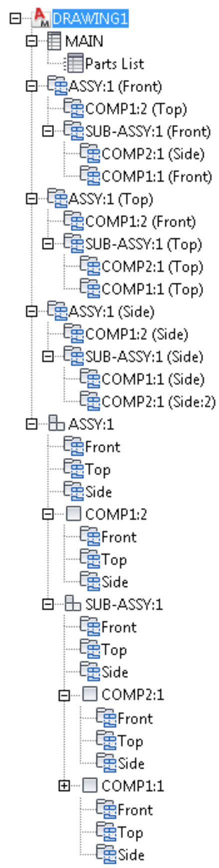
In the final exercise of browser restructure, you restructure SUB-ASSY1 to be a subassembly of ASSY1.

To restructure components

1 In the Mechanical Browser, drag SUB-ASSY:1 ➤ Front to ASSY:1 ➤ Front. The Restructure components dialog box is displayed.

2 Drag SUB-ASSY:1 (Top) to ASSY:1 (Top) and SUB-ASSY:1 (Side) to ASSY:1 (Side).

3 Click OK. SUB-ASSY1 is restructured as a subassembly of ASSY:1



External Reference Components

In AutoCAD Mechanical, you can save individual parts and subassemblies in external files and share them between designs. When a part is modified, the changes are propagated to all instances, ensuring that assembly drawings are always synchronized with their related part drawings.

Inserting External Components

In this exercise, you insert a Gripper on to a Gripper Plate drawing.

1 Open the file *Tut_Gripper_Plate.dwg* in the tutorials folder.

Ribbon	None.
Menu	File ➤ Open...
Command	OPEN

NOTE The path to the tutorials folder is;

- **Windows Vista®**:*C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows® XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains two views of a gripper plate and contains two construction lines.

2 To keep the original file intact, save the file as *Gripper.dwg*

3 Display the Structure Catalog.

Ribbon	None.
Menu	Tools ➤ Palettes ➤ Structure Catalog
	Structure ➤ Structure Catalog...
Command	AMSCATALOG

4 In the External Drawings tab, navigate to the tutorials folder and select *Tut_Gripper.dwg*.

NOTE In the Structure Catalog, navigate to the following folders:

- **Windows Vista:** *C:\Users\Public\Documents\Autodesk\ACADM*

  2010\Acadm\Tutorial
- **Windows XP:** *C:\Documents and Settings\All Users\Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The Mechanical Structure panel shows the mechanical structure components in the drawing and the Preview panel shows a preview of the drawing.

5 In the structure panel, double click GRIPPER to reveal the components list.

6 Click and drag GRIPPER ➤ Front to model space.

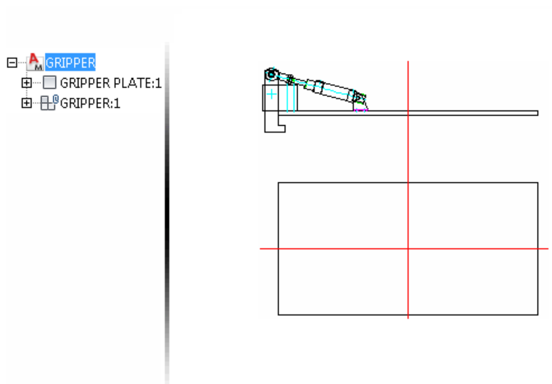
7 Respond to the prompts as shown:

`Specify the insertion point or [change Base point/Rotate 90/select next View]:`

Pick the upper left corner of the smaller rectangle

`Specify rotation angle <0>:` *Press* ENTER

Note the Mechanical Browser. The external reference (xref) component is indicated by a blue colored marker.



Once one view of an xref component is inserted, the other views can be inserted as normal.

To insert another view of the xref component

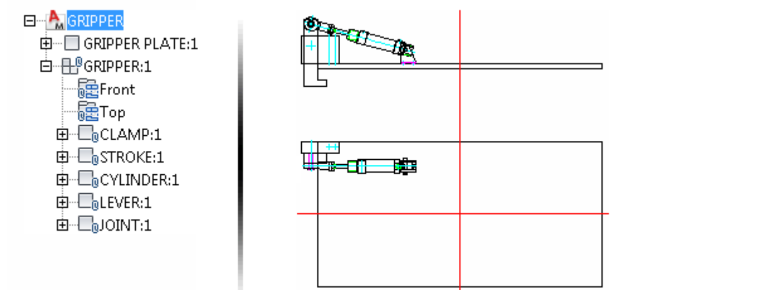
1 In the Mechanical Browser, right-click GRIPPER1 and select Insert from Xref Drawing ➤ Component View ➤ Top.

2 Respond to the prompts as shown:

`Specify the insertion point or [change Base point/Rotate 90/Select nextView]:`

Pick the upper left corner of the larger rectangle

`Specify rotation angle <0>:` *Press* ENTER



To insert more instances of the xref component

1. On the command line enter *MIRROR* and press ENTER.
2. Respond to the prompts as shown:

   `Select objects:`

   Ensure that the selection mode is set to TOP-DN and in model space, click both xref views you inserted, press ENTER

   `Specify first point of mirror line:`

   Click anywhere on the vertical construction line

   `Specify second point of mirror line:`

   Click elsewhere on the vertical construction line

   `Erase source objects? [Yes/No] <N>:` *Enter N and press* ENTER

3. In the last column of the Component View Instance Created dialog box, select New.

   A new instance of the component, GRIPPER:2 is created. You will now mirror the top views of GRIPPER:1 and GRIPPER:2 to draw two more grippers on the top view of the plate.

4. On the command line enter *MIRROR* and press ENTER.
5. Respond to the prompts as follows:

   `Select objects:`

   Ensure that the selection mode is set to TOP-DN and in model space, click the top views of the two grippers, press ENTER

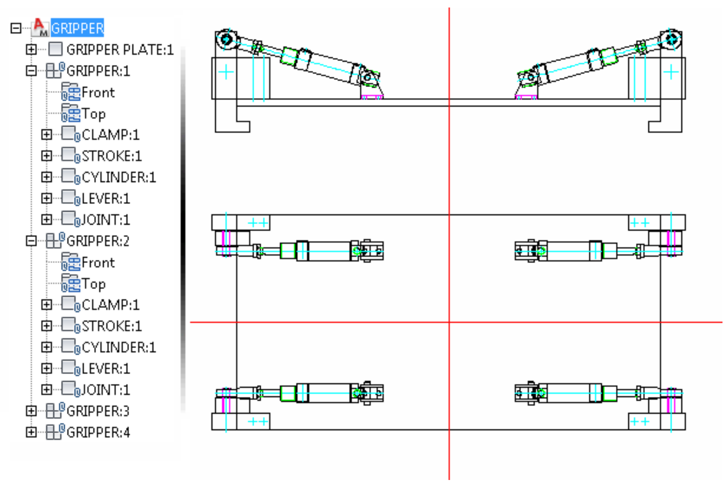
   `Specify first point of mirror line:`

   Click anywhere on the horizontal construction line

   `Specify second point of mirror line:`

Click elsewhere on the horizontal construction line

`Erase source objects? [Yes/No] <N>:` *Enter N and press* ENTER



Next, you assemble the components under an assembly, named GRIPPER ASSEMBLY.

To assemble components

1 Right-click a vacant area in the Mechanical Browser, and select New ➤ Component.

2 Respond to the prompts as shown:

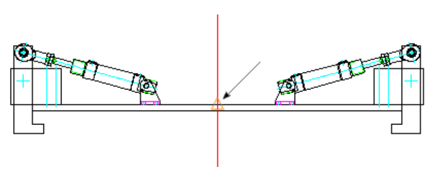
`Enter new view name <Top>:` *Enter Front and press* ENTER

`Select objects for new component view:`

Ensure that the selection mode is set to TOP-DN and window select the smaller rectangle and the two grippers connected to it and press ENTER

`Specify base point:`

Pick the intersection of the construction line with the upper edge of the rectangle



The Component Restructure dialog box is displayed.

3 In the Destination Components list, right-click a vacant area, and select Create New View.

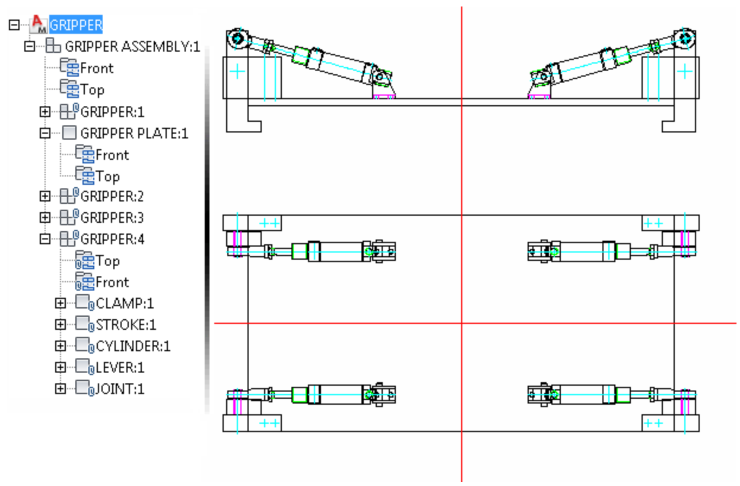
4 Respond to the prompts as shown:

`Enter new view name <Top>:` *Press* ENTER

`Select objects for new component view:`

Ensure that the selection mode is set to TOP-DN and window select the larger rectangle and the four grippers connected to it and press ENTER

`Specify base point:` *Pick the intersection of the two construction lines*



5 Save the file as *Gripper Assembly.dwg*.

Editing External Components In-place

In AutoCAD Mechanical, you can edit xref components in-place. Although this is very convenient, if you accidentally modify a component, the mistake affects all drawings that use this xref component. As a precaution, you must do one of the following before you edit an xref component:

- Release the R-LOCK status bar button.
- Activate the xref component view or folder to be edited.

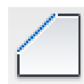
In the next exercise, you modify the gripper lever using the activate method.

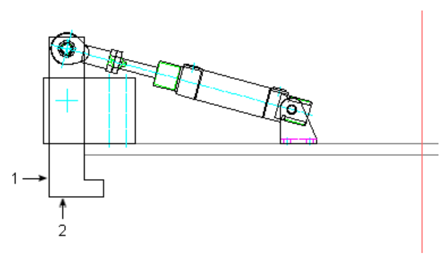
To edit an xref component in place

1. In the Mechanical Browser, double-click Gripper ➤ Front to activate it.

   Notice that locks appear on all instances of the gripper in the Mechanical Browser. This indicates that the source file containing the gripper is now locked and no one else can modify it.

2. Start the Chamfer command.

Ribbon	Content Edit tab ➤ Modify panel ➤ Chamfer. 
Menu	Modify ➤ Chamfer
Command	AMCHAM2D



3. Respond to the prompts as shown:

   Select first object or [Polyline/Setup/Dimension]: <Setup>

   Press ENTER

4. In the Chamfer dialog box, select 10 as the first and second chamfer lengths, and click OK

5. Respond to the prompts as shown:

   Select first object or [Polyline/Setup/Dimension]: <Setup>

   Select the left vertical line of the gripper lever (1)

   Select second object or <Return for polyline>:

   Select the lower horizontal line of the gripper lever (2)

   Select object to create original length: *Press* ESC

6. In the Mechanical Browser, double-click a vacant area to reset activation.

   Note that although the xref component view is no longer the active edit target, the gripper continues to be locked.

7 In the Mechanical Browser, right-click a vacant area, and select Purge All Locks.

8 In the Purge Locks message box, click OK.

To verify if the changes were written back to the source file

- In the Mechanical Browser, right-click GRIPPER1 and select Open to Edit. The Gripper source file opens.

Note that the component view Open Position has also been modified. How did this happen?

Expand the component Lever1. Notice that it has two instances of the component view Front. Another example of how mechanical structure can eliminate repetitive tasks.

Localizing and Externalizing

To modify a part without effecting other drawings that use the part, you can localize the xref component. By localizing you copy the definition of the xref component to the current drawing and the link with the xref file is severed.

To localize an xref component

1 From the Window menu, switch to the Gripper Assembly drawing.

2 In the Mechanical Browser, right-click the GRIPPER assembly node and select Localize.

3 In the Xref Info message box, click Yes.

The Gripper is no longer an xref component.

To detail a part without loosing associativity between the detail and assembly, you can externalize the part to a file and detail it in that file. In the next exercise you externalize the cylinder component.

To externalize a component

1 In the Mechanical Browser, expand one of the Gripper components and right click CYLINDER:1.

2 Select Externalize.

3 In the New External File dialog box, accept the defaults and click Save.

Note that CYLINDER:1 is an xref component in all instances of the GRIPPER component.

Annotation Views

In some cases, externalizing to detail may be considered excessive. Mechanical Structure provides for creating *Annotation Views*, an associative view of a component purely for the purpose of detailing. Annotation views have no effect on the BOM.

In the next exercise, you create an annotation view for the LEVER component.

To create an annotation view

1. In the Mechanical Browser, expand one of the Gripper components and right-click LEVER:1
2. Select New ➤ Annotation View.
3. Respond to the prompts as follows:

   `Enter annotation view name <LEVER(AV1)>:` *Press* ENTER

   `Select placement location`

   `[Modelspace/existing Layout/ New layout] <existing Layout>:`

   Press ENTER

   `Enter existing layout name <Layout1>:` *Press* ENTER

   `Enter scale or [Calculate] <1:2>:` *Press* ENTER

   `--- Switch to Paperspace ---`

   `Restoring cached viewports - Regenerating layout.`

   `Create labels for all subviews [Yes/No] <No>:` *Press* ENTER

   `Specify base point:`

   Select a point at the center of the A3 paper for the annotation views

   `Specify the insertion point or [change Base point/Rotate 90/select next View/Done] <Done>:`

   Select a point below and to the right of the point you clicked on previously

   `Specify rotation angle <0>:` *Press* ENTER

   `Specify the insertion point or [change Base point/Rotate 90/select next View/Done] <Done>:`

Use object tracking mode for alignment, select a point directly below the point you clicked on previously.

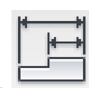
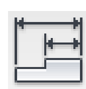
`Specify rotation angle <0>:` *Press* ENTER

`Specify the insertion point or [change Base point/Rotate 90/select next View/Done] <Done>:` *Press* ENTER

NOTE You can type AMSNEW at the command line to display the New dialog box to create annotation views.

To annotate the geometry in the annotation view

1 Start the Automatic Dimension command.

Ribbon	Home tab ➤ Dimension panel ➤ Power Dimension drop-down ➤ Multiple Dimension.  Annotate tab ➤ Dimension panel ➤ Multiple Dimension. 
Menu	Annotate ➤ Multiple Dimensioning
Command	AMAUTODIM

The Automatic Dimensioning dialog box is displayed.

2 In the Type drop-down list, select Chain and click OK.

3 Respond to the prompts as follows:

`Select objects [Block]:`

Window-select the larger of the two views in the annotation view

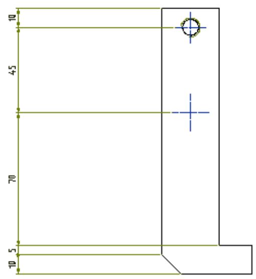
`Select objects [Block]:` *Press* ENTER

`First extension line origin:` *Pick the upper left corner of the geometry*

`Specify dimension line location or [Options/Pickobj]:`

Select a point to the left of the geometry

`Starting point for next extension line:` *Press* ENTER



4 Note the dimension of the chamfer section.

To modify the chamfer in the assembly

1 Switch to model space. In the Mechanical Browser, expand GRIPPER:1, right-click LEVER:1 ➤ Front and select Zoom to.

2 Start the Power Edit command.

Ribbon	None.
Menu	Default Menu ➤ Power Edit
	Edit Menu ➤ Power Edit
	Modify ➤ Power Edit
Command	AMPOWEREDIT

3 Respond to the prompts as shown:

`Select object:` *Select the Chamfer*

4 In the Chamfer dialog box, select 2.5 as the First Chamfer Length and 5 as the Second Chamfer Length, and click OK.

5 Switch to layout1. In the Mechanical Browser, right click LEVER(AV1:1) and select Zoom to.

Note that the Lever shape is changed and the dimensions are updated.

Associative Hide

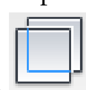
Mechanical structure is all about reuse, especially reuse of components in an assembly to show multiple instances of a component and reuse of component views in the assembly and in the part detail. Component view instances are often obscured in the assembly, sometimes the same view is even obscured differently in different instances. This requires a mechanism to make a folder or component view instance partially or fully hidden without effecting other view instances. Use Associative hide (AMSHIDE) to do that.

Basics of AMSHIDE

In the next exercise, you create a hide situation between two folders.

To create a hide situation

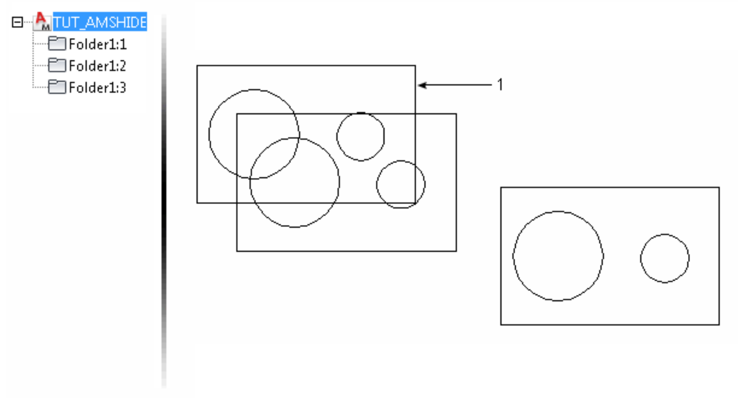
1 Open the file *Tut_AMSHIDE.dwg* in the tutorials folder.

Ribbon	Home tab ➤ Draw Tools panel ➤ Hide Situation drop-down ➤ Create. 
	Structure tab ➤ Hide panel ➤ Create Associative Hide Situation. 
Menu	Modify ➤ Associative Hide ➤ Create Associative Hide Situation
Command	AMSHIDE

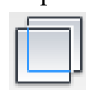
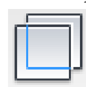
NOTE The path to the tutorials folder is;

- **Windows Vista:** *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows XP:** *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains three instances of a folder, where two overlap each other.



2 Start the Associative Hide command.

Ribbon	Home tab ➤ Draw Tools panel ➤ Hide Situation drop-down ➤ Create. 
	Structure tab ➤ Hide panel ➤ Create Associative Hide Situation. 
Menu	Modify ➤ Associative Hide ➤ Create Associative Hide Situation
Command	AMSHIDE

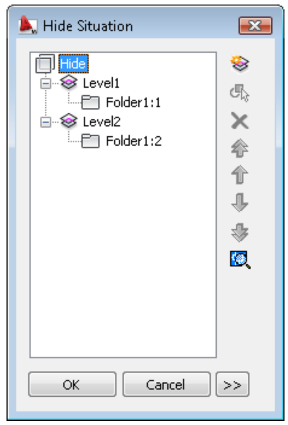
3 Respond to the prompts as follows:

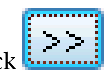
`Select foreground objects:`

Ensure that the selection mode is set to TOP-DN and click the upper rectangle (1)

`Select foreground objects:` *Press* ENTER

4 In the Hide Situation dialog box, expand Level1 and Level2. Note how Folder1 is selected for the foreground (Level1) and Folder2 is selected for the background.



5 If the dialog box is collapsed, as shown in the image above, click  to expand it.

6 Click the Hide node on the tree in the dialog box.

7 Clear the Display hidden lines check box.

The hidden lines are set to invisible. The change is immediately reflected in model space.

8 In the Name box, enter *Test Hide*.

9 To swap the foreground and background, select Level1 on the tree and click the Send to Back button on the toolbar of the dialog box. Note that the position of Level1 changes in the tree and model space reflects the change immediately.

10 Click OK.

In the next exercise, you edit the hide situation and add the third folder to the hide situation.

To edit a hide situation

1 In the Mechanical Browser, if a node named Hide Situations is not visible:

  a Right-click TUT_AMSHIDE on the Mechanical Browser. The Browser Options dialog box is displayed.

  b In the Hide Situations section, select the Display hide situations check box.

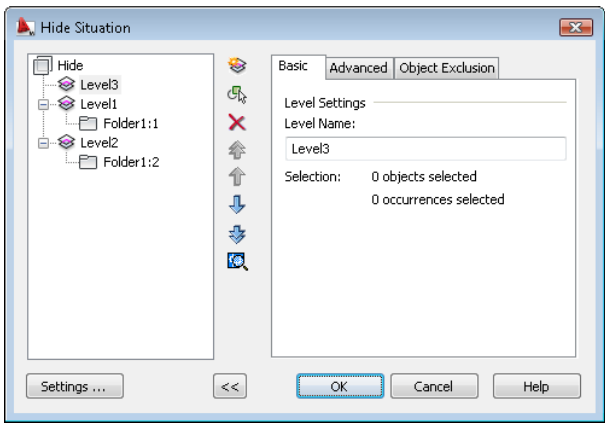
c Click OK.

2 Select Test Hide, from below the Hide Situations node on the Mechanical Browser. Note that the entities involved in the hide are highlighted in model space.

3 Double-click Test Hide.

The Hide situations dialog box is displayed.

4 In the Toolbar of the Hide Situation dialog box, click the  button. Level3 is added to the top of the tree.



5 Click  to select objects for Level3.

6 Respond to the prompts as follows:

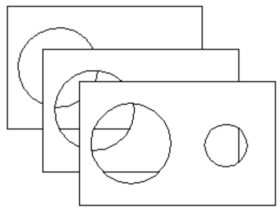
`Select objects:`

Set selection mode to TOP-DN and click the rectangle on the extreme right

`Select objects:` *Press* ENTER

7 Click OK.

8 Use the MOVE command to move the contents of Folder 1:3 on top of Folder1:2 and Folder 1:3.



Using AMSHIDE in Assemblies

In this section, you create a hide situation on an assembly and save it to the appropriate position in the mechanical structure.

To open the sample files

- Open the file *Tut_Robot_Arm.dwg* in the tutorials folder.

Ribbon	None.
Menu	File ➤ Open...
Command	OPEN

NOTE The path to the tutorials folder is;

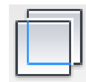
- **Windows Vista:** *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows XP:** *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

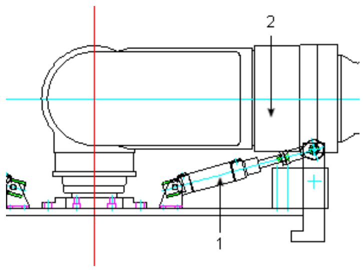
The drawing contains an assembly of a robotic arm, and has external references to the gripper assembly you created during the exercise for external references.

To create a hide situation between the gripper and the axis of the robotic arm

1 Start the Associative Hide command.

Ribbon	Home tab ➤ Draw Tools panel ➤ Hide Situation drop-down ➤ Create. 

	Structure tab ➤ Hide panel ➤ Create Associative Hide Situation. 
Menu	Modify ➤ Associative Hide ➤ Create Associative Hide Situation
Command	AMSHIDE



2 Respond to the prompts as shown:

`Select foreground objects:`

Continue clicking (1) until you see GRIPPER:2 (Front) in the tooltip

`Select foreground objects:` *Press* ENTER

3 Click Level2 in the tree view and select the  button.

4 Respond to the prompts as shown:

`Select objects:`

Continue clicking (2) until you see AXIS:1 (Front) in the tooltip

`Select objects:` *Press* ENTER

5 In the tree view of the Create Hide Situations dialog box, click the Hide node.

In the next step, you select where in the Mechanical Browser the hide situation is stored. The most logical place to store the hide situation is on the Front view of the ROBOT:1 assembly.

6 In the Store Hide Situation on list, select ROBOT:1 (Front).

7 Click OK. The Hide Situation is created and stored under ROBOT:1 (Front) in the Mechanical Browser.

This is the end of the tutorial.

Designing Levers

5

In this tutorial, you start with a lever inserted from the parts library, and then you refine the design using many of the design options available in AutoCAD® Mechanical. You also create a drawing detail and add dimensions to it.

Key Terms

Term	Definition
construction lines	Lines, which are infinite in both directions or rays, which are infinite starting at a point that can be inserted into the drawing area. You use construction lines to transfer important points (for example, center points of holes) into other views or drawing areas.
construction geometry	A line or an arc created with construction lines. Using construction geometry in 2D drawings helps define the shape of a contour.
detail	A portion of a design drawing that cannot be clearly displayed or dimensioned in the overall representation but can be enlarged to show the details.
distance snap	To give the dimensions in a drawing a uniform appearance, Power Dimensioning and Automatic Dimensioning enable automatic insertion of the dimension line at a defined distance from the object being dimensioned. While dragging the dimension line dynamically, you will find that it remains "fixed" and is highlighted in red as soon as the required distance to the object being dimensioned is reached.

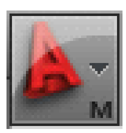
Term	Definition
library	A feature that makes it possible to store parts such as blocks and drawings in a library. For every inserted part, an icon can be created. The icon is put in the display section on the right side of the dialog box along with an assigned name.
power command	A collective term for the Power Copy, Power Recall, Power Edit, Power Dimensioning, Power Erase, and Power View commands.
Power Dimensioning	Power Dimensioning is a very useful tool for generating linear, radial and diameter dimensions, which minimizes the number of the individual actions required while generating a dimension. Power Dimensioning selects the type of linear dimension (horizontal, vertical, or aligned), based on the selected point, and the dimensions of the drawing can have a uniform style using the distance snap.

Extending Designs

First, you start a new drawing template with ISO standard. Then you load the initial drawing using the Library.

To open a template

1 Open a new drawing.

Ribbon	 ➤ New ➤ Drawing
Menu	File ➤ New
Command	NEW

The Select template dialog box opens.

2 In the Select template dialog box, select the template *am_iso.dwt*.

This opens a new drawing template. Now you insert the drawing from the library.

Using Libraries to Insert Parts

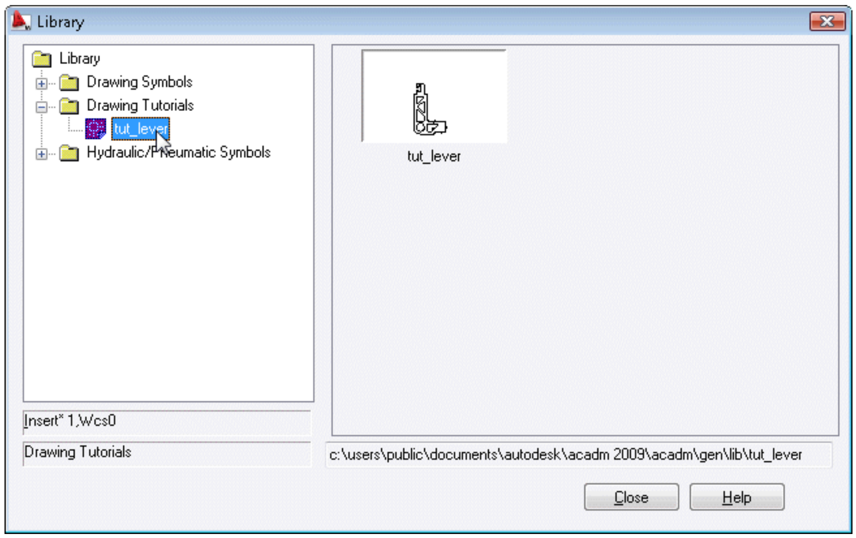
Insert the required part from the library.

To insert a drawing from the library

1 Start the Library.

Ribbon	None.
Menu	Tools ➤ Library...
Command	AMLIBRARY

2 Double-click the *tut_lever.dwg* file in the Library.



3 Respond to the prompt as follows:

`Specify insertion point:` *Specify any point in the drawing*

4 Start the Zoom Window command.

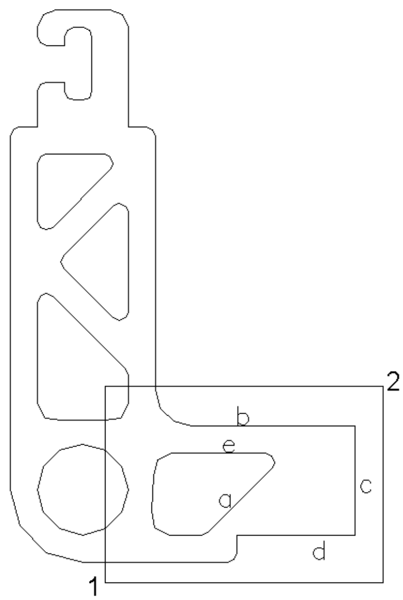
Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

Respond to the prompts as follows:

```
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real
time>:
```
Enter window, press ENTER

`Specify first corner:` *Specify first corner (1)*

`Specify opposite corner:` *Specify opposite corner (2)*



Save your file.

Before starting the design, define the object snaps that you will use in later operations.

Configuring Snap Settings

In addition to the AutoCAD® snap, mechanical snap options like arc radial, arc tangent, and so forth are available. You also have four different snap settings, which can be configured separately for a quick switch to a different snap setting. For example, you can use different snap settings for detailing or general design.

NOTE The snap defaults can be set in the Options dialog box on the AM:Preferences tab.

To configure Power Snap settings

1 Start the Power Snap settings.

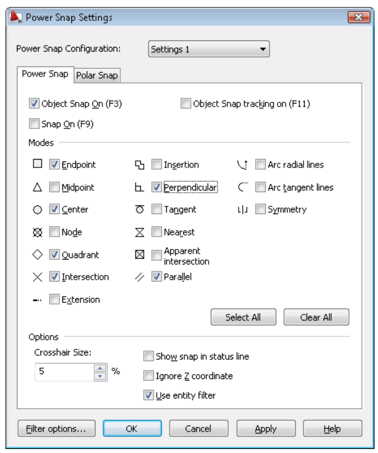
Ribbon	None.
Menu	Object Snap Cursor Menu ➤ Power Snap Settings
Command	AMPOWERSNAP

2 In the Power Snap Settings dialog box:

Settings 1: Endpoint, Intersection

Settings 2: Endpoint, Center, Quadrant, Intersection, Parallel

Settings 3: Perpendicular



3 After configuring the settings, activate Setting 1, by selecting Settings 1 from the Power Snap Configuration drop-down list and then click OK.

Save your file.

NOTE Within a command, the various object snap functions are also accessible. Hold down the SHIFT key, and right-click.

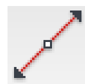
Creating Construction Lines (C-Lines)

Construction lines are very useful when you start your design process. With their help, you draw a design grid with your defined values for distance and angles. After generating the design grid, you simply trace your contour with the contour layer.

Now insert the construction lines, which will help you draw the contour lines.

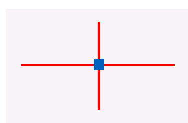
To create construction lines

1 Start the Draw Construction Lines command.

Ribbon	Home tab ➤ Draw Tools panel ➤ Construction Lines. 
Menu	Draw ➤ Construction Lines ➤ Draw Construction Lines...
Command	AMCONSTLINES

The Construction Lines dialog box opens.

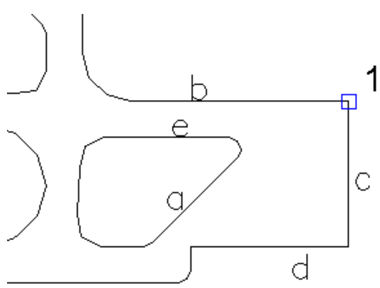
2 In the Construction Lines dialog box, choose the option next to the icon shown below and click OK.



3 Respond to the prompts as follows:

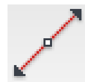
`Specify insertion point:` *Specify the intersection of line b and line c (1)*

`Specify insertion point:` *Press* ENTER



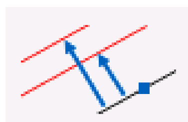
4 Next, draw two lines parallel to the vertical and horizontal lines of the construction line cross.

5 Start the Draw Construction Lines command.

Ribbon	Home tab ➤ Draw Tools panel ➤ Construction Lines. 
Menu	Draw ➤ Construction Lines ➤ Draw Construction Lines...
Command	AMCONSTLINES

The Construction Lines dialog box is displayed.

6 In the Construction Lines dialog box, choose the option next to Parallel construction line with full distance icon shown below and click OK.



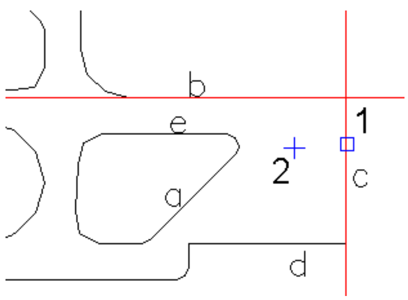
7 Respond to the prompts as follows:

`Select line, ray or xline:` *Select line c (1)*

`Specify insertion point or Distance (xx|xx|xx..) <10|20|30>:`

Enter 3|9, press ENTER

`Specify point on side to offset:` *Specify a point to the left of line c (2)*



8 Insert the second set of parallel lines, and respond to the prompts as follows:

`Select line, ray or xline:` *Select line b*

`Specify insertion point or Distance (xx|xx|xx..) <3|9>:`

Enter 4.5|9.5, press ENTER

`Specify point on side to offset:` *Specify a point below line b (2)*

9 Press ENTER.

Save your file.

Creating additional C-Lines

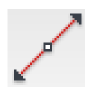
AutoCAD Mechanical offers a large choice of C-line options.

To create additional construction lines

1 Activate snap setting 2.

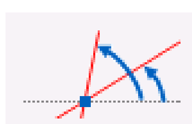
Ribbon	None.
Menu	Object Snap Cursor Menu ➤ Power Snap Settings 1-4 ... ➤ Power Snap Configuration 2 Tools ➤ Drafting Settings ➤ Power Snap Configuration 2
Command	AMPSNAP2

2 Start the Draw Construction Lines command.

Ribbon	Home tab ➤ Draw Tools panel ➤ Construction Lines. 
Menu	Draw ➤ Construction Lines ➤ Draw Construction Lines...
Command	AMCONSTLINES

The Construction Lines dialog box is displayed.

3 In the Construction Lines dialog box, choose the option next to the Construction line by defining two points or an angle icon shown below, and click OK.

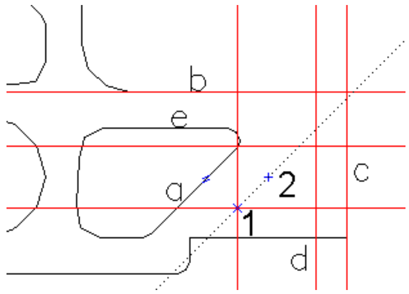


4 Respond to the prompts as follows:

`Specify first point:` *Select the first point (1)*

`Specify second point or Angle (xx|xx|xx..) <30|45|60>:`

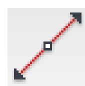
Move the cursor over line a and back to the rectangle until the Parallel symbol appears, click (2)



5 Press ENTER to finish the command.

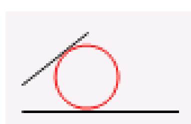
Now, you draw tangential circles between the diagonal C-line and the right vertical line and lower horizontal line of the rectangle.

6 Start the Draw Construction Lines command.

Ribbon	Home tab ➤ Draw Tools panel ➤ Construction Lines. 
Menu	Draw ➤ Construction Lines ➤ Draw Construction Lines...
Command	AMCONSTLINES

The Construction Lines dialog box is displayed.

7 In the Construction Lines dialog box, choose the option next to the Construction line circle tangent to 2 lines icon, shown below and click OK.



8 Draw the two circles by responding to the prompts as follows:

`Select first tangent:` *Select tangent point (1)*

`Select second tangent:` *Select tangent point (2)*

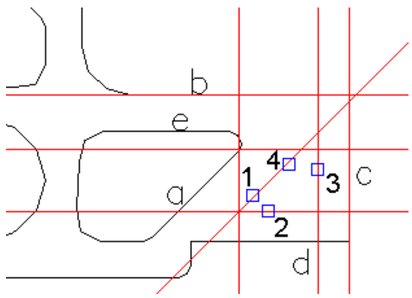
`Specify diameter:` *Enter 2, press* ENTER

`Select first tangent:` *Select tangent point (3)*

`Select second tangent:` *Select tangent point (4)*

`Specify diameter <2>:` *Enter 2, press* ENTER

9 Press ENTER to end the command.



All construction lines have been inserted, and the contour can be generated.

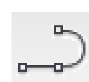
Save your file.

Creating Contours and Applying Fillets

Now, you connect the two tangential circles with the right part of the rectangle, to build a filleted triangle.

To create and edit a contour

1 Start the Polyline command.

Ribbon	Home tab ➤ Draw panel ➤ Polyline. 
Menu	Draw ➤ Polyline
Command	PLINE

2 Create the contour by responding to the prompts as follows:

`Specify start point:` *Specify the intersection point (1)*

`Specify next point or [Arc/Halfwidth/Length/Undo/Width]:`

Specify next point (2)

`Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:`

Enter A, press ENTER

`Specify endpoint of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/ Radius/Second pt/Undo/Width]:` *Specify next point (3)*

`Specify endpoint of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/ Radius/Second pt/Undo/Width]:` *Enter L, press* ENTER

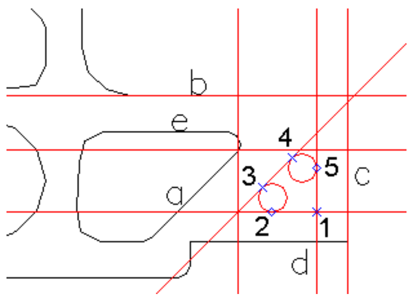
`Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:`

Specify next point (4)

`Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:`
Enter A, press ENTER

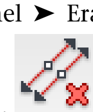
`Specify endpoint of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/ Radius/Second pt/Undo/Width]:` *Specify next point (5)*

`Specify endpoint of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/ Radius/Second pt/Undo/Width]:` *Enter CL, press* ENTER



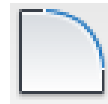
Now, erase the C-Lines. You can erase all C-lines by calling one command.

3 Erase all C-Lines.

Ribbon	Home tab ➤ Draw Tools panel ➤ Erase Construction Lines drop-down ➤ All. 
Menu	Modify ➤ Erase ➤ Erase all Construction Lines
Command	AMERASEALLCL

NOTE You can switch construction lines on and off temporarily by choosing Assist ➤ Layer/Layergroup ➤ Construction Line On/Off.

4 Apply a fillet to the corner of the triangle.

Ribbon	Home tab ➤ Modify panel ➤ Fillet. 
Menu	Modify ➤ Fillet
Command	AMFILLET2D

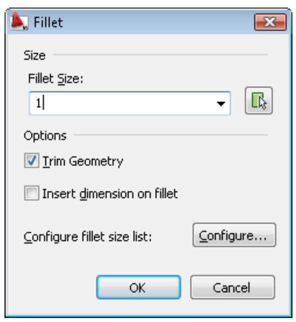
5 Respond to the prompts as follows:

```
(Dimension mode:OFF)(Trim mode) Current fillet radius = 2.5
Select first object or [Polyline/Setup/Dimension] <Setup>:
```

Press ENTER. The Fillet dialog box is displayed.

6 In the Fillet size list, select 1.

7 Select the Trim Geometry check box:



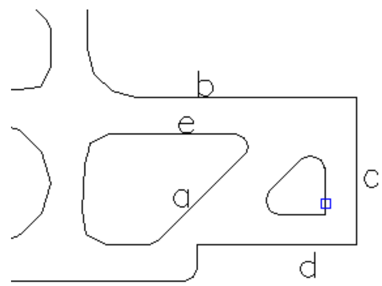
Click OK.

8 Respond to the prompts as follows:

```
(Dimension mode:OFF)(Trim mode) Current fillet radius = 1
Select first object or [Polyline/Setup/Dimension] <Setup>:
```

Enter P, press ENTER

`Select polyline:` *Select a point on the polyline near the corner*



9 Press ESC to cancel the command.
The triangular contour is complete.
Save your file.

Trimming Projecting Edges on Contours

Now, you create another part of the contour and trim projecting edges.

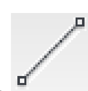
To edit a contour

1 Activate Power Snap Setting 3 command.

Ribbon	None.
Menu	Object Snap Cursor Menu ➤ Power Snap Settings 1-4 ... ➤ Power Snap Configuration 3
	Tools ➤ Drafting Settings ➤ Power Snap Configuration 3
Command	AMPSNAP3

Next, insert the next contour.

2 Start the Line command.

Ribbon	Home tab ➤ Draw panel ➤ Line. 
Menu	Draw ➤ Line
Command	LINE

3 Respond to the prompts as follows:

`Specify first point:`

Hold down the SHIFT *key, right-click, and choose Intersection from the menu* `_int of:` *Select line a (1)*

`and:` *Select intersection on line b (2)*

`Specify next point or [Undo]:`

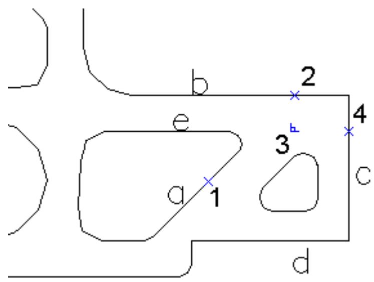
Hold down the SHIFT key*, right-click, and choose Perpendicular from the menu*

`_per to:` *Select line e*

`Specify next point or [Undo]:`

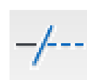
Drag the cursor to the right, crossing over line c, and select intersection point (4)

`Specify next point or [Close/Undo]:` *Press* ENTER



Now, trim the projecting edges at the upper edge of the lever.

4 Start the Trim command.

Ribbon	Home tab ➤ Modify panel ➤ 
Menu	Modify ➤ Trim
Command	TRIM

5 Respond to the prompts as follows:

`Current settings: Projection = UCS, Edge = None`

`Select cutting edges:`

`Select Objects:` *Select cutting edge (1)*

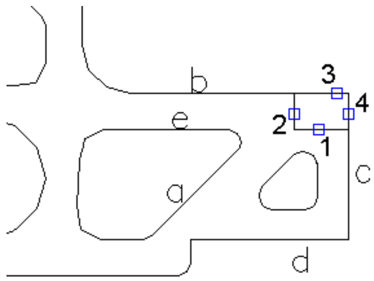
`Select Objects:` *Select cutting edge (2)*

`Select Objects:` *Press* ENTER

`Select object to trim or shift-select to extend or [Project/Edge/Undo]:` *Select object to trim (3)*

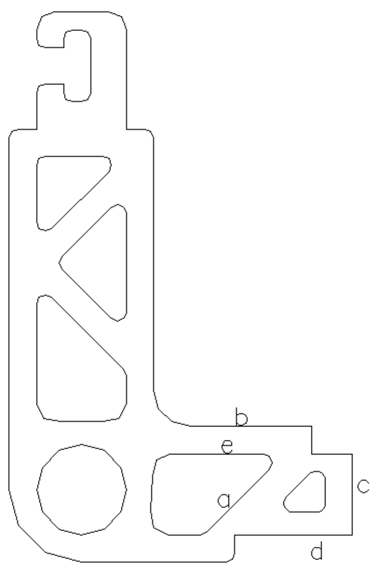
`Select object to trim or shift-select to extend or [Project/Edge/Undo]:` *Select object to trim (4)*

`Select object to trim or shift-select to extend or [Project/Edge/Undo]:` *Press* ENTER



6 Zoom to the extents of the lever.

The contour is complete and looks like this. Save your file.

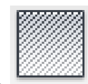


Applying Hatch Patterns to Contours

There are a number of predefined hatch patterns available in AutoCAD Mechanical. Choose one of the predefined hatching styles, and then specify a point within a contour to apply the hatching.

To apply hatching to a contour

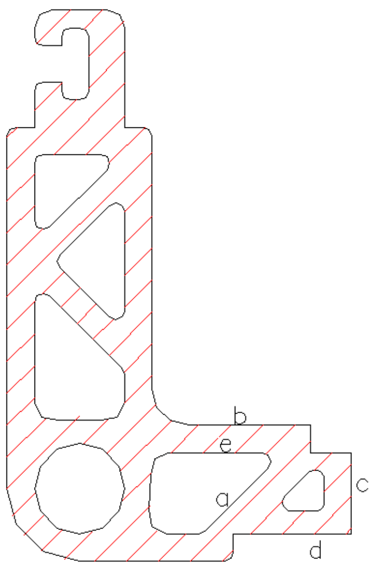
1 Start the Hatch command, using an angle of 45 degrees and 2.5 mm / 0.1 inch spacing.

Ribbon	Home tab ➤ Draw Tools panel ➤ Hatch drop-down ➤ 45 deg. 2.5 mm/0.1 inch. 
Menu	Draw ➤ Hatch ➤ Hatch 45 deg. 2.5 mm/0.1 inch
Command	AMHATCH_45_2

2 Respond to the prompt as follows:

`Select additional boundary or point in area to be hatched or [Select objects]:` *Click a point inside the contour (outside the cutouts)*

The lever is hatched. It looks like this:



Save your file.

Dimensioning Contours

Now, dimension the lever, using the Power Dimensioning command.

To dimension a contour

1. Start the Power Snap Setting 1 command.

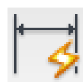
   Ribbon None.

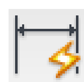
   Menu Object Snap Cursor Menu ➤ Power Snap Settings 1-4 ... ➤ Power Snap Configuration 1

   Tools ➤ Drafting Settings ➤ Power Snap Configuration 1

   Command AMPSNAP1

2. Start the Power Dimensioning command.

   Ribbon Home tab ➤ Dimension panel ➤ Power Dimension. 

   Annotate tab ➤ Dimension panel ➤ Power Dimension. 

   Menu Annotate ➤ Power Dimensioning

   Command AMPOWERDIM

3. Respond to the prompts as follows:

   ```
   Specify first extension line origin or
   [Linear/Angular/Radial/Baseline/Chain/Options/Update] <select
   object>:
   ```

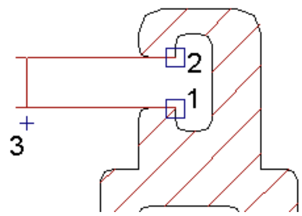
   Select the first corner point of the lever opening (1)

   ```
   Specify second extension line origin:
   ```

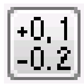
   Select the second corner point (2)

   ```
   Specify dimension line location or
   [Horizontal/Vertical/Aligned/Rotated/Placement options]:
   ```

   Drag the dimension line to the left until it is highlighted in red, click (3)



4 In the Power Dimensioning dialog box, click the Add Tolerance button

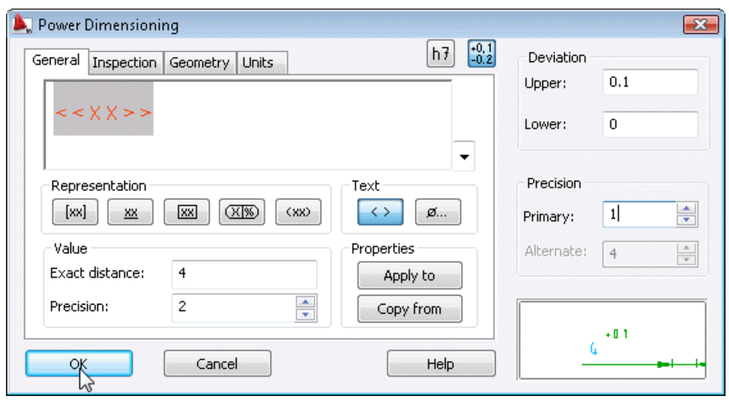


 and specify:

`Deviation: Upper:` *0.1*

`Deviation: Lower:` *0*

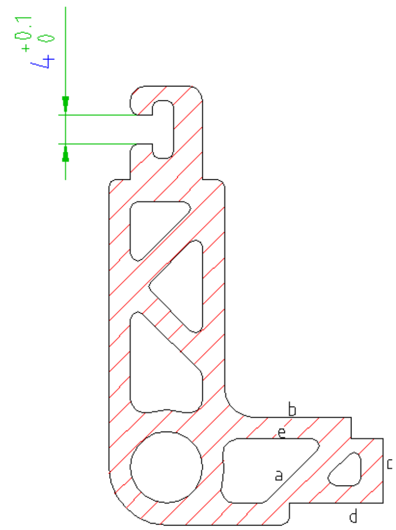
`Precision: Primary:` *1*



Click OK.

5 Press ENTER twice to finish the command.

The lever looks like this:



Save your file.

Creating and Dimensioning Detail Views

Now, define a detail of the upper part of the lever.

To create a detail

1 Start the Detail command.

Ribbon	Home tab ➤ Draw Tools panel ➤ Detail View. 
Menu	Draw ➤ Detail...
Command	AMDETAIL

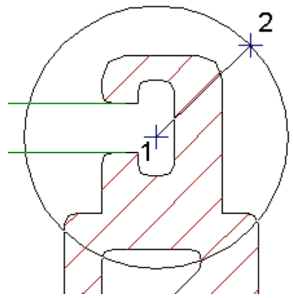
2 Respond to the prompts as follows:

`Center of circle or [Rectangle/Object]:`

Click a point in the center of the area to be detailed (1)

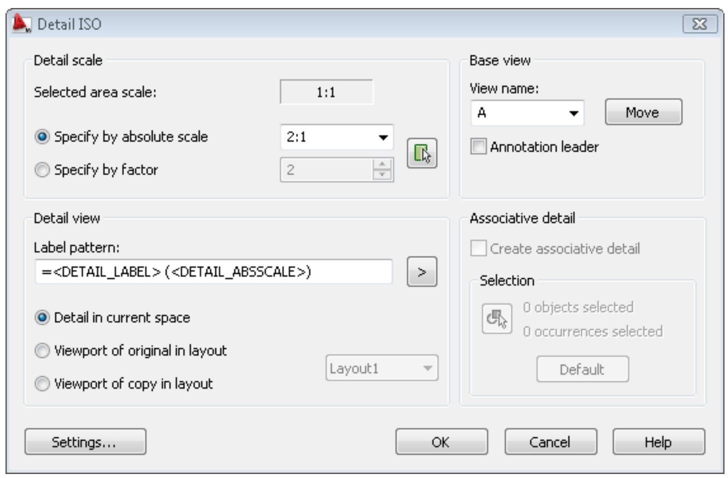
`Specify radius or [Diameter]:`

Drag the radius to the appropriate size, click (2)



3 In the Detail dialog box, specify:

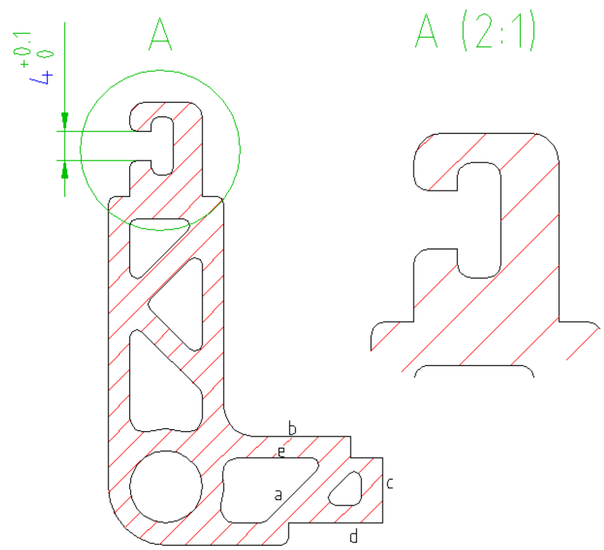
`Detail View: Detail in Current Space`



4 Click OK, and respond to the prompts as follows:

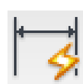
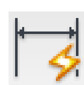
`Place the detail view:` *Select a location to the right of the lever*

NOTE Some entities such as dimensions and symbols are automatically filtered out in the detail function.



Now, add a dimension to the detail.

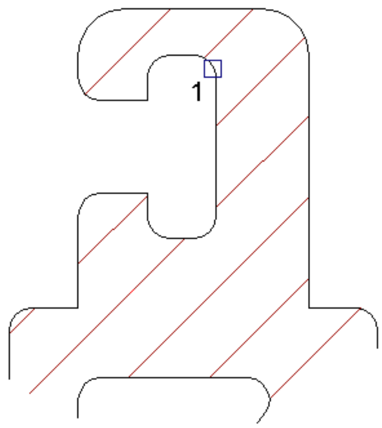
5 Start the Power Dimensioning command.

Ribbon	Home tab ➤ Dimension panel ➤ Power Dimension. 
	Annotate tab ➤ Dimension panel ➤ Power Dimension. 
Menu	Annotate ➤ Power Dimensioning
Command	AMPOWERDIM

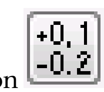
6 Respond to the prompts as follows:

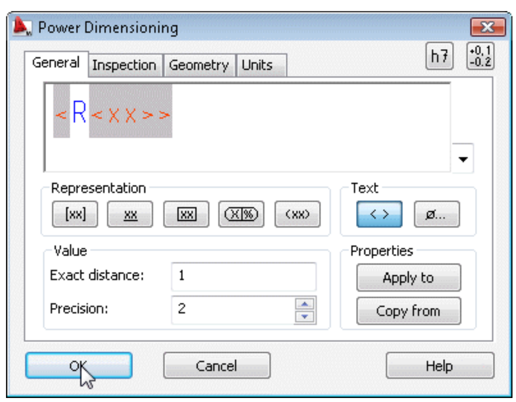
`Specify first extension line origin or [Linear/Angular/Radial/Baseline/Chain/Options/Update] <select object>:` *Press* ENTER

`Select arc, line, circle or dimension:` *Select the radius (1)*



7 Select an appropriate position for the dimension.

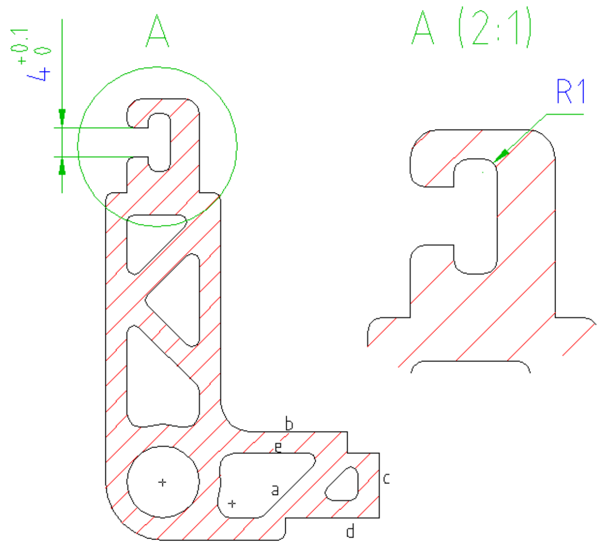
8 In the Power Dimensioning dialog box, click the tolerances button 
to deactivate the tolerances.



Click OK.

9 Press ENTER twice to finish the command.

Now, your lever looks like this:



The Power Dimensioning command recognizes the different scale area. If you dimensioned the radius in the original drawing, the dimension value would be the same. The text height is also the same, as per the selected drafting standard.

This is the end of this tutorial chapter.

Save your file.

Working with Model Space and Layouts

6

In this tutorial, you work with layouts in AutoCAD® Mechanical, to create scale areas, viewports, and detail views in model space. You learn how to freeze objects in viewports without affecting the model and other layouts.

Key Terms

Term	Definition
base layer	A layer made up of working layers and standard parts layers. Base layers are repeated in every layer group.
detail	Enlargement of a portion of the design drawing that cannot be clearly displayed or dimensioned. The overall representation (surface texture symbols, etc.) can be enlarged.
drawing	A layout of drawing views in model space or layout.
layer group	A group of associated or related items in a drawing. A major advantage of working with layer groups is that you can deactivate a specific layer group and a complete component. The drawing and its overview are enhanced by reduction in regeneration time.
layout	The tabbed environment in which you create and design floating viewports to be plotted. Multiple layouts can be created for each drawing.

Term	Definition
Power Dimensioning	A command useful for generating linear, radial, and diameter dimensions, which minimizes the number of the individual actions while generating a dimension. Power Dimensioning automatically selects the type of the linear dimension (horizontal, vertical, aligned), based on the selected point.
scale area	Defines the scale for an area of the drawing.
scale monitor	A function to view and control the scale for any scale area.
viewport	A scaled view of the model defined in a layout.
view scale	The scale of a base drawing relative to the model scale. Also, the scale of dependent views relative to the base view.
working layer	The layer where you are currently working.

Working with Model Space and Layouts

Using model space and layouts, you can create different views with different scales from the same model. The main advantage of working with layouts is that views are associative. If you make changes in one viewport, those changes are made in all other viewports as well, since each viewport is another view of the same model. You can also freeze objects in a new viewport without affecting objects in other views.

Getting Started

In this tutorial, you work with viewports. You generate an associative detail and create a subassembly drawing.

To open a file

- Open the file *tut_engine.dwg* in the tutorials folder.

  Ribbon  ➤ Open ➤ Drawing

  Menu File ➤ Open

Command	OPEN

NOTE The path to the folder containing tutorial files is;

- **Windows Vista:®** *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains parts of a four-stroke engine.

Save your file under a different name or to a different directory to preserve the original tutorial file.

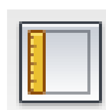
Creating Scale Areas

To generate correct views with correct zoom factors in a layout, you must define a scale area in model space.

Create the scale area.

To create a scale area

1 Start the Viewport/Scale Area command.

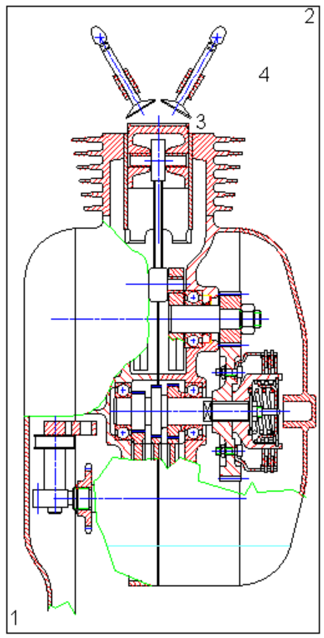
Ribbon	View tab ➤ Viewports panel ➤ Viewport/Scale Area. 
Menu	View ➤ Viewports ➤ Viewport/Scale Area Annotate ➤ Drawing Title and Revision ➤ Viewport/Scale Area
Command	AMSCAREA

2 Respond to the prompts as follows:

`Define the border....`

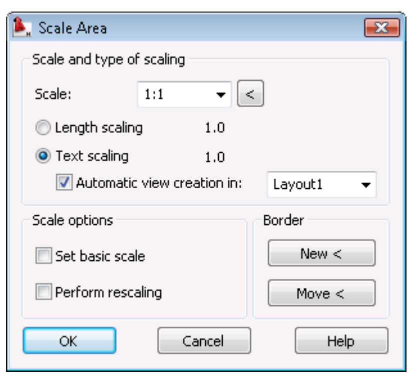
`Specify first point or [Circle/Object]:` *Specify the first corner point (1)*

`Specify second point:` *Specify the second corner point (2)*



3 In the Scale Area dialog box, specify:

`Scale: 1:1`



Click OK.

Since you now have a defined scale area, you can automatically create a viewport.

To create a viewport automatically

1 Start the Viewport Auto Create command.

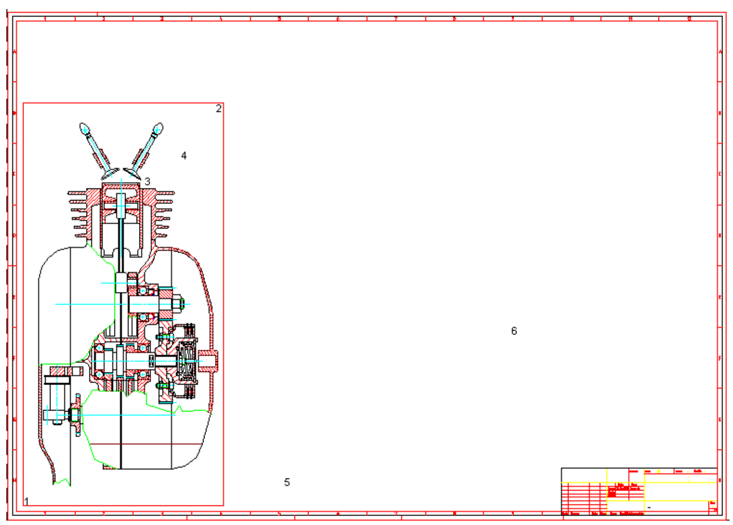
Ribbon	None.
Menu	View ➤ Viewports ➤ Viewport Auto Create
Command	AMVPORTAUTO

2 Respond to the prompts as follows:

`Enter layout name (<Return> for "Layout1"):` *Press* ENTER

`Select target position (<Return> for current position):`

Place the viewport on the left, inside the drawing border



Save your file.

Creating Detail Views

There are two types of detail views; associative and non associative. In this exercise, you create an associative detail, because you use a viewport.

Create an associative detail of the valve.

To create a detail

1 Start the Detail command.

Ribbon	Home tab ➤ Draw Tools panel ➤ Detail View. 
Menu	Draw ➤ Detail...
Command	AMDETAIL

The viewport is activated automatically.

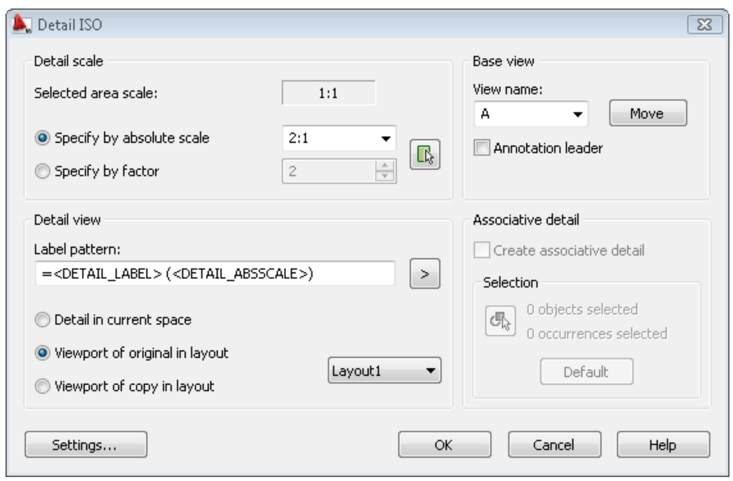
2 Respond to the prompts as follows:

`Define the enlargement area for the detail ...`

`Center of circle or [Rectangle/Object]:` *Select the center of the detail (3)*

`Specify radius or [Diameter]:` *Drag the radius to the desired size (4)*

3 In the Detail dialog box, specify the settings shown in the illustration.

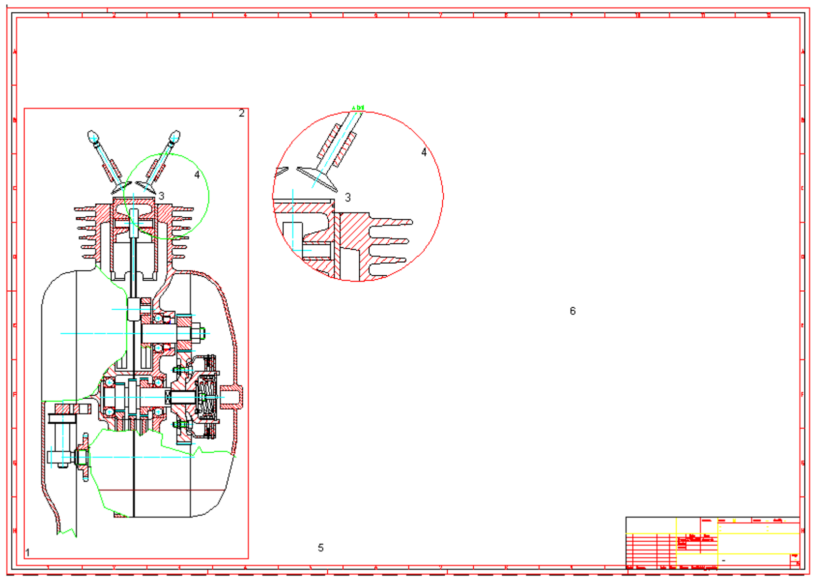


Click OK.

4 Respond to the prompt as follows:

`Select target position (<Return> for current position):`

Place the detail to the right of the current viewport



Save your file.

Generating New Viewports

Now, you create a viewport inside a layout.

To create a viewport in the layout

1 Start the Viewport/Scale Area command.

Ribbon	None.
Menu	None.
Command	AMVPORT

2 Respond to the prompts as follows:

`Specify first point or [Circle/Border/Object]:`

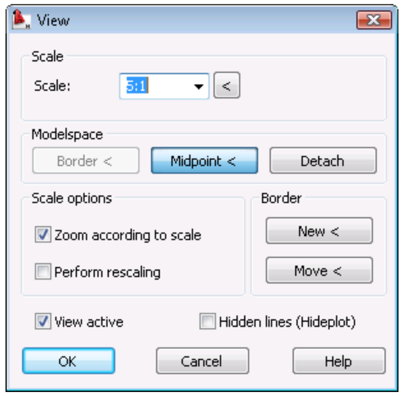
Select point 5 in the drawing

`Specify second point:` *Select point 6 in the drawing*

3 In the View dialog box, specify:

`Scale: 5:1`

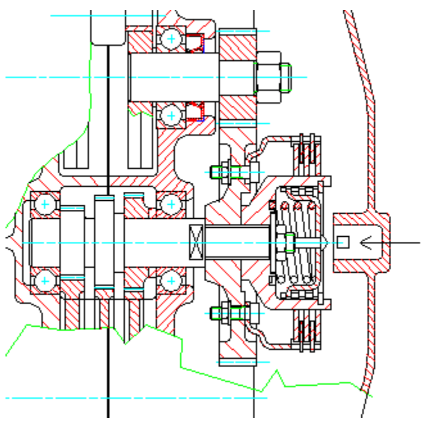
Choose Midpoint.



The drawing is changed to model space so that you can define the midpoint.

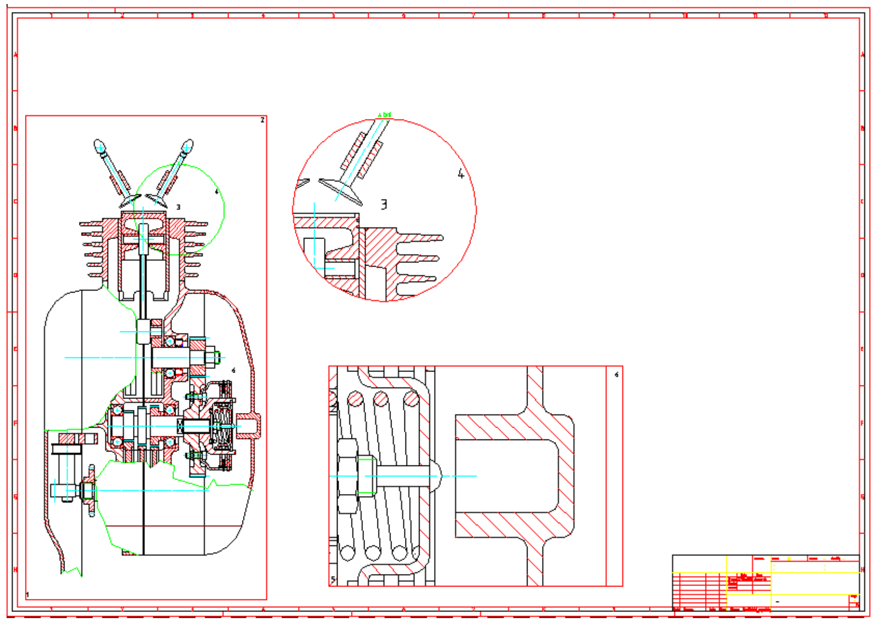
4 Respond to the prompt:

`Select view center:` *Select the endpoint of the centerline*



5 In the View dialog box, Click OK.

Your drawing looks like this:



Save your file.

Inserting Holes Within Viewports

To demonstrate the main advantage of working with layouts, insert a hole in the housing. When you make this change, it is immediately displayed in every view.

Insert a user through hole in the previously created viewport.

To insert a through hole

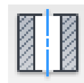
1 Activate the previously created viewport.

Ribbon	None
Menu	None
Command	MSPACE

The viewport has a thick (highlighted) frame.

2 Start the Through Hole command.

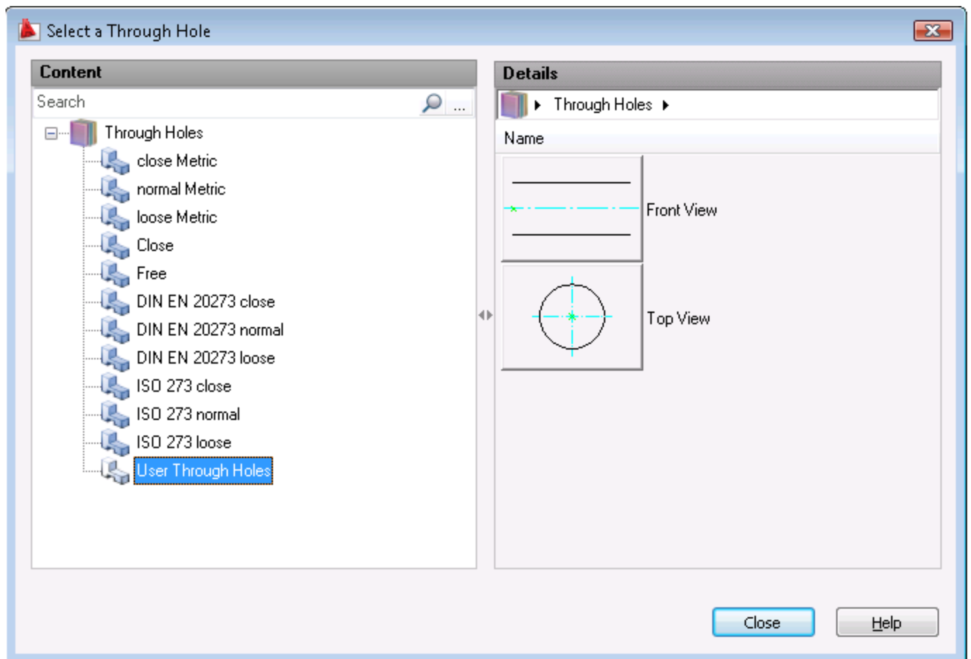
Ribbon Content tab ➤ Holes panel ➤ Through Hole.



Menu Content ➤ Holes ➤ Through Holes...

Command AMTHOLE2D

3 In the Select a Through Hole dialog box, scroll to and select User Through Holes, and then click Front View.



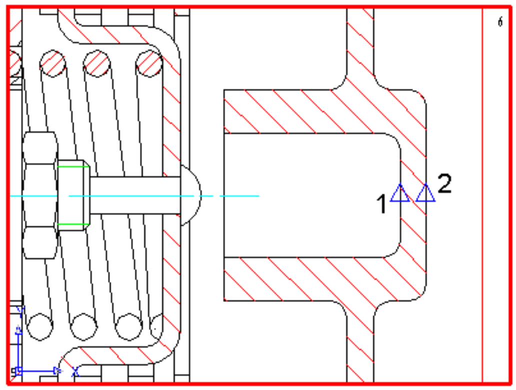
4 Respond to the prompts as follows:

`Specify insertion point:`

Hold down the SHIFT *key and right-click, and then choose Midpoint*

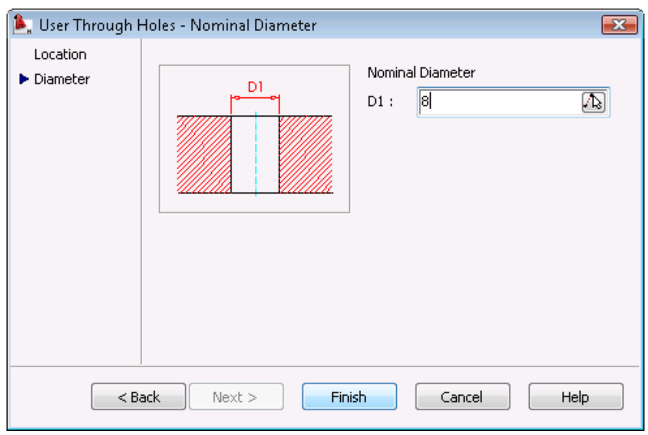
5 `Specify insertion point: _mid of` *Select the midpoint of the housing (1)*

`Specify hole length:` *Select the endpoint of the hole (2)*



6 In the User Through Holes - Nominal Diameter dialog box, specify:

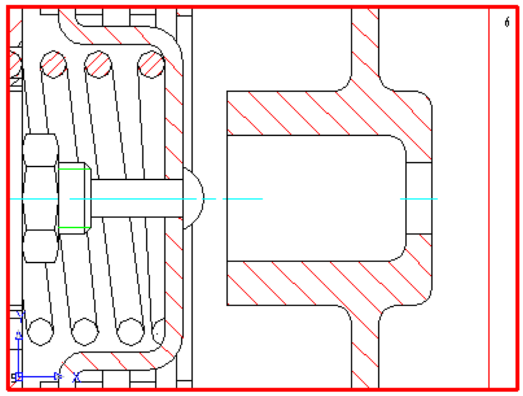
`Nominal Diameter:` 8



Choose Finish.

The user through hole is inserted into your drawing.

The drawing looks like this:



Because of the associativity, the through hole created in the viewport also appears in the original view.

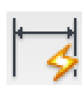
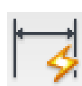
In the next step, you dimension the through hole diameter in the viewport. Since the dimension is to appear only in the detail view, you generate the dimension directly in the layout without having a viewport active.

To apply a dimension in the layout

1 Change to the layout.

Ribbon	None ➤ None ➤
Menu	Does not exist in the Menu.
Command	PSPACE

2 Start the Power Dimensioning command.

Ribbon	Home tab ➤ Dimension panel ➤ Power Dimension.  Annotate tab ➤ Dimension panel ➤ Power Dimension. 
Menu	Annotate ➤ Power Dimensioning
Command	AMPOWERDIM

3 Respond to the prompts as follows:

```
Specify first extension line origin or
[Linear/Angular/Radial/Baseline/Chain/Options/Update] <select
object>:
```

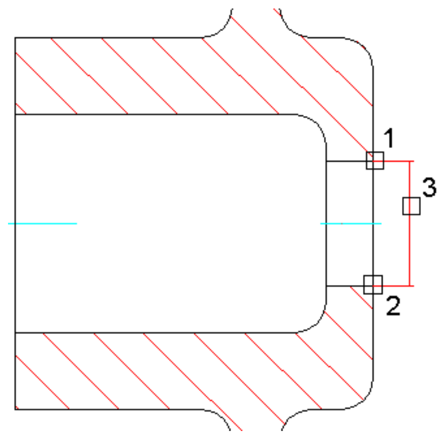
Select the first edge of the hole (1)

```
Specify second extension line origin:
```

Select the second edge of the hole (2)

```
Specify dimension line location or
[Horizontal/Vertical/Aligned/Rotated/Placement options]:
```

Drag the dimension line toward point 3 until it turns red, and then click



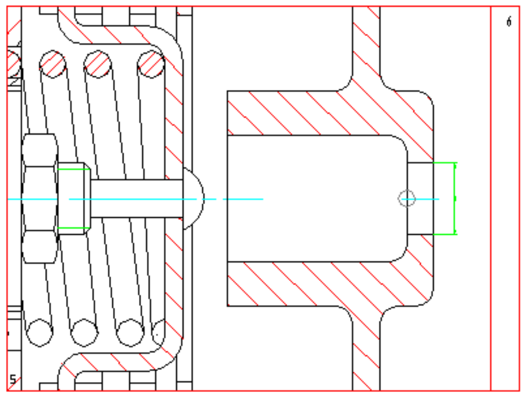
4 In the Power Dimensioning dialog box, Click OK.

5 Continue to respond to the prompts as follows:

`Specify first extension line origin or`
`[Linear/Angular/Radial/Baseline/Chain/Options/Update] <select`
`object>:` *Press* ENTER

`Select arc, line, circle or dimension:` *Press* ENTER

The viewport looks like this:



NOTE You can also dimension the hole in model space and turn off the layer of one specific viewport. In that case, the dimension text is correct only in the 1:1 viewport, and not in the detail view. Therefore, it is best that you dimension directly on the layout.

Save your file.

Creating Subassemblies in New Layouts

If you use layer groups in your assembly drawing, you can create detail and subassembly drawings in layouts. You can switch off selected layer groups in a viewport so that only the detail or subassembly is visible.

Before you create a subassembly in a new layout, freeze the model and other views. Then when you create a new viewport in Layout 2, only the specified subassembly is displayed, and objects are not hidden in the model and other views.

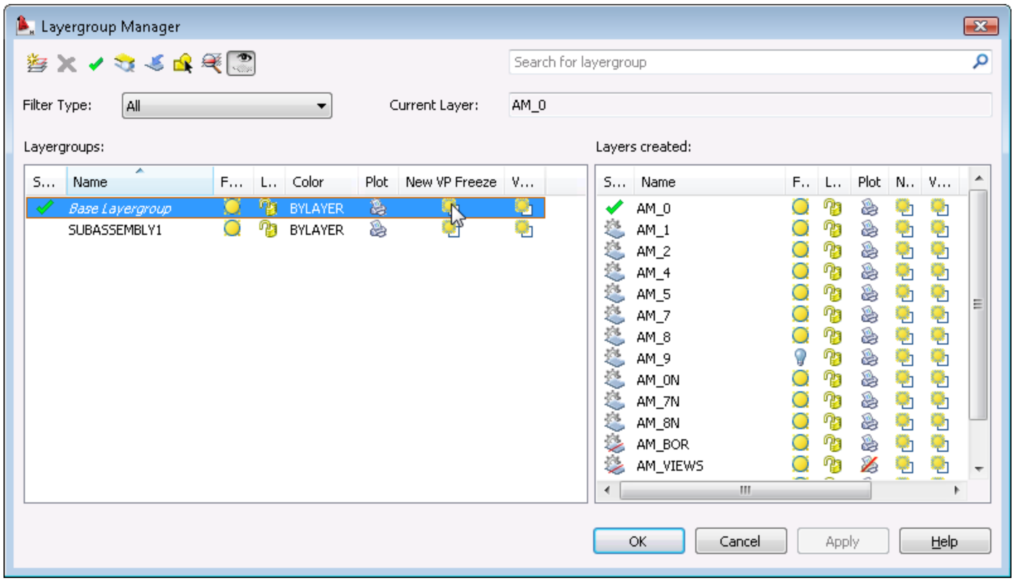
To freeze the model and other layouts

1. Select the Layout 2 tab on the bottom of your drawing area. Layout 2 is displayed.

2. Start the Layer Group Control.

Ribbon	None.
Menu	Tools ➤ Layergroup ➤ Layergroup Manager
Command	AMLAYER

3 In the Layergroup Manager, Layergroups list, click the icon in the Base Layer Group row, New VP Freeze column to freeze it.



Click OK.

Create an associative view of a subassembly in layout 2.

To create an associative view of a subassembly

1 Select the Layout 2 tab on the bottom of your drawing area. Layout 2 is displayed.

2 Start the Viewport/Scale Area command.

Ribbon	None.
Menu	None.
Command	AMVPORT

3 Respond to the prompts as follows:

`Specify first point or [Circle/Border/Object]:`

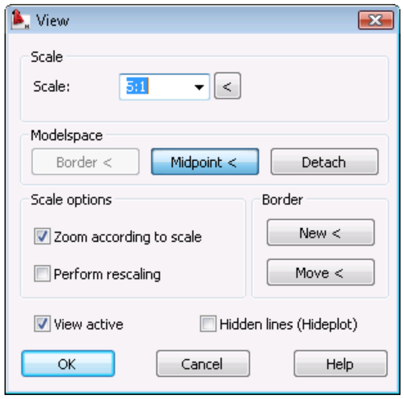
Select point 7 in the drawing

`Specify second point:` *Select point 8 in the drawing*

4 In the View dialog box, specify:

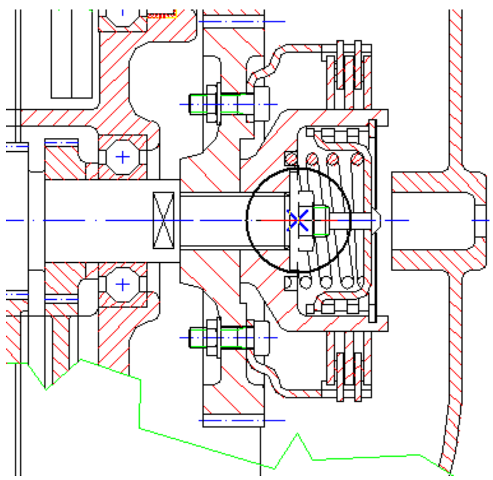
`Scale: 5:1`

Choose Midpoint.



The drawing is changed to model space.

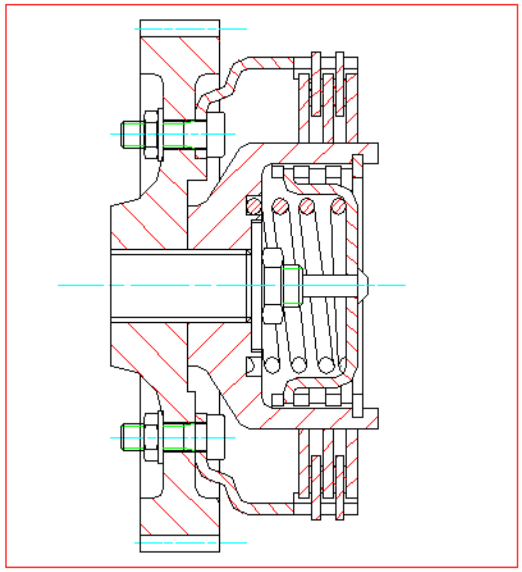
5 Specify the point, as shown in the following drawing:



6 In the View dialog box, Click OK.

In the new viewport, only the subassembly you specified is visible. AutoCAD Mechanical freezes the Base Layer Group.

Your drawing looks like this:



Finish your detail drawing with text, remarks, annotations, and so on.

NOTE When you plot the drawing, the red viewport frame is turned off automatically. If you have a plotter or printer driver installed, use the plot command, and preview the drawing.

This is the end of this tutorial chapter.

Save your file.

Dimensioning

7

In this tutorial, you learn how to add dimensions to your drawing with the automatic dimensioning in AutoCAD® Mechanical, change the dimensions with Power Commands, and insert a drawing border.

Key Terms

Term	Definition
baseline dimension	A dimension that is aligned to extension lines and read from the bottom or right side of the drawing.
centerline	Line in the center of a symmetrical object.
drawing border	A standardized frame that is used for technical drawings.
fit	Range of tightness or looseness in mating parts (for example shafts or holes). Tolerances in these dimensions are expressed in standard form.
fit name	Name of the selected fit (for example, H7).
multi edit	An option where you determine a selection set of dimensions and edit them together.
Power Dimensioning	Power Dimensioning is a tool for generating linear, radial, angular, and diameter dimensions, which minimizes the number of the individual actions required while generating a dimension. Power Dimensioning selects the type of linear dimension (horizontal, vertical, or aligned), based on the selected point, and

Term	Definition
	the dimensions of the drawing can have a uniform style using the distance snap.
Power Erase	Command for deleting. Use Power Erase when you delete part reference numbers or dimensions that were created with Power Dimensioning and Automatic Dimensioning.
title block	A title block contains a series of attributes. Some already have values. The pre-assigned values can be modified, and the vacant attributes can be completed with new values.
tolerance	The total amount by which a given dimension (nominal size) may vary (for example, 20 0.1).

Adding Dimensions to Drawings

AutoCAD Mechanical offers various dimensioning tools. Use automatic dimensioning to add dimensions to a bushing, and then change these dimensions.

To open a file

- Open the file *tut_bushing.dwg* in the tutorials folder.

Ribbon  ➤ Open ➤ Drawing

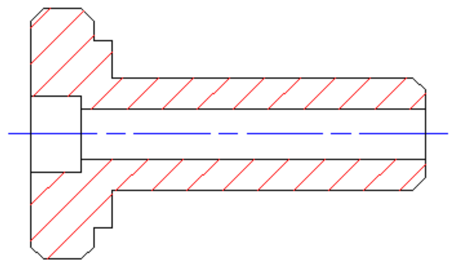
Menu File ➤ Open...

Command OPEN

NOTE The path to the folder containing tutorial files is;

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The file contains a drawing of a bushing.



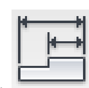
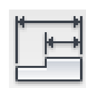
Save your file under a different name or to a different directory to preserve the original tutorial file.

Adding Multiple Dimensions Simultaneously

Dimension the bushing using automatic dimensioning.

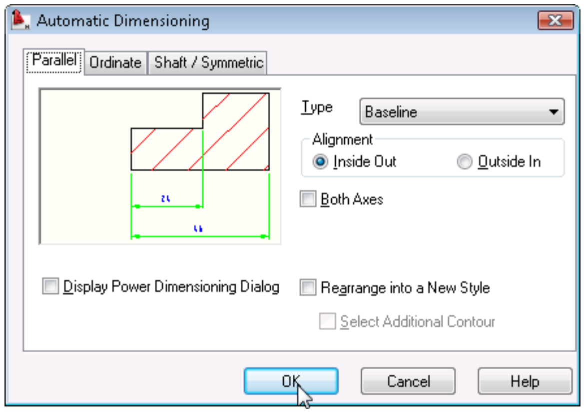
To dimension a contour

1 Start the Automatic Dimensioning command.

Ribbon	Home tab ➤ Dimension panel ➤ Power Dimension drop-down ➤ Multiple Dimension.  Annotate tab ➤ Dimension panel ➤ Multiple Dimension. 
Menu	Annotate ➤ Multiple Dimensioning
Command	AMAUTODIM

2 In the Automatic Dimensioning dialog box, Parallel tab, specify:

`Type: Baseline`



Click OK.

3 Respond to the prompts as follows:

`Select objects [Block]:`

Select the complete bushing by creating a window around it

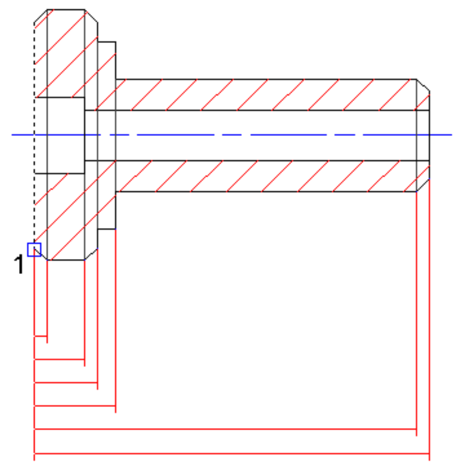
`Select objects [Block]:` *Press* ENTER

`First extension line origin:`

Select the lower left most corner of the bushing (1)

```
Specify dimension line location or
[Horizontal/Vertical/Rotated/Placement options]:
```

Drag the dimensioning downwards until it snaps in (highlighted red), and then click

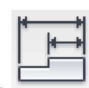
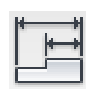


Starting point for next extension line: *Press* ENTER *to end the command*

Generate the diameter dimensions using shaft dimensioning.

To dimension a shaft

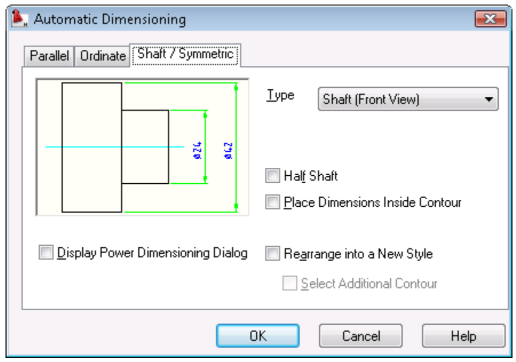
1 Start the Automatic Dimensioning command.

Ribbon	Home tab ➤ Dimension panel ➤ Power Dimension drop-down ➤ Multiple Dimension.  Annotate tab ➤ Dimension panel ➤ Multiple Dimension. 
Menu	Annotate ➤ Multiple Dimensioning
Command	AMAUTODIM

2 In the Automatic Dimensioning dialog box, Shaft/Symmetric tab, specify:

`Type: Shaft (Front View)`

Click OK.



3 Respond to the prompts as follows:

`Select objects [Block]:`

Select the complete bushing by creating a window around it

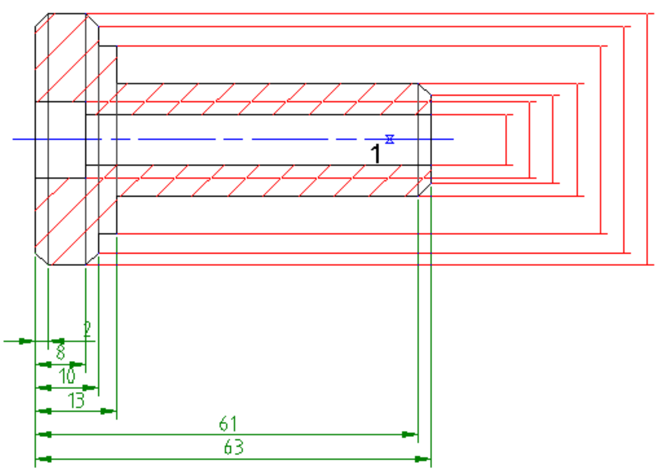
`Select objects [Block]:` *Press* ENTER

`Select Centerline or new starting point:`

Select the centerline of the bushing (1)

`Specify dimension line location or [Placement options]:`

Drag the dimensioning to the right until it snaps in (highlighted red), and then click

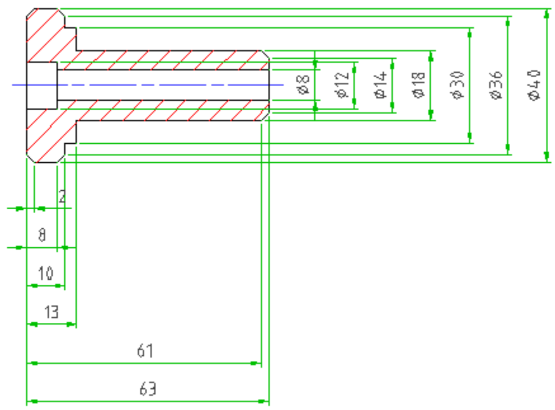


4 Continue responding to the prompt:

`Starting point for next extension line:`

Press ENTER *to end the command*

Your drawing looks like this.



Save your file.

Editing Dimensions with Power Commands

Some dimensions in the drawing are not necessary. In the next step, you delete the dimensions that you don't need.

To delete dimensions

1 Start the Power Erase command.

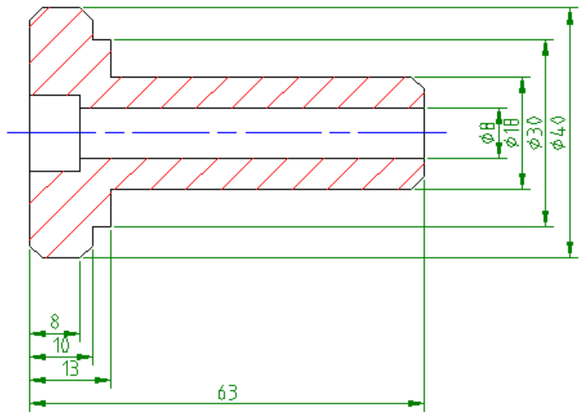
Ribbon	Home tab ➤ Modify panel ➤ Erase. 
Menu	Modify ➤ Power Erase
Command	AMPOWERERASE

2 Respond to the prompt as follows:

```
Select objects:
```

Select baseline dimensions 2 and 61, and diameter dimensions 12, 14, and 36, press ENTER

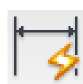
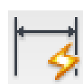
The dimensions are deleted, and the remaining dimensions are rearranged. Your drawing looks like this:



Add a single dimension with a fit using Power Dimensioning.

To add a dimension with a fit

1 Start the Power Dimensioning command.

Ribbon	Home tab ➤ Dimension panel ➤ Power Dimension. 
	Annotate tab ➤ Dimension panel ➤ Power Dimension. 
Menu	Annotate ➤ Power Dimensioning
Command	AMPOWERDIM

2 Respond to the prompts as follows:

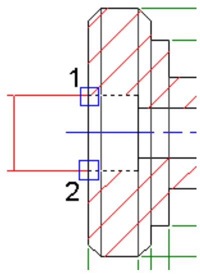
`Specify first extension line origin or [Linear/Angular/Radial/Baseline/Chain/Options/Update] <select object>:`

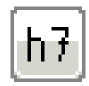
Select the first point (1)

`Specify second extension line origin:` *Select second point (2)*

`Specify dimension line location or [Horizontal/Vertical/Aligned/Rotated/Placement options]:`

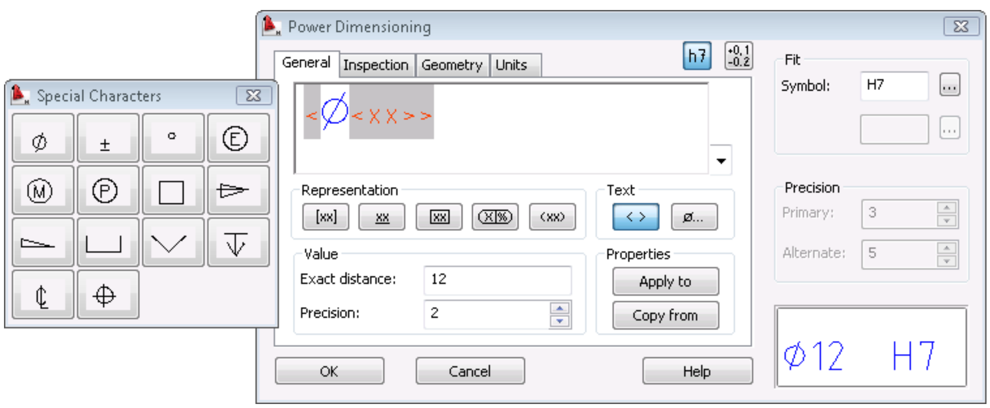
Drag the dimensioning to the left until it is highlighted red, and then click



3 In the Power Dimensioning dialog box, click  , and then specify:

`Fit: Symbol:` *H7*

4 Under Text, click  , and then select the diameter symbol (upper left).



Click OK.

Apply angular dimensioning.

To apply an angular dimension

1 Respond to the prompts as follows:

```
Specify first extension line origin or
[Linear/Angular/Radial/Baseline/Chain/Options/Update] <select
object>:
```

Enter A, press ENTER

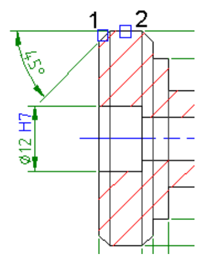
(Single) Select arc, circle, line or [Baseline/Chain/eXit]
<specify vertex>:

Select the line (1)

Select second line: *Select the second line (2)*

Specify dimension arc line location:

Drag the dimension to a suitable position, and then click



2 Press ENTER twice to finish the command.

Add a fit to the shaft dimensions using Multi Edit.

To add a fit using Multi Edit

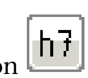
1 Start the Multi Edit command.

Ribbon	Annotate tab ➤ Dimension panel drop-down ➤ Multi Edit. 
Menu	Annotate ➤ Edit Dimensions ➤ Multi Edit
Command	AMDIMMEDIT

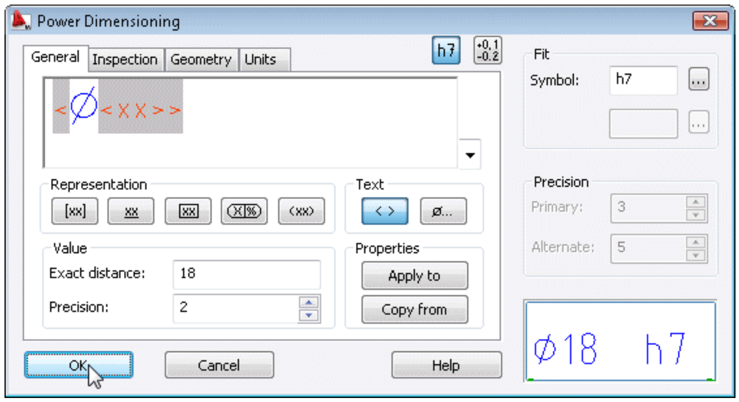
2 Respond to the prompts as follows:

Select dimensions: *Select the dimensions 18 and 30*

Select dimensions: *Press* ENTER

3 In the Power Dimensioning dialog box, choose the Add Fit button  , and then specify:

Fit: Symbol: *h7*



Click OK.

The fit description h7 is added to the dimensions.

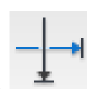
Save your file.

Breaking Dimension Lines

The automatic dimensioning process created intersecting dimension lines. The drawing appearance can be improved by breaking these lines.

To break dimension lines

1 Start the Break Dimension command.

Ribbon	Annotate tab ➤ Dimension panel drop-down ➤ Break Dimension. 
Menu	Annotate ➤ Edit Dimensions ➤ Break Dimension
Command	AMDIMBREAK

2 Respond to the prompt as follows:

```
Select dimension or extension line to break <Multiple>:
```

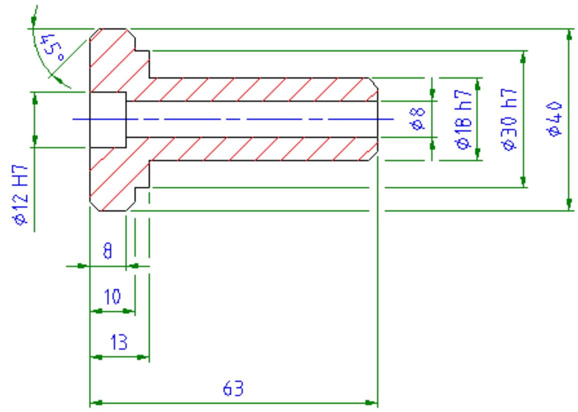
Press ENTER

```
Select dimensions:
```

Select baseline dimension 10 and 13, and diameter dimensions 18, 30, and 40, press ENTER

Select Objects [Restore] <Automatic>: *Press* ENTER

The selected dimensions are broken automatically and your drawing looks like this:



Save your file.

Inserting Drawing Borders

Insert a drawing border.

To insert a drawing border

1 Start the Drawing Title/Borders command.

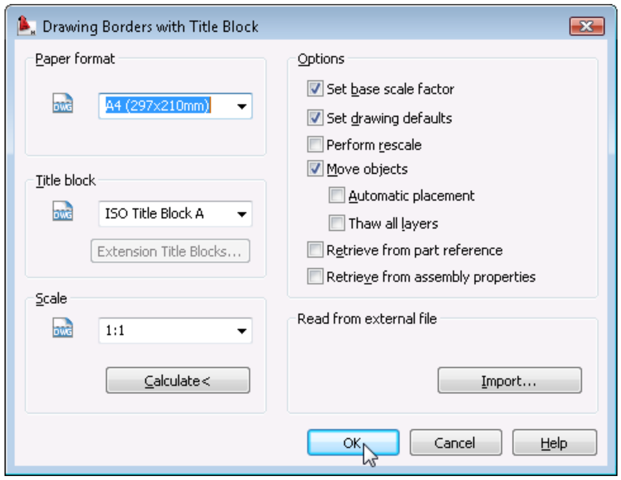
Ribbon	Annotate tab ➤ Layout panel ➤ Title Border. 
Menu	Annotate ➤ Drawing Title and Revision ➤ Drawing Title/Borders...
Command	AMTITLE

2 In the Drawing Borders with Title Block dialog box, specify:

Paper Format: A4 (297x210mm)

Title Block: ISO Title Block A

Scale: 1:1



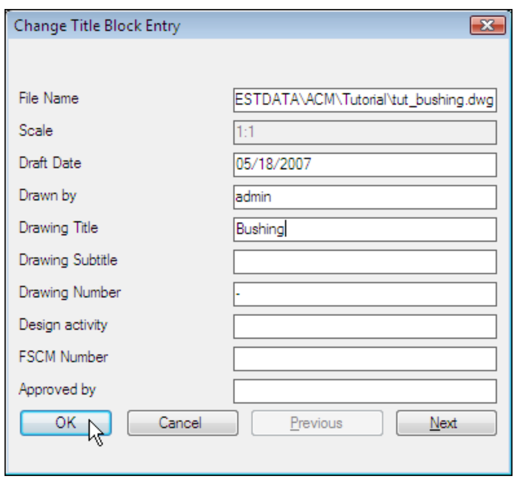
Click OK.

3 Respond to the prompt as follows:

`Specify insertion point:` *Enter -150,0, press* ENTER

4 In the Change Title Block Entry dialog box, specify:

`Drawing Title:` *Bushing*



Click OK.

5 Respond to the prompts as follows:

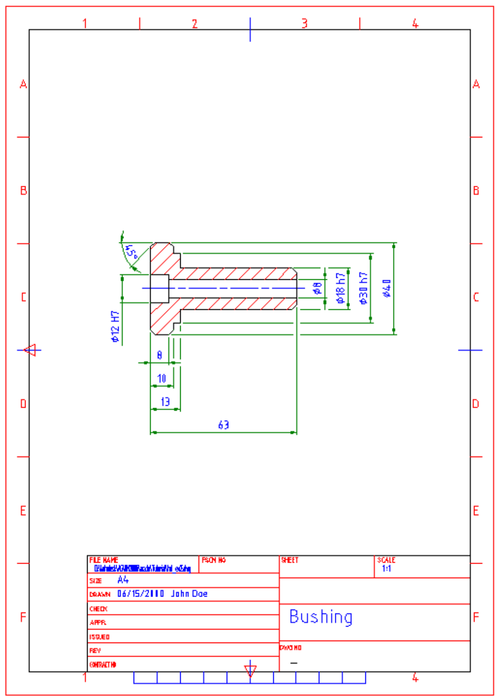
Select Objects: *Select the complete bushing including dimensions*

Select Objects: *Press* ENTER

New location for objects: Click View ➤ Zoom ➤ Extents.

Place the bushing in the middle of the drawing border

Your drawing looks like this:



Save your file.

Inserting Fits Lists

Insert a fits list. Fits lists describe all fits existing in a drawing.

To insert a fits list

1 Start the Fits List command.

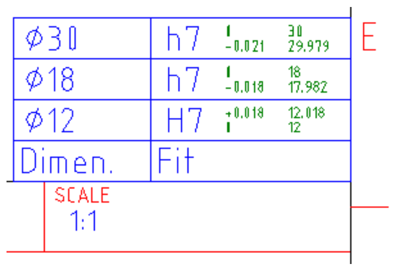
Ribbon	Annotate tab ➤ Table panel ➤ Fits List. 
Menu	Annotate ➤ Fits List
Command	AMFITSLIST

2 Respond to the prompts as follows:

`Fits lists [Update all/Order/New] <New>:` *Press* ENTER

`Specify insertion point:` *Specify the upper right corner of the title block*

The fits list is inserted above the title block, and looks like this.



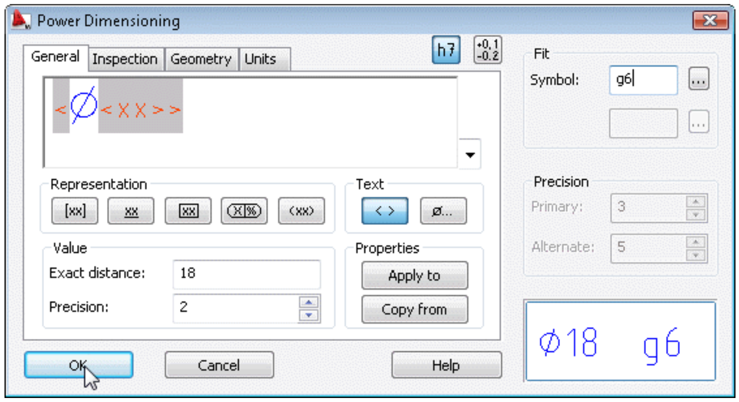
Edit a dimension with a fit. The fits list is updated.

To edit a dimension

1 In the drawing, double-click the diameter dimension (not the dimension line) 18 h7.

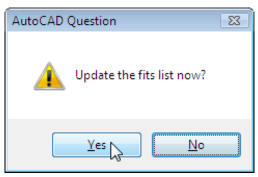
2 In the Power Dimensioning dialog box, specify:

`Fit symbol:` *g6*



Click OK.

3 In the AutoCAD Question dialog box, choose Yes.



The fits list is updated, too. Save your file.

This is the end of this tutorial chapter.

Working with 2D Hide and 2D Steel Shapes

8

In this tutorial, you learn how to work with 2D steel shapes. The features in AutoCAD® Mechanical for defining 2D hide situations have already been covered in Basics of AMSHIDE on page 95. However, the tutorial is intended for new users who may have to work with AM2DHIDE supported legacy drawings.

Key Terms

Term	Definition
background	A contour that is covered by another contour or by objects that are lying behind another contour, in the 3D sense. A background may be a foreground for an additional contour.
foreground	Objects which are lying in front of another contour, in the 3D sense. A foreground may also be a background for an additional contour.
hidden line	Line that is not visible in a specified view. For example, in a front view, lines behind the front plane are not visible.
steel shapes	Steel shapes are standardized steel geometries and profiles that are used for steel and plant construction.

1 Open the file *Tut_Gripper_Plate.dwg* in the tutorials folder.

Ribbon	None.
Menu	File ➤ Open...
Command	OPEN

NOTE The path to the tutorials folder is;

- **Windows Vista®**:*C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows® XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains two views of a gripper plate and contains two construction lines.

2 To keep the original file intact, save the file as *Gripper.dwg*

3 Display the Structure Catalog.

Ribbon	None.
Menu	Tools ➤ Palettes ➤ Structure Catalog
	Structure ➤ Structure Catalog...
Command	AMSCATALOG

4 In the External Drawings tab, navigate to the tutorials folder and select *Tut_Gripper.dwg*.

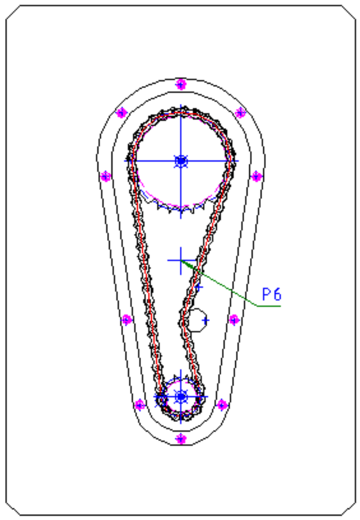
NOTE In the Structure Catalog, navigate to the following folders:

- **Windows Vista:** *C:\Users\Public\Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows XP:** *C:\Documents and Settings\All Users\Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The Mechanical Structure panel shows the mechanical structure components in the drawing and the Preview panel shows a preview of the drawing.

5 **In** the structure panel, double click GRIPPER to reveal the components list.

6 Click and drag GRIPPER ➤ Front to model space.



Defining 2D Hide Situations

Define a 2D hide situation. You can define foreground and background contours and the settings for the representation of the hidden objects.

To define a 2D hide situation

1 Start the Hide Invisible Edges command.

Ribbon	None.
Menu	Modify ➤ 2D Hide ➤ Hide Invisible Edges
Command	AM2DHIDE

2 Respond to the prompts as follows:

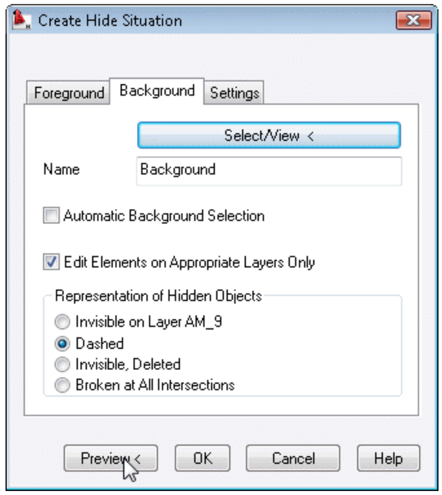
`Select objects for foreground:` *Select the chain*

`Select objects for foreground:` *Press* ENTER

3 In the Create Hide Situation dialog box, Background tab, specify:

`Representation of Hidden Objects:` Dashed

Choose Preview.



NOTE As you can see, the parts of the sprockets that should be visible appear as hidden lines. This shows that the complete area inside the outer chain contour is defined as foreground.

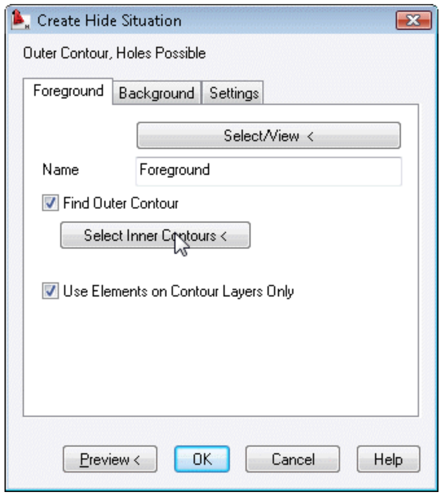
Define the 2D hide situation in a way that the chain has an inner contour.

4 Respond to the prompt as follows:

```
Accept preview and exit command [Yes/No] <Yes>:
```

Enter N, press ENTER

5 In the Create Hide Situation dialog box, Foreground tab, click Select Inner Contours.



6 Respond to the prompt as follows:

`Select point inside a hole or select a loop to remove:`

Select a point inside the chain (1)

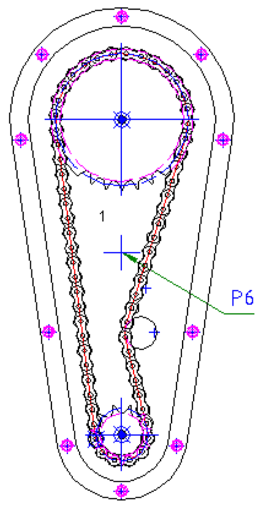
The inner contour of the chain is displayed green.

7 Respond to the prompt as follows:

`Select point inside a hole or select a loop to remove:` *Press* ESC

8 In the Create Hide Situation dialog, choose Preview.

The sprocket is no longer displayed as a hidden line and the chain drive is displayed correctly.



9 Respond to the prompt as follows:

`Accept preview and exit command [Yes/No] <Yes>:` *Press* ENTER

The 2D hide situation is defined correctly, and you can proceed with your drawing.

Save your file under a different name or to a different directory to preserve the original tutorial file.

Inserting 2D Steel Shapes

Steel Shapes can easily be inserted through a selection dialog box, where you can define the standard, profile, size, and length of the steel shape.

Insert a steel shape with a square hollow section on the left edge of the I-shaped girder.

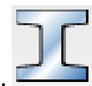
To insert a 2D steel shape

1 Start the Zoom All command.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 
Menu	View ➤ Zoom ➤ Window

Command ZOOM

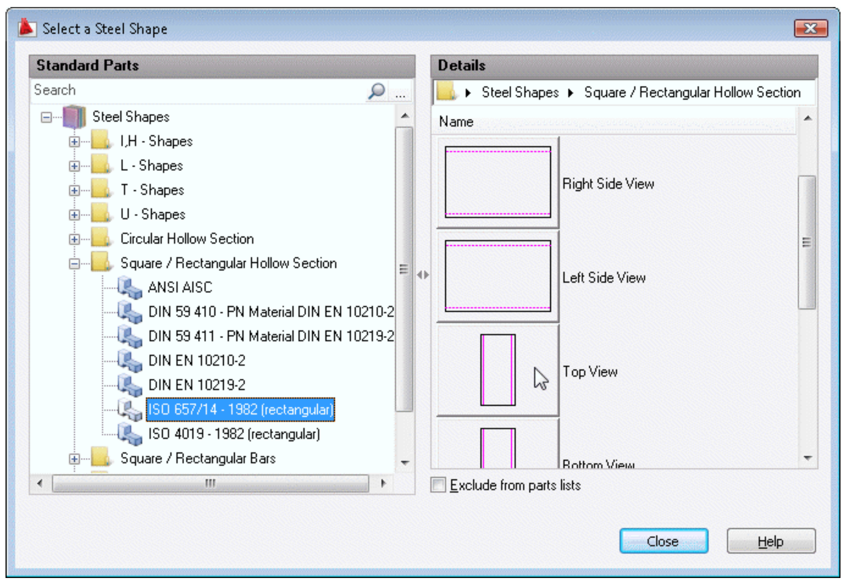
2 Start the Steel Shape command.

Ribbon Content tab ➤ Misc panel ➤ Steel Shapes. 

Menu Content ➤ Steel Shapes...

Command AMSTLSHAP2D

3 In the Select a Steel Shape dialog box, select Steel Shapes ➤ Square/Rectangular Hollow Section, and then select ISO 657/14-1982 (Rectangular) and Top View.



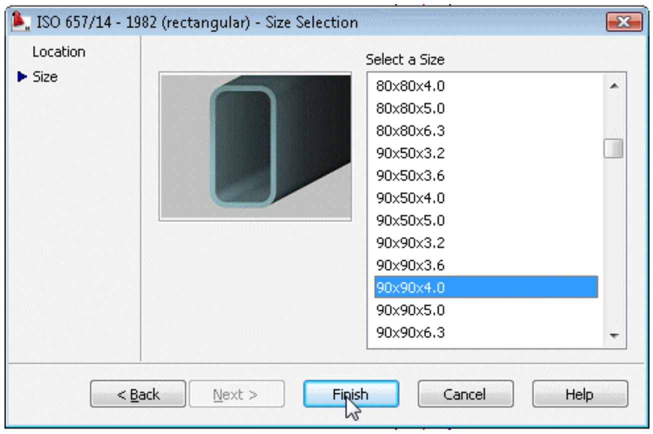
4 Respond to the prompts as follows:

`Specify insertion point:` *Select point P1*

`Specify rotation angle <0>:` *Press* ENTER

5 In the ISO 657/14 - 1982 (Rectangular) - Size Selection dialog box, specify:

`Select a Size:` 90x90x4.0

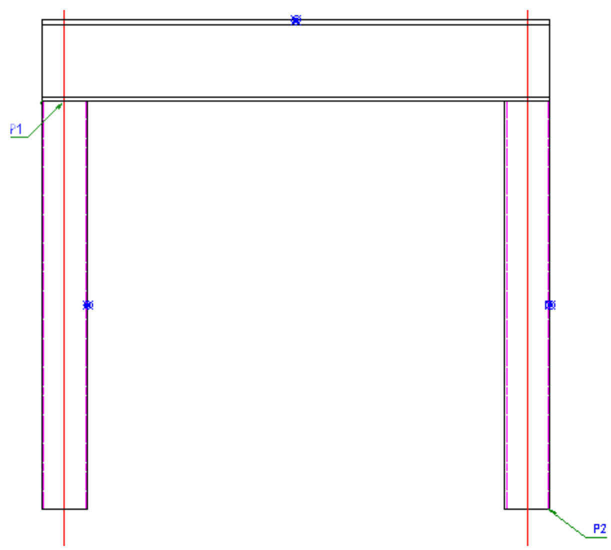


Choose Finish.

6 Respond to the prompt as follows:

`Drag Size:` *Select point P2*

The steel shape is inserted. Your drawing looks like this:



Save the file.

Modify the steel shapes using the Power Commands.

Modifying Steel Shapes Using Power Commands

With the Power Commands, you can create different views of the steel shapes. You can copy, multiply, or edit the steel shapes.

Insert the steel shapes in the top view of the assembly using Power View and Power Copy.

To modify a steel shape using a Power Command

1 Start the Power View command.

Ribbon	None.
Menu	Modify ➤ Power View
Command	AMPOWERVIEW

2 Select the previously inserted steel shape.

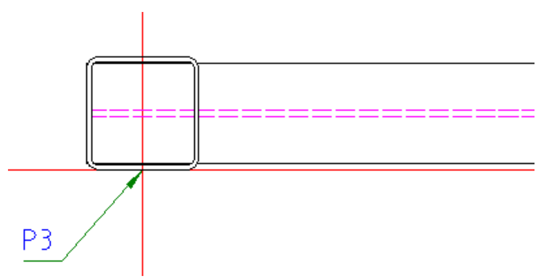
3 In the Select new view dialog box, select the Front View.

4 Respond to the prompts as follows:

`Specify insertion point:` *Select point P3*

`Specify rotation angle <0>:` *0, press* ENTER

The steel shape is inserted in the top view of the assembly. Your drawing looks like this:



Copy the previously inserted view to the other edge of the girder.

5 Start the Power Copy command.

Ribbon

Home tab ➤ Modify panel ➤ Copy. 

Content Edit tab ➤ Modify panel ➤ Power Copy.

Menu	Default Menu ➤ Power Copy
	Edit Menu ➤ Power Copy
	Modify ➤ Power Copy
Command	AMPOWERCOPY

6 Respond to the prompts as follows:

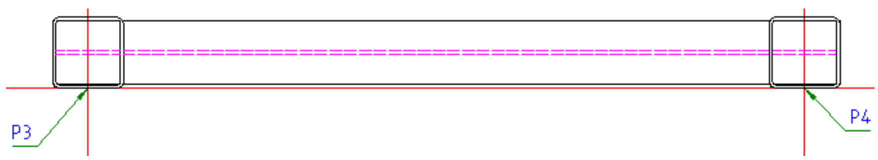
`Select object:` *Select the previously inserted steel shape at point P3*

`Enter an option [Next/Accept]<Accept>:` *Press* ENTER

`Specify insertion point:` *Select point P4*

`Specify rotation angle <0>:` *Press* ENTER

The steel shape is copied. Your drawing looks like this:



Save your file.

Editing 2D Hide Situations

The insertion of the steel shapes in the top view of the assembly created a 2D hide situation automatically. This 2D hide situation is not correct. Use the command AM2DHIDEDIT when mechanical structure is disabled.

Edit the 2D hide situation.

To edit a 2D hide situation

1 Start the Edit Hidden Edges command.

Ribbon	None.
Menu	Modify ➤ 2D Hide ➤ Edit Hidden Edges
Command	AM2DHIDEDIT

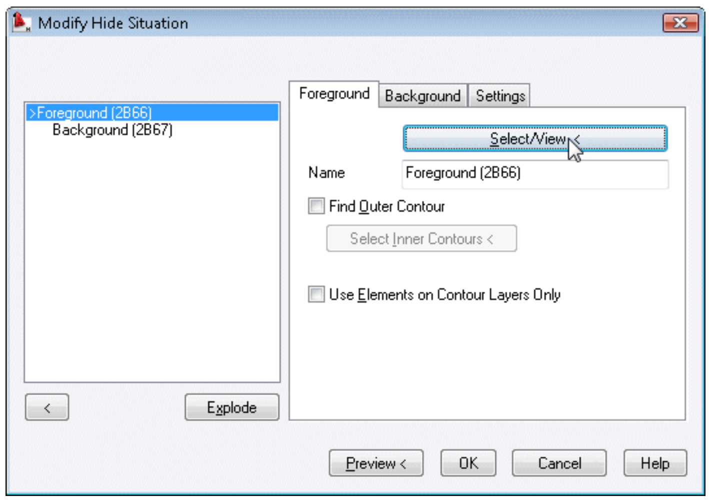
2 Respond to the prompts as follows:

`Edit the behind situation [modifY/Move/Restore/Genius12]`
`<Update>:` *Enter Y, press* ENTER

`Select objects:` *Select the square hollow section on the left*

`Select objects:` *Press* ENTER

3 In the Modify Hide Situation dialog box, Foreground tab, choose Select View.



4 Respond to the prompts as follows:

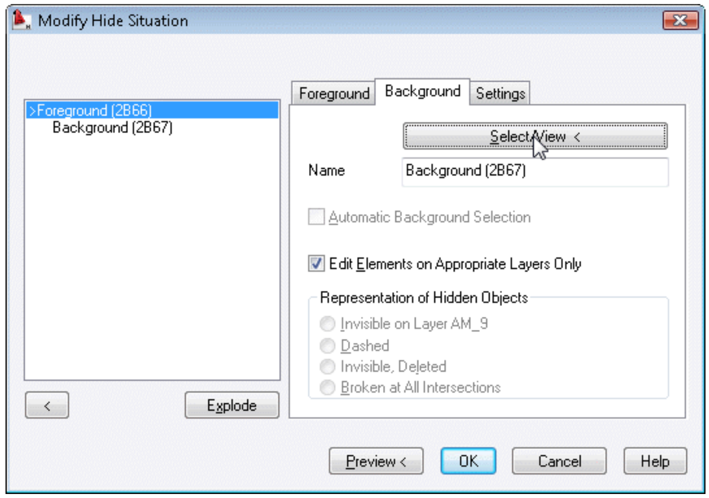
`Select objects for foreground:` *Select the I-shaped girder*

`Select objects for foreground:`

Press SHIFT *while you click the square hollow section on the left to deselect it*

`Select objects for foreground:` *Press* ENTER

5 In the Modify Hide Situation dialog box, Background tab, choose Select View.



6 Respond to the prompts as follows:

`Select objects for background:`

Select the square hollow section on the left

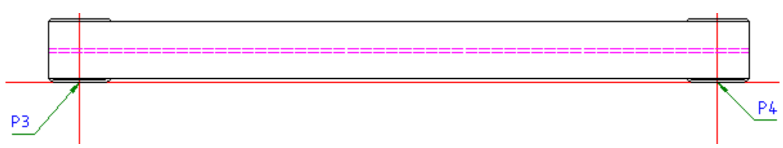
`Select objects for background:`

Select the square hollow section on the right

`Select objects for background:` *Press* ENTER

7 In the Modify Hide Situation dialog box, click Preview.

Your drawing looks like this:



8 Respond to the prompts:

`Accept preview and exit command [Yes/No] <Yes>:` *Press* ENTER

`Edit the behind situation [modifY/Move/Restore/Genius12] <Update>:` *Press* ENTER

`Select objects:` *Press* ENTER

The 2D hide situation is corrected.

Save your file.

Copying and Moving 2D Hide Situations

If you copy or move assemblies that contain 2D hide situations, the 2D hide information is not lost.

Copy the girder assembly.

To copy a 2D hide situation

1. Select the I-shaped girder and the two square hollow sections.
2. Right-click the graphics area background, and then choose Copy with Base Point.

   Respond to the prompt as follows:

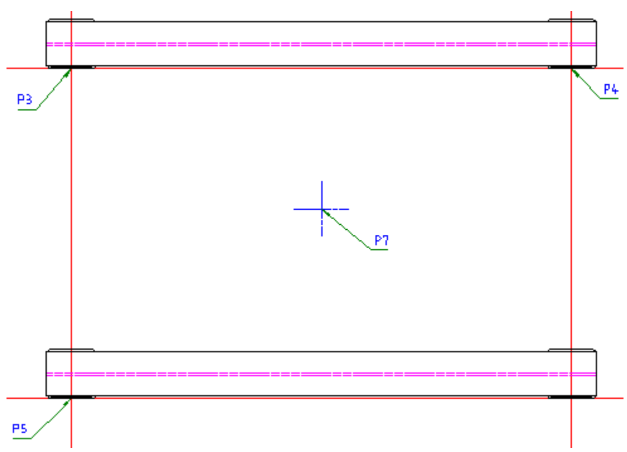
   `Specify base point:` *Select point P3*

3. Right-click, and then choose Paste.

   Respond to the prompt as follows:

   `Specify insertion point:` *Select point P5*

   The girder assembly is copied to the new location. Your drawing looks like this.

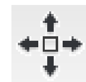
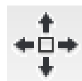


Save your file.

Move the chain drive from the beginning of the chapter to the top view of the assembly.

To move a 2D hide situation

1 Start the Move command.

Ribbon	Home tab ➤ Modify panel ➤ Move. 
	Content Edit tab ➤ Modify panel ➤ Move. 
Menu	Edit Menu ➤ Move
	Modify ➤ Move
Command	MOVE

2 Respond to the prompts as follows:

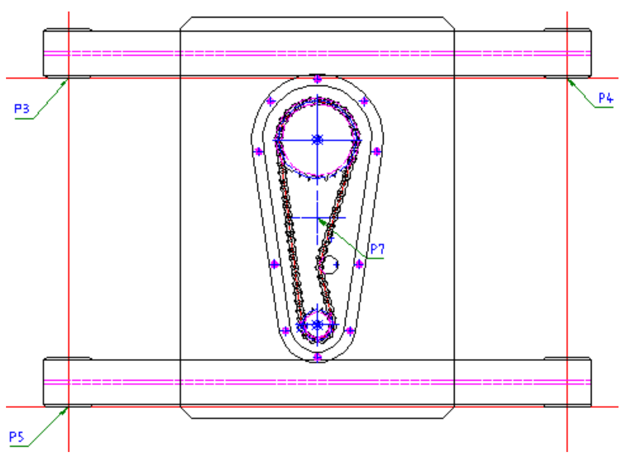
`Select objects:` *Select the complete chain drive using a window*

`Select objects:` *Press* ENTER

`Specify base point or displacement:` *Select point P6*

`Specify second point of displacement or <use first point as displacement>:` *Select point P7*

The complete chain drive is moved to the top view of the assembly. Your drawing looks like this:



Define the 2D hide situation for the girder assembly and the chain drive.

To define a 2D hide situation

1 Start the Hide Invisible Edges command.

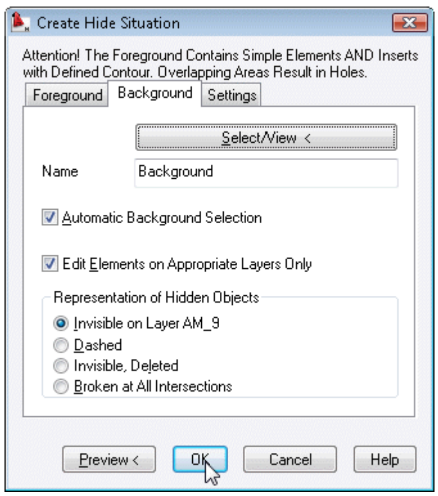
Ribbon	None.
Menu	Modify ➤ 2D Hide ➤ Hide Invisible Edges
Command	AM2DHIDE

2 Respond to the prompts as follows:

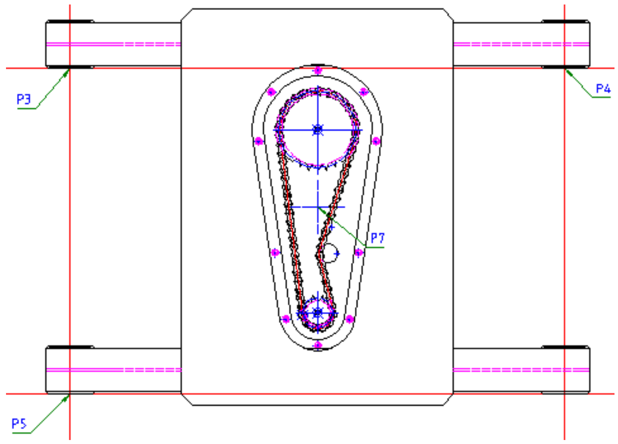
`Select objects for foreground:` *Select the complete chain drive*

`Select objects for foreground:` *Press* ENTER

3 In the Create Hide Situation dialog box, Click OK.



Now, the girder assembly is hidden by the chain drive. Your drawing looks like this:



Save your file. This is the end of this exercise.

Working with Standard Parts

9

In this tutorial, you learn to work with standard parts in AutoCAD® Mechanical. You insert a screw connection, a hole, and a pin. You also edit the standard parts with power commands.

Key Terms

Term	Definition
background	A contour that is covered by another contour or by objects that are lying behind another contour, in the 3D sense. A background may be a foreground for an additional contour.
C-line (construction line)	A line that is infinite in both directions or infinite starting at a point which can be inserted into the drawing area. You use C-lines to transfer important points (for example, center points of holes) into other views or drawing areas.
countersink	A chamfered hole that allows bolt and screw heads to be flush or below the part surface.
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 or length. The values (sizes) are taken from the Standard parts database.
Power Command	Summary term for Power Copy, Power Recall, Power Edit, Power Dimensioning, Power Erase and Power View.

Term	Definition
Power Copy	A command that copies a drawing object to another position in the drawing. Power Copy produces an identical copy of the original object.
Power Edit	An edit command for all objects in your drawing.
Power Erase	A command for intelligent deleting. Use Power Erase when you delete part reference numbers or when you delete dimensions that have been created with Power Dimensioning and Automatic Dimensioning.
Power Recall	A command that lets you click an existing drawing object and places you in the correct command for creating that object.
Power View	A command where you can quickly and easily create a standard part top view or bottom view of a side view and vice versa.
representation	Standard parts representation in a drawing in normal, simplified, or symbolic mode.

Working with Standard Parts

AutoCAD Mechanical provides a large selection of standard parts to work with, including regular and fine threads, many types of holes, fasteners, and other standard parts. You can insert complete screw connections (screws with holes and nuts) in one step. Some intelligence is built into this process. For example, if you select a screw with a metric thread, you get only metric threads when you add any additional parts such as tapped holes or nuts.

NOTE It is required that the ISO standard parts be installed for this tutorial exercise.

Open the initial drawing.

To open a drawing

1 Open the file *tut_std_pts.dwg* in the Tutorial folder.
Ribbon

➤ Open ➤ Drawing

Menu	File ➤ Open...
Command	OPEN

NOTE The path to the folder containing tutorial files is;

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains a motor with a gearbox. Some construction lines are inserted to help you work through the tutorial exercise. The gearbox is not completed yet. We want to add standard components and show how easy it is to edit standard parts with an automatic update of the background objects.

Before you proceed, you must enable mechanical structure. If you proceed without mechanical structure enabled, some command line prompts will differ from the prompts in the exercise.

2 Zoom in to the area of interest.

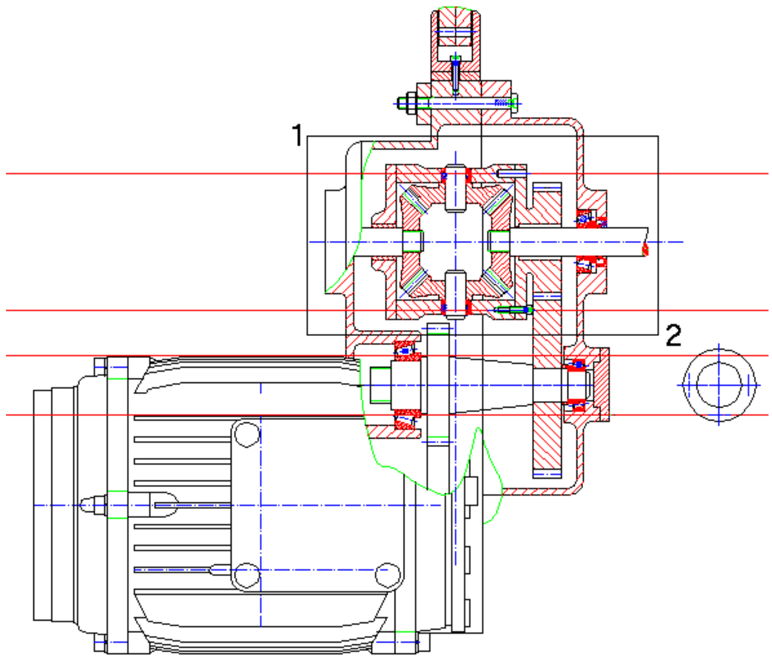
Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

3 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` *Enter W, press* ENTER

`Specify first corner:` *Specify the first corner point (1)*

`Specify opposite corner:` *Specify the second corner point (2)*



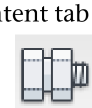
Save your file under a different name or to a different directory to preserve the original tutorial file.

Inserting Screw Connections

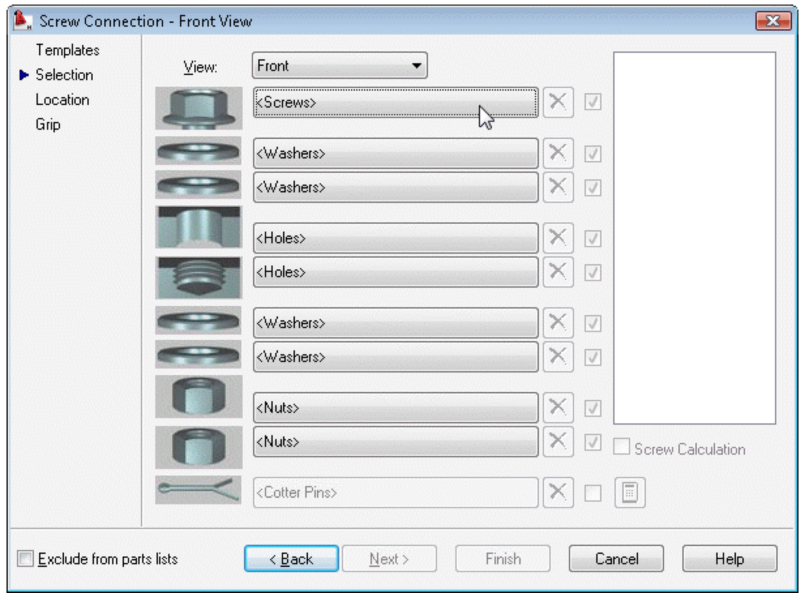
Insert a screw connection in the differential gear housing.

To insert a screw connection

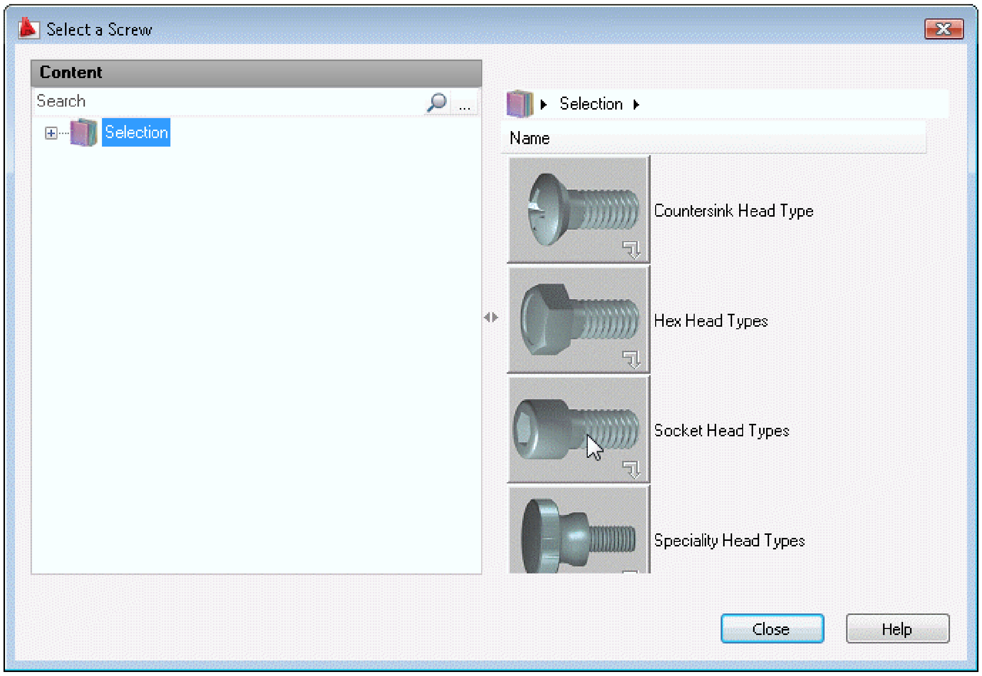
1 Start the Screw Connection command.

Ribbon	Content tab ➤ Fasteners panel ➤ Screw Connection. 
Menu	Content ➤ Screw Connection...
Command	AMSCREWCON2D

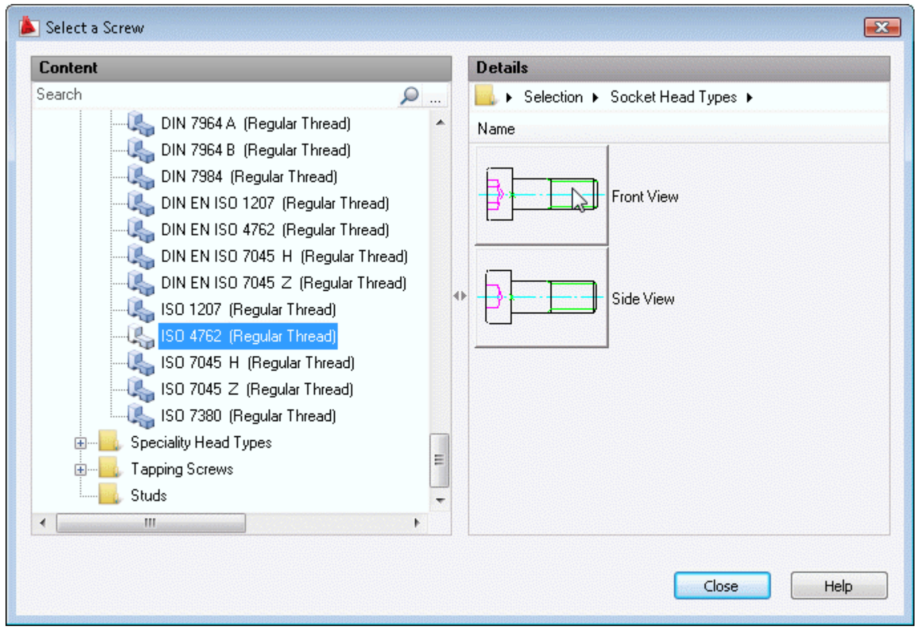
2 In the Screw Connection dialog box, click Screws.



3 In the Select a Screw dialog box, select Socket Head Types.



4 Select ISO 4762 and Front View.



You are returned to the Screw Connection - Front View dialog box.

5 In the Screw Connection - Front View dialog box, click the upper Holes button. Then select Through Cylindrical, and ISO 273 normal.

6 In the Screw Connection - Front View dialog box, click the lower Holes button. Then select Tapped Holes, Blind, and ISO 262 (Regular Thread).

NOTE The screw types available and the order depend on the standard selected to be active in AMOPTIONS, AM:Standard Parts.

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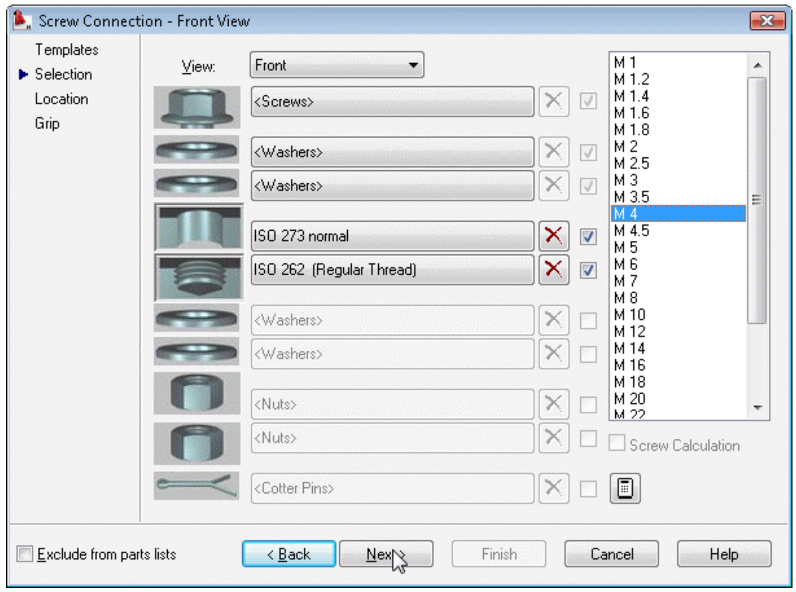
7 In the Screw Connection dialog box, specify the size M4.



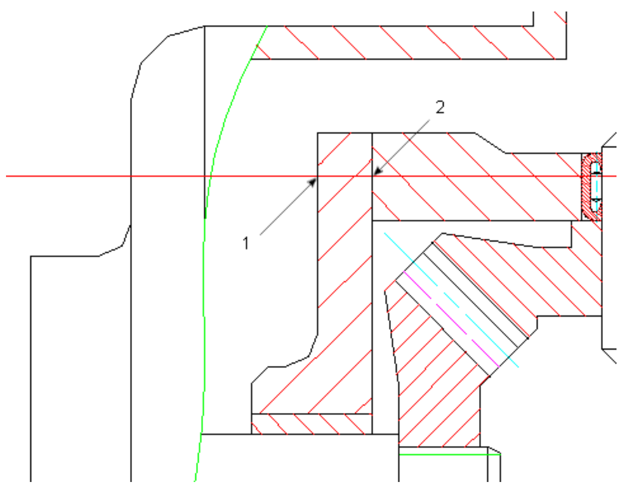
Click Next.

8 Respond to the prompts as follows:

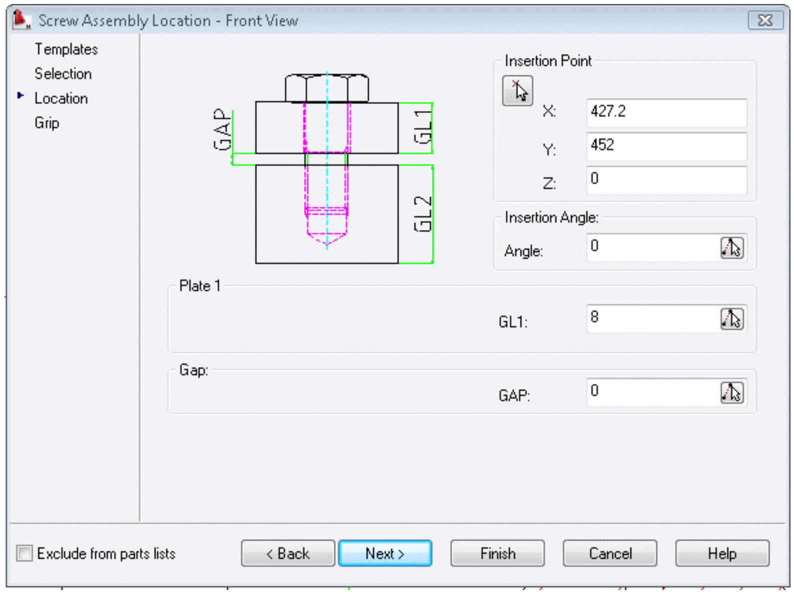
`Specify insertion point of first hole:` *Specify first point (1)*

`Specify endpoint of first hole [Gap between holes]:`

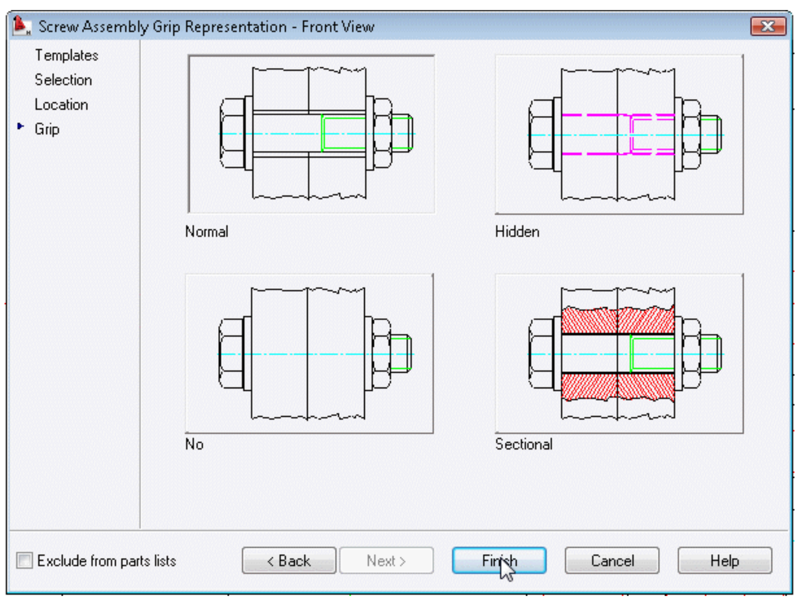
Specify second point (2)



9 In the Screw Assembly Representation - Front View dialog box, click Next.



10 In the Screw Assembly Grip Representation - Front View dialog box, click Finish.



11 Respond to the prompts as follows:

`Drag Size:`

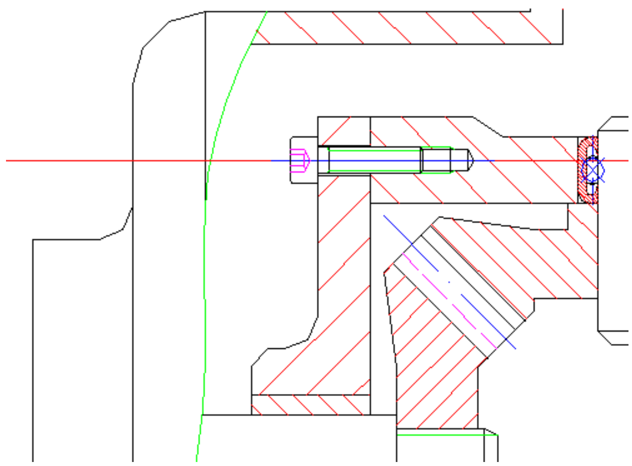
Drag the screw connection dynamically to size M4 x 16, and then click

`Drag Size:` *Enter* `12`*, press* ENTER

The screw connection is inserted with a specified a screw length of 16 mm and a blind hole depth of 12 mm.

NOTE During dragging, the size of the screw is shown as a tooltip and in the status bar, where the coordinates are usually displayed.

The background is automatically hidden, and your drawing looks like this:



Save your file.

Copying Screw Connections with Power Copy

With Power Copy, you can copy complete objects, including the information attached to those objects. In the case of a screw connection, you copy the whole screw connection to another location. The background is automatically updated.

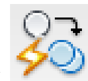
Copy the new screw connection using the Power Copy command.

To copy a screw connection

1 Start the Power Copy command.

Ribbon

Home tab ➤ Modify panel ➤ Copy. 

Content Edit tab ➤ Modify panel ➤ Copy. 

Menu Modify ➤ Power Copy

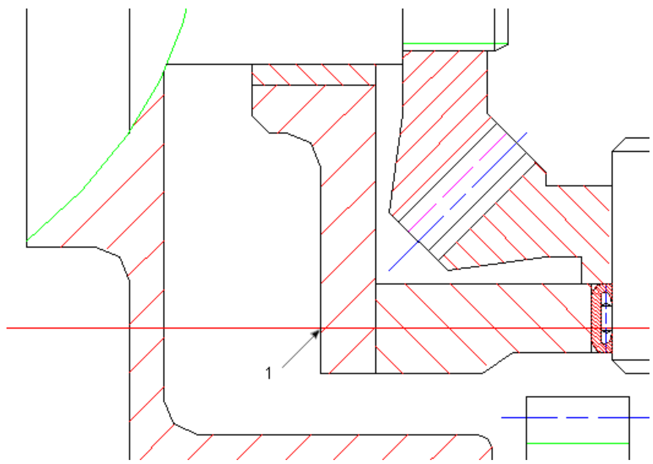
Command AMPOWERCOPY

2 Respond to the prompts as follows:

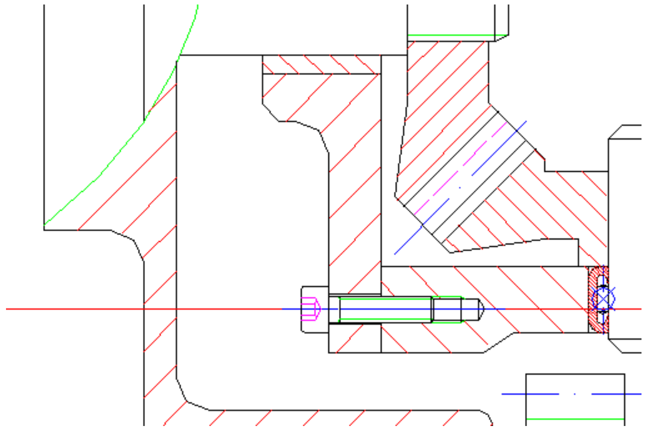
`Select object:` *Select the previously inserted screw*

`Specify insertion point:` *Specify a point (1)*

`Specify direction:` *Press* ENTER



The screw is copied to the specified location. Your drawing looks like this:



Save your file.

Creating Screw Templates

Create a screw template and store it for repeated use. This makes the insertion of identical or similar screw connections much faster.

Before you create and insert the screw template, zoom to the cover plate.

To zoom to a window

1 Zoom to the extents of the drawing.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down  ➤ Extents.
Menu	View ➤ Zoom ➤ Extents
Command	ZOOM

2 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` *Enter E, press* ENTER.

3 Zoom in to the coverplate.

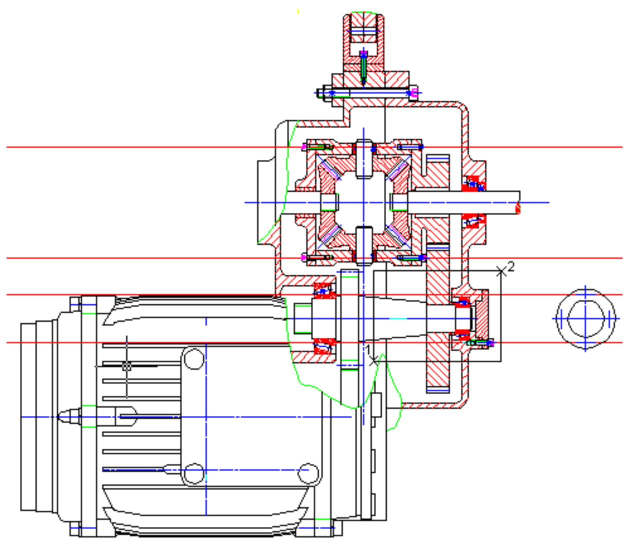
Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

4 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` *Enter W, press* ENTER

`Specify first corner:` *Specify first corner point (1)*

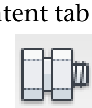
`Specify opposite corner:` *Specify second corner point (2)*



Start the screw connection and create a screw template.

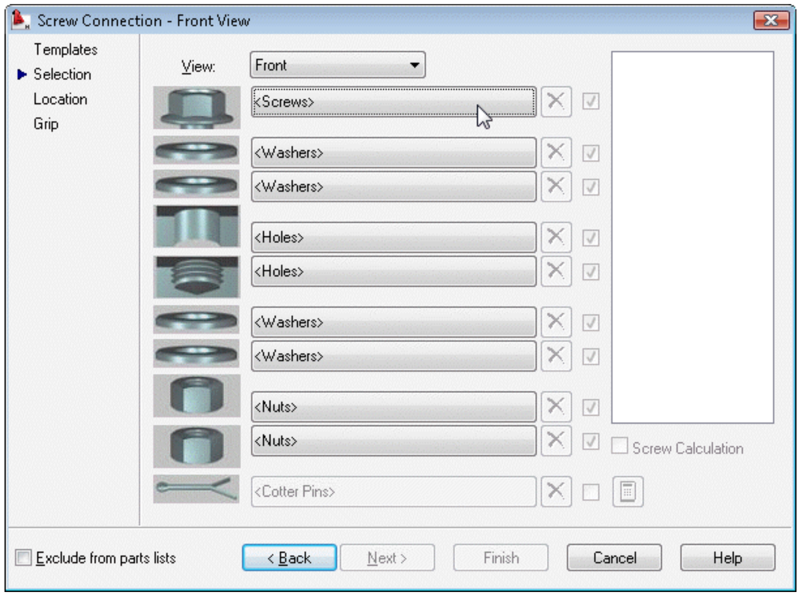
To create a screw template

1 Start the Screw Connection command.

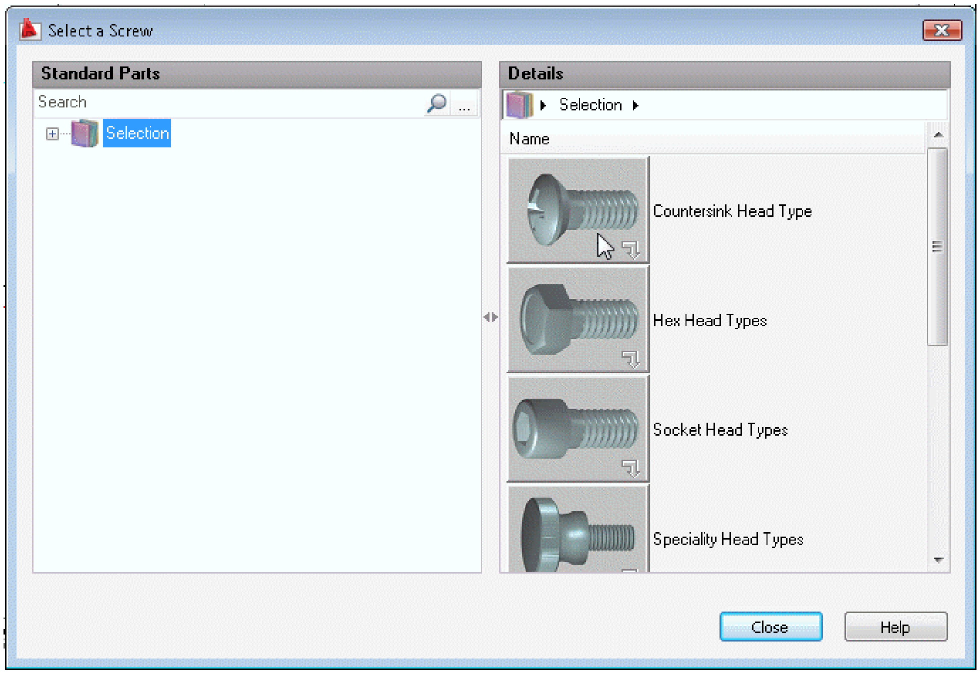
Ribbon	Content tab ➤ Fasteners panel ➤ Screw Connection. 
Menu	Content ➤ Screw Connection...

Command	AMSCREWCON2D

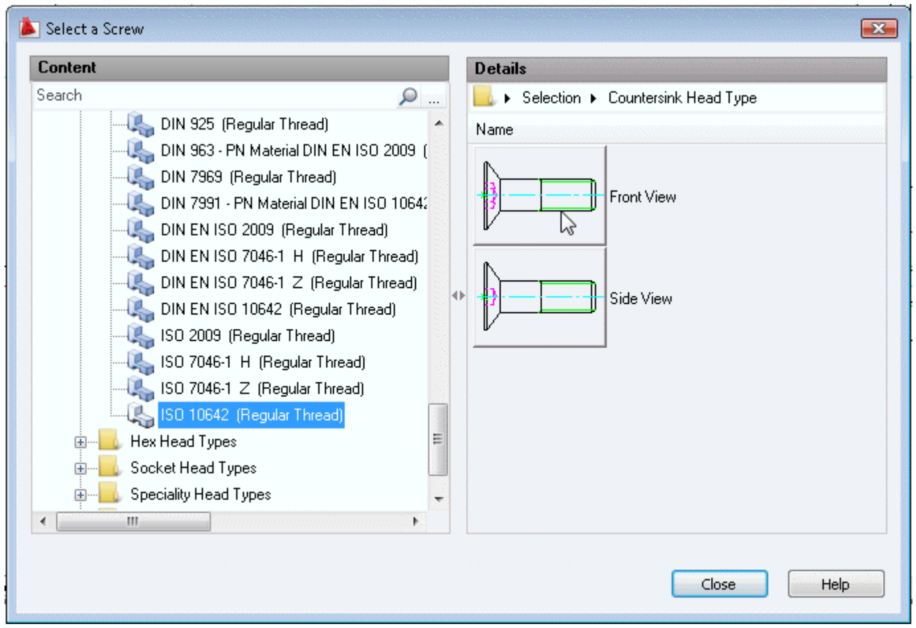
2 In the Screw Connection dialog box, click the Screws button.



3 In the Select a Screw dialog box, select Countersink Head Type.



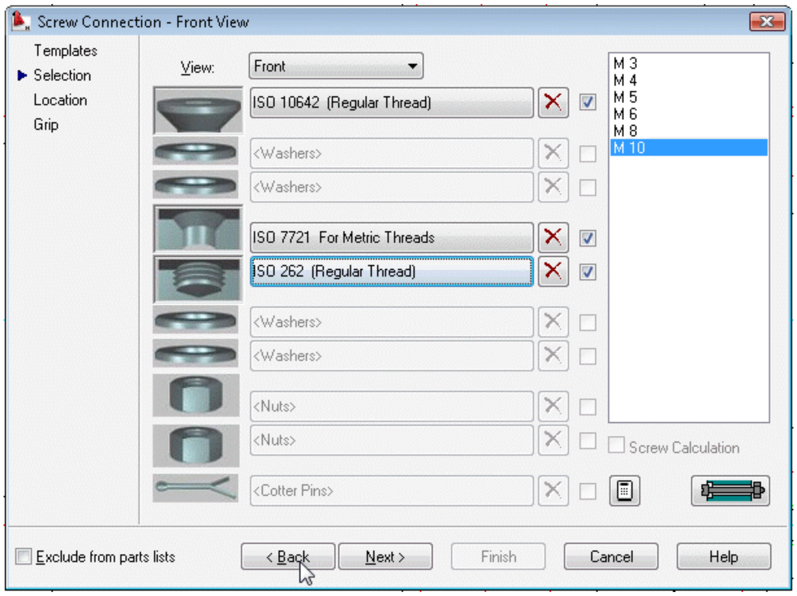
4 Select ISO 10642, and Front View.



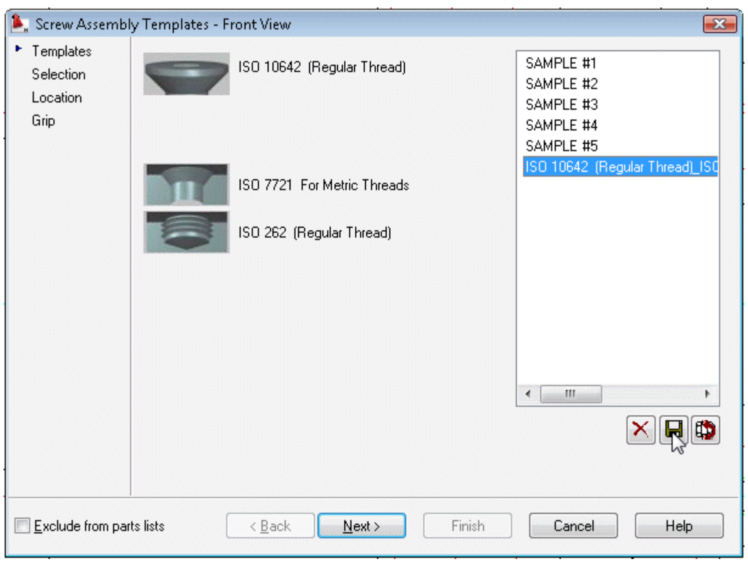
5 In the Screw Connection - Front View dialog box, click the upper Holes button. Then select Countersinks, and ISO 7721.

6 In the Screw Connection - Front View dialog box, click the lower Holes button. Then select Tapped Holes, Blind, and ISO 262.

7 In the Screw Connection - Front View dialog box, click Back to store the screw template.



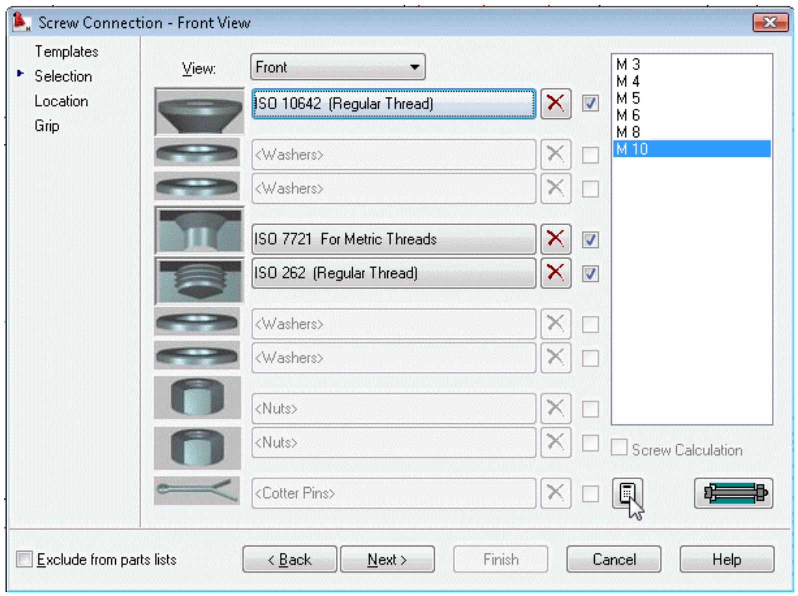
8 In the Screw Assembly Templates dialog box, click the Save icon. Your screw connection is stored as a template and is added to the list.



Click Next.

NOTE The screw template contains the combination of the used standard parts. It contains no sizes, like diameters or lengths.

9 In the Screw Connection dialog box, click the Pre-calculation icon.



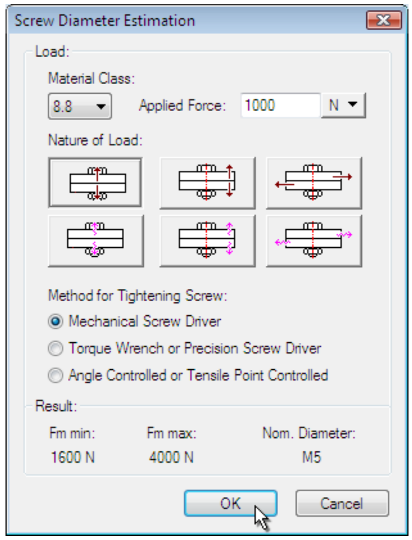
10 In the Screw Diameter Estimation dialog box, specify:

`Material Class:` 10.9

`Applied Force:` 1500

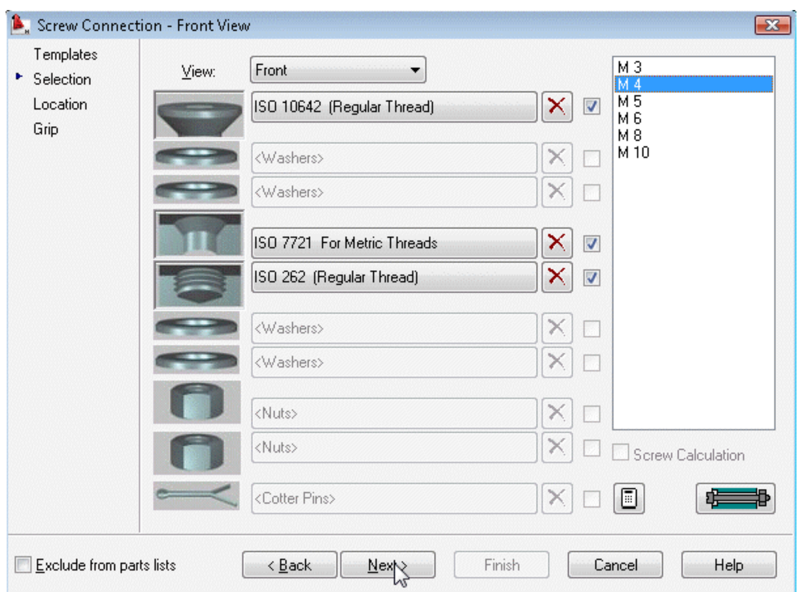
`Nature of Load:` Static and Centric applied Axial Force *(upper-left icon)*

`Method for Tightening Screw:` Mechanical Screw Driver



The Result field displays a nominal diameter size of M4. Click OK.

11 In the Screw Connection - Front View dialog box, the pre calculation routine has marked M4.



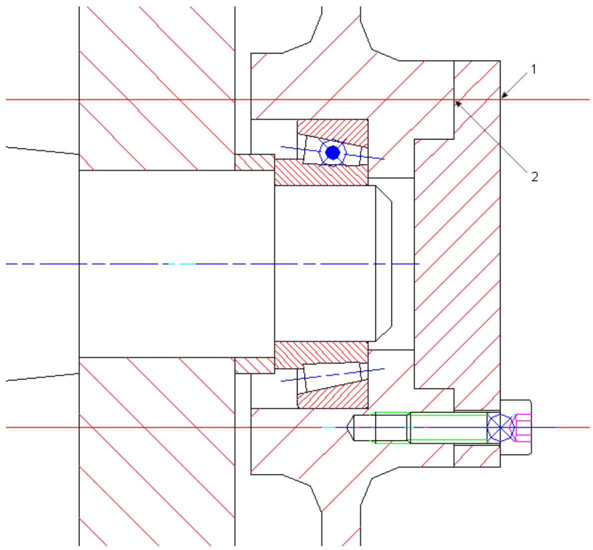
Click Next.

12 Respond to the prompts as follows:

`Specify insertion point of first hole:` *Specify first point (1)*

`Specify endpoint of first hole [Gap between holes]:`

Specify second point (2)



13 In the Screw Assembly Location - Front View dialog box, click Next.

14 In the Screw Assembly Grip Representation - Front View dialog box, click Finish.

15 Respond to the prompts as follows:

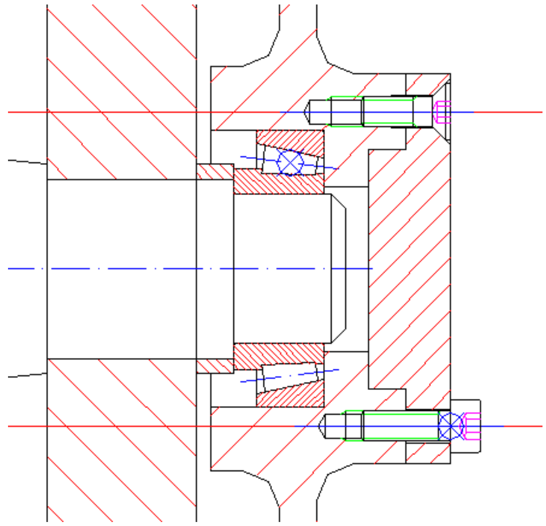
`Drag Size:`

Drag screw connection dynamically to size M4 x 12, and then click

`Drag Size:` *Enter 8, press* ENTER

The screw connection is inserted with a screw length of 12 mm and a blind hole depth of 8 mm.

Your drawing looks like this:



Save your file.

Editing Screw Connections with Power Edit

Rather than use different editing commands for different objects, you can use only one command, Power Edit, for editing all objects in a drawing with built-in intelligence. When you use Power Edit on a screw connection, the whole assembly can be edited and is updated in your drawing with an automatic background update.

Change the screw connections to the appropriate length.

To edit a screw connection that is not yet structured

1 Start the Power Edit command.

Ribbon	None.
Menu	Default Menu ➤ Power Edit
	Edit Menu ➤ Power Edit
	Modify ➤ Power Edit
Command	AMPOWEREDIT

2 Respond to the prompts as follows:

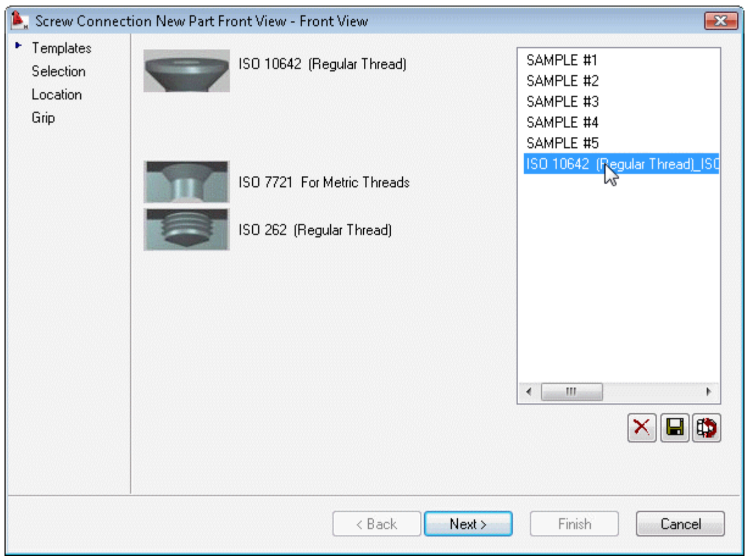
`Select object:` *Select the lower screw of the coverplate*

`Enter an option [Next/Accept] <Accept>:` *Press ENTER*

NOTE You can also start Power Edit by double-clicking the desired part.

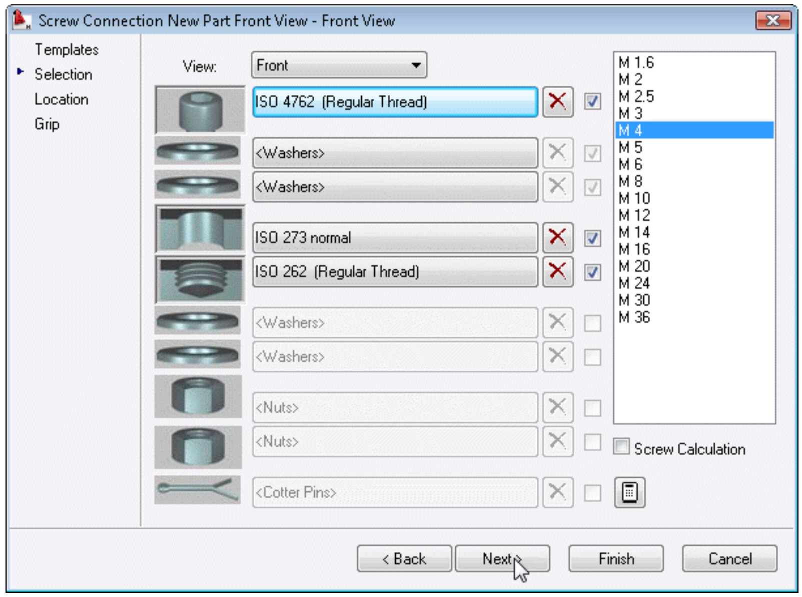
3 In the Screw Connection New Part Front View - Front View dialog box, click Back.

4 On the Templates page, double-click the ISO 10642 screw template in the list, or select it and click the Load the template icon.



The Screw Connection New Part Front View - Front View dialog box contains the screw connection as it has been stored in the template.

5 Select the size M4, and then click Next.



6 Respond to the prompts as follows:

`Specify insertion point of first hole:` *Press* ENTER

`Specify endpoint of first hole [Gap between holes]:` *Press* ENTER

7 In the Screw Connection New Part Front View - Front View dialog box, Location representation, click Next.

8 In he Screw Connection New Part Front View - Front View dialog box, Grip representation, click Finish.

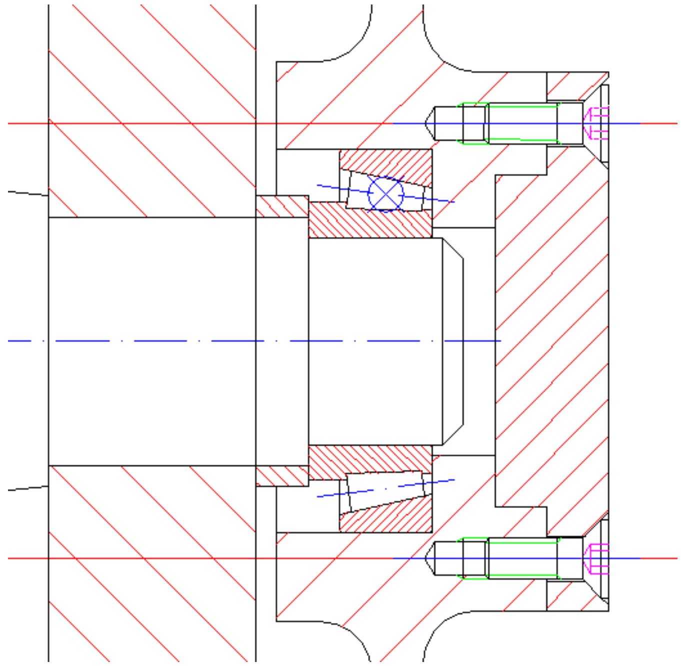
`Drag Size:`

Drag the screw connection dynamically to the size M4 x 12, and then click

`Drag Size:` *Enter 8, press* ENTER

The screw connection is edited to a screw length of 12 mm and a blind hole depth of 8 mm.

Your drawing looks like this:



Save your file.

Working with Power View

With Power View, you can quickly generate a top or bottom view of a side view of a standard part and vice versa.

Before you complete the top view of the coverplate, you have to zoom into it.

To zoom to the cover plate

1 Zoom to the extents of the drawing.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down  ➤ Extents.
Menu	View ➤ Zoom ➤ Extents
Command	ZOOM

2 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` *Enter E, press* ENTER

3 Zoom in to the coverplate.

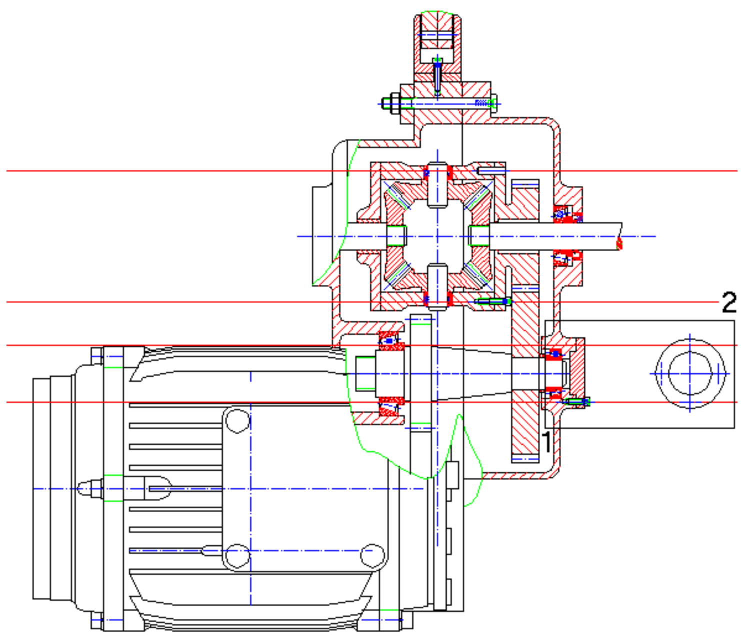
Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down  ➤ Window.
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

4 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` *Enter W, press* ENTER

`Specify first corner:` *Specify first corner point (1)*

`Specify opposite corner:` *Specify second corner point (2)*



Use Power View to insert the screws into the top view of the coverplate.

To insert a standard part using Power View

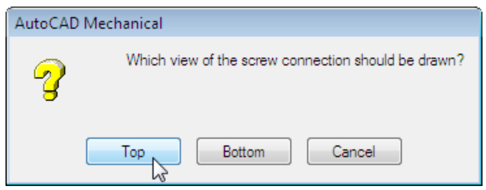
1 Start the Power View command.

Ribbon	None.
Menu	Modify ➤ Power View
Command	AMPOWERVIEW

2 Respond to the prompt as follows:

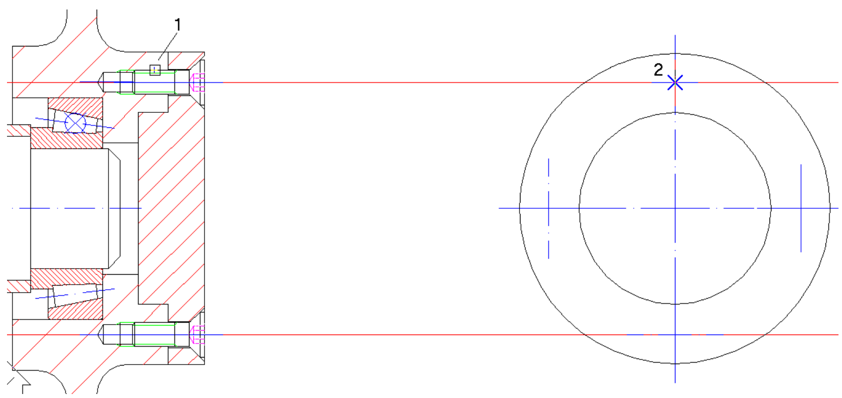
`Select object:` *Select the screw at cover plate (1)*

The AutoCAD Mechanical message box is displayed.

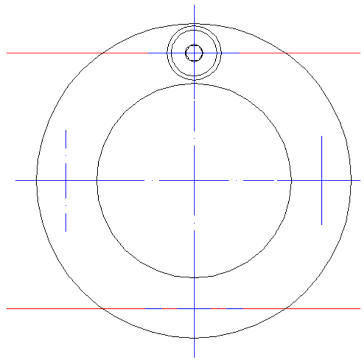


3 Click Top, and respond to the prompt as follows:

`Specify insertion point:` *Specify the centerline cross at top view (2)*

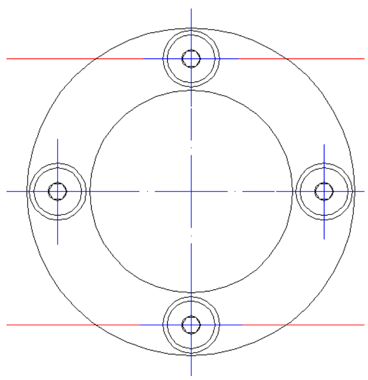


The top view of the screw connection is inserted into the top view of the coverplate. Your drawing should look like this:



4 Repeat steps 1 and 2 to insert the top view of the screw at the other three centerline crosses of the top view of the coverplate.

The coverplate should look like this:



Save your file.

Deleting with Power Erase

Power Erase is an intelligent erase command. It detects the object information of a part. If you delete a screw connection with Power Erase, the representation of the background is automatically corrected.

Before you delete the standard part, you have to zoom into it.

To zoom to the standard part to delete

1 Zoom to the extents of the drawing.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Extents. 
Menu	View ➤ Zoom ➤ Extents
Command	ZOOM

2 Respond to the prompts as follows:

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: *Enter E, press* ENTER

3 Zoom in to the area of interest.

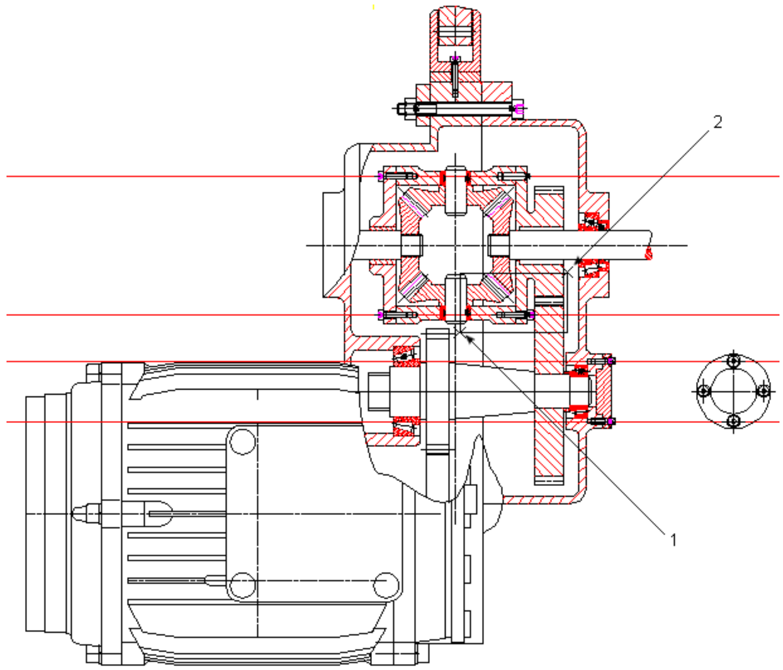
Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

4 Respond to the prompts as follows:

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: *Enter W, press ENTER*

Specify first corner: *Specify first corner point (1)*

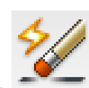
Specify opposite corner: *Specify second corner point (2)*



Delete a screw using the Power Erase command.

To delete a standard part

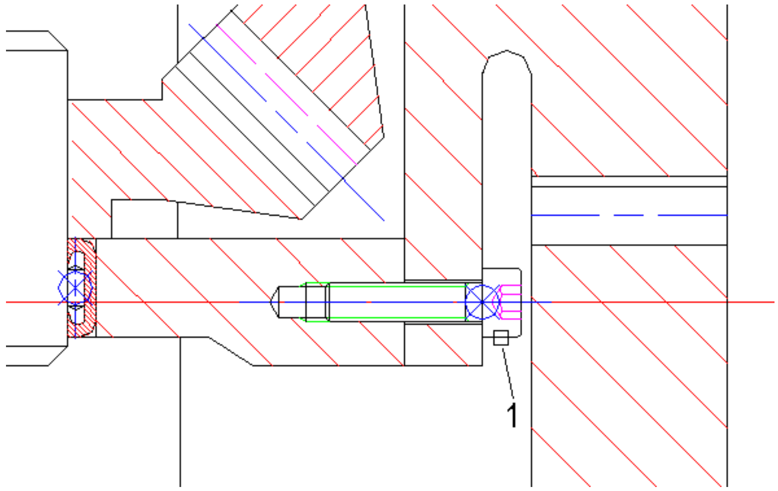
1 Start the Power Erase command.

Ribbon	Home tab ➤ Modify panel ➤ Erase. 
Menu	Modify ➤ Power Erase
Command	AMPOWERERASE

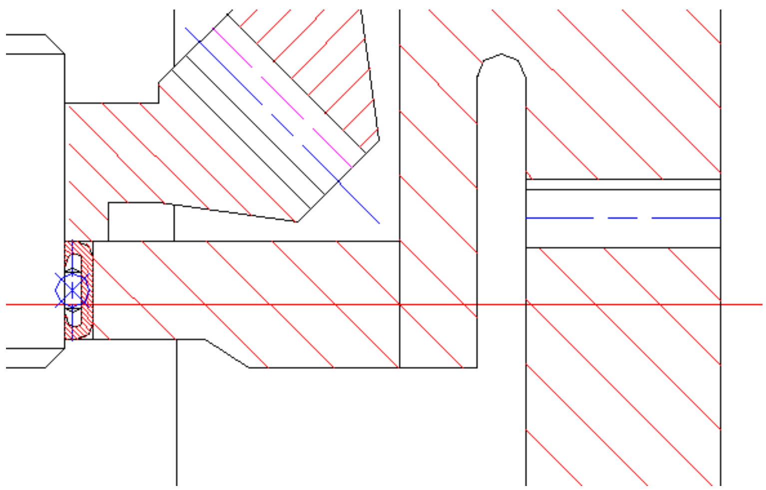
2 Respond to the prompts as follows:

`Select object:` *Select the screw (1)*

`Select object:` *Press* ENTER



The screw connection is deleted and the lines and hatch are restored.



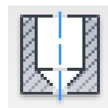
Save your file.

Inserting Holes

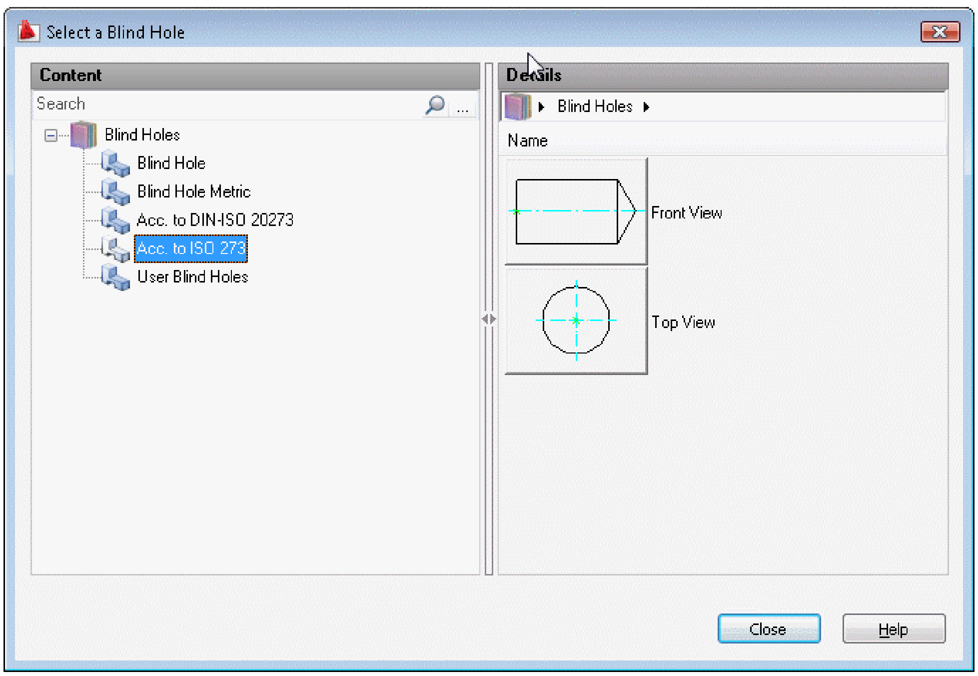
Replace the previously deleted screw connection with a pin. First you insert a blind hole for the pin.

To insert a hole

1 Start the Blind Hole command.

Ribbon	Content tab ➤ Holes panel ➤ Blind Hole. 
Menu	Content ➤ Holes ➤ Blind Holes...
Command	AMBHOLE2D

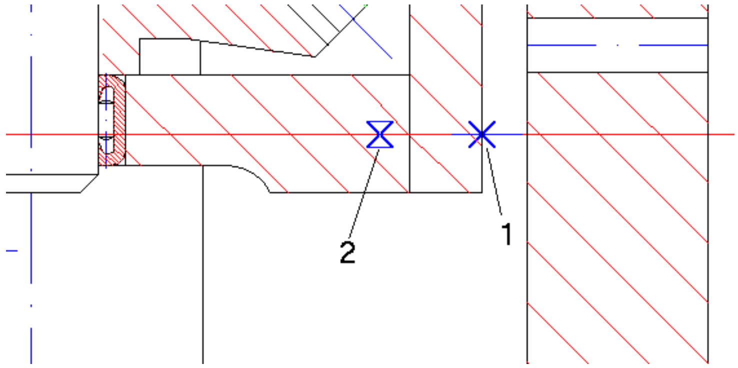
2 In the Select a Blind Hole dialog box, select Acc. to ISO 273, and Front View.



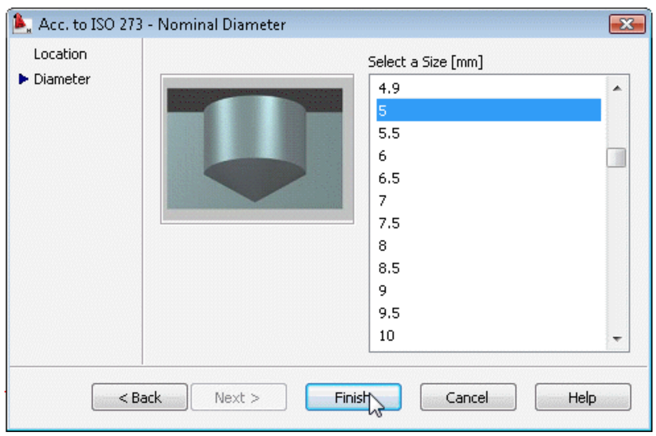
3 Respond to the prompts as follows:

`Specify insertion point:` *Specify insertion point (1)*

`Specify rotation angle <0>:` *Specify a point to define insertion angle (2)*



4 In the Acc. to ISO 273 - Nominal Diameter dialog box, select a size of 5, and then click Finish.

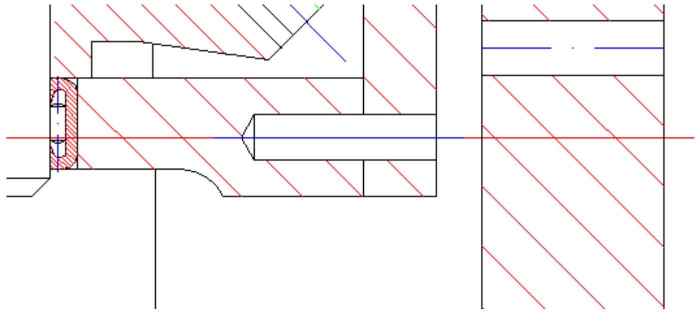


5 Continue to respond to the prompts as follows:

`Drag Size:` *Enter 20, press* ENTER

The blind hole is inserted.

Your drawing should look like this:



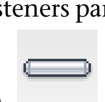
Save your file.

Inserting Pins

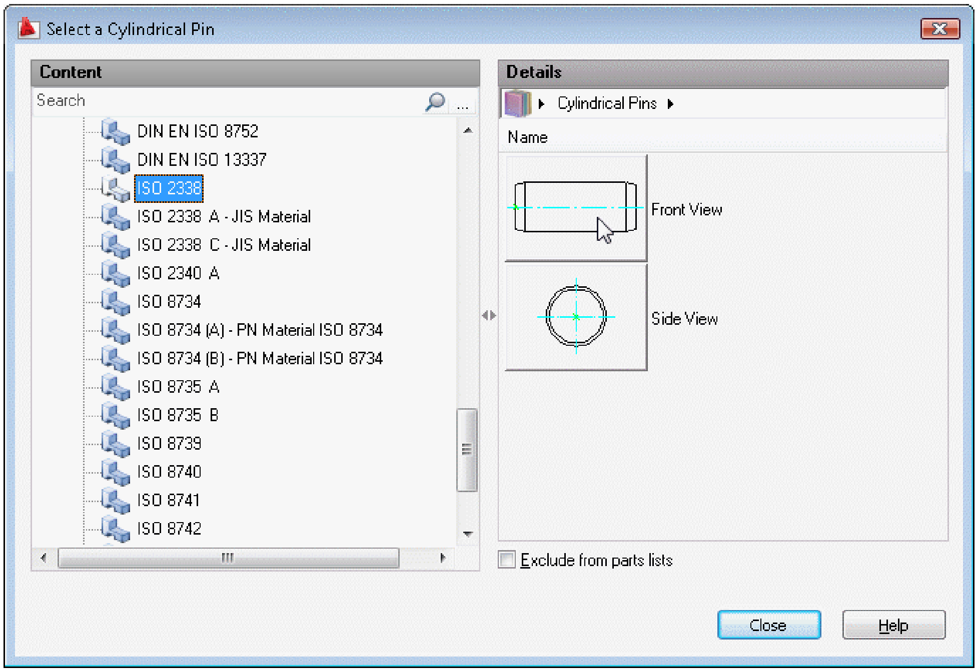
Insert a pin into the blind hole.

To insert a pin

1 Start the Cylindrical Pins command.

Ribbon	Content tab ➤ Fasteners panel ➤ Pins drop-down ➤ Cylindrical Pin. 
Menu	Content ➤ Fasteners ➤ Cylindrical Pins...
Command	AMCYLPIN2D

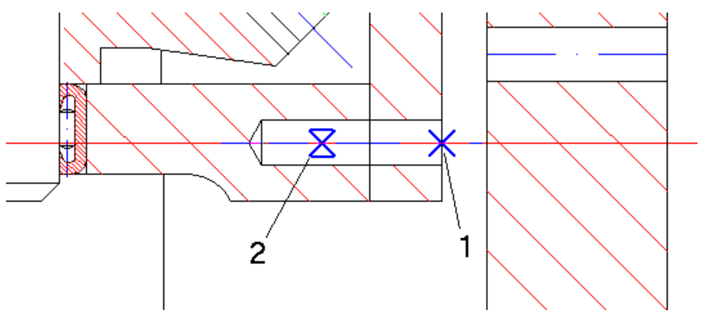
2 In the Select a Cylindrical Pin dialog box, select ISO 2338 and Front View.



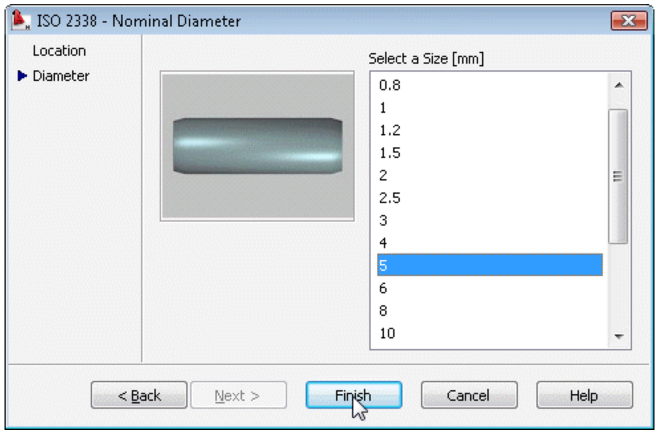
3 Respond to the prompts as follows:

`Specify insertion point:` *Specify insertion point (1)*

`Specify rotation angle <0>:` *Specify a point to define insertion angle (2)*



4 In the ISO 2338 - Nominal Diameter dialog box, select a size of 5.



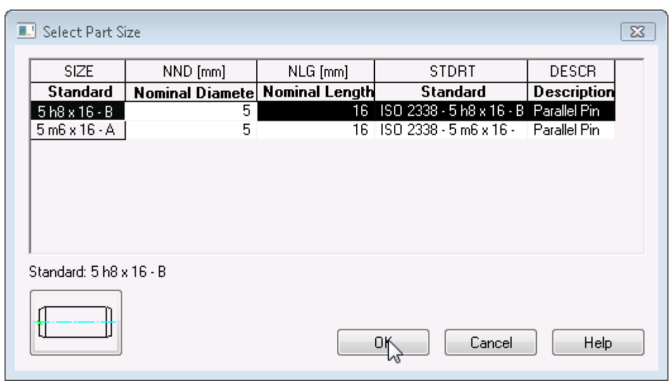
5 Click Finish, and then continue to respond to the prompt as follows:

`Drag Size:` *Drag the pin to size 5 h8 x 16 - B, and then click*

NOTE Turn the object snap (OSNAP) option off to snap to the correct pin size.

---

6 In the Select Part Size dialog box, select 5 h8 x 16 - B, and then Click OK.



You inserted the blind hole first, and then the pin. This results in overlapping centerlines. In order to have a correct plot, turn one centerline off.

To turn off a centerline

1 Select the previously inserted cylindrical pin.

2 Right-click, and on the shortcut menu deactivate Centerlines on/off.

With the centerline of the pin turned off, only the centerline of the blind hole is displayed.

Save your file.

Turning Off Centerlines in Configurations

If your drawing already contains holes with centerlines, and you want to add standard parts, it is recommended to turn off the centerlines for standard parts in the configuration. This avoids removing overlapped centerlines.

To turn off centerlines in the configuration

1 Open the Options dialog box.

Ribbon	None.
Menu	Tools ➤ Options...
Command	AMOPTIONS

2 On the AM:Standard Parts tab, clear the Draw Centerlines check box.

Click Apply, and then Click OK.

Hiding Construction Lines

For a better overview, you can hide the construction lines by turning them off temporarily.

Zoom to the extents of the drawing.

To zoom to the extents

1 Zoom to the extents of the drawing.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Extents. 
Menu	View ➤ Zoom ➤ Extents
Command	ZOOM

2 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` *Enter E, press* ENTER

To turn off C-lines

- Start the Construction Line On/Off command.

Ribbon	Home tab ➤ Layers panel drop-down ➤ Construction Lines On/Off. 
Menu	Format ➤ Layer Tools ➤ Construction Line On/Off
Command	AMCLINEO

All construction lines are turned off.

Save your file.

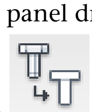
Simplifying Representations of Standard Parts

In some cases, such as in complex assemblies, it is helpful to have a simplified representation of the standard parts for a better overview. With AutoCAD Mechanical, you can switch between different representation types without losing object or part information.

Change the representation of the differential gear screws.

To change the representation of a standard part

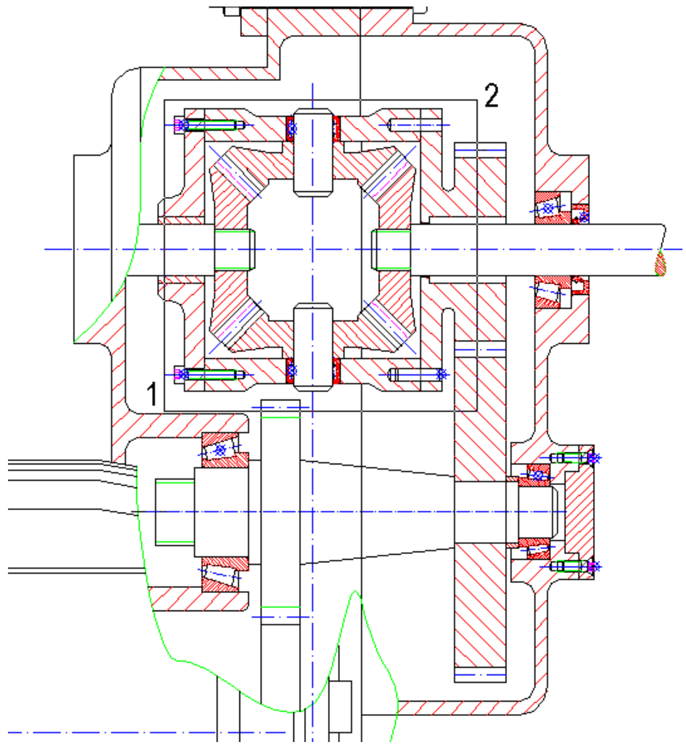
1 Start the Change Representation command.

Ribbon	Content tab ➤ Library panel drop-down ➤ Change Representation. 
Menu	Content ➤ Change Representation...
Command	AMSTDPREP

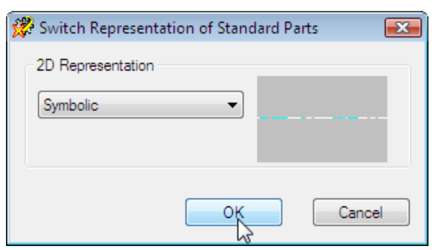
2 Respond to the prompts as follows:

`Select objects:` *Select the differential gear with a window (1, 2)*

`Select objects:` *Press* ENTER

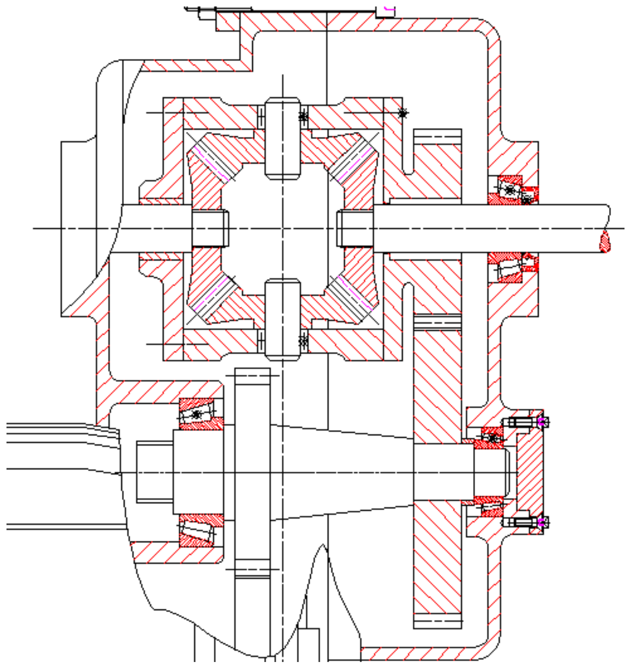


3 In the Switch Representation of Standard Parts dialog box, select Symbolic.



Click OK.

The representation of the selected standard parts is symbolic. Your drawing should look like this:



All of the standard parts you inserted in this exercise are listed in the mechanical browser.

Save your file. This is the end of this tutorial chapter.

Working with BOMs and Parts Lists

10

In AutoCAD® Mechanical, you can create parts lists and bills of material (BOMs), and modify part references and balloons. In this chapter, you insert and edit a parts list, and work with the bill of material (BOM) database.

Key Terms

Term	Definition
balloon	Circular annotation tag that identifies a bill of material item in a drawing. The number in the balloon corresponds with the number of the part in the bill of material.
bill of material	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.
BOM attribute	An entity that contains attributes by default (the attribute is invisible) that can add information to and describe details of a part in the drawing. The values of these attributes are transformed into the parts list attributes when converting BOM attributes and creating a parts list.
part reference	Part information for a bill of material, which is attached to the part in the drawing.

Term	Definition
parts list	A dynamic list of parts and associated attributes generated from a bill of material database. The parts list automatically reflects additions and subtractions of parts from an assembly.

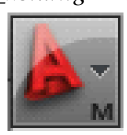
Working with Parts Lists

The drawing used for this exercise is not structured. In structured drawings, BOMs and parts list are generated automatically, and it is not necessary to insert part references manually.

Open the initial drawing.

To open a drawing

1 Open the file *tut_pts_list.dwg* in the Tutorial folder.

Ribbon	 ➤ Open ➤ Drawing
Menu	File ➤ Open...
Command	OPEN

NOTE The path to the folder containing tutorial files is;

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains a shaft with a housing.

2 Zoom in to the area of interest.

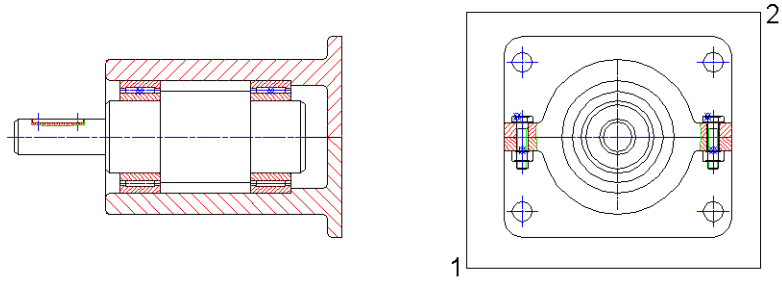
Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

3 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` *Enter W, press* ENTER

`Specify first corner:` *Specify the first corner point figure (1)*

`Specify opposite corner:` *Specify the second corner point (2)*



Save your file under a different name or to a different directory to preserve the original tutorial file.

Inserting Part References

Part references contain the part information required for a bill of material. The information in a part reference is available in the BOM database for creating a parts list.

Use the part reference command to enter part information for your part.

To insert a part reference

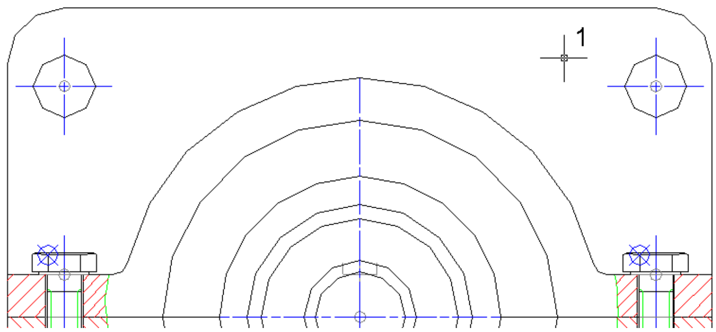
1 Start the Part Reference command.

Ribbon	Annotate tab ➤ BOM panel ➤ Part Reference drop-down ➤ Create. 
Menu	Annotate ➤ Part Reference
Command	AMPARTREF

2 Respond to the prompts as follows:

`Select point or [Block/Copy/Reference]:`

Specify a point on the part (1).

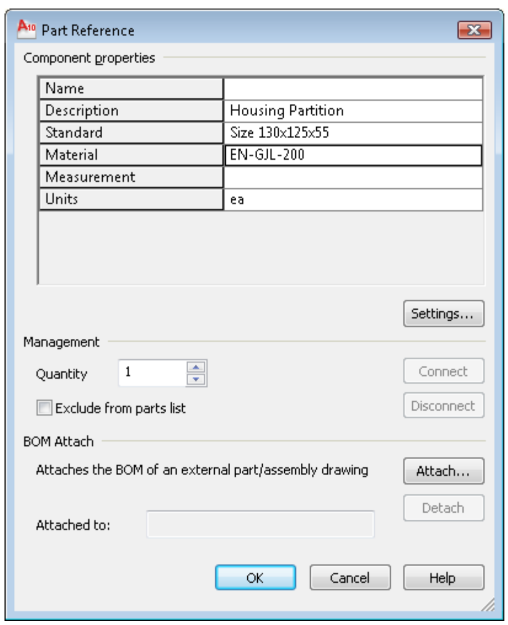


3 In the Part Reference dialog box, specify:

`Description:` Housing Partition

`Standard:` Size 130x125x55

`Material:` EN-GJL-200

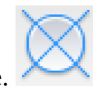


Click OK.

The Part Reference is inserted into the drawing. In the next step, you create a part reference by reference.

To insert a part reference by reference

1. Start the Part reference command again.

Ribbon	Annotate tab ➤ BOM panel ➤ Part Reference drop-down ➤ Create. 
Menu	Annotate ➤ Part Reference
Command	AMPARTREF

2. Respond to the prompts as follows:

   `Select point or [Block/Copy/Reference]:`

   Enter R at the command prompt to select Reference.

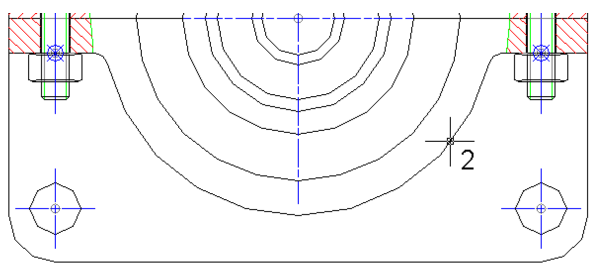
3. In the drawing, select the previously inserted part reference to create a reference.

   NOTE You can use the option Copy to create a new part with similar text information.

4. Respond to the prompts as follows:

   `Enter an option [Next/Accept]<Accept>:` *Press* ENTER

5. `Select point or:` *Specify the insertion point at the circular edge (2)*



6. In the Part Reference dialog box, click OK.

   NOTE This part reference looks different, because it has been attached to an object (the circular edge) of the part.

   Subsequently, when you generate the parts list, it shows a quantity of 2 for this item.

   Save your file.

Editing Part References

In this exercise, you edit an existing part reference.

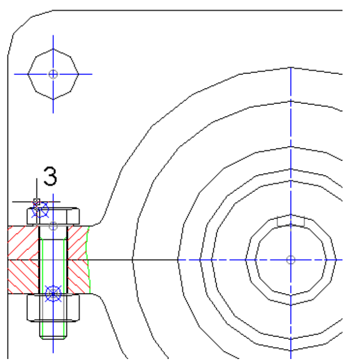
To edit a Part Reference

1 Start the Part Reference Edit command.

Ribbon	Annotate tab ➤ BOM panel ➤ Part Reference drop-down ➤ Edit. 
Menu	Annotate ➤ Part Reference Edit
Command	AMPARTREFEDIT

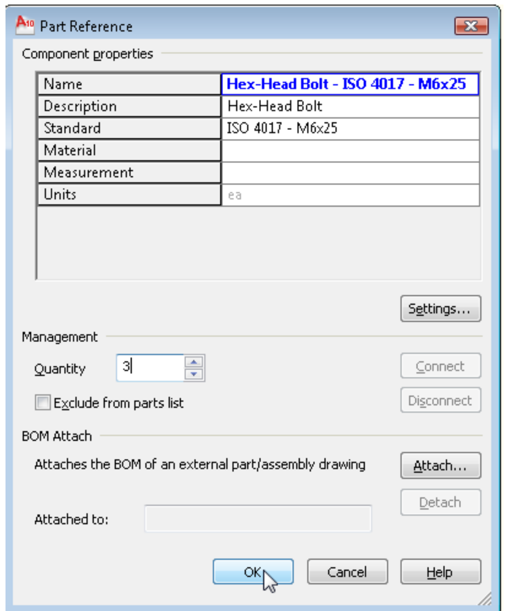
2 Respond to the prompts as follows:

`Select part reference or <Assembly properties>:` *Specify the part reference of the left bolt (3)*



`Enter an option [Next/Accept]<Accept>:` *Press* ENTER

3 In the Part Reference dialog box, Reference Quantity field, enter *3*, and then click OK.



4 Zoom to the extents to display the entire drawing.

Ribbon View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 

Menu View ➤ Zoom ➤ Window

Command ZOOM

Save your file.

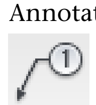
Placing Balloons

Create balloons from the part references in the drawing.

To place a balloon

1 Start the Balloon command.

Ribbon Annotate tab ➤ Balloon panel ➤ Balloons.



Menu	Annotate ➤ Balloons
Command	AMBALLOON

2 Respond to the prompt as follows:

`Select part/assembly or [auTo/autoAll/set Bom/Collect/arrow Inset/Manual/One/Renumber/rEorganize/annotation View]:` *Enter B*

NOTE At this stage the drawing doesn't contain a BOM database. As with the AMPARTLIST command, the AMBALLOON command creates a BOM database automatically. All part references are added to the database and item numbers are created inside the database. However, unless specifically instructed the commands create only the main BOM database. For the purpose of this tutorial, you must create a BOM database to contain part references held within the border; a border BOM. This is why you are instructed to enter B, to trigger the set BOM option of the AMBALLOON command.

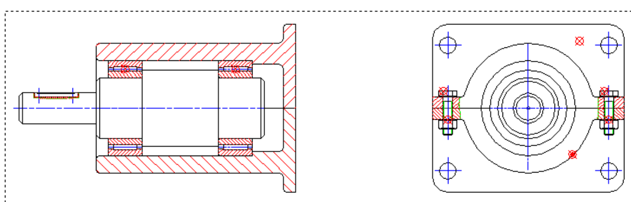
NOTE To create and edit a database manually, use the AMBOM command

`Specify BOM to create or set current [Main/?] <BORDER1>:` *Press* ENTER

`Select part/assembly or:`

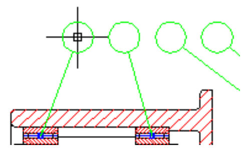
`Select part/assembly or [auTo/autoAll/set Bom/Collect/arrow Inset/Manual/One/Renumber/rEorganize/annotation View]:` *Enter A*

`Select part reference or:` *Use a window to select all objects, press* ENTER



NOTE Press ENTER to change the type of arrangement (horizontal, vertical, angle or stand-alone).

3 Place the balloons horizontally, above the assembly.



Because the balloons are numbered automatically, depending on where you located the part references, the appearance of your drawing can be different.

In the next step, you must renumber the balloons.

To renumber balloons

1 Start the Balloon command again.

Ribbon	Annotate tab ➤ Balloon panel ➤ Balloons.
Menu	Annotate ➤ Balloons
Command	AMBALLOON

2 Respond to the prompt as follows:

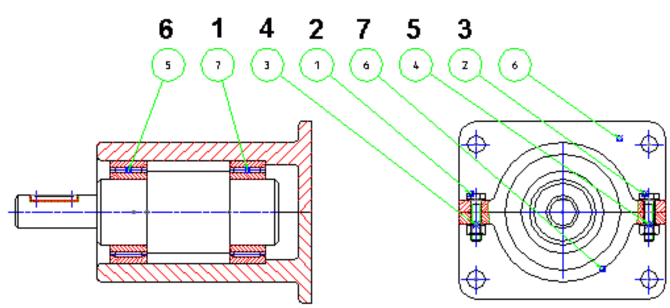
`Select part/assembly or [auTo/autoAll/set Bom/Collect/arrow Inset/Manual/One/Renumber/rEorganize/annotation View]:` *Enter R*

`Enter starting item number: <1>:` *Press* ENTER

`Enter increment: <1>:` *Press* ENTER

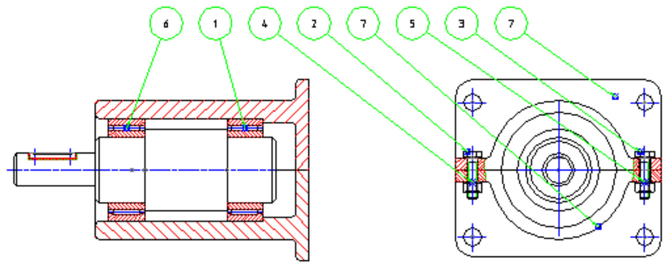
`Select balloon:` *Select the balloons in numerical order from 1 to 7*

`Select balloon:` *Press* ENTER



NOTE Since the right most balloon contains the same number as another balloon you already selected, there is no need to explicitly select it.

Your drawing must look like the following image for you to continue:



Rearrange the balloons for a better representation.

To rearrange balloons.

1 Start the Balloon command again.

Ribbon	Annotate tab ➤ Balloon panel ➤ Balloons. 
Menu	Annotate ➤ Balloons
Command	AMBALLOON

2 Respond to the prompt as follows:

`Select part/assembly or [auToi/autoAll/set Bom/Collect/arrow Inset/Manual/One/Renumber/rEorganize/annotation View]:` *Enter E*

`Select balloon:` *Select the six balloons on the right*

`Select balloon:` *Press* ENTER

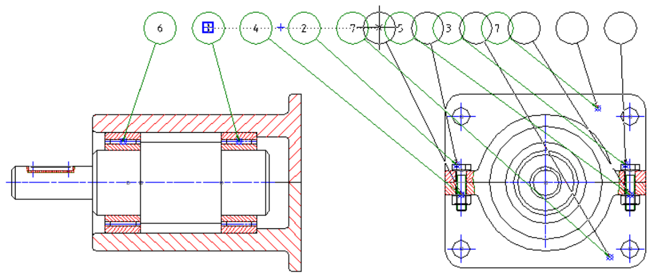
`Align [Angle/Standalone/Horizontal/Vertical]<Vertical>:`

Enter h, press ENTER

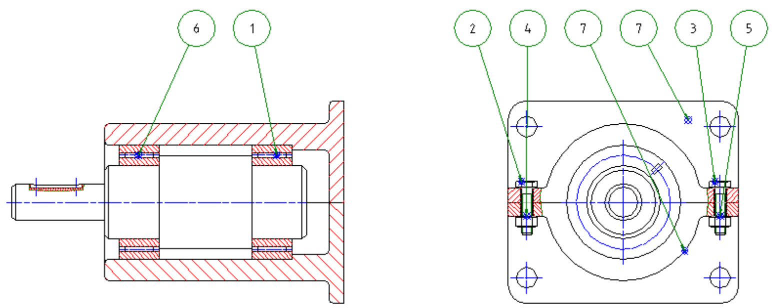
3 Move the cursor through the center of balloon 1 to get the horizontal tracking line.

NOTE Make sure that the OTRACK function is active.

4 Move the cursor to the right, and snap along the tracking line until you reach a distance of 120, and then click.



The result must look like the following image:

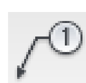


NOTE You can control snap distance within the Balloon Properties dialog box.

Create a part reference and a balloon in one step with the manual option.

To create a part reference and a balloon using the manual option

1 Start the Balloon command again.

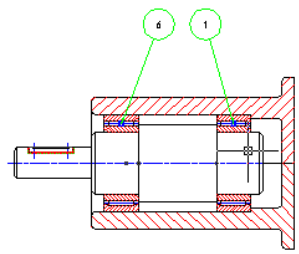
Ribbon	Annotate tab ➤ Balloon panel ➤ Balloons. 
Menu	Annotate ➤ Balloons
Command	AMBALLOON

2 Respond to the prompt as follows:

```
Select part/assembly or [auTo/autoAll/set Bom/Collect/arrow
Inset/Manual/One/Renumber/rEorganize/annotation View]:
```

Enter M, press ENTER

`Select point or [Block/Copy/Reference]:` *Select a point on the shaft*



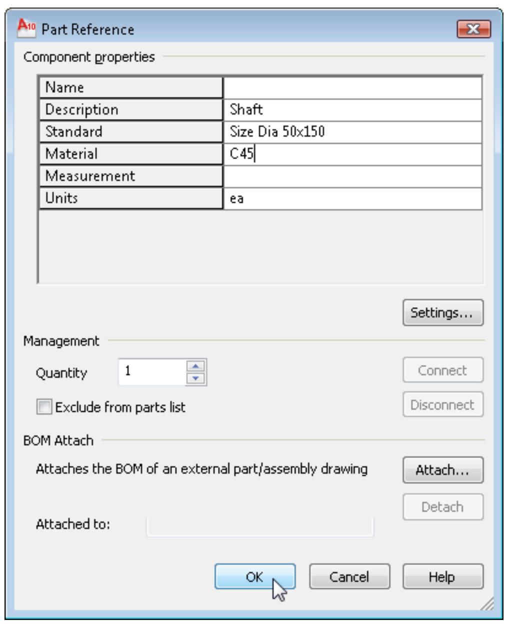
NOTE Instead of selecting a point to create a part reference, you can use Copy or Reference from the Manual option to get the information from an existing balloon or part reference.

3 In the Part Reference dialog box, specify:

`Description:` Shaft

`Standard:` Size Dia 50x150

`Material:` C45

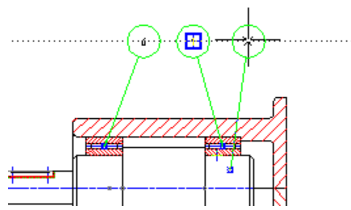


Click OK.

4 Press ENTER to start the leader line of the balloon in the center of the part reference.

5 Move the cursor through the center of balloon 1 to get the tracking line and the snap distance, and then click the insertion point.

NOTE Instead of entering the insertion point, you can select another point to create an extended leader line.



6 Press ENTER.

Save your file.

Creating Parts Lists

Generate a parts list from the part reference information.

To create a parts list

1 Start the Parts List command.

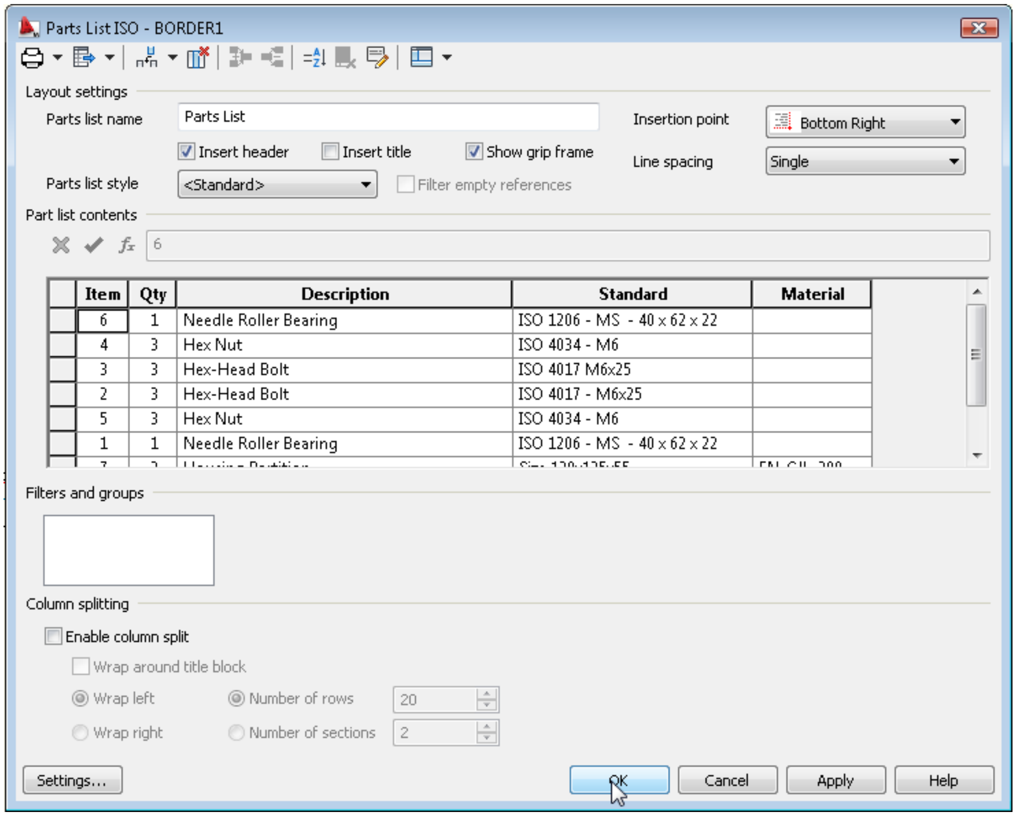
Ribbon	Annotate tab ➤ Table panel ➤ Parts List. 
Menu	Annotate ➤ Parts List
Command	AMPARTLIST

2 Respond to the prompt as follows:

```
Select border/annotation view or specify BOM to create/use
[Main/?] <MAIN>:
```

Move the cursor over the border until tooltip ISO_A2 is displayed, click the highlighted border

The Parts List dialog box is displayed.



Click OK.

The parts list appears dynamically on the cursor.

3 Move the cursor to position the parts lists above the title block, and then click to insert the parts list.

The parts list looks like the following:

8	1	Shaft	Size Dia 50x150	C45
4	3	Hex Nut	ISO 4034 - M6	
1	1	Needle Roller Bearing	ISO 1206 - MS - 40 x 62 x 22	
7	2	Housing Partition	Size 130x15x55	EN-GJL-200
2	3	Hex-Head Bolt	ISO 4017 - M6x25	
3	3	Hex-Head Bolt	ISO 4017 M6x25	
5	3	Hex Nut	ISO 4034 - M6	
6	1	Needle Roller Bearing	ISO 1206 - MS - 40 x 62 x 22	
Item	Qty	Description	Standard	Material

NOTE

- Because the balloons were originally numbered automatically, depending on where you located the part references, the order that parts are listed can be different in your drawing.
- If a drawing contains more than one border, the borders are listed in the BOM dialog box. From there you can select a particular border and view the associated parts list.

In the next exercise, you edit balloon and parts list information using several methods.

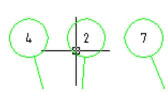
To edit parts list information

1 Start the Edit Part List/Balloon command.

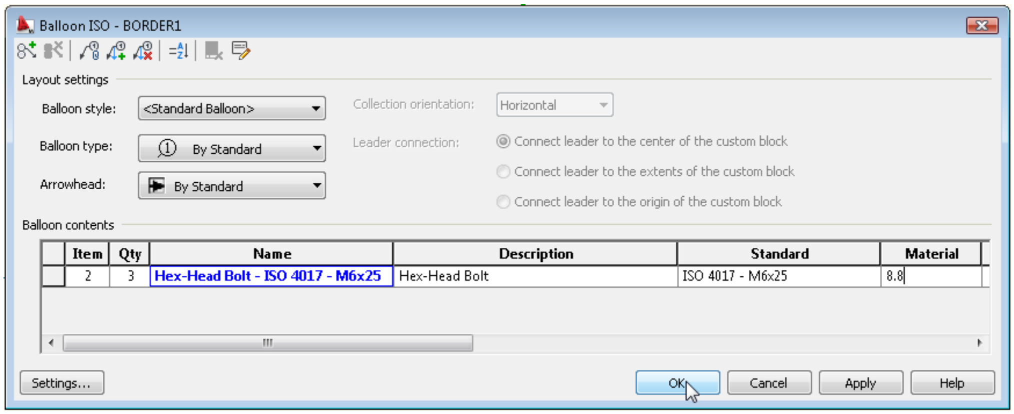
Ribbon	None.
Menu	Modify ➤ Power Edit
Command	AMPOWEREDIT

2 Respond to the prompt as follows:

`Select object:` *Select balloon 2*



3 In the Balloon dialog box, Material column, enter *8.8*.



Click OK.

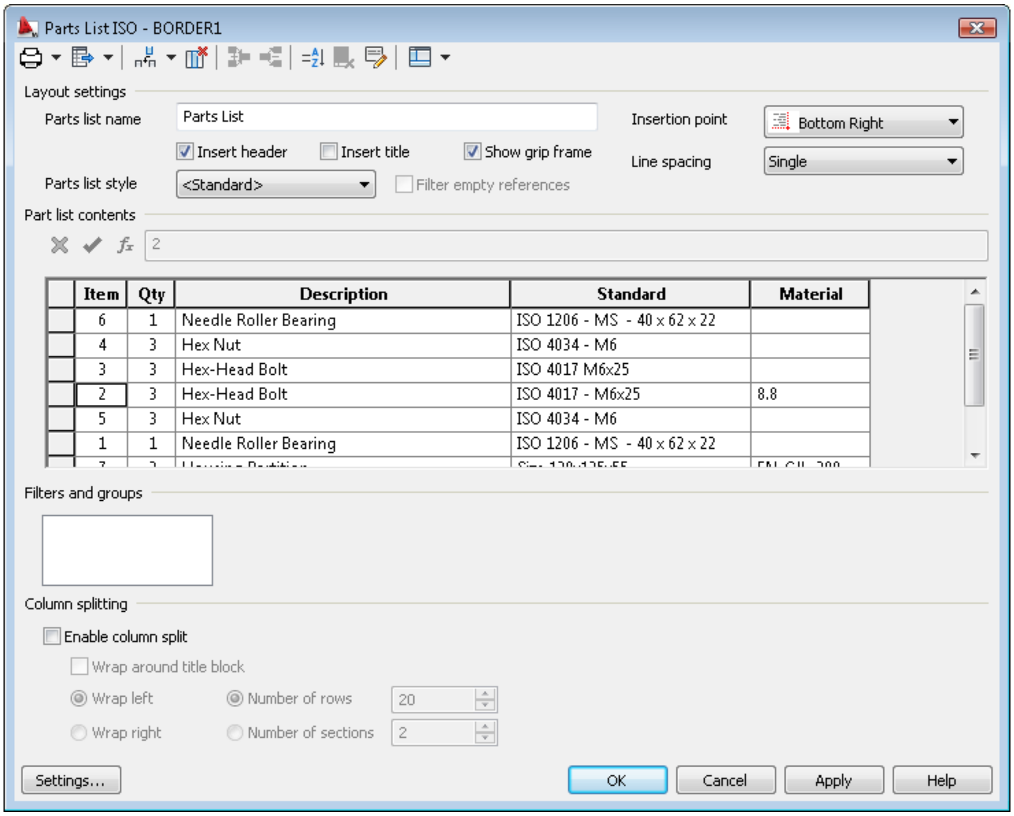
The parts list reflects the material value you added.

Item	Qty	Description	Standard	Material
8	1	Shaft	Size Dia 50x150	C45
4	3	Hex Nut	ISO 4034 - M6	
1	1	Needle Roller Bearing	ISO 1206 - MS - 40 x 62 x 22	
7	2	Housing Partition	Size 130x15x55	EN-GJL-200
2	3	Hex-Head Bolt	ISO 4017 - M6x25	8.8
3	3	Hex-Head Bolt	ISO 4017 M6x25	
5	3	Hex Nut	ISO 4034 - M6	
6	1	Needle Roller Bearing	ISO 1206 - MS - 40 x 62 x 22	

NOTE Choose Apply to see the results in the drawing immediately without leaving the dialog box. All changes made in the dialog box are associative and change the data in the drawing immediately.

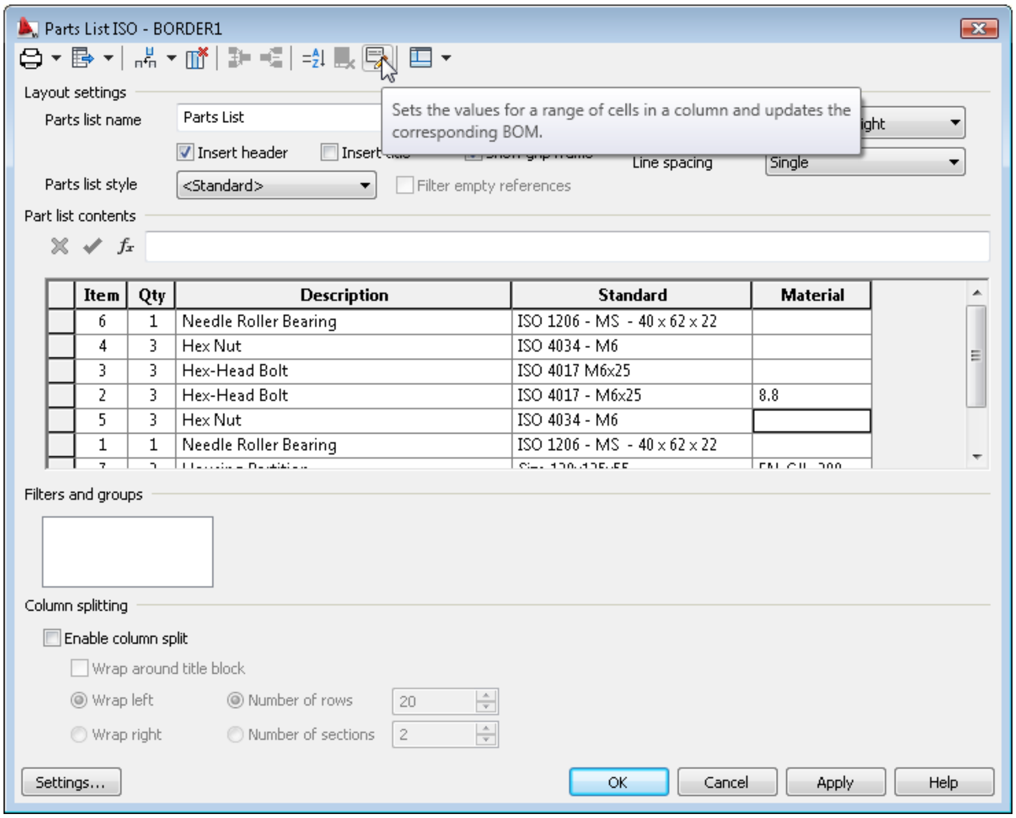
4 Double-click the parts list.

The Parts List dialog box is displayed.



You can edit your data in this dialog box. Some examples are shown next.

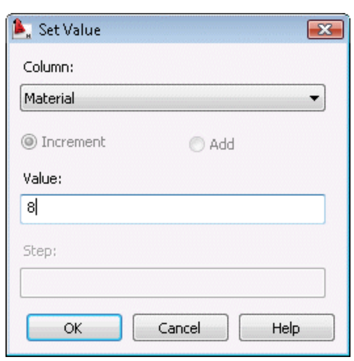
5 Select the Hex Nut entry, and then choose the Set values icon.



6 In the Set Value dialog box, specify:

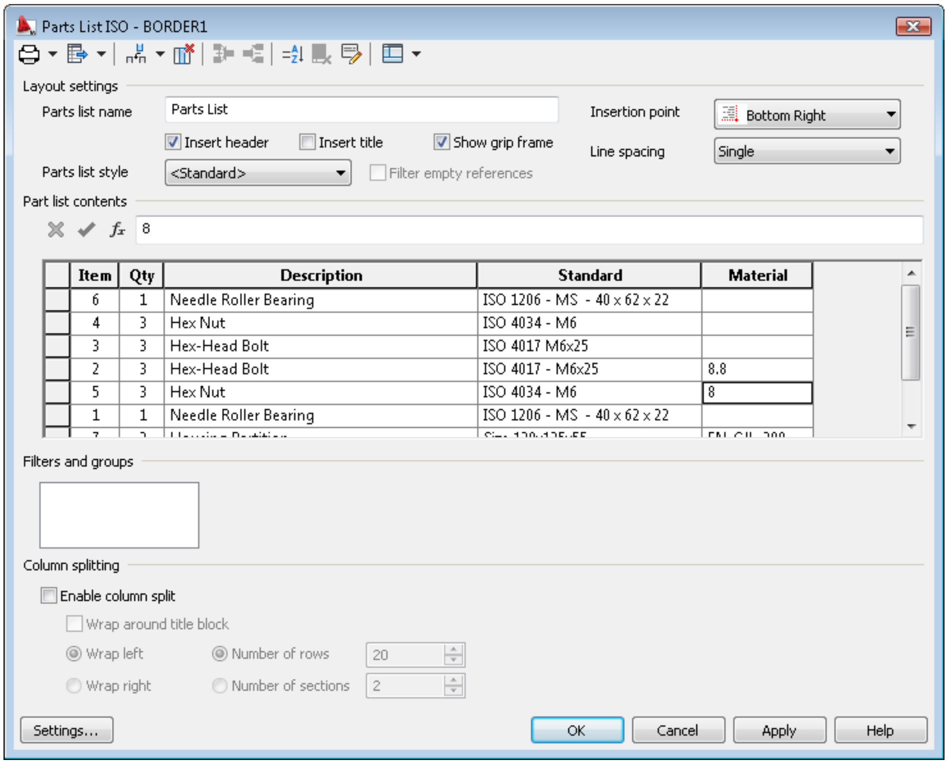
`Column:` Material

`Value:` 8



Click OK.

The material value is added to the Parts List.



7 Now, change the material of the second bolt and nut accordingly.

NOTE Use the shortcut menu inside a field to cut, copy, and paste.

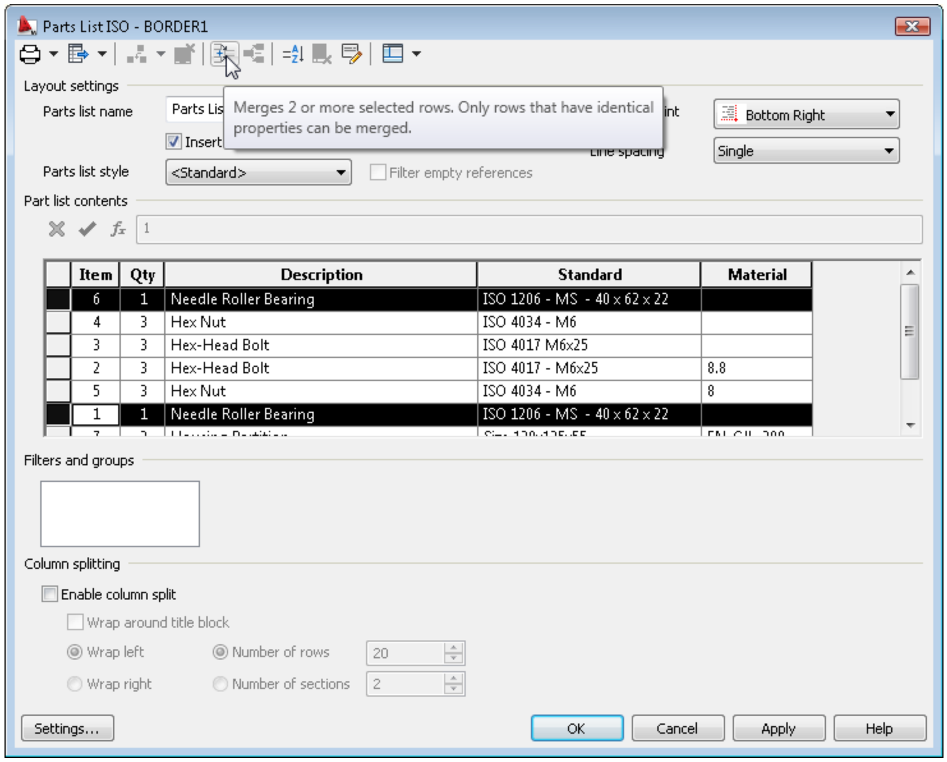
Merging and Splitting Items In Parts Lists

Use the Parts List function to merge like items that are listed repeatedly.

To merge items in a parts list

1 In the Parts List dialog box, select the repeated items -Needle Roller Bearing. Click the row heading (the button in the left most column) of item 1 and with the CTRL key pressed click the row heading of item 6.

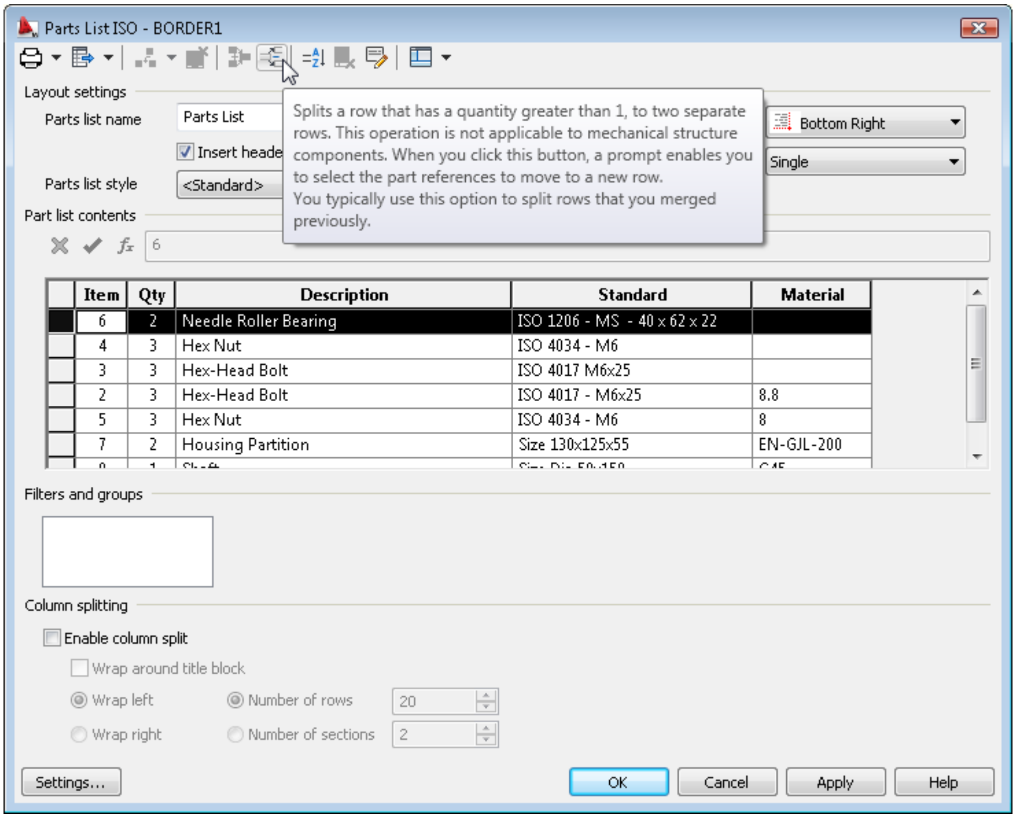
2 Click the Merge Items toolbar button.



The two rows are merged. In the Parts List dialog box, Item 6 now has a quantity of 2, and item 1 is missing.

You can select several rows to merge or split items. To merge rows, the part data in the selected rows must be the same.

3 Click Apply to display the changes in the drawing. Note how the balloon changes to reflect the same item number.



If you click the Split item button with the Needle Roller Bearing row selected, the previously merged items can be split to become two separate items once again. We, however, will not do it now, because in the next exercise we replace the two separate balloons with a single balloon that points to both Needle Roller Bearings.

Click OK to exit the Parts List dialog box.

To delete a balloon

1. Use Power Erase, and select the left balloon with the item number 1 (or 6, as the case may be).
2. Press ENTER to delete the balloon.

NOTE Deleting a balloon in the drawing doesn't delete any data. Data is lost only if you delete a part reference. You can add more than one balloon to a part reference. For example, you can create a balloon with the same item number for the same part in another view.

To add an additional leader

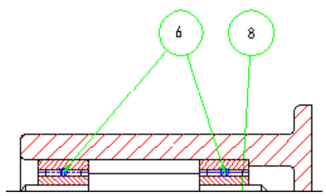
1 Select the remaining balloon 1.

2 Right-click to display the shortcut menu. Select Add Leader and respond to the prompts as follows:

`Select object to attach:` *Select the left bearing*

`Next point (or F for first point):` *Select a point inside the balloon 1*

The leader is added and your drawing should look like the following:



Save your file.

Collecting Balloons

You can collect balloons to place balloons of related parts to one leader line. For example, you can place the balloons of a screw and a nut to one common leader line.

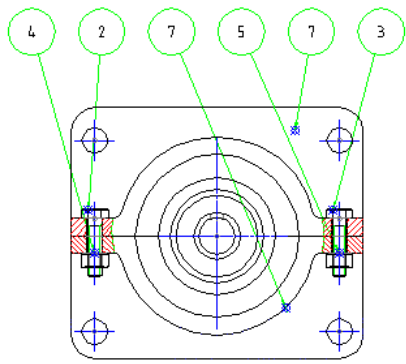
Use Zoom Window to zoom in the top view of the drawing.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤  Window.
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

To collect balloons

1 Start the AMBALLOON command.

Ribbon	Annotate tab ➤ Balloon panel ➤ Balloons. 
Menu	Annotate ➤ Balloons
Command	AMBALLOON



2 Respond to the prompt as follows:

`Select part/assembly or [auTo/autoAll/set Bom/Collect/arrow Inset/Manual/One/Renumber/rEorganize/annotation View]:`

Enter C, press ENTER

3 Continue to respond to the prompts as follows:

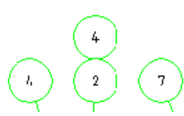
`Select pick object or balloon:` *Select the part reference of the left nut*

4 Continue to respond to the prompts as follows:

`Select pick object or balloon:` *Press* ENTER

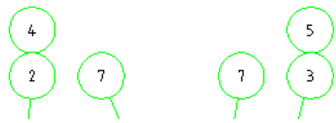
`Select balloon:` *Select balloon 2*

`Pick orientation:` *Select a vertical orientation*



5 Repeat the collect balloon command for the screw and nut on the right side.

The result should look like this:



Save your file.

Sorting and Renumbering Items In Parts Lists

You can sort a parts list for manufacturing and sort standard parts with updated item numbers.

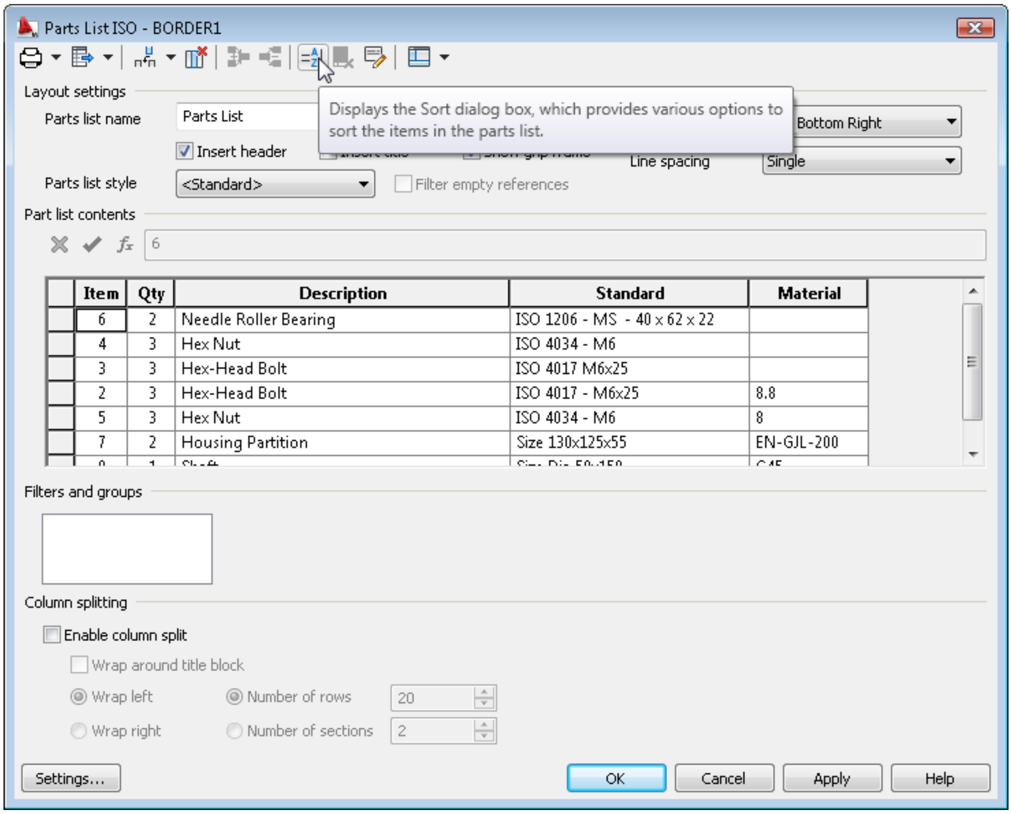
To sort a parts list

1 Zoom to the extents of the drawing.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down  ➤ Window.
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

2 Double-click the parts list to display the Parts List dialog box.

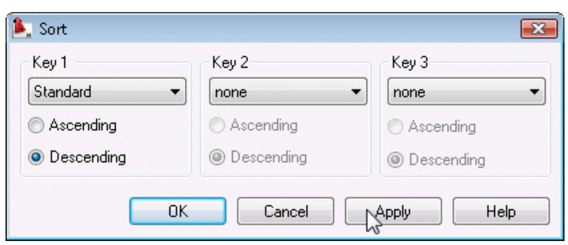
3 Choose the Sort icon.



The Sort dialog box opens.

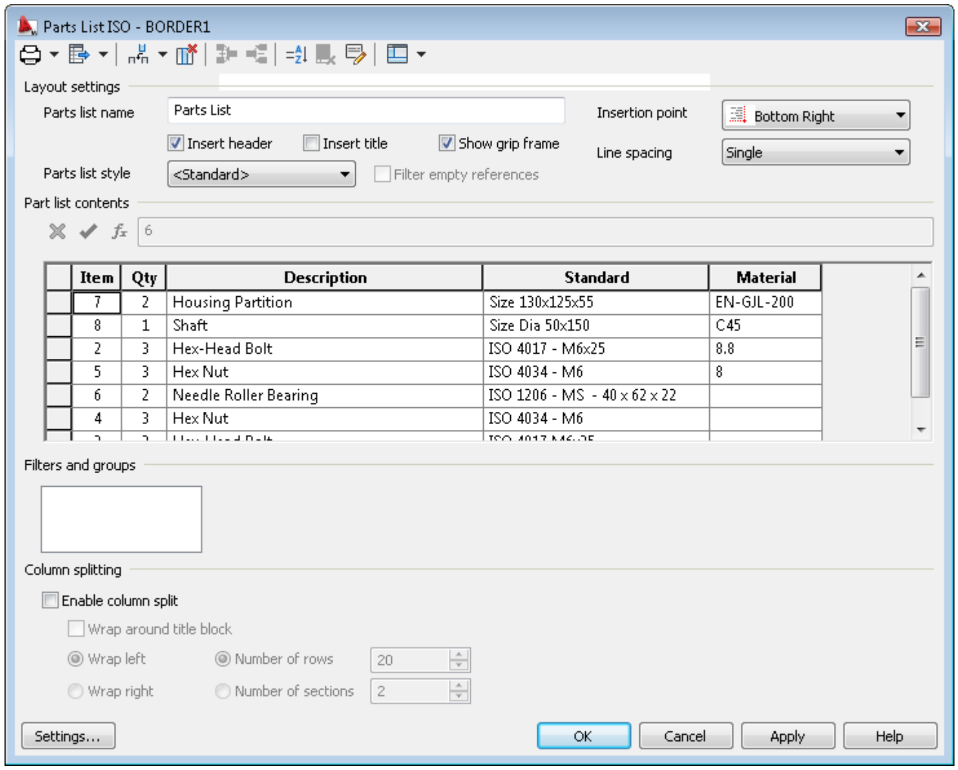
NOTE You can sort within a selection set, otherwise you are sorting all items.

4 In the Sort dialog box, specify as shown in the following image.



Click OK.

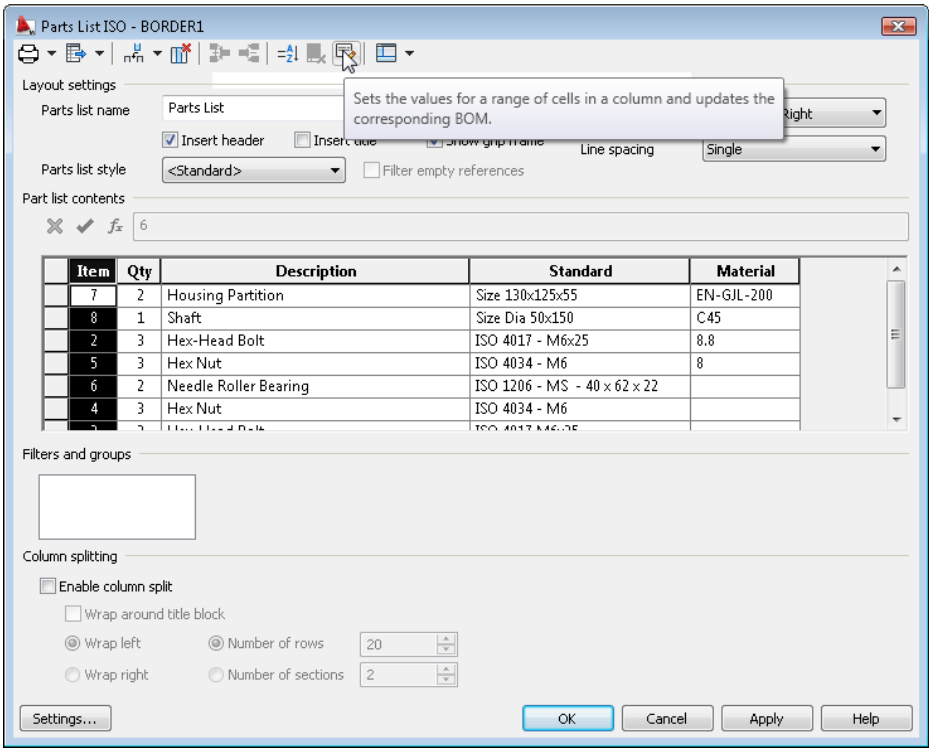
The result should look like this:



In the next step, you renumber the items.

To renumber parts list items

1 Click the column heading for the Item column.



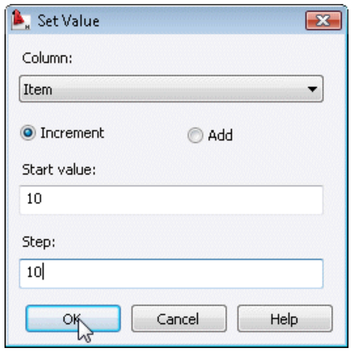
2 Click the Set values button.

3 In the Set Value dialog box, specify:

`Column:` Item

`Start value:` 10

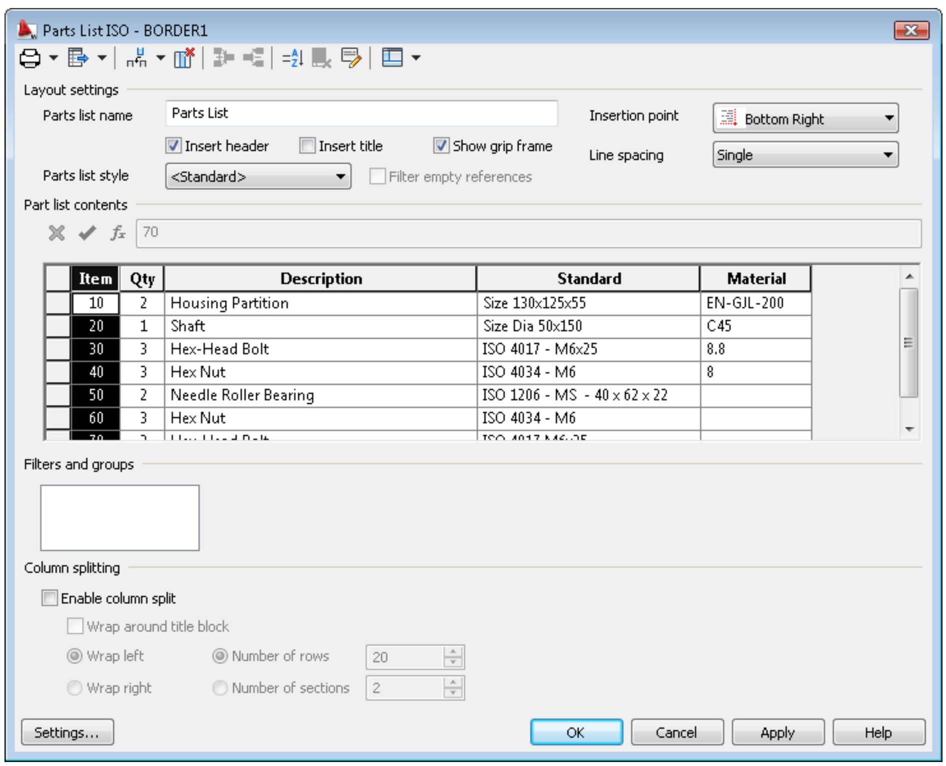
`Step:` 10



4 Click OK to return to the Parts List dialog box.

5 Choose Apply to see the results.

The result should look like the following.



6 Choose OK to return to the drawing.

Save your file.

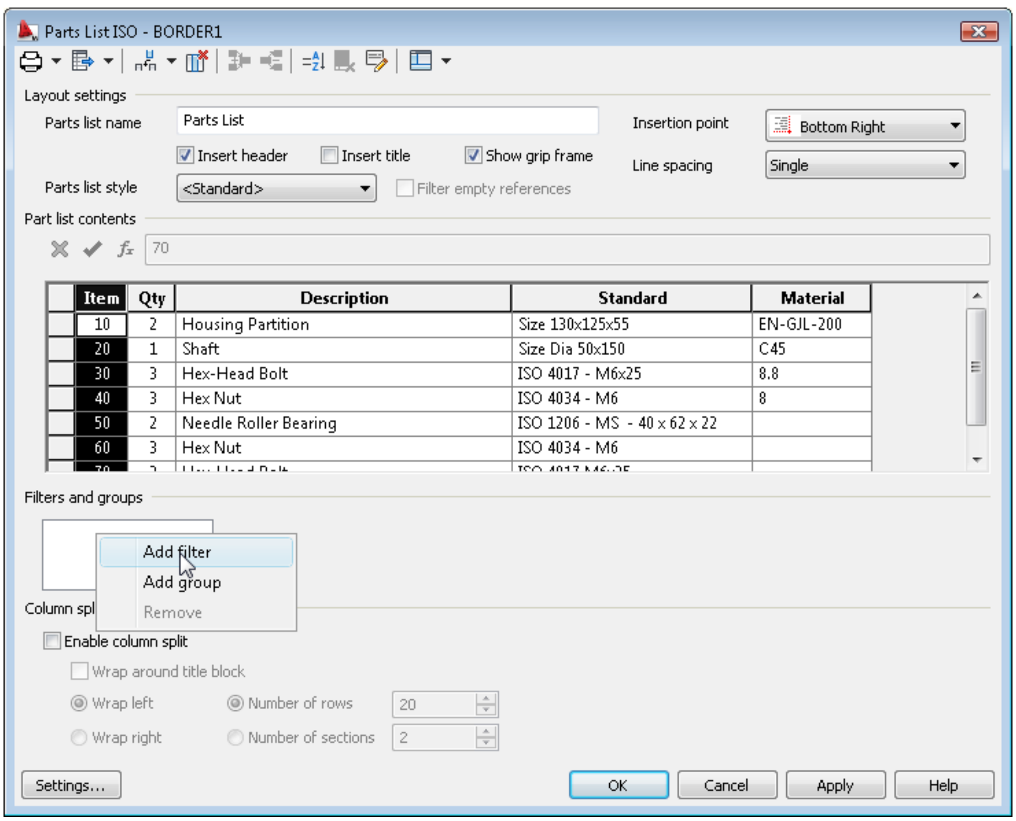
Using Filters

You can create and use one or more filters for every parts list you have inserted in the drawing.

To use filters in a parts list

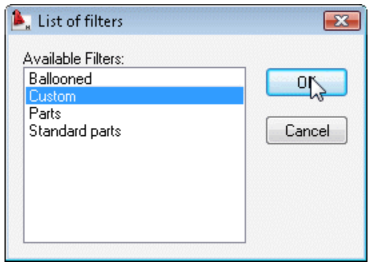
1 Double-click the parts list to display the Parts List dialog box.

2 Right-click the blank list in the Filters and groups section.



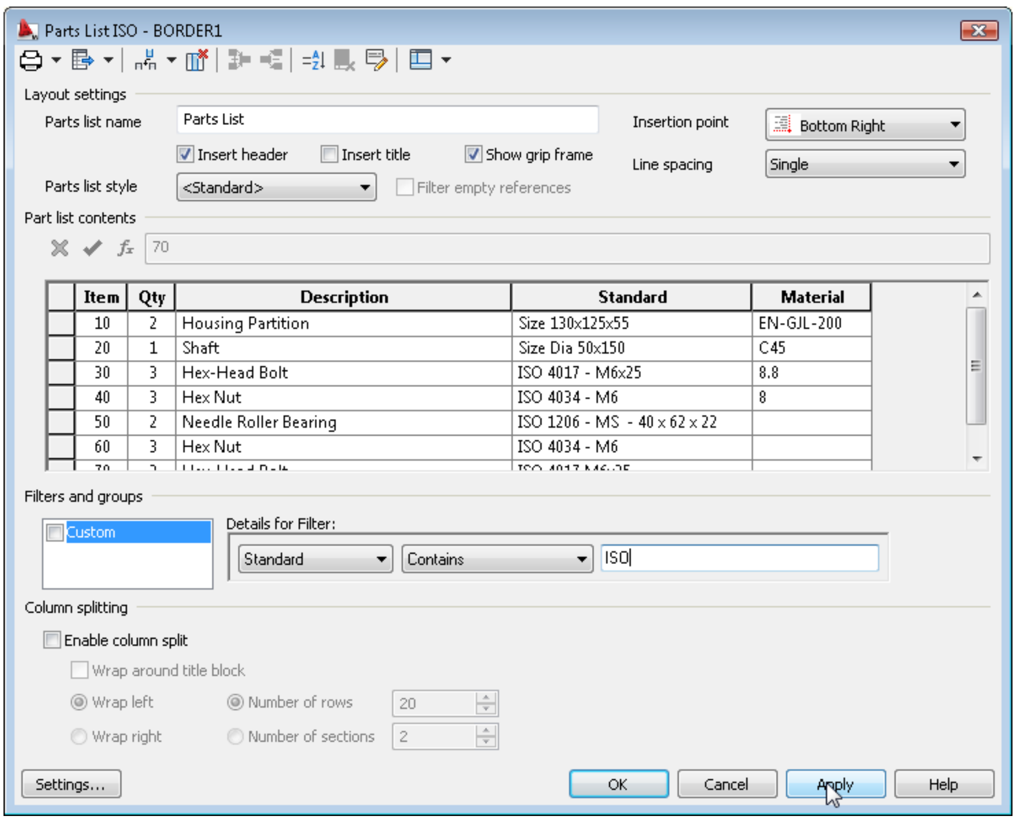
3 Select Add Filter to display the List of Filters dialog box.

4 Select Custom and click OK.



The details for this filter are displayed in the Filter and groups section of the Parts List dialog box.

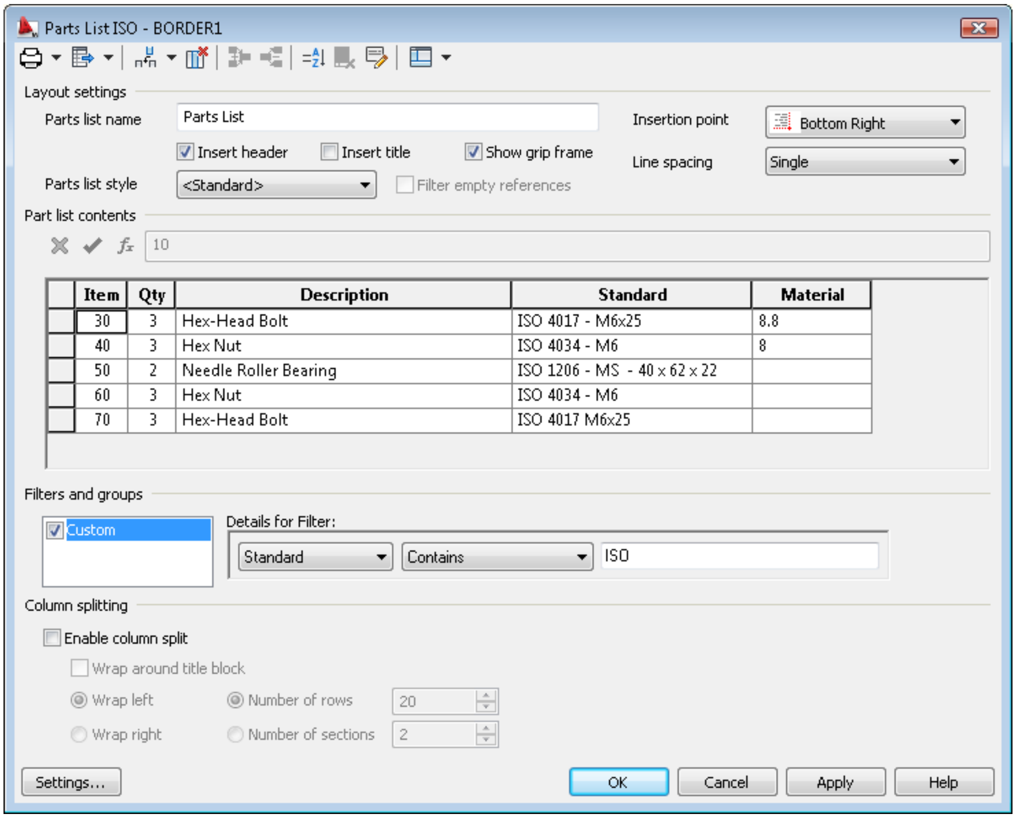
5 Set the following values to define the filter.



6 Activate the filter with the Custom check box.

7 Select the check box next to Custom and click Apply.

The Standards that contain ISO are displayed.



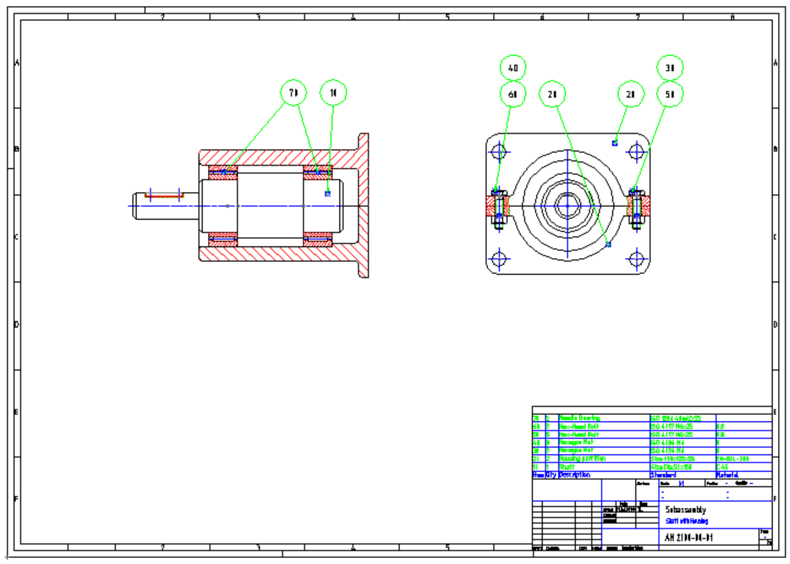
The filtered parts list is displayed in the drawing. The defined filters are saved with the parts list and can be used again later.

To print only the filtered list, choose the Print icon.

8 Deactivate the custom filter, and then click OK to close the dialog box.

The filter is used in this drawing.

The result looks like the following:



Save your file. This is the end of this tutorial chapter.

Creating Shafts with Standard Parts

11

In this tutorial, you work with the automated shaft generator and standard parts in AutoCAD® Mechanical to create and edit a shaft, and insert bearings. The standard parts you use are automatically structured in the mechanical browser.

Key Terms

Term	Definition
bearing calculation	Calculates limiting value, dynamic and static load rating, dynamic and static equivalent load, and fatigue life in revolutions and hours.
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 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.
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.

Term	Definition
gear	Any of several arrangements, especially of toothed wheels in a machine, which allow power to be passed from one part to another to control the power, speed, or direction of movement.
radius reflection line	Thin line that represents the radius in the side or top view.
shaft break	Interruption of a shaft. A shaft can be interrupted at a point, and the shaft break symbols are inserted in a suitable size.
shaft generator	Tool to draw rotationally symmetrical parts. A shaft is usually created from left to right using different sections. These sections are positioned automatically one after the other. Additionally, any shaft section can be inserted, deleted, or edited.

Creating Shafts

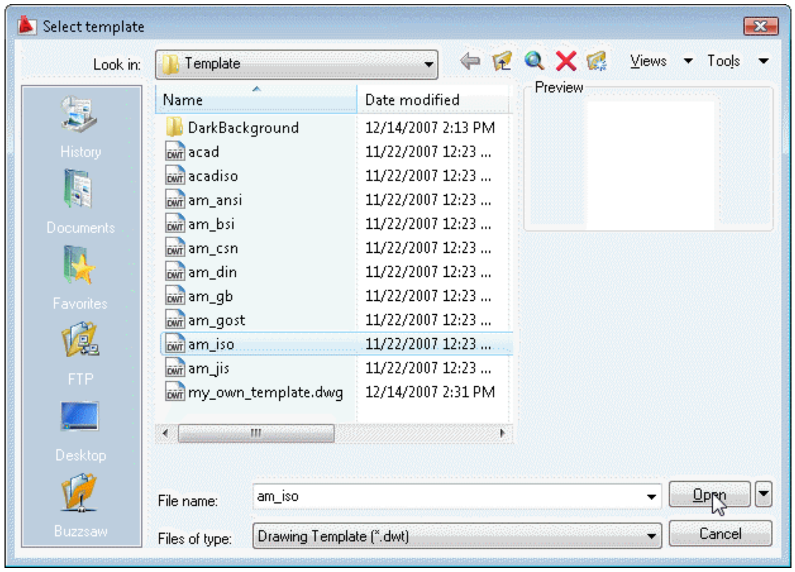
In this section, you use the shaft generator to create a shaft with standard parts. You also perform a bearing calculation.

To open a template

1 Create a new drawing.

Ribbon	➤ New ➤ Drawing
Menu	File ➤ New
Command	NEW

2 In the Select template dialog box, click the template *am_iso.dwt*, and then click Open.



This creates a new drawing based on the am_iso template. Use Save As to save the drawing file with an appropriate name.

NOTE The ISO standard part standard has to be installed for this tutorial exercise.

Ensure that mechanical structure is enabled

To enable mechanical structure

1 Click the STRUCT status bar button and latch it down to enable mechanical structure.

2 If the mechanical browser is not visible, on the command line, enter *AMBROWSER*.

3 When prompted, enter *ON*.

Configuring Snap Options

Configure the snap options.

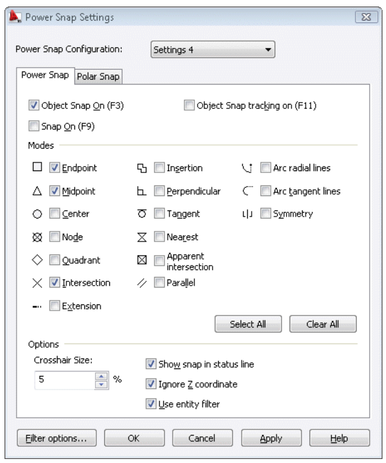
To configure the snap options

1 Start the Power Snap Settings command.

Ribbon	None.
Menu	Object Snap Cursor Menu ➤ Power Snap Settings
Command	AMPOWERSNAP

2 In the Power Snap Settings dialog box, in the Power Snap Configuration list, select Settings 4 and specify:

`Snap Modes`: Endpoint, Midpoint, Intersection



Click OK

Save your file.

Configuring Shaft Generators

In the next steps, you start and configure the shaft generator.

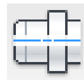
NOTE Turn off Dynamic Input

Click on the Dynamic Input status bar icon and ensure that it is not lighted up.

To start and configure the shaft generator

1 Start the Shaft Generator command.

Ribbon Content tab ➤ Shaft panel ➤ Shaft Generator.



Menu Content ➤ Shaft Generator...

Command AMSHAFT2D

2 Respond to the prompts as follows:

`Enter shaft component name <Shaft1>:` *Press* ENTER

`Specify starting point or select center line:`

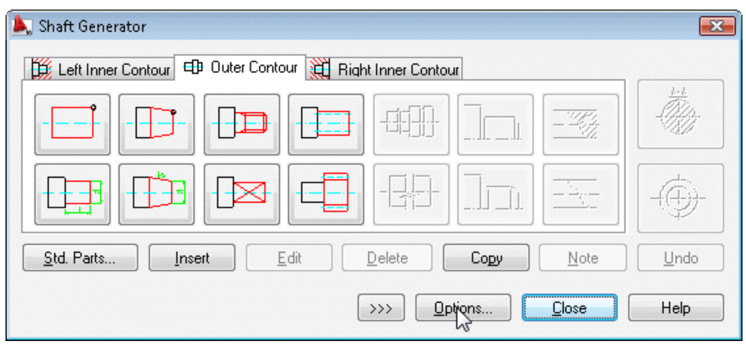
Enter 150,150, press ENTER

`Specify centerline endpoint:` *Enter 240,150, press* ENTER

NOTE The start and endpoints of the centerline are only important in determining the direction. The length of the centerline is automatically adapted to the length of the shaft.

---

3 In the Shaft Generator dialog box, click Options.



4 In the Shaft Generator - Configuration dialog box, specify:

`For Segment inserted:` Insert

`Stationary Shaft End:` Left

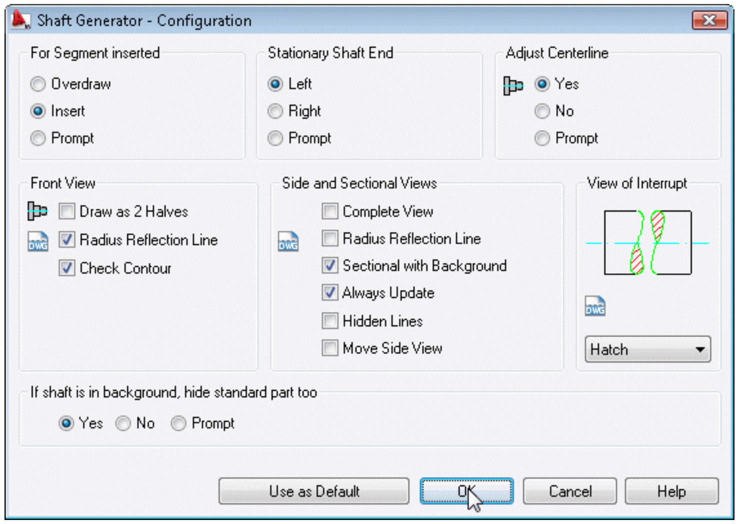
`Adjust Centerline:` Yes

`Front View:` Radius Reflection Line, Check contour

`Side and Sectional Views:` Sectional with Background, Always update

`View of Interrupt:` Hatch

`If shaft is in background, hide standard part too:` Yes



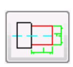
Click OK.

You return to the Shaft Generator dialog box.

Creating Cylindrical Shaft Sections and Gears

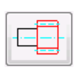
The shaft generator is configured. Now you want to generate the first shaft segments. Verify that the Outer Contour tab is selected.

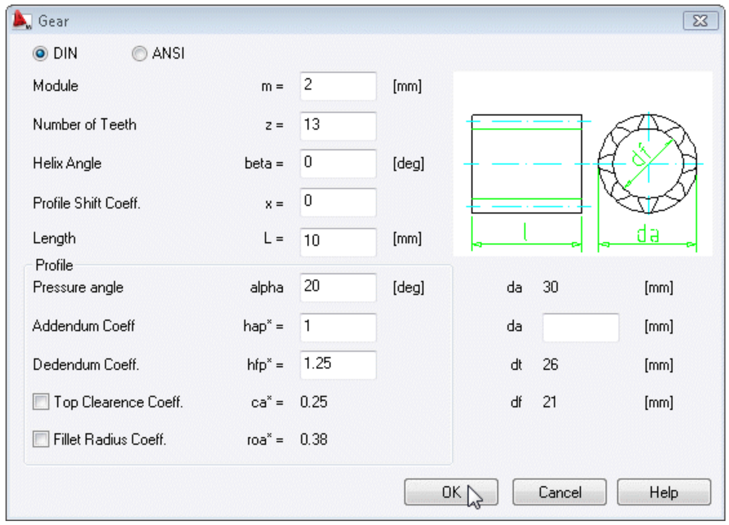
To create shaft segments

1  Click the lower cylinder button to define a cylinder section, and respond to the prompts as follows:

`Specify length <50>:` *Enter 12, press* ENTER

`Specify diameter <40>:` *Enter 20, press* ENTER

2  Click the gear button, and then enter the values for module, number of teeth, and length as shown in the following figure:



NOTE Here, the DIN standard requires that you indicate the module. The ANSI standard requires the Diametral Pitch 1/module. You can switch between these two representations using the DIN and ANSI options.

3 Close the Shaft Generator dialog box.

In the mechanical structure browser, the shaft is added as a component. Add an assembly to structure the shaft components you create in this exercise.

To add an assembly to the mechanical browser

1 In the mechanical browser, right click the file name node (the root node) and click New ➤ Component.

2 Respond to the prompts:

`Enter new component name <COMP1>:` *Enter shaftassembly, press* ENTER

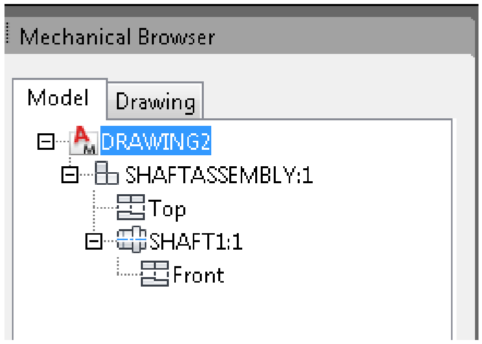
`Enter new view name <Top>:` *Enter front, press* ENTER

`Select objects for new component view:` *Select the shaft with a window*

`Select objects for new component view:` *Press* ENTER

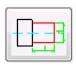
`Specify base point:` *Specify a point at the upper left of the shaft*

The shaft assembly is listed at the top of the browser, and the existing shaft components are listed within the assembly. As you add more components to the shaft, they are automatically structured in the assembly.



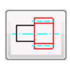
Return to the shaft generator.

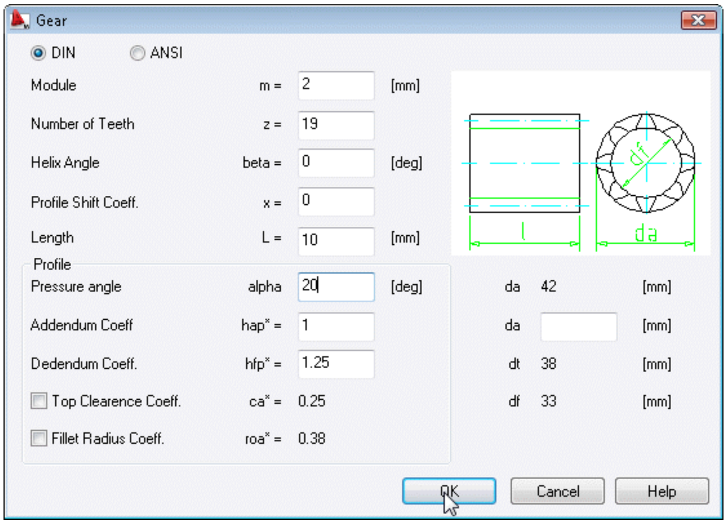
Double-click the left shaft segment in the drawing and then press ESC.

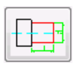
3  Click the lower cylinder button to define a further cylinder section and respond to the prompts as follows:

`Specify length <10>:` *Enter 5, press* ENTER

`Specify diameter <20>:` *Enter 20, press* ENTER

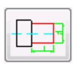
4  Click the gear button, and then enter the values for module, number of teeth, and length as shown in the following figure:



5  Click the lower cylinder button to define another cylinder section, and then respond to the prompts as follows:

`Specify length <10>:` *Enter 4, press* ENTER

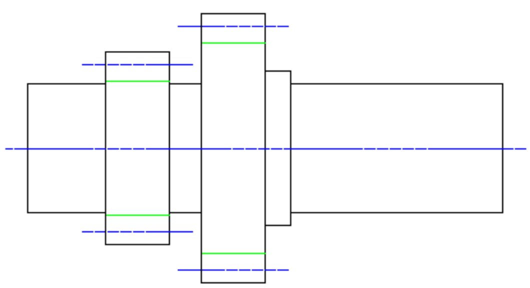
`Specify diameter <20>:` *Enter 24, press* ENTER

6  Click the lower cylinder button to define another cylinder section, and then respond to the prompts as follows:

`Specify length <4>:` *Enter 33, press* ENTER

`Specify diameter <24>:` *Enter 20, press* ENTER

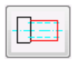
The first five sections of the shaft are created, as represented in the following figure:



Inserting Spline Profiles

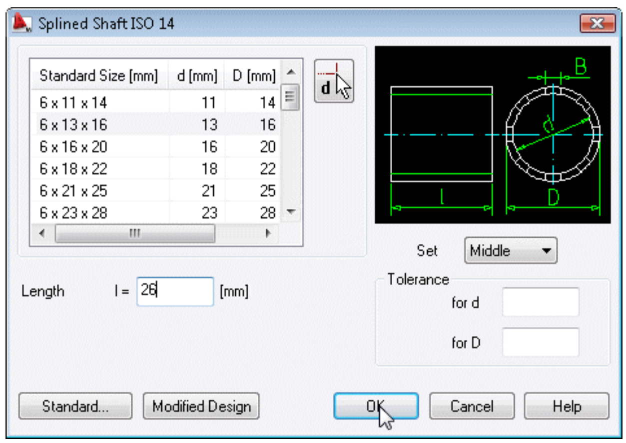
Add a spline profile to the shaft.

To create a profiled segment

1  Click the Profile button.

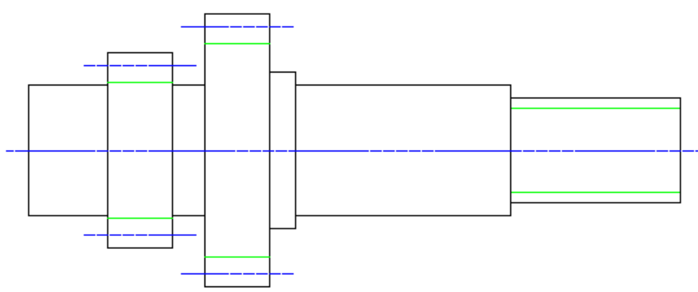
2 In the Profile dialog box, click ISO 14 in the Details panel.

3 In the Splined Shaft ISO 14 dialog box, select the standard size 6 x 13 x 16 and enter a length of *26*.



Click OK.

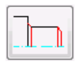
You created another section of the shaft, as shown in the following figure:



Inserting Chamfers and Fillets

Apply a chamfer and a fillet to the shaft.

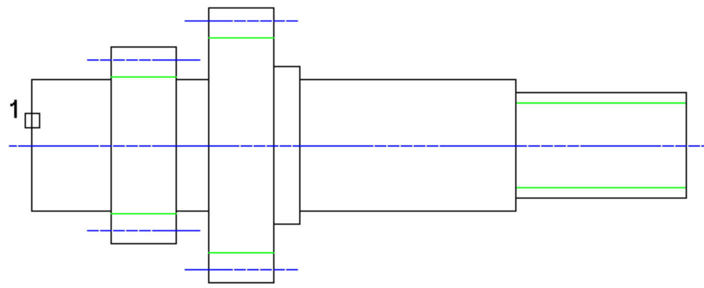
To apply a chamfer and a fillet

1.  Click the Chamfer button to apply a chamfer to a shaft section, and then respond to the prompts as follows:

   `Select object:` *Select the left most cylinder section (1)*

   `Specify length (max. 12) <2.5>:` *Enter 2, press* ENTER

   `Specify angle (0-79) or [Distance] <45>:` *Enter 45, press* ENTER

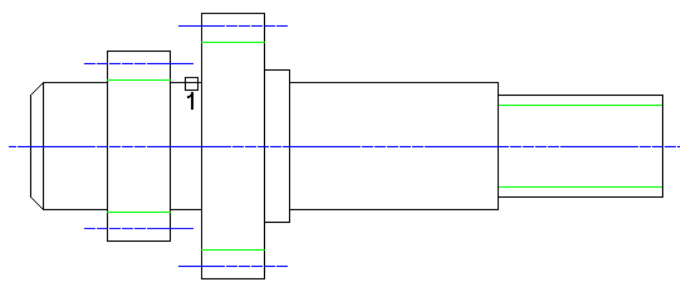
   

2.  Click the Fillet button to apply a fillet to a shaft section, and then respond to the prompts as follows:

   `Select object:`

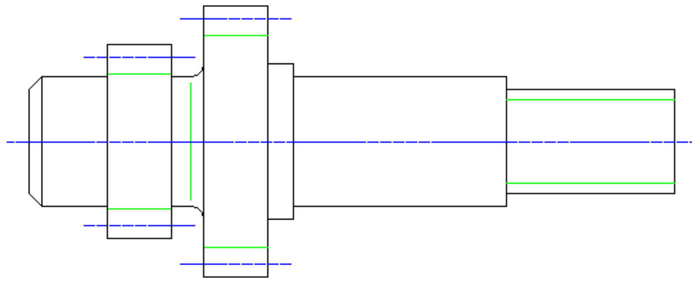
   Select the cylinder section between the two gears near the second gear (1)

   `Enter radius (max. 10.00) <2.50>:` *Enter 2, press* ENTER

NOTE The fillet is applied to the edge of the selected section that is closer to the selected point.

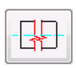
The shaft looks like the following figure:



Inserting Shaft Breaks

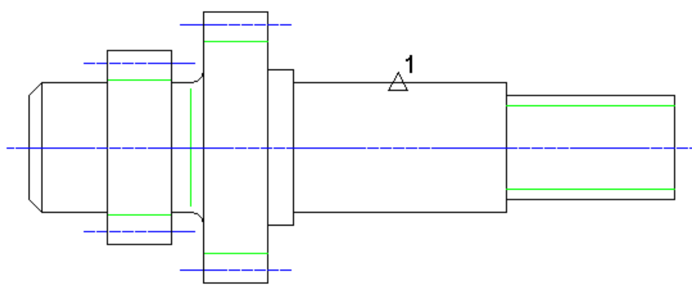
Insert a shaft break in the drawing.

To insert a shaft break

■  Click the Break button to insert a shaft break, and then respond to the prompts as follows:

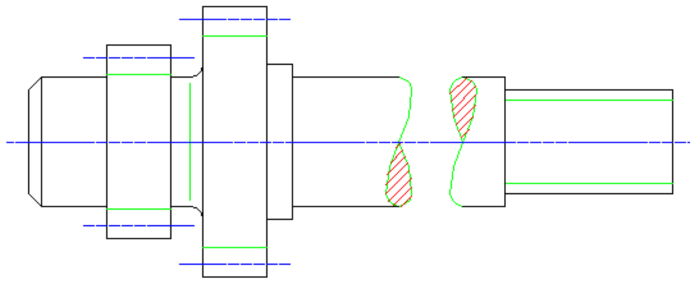
`Specify point:` *Select the midpoint of the cylindrical section (1)*

`Specify length (min. 4.00) <6>:` *Enter 10, press* ENTER



NOTE You can insert the break to the left if you enter a negative value.

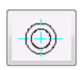
The shaft break is inserted.



Creating Side Views of Shafts

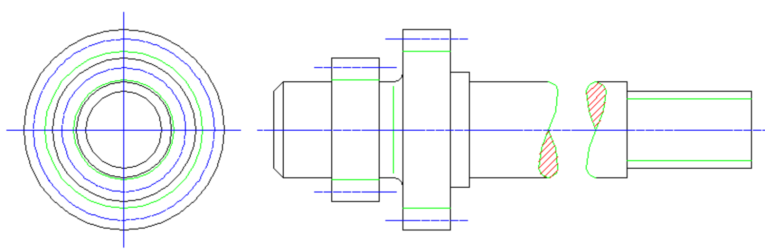
Insert a side view of the shaft.

To insert a side view

1.  Click the Side view button.
2. In the Side view from dialog box, select Right. Click OK.
3. Respond to the prompt as follows:

   `Specify insertion point:` *Press* ENTER

   The right side view is inserted at the proposed position.

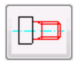
   

   In the mechanical browser, the new right side view is listed within the shaft component along with the existing front view. The right side view includes its hide situations.

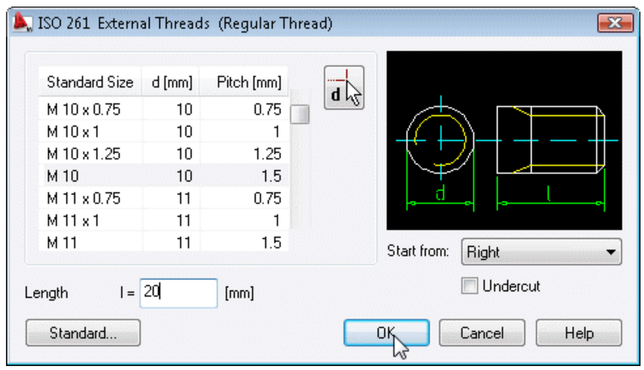
Inserting Threads on Shafts

Add a thread to the shaft.

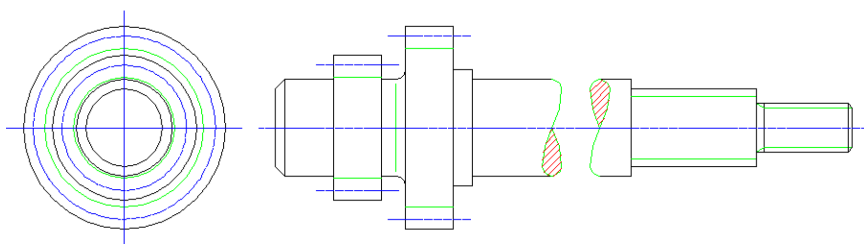
To insert a thread on a shaft

1  Click the Thread button to insert a thread, and then select ISO 261 External from the Details panel.

2 In the ISO 261 ExternalThreads (Regular Thread) dialog box, select M10 and enter a length of *20*. Click OK.



The thread is added to the shaft, which looks like this:



NOTE If Always Update is unchecked in Options, AM:Shaft tab, you are prompted to update associated views when you close the Shaft Generator.

Editing Shafts and Inserting Sections

Edit an existing shaft section and insert a new section. You use the Edit button in the shaft generator to turn on AMPOWEREDIT.

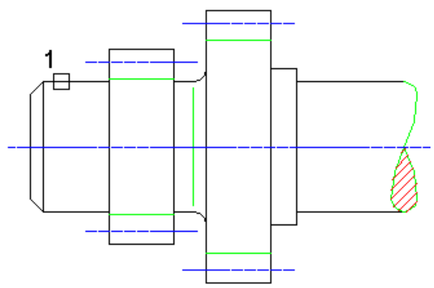
To edit and insert a shaft section

1 Click the Edit button, and then respond to the prompts as follows:

`Select object:` *Select the first cylindrical section (1)*

`Specify length <12>:` *Press* ENTER

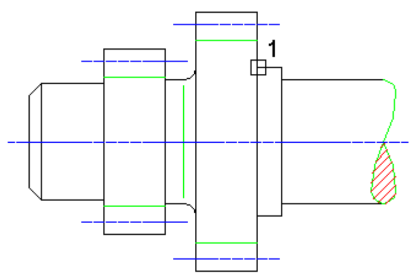
`Specify diameter <20>:` *Enter 18, press* ENTER

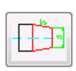


The diameter is changed to 18 while the length remains 12.

2 Click the Insert button, and then respond to the prompt as follows:

`Specify point:` *Select a point after the second gear (1)*



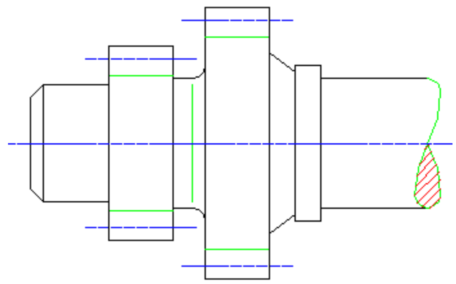
3  Click the Slope button, and then respond to the prompts as follows:

`Specify length or [Dialog] <20>:` *Enter 4, press* ENTER

`Specify diameter at starting point <24>:` *Enter 28, press* ENTER

`Specify diameter at endpoint or [Slope/Angle] <20>:`

Enter 22, press ENTER



Replacing Shaft Sections

The previously inserted slope needs to be deleted again.

To replace a shaft section

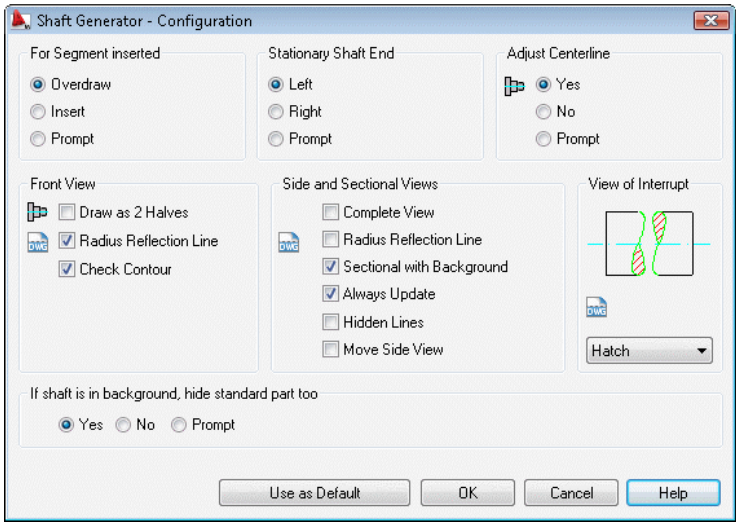
1. Click the Undo button.

   The previous slope insertion is undone.

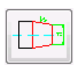
   Replace an existing shaft section. To do this, change the settings in the configuration.

2. Click the Options button to start the shaft generator configuration, and then specify:

   `For Segment inserted:` Overdraw

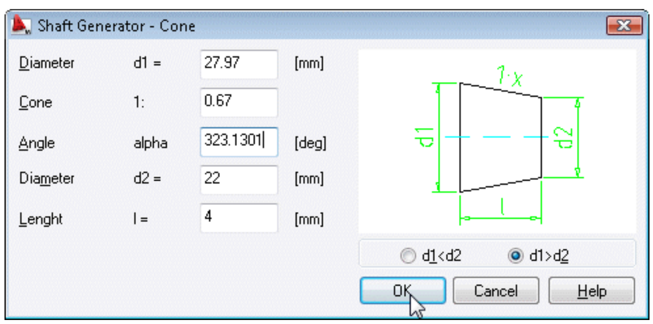


Click OK.

3  Click the Slope button, and then respond to the prompt as follows:

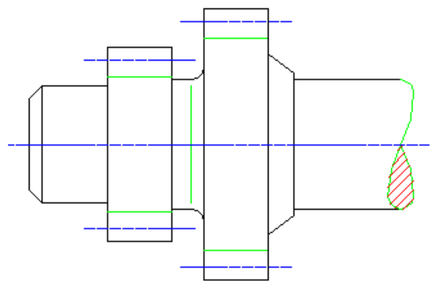
`Specify length or [Dialog] <4>:` *Enter D, press* ENTER

4 In the Shaft Generator - Cone dialog box, specify the following settings.



Click OK.

The slope replaces the cylindrical shaft section.



Inserting Bearings

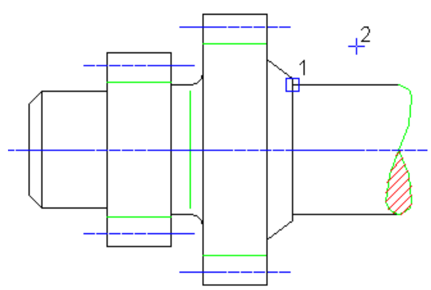
Insert a bearing and perform a bearing calculation.

To insert a bearing

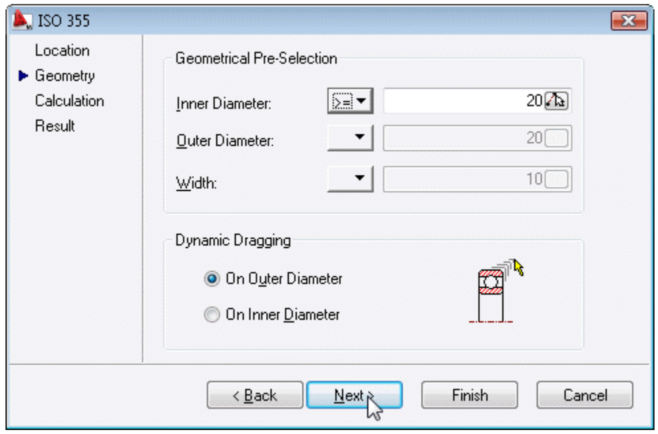
1 Click the Standard Parts button, and then select Roller Bearings ➤ Radial ➤ ISO 355. Respond to the prompts as follows:

`Specify insertion point on shaft contour:` *Specify insertion point (1)*

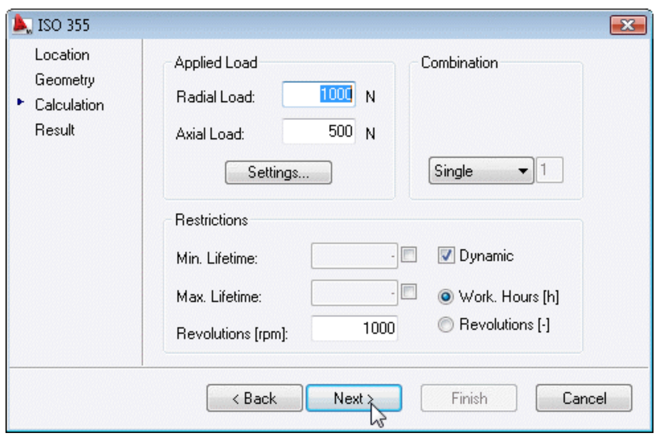
`Direction to [Left]:` *Select a point to the right (2)*



2 In the ISO 355 dialog box, click Next.

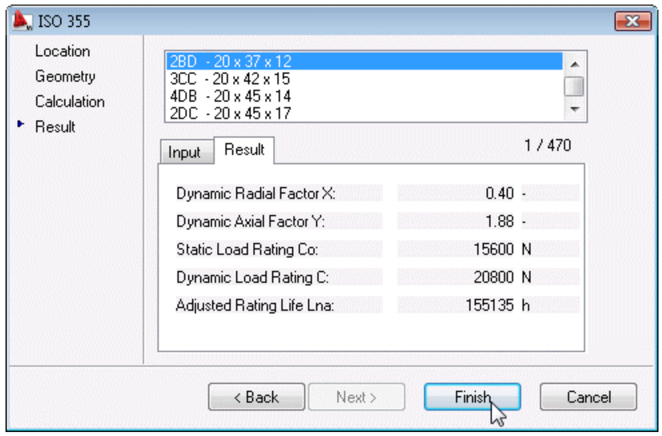


3 Specify the loads, and activate Work Hours as shown in the following.



Click Next.

4 In the ISO 355 dialog box, select the bearing 2BD - 20 x 37 x 12, and then click Finish.

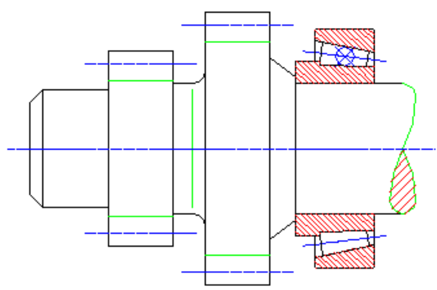


You can drag the cursor to see all available bearing sizes.

5 Drag to the size 2BD - 20 x 37 x 12, and then press ENTER.

6 In the Create Hide Situation dialog box, click OK.

The bearing is inserted.



7 Close the Shaft Generator dialog box.

In the mechanical structure browser, the roller bearing component is added to the assembly.

Save your file. This is the end of this tutorial chapter.

Engineering Calculations

The tutorials in this section teach you how to calculate moments of inertia and deflection lines, create and calculate chains, springs and cams. The drawing files for each lesson can be found in the *Acadm/tutorial/* folder of the AutoCAD® Mechanical installation folder. These drawing files provide design elements that help you understand several AutoCAD Mechanical concepts.

Calculating Shafts

12

In this tutorial, you use the shaft generator in AutoCAD® Mechanical to perform a calculation on an existing shaft, and apply various loads to a supported shaft. Then you insert the results into a drawing.

Key Terms

Term	Definition
deflection line	A curve representing the vertical displacement of different points along the member subjected to a load.
bending moment	The moment of all forces that act on the member to the left of a section (a point along the member where bending moment needs to be calculated) taken about the horizontal axis of the section.
fatigue factor	Safety to endurance or fracture under repetitive cycles of loads.
fixed support	A support that prevents translation as well as rotation about all axes.
gear	Any of several arrangements in a machine, especially toothed wheels, that allow power to be passed from one part to another to control the power, speed, or the direction of movement.
load	The forces and moments that act on a part.
movable support	A support that prevents translation as well as rotation about all axes.
notch	A change of cross section, such as an undercut, groove, hole or shoulder. A notch leads to higher stress in the part. The flux of the stress is interrupted or redirected.

Term	Definition
point force	A force that is concentrated on a point.
strength	A summary term for all forces and moments, thus loads and stress, which act on a part.
stress	Force or pressure on a part. Stress is the force per unit area.
yield point	Safety to the stress beyond which the material exhibits permanent deformation.

Calculating Shafts

With AutoCAD Mechanical, you can perform a shaft calculation using a contour created with the Shaft Generator, or any other symmetric shaft contour. The function provides a static calculation, which is important for the design of the shaft and the bearing load.

In this tutorial, you calculate a gearbox shaft. The general way to calculate an existing shaft is to define the contour and insert forces and supports. The routine calculates all necessary values and draws the respective graphs for moment and deflection.

Mechanical structure does not impact this engineering structure routine. You can calculate shafts with mechanical structure enabled or disabled.

Load the initial drawing.

To open a file

1 Open the file *tut_shafts.dwg* in the Tutorial folder.

Ribbon	 ➤ Open ➤ Drawing
Menu	File ➤ Open
Command	OPEN

NOTE The path to the folder containing tutorial files is;

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains a shaft in front and side view.

2 Zoom in to the shaft.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

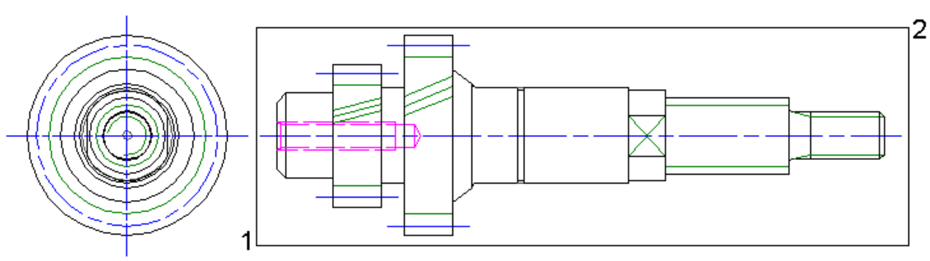
3 Respond to the prompts as follows:

```
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real
time>:
```

Enter W, press ENTER

`Specify first corner:` *Specify the first corner point (1)*

`Specify opposite corner:` *Specify the second corner point (2)*



Save your file under a different name or to a different directory to preserve the original tutorial file.

Creating Shaft Contours

Before you can perform any calculations on a shaft, you have to create the shaft contour.

To create a shaft contour

1 Start the Shaft Calculator.

2 **Ribbon** Content tab ➤ Calculation panel drop-down ➤ Shaft Calculation. 

Menu Content ➤ Calculations ➤ Shaft Calculation...

Command AMSHAFTCALC

3 Respond to the prompts as follows:

Select contour or [Create contour/Strength] <Create>:

Enter C, press ENTER

Select objects: *Select the complete shaft*

Select objects: *Press* ENTER

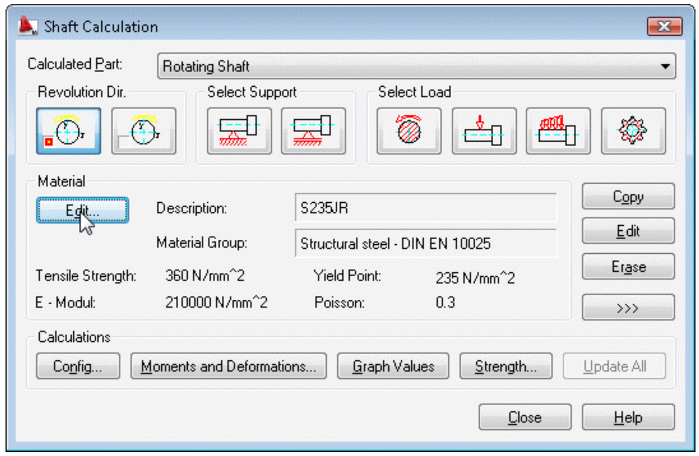
4 In the AutoCAD Question dialog box, click Yes.

5 Respond to the prompts as follows:

Specify contour position: *Press* ENTER

NOTE The calculation routine recognizes hollow shafts and uses the contour for the calculation.

After you create the shaft contour, the Shaft Calculation dialog box is displayed so that you can select the boundary conditions, the material, and the representation of the calculation results.



Specifying Material

You specify the material by selecting it from a table containing the most commonly used materials. You can also to enter the characteristics for other materials using the option Edit.

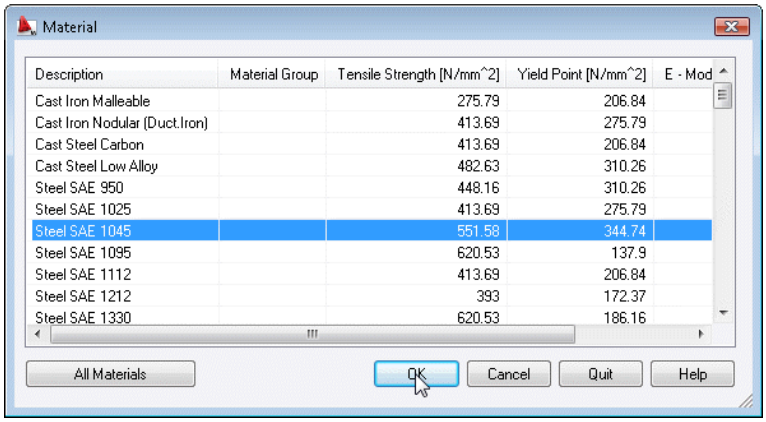
To specify a material

1. In Material, click Edit.

   The Material Properties dialog box is displayed.

   TIP There are two Edit buttons in the dialog box. Ensure that you click the Edit button in the Materials section.

2. In the Material Properties dialog box, click Table.
3. In the Details panel of the Material Dialog box, click ANSI Material.
4. In the Material dialog box, select the material Steel SAE 1045 from the table.



Click OK.

NOTE If the ANSI standard is not installed on your system, you can select a different standard, but the results may differ from the results in this tutorial. For example, if you select DIN, you can select a similar material, like E335, to achieve similar results.

NOTE Some material properties are not complete. In this case, you have to complete them to obtain calculation results.

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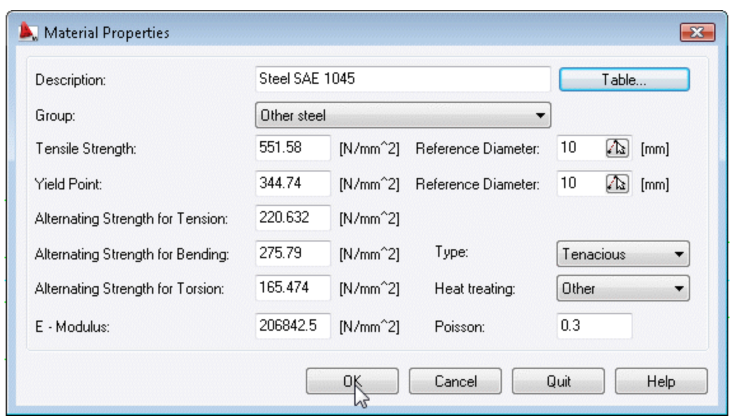
5 In the Material Properties dialog box, complete the ANSI material properties, if necessary.

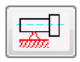


Click OK.

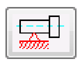
Placing Shaft Supports

Specify the shaft supports.

To place a support

1  In the Shaft Calculation dialog box, select the Movable Support icon, and then respond to the prompt as follows:

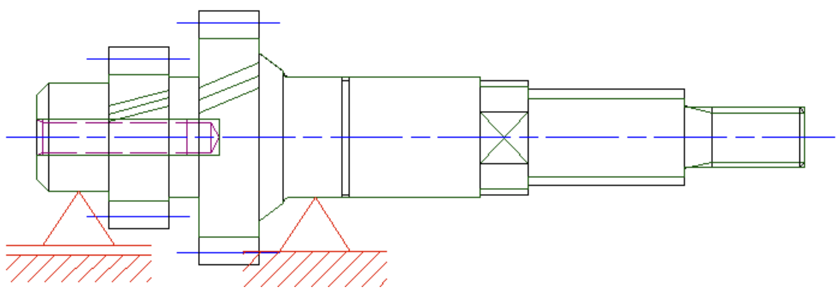
`Specify insertion point:` *Select the midpoint of the left most shaft section*

2  Select the Fixed Support icon, and then respond to the prompt as follows:

`Specify insertion point:`

Select the midpoint of the third cylindrical shaft section

The shaft supports are specified, and the result looks like this:



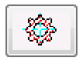
Specifying Loads on Shafts

Specify the effective loads. AutoCAD Mechanical uses geometry from the drawing for load calculations.

The loads depend on the Calculated Part setting. There are three possibilities: Rotating Shaft, Rotating Axle, and Not rotating Axle. Shafts transfer torque and rotating axles results in different stress values than static axles results.

To specify a load

1 From the Calculated Part drop-down list, click Rotating Shaft.

2  Click the Gear icon, and then respond to the prompt as follows:

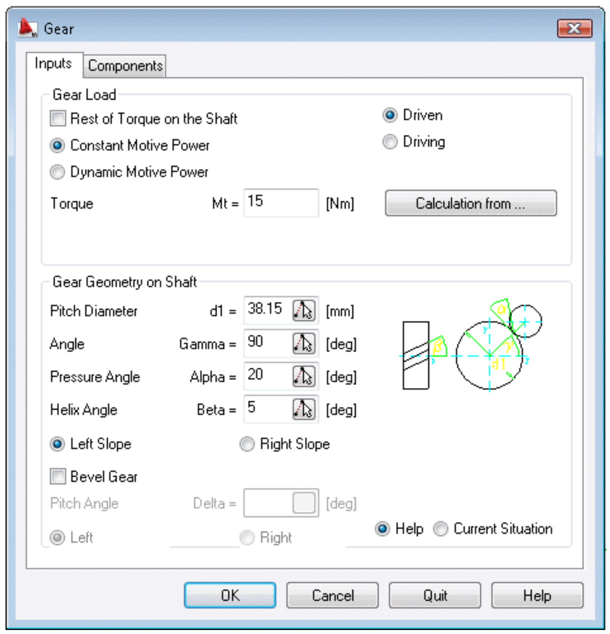
`Specify insertion point:`

Select the midpoint of the second gear from the left

3 In the Gear dialog box, Inputs tab, specify:

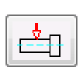
`Gear Load: Constant Motive Power, Driven`

`Torque:` *15*



Click OK.

NOTE The Components tab displays the force components. Changes in one tab are automatically reflected in the other tab.

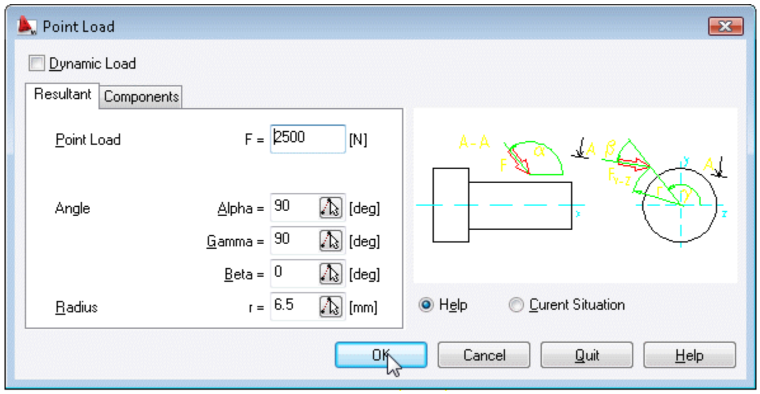
4  Click the Point Load icon, and then respond to the prompts as follows:

`Specify insertion point:` *Select the midpoint of the profile section*

`Specify rotation angle:` *Press* ENTER

5 In the Point Load dialog box, Resultant tab, specify:

`Point Load:` 2500



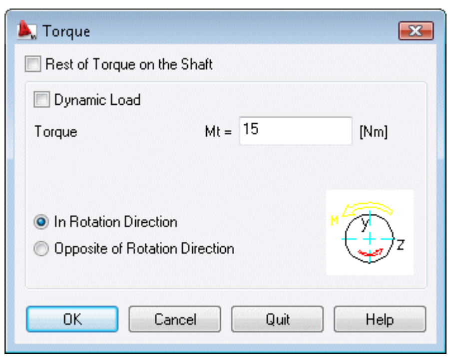
Click OK.

6  Click the Torque icon, and then respond to the prompt as follows:

`Specify insertion point:` *Select the midpoint of the profile section*

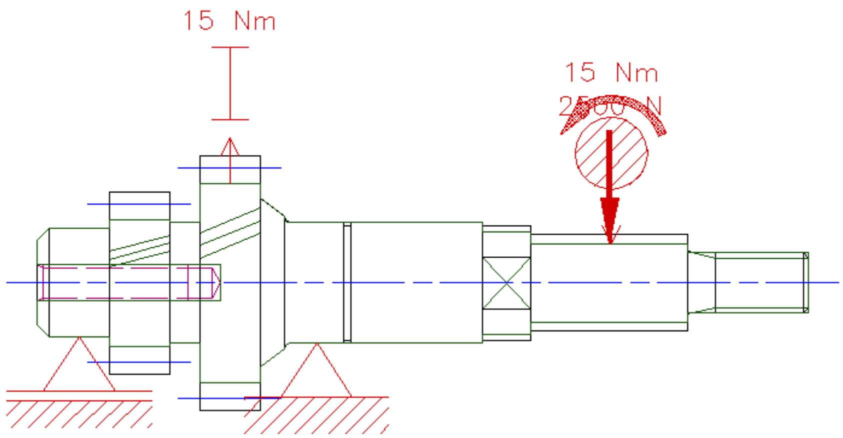
7 In the Torque dialog box, specify:

`Torque: Mt=:` *15*



Click OK.

The loads are specified, and the result looks like this:



All boundary conditions necessary for a shaft calculation are specified.

Calculating and Inserting Results

Perform a calculation of the moments and deformations, and insert the results in your drawing.

To perform a shaft calculation

1. In the Shaft Calculation dialog box, click the Moments and Deformations button.
2. In the Select Graph dialog box, specify:

   `Bend:` Bending Moment in Y - Axis, Deflection in Y - Axis

   `Torsion:` Torsion Moment in X - Direction

   `Stresses:` Result Bending Stress

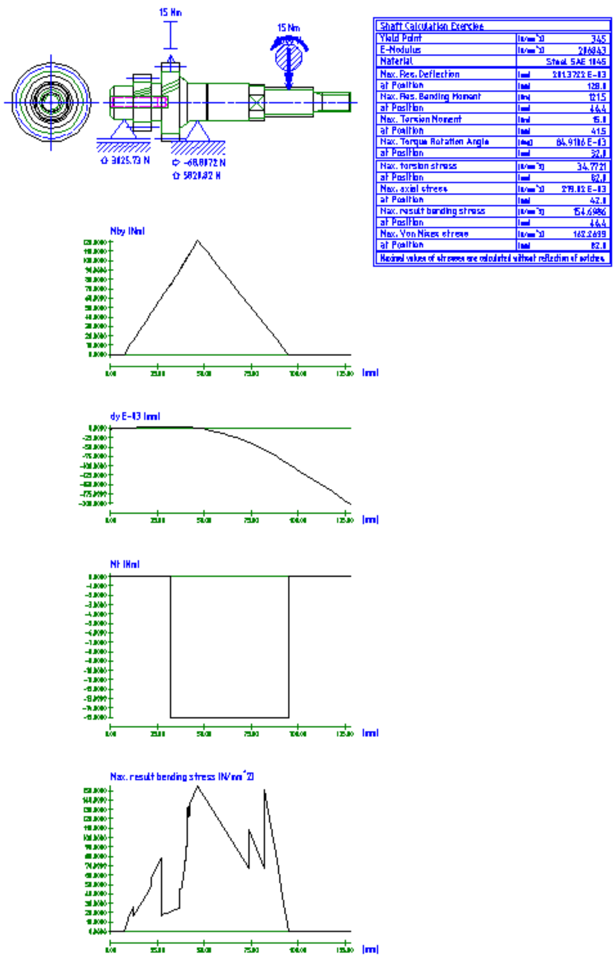
   `Table Title:` Shaft Calculation Exercise
3. Click OK, and then respond to the prompts as follows:

   `Specify insertion point:`

   Select an appropriate point to the right of the shaft

   The result block and the deflection and torsion moment graphs are inserted.
4. Close the Shaft Calculation dialog box.

   Your drawing looks like this:



The result block provides the most important information about your calculated shaft, such as the maximum stress deflection and moment values.

Calculated Values		
Yield Point	[N/mm^2]	345
E-Modulus	[N/mm^2]	206843
Material		Steel SAE 1045
Max. Res. Deflection	[mm]	201.3722 E-03
at Position	[mm]	128.0
Max. Res. Bending Moment	[Nm]	121.5
at Position	[mm]	46.4
Max. Torsion Moment	[Nm]	15.0
at Position	[mm]	41.5
Max. Torque Rotation Angle	[deg]	84.9106 E-03
at Position	[mm]	32.0
Max. torsion stress	[N/mm^2]	34.7721
at Position	[mm]	82.0
Max. axial stress	[N/mm^2]	219.02 E-03
at Position	[mm]	43.0
Max. result bending stress	[N/mm^2]	154.6986
at Position	[mm]	46.4
Max. Von Mises stress	[N/mm^2]	162.2699
at Position	[mm]	82.0
Maximal values of stresses are calculated without reflection of notches.		

5 Close the Shaft Calculation dialog box.
Save your file.

Calculating Strengths of Shafts

Check the strength at a critical place of the shaft, such as at a notch.

To calculate the strength at a notch

1 Restart the Shaft Calculation.

Ribbon	Content tab ➤ Calculation panel drop-down ➤ Shaft Calculation. 
Menu	Content ➤ Calculations ➤ Shaft Calculation...
Command	AMSHAFTCALC

2 Respond to the prompt as follows:

Select contour or [Create contour/Strength] <Create>:

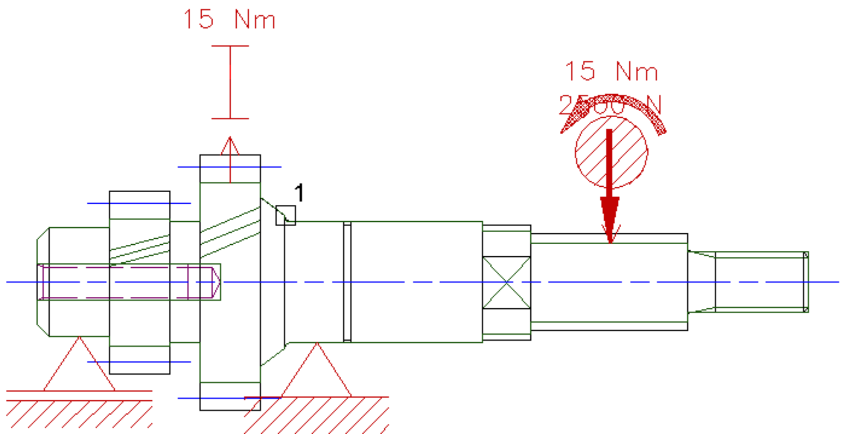
Select the shaft contour

The Shaft Calculation dialog box opens. Continue with calculations on the previously specified shaft.

3 In the Shaft Calculation dialog box, click the Strength button, and then respond to the prompt as follows:

Specify calculation position on shaft or [Graph]:

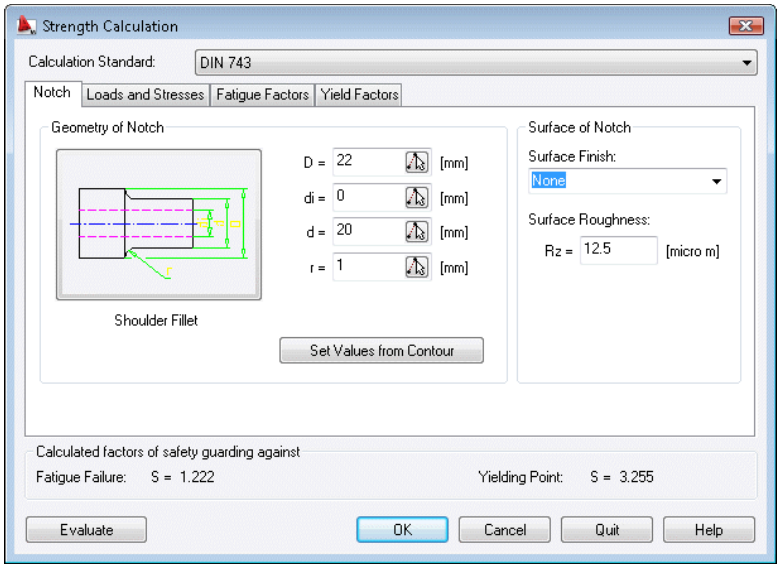
Specify the notch at the end of the conical section (1) (do not select the endpoint of the cylindrical shaft section)



NOTE This notch was selected because the calculation established that the highest bending stress is close to this place.

The Strength Calculation dialog box opens.

Use the Strength Calculation dialog box to specify the properties of the notch in more detail and display the strength values and factors.



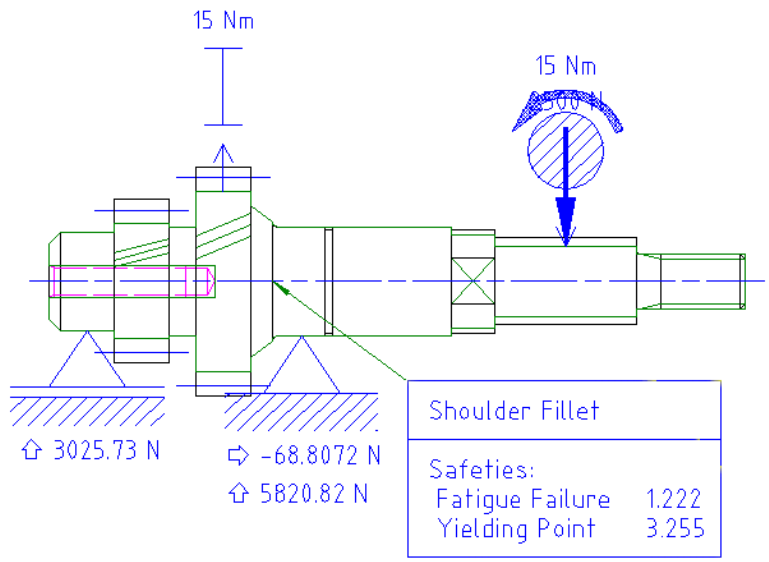
Click OK.

4 Respond to the prompts as follows:

`Specify next point <Symbol>:` *Specify a point below the shaft*

`Specify next point <Symbol>:` *Press* ENTER

The result block is inserted in the drawing.



The safety factors are greater than 1.0, so the shaft does not need to be redesigned at this notch.

5 Close the Shaft Calculation dialog box.

Save your file. This is the end of this tutorial chapter.

Calculating Moments of Inertia and Deflection Lines

13

Many engineering calculations are automated in AutoCAD® Mechanical. This tutorial illustrates how you calculate the moment of inertia for a profile section, and calculate the deflection line on a beam based on the profile calculation.

Key Terms

Term	Definition
bending moment	The moment of all forces that act on the member to the left of a section (a point along the member where bending moment needs to be calculated) taken about the horizontal axis of the section.
deflection line	A curve representing the vertical displacement of different points along the member subjected to a load.
distributed load	A load or force that is exerted over a certain length.
fixed support	A support that prevents translation as well as rotation about all axes.
load	Force or moment acting on a member or body.
moment of inertia	An important property of areas and solid bodies. Standard formulas are derived by multiplying elementary particles of area and mass by the squares of their distances from reference axes. Mo-

Term	Definition
	ments of inertia, therefore, depend on the location of reference axes.
movable support	A support that prevents rotation in all axes, but allows translation along one axis.
point force	A force that is concentrated on a point.

Performing Calculations

The measurement unit for the moment of inertia is mm4 or inches4. These are geometric values, which appear in deflection, torsion, and buckling calculation. AutoCAD Mechanical uses the result of the moment of inertia calculation for the deflection line calculation.

Moment of inertia calculations are performed on cross sections of beams or on other objects that can be represented as closed contours. Calculations can be performed on a cross section of any shape, as long as the geometry of the cross section forms a closed contour.

AutoCAD Mechanical determines the center of gravity for a cross section, draws the main axes, and calculates the moment of inertia about each of those axes. You can also select a load direction for a cross section; AutoCAD Mechanical calculates the moment of inertia and angle of deflection for that load.

NOTE Before you perform this exercise, verify that the ISO standard part standard is installed.

Load the initial drawing.

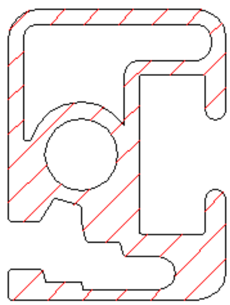
- Open the file *tut_calc* in the Tutorial folder.

Ribbon	➤ Open ➤ Drawing
Menu	File ➤ Open
Command	OPEN

NOTE The path to the folder containing tutorial files is;

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains this profile:



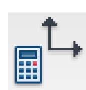
Save your file under a different name or directory to preserve the original tutorial file.

Calculating Moments of Inertia

In order to perform any calculations on a profile, you need to know its moment of inertia.

To calculate the moment of inertia

1 Start the calculation for the moment of inertia.

Ribbon	Content tab ➤ Calculation panel ➤ Moment of Inertia. 
Menu	Content ➤ Calculations ➤ Moment of Inertia
Command	AMINERTIA

2 Respond to the prompts as follows:

`Specify interior point:` *Click a point inside the profile*

`Specify interior point:` *Press* ENTER

`Is the area filled correctly? (Yes/No)? <Yes>:` *Press* ENTER

The coordinates of the centroid and the moment of inertia along the principle axes are displayed on the command line, as follows:

`Coordinates of centroid (in user coordinates):`

`X coordinate: 228.071933 Y coordinate: 150.027674`

`Moments of inertia along principal axes:`

`I1: 2.359e+004 I2: 1.4095e+004`

`Axis angle for major moment (I1): 5.3`

Define the direction of the loads. They must be in one plane.

3 Respond to the prompts as follows:

`Specify direction of load forces (must all lie in one plane):`

Enter 270, press ENTER

The data for this load direction is displayed on the command line, as follows:

`Effective moment of inertia for this load direction: 2.341e+004`

`Angle of deflection: 266.5`

`Maximum distances neutral line - border:`

`Extension side: 16.690 Compression side: 14.444`

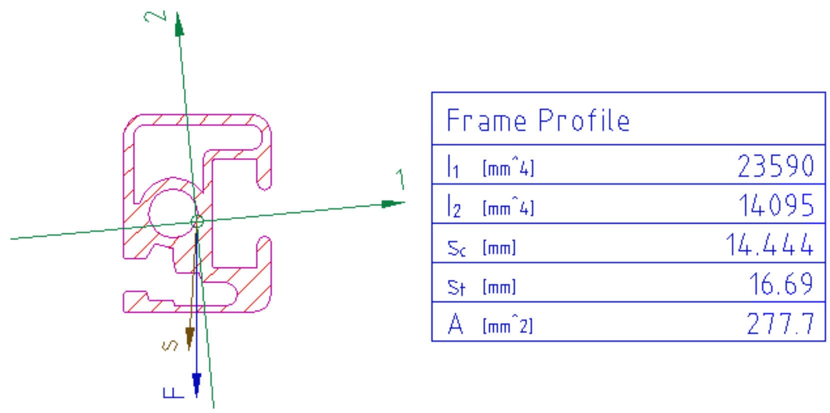
Enter a description for the calculated profile and locate the block with the calculation data in the drawing.

4 Respond to the prompts as follows:

`Enter description:` *Enter Frame Profile, press* ENTER

`Specify insertion point:` *Place the calculation block next to the profile*

Your drawing looks like this:



NOTE The main axes, 1 and 2, are the axes with the most and least deflection. The F arrow displays the direction of the force, the s arrow displays the resultant deflection. The moment of inertia block shows the moments related to the main axis, the maximum distances from the edges, and the calculated area. For more detailed information, see Help.

A side view of the profile has been created for the deflection line.

5 Zoom to the extents of the drawing.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Extents. 
Menu	View ➤ Zoom ➤ Extents
Command	ZOOM

Save your file.

Calculating Deflection Lines

The calculation of the deflection line requires the calculation result from the moment of inertia calculation.

Calculate the deflection line under a specific load situation.

To calculate the deflection line

1 Start the deflection line calculation.

Ribbon	Content tab ➤ Calculation panel ➤ Deflection Line. 
Menu	Content ➤ Calculations ➤ Deflection Line...
Command	AMDEFLINE

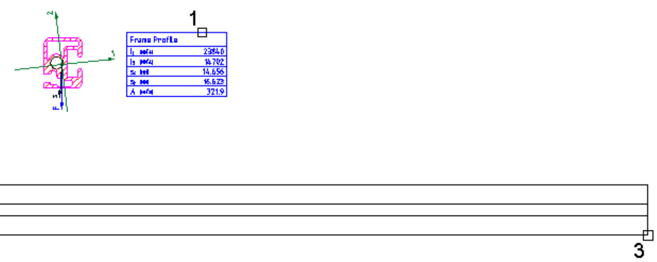
2 Respond to the prompts as follows:

`Select moment of inertia block:` *Select the calculation block (1)*

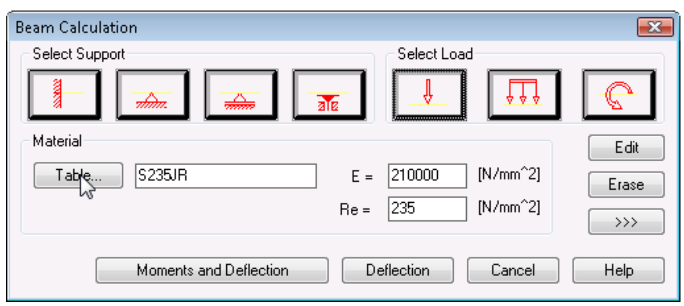
`Specify starting point or [Existing beam]:`

Select the left end of the beam (2)

`Specify endpoint:` *Select the right end of the beam (3)*



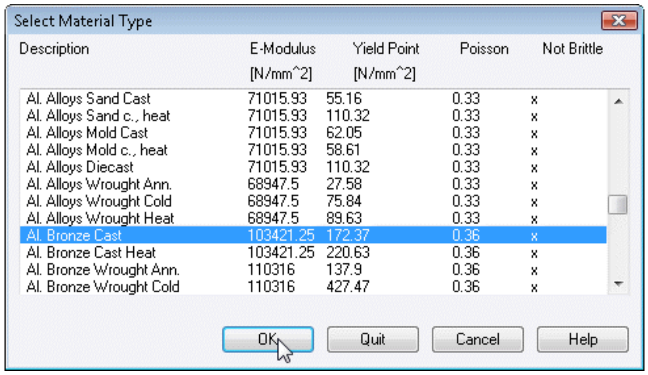
3 In the Beam Calculation dialog box, click Table.



4 In the Select Standard for Material dialog box, select ANSI Material.

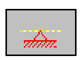
5 In the Material Type dialog box, select ANSI standard and the material Al. Bronze Cast.

NOTE If you have not installed ANSI standard, selecting a different standard according to your preference is also possible, but the results will differ from the results in this tutorial exercise. For example, if you select DIN, you can select a similar material, like AlMgSi0.5F22, to achieve similar results.

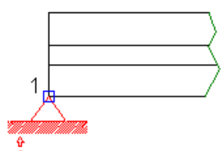


Click OK.

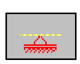
Define the supports and the loads.

6  Click the Fixed Support icon, and then respond to the prompt as follows:

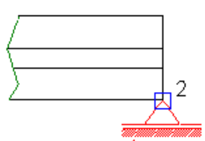
`Specify insertion point:` *Select the left edge of the beam (1)*

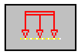


NOTE The support can only be placed along the beam.

7  Click the Movable Support icon, and then respond to the prompt as follows:

`Specify insertion point:` *Select the right edge of the beam (2)*

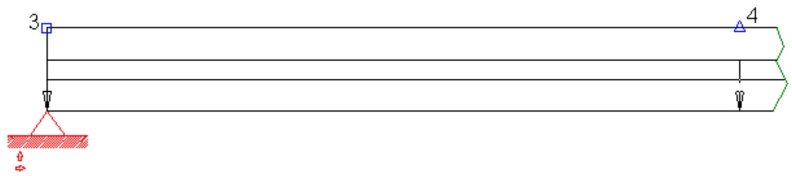


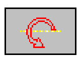
8  Click the Uniform Load icon, and then respond to the prompts as follows:

`Specify insertion point:` *Select the left edge of the beam (3)*

`Specify endpoint:` *Select the midpoint of the beam using midpoint snap (4)*

`Line Load [N/mm]<50>:` *Enter* `10`, *press* ENTER

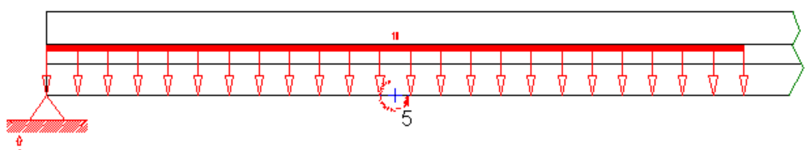


9  Click the Moment icon, and then respond to the prompts as follows:

`Specify insertion point:`

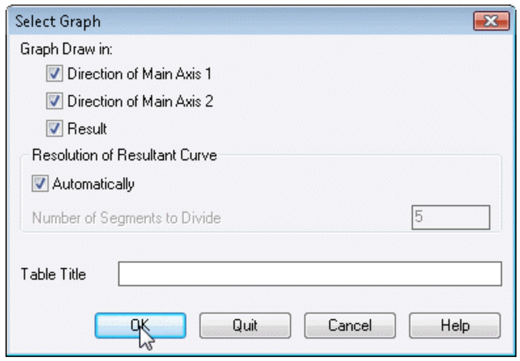
Select a point in the center of the uniform load (5)

`Bending moment (Nm)<10>:` *Enter 3, press* ENTER



10 In the Beam Calculation dialog box, click Moments and Deflection.

11 In the Select Graph dialog box, select the options as shown in the following figure, and then Click OK.



12 Respond to the prompts as follows:

`Enter scale for bending moment line (drawing unit:Nm)<1:1.3913>:`

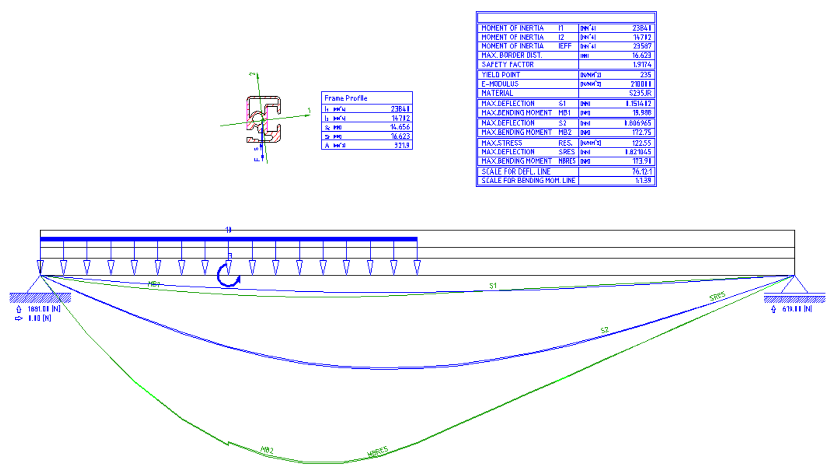
Press ENTER

`Enter scale for deflection (drawing unit:mm)<37.208:1>:`

Press ENTER

`Specify insertion point:` *Select a point in the drawing*

The result looks like this:



The calculation result block displays all important data on your calculation:

Moment of Inertia	I1	[mm^4]	23590
Moment of Inertia	I2	[mm^4]	14095
Moment of Inertia	Ieff	[mm^4]	23411
Max. Border Dist.		[mm]	16.69
Safety Factor			1.3873
Yield Point		[N/mm^2]	172
E-Modulus		[N/mm^2]	103421
Material			Al. Bronze Cast
Max.Deflection	S1	[mm]	0.257708
Max.Bending Moment	Mb1	[Nm]	16.064
Max.Deflection	S2	[mm]	1.659856
Max.Bending Moment	Mb2	[Nm]	173.16
Max.Stress	Res.	[N/mm^2]	123.98
Max.Deflection	Sres	[mm]	1.679743
Max.Bending Moment	Mbres	[Nm]	173.90
Scale for Defl. Line			37.21:1
Scale for Bending Mom. Line			1:1.39

Save your file. This is the end of this tutorial chapter.

14 Calculating Chains

In this AutoCAD® Mechanical tutorial, you calculate a chain length, and insert sprockets and chain links into a drawing.

Key Terms

Term	Definition
partition	Distance in mm or inches between centers of adjacent joint members. Other dimensions are proportional to the pitch. Also known as pitch.
pitch diameter	The diameter of the pitch circle that passes through the centers of the link pins as the chain is wrapped on the sprocket.
roller chain	A roller chain is made up of two kinds of links: roller links and pin links alternately and evenly spaced throughout the length of the chain.
sprocket	A toothed wheel that transfers the power from the chain to the shaft or the other way round.

Chain Calculations

NOTE Before you begin this tutorial exercise, be sure the ISO standard parts are installed on your screen.

Before you begin this tutorial...

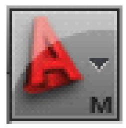
This tutorial requires the mechanical browser. If the mechanical browser is not visible:

1 Enter *AMBROWSER* on the command line and press ENTER.

2 When prompted, enter *ON* and press ENTER.

To load the tutorial drawing

1 Open the file *tut_chain.dwg* in the Tutorial folder.

Ribbon  ➤ Open ➤ Drawing

Menu File ➤ Open

Command OPEN

NOTE The path to the folder containing tutorial files is;

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

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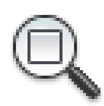
2 Save your file under a different name to preserve the original tutorial file.

3 Use a window to Zoom in to the chain housing.

Ribbon View tab ➤ Navigate panel ➤ Extents drop-down ➤ 

Menu View ➤ Zoom ➤ Zoom, Window

Command ZOOM

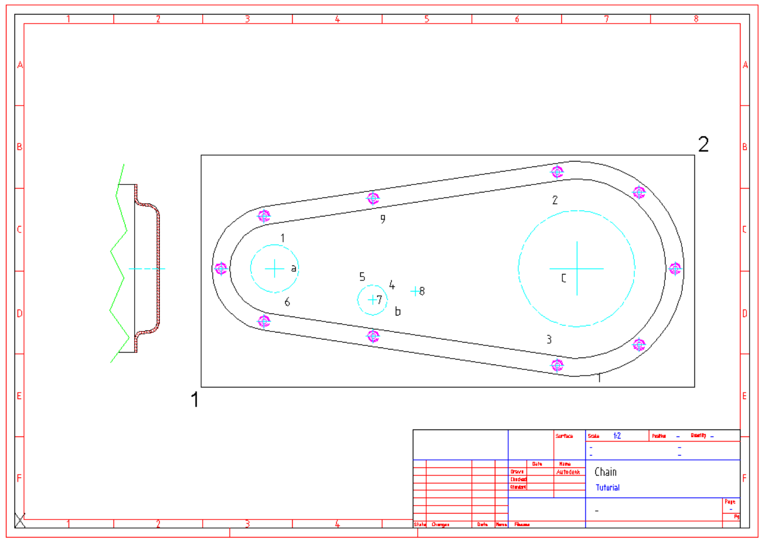
4 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:`

Enter W, press ENTER

`Specify first corner:` *Specify first corner point (1)*

`Specify opposite corner:` *Specify second corner point (2)*



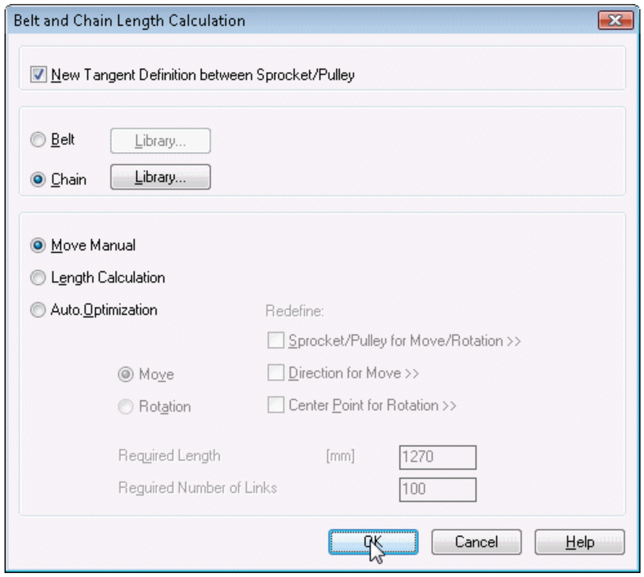
Performing Length Calculations

To calculate the required length of the chain

1 Start the Length Calculation command..

Ribbon	Content tab ➤ Tools panel ➤ Chains/Belts drop-down ➤ Length Calculation. 
Menu	Content ➤ Chains / Belts ➤ Length Calculation...
Command	AMCHAINLENGTHCAL

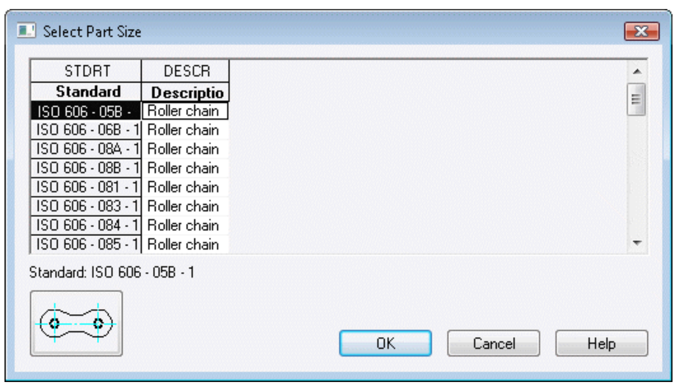
2 In the Belt and Chain Length Calculation dialog box, click Library.



3 In the Select a Chain dialog box, in the Details panel, select ISO 606 metric.

4 In the Select Part Size dialog box, specify:

`Standard: ISO 606 - 05B - 1`



Click OK.

5 In the Belt and Chain Length Calculation dialog box, Click OK, and then respond to the prompts as follows:

`Specify 1st point for tangent or [Undo] <exit>:` *Select circle a (1)*

Specify 2nd point for tangent: *Select circle c (2)*

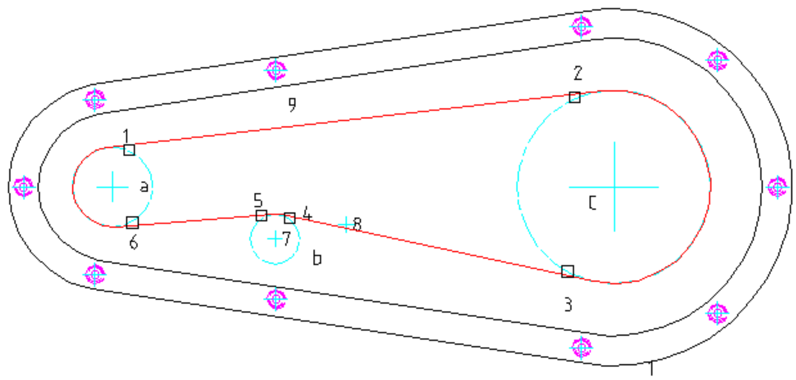
Specify 1st point for tangent or [Undo] <exit>: *Select circle c (3)*

Specify 2nd point for tangent: *Select circle b (4)*

Specify 1st point for tangent or [Undo] <exit>: *Select circle b (5)*

Specify 2nd point for tangent: *Select circle a (6)*

Specify 1st point for tangent or [Undo] <exit>: *Press* ENTER



The tangent definition is finished, and the length of the chain is calculated. Because the length is divided into whole numbers of links, one sprocket has to be moved to achieve such a length.

6 Continue responding to the prompts as follows:

Select pulleys or sprockets to be moved. Select objects:

Select circle b

Select objects: *Press* ENTER

Specify base point of displacement: *Select the center of circle b*

Specify second point of displacement: *Select the center of the cross (8)*

Select pulleys or sprockets to be moved.

Select objects: *Press* ENTER

AutoCAD Mechanical calculated the new length, which is still not a multiple of the chain division:

Number of links in chain: 121 Distance to next link: 6.88567 mm

Length: 974.8857

NOTE You can view the results by resizing the command line or opening the AutoCAD® Text Window using F2.

The chain arrangement has to be optimized to a length that is a multiple of the chain division.

Save your file.

Optimizing Chain Lengths

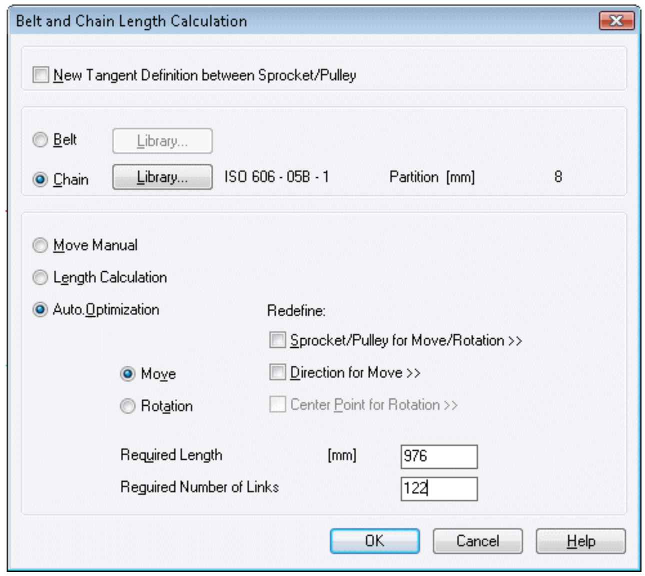
To optimize the chain length

1 Start the Length Calculation command.

Ribbon	Content tab ➤ Tools panel ➤ Chains/Belts drop-down ➤ Length Calculation. 
Menu	Content ➤ Chains / Belts ➤ Length Calculation...
Command	AMCHAINLENGTHCAL

2 In the Belt and Chain Length Calculation dialog box, select Auto Optimization and Move, and then specify:

`Required Number of Links:` 122



Click OK.

3 Respond to the prompts as follows:

`Select pulleys or sprockets to be moved.`

`Select objects:` *Select the relocated circle b*

`Select objects:` *Press* ENTER

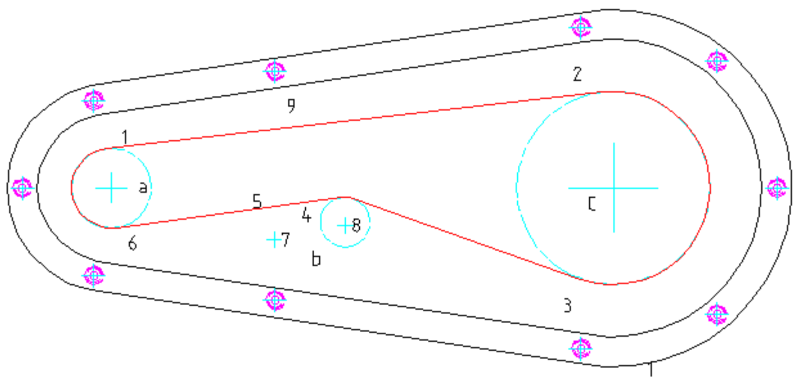
`Specify direction angle to move:` *Enter 90, press* ENTER

Sprocket b is moved until a chain length of 122 links is achieved.

4 In the Belt and Chain Length Calculation dialog box, Click OK.

Close the dialog box by clicking Cancel.

Your drawing looks like this:



Save your file.

Inserting Sprockets

To insert the sprocket

1 Start the Draw Sprocket/Pulley command.

Ribbon	Content tab ➤ Tools panel ➤ Chains/Belts drop-down ➤ Sprocket/Pulley. 
Menu	Content ➤ Chains / Belts ➤ Draw Sprocket/Pulley...
Command	AMSPROCKET

2 In the Select Pulley and Sprocket dialog box, Details panel, click Sprockets ➤ Front view.

Respond to the prompts:

`Specify insertion point:` *Select the center of circle a*

`Specify rotation angle < 0 >:` *Enter 360, press* ENTER

3 In the Sprockets - Size Selection dialog box, select ISO 606 05B-1, and then click Next.

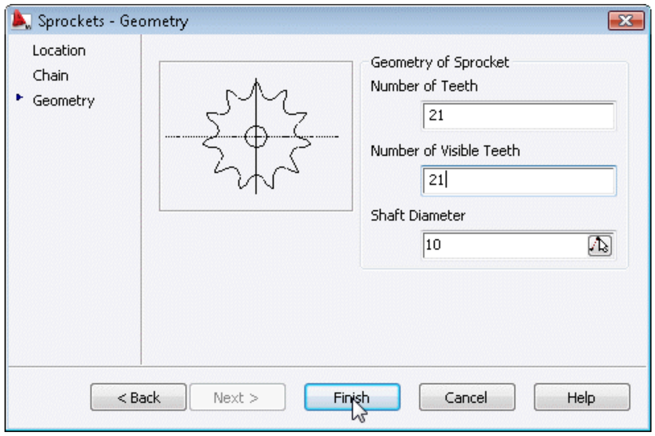
4 In the Sprockets - Geometry dialog box, specify:

`Geometry of Sprocket:`

`Number of teeth:` *21*

`Number of Visible Teeth:` *21*

`Shaft Diameter:` *10*



Click Finish.

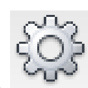
The sprocket is inserted into the drawing, and the Create Hide Situation dialog box is displayed.

5 In the Hide Situation dialog box, click OK.

A hide situation is created.

Insert the next two sprockets.

6 Start the Draw Sprocket/Pulley command again.

Ribbon	Content tab ➤ Tools panel ➤ Chains/Belts drop-down ➤ Sprocket/Pulley. 
Menu	Content ➤ Chains / Belts ➤ Draw Sprocket/Pulley...
Command	AMSPROCKET

7 In the Select Pulley and Sprocket dialog box, Buttons tab, click Sprockets ‰ Front view.

Respond to the prompts:

`Specify insertion point:` *Select the center of circle b*

`Specify rotation angle < 0 >:` *Enter 360, press* ENTER

8 In the Sprockets - Size Selection dialog box, select ISO 606 05B-1, and then click Next.

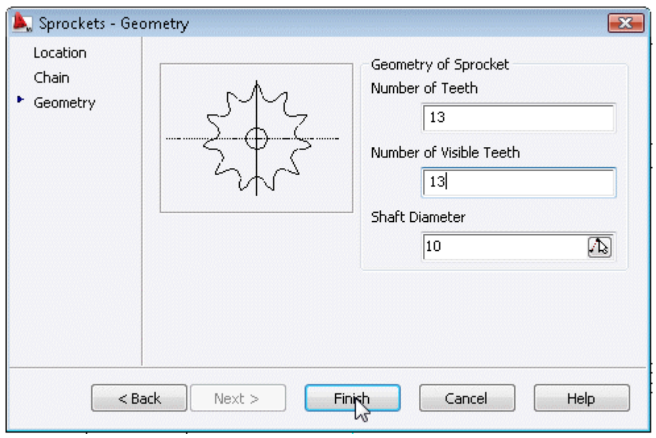
9 In the Sprockets - Geometry dialog box, specify:

`Geometry of Sprocket:`

`Number of teeth:` *13*

`Number of Visible Teeth:` *13*

`Shaft Diameter:` *10*



Click Finish.

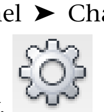
10 In the Create Hide Situation dialog box, click OK.

A hide situation is created, and is listed in the mechanical browser.

The sprocket is inserted into the drawing.

Create the next sprocket.

11 Start the Draw Sprocket/Pulley command again.

Ribbon	Content tab ➤ Tools panel ➤ Chains/Belts drop-down ➤ Sprocket/Pulley. 
Menu	Content ➤ Chains / Belts ➤ Draw Sprocket/Pulley...
Command	AMSPROCKET

12 In the Select Pulley and Sprocket dialog box, Details panel, click Sprockets ➤ Front view.

Respond to the prompts:

`Specify insertion point:` *Select the center of circle c*

`Specify rotation angle < 0 >:` *Enter 360, press* ENTER

13 In the Sprockets - Size Selection dialog box, select ISO 606 05B-1, and then click Next.

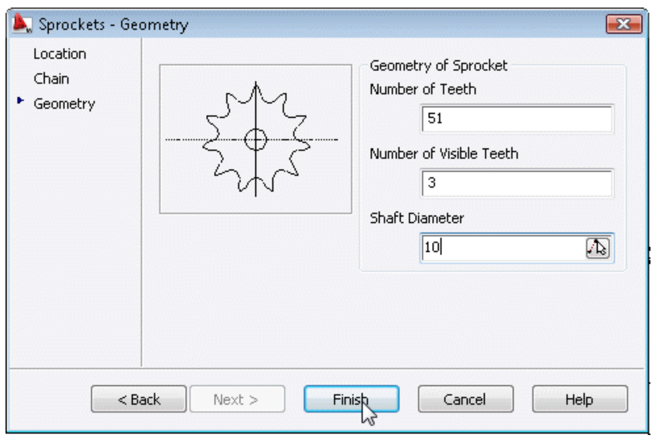
14 In the Sprockets - Geometry dialog box, specify:

`Geometry of Sprocket:`

`Number of teeth:` *51*

`Number of Visible Teeth:` *3*

`Shaft Diameter:` *10*

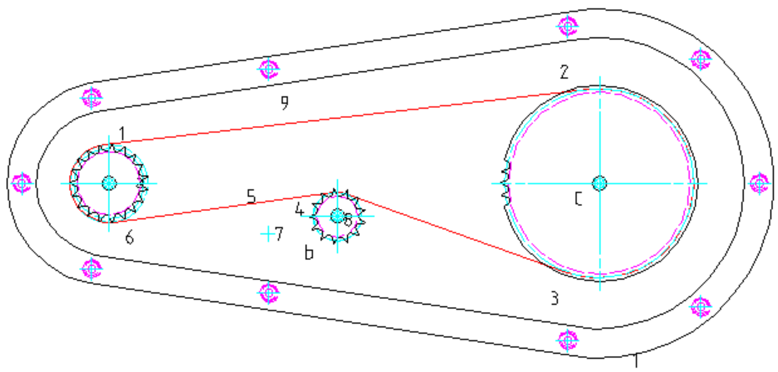


Click Finish.

15 In the Create Hide Situation dialog box, click OK.

A hide situation is created, and is listed in the mechanical browser.

The last sprocket is inserted as a simplified representation with only three teeth, as specified in the dialog box. Your drawing looks like this:



Save your file.

Inserting Chains

To insert a chain

1 Start the Draw Chain/Belt Links command.

Ribbon	Content tab ➤ Tools panel ➤ Chains/Belts drop-down ➤ Chain/Belt Links. 
Menu	Content ➤ Chains / Belts ➤ Draw Chain/Belt Links...
Command	AMCHAINDRAW

2 In the Select Belt and Chain dialog box, Details panel, click Chains.

Respond to the prompts:

`Select polyline:` *Select the polyline near point 9*

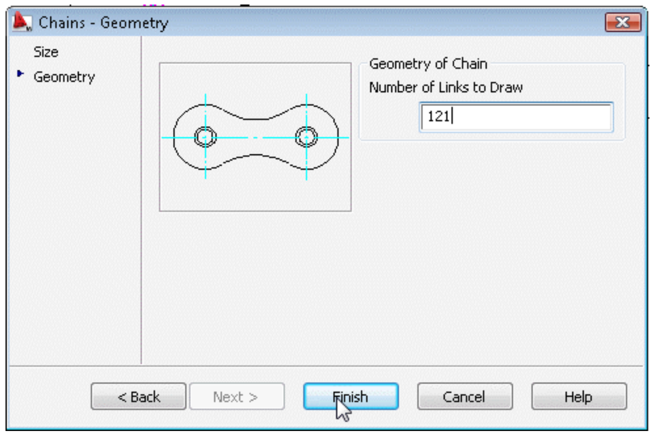
`Select starting point on polyline:` *Select a point on the polyline*

3 In the Select a Chain dialog box, select ISO 606 Metric.

4 In the Chains - Size Selection dialog box, select ISO 606 05B - 1, and then click Next.

5 In the Chains - Geometry selection dialog box, specify:

`Number of Links:` 121



Click Finish.

6 Respond to the prompts:

`Specify direction of Links [Flip/Accept] <Accept>:` *Press* ENTER

`Specify orientation of Links [Flip/Accept] <Accept>:`

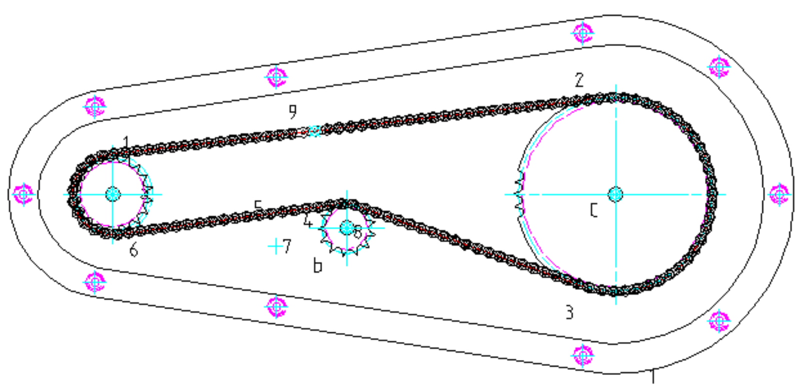
Enter F, press ENTER

`Specify orientation of Links [Flip/Accept] <Accept>:` *Press* ENTER

7 In the Hide Situation dialog box, click OK.

The chain is inserted into the drawing, and a hide situation is created.

Your drawing looks like this:



The mechanical browser reflects the standard components you created in the drawing.

Save your file. This is the end of this tutorial chapter.

Calculating Springs

15

In this tutorial, you calculate a spring for existing boundary conditions and insert the spring into a drawing. You copy and edit the spring using the Power Copy and Power Edit commands in AutoCAD® Mechanical.

Key Terms

Term	Definition
Belleville spring washer	A washer-type spring that can sustain relatively large loads with small deflections. The loads and deflections can be increased by stacking the springs.
compression spring	A spring type that can be compressed and can absorb pressure forces.
dynamic dragging	The act of determining the size of a standard part with the cursor while inserting the part 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.
extension spring	A spring type that can absorb tension forces.
Power Copy	A command that copies a drawing object to another position in the drawing. Power Copy produces an identical copy of the copied object.
Power Edit	A single edit command for all objects in a drawing.
torsion spring	A spring type that can absorb torque forces.

Calculating Springs

With the AutoCAD Mechanical spring function, you can insert compression, extension, and torsion springs, as well as Belleville spring washers. The calculation is carried out in accordance with DIN 2098 or ANSI. The standard sizes of the springs can be selected from various standard catalogs.

NOTE The ISO standard parts have to be installed for this tutorial exercise.

In this tutorial, you create a compression spring in two different compression situations. You calculate and insert the springs in the existing drawing.

Perform this tutorial with mechanical structure disabled.

To open a drawing

1 Open the file *tut_spring.dwg* in the tutorials folder at:

Ribbon	➤ Open ➤ Drawing
Menu	File ➤ Open
Command	OPEN

NOTE The path to the folder containing tutorial files is;

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

2 Zoom in to the area of the spring housings.

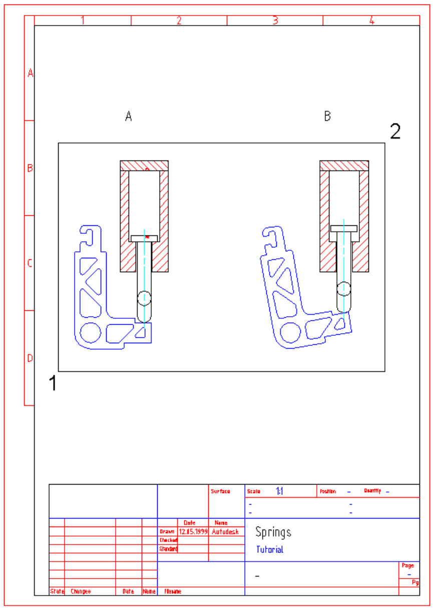
Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window.
Menu	View ➤ Zoom ➤ Window
Command	ZOOM

3 Respond to the prompts as follows:

`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` *Enter W, press* ENTER

4 `Specify first corner:` *Specify first corner (1)*

`Specify opposite corner:` *Specify opposite corner (2)*



The drawing shows two views (A and B) of the lever and spring housing, to reflect two different states of compression.

Save your file under a different name or to a different directory to preserve the original tutorial file.

Starting Spring Calculations

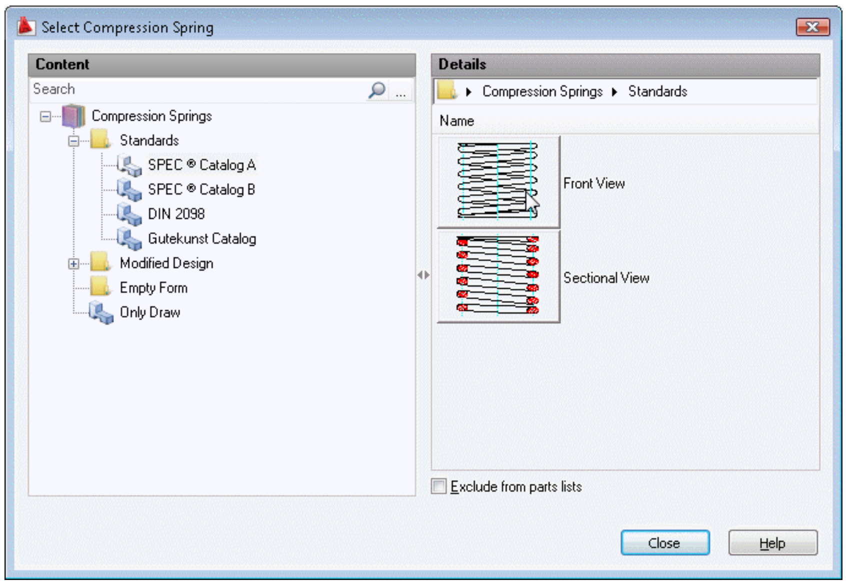
Specify the spring and the location.

To specify a spring

1 Start the Compression Spring command..

Ribbon	Content tab ➤ Tools panel ➤ Springs drop-down ➤ Compression. 
Menu	Content ➤ Springs ➤ Compression...
Command	AMCOMP2D

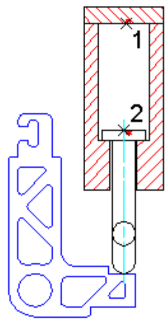
2 In the Select Compression Spring dialog box, click Standards ➤ SPEC® Catalog A ➤ Front View.



3 Respond to the prompts as follows:

`Specify starting point:` *Specify the starting point (1)*

`Specify direction:` *Specify endpoint (2)*



Specifying Spring Restrictions

Specify the spring restrictions. Use the Compression Springs dialog box to restrict the spring selection in various ways.

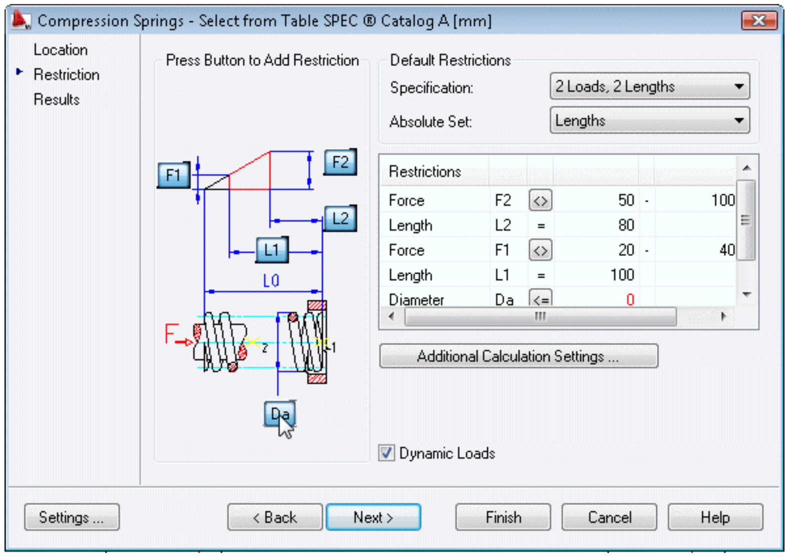
To specify the spring restrictions

1 In the Compression Springs - Select from Table SPEC® Catalog A [mm] dialog box, specify:

```
Specification: 2 Loads, 2 Lengths
```

```
Absolute Set: Lengths
```

Click the Da button.

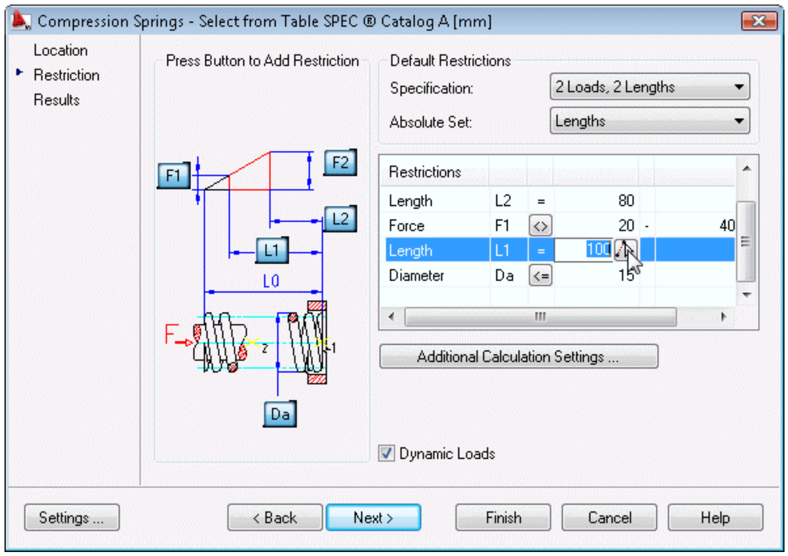


A row for specifying the outer diameter Da is added to the restrictions table.

2 Click the value field for the diameter Da. You can pick a point on the inner spring housing to specify the diameter, or enter a value. In this instance, enter the value *15*.

Define the initial spring length.

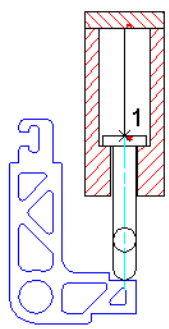
3 In the Compression Springs - Select from Tables SPEC® Catalog A [mm] dialog box, click the value field for the length L1, and then choose the pick icon.



4 Respond to the prompts as follows:

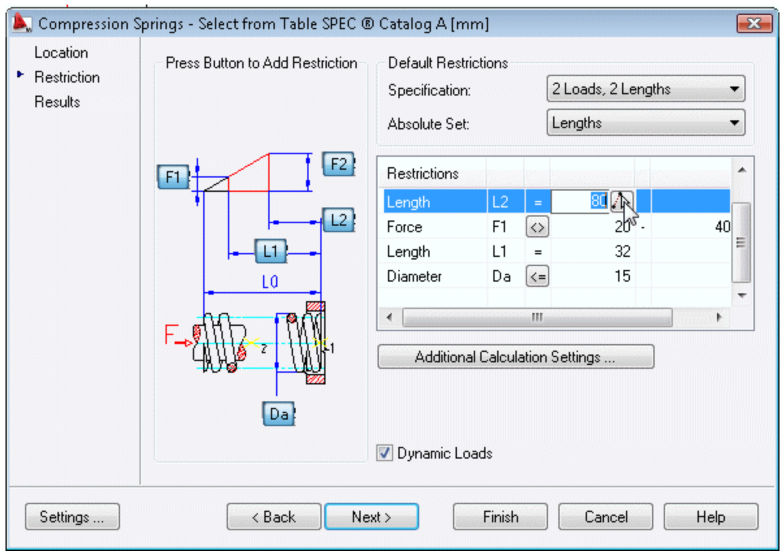
```
Specify point for spring length L1:
```

Select a point on the spring pressure plate (1)



Use view B of the lever and spring housing to define the compressed spring length.

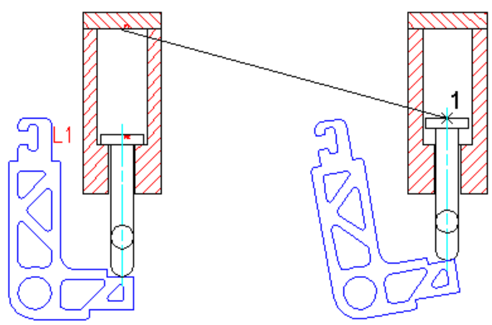
5 In the Compression Springs dialog box, click the value field for the length L2, and then choose the pick icon.



6 Respond to the prompts as follows:

```
Specify point for spring length L2:
```

Select a point on the spring pressure plate in view B (1)



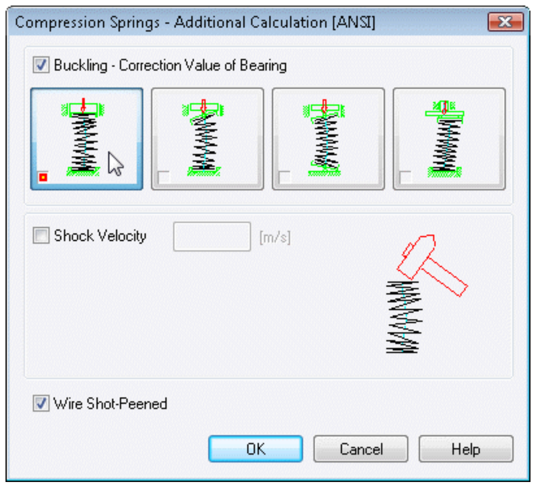
The geometric boundary conditions are defined, and you can proceed with the calculation.

Calculating and Selecting Springs

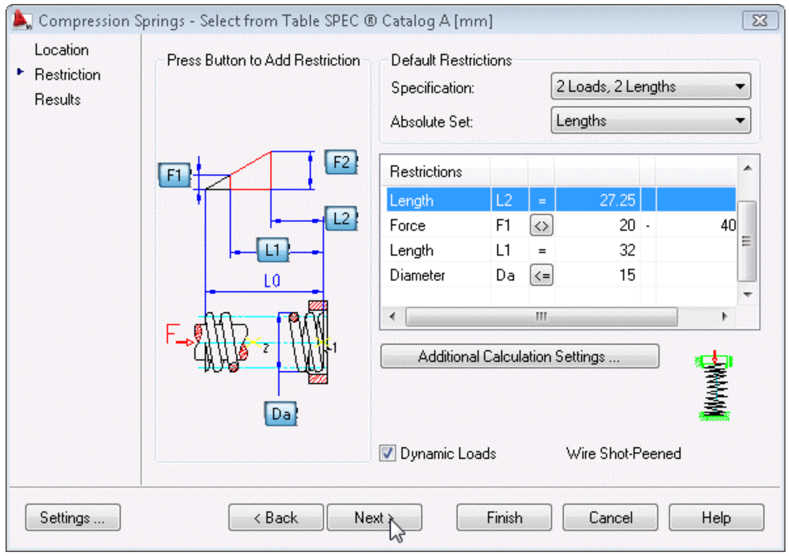
Make the calculation settings and calculate the possible springs.

To calculate and select a spring

1. In the Compression Springs - Select from Tables SPEC® Catalog A [mm] dialog box, choose the Additional Calculation Settings button.
2. In the Compression Springs - Additional Calculation [ANSI] dialog, select the left buckling case, and then Click OK.

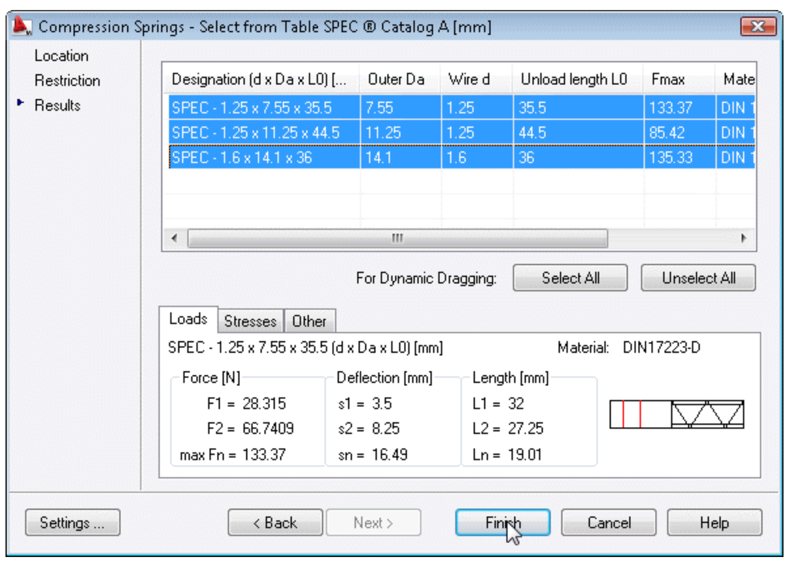


3. In the Compression Springs - Select from Tables SPEC® Catalog A [mm] dialog box, choose Next.



The possible springs are calculated and the results are displayed in the Compression Springs - Select from Tables SPEC® Catalog A [mm] dialog box.

4 Choose Select All to select all possible springs for the dynamic dragging process.



Choose Finish.

Inserting Springs

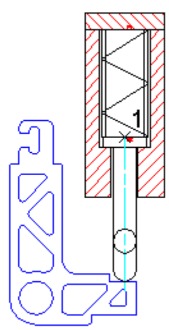
Drag the cursor dynamically to switch between the selected possible springs. The outline of the spring is displayed in the drawing and the spring description is displayed in the tooltip.

To insert a spring

1. Drag the cursor until the tooltip reads SPEC - 1.6 x 14.1 x 36, and then click.
2. Respond to the prompts as follows:

   Topical Length (14.28 - 36) [Force/Deflection] <32.01>:

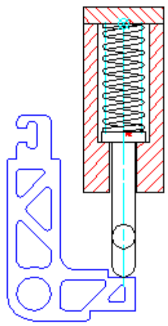
   Select a point on the spring pressure plate (1)

3. Continue to respond to the prompts as follows:

   Select rod (only closed contours) <Enter=continue>: *Press* ENTER

   The spring is inserted as shown below.



Save your file.

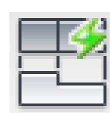
Creating Views of Springs with Power View

In order to adjust the length of the spring in view B, the springs in the two views need to be different components rather than instances of the same component.

Use the previously inserted spring in view A to create a spring for view B, using the Power View command.

To create a view of a spring with Power View

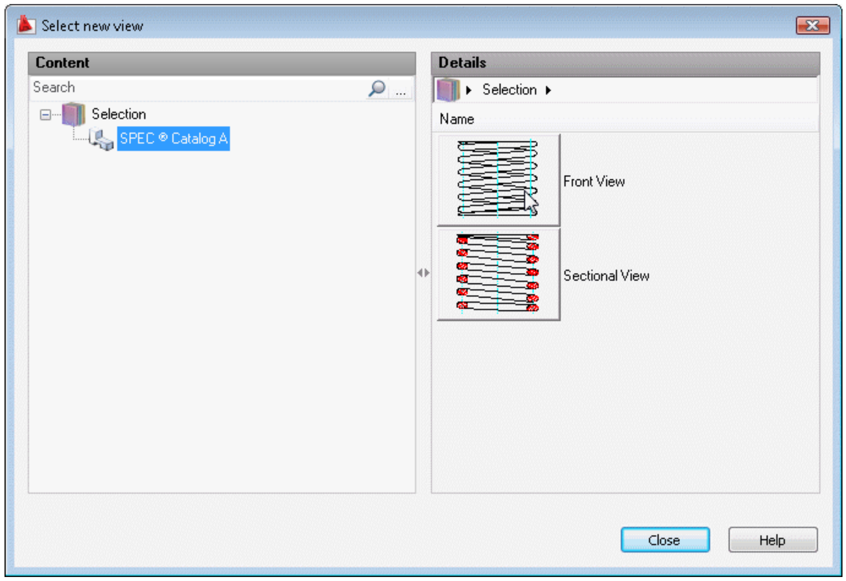
1 Start the Power View command.

Ribbon	Tools ➤ Content ➤ 
Menu	Modify ➤ Power View
Command	AMPOWERVIEW

2 Respond to the prompts as follows:

`Select objects:` *Select the spring in view A*

3 In the Select New View dialog box, select Front View.



4 Respond to the prompts:

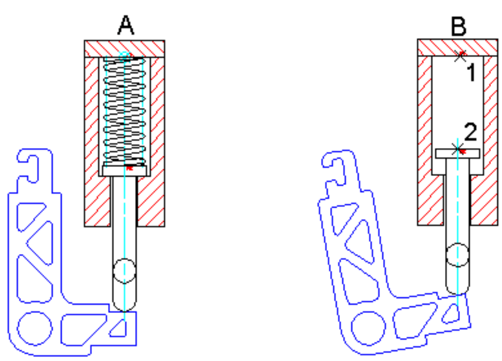
`Specify starting point:` *Select point (1) in view B*

`Specify direction:` *Select point (2) in view B*

`Topical Length (14.28 - 36)[Force/Deflection]<32.01>:`

Select the lower contact point of the compressed spring

`Select rod (only closed contours) <Enter=continue>:` *Press* ENTER



The spring is copied into view B in its compressed length.

Save your file. This is the end of this tutorial chapter.

Calculating Screw Connections

16

In this tutorial, you calculate a screw connection using the stand-alone screw calculation function in AutoCAD® Mechanical.

Key Terms

Term	Definition
axial force	A force parallel to the screw axis.
contact area	The touching surfaces of the plates, which are effective for the calculation.
safety factor	The safety factor is the ratio of effective load and safe load.
shear force	A force perpendicular to the screw axis.
stress	The force acting on a member or body per unit area.

Methods for Calculating Screws

The Screw Calculation provides two different ways to calculate a screw connection:

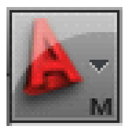
- Stand-alone calculation: All data and properties are specified by the user.

■ Calculation of an existing screw connection: The user selects an existing screw connection to be calculated. All geometric and standard-related data is taken from the screw connection and cannot be edited.

In this exercise, you use the stand-alone Screw Calculation. With the standalone calculation, you can calculate a screw connection without any prerequisites. You can specify the screw connection in detail (material, geometry, load, settlement and tightening properties). In this exercise, you are provided with the drawing of a screw calculation. Some values are selected from tables, some are entered manually, and some are taken directly from the drawing.

To open the initial drawing

1 Open the file *tut_screw.dwg* in the tutorials folder.

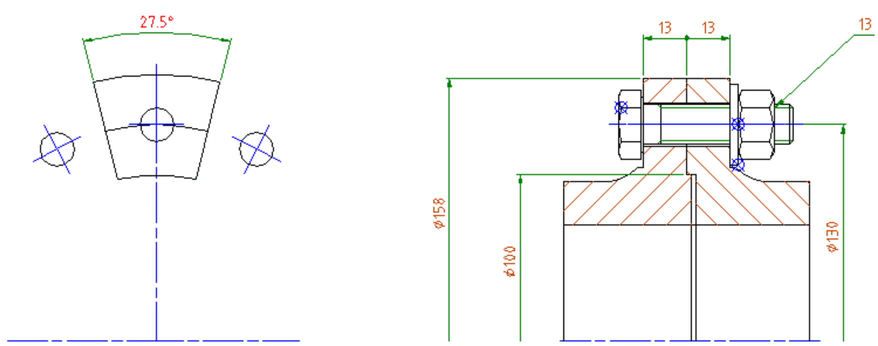
Ribbon	 ➤ Open ➤ Drawing
Menu	File ➤ Open
Command	OPEN

NOTE The path to the folder containing tutorial files is;

■ **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

■ **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

The drawing contains the representation of a screw connection.



2 Save your file under a different name or to a different directory to preserve the original tutorial file.

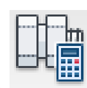
Problem for this exercise

- Two hollow shafts made of Cq 45 with forged coupling flanges are to be connected by 13 hex-head bolts ISO 4017 M12 x 45 - 10.9, which are arranged at a pitch diameter of 130 mm.
- The through holes are according to ISO 273 close.
- The bolts are safeguarded against loosening by gluing the threads $(\mu = 0.14)$. The tightening takes place manually using a torque wrench (k = 1.8).
- The flanged connection is to be designed for a alternating torque of T = 2405 Nm and non-skid (seal safety of plates $\geq$ 1).

Using Stand Alone Screw Calculations

To start the Screw Calculation

1 Start the Screw Calculation command.

Ribbon	Content tab ➤ Calculation panel drop-down ➤ Screw Calculation. 
Menu	Content ➤ Calculations ➤ Screw Calculation...
Command	AMSCREWCALC

2 Respond to the prompts as follows:

`Select screw connection <Stand alone calculation>:` *Press* ENTER

The Screw Calculation dialog box opens.

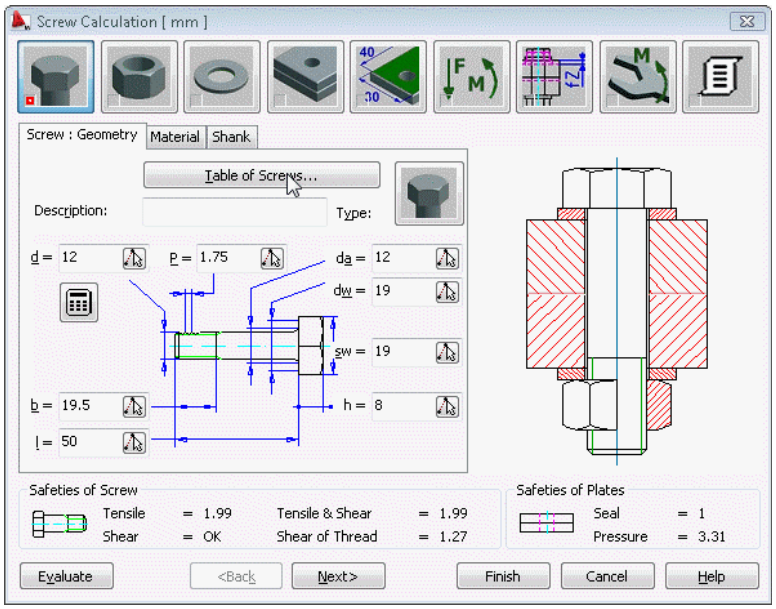
Specify the screw connection.

Selecting and Specifying Screws

In the Definition of SCREW section of the screw calculation, you can select the screw standard and size and the material properties. You can also enter the geometric properties of a user-defined screw, for example in detail.

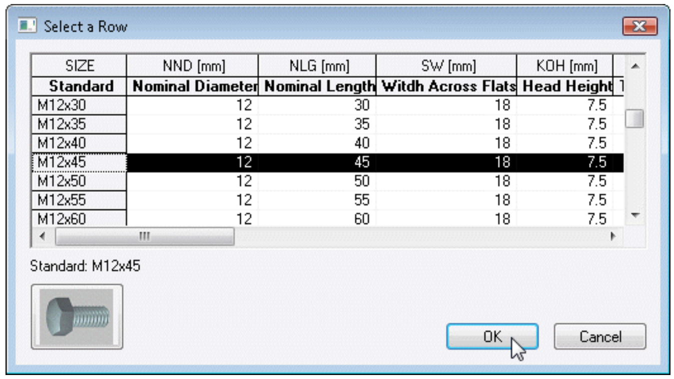
To specify a screw

1 On the Screw: Geometry tab, click Table of Screws.



2 In the Select a Screw dialog box, in the Details panel, click Hex Head Types, and then click ISO 4017 (Regular Thread).

3 In the Select a Row dialog box, choose the standard M12x45.

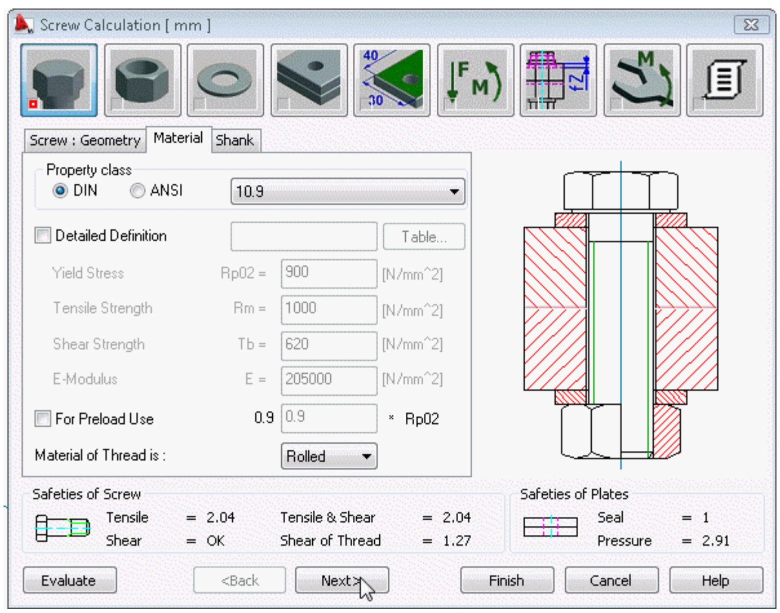


Click OK.

The geometric values of the standard screw ISO 4017 M12x45 are entered. Specify the property class.

4 Click the Material tab and then specify:

`Property class:` DIN 10.9



The screw is specified completely.

Specify the nut.

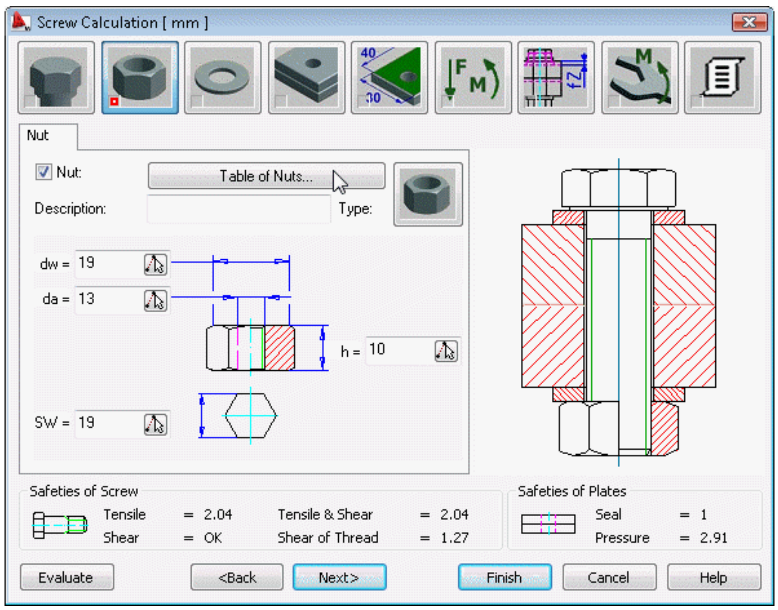
5 Click Next or the Definition of NUT icon in the top row to proceed.

Selecting and Specifying Nuts

In the Definition of NUT section of the screw calculation, you can select a nut standard and size.

To specify a nut

1 On the Nut tab, click Table of Nuts.



2 In the Select a Nut dialog box, in the Details pane, click Hex Nuts and then, ISO 4032 (Regular Thread).

You do not need to specify a size, because the size is determined by the screw size.

Specify the washers.

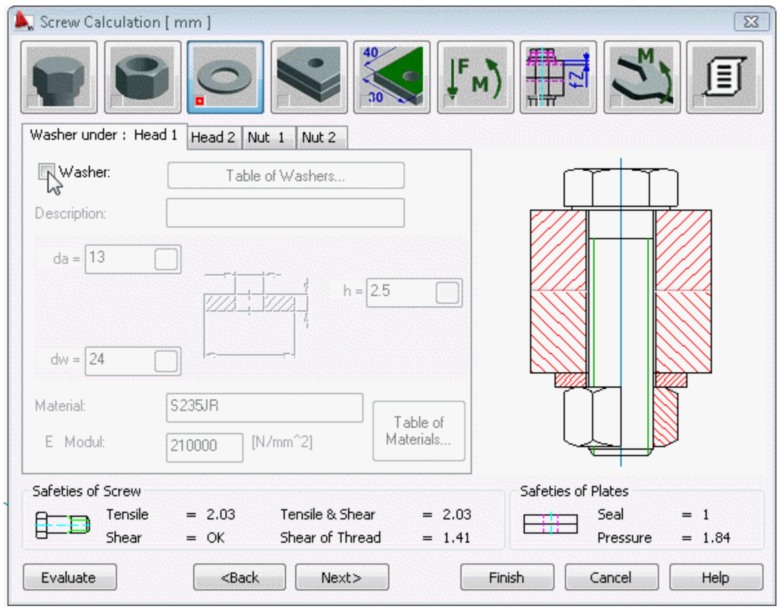
3 Click Next.

Selecting and Specifying Washers

In the Definition of WASHERS section of the screw calculation, you can select the washer standard and size and the positions of the washers.

To specify a washer

1 On the Washer under: Head 1 tab, clear the Washer check box.



2 Click the Nut 1 tab, and then click the Table of Washers button.

3 In the Select a Washer dialog box, choose ISO 7091.

Specify the plates.

4 Click Next.

Specifying Plate Geometry and Properties

In the Definition of PLATES section of the screw calculation, you can select plate materials and their geometric properties.

To specify the plates

1 On the Plates tab, specify:

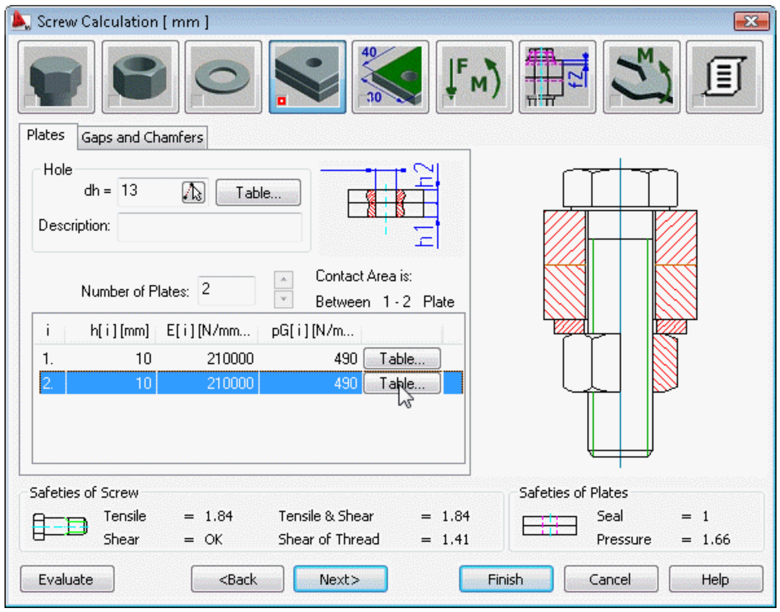
`Hole: dh:` *13*

`Number of Plates:` *2*

`Height of plate 1 h1:` *10*

`Height of plate 2 h2:` *10*

2 Click any of the Table buttons.



3 In the Please Select a Part dialog box, in the Details panel, click DIN material.

4 Choose the material Cq 45, and then Click OK.

5 Repeat steps 3 and 4 for the other Table button.

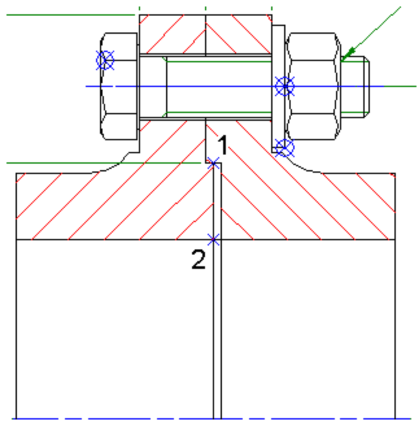
Specify the contact area.

6 On the Gaps and Chamfers tab, click the pick button of the value gr.

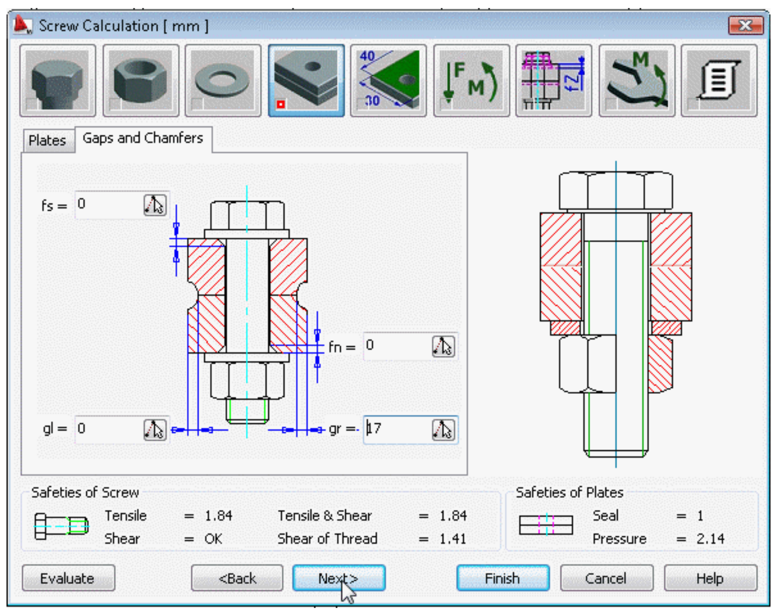
7 Respond to the prompts as follows:

`Specify first point:` *Select the point (1)*

`Second point:` *Select the point (2)*



The value for gr is changed to 17, as shown in the illustration.



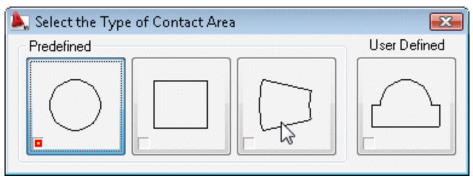
8 Click Next.

Specifying Contact Areas

In the Definition of CONTACT AREA section of the screw calculation, you can specify the geometric properties of the contact area.

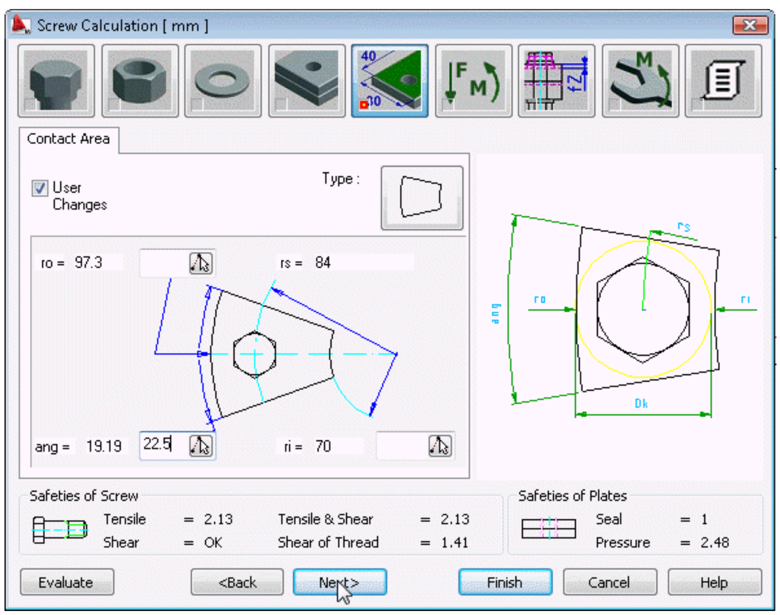
To specify the contact area

1. On the Contact Area tab, click the Type button.
2. In the Select the Type of Contact Area dialog box, click the third button from the left.



3. Select the User Changes check box.
4. In the entry field, specify:

`ang:` *22.5*



5 For the outer radius ro, click the pick button next to the entry field and respond to the prompts as follows:

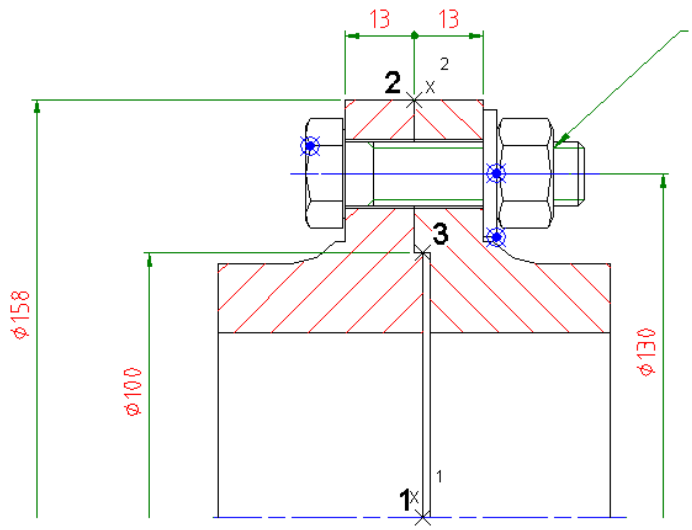
`Specify first point:` *Select the point (1)*

`Second point:` *Select the point (2)*

6 For the inner radius ri, click the pick button next to the entry field and respond to the prompts as follows:

`Specify first point:` *Select the point (1)*

`Second point:` *Select the point (3)*



Specify the loads and moments.

7 Click Next.

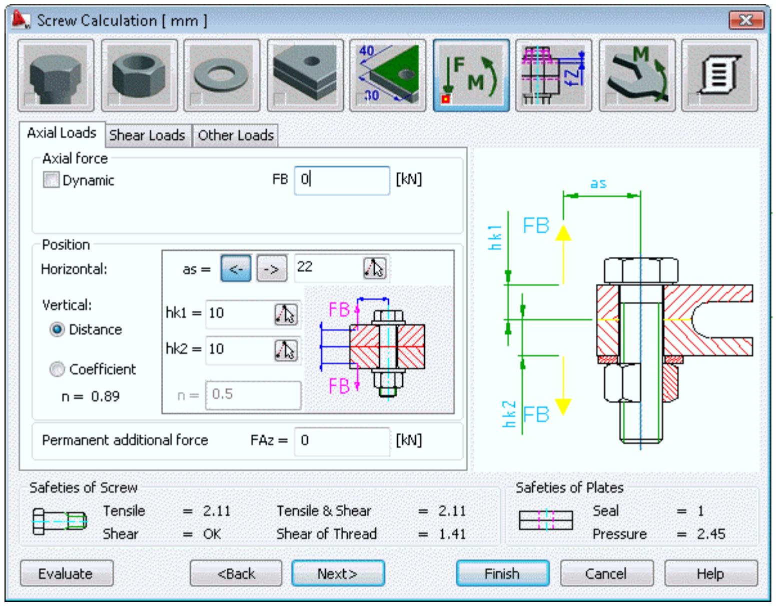
Specifying Loads and Moments

In the Definition of LOADS section of the screw calculation, you can specify the loads and moments and their points of application.

To specify loads and moments

1 On the Axial Loads tab, clear the Dynamic check box and specify:

`Axial force: FB:` *0*



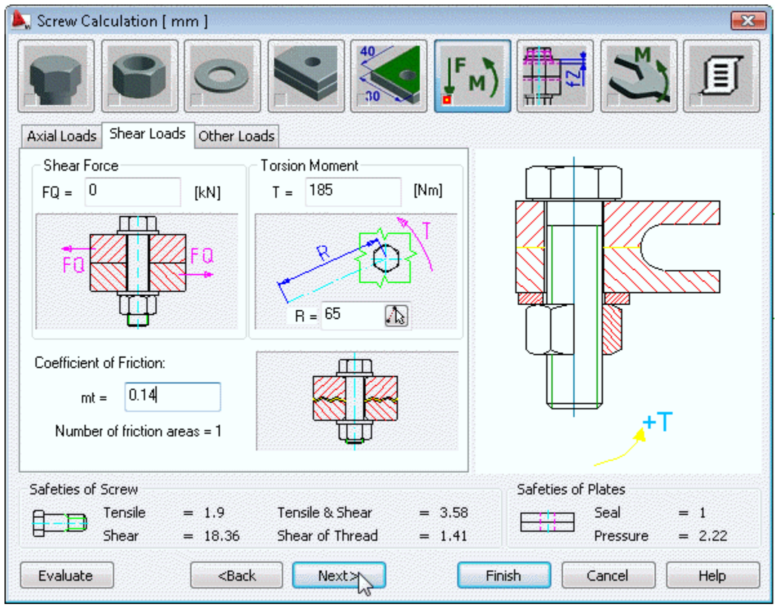
2 Click the Shear Loads tab and specify:

`Torsion Moment T =:` *185* `[Nm]`

`Radius R:` *65*

`Coefficient of Friction: mt=:` *0.14*

NOTE The torsion moment of 185 Nm results from the total torsion moment of 2405 Nm as given in the terms of reference divided by the 13 bolts.



Specify the settlement.

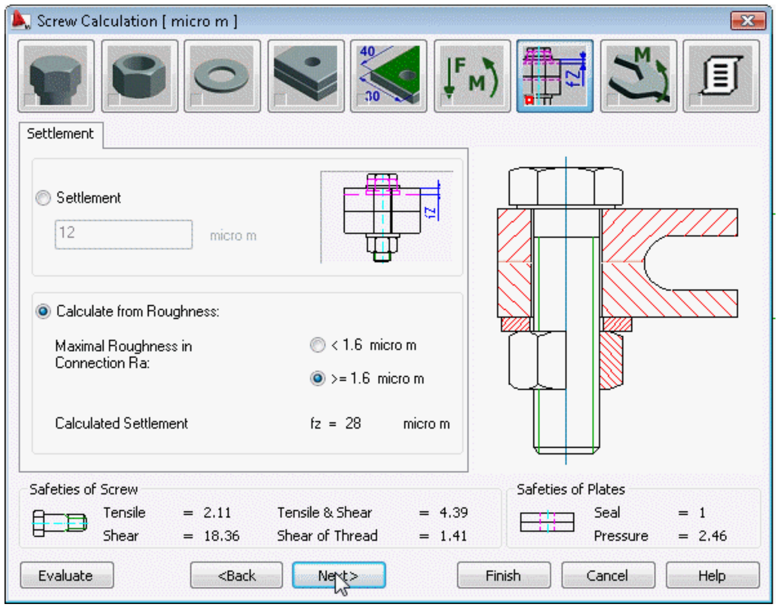
3 Click Next.

Specifying Settlement Properties

In the Definition of SETTLEMENT section of the screw calculation, you can specify settlement properties.

To specify the settlement

1 Click Calculate from Roughness and >= 1.6 micro m.



Specify the tightening.

2 Click Next.

Specifying Tightening Properties

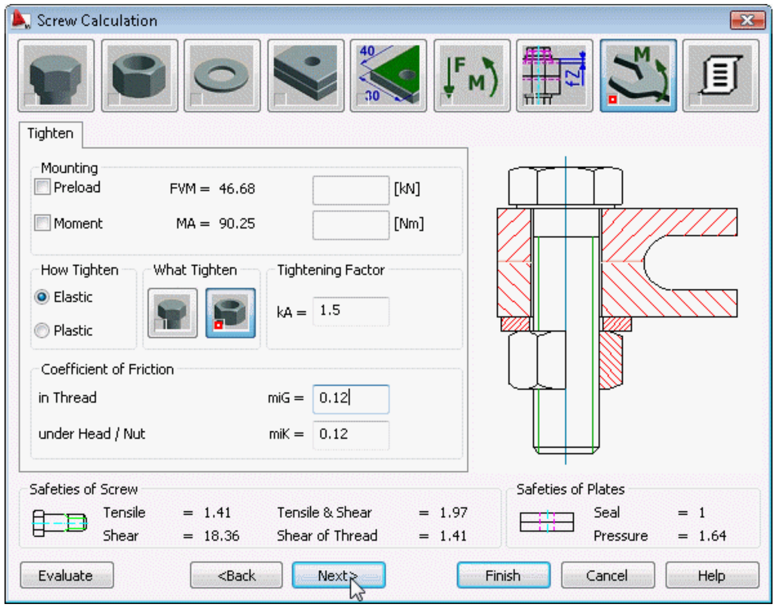
In the Definition of TIGHTEN section of the screw calculation, you can specify the tightening method and properties.

To specify the tightening

1 Specify as follows:

`Tightening Factor: kA=:` *1.5*

`Coefficient of Friction: in Thread miG =:` *0.12*



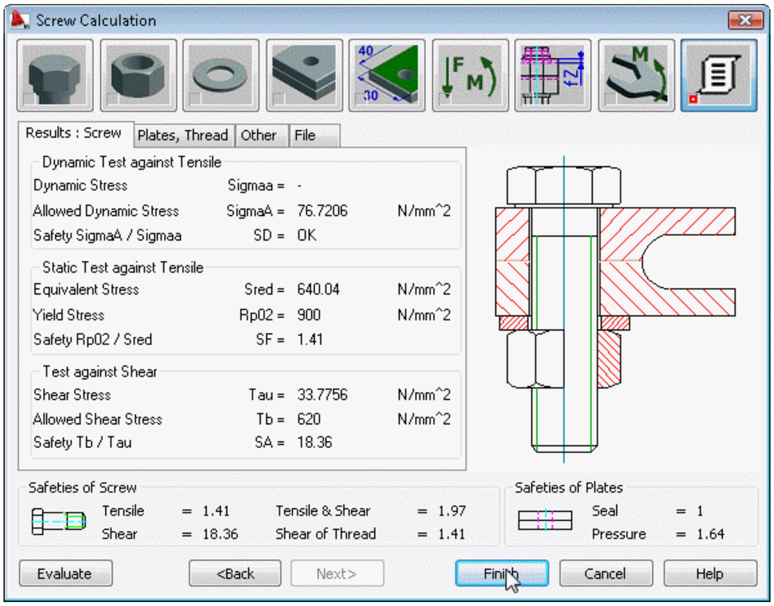
Insert the result block.

2 Click Next.

Creating and Inserting Result Blocks

In the Results section of the screw calculation, you can take a look at the results.

You have a complete overview of the results of the screw calculation.



Insert the result block.

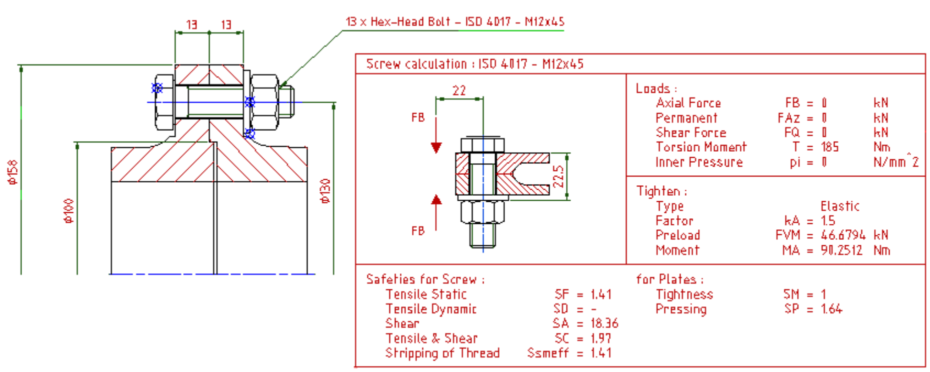
To insert a result block

■ Click Finish and respond to the prompts as follows:

`Specify start point:` *Specify a point right of the screw connection*

`Specify next point <Symbol>:` *Press* ENTER

The result block is inserted at the specified location.



Save your file. This is the end of this tutorial chapter.

Calculating Stress Using FEA

17

In this tutorial, you calculate the stresses in a lever using the finite element analysis (FEA) in AutoCAD® Mechanical. You use the results to improve the design of the lever.

Key Terms

Term	Definition
distributed load	A load or force that is exerted over a certain length.
FEA	Finite Element Analysis. A calculation routine based on analyzing a rigid body subject to loads and restraints for stress, strain, and deformation.
fixed support	A support that prevents translation as well as rotation about all axes.
load	Force or moment acting on a member or body.
movable support	A support that prevents rotation in all axes, but allows translation along one axis.
Power Edit	A single edit command for the objects in your drawing.
stress	The force acting on a member or body per unit area.

2D FEA

To determine the stability and durability of a given structure under various loading situations, you need to observe the stress and deformation in the components while they are being loaded. A structure is considered to be durable if the maximum stress is less than what the material permits.

There are various computational methods for calculating deformation and stress conditions. One of these methods is called the Finite Element Analysis.

The knowledge gained from this stress rating may lead to changing the structure in certain areas, which in turn necessitates changes to the design.

The FEA routine uses its own layer group for input and output.

Note that FEA is not designed for solving all special FEA tasks. Its purpose is to provide you with a quick idea of the stress and deformation distributions.

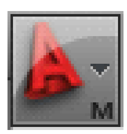
NOTE The ISO standard parts have to be installed for this tutorial exercise.

For this exercise, work with mechanical structure disabled.

To open the initial drawing

1 Open the file *tut_fea.dwg* in the tutorials folder at:

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

Ribbon	 ➤ Open ➤ Drawing
Menu	File ➤ Open
Command	OPEN

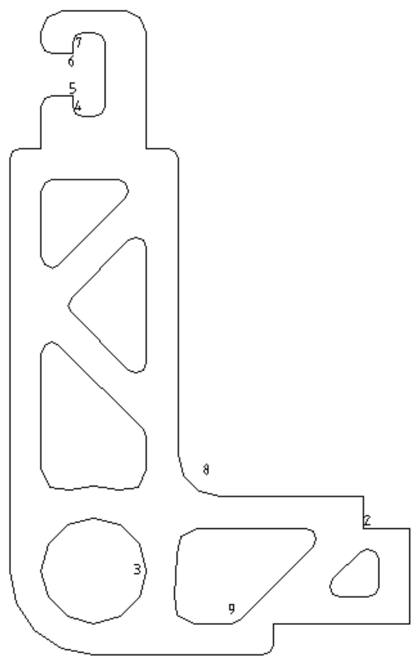
The drawing contains a lever, which is the basis for your calculations.

2 Zoom in to the lever.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Window. 

Menu	View ➤ Zoom ➤ Window
Command	ZOOM

The complete lever is displayed on your screen.



Save your file under a different name or to a different directory to preserve the original tutorial file.

Regenerate the drawing.

- **Ribbon** None.

  Menu View ➤ Regen All

  Command REGENALL

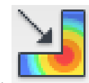
  The drawing is regenerated.

Calculating Stress In Parts

Before you calculate the stress in a part, specify the border conditions.

To specify the border conditions

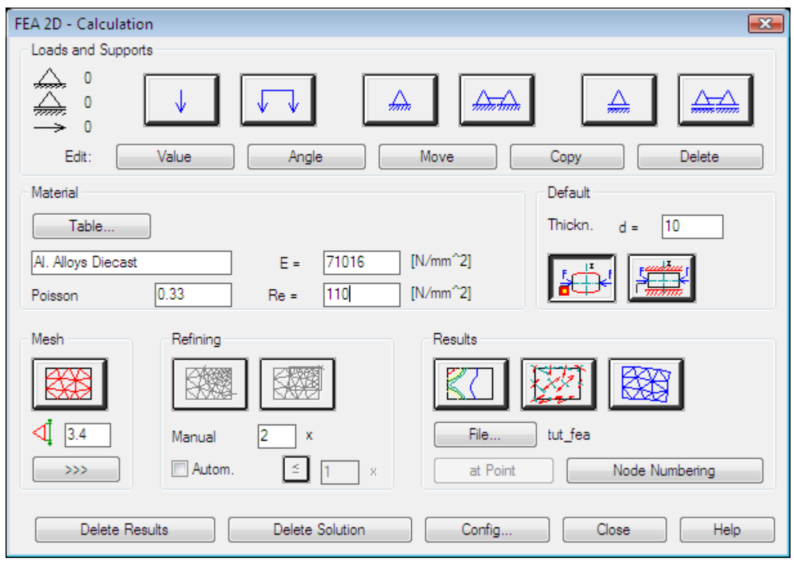
1 Activate the FEA calculation.

Ribbon	Content tab ➤ Calculation panel ➤ FEA. 
Menu	Content ➤ Calculations ➤ FEA...
Command	AMFEA2D

2 Respond to the prompts as follows:

`Specify interior point:` *Specify a point inside the contour*

The FEA 2D Calculation dialog box opens so that you can define border conditions and perform calculations.



Select the thickness and the material of the lever.

3 In the Default section, specify a thickness of *10*.

4 In the Material section, click Table.

5 In the Select Standard for Material dialog box, in the Details panel, click ANSI Material, and from the Select Material Type dialog box, select Al. Alloys Diecast.

6 Click the Config button, and in the FEA Configuration dialog box, and specify:

Scale Factor for Symbols: *0.1*

7 Click OK to return to the FEA 2D - Calculation dialog box.

Defining Loads and Supports

To perform calculations, you need to define the loads and supports.

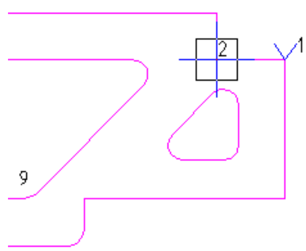
To specify loads and supports

1  Click the fixed line support button, and respond to the prompts as follows:

Specify insertion point <Enter=Dialogbox>: *Specify point (1)*

Specify endpoint: *Specify point (2)*

Specify side from endpoint: *Specify a point above the contour*

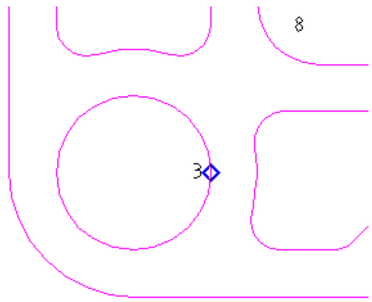


2  Click the movable line support button, and respond to the prompts as follows:

Specify insertion point <Enter=Dialogbox>:

Hold down SHIFT*, right-click and click Quadrant, specify point (3)*

Specify endpoint: *Press* ENTER *to define the starting point as the endpoint*



3  Click the line force button, and respond to the prompts as follows:

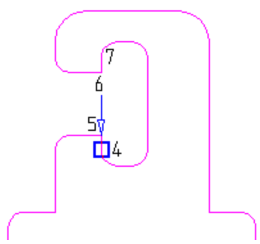
`Specify insertion point <Enter=Dialogbox>:` *Specify point (5)*

`Specify endpoint:` *Specify point (4)*

`Specify side from endpoint:`

Specify a point to the right of the specified points

`Enter a new value <1000 N/mm>:` *Enter 500, press* ENTER



4  Click the line force button again, and respond to the prompts as follows:

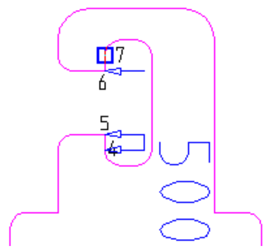
`Specify insertion point <Enter=Dialogbox>:` *Specify point (6)*

`Specify endpoint:` *Specify point (7)*

`Specify side from endpoint:`

Specify a point to the right of the specified points

`Enter a new value <1000 N/mm>:` *Enter 500, press* ENTER



Calculating Results

Before you calculate the results, generate a mesh.

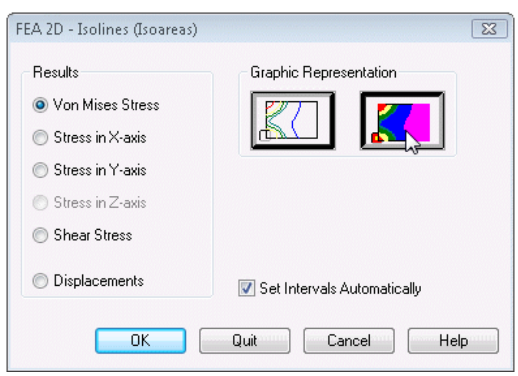
NOTE If you calculate results without creating a mesh in advance, the mesh will be created automatically.

To calculate the results

1.  In the Mesh section, click the mesh button, and then press ENTER to return to the dialog box.

2.  In the Results section, click the isolines (isoareas) button.

3. In the FEA 2D Isolines (Isoareas) dialog box, select the Graphic -Representation button on the right.

   Click OK.

4. Respond to the prompts as follows:

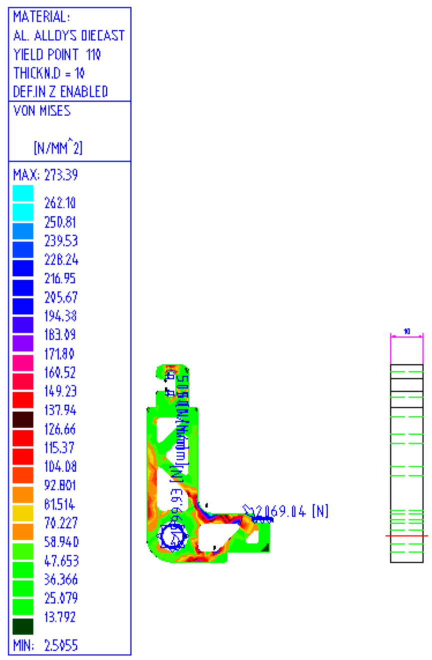
Specify base point <Return = in boundary>:

Press ENTER *to place the isoareas in the boundary*

Insertion point: *Select a point to place the table to the left of the part*

<Return>: *Press* ENTER *to return to the dialog box*

The result looks like this:

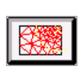


After calculation, the support forces are displayed near the support symbol.

Evaluating and Refining Mesh

The stress table allocation relative to the lever shows heavy concentration of local stress near drawing points 8 and 9. Refine the mesh near these points to obtain more exact calculation results for the points of interest.

To refine the mesh

1  In the Refining section, click the left refining button, and respond to the prompts as follows:

Specify center point 1 <Return=Continue>:

Specify a point near point 8

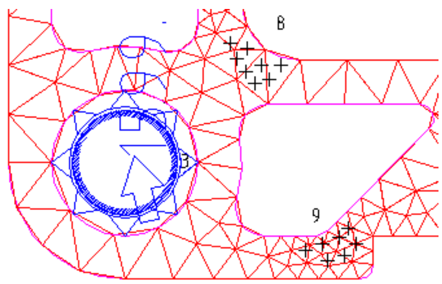
Specify center point 2 <Return=Continue>:

Specify a point near point 9

Specify center point 3 <Return=Continue>:

Press ENTER *to continue meshing*

<Return>: *Press* ENTER *to return to the dialog box*

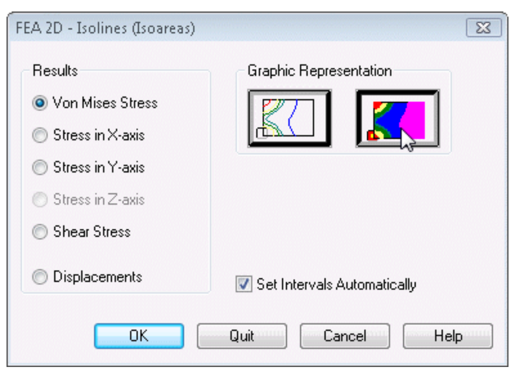


The mesh is refined at the specified points.

Recalculate the stress representation.

2  Click the isolines (isoareas) button.

3 In the FEA 2D Isolines (Isoareas) dialog box, click the Graphic Representation button on the right.



Click OK

4 Respond to the prompts as follows:

`Specify base point <Return = in boundary>:` *Press* ENTER

`Insertion point:` *To the left of the part, select a location for the table*

`<Return>:` *Press* ENTER *to return to the dialog box*

Refining Designs

The results show a critical area around point 8 that can be improved by applying a larger radius. Before changing the geometry, the results and solutions have to be deleted.

To edit the geometry

1 Click the Delete Solution button.

2 In the AutoCAD Question dialog box, click Yes to delete the solutions and results.

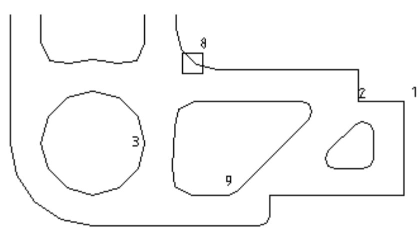
3 In the AutoCAD Question dialog box, click No to keep the loads and supports.

4 To change the radius, start the Power Edit command.:

Ribbon	None.
Menu	Modify ➤ Power Edit
Command	AMPOWEREDIT

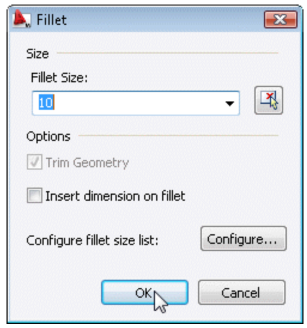
Respond to the prompt as follows:

`Select object:` *Select the arc segment at point 8*

5 

6 In the Fillet dialog box, specify:

`Fillet Size:` 10



Click OK.

7 Respond to the prompt:

`Select objects:` *Press* ENTER *to exit the command*

The radius of the fillet is changed to 10.

8 Zoom to the extents of the drawing.

Ribbon	View tab ➤ Navigate panel ➤ Zoom drop-down  ➤ Extents.
Menu	View ➤ Zoom ➤ Extents
Command	ZOOM

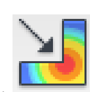
Save your file.

Recalculating Stress

Before recalculating the stress division of the lever, calculate and display the deformation.

To calculate the stress

1 Restart the FEA routine.

Ribbon	Content tab ➤ Calculation panel ➤ FEA. 
Menu	Content ➤ Calculations ➤ FEA...
Command	AMFEA2D

2 Respond to the prompts as follows:

`Specify interior point:` *Specify a point inside the contour*

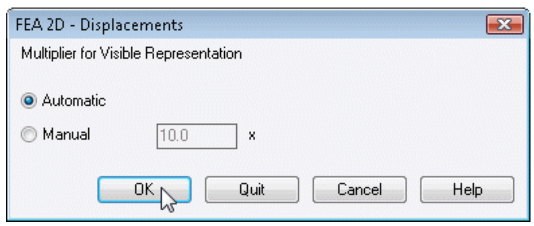
Select the thickness and the material of the lever again, as you did it before.

3 In the Default section, enter a thickness of *10*.

4 Click Table, and select the material from your preferred standard table. Select Al. Alloys Diecast if you prefer to use ANSI materials.

5  Click the deformation button in the Results field.

6 In the FEA 2D - Displacements dialog box, Click OK.



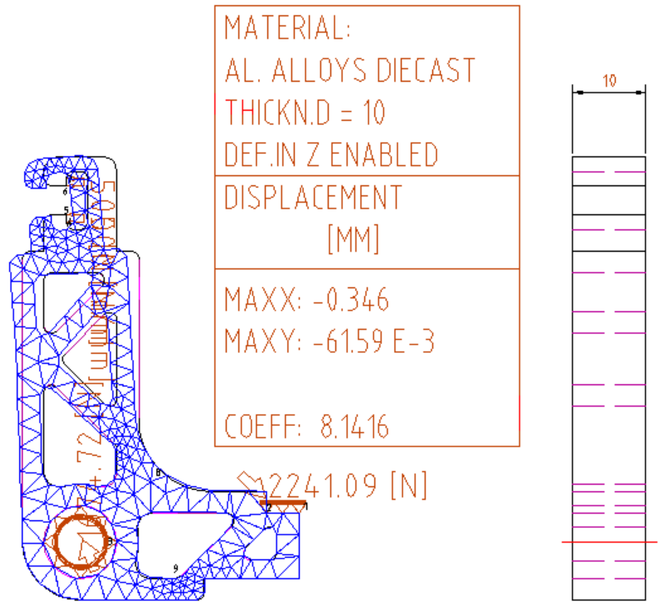
7 Respond to the prompts as follows:

Specify base point <Return = in boundary>: *Press* ENTER

Insertion point: *To the right of the part, select a location for the table*

<Return>: *Press* ENTER *to return to the dialog box*

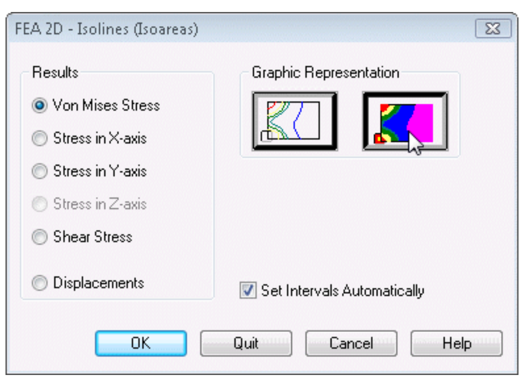
The result looks like this:



Recalculate the stress division of the lever.

1  Click the isolines (isoareas) button.

2 In the FEA 2D Isolines (Isoareas) dialog box, click the Graphic Representation button on the right.



Click OK.

3 Respond to the prompts as follows:

`Specify base point <Return = in boundary>:` *Press* ENTER

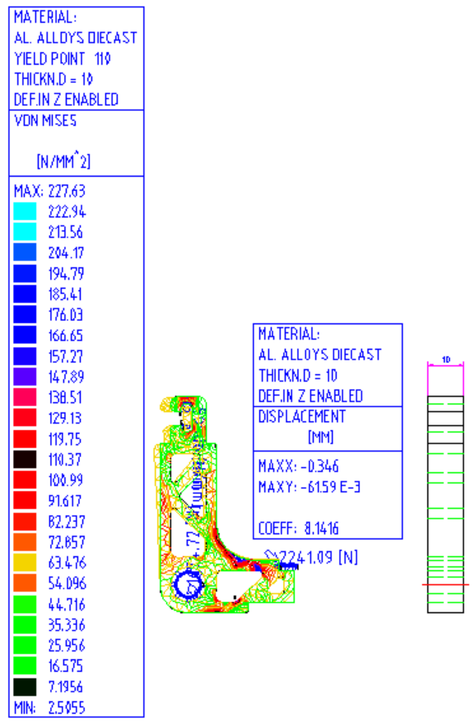
`Specify insertion point:`

To the left of the part, select a location for the table

`<Return>:` *Press* ENTER *to return to the dialog box*

4 Click Close to leave the FEA 2D - Calculation.

The final result looks like this:



NOTE You can return to the FEA 2D - Calculation using Power Edit.

Save your file. This is the end of this tutorial chapter.

Designing and Calculating Cams

18

In this tutorial you use the automated cam design and calculation functionality in AutoCAD® Mechanical to create a cam, perform calculations, and generate data for NC production.

Key Terms

Term	Definition
acceleration	Rate of change in velocity.
cam	Types of gears for obtaining unusual and irregular motions that would be difficult to produce otherwise.
curve path	Geometric shape of the cam.
motion diagram	Diagram illustrating the lift or rotation of the follower for each degree of rotation or translation of the cam plate.
motion section	Part of the motion diagram. Some sections are defined by design. For example, the maximum lift of 15 mm is reached at an angle of 90 degrees.
NC	Numerical Control. Used in manufacturing to represent the control on machine tool motion through numeric data for 2 to 5 axis machining.
resolution	Controls the precision of curves. A low value increases computing time. Use a higher value for initial design.

Term	Definition
step width	Graph of the speed of the straight driven element, or the rotation angle of a rocker and the cam plate angle of rotation.

Designing and Calculating Cams

With the cam design and calculation functionality in AutoCAD Mechanical, you can implement all motions required in the scope of process control with a minimum number of gear elements. The basis for systematic design procedures is offered using standardized laws of movement in the development of new cam gears.

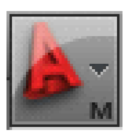
With the automated cam features, you can create cams (linear, circular, and cylindrical cams) based on sections drawn in a motion diagram. You can also calculate velocity and acceleration of an existing section of the motion diagram. The cam curve path can be determined with the calculated cam sections. An existing curve path can be scanned and transferred in the motion diagram. A driven element can be coupled to the cam. NC data can be created using the curve path.

In the following exercise, you generate a circular cam and a swinging follower with a single roller. You also calculate the spring of the follower. The cam and the follower are inserted into the drawing together with the motion diagrams. At the end you generate the NC data for the cam production.

Start with an ISO drawing template.

To open a template

1 Start a new drawing.

Ribbon	 ➤ New ➤ Drawing
Menu	File ➤ New
Command	NEW

2 The Select template dialog box opens. Select the template *am_iso.dwt* and click Open. This creates a new drawing based on the *am_iso* template. Use Save As to save the drawing file with an appropriate name.

Starting Cam Designs and Calculations

To start a cam design and calculation

1 Open the cam design and calculation tool.

Ribbon	Content tab ➤ Calculation panel drop-down ➤  Cam Design and Calculation.
Menu	Content ➤ Cams...
Command	AMCAM

Specify the cam type.

2 In the Cam Design and Calculation dialog box, on the Cam tab, specify:

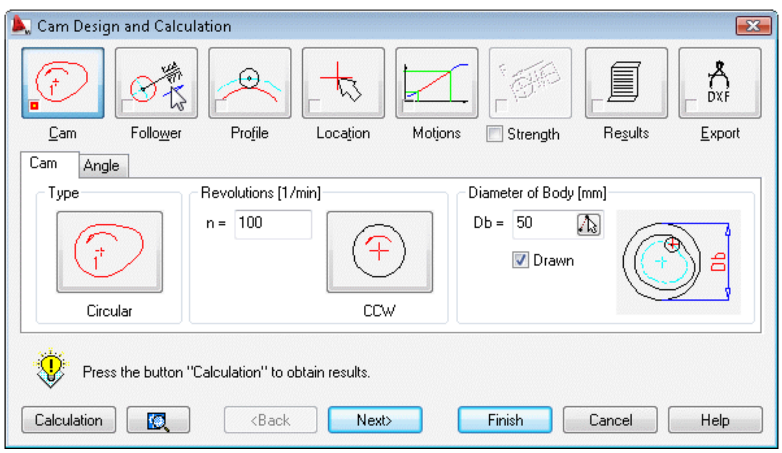
`Type: Circular`

In the Type of Cam dialog box, click the center Circular icon and specify:

`Revolutions [1/min]:` *100*

`Drawn:` *Select the check box*

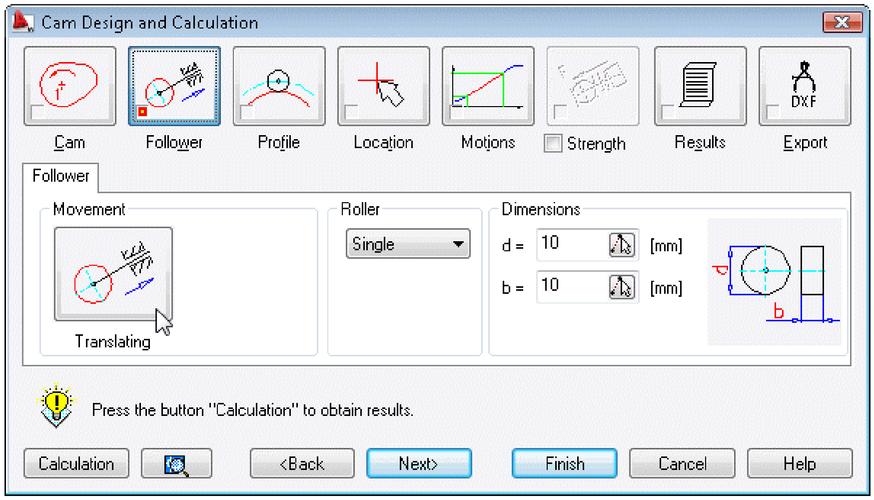
`Diameter of Body [mm]:` *50*



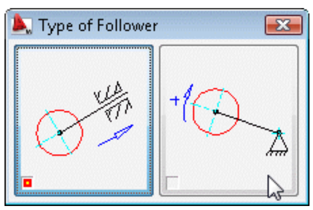
3 Click the Follower button.

NOTE You can also step through the dialog using the Next button.

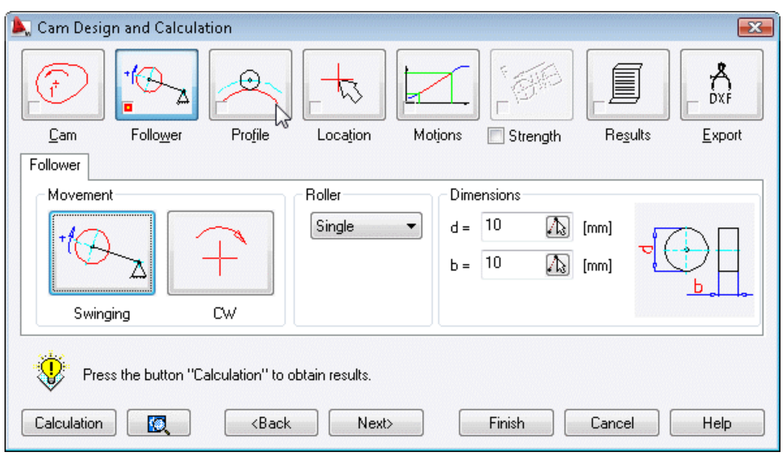
4 On the Follower tab, Movement section, click the Translating button.



5 In the Type of Follower dialog box, click the Swinging button. You are returned to the CAM Design and Calculation dialog box.



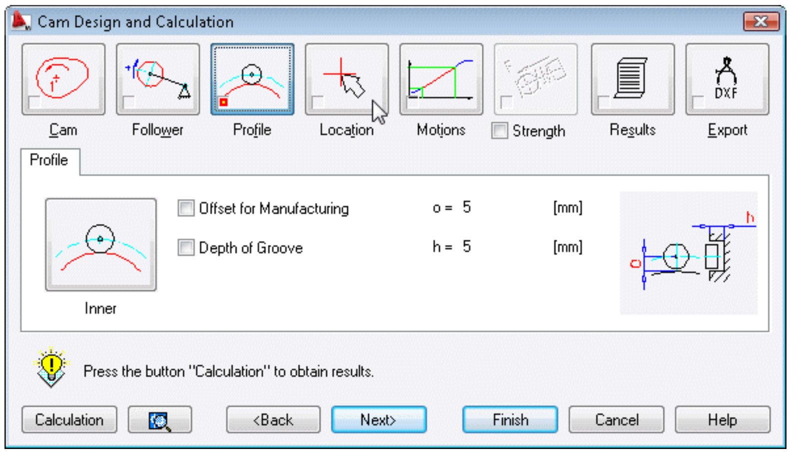
Specify the following settings.



6 Click the Profile button, and define the profile.

You can select between a power-contact profile (inner or outer) or a form-contact profile (both outer). Specify an inner profile, which requires a spring to keep contact.

Specify the following settings.



7 Click the Location button.

The dialog box is hidden so you can specify a location for the cam and the follower in the drawing.

8 Respond to the prompts as follows:

`Specify center of cam:` 100,100, *press* ENTER

`Specify center of follower swing [Undo]:` @100,0, *press* ENTER

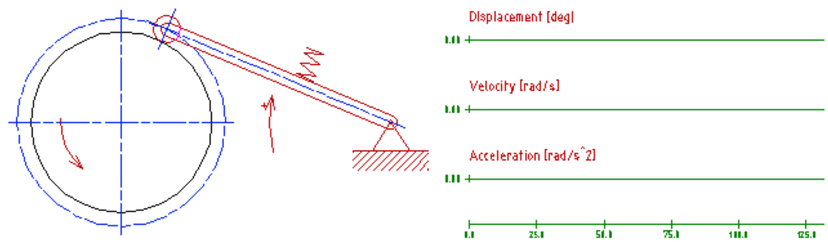
`Specify start of movement [Undo]:` @90<157.36, *press* ENTER

`Specify origin of movement diagram [Undo/Window] <Window>:`

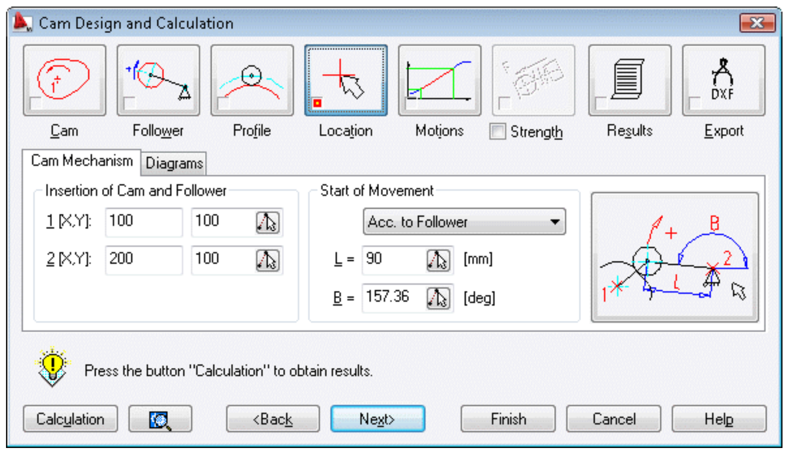
Specify a point next to the cam

`Specify length of movement diagram [Undo]:` @360,0, *press* ENTER

The cam and the follower are inserted into the drawing with the motion diagram. Your drawing looks like this:



The Cam Design and Calculation dialog box is opened again.

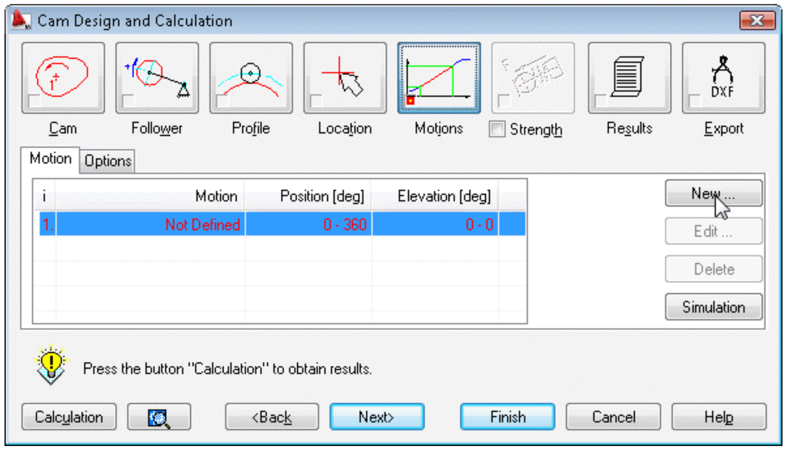


Defining Motion Sections

Define five motion sections to describe the cam.

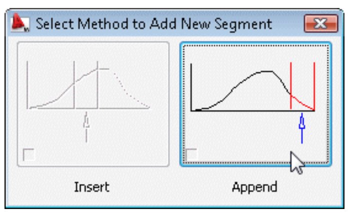
To specify motions

1 In the Cam Design and Calculation dialog box, click the Motions button, and then click the New button.



In the Select Method to Add New Segment dialog box, you can either insert or append a new motion section.

2 Click Append.

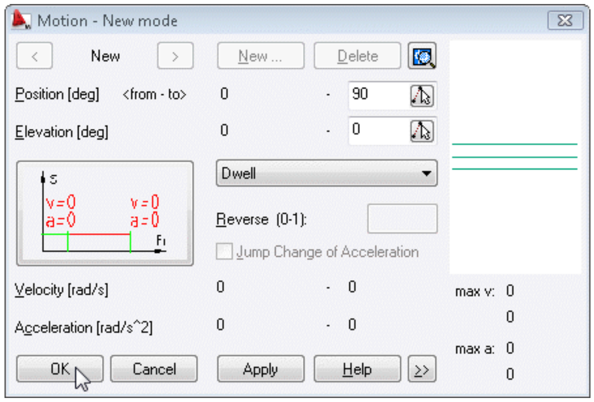


Define the first motion section.

3 In the Motion - New mode dialog box, specify the following settings.

`Position [deg] <from - to> 0 -:` *90*

`Elevation [deg] 0 -:` *0*

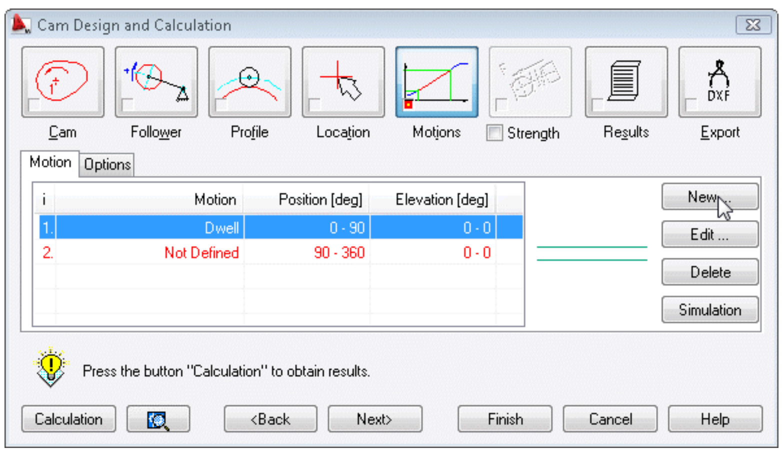


Click OK.

The motion is inserted into the drawing and you are reverted back to the Cam Design and Calculation dialog.

Define the next motions to describe the cam.

1 In the Cam Design and Calculation dialog box, Motion tab, click New.

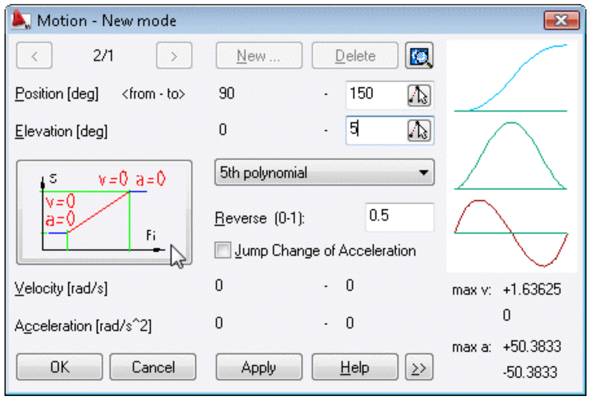


2 In the Select Method to Add New Segment dialog box, click Append.

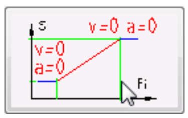
3 In the Motion - New mode dialog box, specify the following settings.

`Position [deg] <from - to> 90 -:` *150*

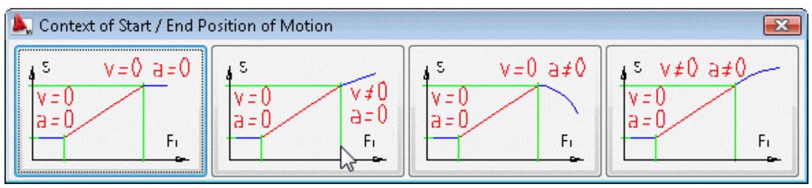
`Elevation [deg] 0 -:` *5*



4 Click the Context of Follower movement button.



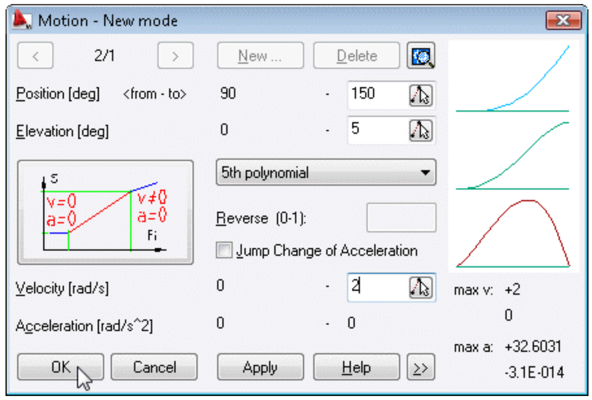
5 Click Dwell - Constant Velocity (second button from left).



6 In the Motion - New mode dialog box, specify the following settings.

`Curve:` 5th polynomial

`Velocity [rad/s] 0 -:` *2*



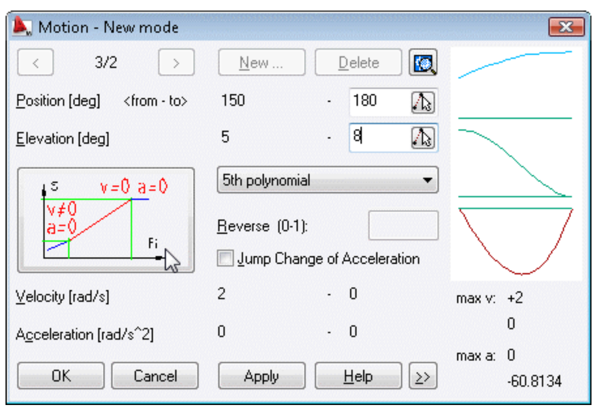
Click OK.

The next motion section has to be ‘Constant Velocity,’ since the motion section before is ‘Dwell - Constant Velocity’.

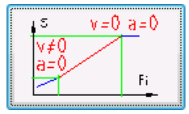
1. In the Cam Design and Calculation dialog box, Motion tab, click New.
2. In the Select Method to Add New Segment dialog box, click Append.
3. In the Motion - New mode dialog, specify the following settings.

   `Position [deg] <from - to> 150 -:` *180*

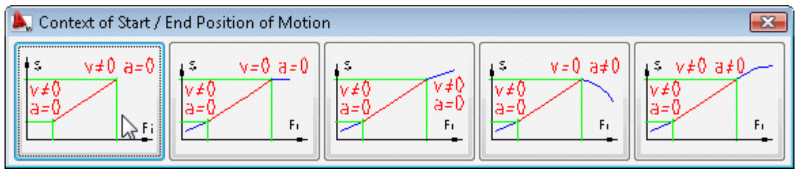
   `Elevation [deg] 5 -:` *8*



4. Click the Context of Follower movement button.



5 Click Constant Velocity (left most button).



The routine recalculates the elevation and inserts the correct value, 10.73, in the Elevation box of the Motion New mode dialog box.

Click OK.

Define the next motion section.

1 In the Cam Design and Calculation dialog box, Motion tab, click New.

2 In the Select Method to Add New Segment dialog box, click Append.

3 In the Motion - New mode dialog box, specify the following settings.

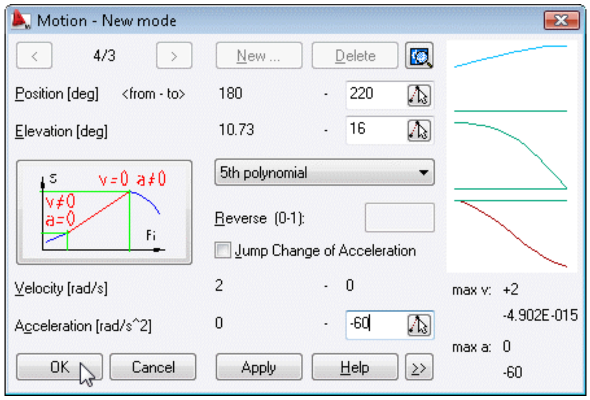
`Position [deg] <from - to> 180 -:` *220*

`Elevation [deg] 10.73 -:` *16*

4 Click the Context of Follower movement button, and then click Constant Velocity - Reverse (fourth button from left).

5 In the Motion - New mode dialog box, specify the following settings.

`Acceleration [rad/s^2] 0 -:` *60*



Click OK.

Define the last motion section to complete the 360 degrees.

1 In the Cam Design and Calculation dialog box, Motion tab, click New.

2 In the Select Method to Add New Section dialog box, click Append.

3 In the Motion - New mode dialog, specify the following settings.

`Position [deg] <from - to> 220 -:` *360*

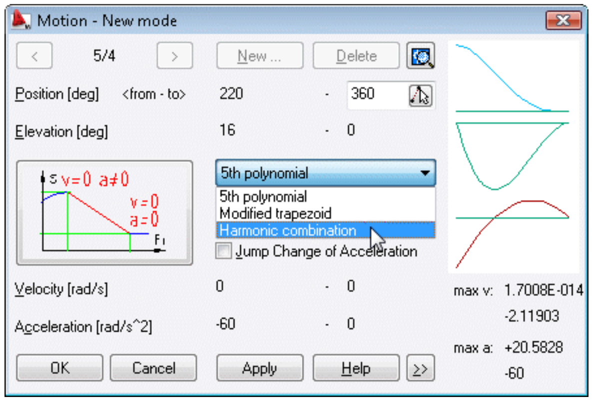
`Elevation [deg] 16-:` *0*

4 Click the Context of Follower movement button.

The routine calculates the correct values for the end position.

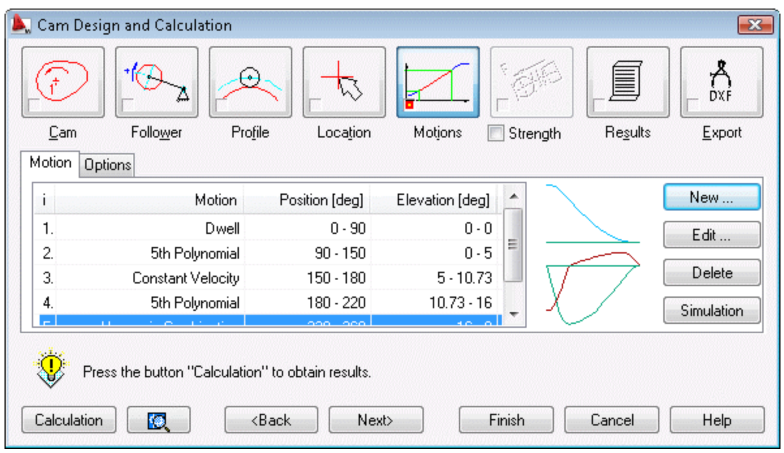
5 In the Motion - New mode dialog box, specify the following settings.

`Curve:` Harmonic Combination



Click OK.

The definition of the motion section is complete, and all motion sections are displayed in the list.



The definition of the geometry is finished.

Calculating Strength for Springs

To calculate the strength for the spring

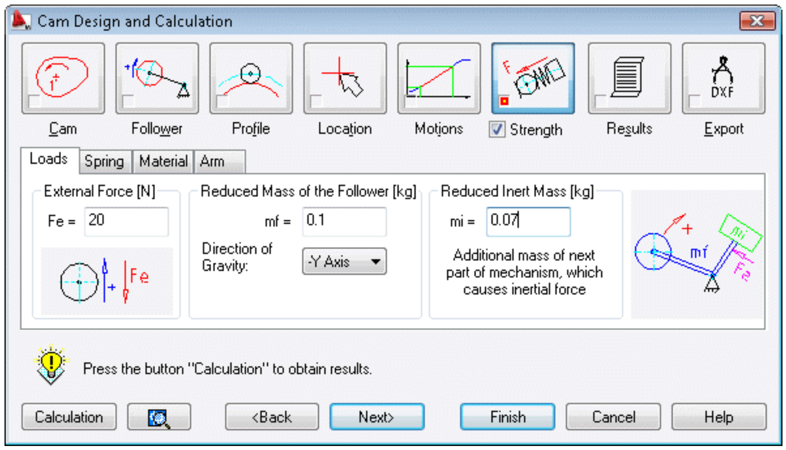
1 In the Cam Design and Calculation dialog box, select the Strength check box, and then click the Strength button.

2 In the Cam Design and Calculation dialog box, Loads tab, specify:

`External Force [N] Fe =:` 20

`Reduced Mass of the Follower [kg] mf =:` *0.1*

`Reduced Inert Mass [kg] mi =:` *0.07*



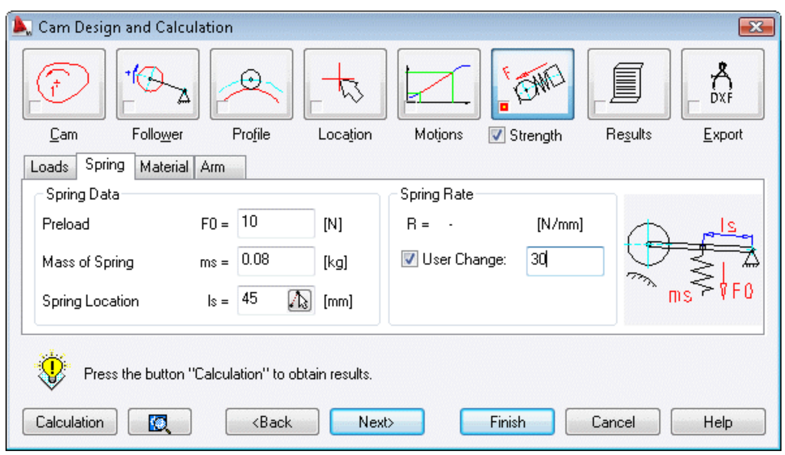
3 On the Spring tab, specify:

`Preload [N] F0 =:` *10*

`Mass of Spring [kg] ms =:` *0.08*

`Spring Location [mm] ls =:` *45*

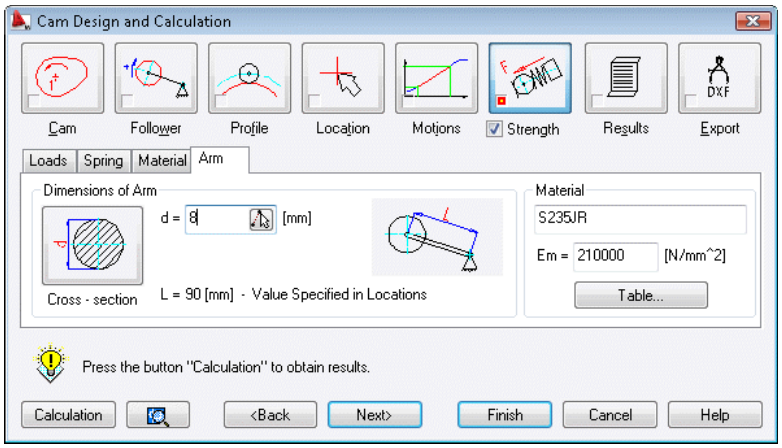
`Spring Rate:` *Select the User Change check box, enter 30*



4 On the Material tab, you can specify the material for cam and roller. In this case, use the default material.

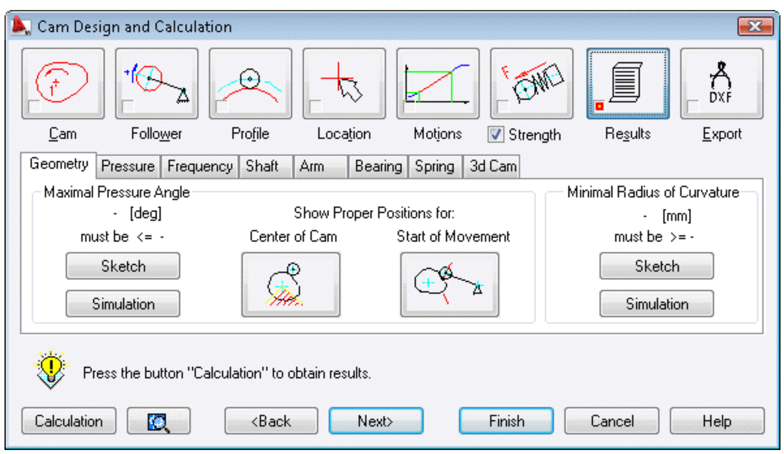
5 On the Arm tab, specify:

`Dimensions of Arm [mm] d =:` *8*



NOTE You can choose other types of cross sections for the arms.

6 Click Results, and then click Calculation.



All calculation results are displayed on the respective tabs:

The Calculation button gives you the results of your design. To optimize your design, you can choose to generate the correct size of the cam based on the pressure angle and the radius of curvature.

To generate a cam design based on pressure angle and radius of curvature

1. Click the Calculation button.

   To optimize the size of the cam, the pressure angle from your design must be less than or equal to a certain value (automatically calculated and displayed at the bottom of the cam Design and Calculation dialog box) while the radius of curvature must be greater than or equal to a certain value (automatically calculated and displayed at the bottom of the cam Design and Calculation dialog box).

2. Click the Results button.
3. In the Geometry tab, click the Center of Cam button.

   Two hatched open triangles are displayed on the screen.

4. Respond to the prompts as follows:

   `Press ESC or ENTER to exit, or [Change center of cam]:`

   Enter C, *press* ENTER

   `Specify center of cam <100,100>:` *Press* ENTER

5. Snap to the apex of the triangle that produces a maximal pressure angle less than or equal to the recommended value and a minimal radius of curvature greater than or equal to the recommended value.

Exporting Cam Data and Viewing Results

To export TXT cam data for an NC machine

1. In the Cam Design and Calculation dialog box, click Export.

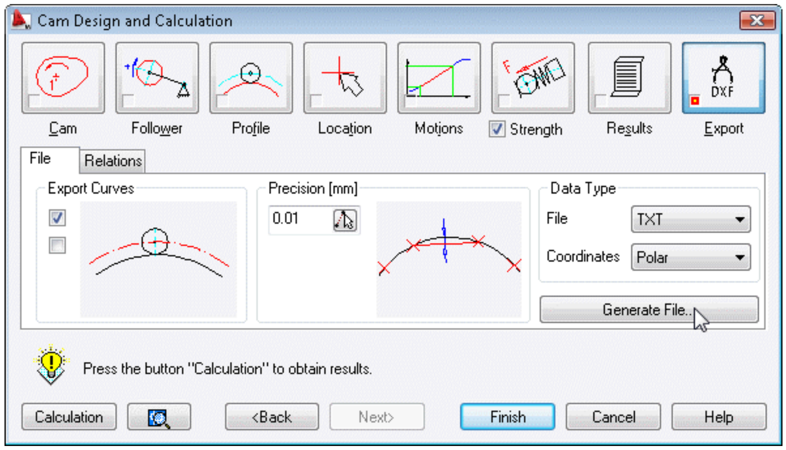
   On the File tab, specify:

   `Export Curves: Inner`

   `Precision [mm]:` *0.01*

   `Data Type: File: TXT`

   Data Type: Coordinates: Polar



Click Generate File.

2 In the Save As dialog box, specify a descriptive file name and a location. Click Save.

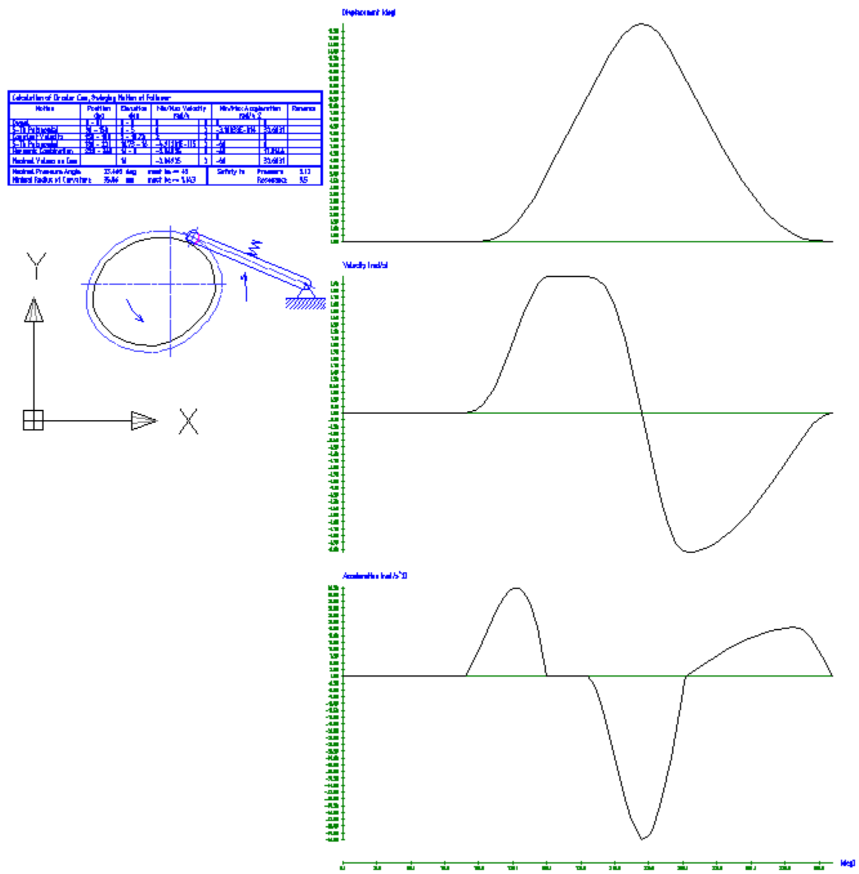
The cam is completely designed and calculated.

3 To view the results, click Finish, and then respond to the prompt as follows:

`Specify insertion point of result table:`

Specify a location for the result table

The table of results is inserted into the drawing.



Save your file. This is the end of the tutorial chapter.

Autodesk Inventor Link

The tutorial in this section teaches you how to import an Autodesk Inventor® file and generate drawing views from them for documentation. The Autodesk Inventor assembly and part drawings required for this tutorial are available in the *Acadm/tutorial/tut_bracket* folder of the AutoCAD® Mechanical installation folder.

Using Autodesk Inventor Link Support

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In this chapter, you learn how to enable AutoCAD® Mechanical to create views and documentation for Autodesk Inventor® assemblies and parts.

Key Terms

Term	Definition
parametric dimensions	A type of dimension associated with an Autodesk Inventor model. Parametric dimensions control the size and positions of geometry. If the dimension value is changed, the size and position of the geometry adjusts to reflect the new value. Parametric dimensions can be changed only from Autodesk Inventor.
power dimensioning	A command useful for generating linear, radial, and diameter dimensions, which minimizes the number of individual actions while generating a dimension. Power dimensioning automatically selects the type of linear dimension (horizontal, vertical, or aligned), based on the selected point.
reference dimensions	A type of dimension that indicates the size and position of geometry. Reference dimensions do not control the geometry size or position, but follow them instead. The type of dimensions created with power dimensioning commands are reference dimensions.

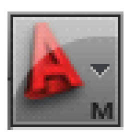
Term	Definition
template	A file with predefined settings to use for new drawings. However, any drawing can be used as a template.
title block	A title block contains a series of attributes. Some already have values. The preassigned values can be modified, and the vacant attributes can be completed with new values.
viewport	A scaled view of the model defined in a layout.
view scale	The scale of the base drawing relative to the model scale. Also, the scale of dependent views relative to the base view.

Linking Autodesk Inventor Part Files

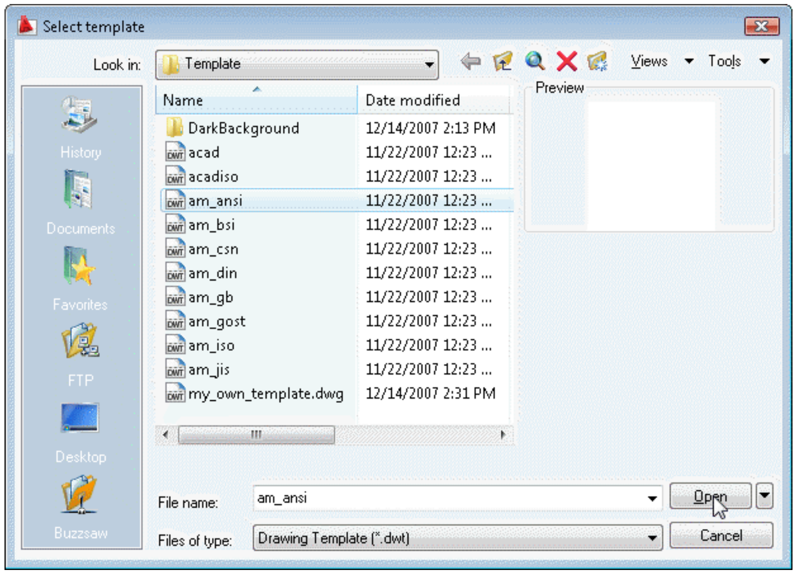
With Autodesk Inventor link support, you can create views of Autodesk Inventor part files while maintaining full model to drawing associativity. Note: You can perform the exercises in this chapter only if AutoCAD Mechanical is installed with the Install Autodesk Inventor link option enabled. Also, before you begin the exercises, download and install Visual Basic for Applications (VBA) at http://autodesk.com/vba-download.

To link an Autodesk Inventor part file

1 Open the Inventor part file.

Ribbon	 ➤ New ➤ Inventor Link
Menu	File ➤ New Inventor Link
Command	NEW

2 In the Select template dialog box, select the template *am_ansi.dwt*, then click Open.



3 In the Link Autodesk Inventor File dialog box, locate the *Tut_Bracket\Bracket Components* folder, within the folder containing tutorial files at:

- **Windows Vista®**: *C:\Users\Public\Public Documents\Autodesk\ACADM 2010\Acadm\Tutorial*
- **Windows®XP**: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2010\Acadm\Tutorial*

4 Click *Holder Bracket.ipt*, then click Open.

Shading and Rotating Geometry

The commands to shade and rotate geometry are on the Mechanical Browser's right-click menu. If the Mechanical Browser is not visible:

➤ On the command line, enter AMBROWSER and respond to the prompts as follows.

```
Desktop browser? [ON/OFf] <ON>:
```

Enter ON, press ENTER

To shade a part

- Right-click the root node of the Mechanical Browser and click Shade. Use the 3D Orbit tool to rotate the part.

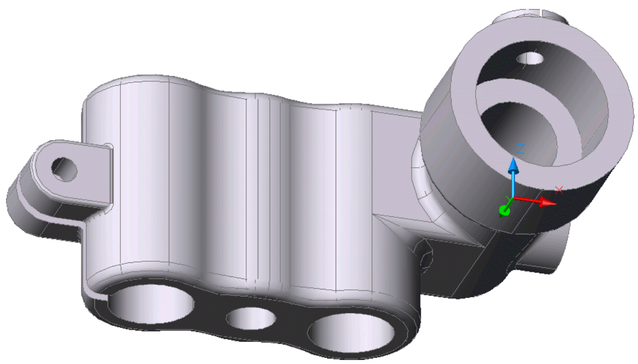
To rotate a part

1 Right-click the root node of the Mechanical Browser and click 3D orbit.

2 Right-click in the drawing area again and select Other Navigation Modes ➤ Free Orbit.

3 Place the cursor in the appropriate location inside or on the Arcball.

4 Click and hold the left mouse button, then rotate the part to a position that resembles the following illustration.



5 Right-click, then click Exit from the menu.

Inserting Drawing Borders

To insert a drawing border

1 Click the Drawing tab in the Mechanical Browser.

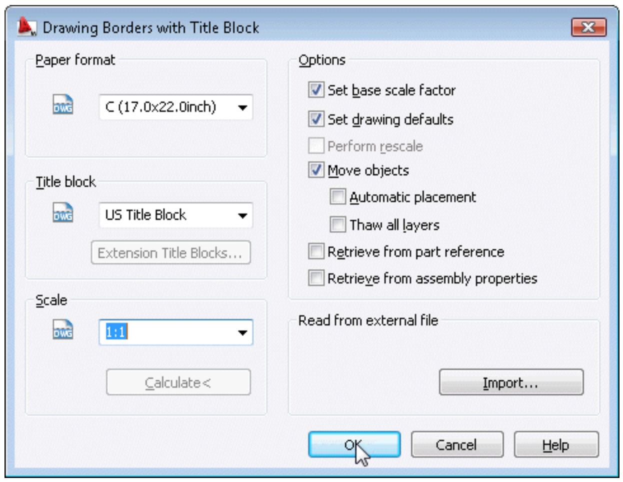
2 Start the Drawing Title/Borders command.

Ribbon	Annotate tab ➤ Layout panel ➤ 
Menu	Annotate ➤ Drawing Title and Revision ➤ Drawing Title/Borders...

Command	AMTITLE

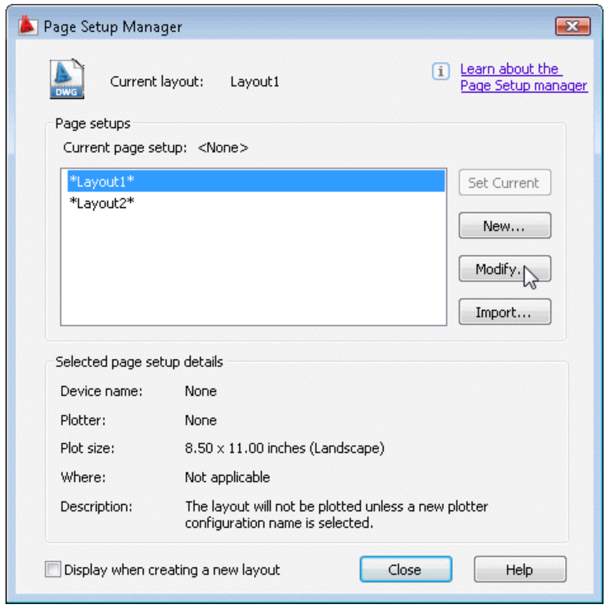
3 In the Drawing Borders with Title Block dialog box, specify:

```
Paper Format: C (17.0x22.0 inch)

Title Block: US Title Block

Scale: 1:1
```



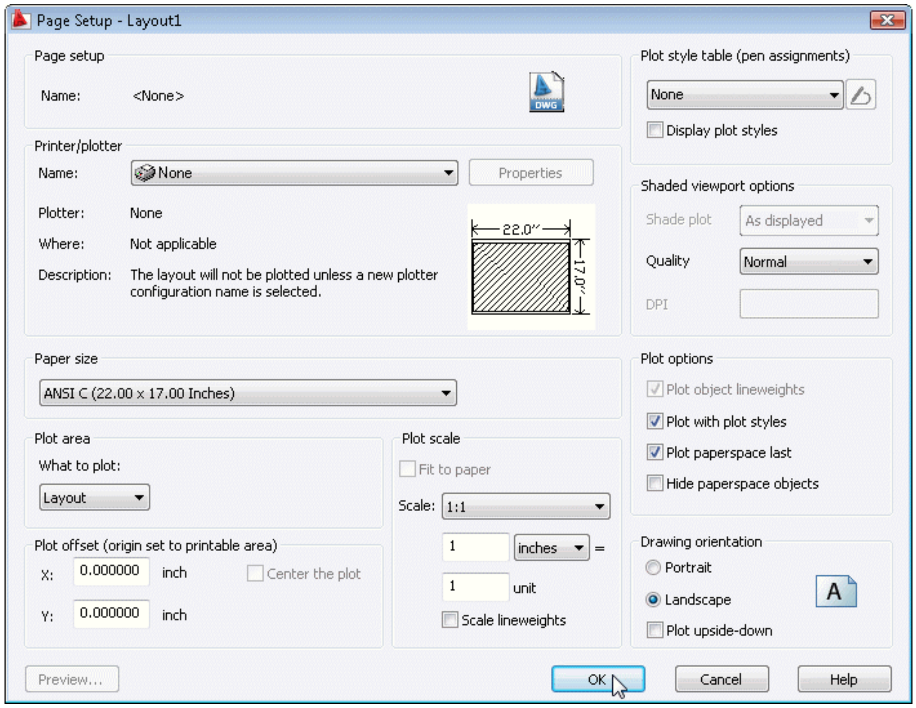
4 Choose OK.

5 In the Page Setup Manager dialog box, select Layout1, then click Modify.



6 In the Page Setup - Layout1 dialog box, specify the following value:

```
Paper size: ANSI C (22.00 x 17.00 Inches)
```



7 Click OK to exit the Page Setup Manager.

8 Click Close.

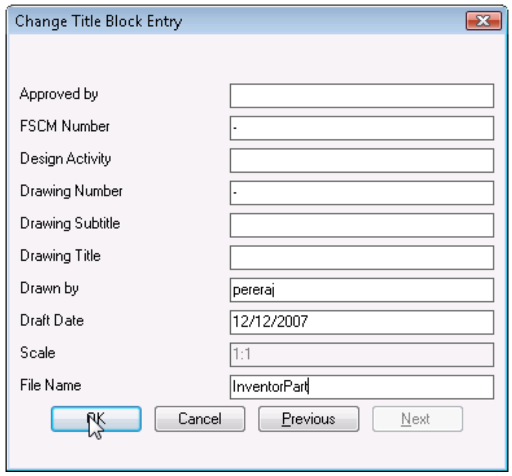
9 Respond to the prompt as follows:

`Specify insertion point:` *Enter -0.25,-0.75, press* ENTER

10 In the Change Title Block Entry dialog box, click Next.

11 In the next page specify:

`File Name:` InventorPart



12 Choose OK.

13 In the Save Title Block Filename dialog box, verify the following settings:

`File Name:` InventorPart.dwg

`File of Type:` Drawing (**.dwg*)

14 Choose Save.

Creating Drawing Views

You can create a variety of drawing view for a part. Any changes made to the part in Autodesk Inventor are automatically updated in the drawing views, when the *.dwg* file is updated.

When you create a drawing view, the link reads parametric dimensions from the model and adds them to the view.

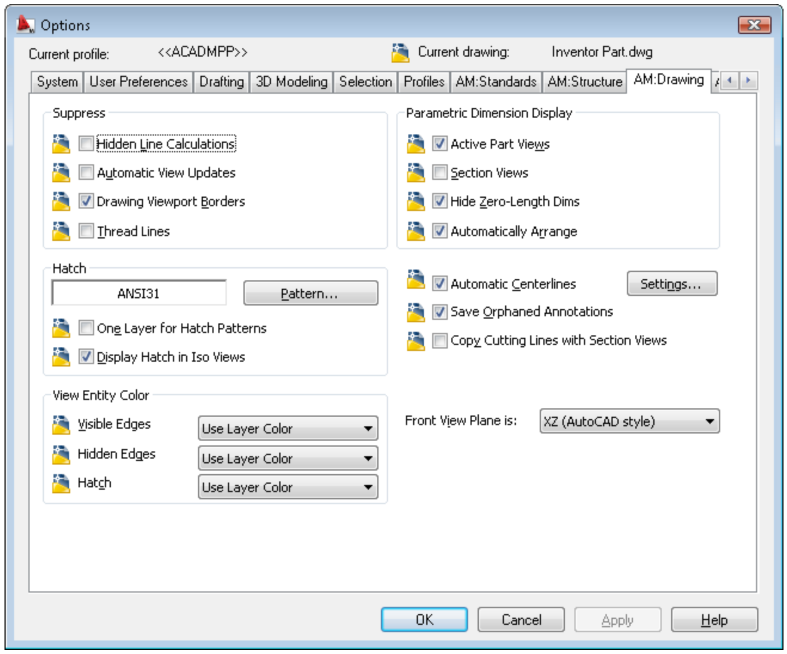
To enable creation of parametric dimensions

1 Open the Options dialog box.

Ribbon	None ➤ None ➤
Menu	Tools ➤ Options
Command	OPTIONS or AMOPTIONS

2 Select the AM:Drawing tab.

3 Under Parametric Dimension Display, select the Active Part Views check box.



4 Choose OK.

To create a base view

1 Create a new drawing view

Ribbon	None
Menu	Drawing Default ➤ New View
Browser	Right-click a Layout icon in the Drawing tab, then choose New View.
Command	AMDWGVIEW

2 In the Create Drawing View dialog box, specify:

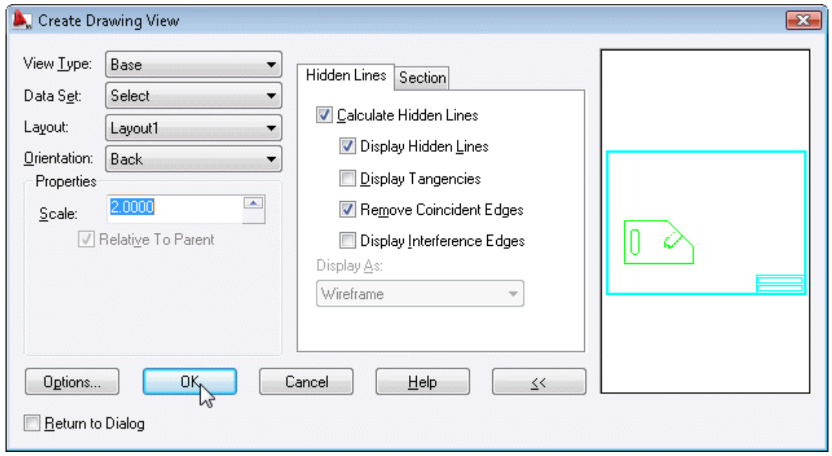
`View Type:` Base

`Data Set:` Select

`Layout:` Layout1

`Orientation:` Back

`Scale:` 2.0000

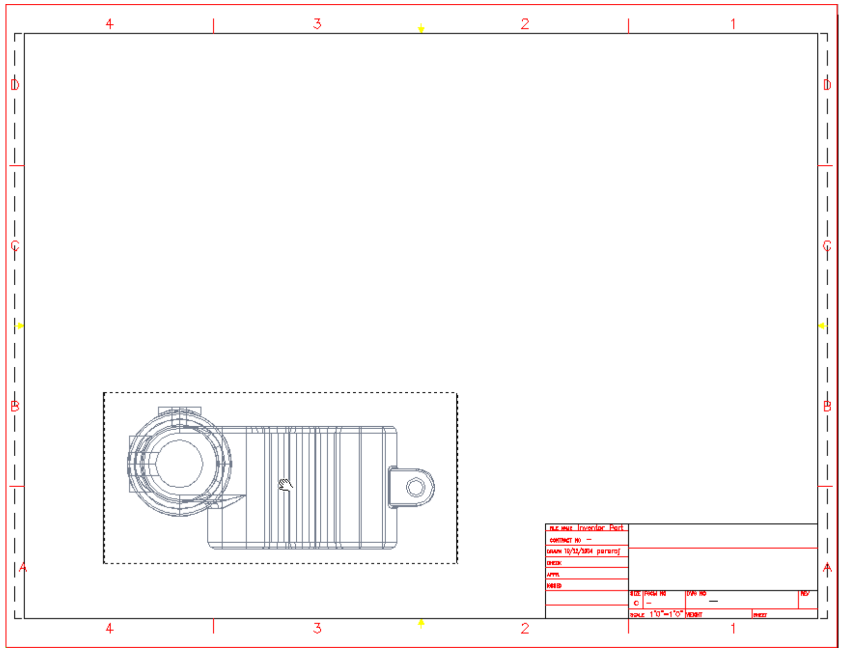


3 Choose OK.

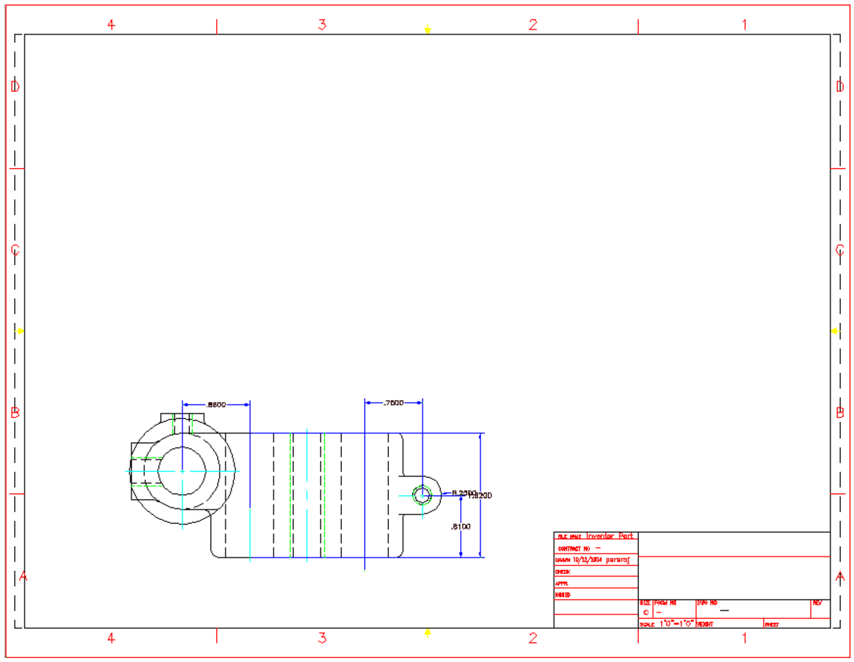
4 Respond to the prompts as follows:

`Specify location for base view:`

Click in the lower left corner of the graphics area, press ENTER



The base view is placed in the lower-left corner of the drawing. Parametric dimensions extracted from the Inventor Part file are displayed.



Create an orthogonal view from the base view.

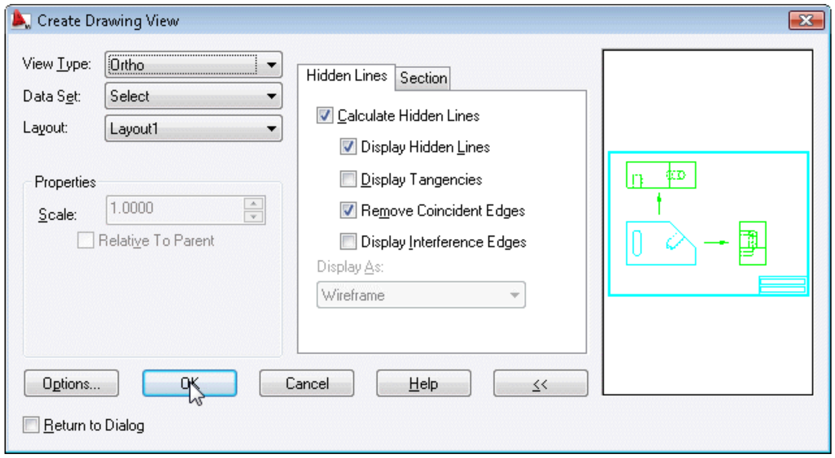
To create an orthogonal view type

1 Create a new drawing view..

Ribbon	None
Menu	Drawing Default ➤ New View
Browser	Right-click the Base icon in the Drawing tab, then choose New View.
Context Menu	Right-click in the graphics area, then choose New View.
Command	AMDWGVIEW

2 In the Create Drawing View dialog box, specify:

`View Type:` Ortho



3 Choose OK.

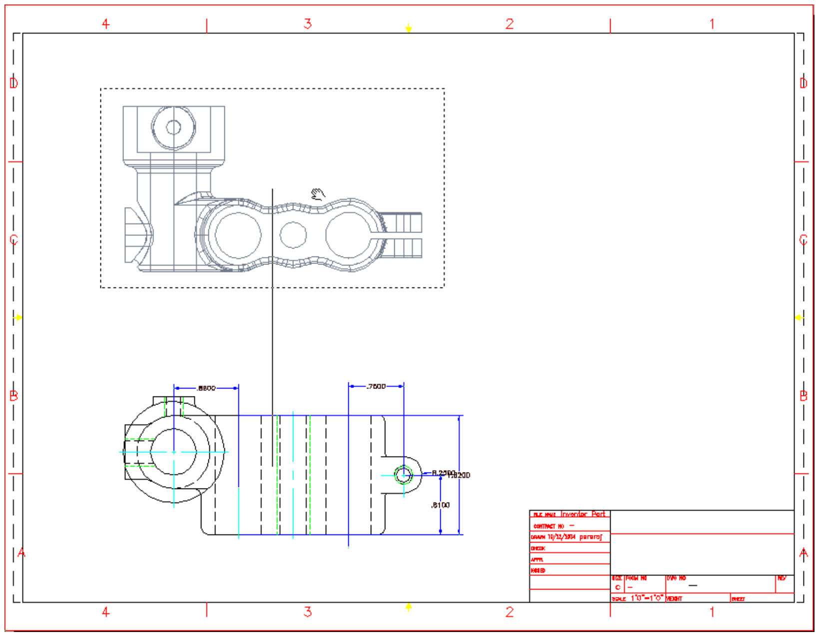
4 Respond to the prompts as follows:

`Select parent view:` *Select the base view*

`Specify location for orthogonal view:`

Drag to a location above the base view, click to select location

`Specify location for orthogonal view:` *Press* ENTER



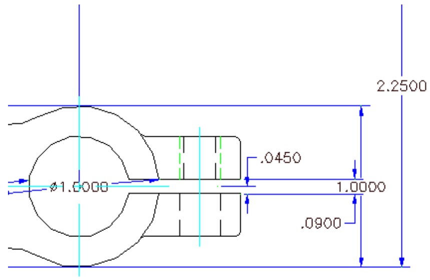
The view is placed in the upper-left corner of the drawing. Parametric dimensions read from the Autodesk Inventor part are displayed

Working with Dimensions

Some of the dimensions need rearranging, while a few may be redundant. You may also need to create dimensions for some entities. Dimensions you add yourself are called reference dimensions. If the part is modified in Autodesk Inventor, these dimensions automatically display the correct part size.

To delete a parametric dimension

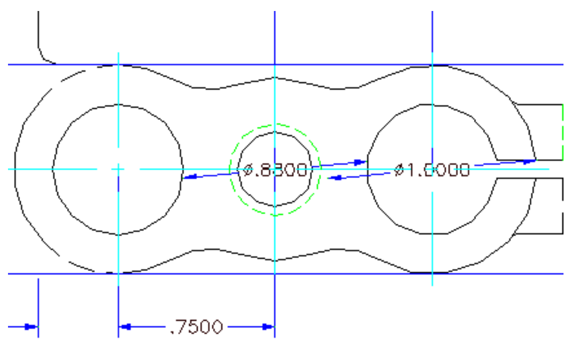
➤ In the orthogonal view, click the dimension that reads .0450 and press DELETE. The dimension is deleted.



NOTE The dimension you deleted may have been entered as a sketch dimension originally, and extruded later resulting in the redundancy of dimensions.

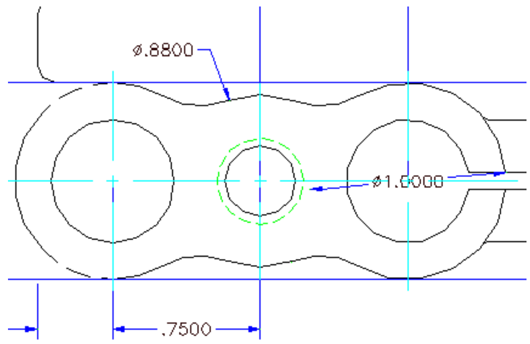
To move dimensions

1 Click the diameter dimension of 0.8800.

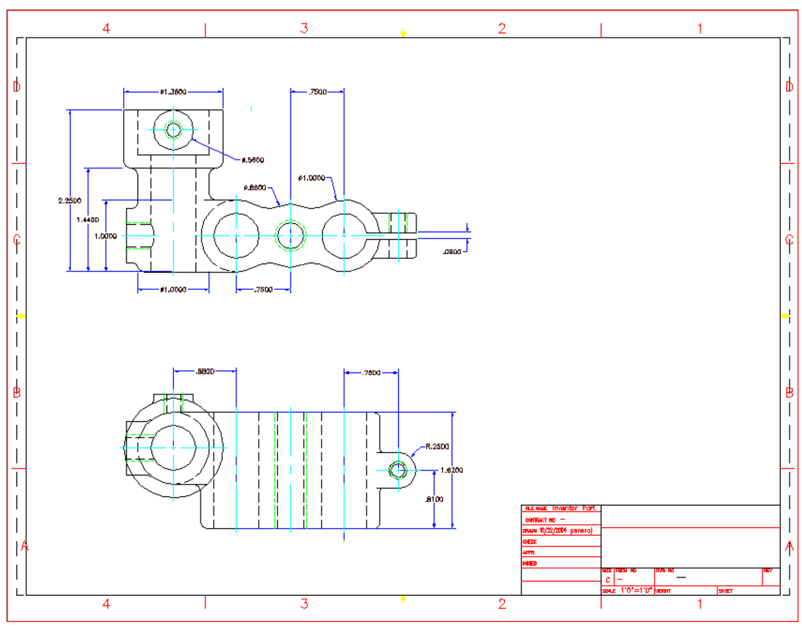


Three grip points are displayed on the dimension.

2 Drag the middle grip outside the bracket, and click. The dimension should be displayed as shown in the following image.



3 You may want to rearrange all the dimensions to tidy up the drawing view.



To add a hole note

1 Start the Leader note command.
Ribbon

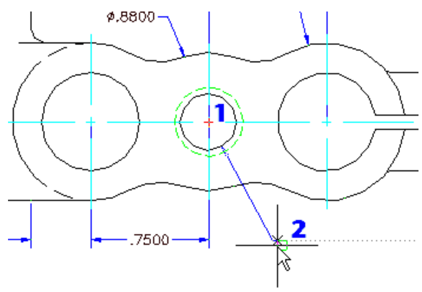
Annotate tab ➤ Symbol panel ➤ 

Menu	Annotate ➤ Leader Note
Command	AMNOTE

2 Respond to the prompts as follows:

```
Select object to attach [rEorganize]:
```

In the orthogonal view, click the center of the hole in the middle (1), drag to a placement point (2), click and press ENTER



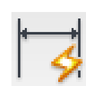
The Note Symbol ANSI dialog box is displayed.

3 Click OK. The Hole Note is added.

NOTE The note text is automatically generated with details extracted from the part file.

To create a vertical reference dimension

1 Start the power dimensioning command.

Ribbon	Annotate tab ➤ Dimension panel ➤ 
Menu	Annotate ➤ Power Dimension
Command	AMPOWERDIM

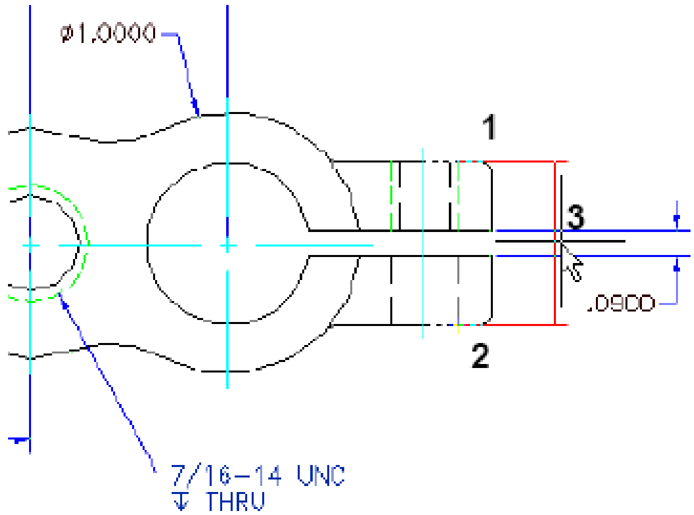
2 Respond to the prompts as follows:

```
Specify first extension line origin or
[Linear/Angular/Radial/Baseline/ Chain/Options/Update] <select
object>:
```

In the orthogonal view, click the end points of the line, (1) and (2)

```
Specify dimension arc line location:
```

Drag the dimension line to the right until it is highlighted in red and click (3)



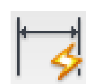
3 In the Power Dimensioning dialog box, click OK.

4 Press ENTER twice to exit the command.

NOTE Parametric dimensions and reference dimensions are shown in different colors.

To create a radial reference dimension

1 Start the power dimensioning command.

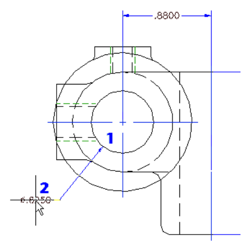
Ribbon	Annotate tab ➤ Dimension panel ➤ 
Menu	Annotate ➤ Power Dimension
Command	AMPOWERDIM

2 Respond to the prompts as follows:

`Specify first extension line origin or [Linear/Angular/Radial/Baseline/ Chain/Options/Update] <select object>:` *Press* ENTER

`Select arc, circle, line or dimension:`

In the base view, click the circle indicating the hole (1), drag the dimension to a placement point (2) and click



3 In the Power Dimensioning dialog box, click OK.

4 Press ENTER twice to exit the command.

Exporting Drawing Views to AutoCAD

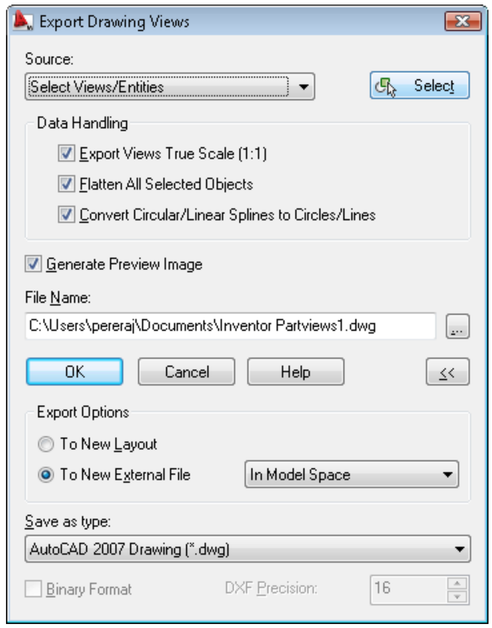
It is possible to export a drawing view of a linked drawing such that it can be viewed in AutoCAD® or AutoCAD® LT.

To export a drawing view

1 Start the Export Views command..

Ribbon	None
Menu	Does not exist in the Menu.
Browser	Right-click the Base icon in the Drawing tab, then choose Export.
Command	AMVIEWOUT

2 In the Export Drawing Views dialog box, from the Source drop-down list, select Select Views/Entities, then click Select.



The dialog box hides.

3 Respond to the prompts:

`Select objects to export <all views>:` *Select the base view*

`Select objects to export <all views>:` *Press* ENTER

You are returned to the Export Drawing Views dialog box.

4 In the File Name box, enter the name of a drawing file to export to.

5 Click OK.

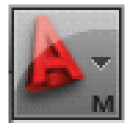
6 Save the file.

7 Close AutoCAD Mechanical, start AutoCAD and open the file that you created in step 5.

Linking Autodesk Inventor Assembly Files

To link an Autodesk Inventor assembly file

1 Open the assembly file.

Ribbon	 ➤ New ➤ Inventor Link
Menu	File ➤ New Inventor Link
Command	NEW

2 In the Select template dialog box, select the template *am_ansi.dwt*, then click Open.

3 Locate and select *Bracket.iam*, then click Open.

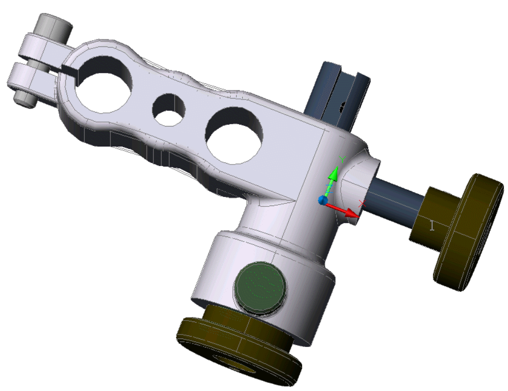
To shade and rotate the assembly

1 Right-click the assembly name in the Mechanical Browser and select Shade.

2 Right-click the assembly name in the Mechanical Browser and select 3D orbit.

3 Place the cursor in the appropriate location inside or on the Arcball.

4 Click and hold the left mouse button, then rotate the part to a position that resembles the following illustration.



Accessing Parts from the Browser

To select a part from the browser

1. Click a part in the Mechanical Browser. The part is highlighted in model space.
2. Right-click a part and select Zoom-to. The display zooms to the part.

Accessing iProperties

When the assembly file is linked, Attached Mechanical is able to access iProperties through its Bill of Materials (BOM).

To access iProperties

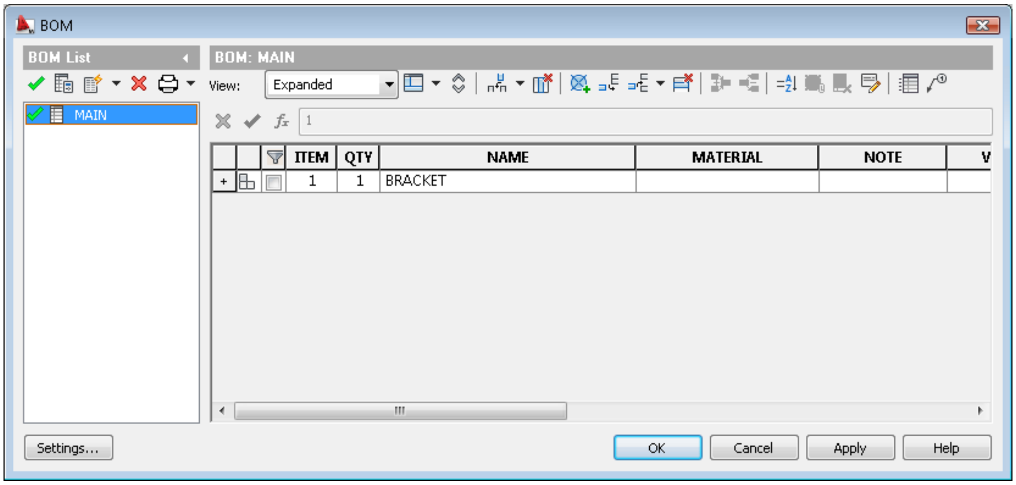
1. Start the BOM Database command.

Ribbon	Annotate tab ➤ BOM panel ➤ 
Menu	Annotate ➤ BOM Database
Command	AMBOM

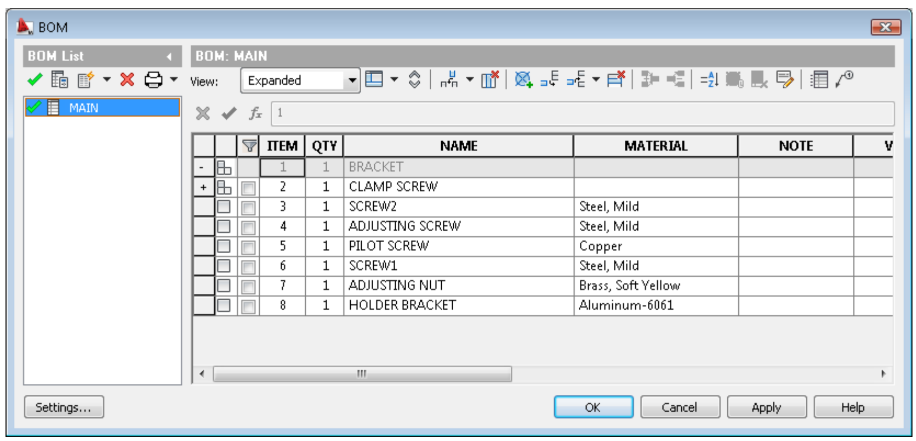
2. Respond to the prompts as follows:

   `Specify BOM to create or set current [Main/?] <MAIN>:` *Press* ENTER

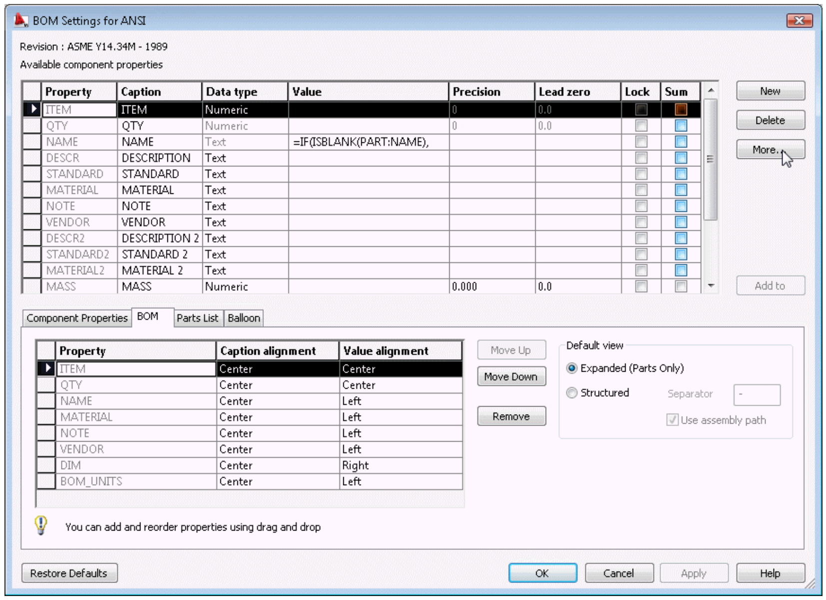
   The BOM dialog box is displayed.



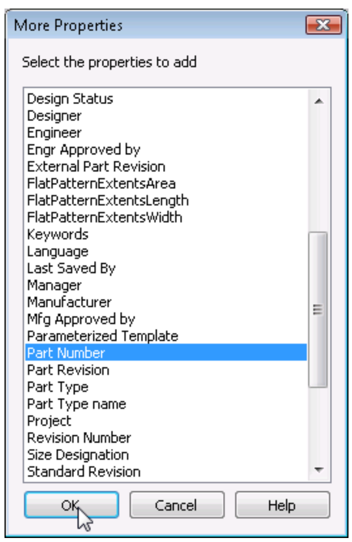
3 Click the + sign in the first column to expand the row.



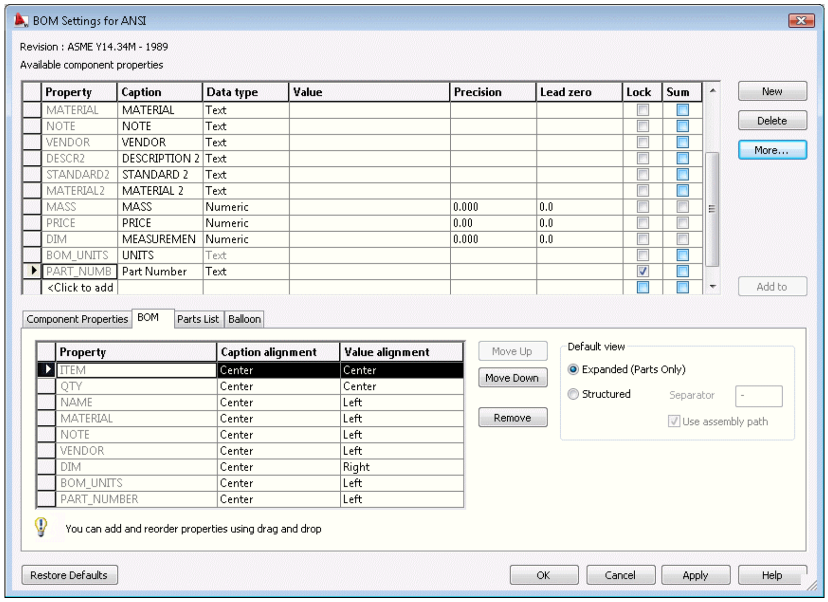
4 Click Settings. The BOM Settings dialog box is displayed.



5 Click the More button to display More Properties dialog box.



6 Select Part Number and click OK. You are returned to the BOM dialog box. Notice the additional row at the bottom of the Available component properties list.



7 Click OK. The BOM Settings dialog box closes and the BOM dialog box becomes accessible again.

8 In the BOM dialog box, use the horizontal scroll bar to inspect the columns in the extreme right.

Note how the iProperty Part Number is listed and automatically filled with data from the Inventor assembly file.

9 Save the file as *Inventor Assembly.dwg*.

Inserting Drawing Borders

To insert a drawing border

1 Click the Drawing tab in the Mechanical Browser.

2 Start the Drawing Title/Borders command.

Ribbon	Annotate tab ➤ Layout panel ➤ 
Menu	Annotate ➤ Drawing Title and Revision ➤ Drawing Title/Borders...

Command AMTITLE

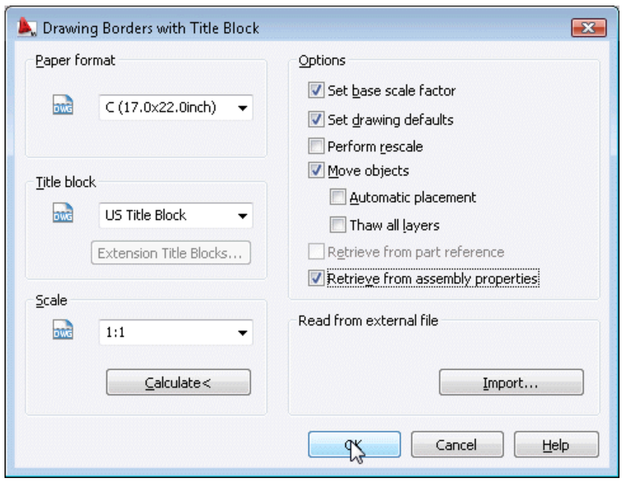
3 In the Drawing Borders with Title Block dialog box, specify:

`Paper Format:` C (17.0x22.0 inch)

`Title Block:` US Title Block

`Scale:` 1:1

`Retrieve from Assembly Properties:` *Select*



4 Click OK.

5 In the Page Setup Manager dialog box, select Layout1, then click Modify.

6 In the Page Setup - Layout1 dialog box, specify the following value:

`Paper size:` ANSI C (22.00 x 17.00 Inches)

7 Choose OK to exit the Page Setup Manager.

8 Click Close.

9 Respond to the prompt as follows:

`Specify insertion point:` *Enter -0.25,-0.75, press* ENTER

10 In the Change Title Block Entry dialog box click Next.

11 In the Drawing Title box, type Adjustable Bracket.

12 Click OK.

Creating Parts Lists and Balloons

You can create a variety of drawing view types for a part, but you must create a base view first. Subsequent changes made to the assembly file in Autodesk Inventor are automatically updated in the drawing views when the drawing file is updated.

To create a base view

1 Create a new drawing view..

Ribbon	None
Menu	Drawing Default ➤ New View
Browser	Right-click a Layout icon in the Drawing tab, then choose New View.
Command	AMDWGVIEW

2 In the Create Drawing View dialog box, specify:

`View Type:` Base

`Data Set:` Select

`Layout:` Layout1

`Orientation:` Top

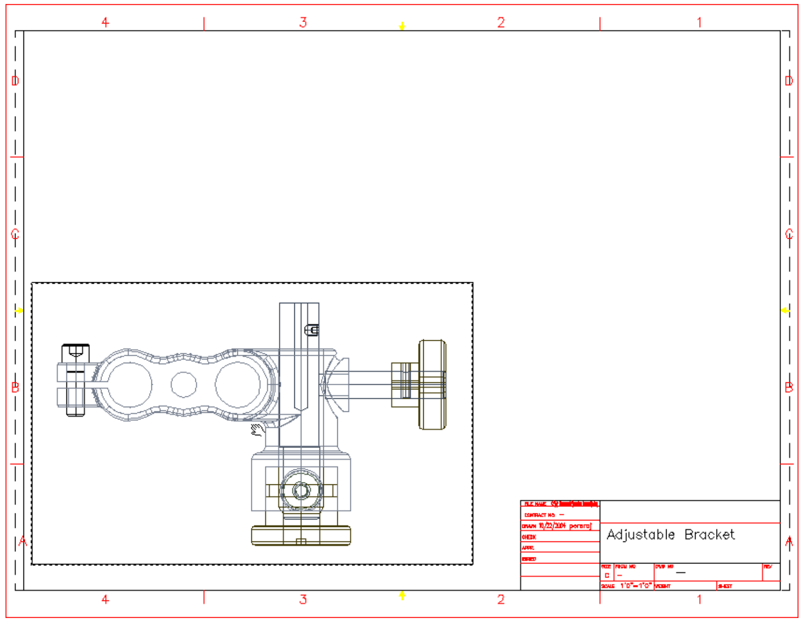
`Scale:` 2.0000

3 Click OK.

4 Respond to the prompts as follows:

`Specify location for base view:`

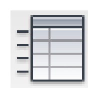
Click in the lower left corner of the graphics area, press ENTER



The base view is placed in the lower-left corner of the drawing.

To create the parts list

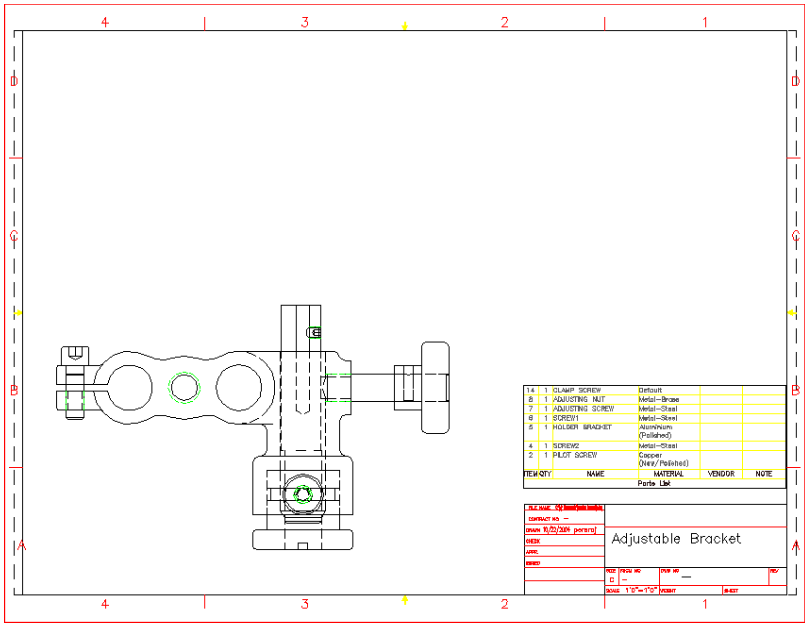
1 Start the Parts List command.

Ribbon	Annotate tab ➤ Table panel ➤ 
Menu	Annotate ➤ Parts List
Command	AMPARTLIST

The Part List ANSI dialog box is displayed.

2 Click OK.

3 Move the cursor to position the parts lists above the title block, then click to insert the parts list.



To create balloons

1 Start the Balloons command..

Ribbon	Annotate tab ➤ Balloon panel ➤ 
Menu	Annotate ➤ Balloons
Command	AMBALLOON

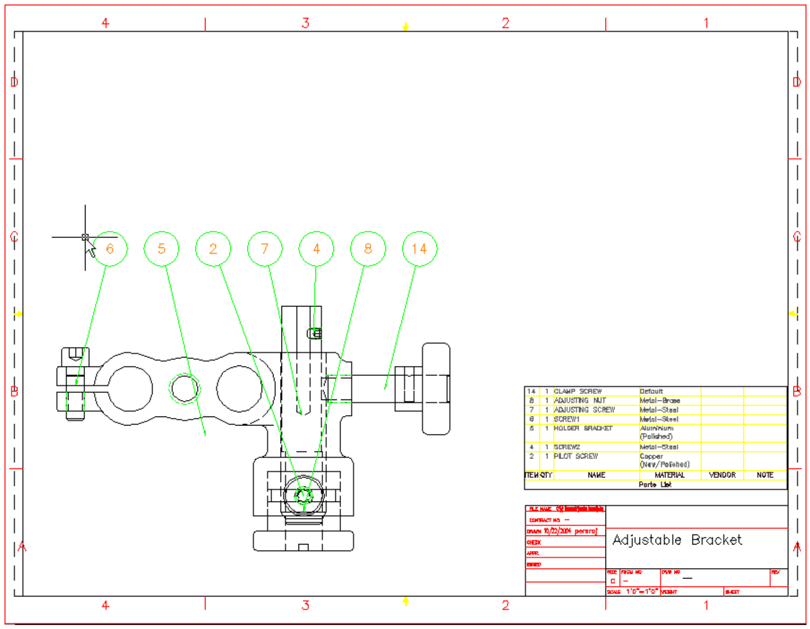
2 Respond to the prompt as follows:

`Select part/assembly or`
`[auTo/autoAll/Collect/Manual/One/Renumber/rEorganize]:` *Enter A*

`Select pick object:` *Window select the entire assembly*

`Select pick object:` *Press* ENTER

3 Place the balloons horizontally above the assembly.



Creating Breakout Section Views

A breakout section view shows hidden details by cutting away portions that block their visibility. In this exercise, you indicate the section to remove by creating a cut line on one view and marking the depth of the cut on another view. Once the breakout section view is generated, you create an isometric view for it.

To create the base view and orthogonal view:

1 Click the Drawing tab in the browser and double-click Layout 2.

2 Start the Drawing Title/Borders command.

Ribbon	Annotate tab ➤ Layout panel ➤ 
Menu	Annotate ➤ Drawing Title and Revision ➤ Drawing Title/Borders...
Command	AMTITLE

The Drawing Borders with Title Block dialog box is displayed

3 Create a new drawing border for Layout2, following the procedure outlined in Inserting Drawing Borders on page 376.

To create a base view and orthogonal view

1 Create a new drawing view.

Ribbon	None
Menu	Drawing Default ➤ New View
Browser	Right-click the Layout2 icon in the Drawing tab, then choose New View.
Command	AMDWGVIEW

2 In the Create Drawing View dialog box, specify:

`View Type:` Multiple

`Data Set:` Select

`Layout:` Layout2

`Scale:` 1.5

`Display Hidden Lines:` *Clear the check box*

3 Choose OK.

4 Respond to the prompt as follows:

`Select planar face, work plane or`

`[sTandard view/Ucs/View/worldXy/worldYz/worldZx]:` *Enter X*

`Select work axis, straight edge or [worldX/worldY/worldZ]:`

Enter X

`Adjust orientation [Flip/Rotate] <Accept>:` *Press* ENTER

`Specify location of base view:`

Position the view in the lower left corner of the graphics area and click

`Specify location of base view or [Done] <next view>:` *Press* ENTER

`Specify location of projected view or [New parent view]:`

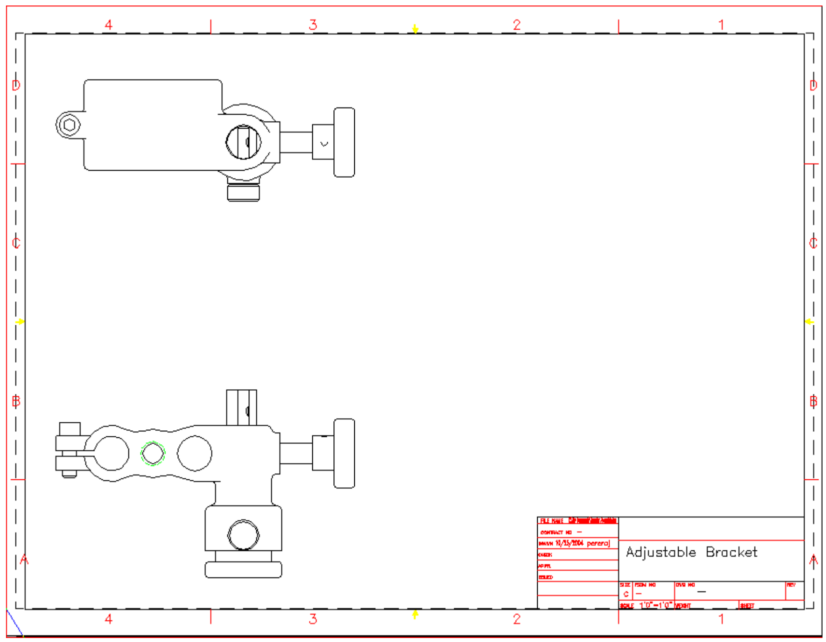
Drag to a location above the base view, click to select location

`Specify location of projected view or [Done] <next view>:`

Press ENTER

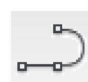
`Specify location of projected view or [New parent view]:`

Press ENTER



To create the cut line:

1 Start the polyline command..

Ribbon	Home tab ➤ Draw panel ➤ 
Menu	Draw ➤ Polyline
Command	PLINE

2 Respond to the prompts as follows:

`Specify start point:` *Click point (1)*

`Specify next point or [Arc/Halfwidth/Length/Undo/Width]:`

Click point (2)

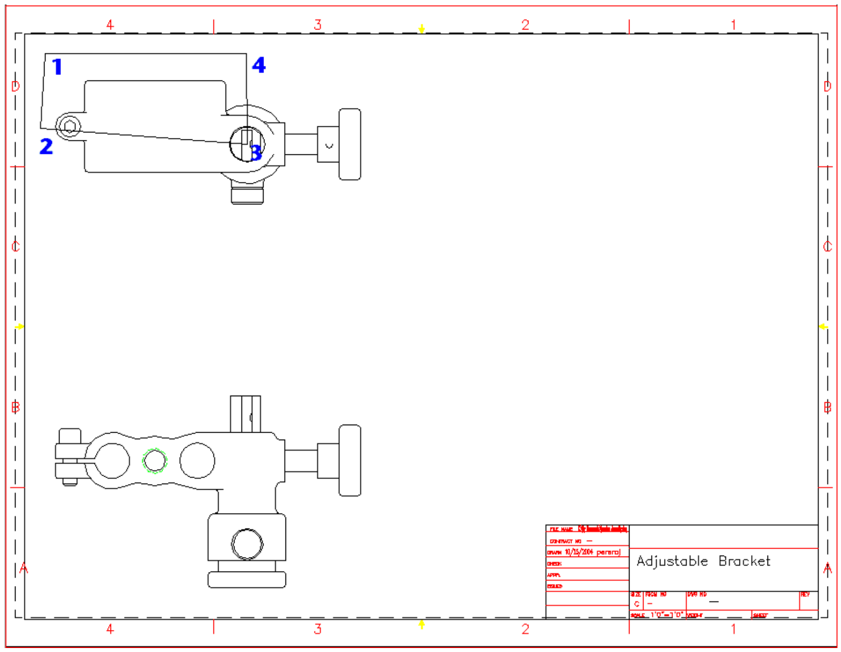
`Specify next point or [Arc/Halfwidth/Length/Undo/Width]:`

Click point (3)

`Specify next point or [Arc/Halfwidth/Length/Undo/Width]:`

Enter close, press ENTER

A closed polyline is created.



To create a breakout section view

1 Create a base view type.

Ribbon	None
Menu	Drawing Default ➤ New View
Browser	Right-click the Ortho icon in the Drawing tab, then choose New View.
Context Menu	Right-click in the graphics area, then choose New View.
Command	AMDWGVIEW

2 In the Create Drawing View dialog box, specify:

`View Type:` Base

3 On the Section Tab, specify

`Type:` Breakout

`Hatch:` Selected

4 Click OK.

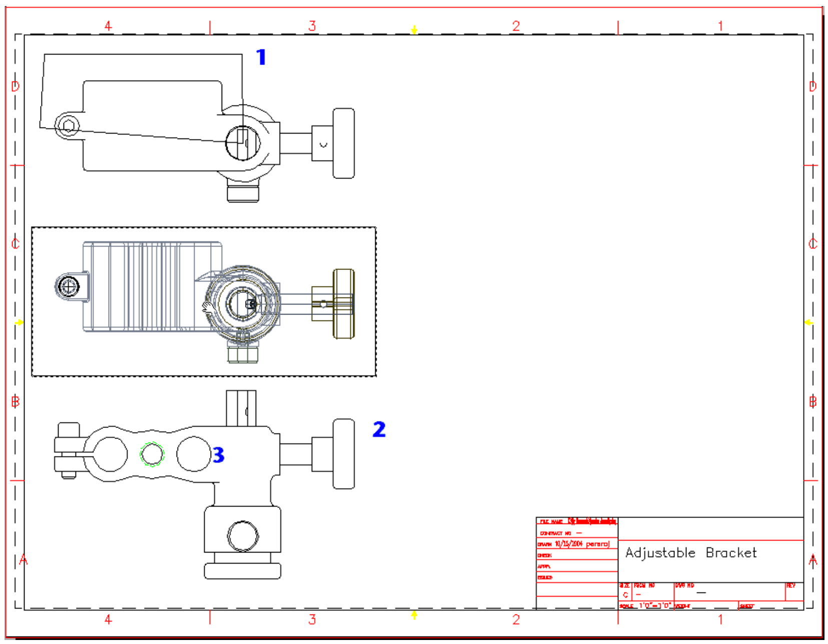
5 Respond to the prompts:

`Select first parent view for breakout view:`

Select the orthogonal view

`Specify location of base view:`

Drag just above the base view, click to select the location, press ENTER



`Select polyline to use as cutline:`

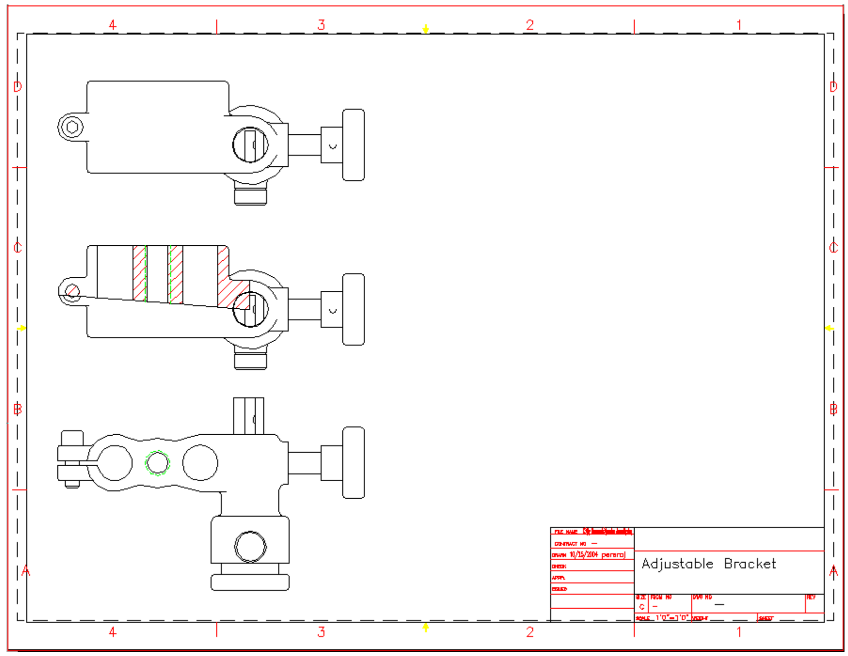
Click the polyline you created in the previous exercise (1)

`Select second parent view for depth selection:`

Select the base view (2)

`Select point for depth of section:` *Select point (3)*

The breakout section view is created.



To create an isometric view of the breakout section view:

1. Create an isometric view type.

Ribbon	None
Menu	Drawing Default ➤ New View
Browser	Right-click the Section icon in the Drawing tab, then choose New View.
Context Menu	Right-click in the graphics area, then choose New View.
Command	AMDWGVIEW

2. In the Create Drawing View dialog box, specify:

   `View Type:` Iso

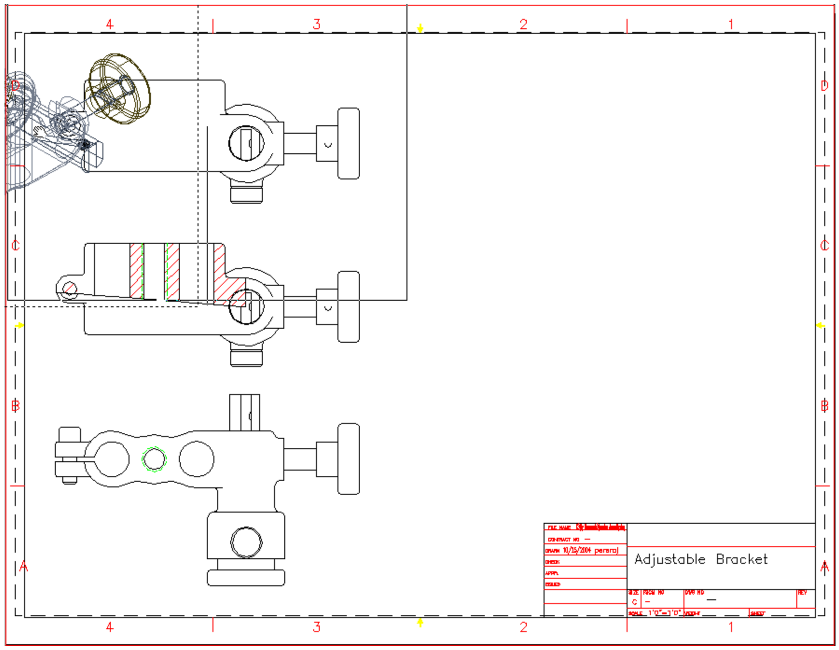
3. Choose OK.

4. Respond to the prompts:

   `Select parent view:` *Select the breakout section view*

   `Specify location of base view:`

Drag to the left of the orthogonal view, click, and press ENTER



NOTE The details shown in the view that is generated depend on where you place the view. When you drag to the left, the isometric view that is generated reveals a hole and a screw. They would not be visible if you placed the view elsewhere.

The isometric view is created.

5 Move the isometric view to the right of the orthogonal view.

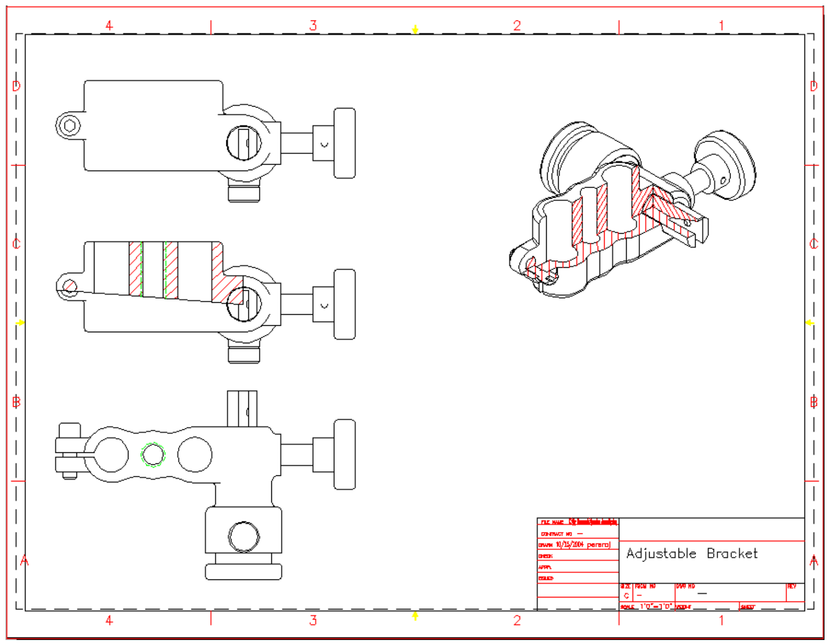
Ribbon	None
Menu	Drawing Default ➤ Move View
Browser	In the Drawing tab, right-click the Iso icon, then choose Move.
Context Menu	Right-click in the graphics area, then choose Move View.
Command	AMMOVEVIEW

6 Respond to the prompts:

`Select view to move:` *Select the isometric view*

`Specify new view location:`

Drag to the right of the orthogonal view, click, and press ENTER



Modifying Breakout Section Views

The cut line used to generate the breakout section view can be modified and breakout section view regenerated. Under normal circumstances, the cut line is not visible. To modify the cut line, you must display it first.

To display the cutline:

1 Start the Edit Paper Space Cut Line command.

Browser	Right-click the Section icon in the Drawing tab, then choose Re-Select Cutline.
Command	AMEDITPSCUTLINE

2 Respond to the prompts:

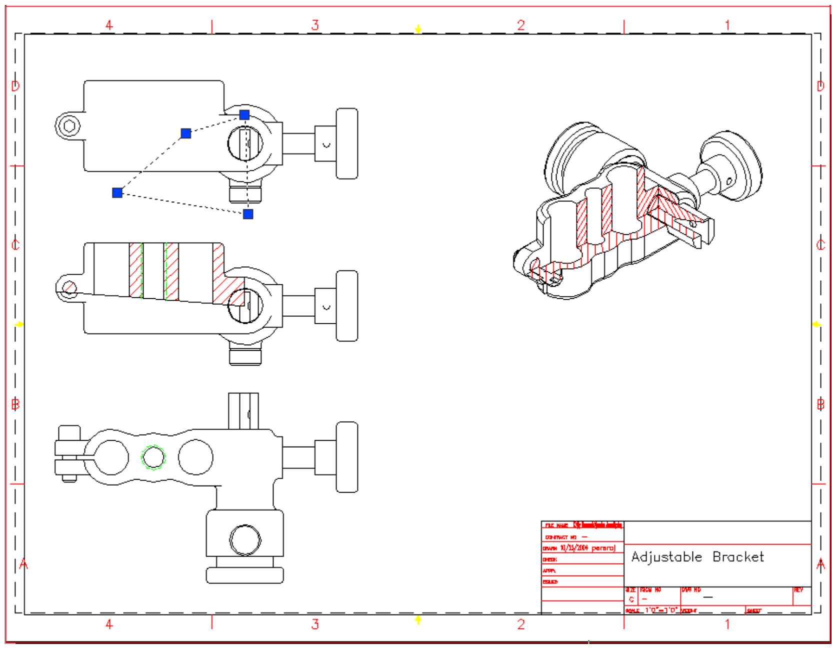
`Select broken-out section view:` *Click the breakout section view*

```
Enter an option for paperspace cutline [Display/Select]
<Display>:
```

Press ENTER

The cut line is displayed.

3 Modify the cutline to any shape you want it to be.



4 Start the Edit Paper Space Cut Line command.

Browser	Right-click the Section icon in the Drawing tab, and then choose Re-Select Cutline.
Command	AMEDITPSCUTLINE

5 Respond to the prompts.

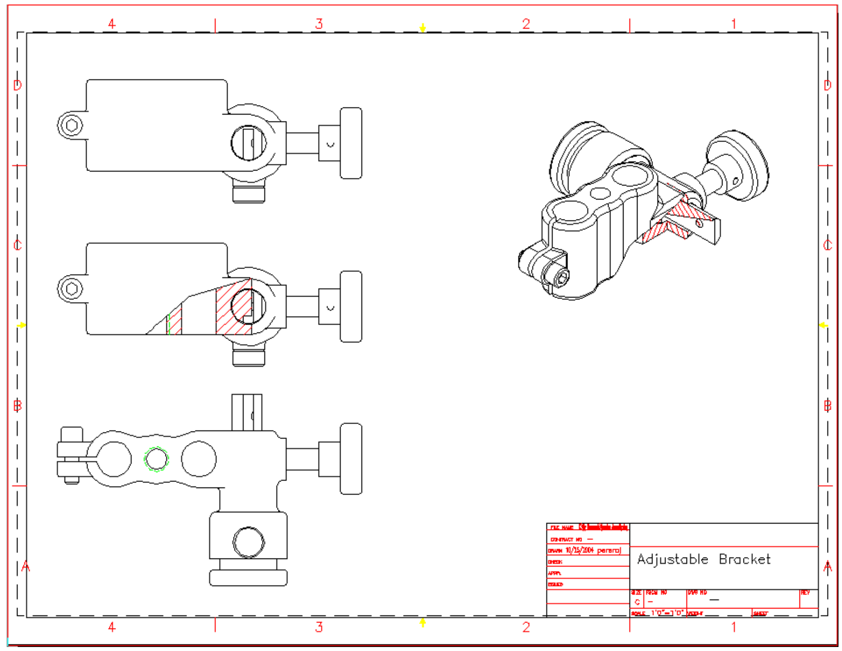
`Select broken-out section view:` *Click the breakout section view*

```
Enter an option for paperspace cutline [Display/Select]
<Display>:
```

Enter S

`Select polyline to use as cutline:` *Click the edited polyline*

The breakout section view and the isometric view update.



Removing Views

You can remove views, even though that view may have been used to derive other views.

To delete the base view:

1. Right-click the base view icon in the browser and select Delete. The Delete dependent views dialog box is displayed.
2. Click No. The base view is deleted.
3. Save the file and close AutoCAD Mechanical.

Updating Autodesk Inventor Parts

If you have access to Autodesk Inventor (version 8 or above), you can modify the part file using Autodesk Inventor, then update the part in AutoCAD Mechanical to reflect the change.

To edit a dimension using Autodesk Inventor

1. Open *Holder Bracket.ipt* in Autodesk Inventor.
2. Edit a feature.
3. Save the modified part file.

When the part file has been modified outside AutoCAD Mechanical, on the browser, the affected views are highlighted in yellow. Additionally, a balloon is displayed on the status bar informing you that a newer version is now available. To bring in the modifications, you must update the part file.

To update the part file

1. Use AutoCAD Mechanical to open the Assembly file or Part file that was used in this exercise.
2. Observe how the browser indicates the parts and views affected by the part modification.
3. Start the update command.

Browser	Right-click the yellow part node, then choose Update.
Command	AMIVUPDATE

4. Verify that your part has been modified.

   You are notified of the number of drawing views that have been updated.

   Save your file. This is the end of this tutorial chapter.

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