ANSYS Workbench SwiftComp GUI

User Manual

Version 1.01

March, 2019

Banghua Zhao, Hamsasew Sertse, Xin Liu, Su Tian and Wenbin Yu

Table of Contents

CHAPTER 1

1 Introduction

1.1 Introduction

Based on the recently invented Mechanics of Structure Genome (MSG), SwiftComp™ provides an efficient and accurate approach for modeling composite materials and structures. It can be used either independently as a tool for virtual testing of composites or as a plugin to power conventional FEA codes with efficient high-fidelity multiscale modeling for composites. SwiftComp™ implements a true multiscale theory which assures the best models at a given level of efficiency to capture both anisotropy and heterogeneity of composites at the microscopic scale or any other scale of user's interest. SwiftComp™ enables engineers to model composites as a black aluminum, capturing details as needed and affordable. This saves orders of magnitude in computing time and resources without sacrificing accuracy, while enabling engineers to tackle complex problems effectively.

To facilitate the use of SwiftComp™, a simple graphic user interface (GUI) based on ANSYS Workbench, called ANSYS Workbench SwiftComp GUI is developed. This Manual focus on the introduction of ANSYS Workbench SwiftComp GUI along with some examples.

CHAPTER 2

2 Installation

2.1 Introduction

First of all, to use ANSYS Workbench SwiftComp GUI, your machine must be preinstalled with the **ANSYS 19.0 or higher**.

2.2 System Requirement

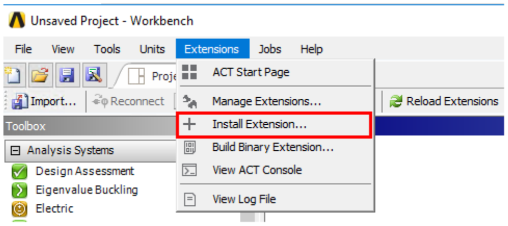
ANSYS Workbench SwiftComp GUI is developed in **Windows 10** environment. Other Windows version may not be stable. It cannot be installed in a Unix/Linux environment.

2.3 Installation of ANSYS Workbench SwiftComp GUI

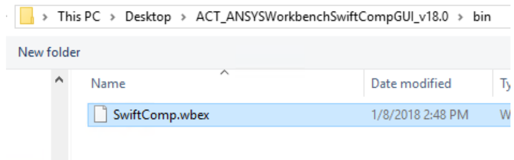
ANSYS Workbench SwiftComp GUI extension is developed based on ANSYS Customization Toolkit (ACT). First, you need to install the extension to Workbench environment. Then load the extension.

First, install from the Extensions menu:

- From the Extensions menu, select the "Install Extension…" option

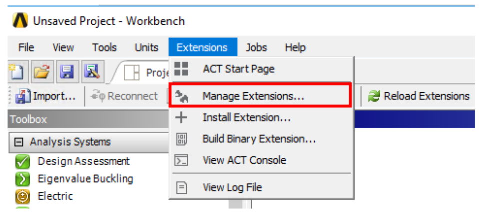


- It will open a file dialog to select the appropriate "*.*wbex*" binary file, which is the extension binary file. It can be found in the bin folder under released package. Click "Open" to install the extension

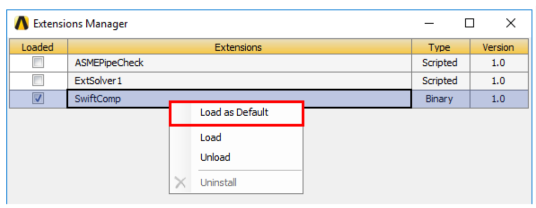


Then, load the extension:

- Select SwiftComp under "Extensions > Manage Extensions" on the menu bar.



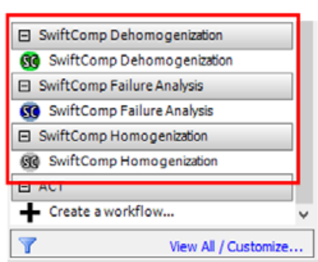
- In the dialog box, toggle "SwiftComp". Then right click "SwiftComp", and select "Load as Default"



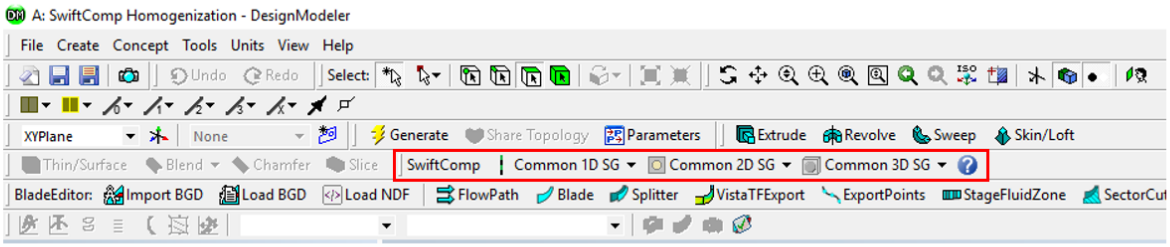
The corresponding demo video can be found at doc/Video demo/installation or online at: https://www.youtube.com/watch?v=wUVjFHZvSSU

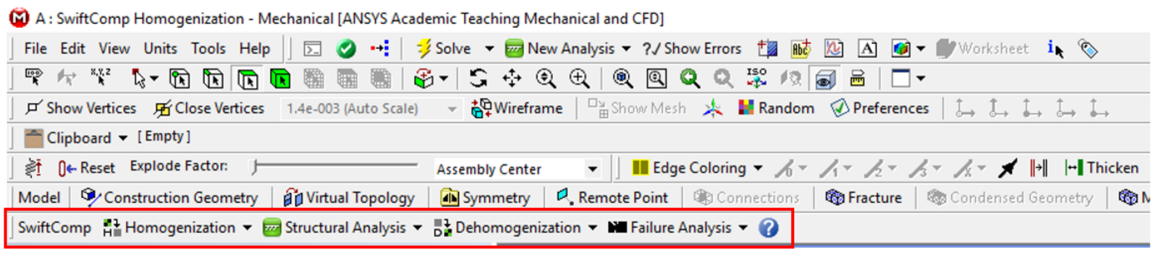
2.4 Check Installation

To check whether ANSYS Workbench SwiftComp GUI is successfully installed in Workbench environment. First, check whether SwiftComp Homogenization, SwiftComp Dehomogenization are available, and SwiftComp Failure Analysis are in the toolbox at Workbench project level.



Then, open Design Modeler and Mechanical environments from SwiftComp Homogenization system. Check that the menus have been added in each GUI environment. If the SwiftComp toolbar does not appear in the Workbench environment, the SwiftComp extension may not be installed successfully.





2.5 Installation of SwiftComp

In order to invoke SwiftComp for homogenization, dehomogenization and failure analysis, **you need to install SwiftComp™ on your local machine**.

To install SwiftComp™ you should request the code from AnalySwift (https://analyswift.com/software-trial/) and follow the instruction inside the SwiftComp release package for installation.

CHAPTER 3

3 Overview

3.1 Introduction

After ANSYS Workbench SwiftComp GUI is successfully installed in ANSYS Workbench environment, SwiftComp Homogenization and SwiftComp Dehomogenization will be available in Toolbox at Workbench project level.

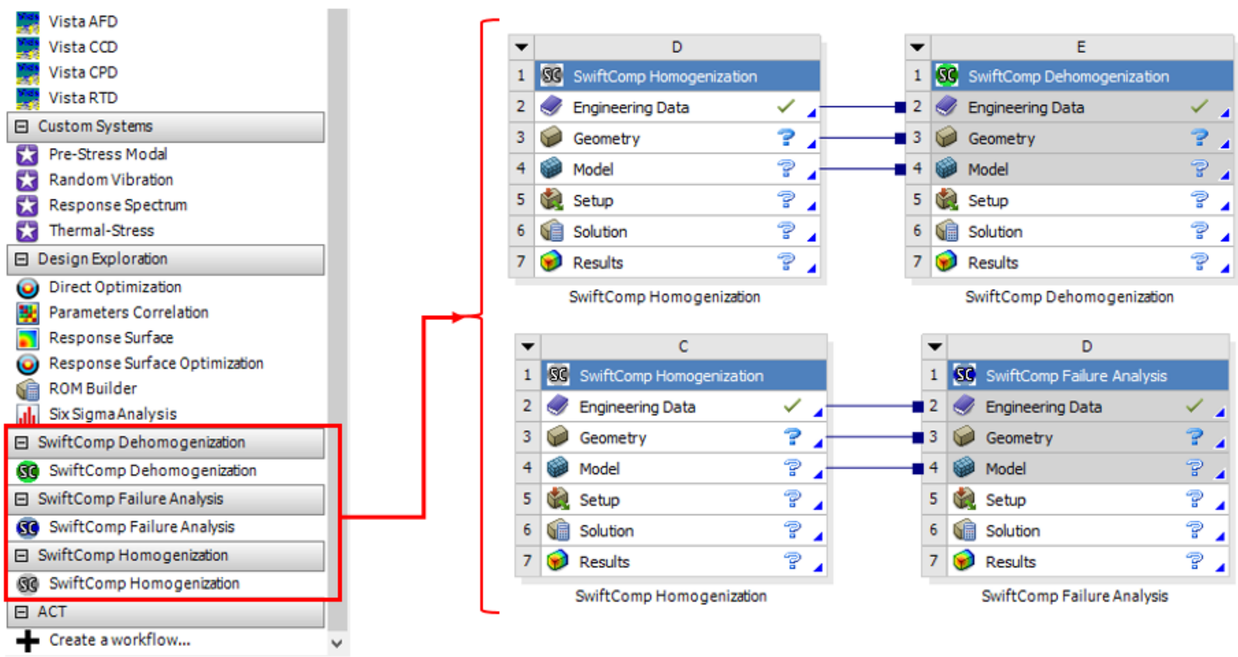


Figure 3.1 SwiftComp Homogenization and SwiftComp Dehomogenization in Toolbox

3.2 Toolbar in DesignModeler

In the DesignModeler, SwiftComp toolbar with Common 1D SG (Structure Genome), Common 2D SG and Common 3D SG will be available.

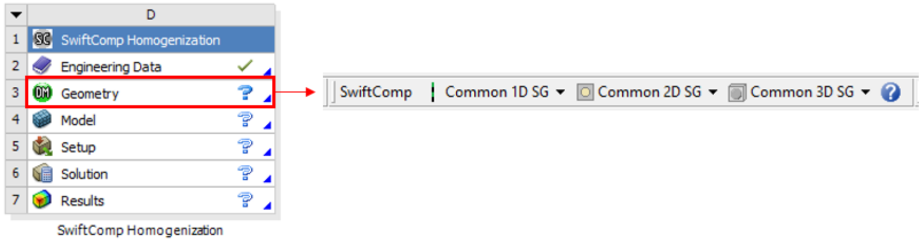


Figure 3.2 SwiftComp toolbar in DesignModeler

3.3 Toolbar in Mechanical Environment

In the Mechanical environment (Model), SwiftComp toolbar with Homogenization, Structural Analysis and Dehomogenization is available.

Figure 3.3 SwiftComp toolbar in Model

3.4 Modeling Workflow

Where are four mainly modeling workflow in ANSYS Workbench SwiftComp GUI. There are explained in the following:

- Multiscale Modeling Workflow for Beam like Structure: 1. Create SG (2D or 3D). 2. Perform homogenization to get beam stiffness matrix. 3. Perform beam model structural analysis. 4. Use the results from structural analysis for dehomogenization.

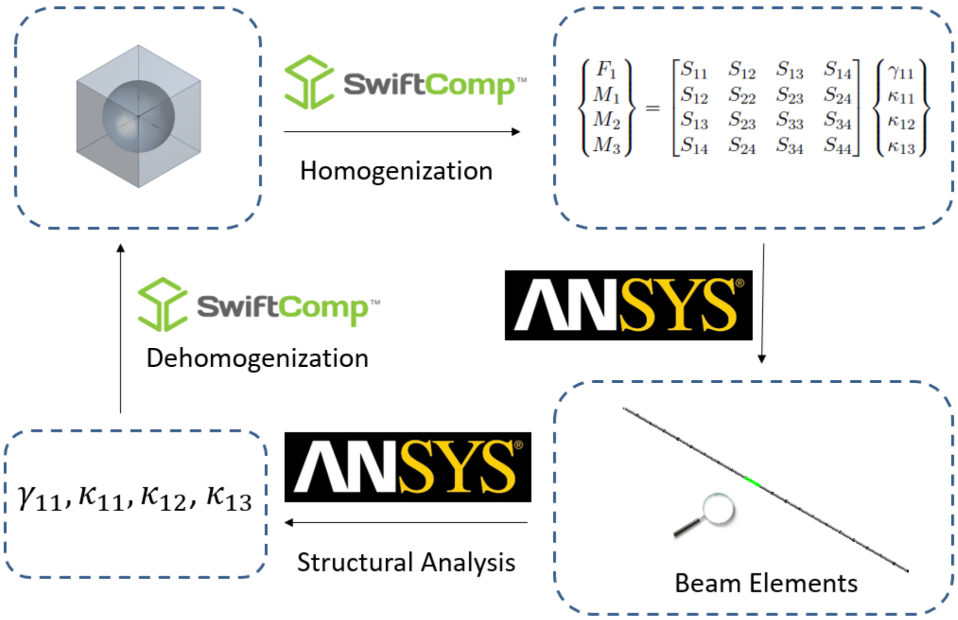


Figure 3.4 Modeling Workflow for Beam like Structure

- Multiscale Modeling Workflow for Plate like Structure: 1. Create SG (1D, 2D or 3D). 2. Perform homogenization to get plate stiffness matrix. 3. Perform plate model structural analysis. 4. Use the results from structural analysis for dehomogenization.

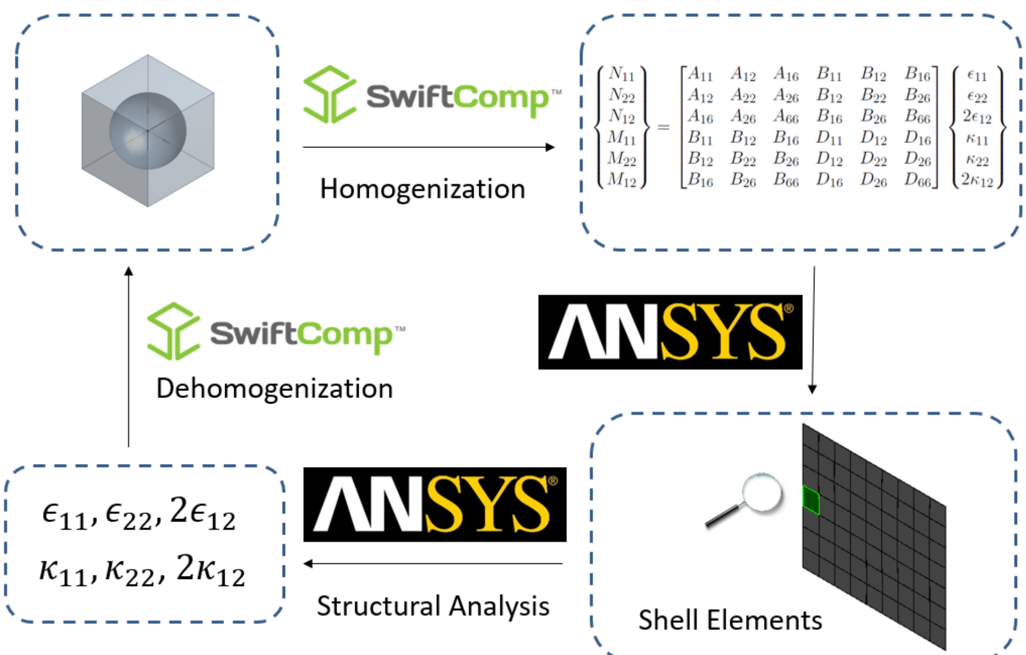


Figure 3.5 Modeling Workflow for Plate like Structure

- Multiscale Modeling Workflow for 3D Structure: 1. Create SG (1D, 2D or 3D). 2. Perform homogenization to get solid stiffness matrix. 3. Perform solid model structural analysis. 4. Use the results from structural analysis for dehomogenization.

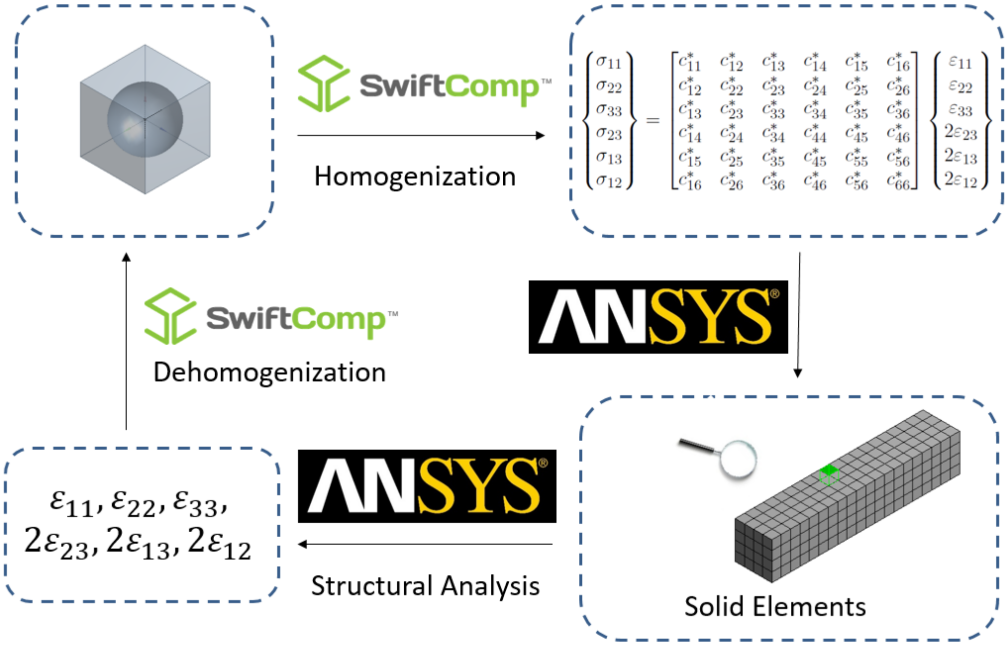


Figure 3.6 Modeling Workflow for Solid like Structure

- Multiscale Modeling Workflow for Failure Analysis: 1. Create SG (1D, 2D or 3D). 2. Perform homogenization to get beam/plate/solid stiffness matrix. 3. Choose failure criterion. 4. Perform failure analysis

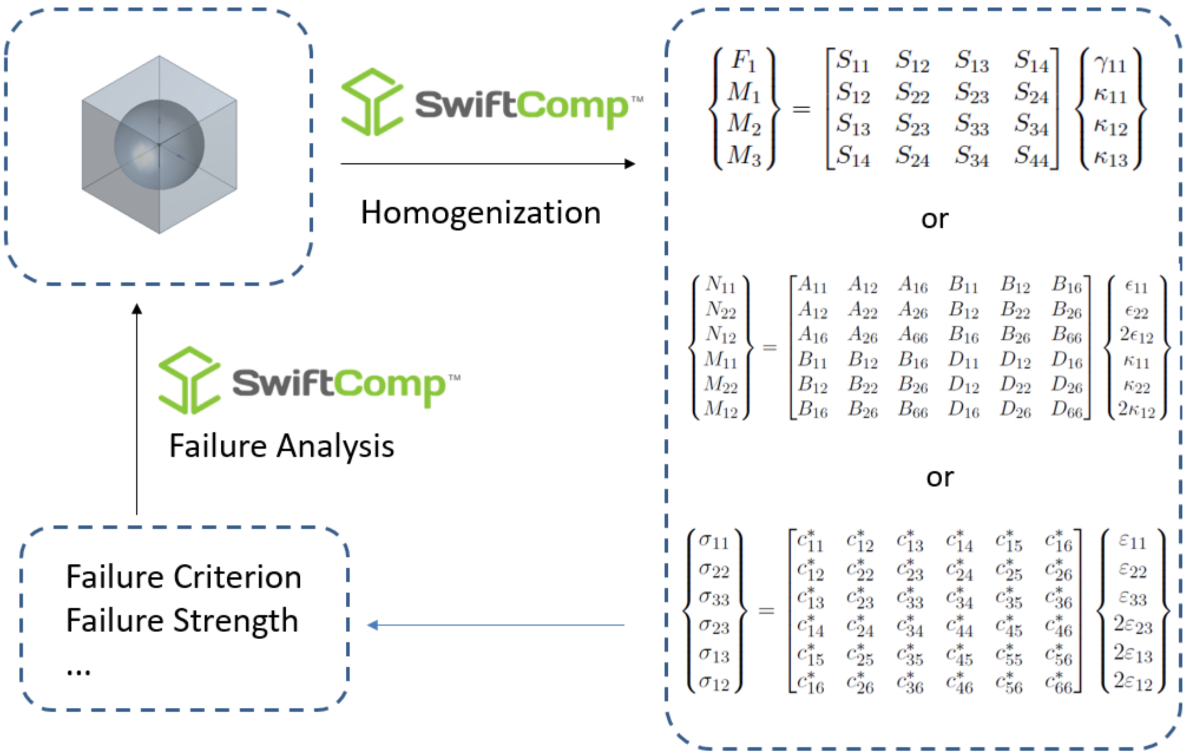


Figure 3.7 Modeling Workflow for Failure Analysis

3.5 Examples

To help user get started, several example videos are created. They are:

- Example 1 2D Square Pack Microstructure: The corresponding demo video can be found at doc/Video demo/example1_2D_Square_Pack_Microstructure or online at: https://www.youtube.com/watch?v=HZZ7dBiuQR8

- Example 2 2D Custom SG: The corresponding demo video can be found at doc/Video demo/example2_2D_Custom_SG or online at: https://www.youtube.com/watch?v=U3iYFnSPUio&t=4s

CHAPTER 4

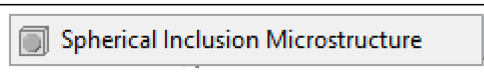
4 Common SG

4.1 Introduction

ANSYS Workbench SwiftComp GUI provides a convenient way to create some common SG models in DesignModeler. Users can easily create the geometry of these models by input the parameters for these models

Currently, ANSYS Workbench SwiftComp GUI provides the following common SGs:

Table 1 Common SG in ANSYS Workbench SwiftComp GUI

SG	Icon in DesignModeler	Model
1D SG		Dummy Plate (for ACP)
2D SG		Square Pack Microstructures 2D
3D SG		Spherical Inclusion Microstructures

4.2 Dummy Plate for ACP

A dummy plate for ACP to generate layup information. No input parameters are need. You just need to click generate.

4.3 Square Pack Microstructures 2D

A SG in which a cylindrical fiber is aligned in the center of a square matrix. To create the SG, you need to input the Fiber Volume Fraction [%]. The valid value is larger than 0 and smaller than 78.5. The default length of the SG is set to be 100. On generation, the radius of fiber will be calculated.

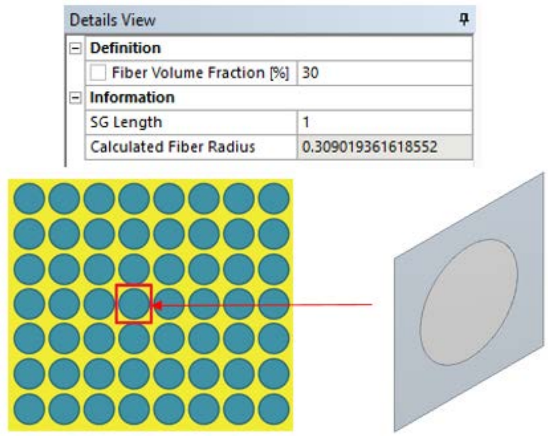


Figure 4.1 Generation of Square Pack Microstructure 2D

4.4 Spherical Inclusion Microstructures

A SG in which a spherical particle is dispersed in the center of a cube matrix. To create the SG, you need to input the Sphere Volume Fraction [%]. The valid value is larger than 0 and smaller than 52.3. The default length of the SG is set to be 100. On generation, the radius of sphere will be calculated.

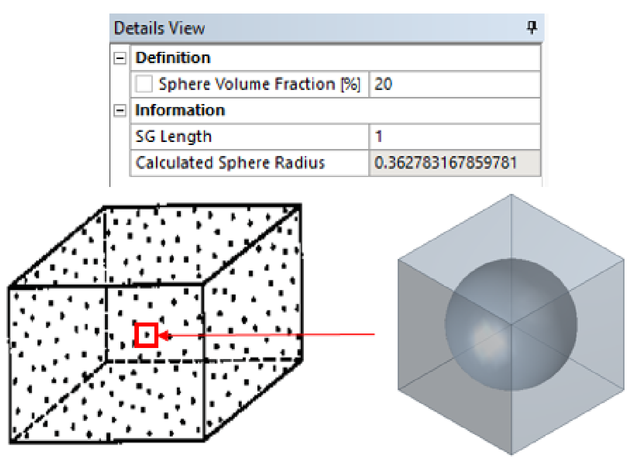


Figure 4.2 Generation of Square Inclusion Microstructure

CHAPTER 4

5 Homogenization

5.1 Introduction

In homogenization analysis, constitutive modeling over the SG is performed to obtain the constitutive models needed for structural analysis,

To perform homogenization, on the SwiftComp toolbar, click "Homogenization" button. Then, choose Beam Model, Plate/Shell Model or Solid Model according to your analysis. Specify all the settings specific to Beam Model Homogenization Setting, Plate/Shell Model Homogenization Setting or Solid Model Homogenization Setting in the Details window. For example, Figure 5.1 shows the homogenization for Solid Model.

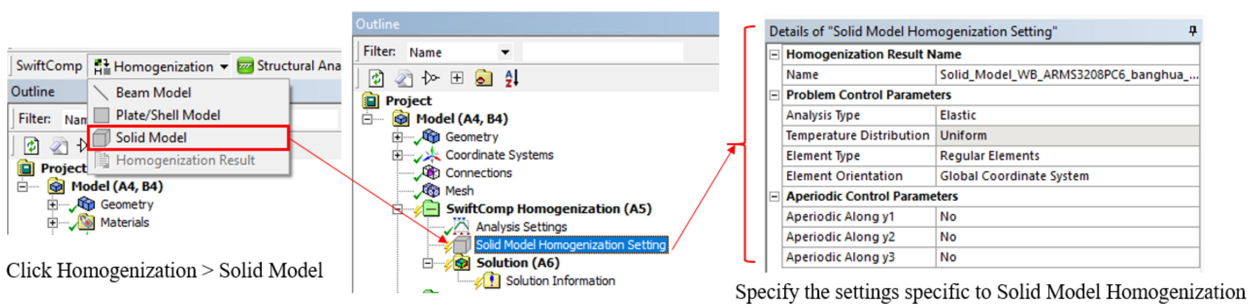


Figure 5.1 Homogenization for Solid Model

For all the model, you need to specify Problem Control Parameters, i.e. Analysis Type, Temperature Distribution, Element Type and Element Orientation.

Analysis Type is the type of analysis for the problem. In the current version, only Elastic is available.

Temperature Distribution inducts whether the temperature is uniform within the SG. For elastic problem, the Temperature Distribution is uniform.

Element Type denotes the type of elements. In the current version, only Regular Elements is available.

Element Orientation indicates whether the transformation of the element orientation is needed. In the current version, only Global Coordinate System is available. In this case, the element orientation is the same as the problem coordinate system and transformation is not needed.

Also, you need to specify Aperiodic Control Parameters. The default values are No. If you choose Yes for a particular direction, then the SG is aperiodic in this direction.

5.2 Beam Model Homogenization

In the macro structural analysis of your model, if one dimension of the structure is much larger than two other dimensions, the structure is called as a beam and you can use beam element for structural analysis. In this case you need to perform Beam Model Homogenization.

If the beam has uniform cross-sections which could be made of homogeneous materials or composites (Figure 5.2 a), its SG is the 2D cross-sectional domain.

If the beam is also heterogeneous in the spanwise direction (Figure 5.2 b), a 3D SG is needed to describe the microstructure of the 1D continuum.

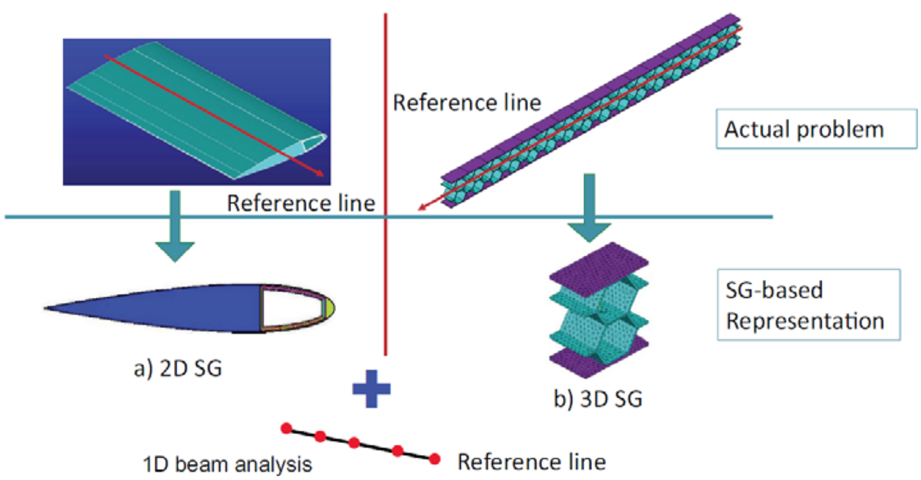


Figure 5.2 Analysis of beam-like structures approximated by a constitutive modeling over SG and a corresponding 1D beam analysis.

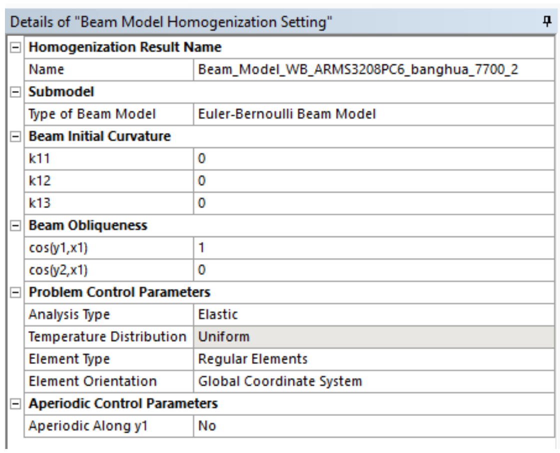
Beam Model Homogenization will provide the constitutive relations needed for structural analysis by homogenizing over the SG in your problem.

For example, the constitutive relation of the Euler-Bernoulli beam model can expressed using the following four equations.

$$
\begin{Bmatrix} F_1 \\ M_1 \\ M_2 \\ M_3 \end{Bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} & S_{14} \\ S_{12} & S_{22} & S_{23} & S_{24} \\ S_{13} & S_{23} & S_{33} & S_{34} \\ S_{14} & S_{24} & S_{34} & S_{44} \end{bmatrix} \begin{Bmatrix} \gamma_{11} \\ \kappa_{11} \\ \kappa_{12} \\ \kappa_{13} \end{Bmatrix} \tag{1}
$$

The 4×4 matrix is called the beam stiffness matrix and its inverse is called the beam compliance matrix. Both beam stiffness matrix and beam compliance matrix will be the output of Beam Model Homogenization.

In the Details window of Beam Model Homogenization Setting. You also need to specify Submodel, Beam Initial Curvature and Beam Obliqueness as shown in Figure 5.3.



Details of "Beam Model Homogenization Setting"

Homogenization Result Name	
Name	Beam_Model_WB_ARMS3208PC6_banghua_7700_2
Submodel	
Type of Beam Model	Euler-Bernoulli Beam Model
Beam Initial Curvature	
k11	0
k12	0
k13	0
Beam Obliqueness	
cos(y1,x1)	1
cos(y2,x1)	0
Problem Control Parameters	
Analysis Type	Elastic
Temperature Distribution	Uniform
Element Type	Regular Elements
Element Orientation	Global Coordinate System
Aperiodic Control Parameters	
Aperiodic Along y1	No

Figure 5.3 Beam Model Homogenization Setting

Submodel denote the specific type of beam model. In the current version, Euler-Bernoulli Beam Model and Timoshenko Beam Model are available.

Beam Initial Curvature k_{11}, k_{12} and k_{13} indicate the initial twist (k_{11})/curvatures (k_{12} and k_{13}) of the beam. If the beam is initially straight, zeroes should be provided (default values).

Beam Obliqueness parameters, cos(y_1, x_1) and cos(y_2, x_1), specify a SG with oblique cross-sections, see Figure 5.4 for a sketch of such a cross-section. The first number is cosine of the angle between normal of the oblique section (y_1) and beam axis x_1. The second number is cosine of the angle between y_2 of the oblique section and beam axis (x_1). For normal cross-sections, we cos(y_1, x_1)=1.0 and cos(y_2, x_1)=0.0.

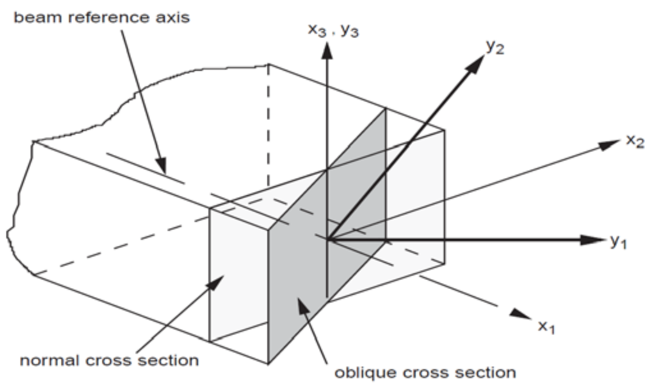


Figure 5.4 Sketch of an oblique reference cross-section

5.3 Plate/Shell Model Homogenization

In the macro structural analysis of your model, if one dimension of the structure is much smaller than two other dimensions, the structure is called as a plate and you can use shell element for structural analysis. In this case you need to perform Plate/Shell Model Homogenization.

If the plate-like structures feature no in-plane heterogeneities (Figure 5.5 a), the SG is the transverse normal line with each segment denoting the corresponding layer.

For a sandwich panel with a core corrugated in one direction (Figure 5.5 b), the SG is 2D.

If the panel is heterogeneous in both in-plane directions (Figure 5.5 c), such as a stiffened panel with stiffeners running in both directions, the SG is 3D.

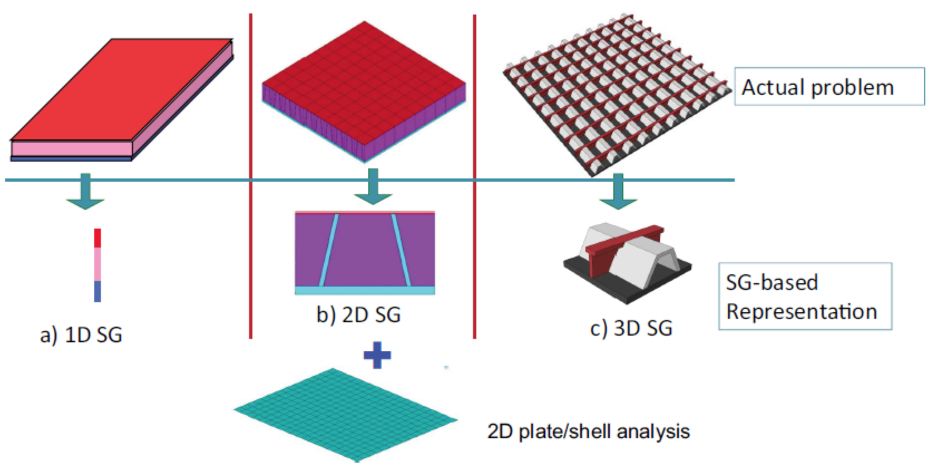


Figure 5.5 Analysis of plate-like structures approximated by a constitutive modeling over SG and a corresponding 2D plate analysis.

Plate/Shell Model Homogenization will provide the constitutive relations needed for structural analysis by homogenizing over the SG in your problem.

For example, the constitutive relation of the Kirchhoff-Love model can expressed using the following six equations.

$$\begin{Bmatrix} N_{11} \\ N_{22} \\ N_{12} \\ M_{11} \\ M_{22} \\ M_{12} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} & B_{11} & B_{12} & B_{16} \\ A_{12} & A_{22} & A_{26} & B_{12} & B_{22} & B_{26} \\ A_{16} & A_{26} & A_{66} & B_{16} & B_{26} & B_{66} \\ B_{11} & B_{12} & B_{16} & D_{11} & D_{12} & D_{16} \\ B_{16} & B_{22} & B_{26} & D_{12} & D_{22} & D_{26} \\ B_{16} & B_{26} & B_{66} & D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ 2\epsilon_{12} \\ \kappa_{11} \\ \kappa_{22} \\ 2\kappa_{12} \end{Bmatrix} \tag{2}$$

The 6×6 matrix is called the plate stiffness matrix (*A*, *B* and *D* matrices) and its inverse is called the plate compliance matrix. Both plate stiffness matrix and plate compliance matrix will be the output of Plate/Shell Model Homogenization.

In the Details window of Plate/Shell Model Homogenization Setting. You also need to specify Submodel, Plate/Shell Initial Curvature as shown in Figure 5.6.

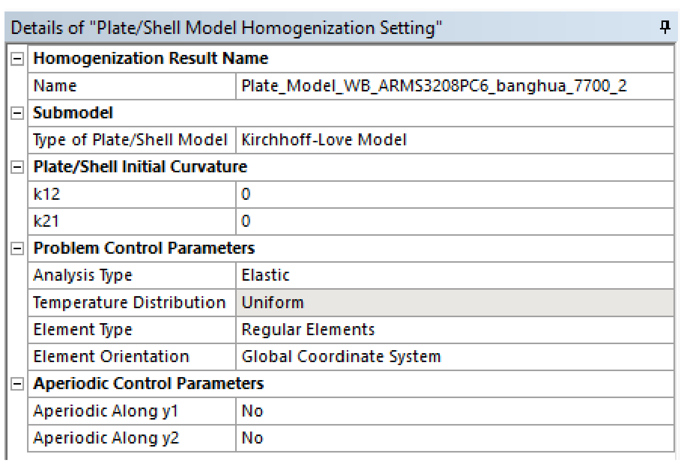


Figure 5.6 Plate/Shell Model Homogenization Setting

Submodel denote the specific type of Plate/Shell model. In the current version, only Kirchhoff-Love Model is available.

Plate/Shell Initial Curvature k_{12} and k_{21} indicate the curvatures of the plate. If the plate is initially straight, zeroes should be provided (default values).

5.4 Solid Model Homogenization

In the macro structural analysis of your model If the three dimensions of a structure are of similar size, the structure is called as a solid (or 3D structure) and you can use solid element for structural analysis. In this case you need to perform Solid Model Homogenization.

For a structure made of composites featuring 1D heterogeneity (e.g. binary composites made of two alternating layers, a), the SG will be a straight line with two segments denoting corresponding phases

For a structure made of composites featuring 2D heterogeneity (e.g. continuous unidirectional fiber reinforced composites, b), the SG will be 2D.

For a structure made of composites featuring 3D heterogeneity (e.g. particle reinforced composites, c), the SG will be a 3D volume.

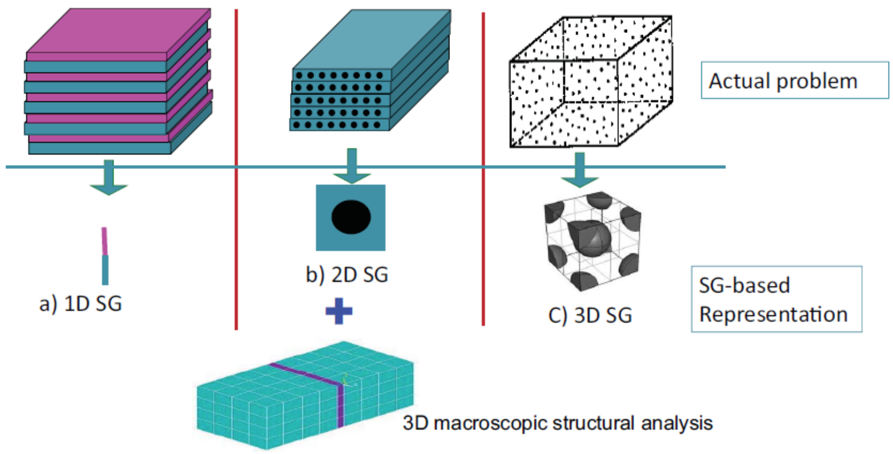


Figure 5.7 Analysis of 3D heterogeneous structures approximated by a constitutive modeling over SG and a corresponding 3D macroscopic structural analysis.

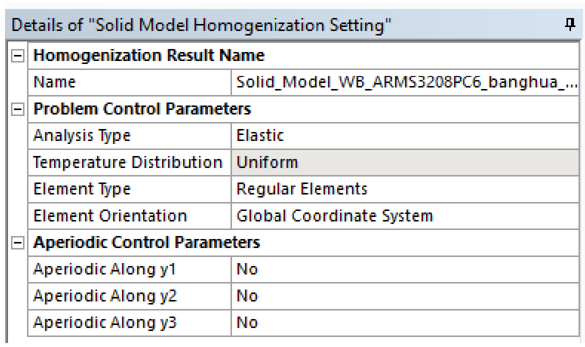
Solid Model Homogenization will provide the constitutive relations needed for structural analysis by homogenizing over the SG in your problem.

For example, the constitutive relation of Cauchy continuum model for the linear elastic behavior are described using the generalized Hooke's law as

$$
\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{13} \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} c_{11}^* & c_{12}^* & c_{13}^* & c_{14}^* & c_{15}^* & c_{16}^* \\ c_{12}^* & c_{22}^* & c_{23}^* & c_{24}^* & c_{25}^* & c_{26}^* \\ c_{13}^* & c_{23}^* & c_{33}^* & c_{34}^* & c_{35}^* & c_{36}^* \\ c_{14}^* & c_{24}^* & c_{34}^* & c_{44}^* & c_{45}^* & c_{46}^* \\ c_{15}^* & c_{25}^* & c_{35}^* & c_{45}^* & c_{55}^* & c_{56}^* \\ c_{16}^* & c_{26}^* & c_{36}^* & c_{46}^* & c_{56}^* & c_{66}^* \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{23} \\ 2\varepsilon_{13} \\ 2\varepsilon_{12} \end{Bmatrix} \tag{3}
$$

The 6×6 matrix c_{ij}^* is called the effective stiffness matrix and its inverse is called the compliance matrix. Both effective stiffness matrix and compliance matrix will be the output of Solid Model Homogenization.

In the Details window of Solid Model Homogenization Setting in Figure 5.8, no other extra input for beam or plate/shell are needed. You just need to provide Problem Control Parameters and Aperiodic Control Parameters.


Details of "Solid Model Homogenization Setting"

Homogenization Result Name	
Name	Solid_Model_WB_ARMS3208PC6_banghua_...
Problem Control Parameters	
Analysis Type	Elastic
Temperature Distribution	Uniform
Element Type	Regular Elements
Element Orientation	Global Coordinate System
Aperiodic Control Parameters	
Aperiodic Along y1	No
Aperiodic Along y2	No
Aperiodic Along y3	No

Figure 5.8 Solid Model Homogenization Setting

5.5 Homogenization Result

To see the homogenization result, you can click Solution Information under the Outline tree. It will give you all the solution information.

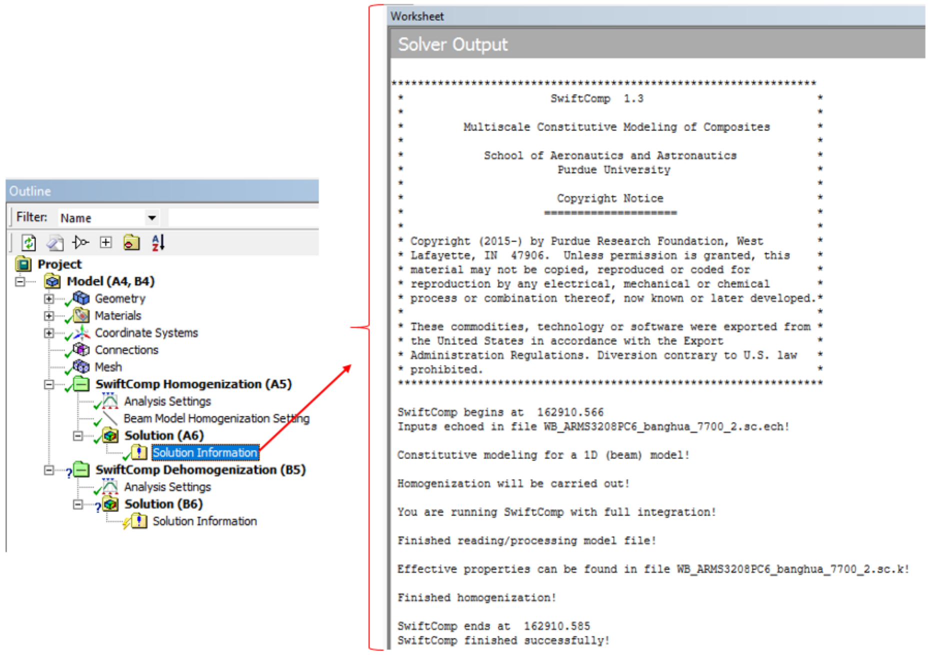


Figure 5.9 Solution Information

If you want to perform a parametric study for the results, you can also click Homogenization > Homogenization Result.

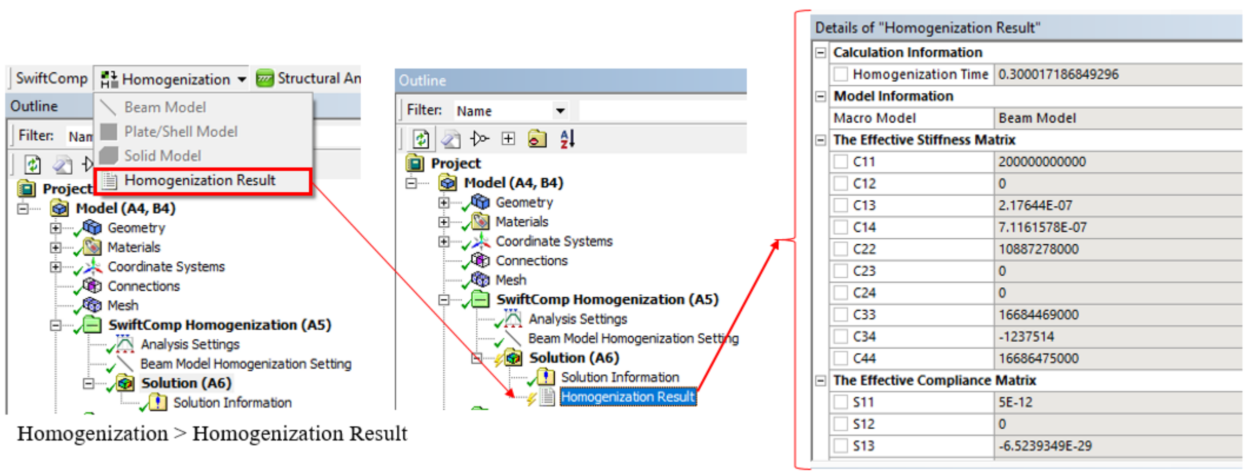


Figure 5.10 Homogenization Result

CHAPTER 6

6 Structural Analysis

6.1 Introduction

In structural analysis, macro structure is analyzed by applying the homogenized material properties determined through homogenization analysis as sectional properties for beam/shell element or material properties for the solid model.

6.2 Import Homogenization Result

You can import homogenization result from homogenization analysis to preform structural analysis as shown in Figure below. You need to select the body to assign the homogenization properties. The model information and effective properties are shown in the Details window.

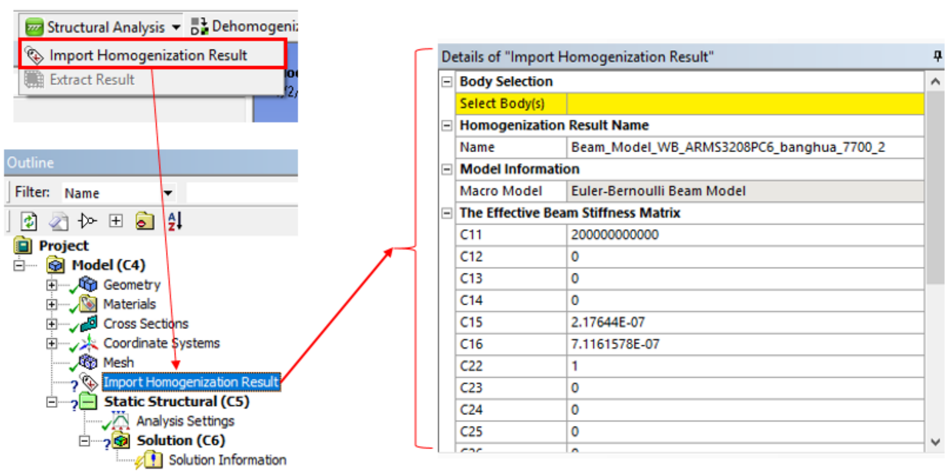


Figure 6.1 Structural Analysis for Beam Model

It is noted that different homogenization models correspond to different macroscopic models as discussed in the following:

6.2.1 Structural Analysis for Beam Model

In this case, the sectional properties (beam stiffness matrix) evaluated from Beam Model Homogenization Analysis are assigned to beam elements.

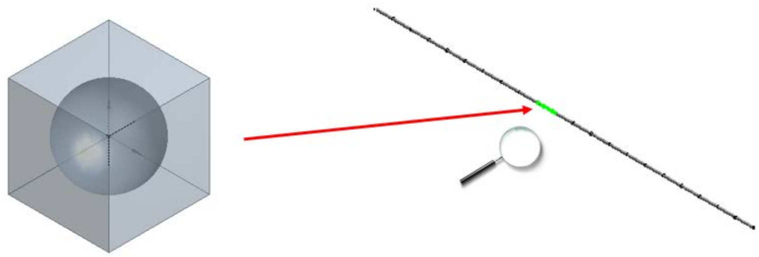


Figure 6.2 Structural Analysis for Beam Model

6.2.2 Structural Analysis for Plate/Shell Model

In this case, the sectional properties (plate stiffness matrix) evaluated from Plate/Shell Model Homogenization Analysis are assigned to shell elements.

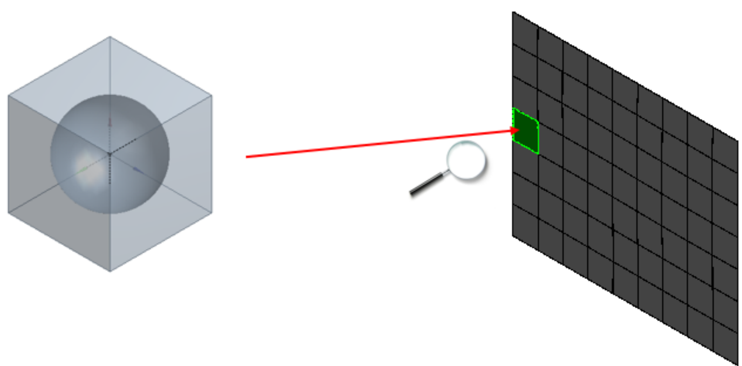


Figure 6.3 Structural Analysis for Plate/Shell Model

6.2.3 Structural Analysis for Solid Model

In this case, the effective material properties (anisotropic material properties) evaluated from Solid Model Homogenization Analysis are assigned to solid elements.

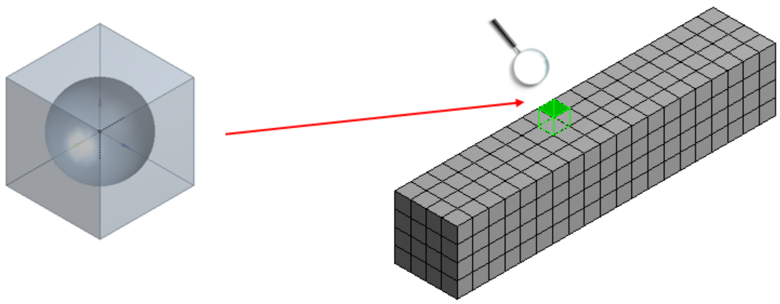


Figure 6.4 Structural Analysis for Solid Model

6.3 Extract Structural Analysis Result

Users can extract the structural analysis result as input for dehomogenization analysis. To perform extractions, on the SwiftComp toolbar, click Structural Analysis” button. Then choose Extract Result and select the element(s) for extraction. You will get the extraction result in Details window, i.e. Macro Displacement, Macro Rotations and Macro Strains (for Beam Model or Plate/Shell Model or Solid Model), which will be used in Dehomogenization Analysis.

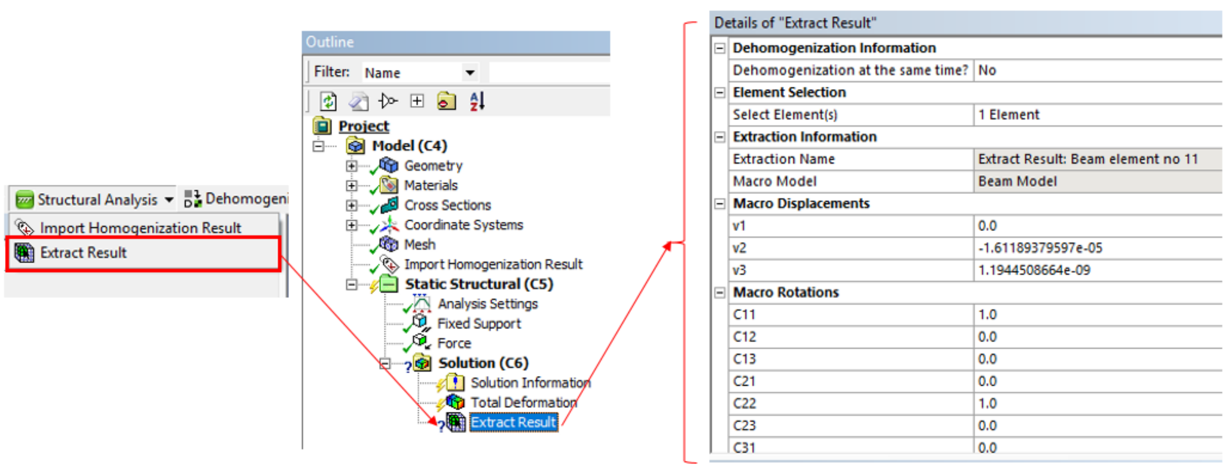


Figure 6.5 Extract Result in Structural Analysis

Users can also preform dehomogenization at the same time. Just select “Yes” for “Dehomogenization at the same time?” box. Gmsh will be used to visualize the dehomogenization result as shown in the Figure below.

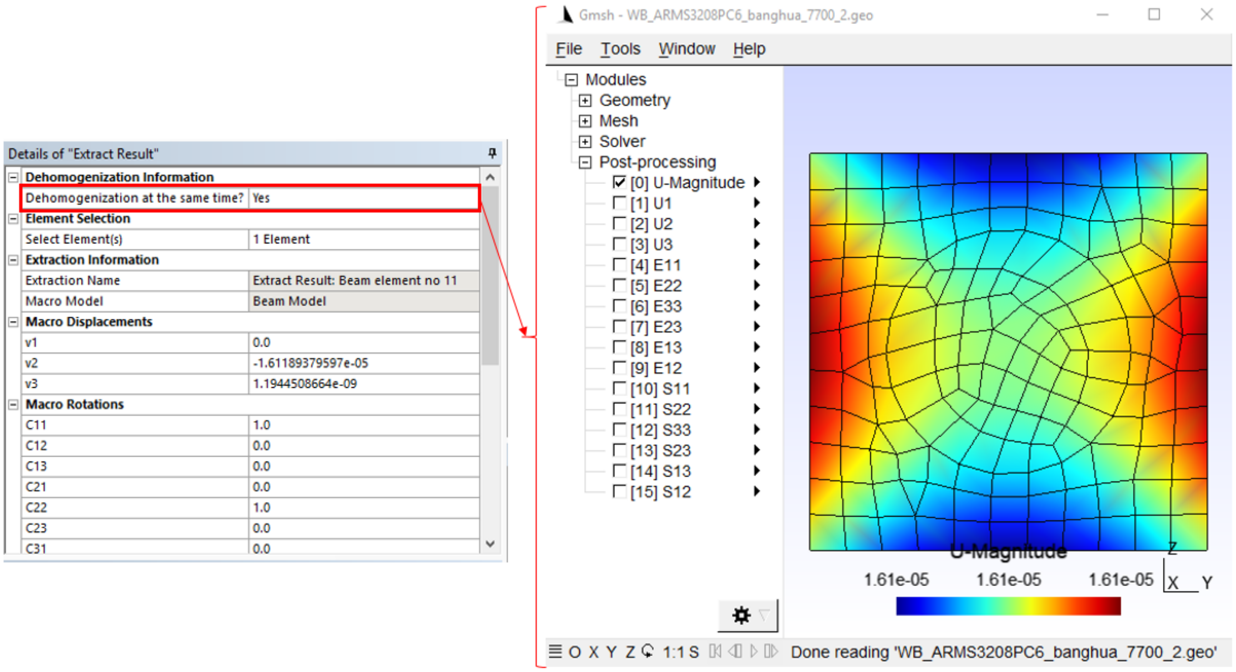


Figure 6.6 Dehomogenization at the same time and visualization by Gmsh

CHAPTER 7

7 Dehomogenization

7.1 Introduction

In dehomogenization analysis, a specific element composing the macro model is extracted to get the macro displacement, macro rotation and macro strain for determining local field, i.e. the local displacement/stress/strain for the SG.

First of all, a corresponding homogenization analysis must be run before carrying dehomogenization analysis. Also, the SwiftComp Dehomogenization system must connect to SwiftComp Homogenization system as shown in Figure 7.1.

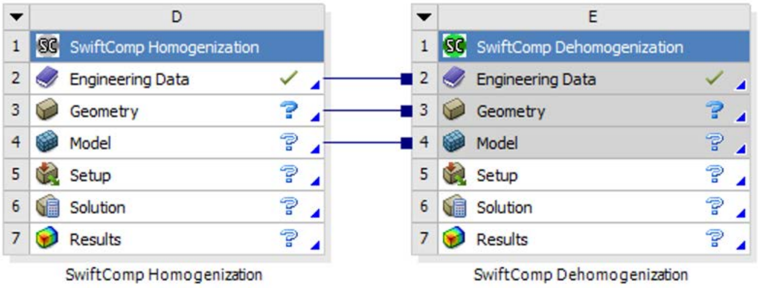


Figure 7.1 Connect SwiftComp Dehomogenization system to SwiftComp Homogenization system

To perform dehomogenization, on the SwiftComp toolbar, click "Dehomogenization" button. Then, choose Beam Model, Plate/Shell Model or Solid Model according to your analysis. Specify all the settings specific to Beam Model Dehomogenization Setting, Plate/Shell Model Dehomogenization Setting or Solid Model Dehomogenization Setting in the Details window. For example, Figure 7.2 shows the dehomogenization for Beam Model.

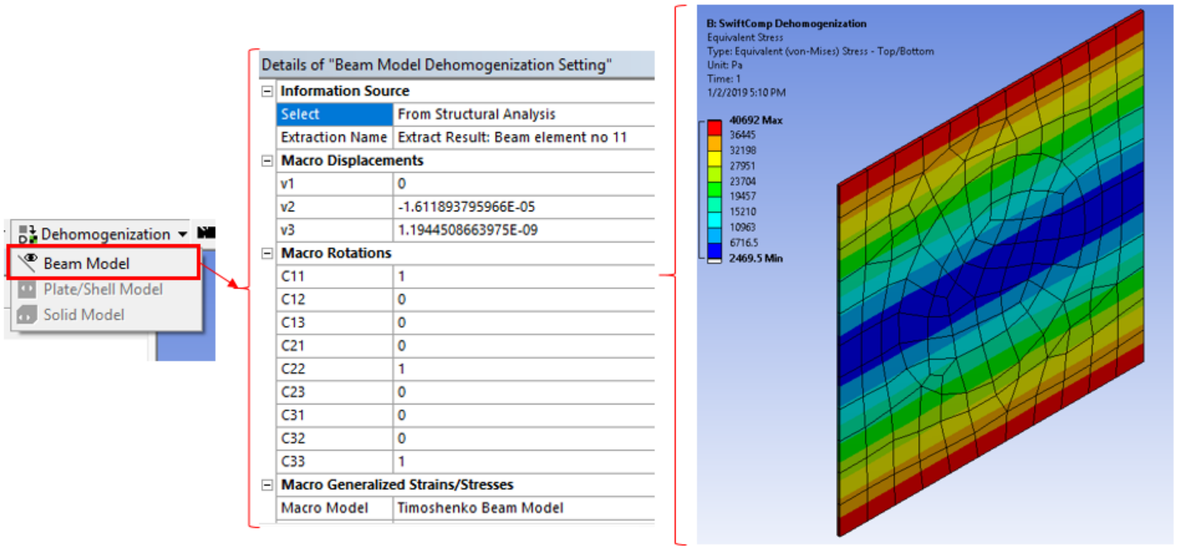


Figure 7.2 Dehomogenization for Solid Model

You can input all of the parameters by yourself or select From Structural Analysis

For all the model, you need to specify, Macro Displacements, Macro Rotations and Macro Generalized Strains/Stresses. Different models have different macro generalized strains/stress as discussed in the following.

7.2 Beam Model Dehomogenization

In Beam Model Dehomogenization Setting, the Macro Generalized Strains for Euler-Bernoulli beam model will be

$$\epsilon_{11}, \kappa_{11}, \kappa_{12}, \kappa_{13}$$

The Macro Generalized Strains for Timoshenko beam model will be

$$\epsilon_{11}, \gamma_{12}, \gamma_{13}, \kappa_{11}, \kappa_{12}, \kappa_{13}$$

The Macro Generalized Stresses for Euler-Bernoulli beam model will be

$$F_1, M_1, M_2, M_3$$

The Macro Generalized Stresses for Timoshenko beam model will be

$$F_1, F_2, F_3, M_1, M_2, M_3$$

7.3 Plate/Shell Model Dehomogenization

In Plate/Shell Model Dehomogenization Setting, the Macro Generalized Strains for Kirchhoff-Love Plate/Shell model will be

$$\epsilon_{11}, \epsilon_{22}, 2\epsilon_{12}, \kappa_{11}, \kappa_{22}, 2\kappa_{12}$$

The Macro Generalized Stresses for Kirchhoff-Love Plate/Shell model will be

$$N_{11}, N_{22}, N_{12}, M_{11}, M_{22}, M_{12}$$

7.4 Solid Model Dehomogenization

In Solid Model Dehomogenization Setting, the Macro Generalized Strains will be

$$\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}, 2\varepsilon_{23}, 2\varepsilon_{13}, 2\varepsilon_{12}$$

CHAPTER 8

8 Failure Analysis

8.1 Introduction

In failure analysis, you can perform failure strength analysis, failure envelop analysis or failure index analysis to get the strength, failure envelop or failure index/strength ratio of the SG.

First of all, a corresponding homogenization analysis must be run before carrying failure analysis. Also, the SwiftComp Failure Analysis system must connect to SwiftComp Homogenization system as shown in Figure 8.1.

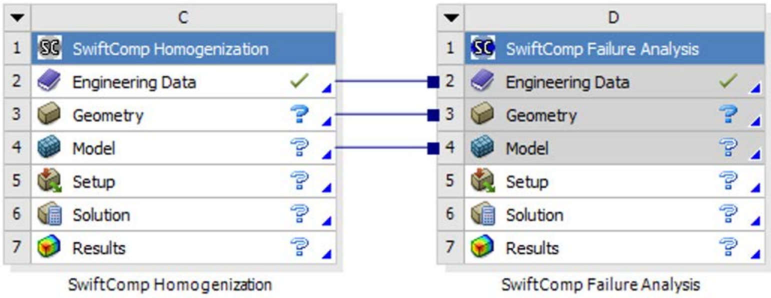


Figure 8.1 Connect SwiftComp Failure Analysis system to SwiftComp Homogenization system

To perform failure analysis, on the SwiftComp toolbar, click "Failure Analysis" button. Then, choose Failure Strength, Failure Envelop or Failure Index according to your analysis. Specify all the settings specific to Failure Strength Analysis, Failure Envelop Analysis or Failure Index Analysis in the Details window.

For all of the analysis, you need to specific Failure Criterion, Characteristic Length, and Generalized Stress/Strain.

For Failure Criterion, if the material is isotropic, you can choose the max principal stress criterion, the max principal strain criterion, the max shear stress criterion, the max shear strain criterion, or the Mises criterion. If the material is not isotropic (transversely isotropic, orthotropic or general anisotropic), you can choose the max stress criterion, the max strain criterion, the Tsai-Hill criterion, the Tsai-Wu criterion, or the Hashin criterion.

It is noted that users can define parameters for each criterion at in Failure Criterion table or at the Engineering Data.

Characteristic Length is a real number indicating the characteristic length used in the nonlocal approach for initial failure analysis. If Characteristic Length is equal to zero, the local approach based on element averaged values will be used.

Generalized Stress/Strain indicates whether the strength is expressed in terms of generalized stresses or generalized strains. If it is equal Stress, the strength is expressed in terms of generalized stresses; if it is equal to Strain, it is expressed in terms of generalized strains.

8.2 Failure Strength Analysis

For Failure Strength Analysis, the results are initial failure strengths in both tensile and compressive directions for the SG.

An example of failure strength analysis is shown in Figure 8.2.

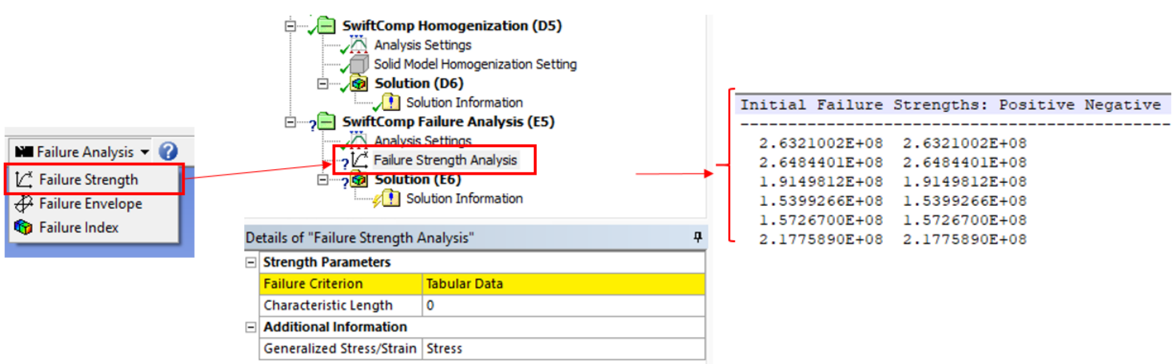


Figure 8.2 Example of Failure Strength Analysis

8.3 Failure Envelope Analysis

For Failure envelope Analysis, you need to also specify the two load directions that one would like to predict a failure envelope for. The default directs are sigma11 and sigma22. In addition, you need to enter the number of divisions along one direction, which is used for searching for failure envelope points. The total failure envelope points will be about 4 times the number of divisions. Although one is free to use any number for the number of divisions, it is recommend to use 10 for a balance between efficiency and accuracy in describing the failure envelope.

The results are failure envelope points with the first number being the number of the failure point, the two trailing read numbers being the values for corresponding two given directions for predicting the failure envelope.

An example of failure Envelope analysis is shown in Figure 8.3.

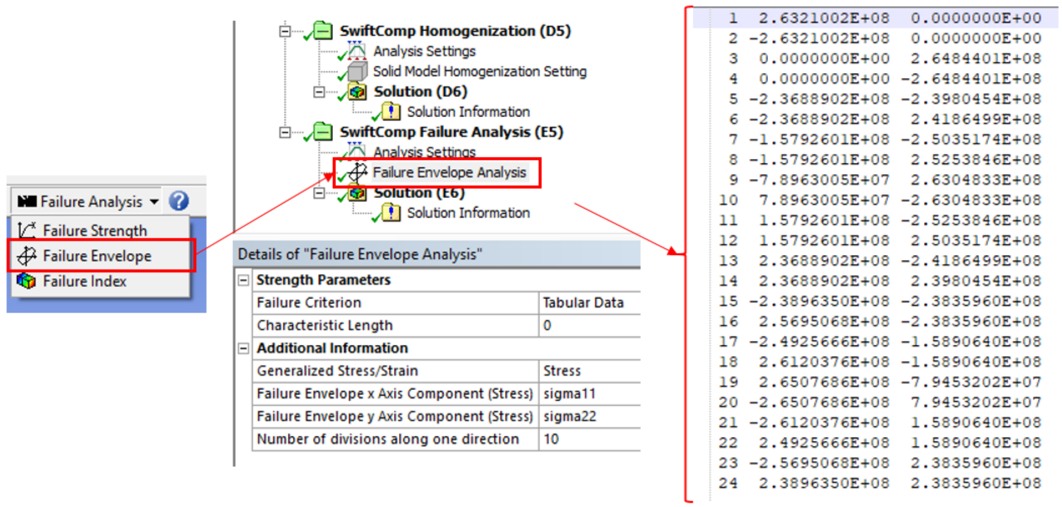


Figure 8.3 Example of Failure Envelope Analysis

8.4 Failure Index Analysis

For Failure Index Analysis, you need to also provide the loads used to compute the strength ratio and the failure index. These values can be given in terms of generalized stresses or generalized strains depending on whether the required strength outputs are in generalized stresses or generalized strains. By default, sigma11, sigma22, sigma33, sigma23, sigma13, and sigma23 are required.

The results are failure index and strength ratio for each element. The first number is an integer indicating the element number and the following two numbers are the initial failure index and the initial strength ratio for each element under given loads.

An example of failure Index analysis is shown in Figure 8.4

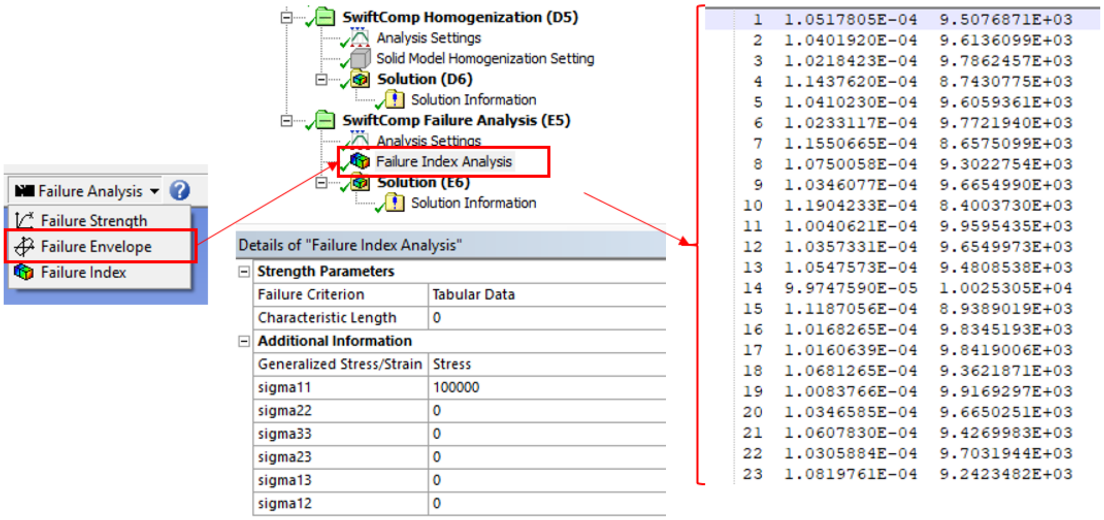


Figure 8.4 Example of Failure Index Analysis