ANSYS-SwiftComp GUI

Version 1.1 User's Manual

August, 2017

Banghua Zhao
Fang Jiang
Zheng Ye
Wenbin Yu

TABLE OF CONTENTS

Page #

1.0 GENERAL INFORMATION

1.0 GENERAL INFORMATION

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, called ANSYS-SwiftComp GUI is developed. This manual focuses on explaining how to use ANSYS-SwiftComp GUI. The ANSYS-SwiftComp GUI is modified from the ANSYS-VAMUCH GUI developed by Zheng Ye and part of the macros are also inherited from ANSYS-VABS GUI developed by Fang Jiang. Both ANSYS-VAMUCH GUI and ANSYS-VABS GUI are superseded by ANSYS-SwiftComp GUI.

1.1 Installation and Uninstallation

The ANSYS-SwiftComp GUI package, *ANSYS-SwiftComp_GUI.zip*, can be downloaded from cdmHUB (https://cdmhub.org/resources/1136). To successfully install and use ANSYS-SwiftComp GUI, follow step 1 to step 4 as shown below.

Step 1. Unzip the package to any location in your computer.

Step 2. In the unzipped folder, find the windows batch file named *Install.bat*. See Figure 1.1. Right click the file and select *Run as administrator*. The Windows Command Prompt will open. Type the ANSYS version number as three digits. For example, if you have ANSYS 18.0, type 180. Then ANSYS-SwiftComp GUI will be automatically installed in your computer. (Here, it is assumed that your computer has already installed the ANSYS version that you have entered).

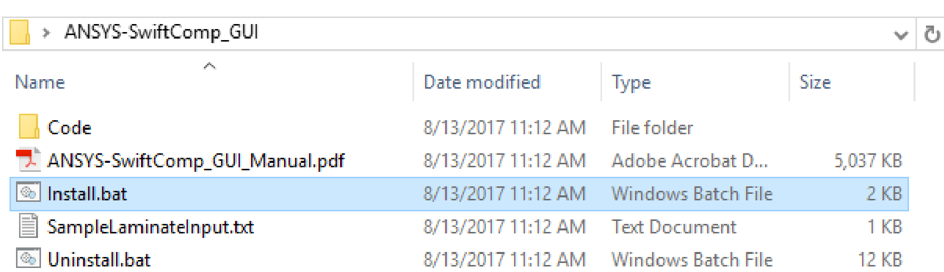


Figure 1.1

Step 3. Launch *Mechanical APDL Product Launcher ###* (### is the version number of the ANSYS). Type in your Working Directory and your Job Name. For example, Launch *Mechanical APDL Product Launcher 14.5*. Type in *C:\Users\bangh\Desktop\project* for Working Directory and *Try* for Job Name (see Figure 1.2).

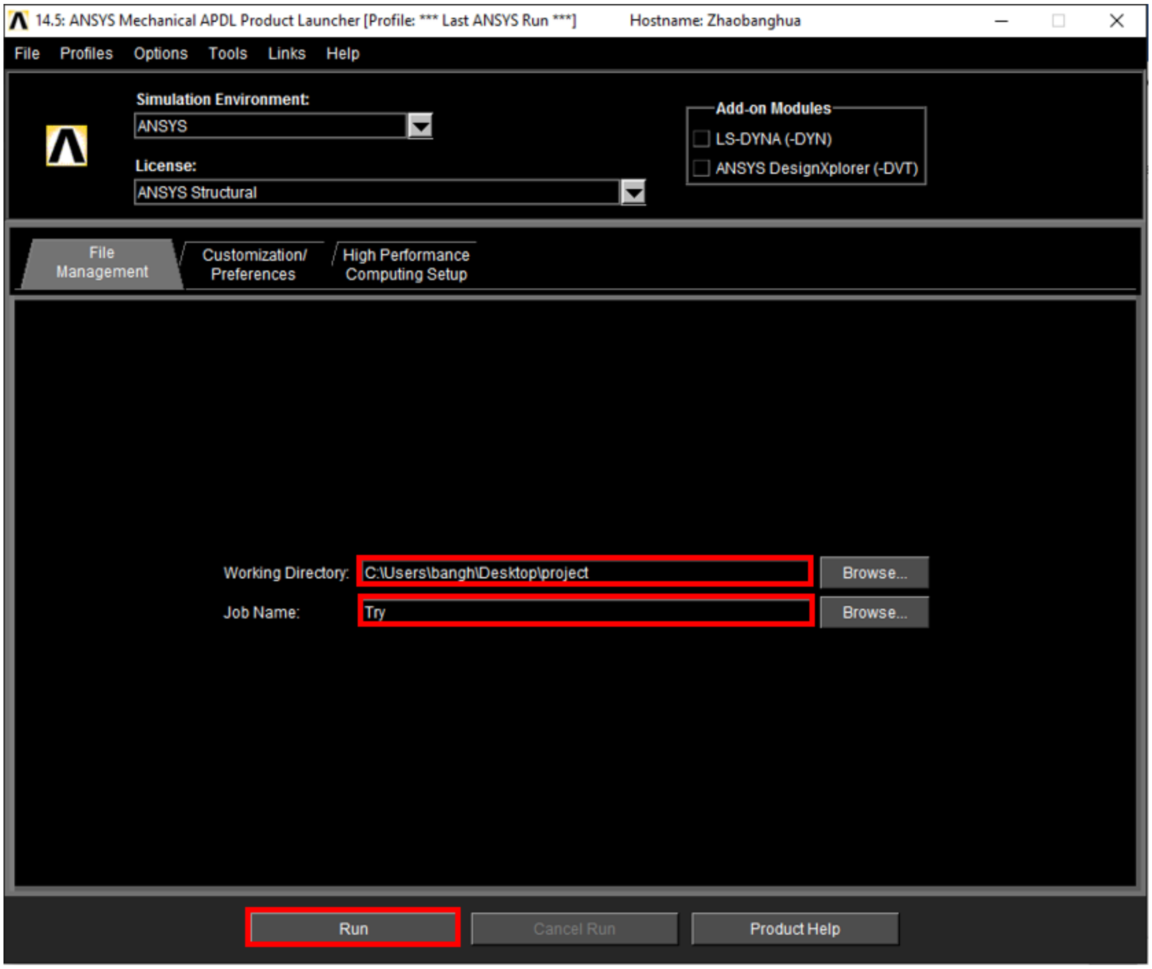


Figure 1.2 Mechanical APDL Product Launcher 14.5

Step 4. Click *Run* bottom (see Figure 1.2) and check the Main Menu on the left side of ANSYS window. If everything is correctly followed from Step 1 to Step 4, newly added menus and functions will appear as shown in the red frames in Figure 1.3.

Of course, you also need to have SwiftComp™ already installed on your computer. 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.

Please note that all the files generated will be stored in the work directory and the valid SwiftComp license requested from AnalySwift should also be stored in your work directory.

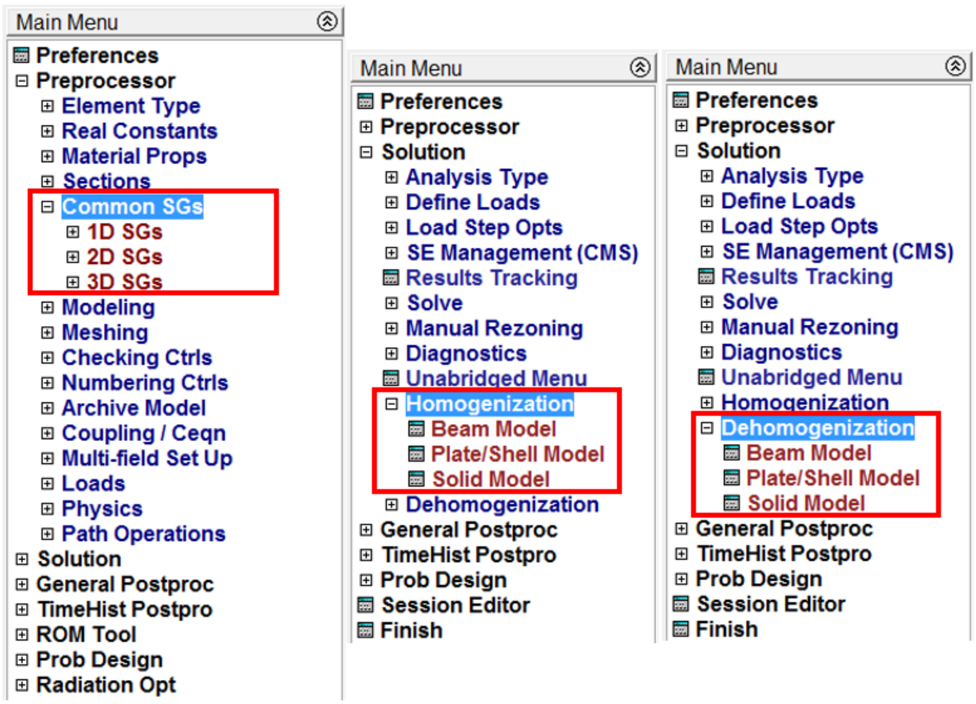


Figure 1.3: Customized menu structure for ANSYS-SwiftComp GUI

To uninstall ANSYS-SwiftComp GUI, follow the instructions below.

In the unzipped folder (the folder from step 1), find the windows batch file named *Uninstall.bat*. See Figure 1.4. Right click the file and select *Run as administrator*. The Windows Command Prompt will open. Type the ANSYS version number as three digits. For example, if you have ANSYS 18.0, type 180. Then ANSYS-SwiftComp GUI will be automatically uninstalled in your computer. (Here, it is assumed that your computer has already installed ANSYS-SwiftComp GUI and ANSYS is not running.)

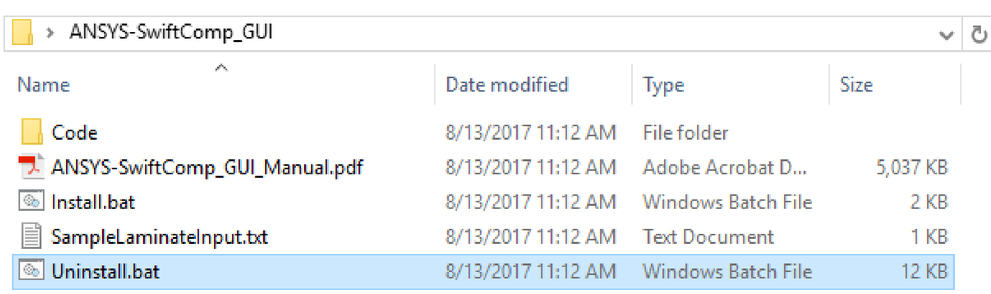


Figure 1.4

1.2 ANSYS-SwiftComp GUI Overview

ANSYS-SwiftComp GUI is a graphic user interface integrated in the Main Menu of ANSYS, which can use the preprocessing and postprocessing capability of ANSYS as a user friendly GUI for SwiftComp™. The newly added functions specific for SwiftComp™ are located in the Preprocessor menu and the Solution menu in ANSYS.

The first level menu of ANSYS-SwiftComp GUI composes of three major parts: Preprocessor, Solution and General Postproc.

Preprocessor

Material Props: Users can define isotropic, orthotropic or general anisotropic materials properties under this menu by using Material Models function.

Modeling: Users can build their own models by elementary entities which contain all basic geometry elements (e.g. point, line, surface, and volume).

Common SGs: Users can create their model through Common SGs, which provides several common fundamental building blocks of composite materials and structures.

Mesh: Generate mesh according to the geometry of model, and user can choose different mesh algorithms to meet their needs.

Solution

Homogenization: Users can invoke SwiftComp™ to compute the effect properties for different structural models (beams, plates/shells or 3D solids).

Dehomogenization: Providing the global behavior, users can invoke SwiftComp™ to compute the local fields including displacements, stresses and strains, and contour plots will be automatically generated after calculation is completed.

General Postproc

Plot Results: After Dehomogenization, users can choose different contour plots (displacements, stresses and strains) in this menu to visualize the results.

1.3 ANSYS-SwiftComp GUI Conventions

Coordinate Convention

The relation of coordinate system between SwiftComp™ and ANSYS is shown in Table 1.1.

Table 1.1 Relation of coordinate system between SwiftComp™ and ANSYS.

SwiftComp	ANSYS
x_1, y_1	X
x_2, y_2	Y
x_3, y_3	Z

Note: Users can create their 2D SGs in XY working plane without worrying about the coordinate convention in SwiftComp™ (In SwiftComp™, 2D SG should be created in YZ working plane. See *SCManual.pdf* for more information). After clicking Homogenization, the model will be automatically rotated to YZ plane.

Element Convention

In ANSYS-SwiftComp GUI, particular element types are used for different element shapes and number of nodes. The relation is shown in Table 1.2.

Table 1.2 Relation of element shape and number of nodes to element type

Element Shape and Number of Nodes	Element Type
6-noded triangular elements	PLANE2
4-noded quadrilateral elements	PLANE182 / PLANE42
8-noded quadrilateral elements	PLANE183 / PLANE82
4-noded tetrahedral elements	SOLID285
10-noded tetrahedral elements	SOLID187 / SOLID92
8-noded brick elements	SOLID185 / SOLID45
20-noded brick elements	SOLID186 / SOLID95

Format of Input File for 1D Structure Genome

The first line has one integer (without indent):

nsegment
denoting the number of laminate segments in this file.

The next *nsegment* blocks are the layup information from bottom to top for each laminate segment.

For each block, four integers are arranged at the first line as (without indent):

nlayer Tab *m* Tab *s* Tab *n*

where the number *nlayer* is a positive integer indicting the number of layers (not counting those formed by repetition and symmetry) in this segment. The number *m* is a positive integer indicting the number of repetition before symmetry. The number *s* is either 0 or 1 indication symmetry (0: non-symmetry, 1: symmetry). The number *n* is a positive integer indicting the number of repetition after symmetry is applied. If there is no repetition, *m* and *n* are equal to unity. Among *nlayer*, *m*, *s* and *n*, a Tab is typed.

The next *nlayer* lines are arranged as following (without indent):

thickness Tab *orientation* Tab *mate_id*
where the number *thickness* is a positive real number representing the thickness of the layer. The number *orientation* is a real number indicating the fiber orientation of the layer. The number *mate_id* is an integer indicating the material number of the layer. Among *thickness, orientation*, and *mate_id*, a Tab is typed.

For example, the input file for laminate $[(0/-45/60)_{2s}/(30/-30)_{s}]$ is shown in Figure 1.5. The release package contains a sample laminate file *SampleLaminateInput.txt*.

```
2
3       2       1       1
0.05    0       1
0.05    -45     1
0.05    60      1
2       1       1       1
0.25E-01        30      2
0.25E-01        -30     2
```

Figure 1.5

Other Conventions

The only way to create 1D SG is to use 1D SGs menu in Common SGs menu. And 1D SGs do not need to mesh.

After homogenization, the effective properties computed by SwiftComp™ stored in **.sc.k* (* is the jobname) will pop up to user. Pleaser close this file before other operations. If user does not close the file, the cursor will appear busy and no other operations can be performed.

Always check ANSYS Mechanical APDL ### (### is the version number) Output Window after homogenization or dehomogenization. Only after SwiftComp™ finished successfully, can user continue other operations.

1.4 Organization of the Manual

Chapter One introduces the general information about ANSYS-SwiftComp GUI, the functions in each part and some conventions.

Chapter Two describes how to use Common SGs to quickly build the geometry and mesh for the model. After running SwiftComp™, the results of homogenization and dehomogenization can be automatically shown in the GUI window.

Chapter Three introduces how to create Square Pack Microstructures and Spherical Inclusion Microstructures, and a rectangular SG with two arbitrary inclusions by elementary entities functions provided by ANSYS, which requires users to create each geometry entity. These three examples should give users a good preparation to build user-defined models according to different analysis needs.

2.0 CREATE COMMON MODEL

2.0 CREATE COMMON MODEL

ANSYS-SwiftComp GUI provides a convenient way to create some common SG models. Engineers can easily create the geometry and mesh of these models, and invoke SwiftComp™ to perform homogenization and dehomogenization for different composites with arbitrary fundamental building blocks (aka SGs).

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

1D SGs: layered materials, laminates.

2D SGs: square pack microstructures (with interphase region), hexagonal pack microstructures (with interphase region).

3D SGs: square pack microstructures, spherical inclusion microstructures.

You can add your own common SGs similarly as what has been done in the APDL macros. Or you can let the developers of this GUI to know your requirements so that we can add other common SGs if needed. In this chapter, we will use some common SGs for different structures (e.g. solid and laminate) to illustrate how to use ANSYS-SwiftComp GUI to perform homogenization and dehomogenization. For solid models, we will show 2D Square Pack Microstructure model and 3D Spherical Inclusion Microstructure. For laminates, we will show three different ways to generate 1D SG of laminates.

First, users need to Launch *Mechanical APDL Product Launcher ###* (### is the version number of the ANSYS). Type in your Working Directory, your Job Name and click *Run* bottom.

2.1 Square Pack Microstructure (2D SG)

Define Materials

Preprocessor → Material Props → Material Models → Structural → Linear → Elastic.

Choose materials type (e.g. Isotropic, Orthotropic, or Anisotropic).

In this example, assume fiber and matrix are both isotropic materials (Material 1: Young's modulus E = 379.3 GPa, Poisson's ratio υ = 0.1; Material 2: Young's modulus E = 68.3 GPa, Poisson's ratio υ = 0.3).

For fiber, add materials properties for Material Model Number 1 (see Figure 2.1).

For matrix, first click Material in the menu bar to create Material Model Number 2. And then add materials properties for Material Model Number 2 (see Figure 2.2).

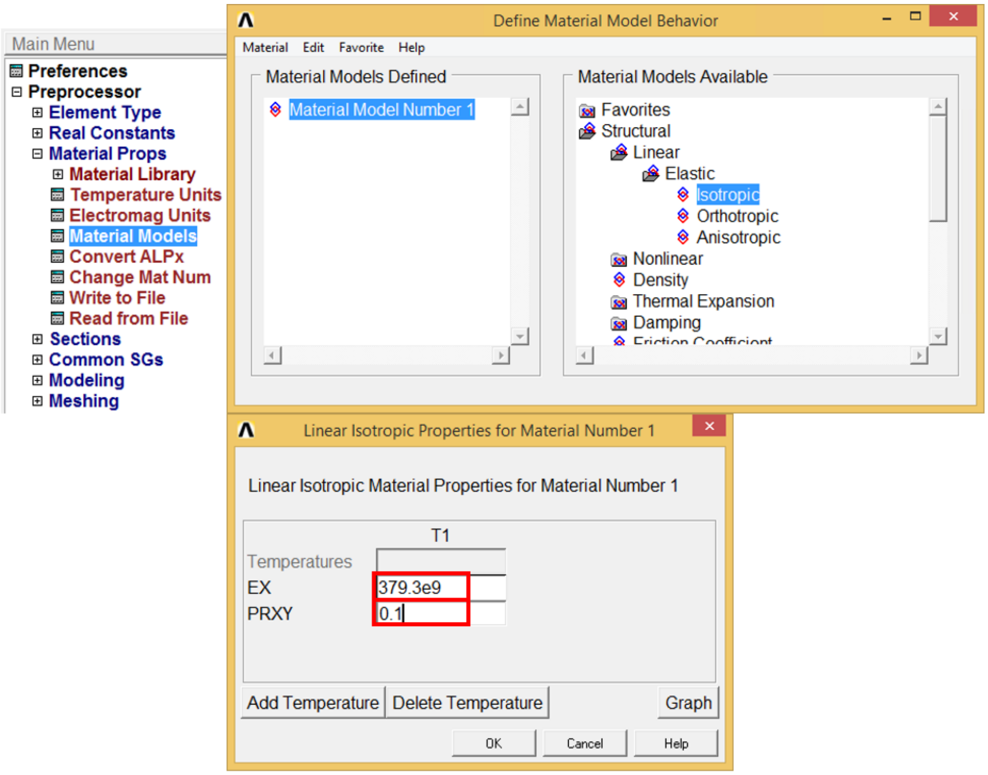


Figure 2.1

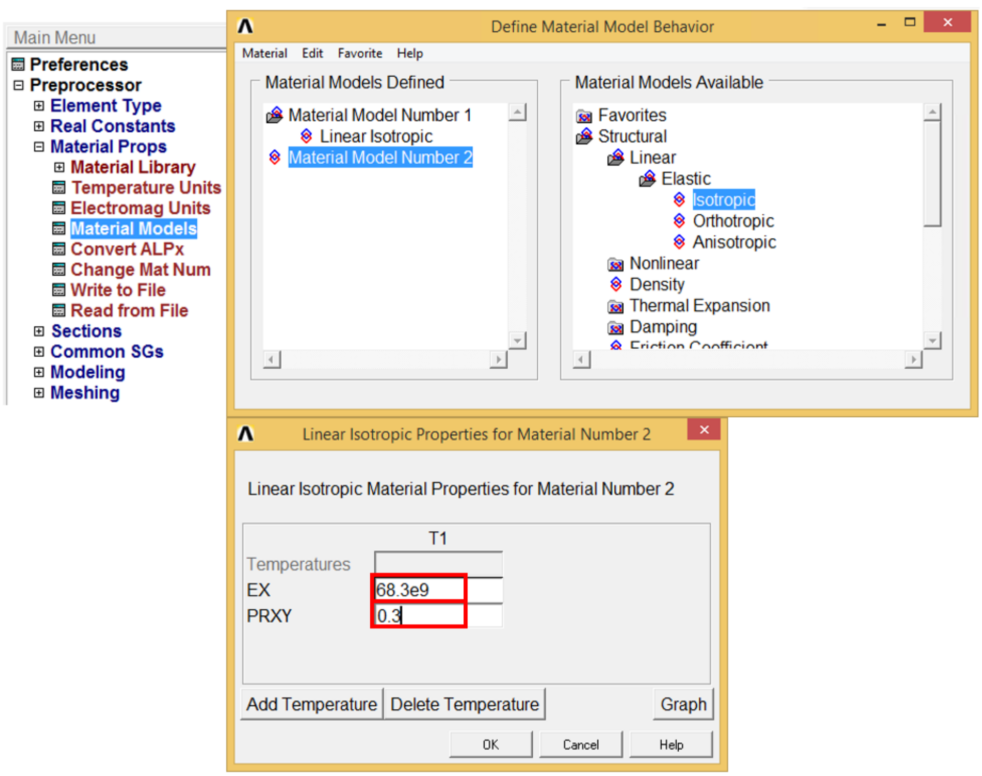


Figure 2.2

Generate Common Model Geometry and Mesh

Preprocessor → Common SGs → 2D SGs → Square Pack.

Input volume fraction of fiber, and keep volume fraction of interphase empty or zero if there is no interphase region in your model. Select corresponding material number to fiber and matrix. If there is no interphase, keep the Material # for Interphase empty or zero. Select element type and mesh size (see Figure 2.3).

Click OK. The SG represented as a square with a circle in the center will be shown on the screen (see Figure 2.4).

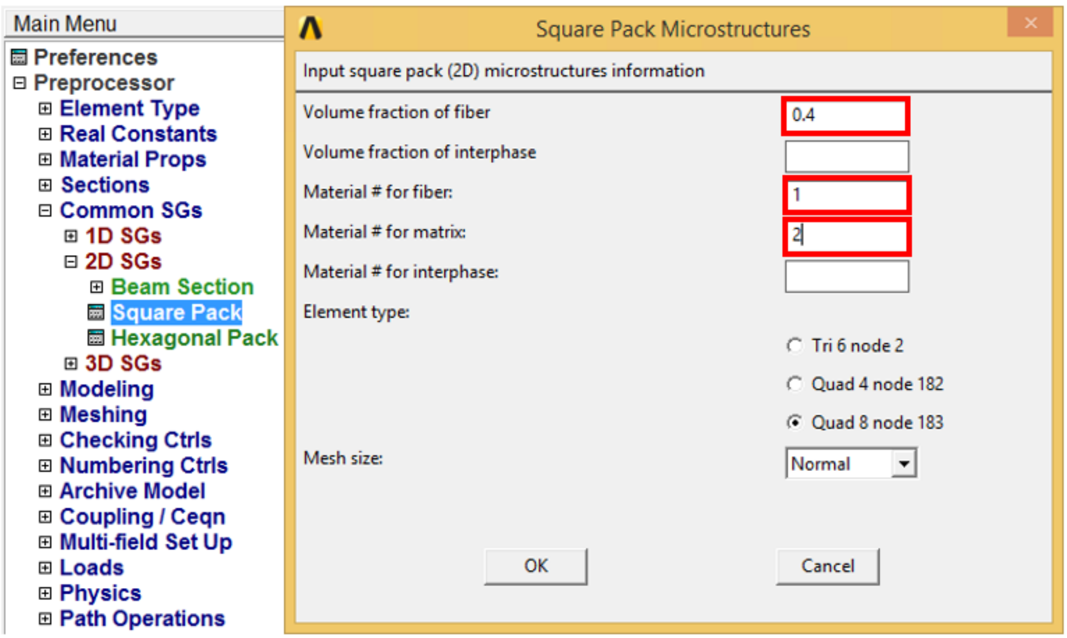


Figure 2.3

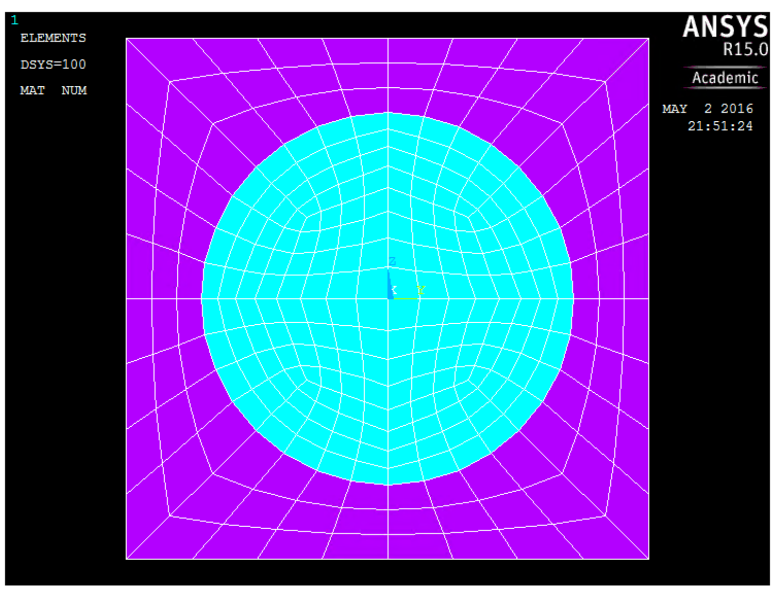


Figure 2.4

Homogenization

Preprocessor → Solution → Homogenization → Solid Model.

Please read the manual of SwiftComp™ to see the meaning of each parameters.

In this example, we consider that the macroscopic structural model is Solid Model, keep all default settings, click Ok (see Figure 2.5).

Wait for preparing the input file of SwiftComp™ and SwiftComp™ to finish the computation. Note that if the SG contains a large number of elements, it will take some time to prepare the input file and to finish the computation. Then the effective properties will pop up automatically (see Figure 2.6).

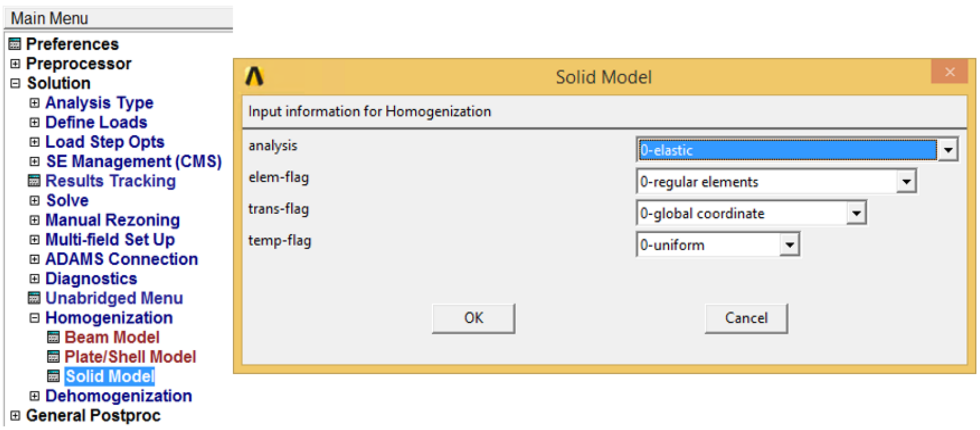
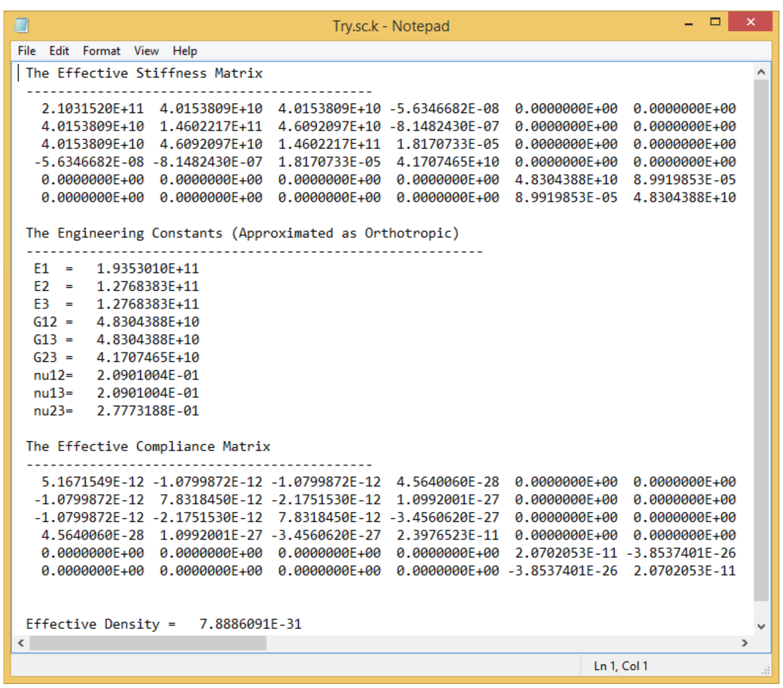


Figure 2.5




```
The Effective Stiffness Matrix
-------------------------------------------
  2.1031520E+11  4.0153809E+10  4.0153809E+10 -5.6346682E-08  0.0000000E+00  0.0000000E+00
  4.0153809E+10  1.4602217E+11  4.6092097E+10 -8.1482430E-07  0.0000000E+00  0.0000000E+00
  4.0153809E+10  4.6092097E+10  1.4602217E+11  1.8170733E-05  0.0000000E+00  0.0000000E+00
 -5.6346682E-08 -8.1482430E-07  1.8170733E-05  4.1707465E+10  0.0000000E+00  0.0000000E+00
  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  4.8304388E+10  8.9919853E-05
  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  8.9919853E-05  4.8304388E+10

The Engineering Constants (Approximated as Orthotropic)
----------------------------------------------------------
 E1  =   1.9353010E+11
 E2  =   1.2768383E+11
 E3  =   1.2768383E+11
 G12 =   4.8304388E+10
 G13 =   4.8304388E+10
 G23 =   4.1707465E+10
 nu12=   2.0901004E-01
 nu13=   2.0901004E-01
 nu23=   2.7773188E-01

The Effective Compliance Matrix
-------------------------------------------
  5.1671549E-12 -1.0799872E-12 -1.0799872E-12  4.5640060E-28  0.0000000E+00  0.0000000E+00
 -1.0799872E-12  7.8318450E-12 -2.1751530E-12  1.0992001E-27  0.0000000E+00  0.0000000E+00
 -1.0799872E-12 -2.1751530E-12  7.8318450E-12 -3.4560620E-27  0.0000000E+00  0.0000000E+00
  4.5640060E-28  1.0992001E-27 -3.4560620E-27  2.3976523E-11  0.0000000E+00  0.0000000E+00
  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.0702053E-11 -3.8537401E-26
  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00 -3.8537401E-26  2.0702053E-11


Effective Density =   7.8886091E-31
```

Figure 2.6

Note: In rare situations that error may occur. So always check ANSYS Mechanical APDL ### (### is the version number) Output Window after homogenization run (see Figure 2.7). Only after SwiftComp™ finished successfully, can user continue other operation.

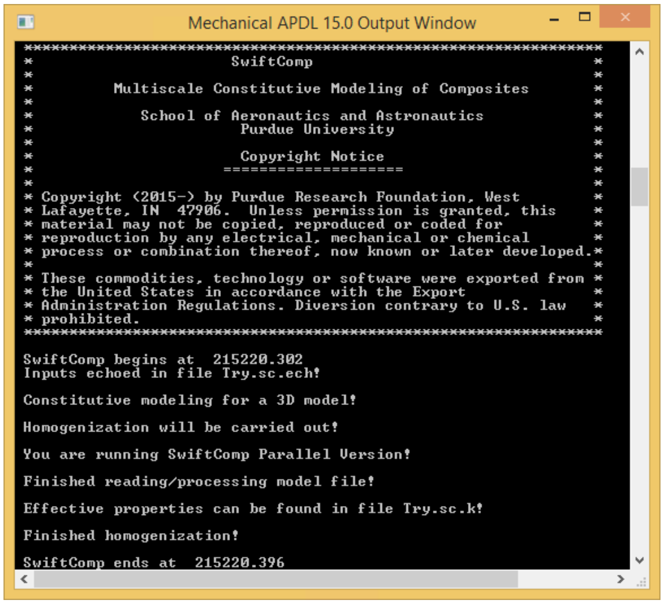


Figure 2.7

Dehomogenization

Preprocessor → Solution → Dehomogenization → Solid Model.

Input the global behavior from the macroscopic structural analysis. Please refer to SwiftComp™ manual for meaning of the global behavior parameters. Click Ok (see Figure 2.8).

Wait for SwiftComp™ to finish the computation. The post-processing results will be automatically loaded. The default value is the magnitude of displacement as in Figure 2.9.

Contour plots are available for all local fields including three displacement components and its magnitude, six strain components and six stress components. General Postproc → Plot Results → Contour Plot.

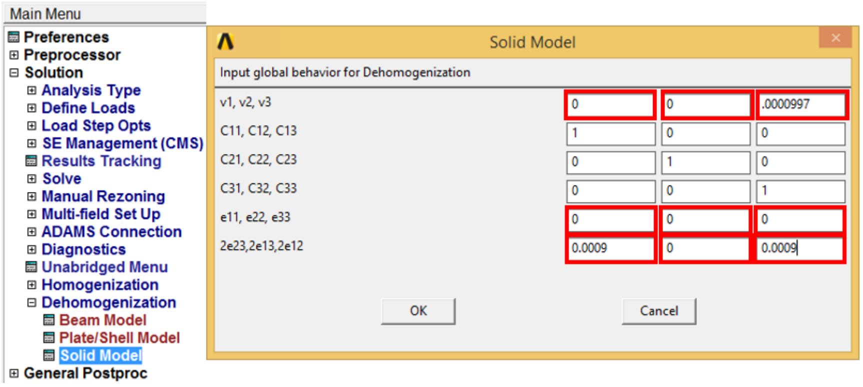


Figure 2.8

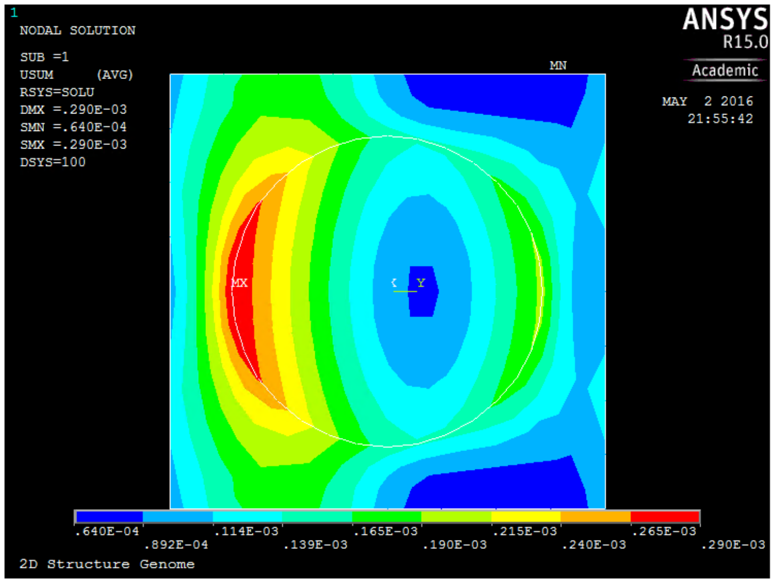


Figure 2.9

2.2 Spherical Inclusions Microstructure (3D SG)

Define Materials

This step is the same as defining materials for 2D common SG in the previous section.

Generate Common Model Geometry and Mesh

Preprocessor → Common SG → 3D SGs → Spherical Inclusions (see Figure 2.10).

Choose Spherical Inclusion and input all the parameters according to your model, click Ok (see Figure 2.11).

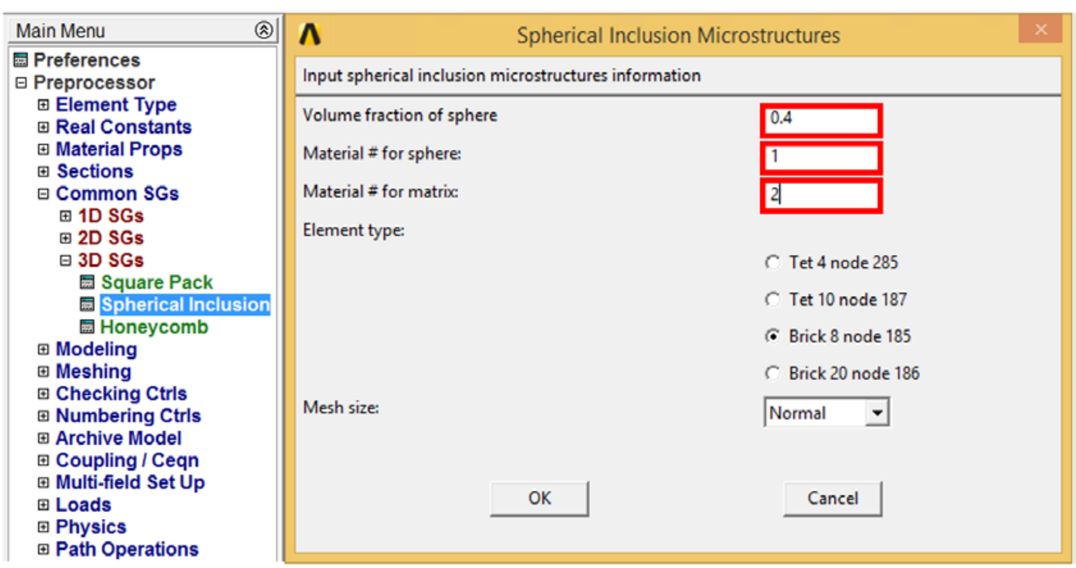


Figure 2.10

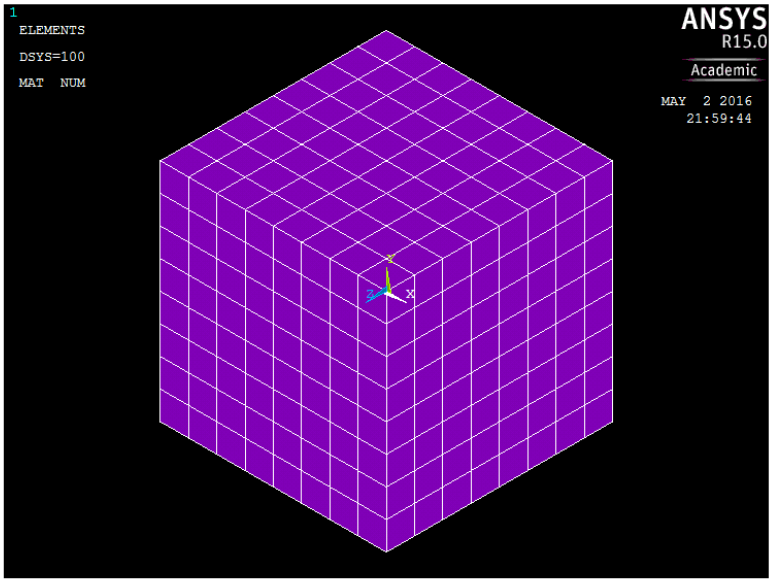


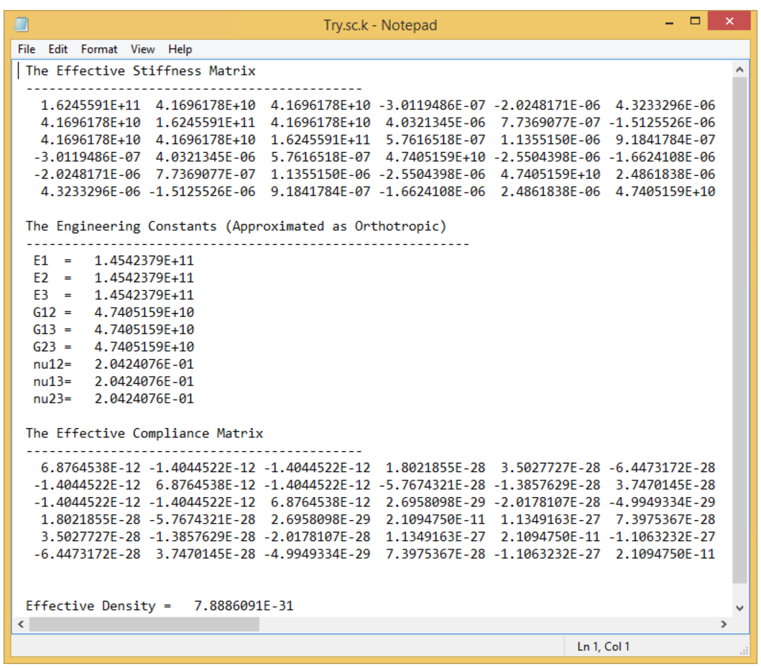
Figure 2.11

Homogenization

Preprocessor → Solution → Homogenization → Solid Model.

Please read the manual of SwiftComp™ to see the meaning of each parameters. Input the parameters for your model (Figure 2.5).

Wait for preparing the input file of SwiftComp™ and SwiftComp™ to finish the computation. Note that if the SG contains a large number of elements, it will take some time to prepare the input file and finish the computation. Then the effective properties will pop up automatically (see Figure 2.12).



```
The Effective Stiffness Matrix
----------------------------------------------
  1.6245591E+11  4.1696178E+10  4.1696178E+10 -3.0119486E-07 -2.0248171E-06  4.3233296E-06
  4.1696178E+10  1.6245591E+11  4.1696178E+10  4.0321345E-06  7.7369077E-07 -1.5125526E-06
  4.1696178E+10  4.1696178E+10  1.6245591E+11  5.7616518E-07  1.1355150E-06  9.1841784E-07
 -3.0119486E-07  4.0321345E-06  5.7616518E-07  4.7405159E+10 -2.5504398E-06 -1.6624108E-06
 -2.0248171E-06  7.7369077E-07  1.1355150E-06 -2.5504398E-06  4.7405159E+10  2.4861838E-06
  4.3233296E-06 -1.5125526E-06  9.1841784E-07 -1.6624108E-06  2.4861838E-06  4.7405159E+10

The Engineering Constants (Approximated as Orthotropic)
-------------------------------------------------------------
 E1  =   1.4542379E+11
 E2  =   1.4542379E+11
 E3  =   1.4542379E+11
 G12 =   4.7405159E+10
 G13 =   4.7405159E+10
 G23 =   4.7405159E+10
 nu12=   2.0424076E-01
 nu13=   2.0424076E-01
 nu23=   2.0424076E-01

The Effective Compliance Matrix
----------------------------------------------
  6.8764538E-12 -1.4044522E-12 -1.4044522E-12  1.8021855E-28  3.5027727E-28 -6.4473172E-28
 -1.4044522E-12  6.8764538E-12 -1.4044522E-12 -5.7674321E-28 -1.3857629E-28  3.7470145E-28
 -1.4044522E-12 -1.4044522E-12  6.8764538E-12  2.6958098E-29 -2.0178107E-28 -4.9949334E-29
  1.8021855E-28 -5.7674321E-28  2.6958098E-29  2.1094750E-11  1.1349163E-27  7.3975367E-28
  3.5027727E-28 -1.3857629E-28 -2.0178107E-28  1.1349163E-27  2.1094750E-11 -1.1063232E-27
 -6.4473172E-28  3.7470145E-28 -4.9949334E-29  7.3975367E-28 -1.1063232E-27  2.1094750E-11


Effective Density =   7.8886091E-31
```


Figure 2.12

Dehomogenization

Preprocessor → Solution → Dehomogenization → Solid Model.

Input the global behavior from the macroscopic structural analysis. Click Ok (see Figure 2.8). Wait for SwiftComp™ to finish the computation. The post-processing results will be automatically loaded. The default value is the magnitude of displacement as in Figure 2.13.

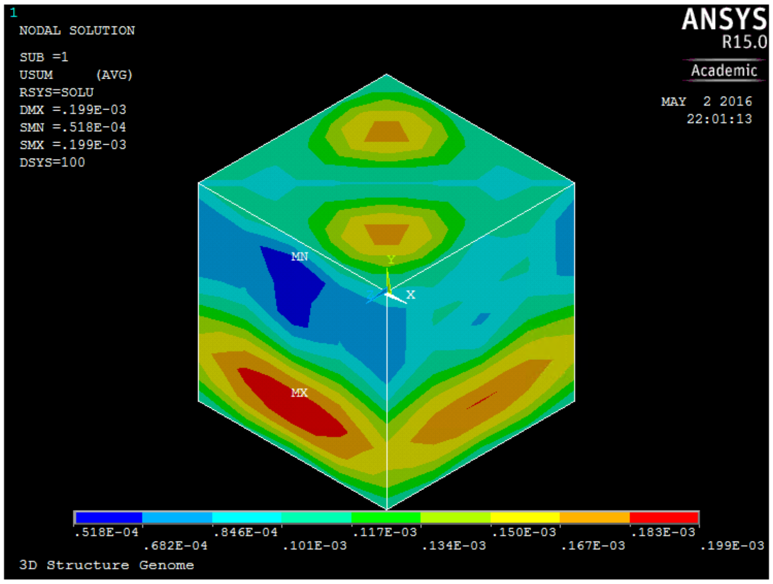


Figure 2.13

2.3 Laminate (1D SG)

Define Materials

Preprocessor → Material Props → Material Models → Structural → Linear → Elastic.

Choose material type which could be isotropic, orthotropic, or general anisotropic, although we usually describe a lamina as an orthotropic material. We assume the following two different lamina properties:

Material Model Number 1: E_1 = 250 GPa, E_2 = 50 GPa, E_3 = 50 GPa
G_{12} = 5 GPa, G_{13} = 2 GPa, G_{23} = 2 GPa
υ_{12} = 0.25, υ_{13} = 0.25, υ_{23} = 0.25

Material Model Number 2: E_1 = 200 GPa, E_2 = 20 GPa, E_3 = 20 GPa
G_{12} = 5 GPa, G_{13} = 2 GPa, G_{23} = 2 GPa
υ_{12} = 0.3, υ_{13} = 0.3, υ_{23} = 0.3

Then we will use three different ways to create 1D SG for different user-defined laminates.

Generate Common 1D SGs

Preprocessor → Common SG → 1D SGs->Fast Generate.

Choose Fast Generate function. This function will provide a fast way to generate laminates. Users can use simple notation to define ply sequence from the bottom to the top. For example, the notation '[0/-45/60]2s' means the following sequence [0/-45/60/0/-45/60/60/-45/0/60/-45/0]. The number after bracket means the repeating times and "s" means symmetry. Choose material number for each lamina and assign thickness for each ply (see Figure 2.14).
Note: For Solid Model, 2-noded elements will be sufficient. And for Plate/Shell Model, 5-noded elements are needed.

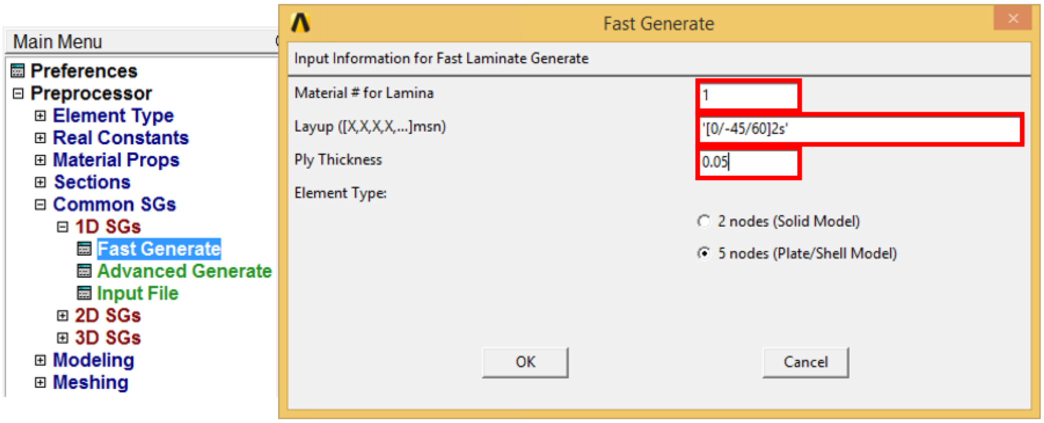


Figure 2.14

After inputting all the parameters, click Ok. The 1D SG for laminates has been created as shown in Figure 2.15. Note that Fast Generate function only deals with constant layer thickness and single material property. As shown in Figure 2.15, each layer with five nodes has been shown by five different color lines.

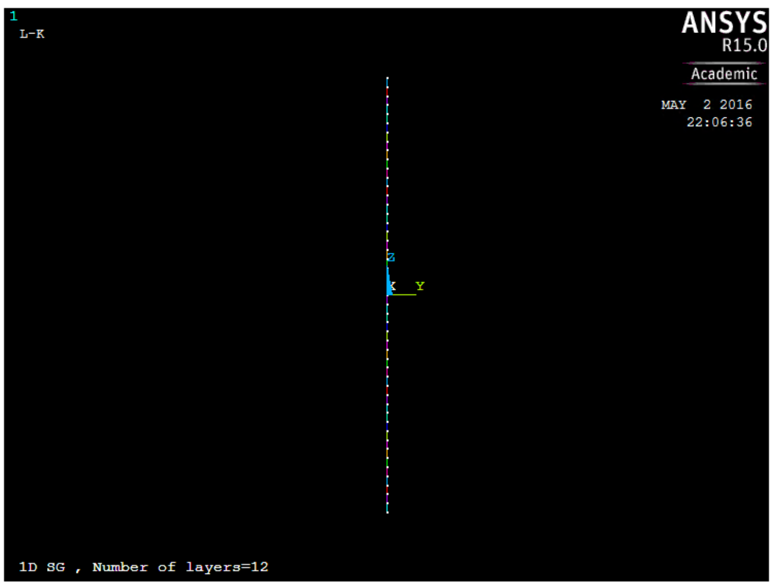


Figure 2.15

Preprocessor → Common SG → 1D SGs->Advanced Generate.

If users want to create laminates with different layer thickness and material property, advanced module in 1D SGs can serve this purpose. Click Advanced Generate, a new window pop out as shown in Figure 2.16. Recall that we have defined two different materials properties. Input parameters as shown in Figure 2.16. The 1D SG has been created as shown in Figure 2.16.

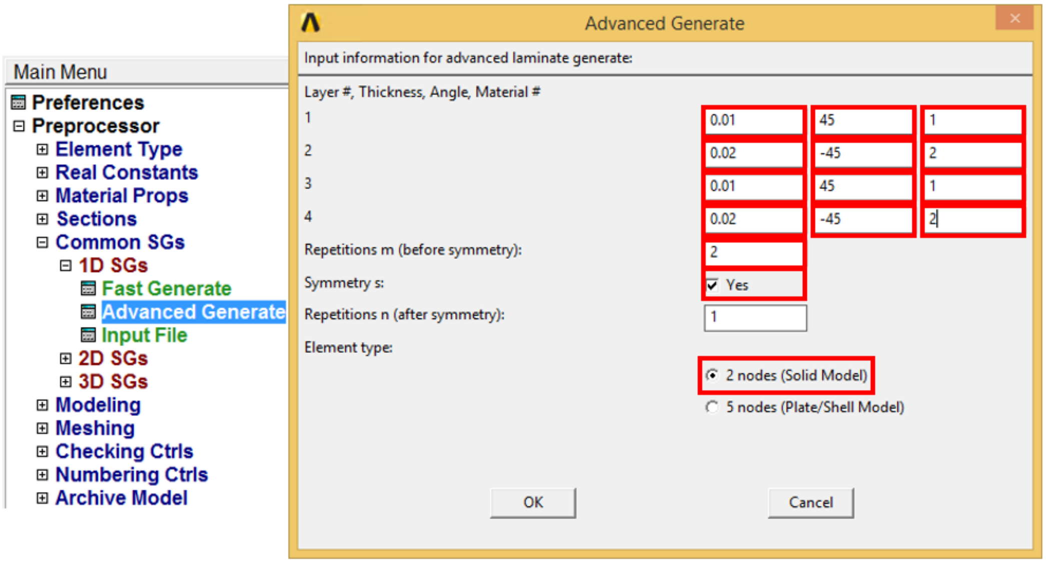


Figure 2.16

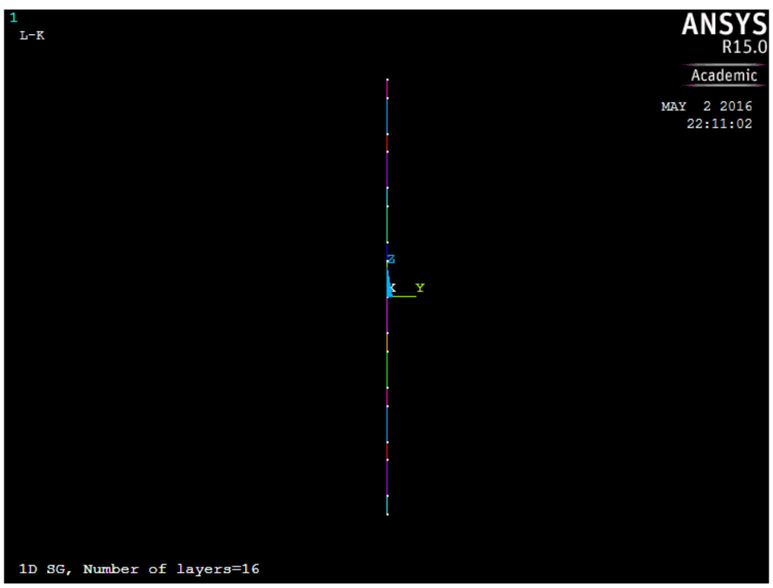


Figure 2.17

Preprocessor → Common SG → 1D SGs → Input File.

In addition, 1D SGs also provides Input File module to let users upload their own data file. This module can handle laminates with complex ply sequence. Click ‘Input File’ in the 1D SGs menu, a new window will pop out. Users can choose the file which contains laminate data (Figure 2.18). The format of the data file is discussed in the first chapter. The release package contains a sample laminate file *SampleLaminateInput.txt*.

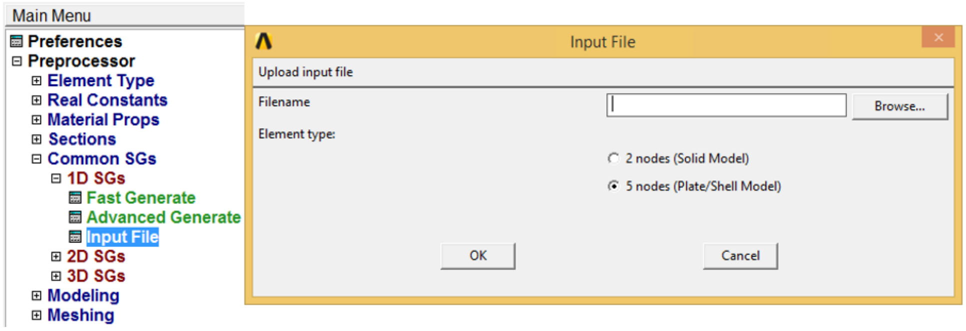


Figure 2.18

Homogenization

Preprocessor → Solution → Homogenization → Plate/Shell Model.

For laminates, the mesh has been generated right after defining laminate. Users can directly go to Homogenization function to get the properties of laminate. We will use the model generated by fast generate to show the results of Homogenization and Dehomogenization. Click Plate/Shell Model in Homogenization function, keep default parameters and click OK (see Figure 2.19). Then the effective properties will pop up automatically (see Figure 2.20)

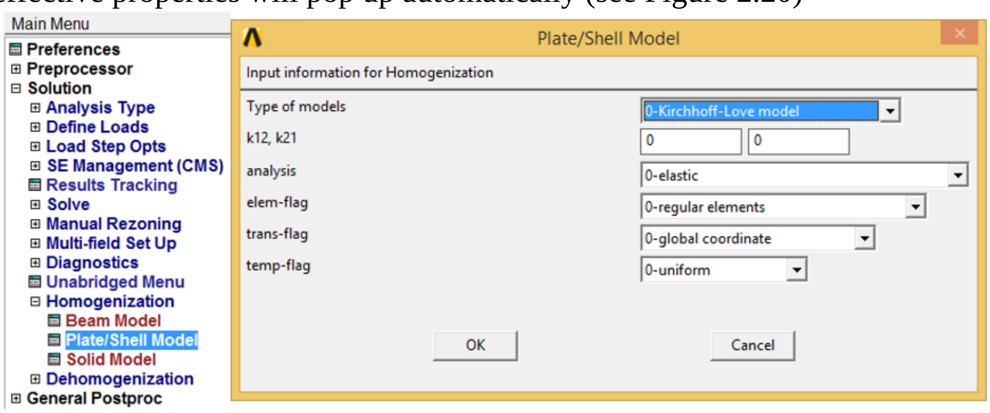


Figure 2.19

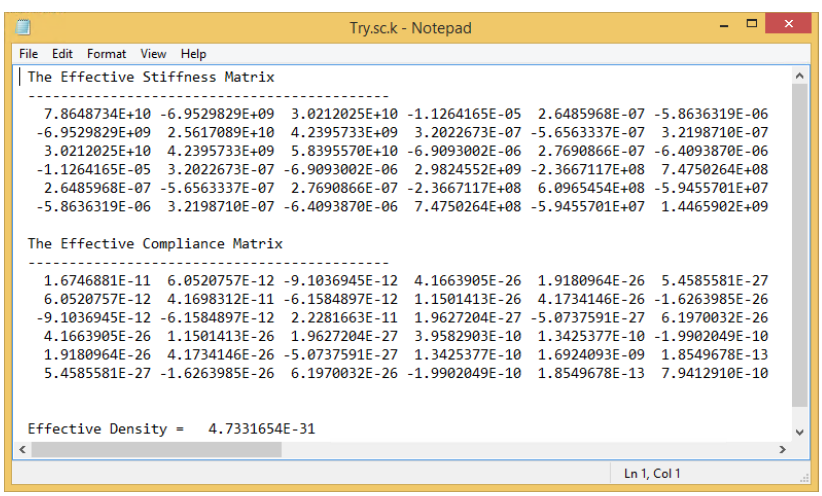


Figure 2.20

Dehomogenization

Preprocessor → Solution → Dehomogenization → Plate/Shell Model.

Input the global behavior from the plate analysis (see Figure 2.21). Click Run. The post-processing results will be automatically loaded. The default value is the magnitude of displacement as in Figure 2.22.

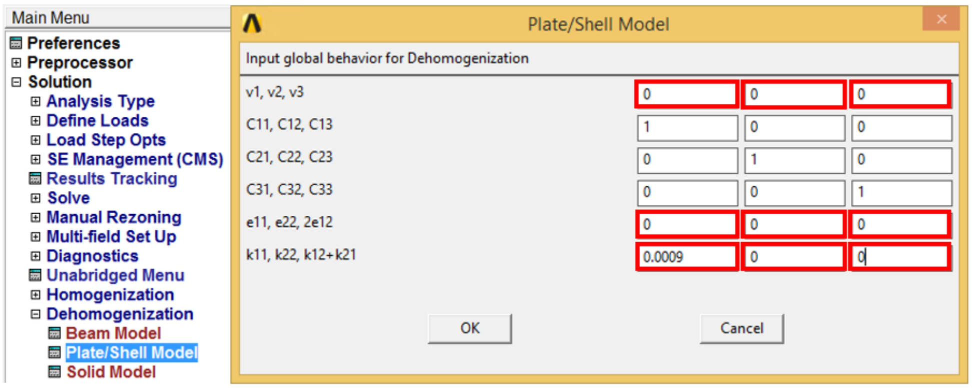


Figure 2.21

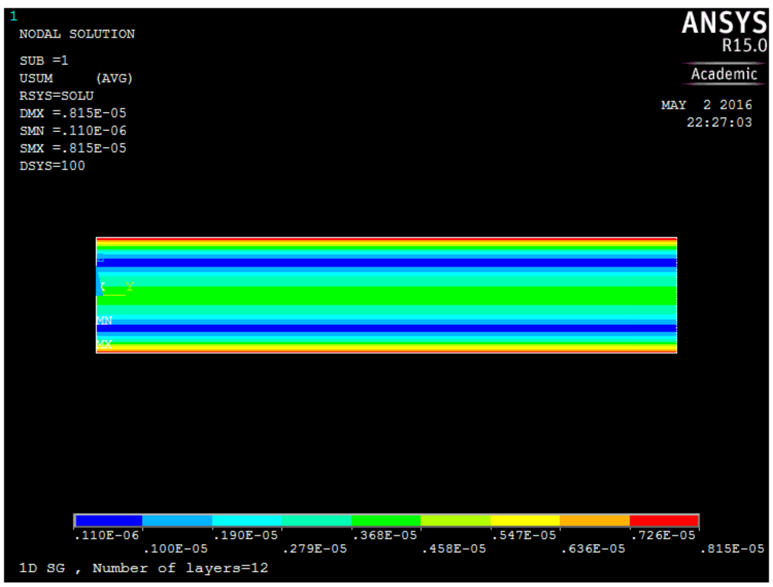


Figure 2.22

2.4 Summary

Common SGs function provides users a convenient way to quickly build the geometry of common SG, and easily specify materials and structures properties they want to test. Users can also test other common models if they are interested.

3.0 CREATE USER-DEFINED MODEL

3.0 CREATE USER-DEFINED MODEL

Although common SG function provides users a convenient way to build the geometry and mesh of the most common models, often users need to build their own models according to the microstructure they are analyzing. Without using the common SG function, this chapter will illustrate how to build the above two common models along with a rectangular SG with two arbitrary inclusions using the simple CAD capabilities provided by ANSYS.

The differences between user-defined model and common SG model are the generation geometry, mesh of model, and assign material properties. The procedures of creating material properties and running SwiftComp™ are the same. So we will only focus on the **geometry and mesh creation** in this chapter. Since we are using the native CAD capabilities in ANSYS, users are encouraged to refer to the ANSYS manual for more detailed instructions on geometry and mesh creation.

3.1 Square Pack Microstructure (2D SG)

Define Materials

This step is the same as defining materials for 2D common SGs in the Chapter Two.

Generate User-defined Model Geometry

Preprocessor → Modeling → Create → Areas → Rectangle → By Dimensions.
Input the two corner coordinates of the rectangle, i.e. x1=0.5, y1=0.5, x2=-0.5, y2=-0.5 (see Figure 3.1). These coordinates are used for defining the SG boundary.

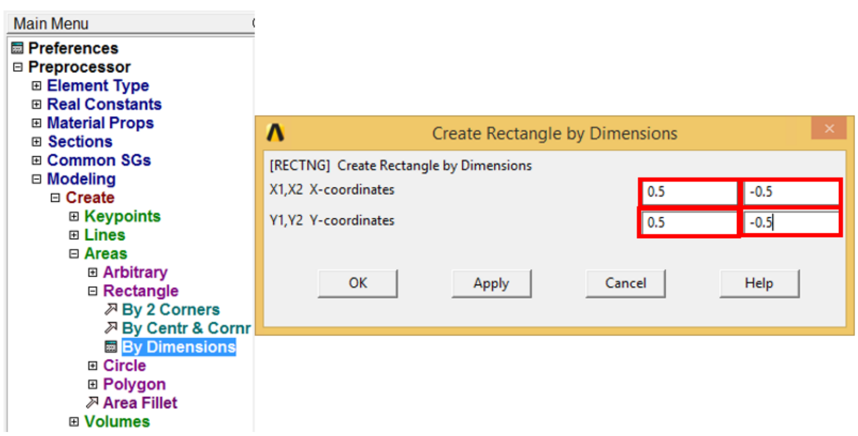


Figure 3.1

Preprocessor → Modeling → Create → Areas → Circle → By Dimensions.
Input the outer radius of the circle, i.e. RAD1=0.3569. This parameter is used for defining the fiber boundary, and needs to be calculated first according to the volume fraction of fiber (see Figure 3.2).

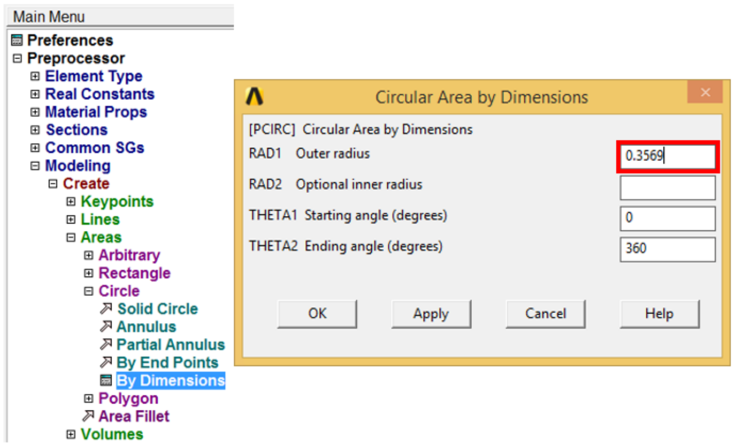


Figure 3.2

Preprocessor → Modeling → Operate → Booleans → Overlap → Areas.
Click 'Pick All' in the Overlap Areas dialog box (see Figure 3.3). This operation will create two separated areas for mesh. And this is the final step for creating geometry of the model. The geometry of the model should look like Figure 3.3.

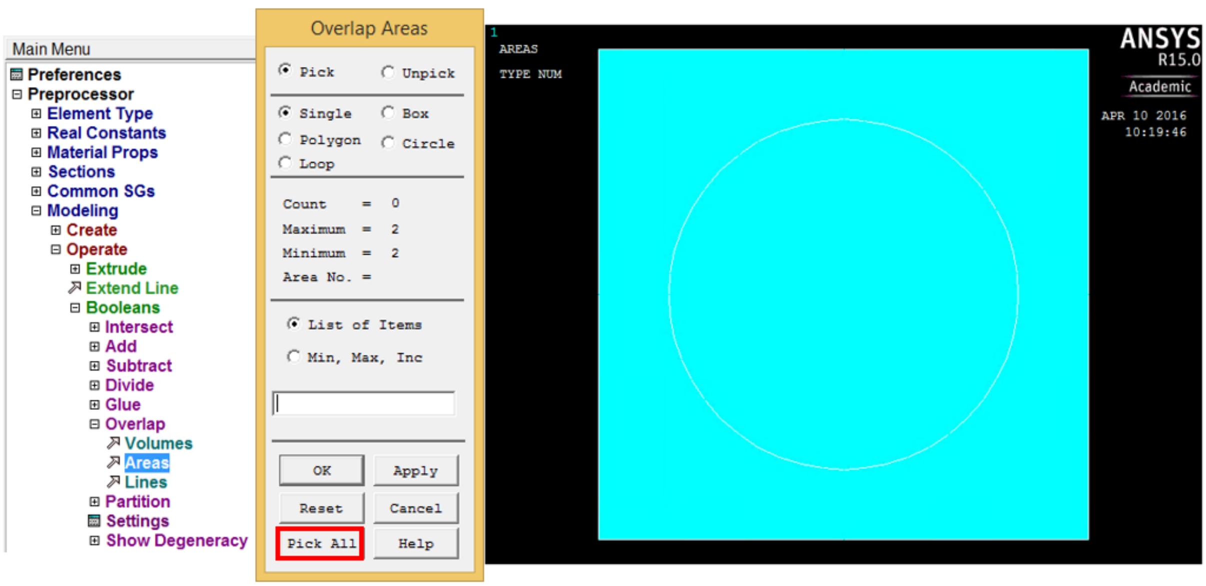


Figure 3.3

Generate User-defined Model Mesh

Preprocessor → Element Type → Add/Edit/Delete → Add…→ Solid → 8 node 183.
This step is to choose element type for mesh. We choose quadrilateral 8 nodes 183 element (see Figure 3.4).

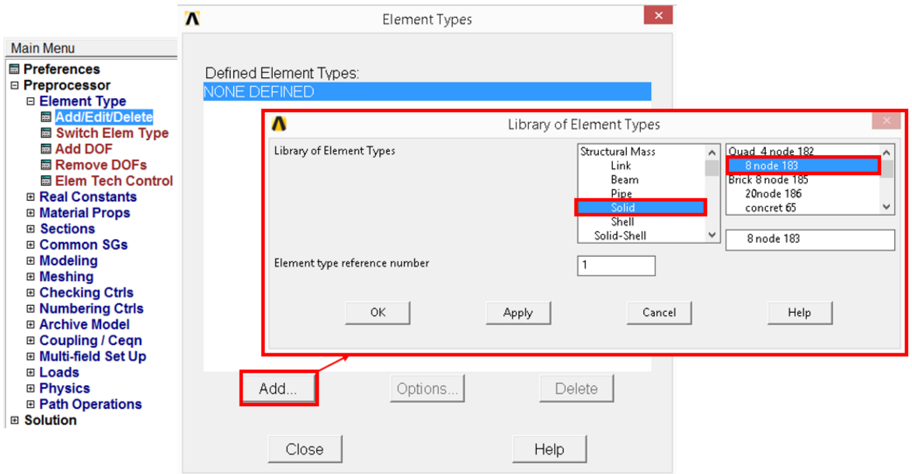


Figure 3.4

Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size.
Input element edge length size, i.e. 1/20. This element size controls how many elements at the boundary of our model, which creates a periodic node distribution at the boundary of the model which are needed for periodic materials.

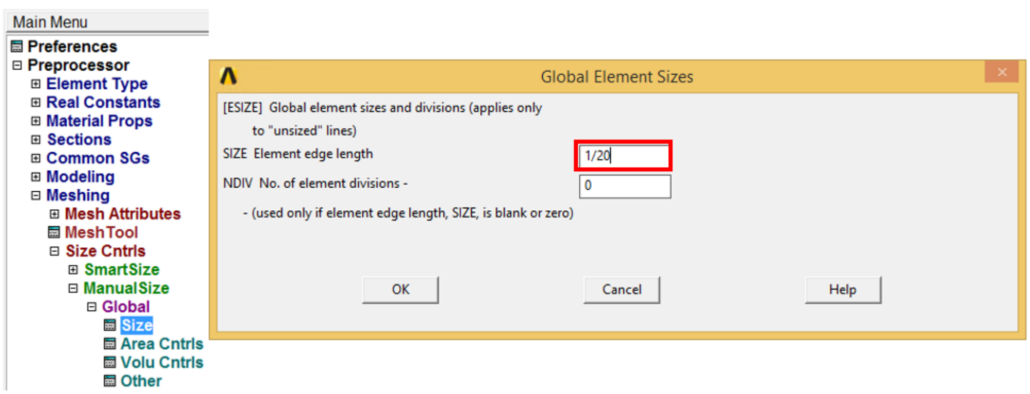


Figure 3.5

Preprocessor → Meshing → MeshTool → Mesh (choose areas).
Mesh fiber (Area at the center of the model) with Material Model Number 1 (default) (see Figure 3.6).

Preprocessor → Meshing → MeshTool → Set (change material number) → Mesh (choose areas).
Mesh matrix with Material Model Number 2. Choose the area that surrounds fiber (see Figure 3.7). This is the final step for meshing the model. After this step, the model should look like Figure 3.8.

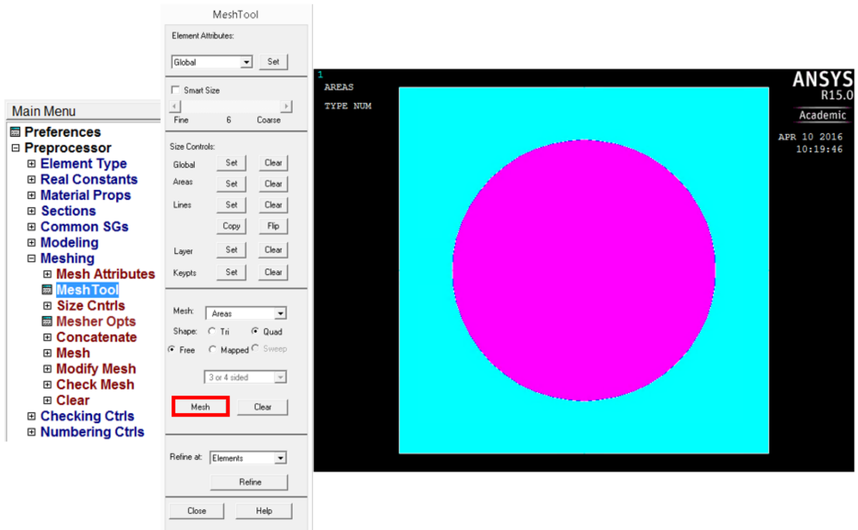


Figure 3.6

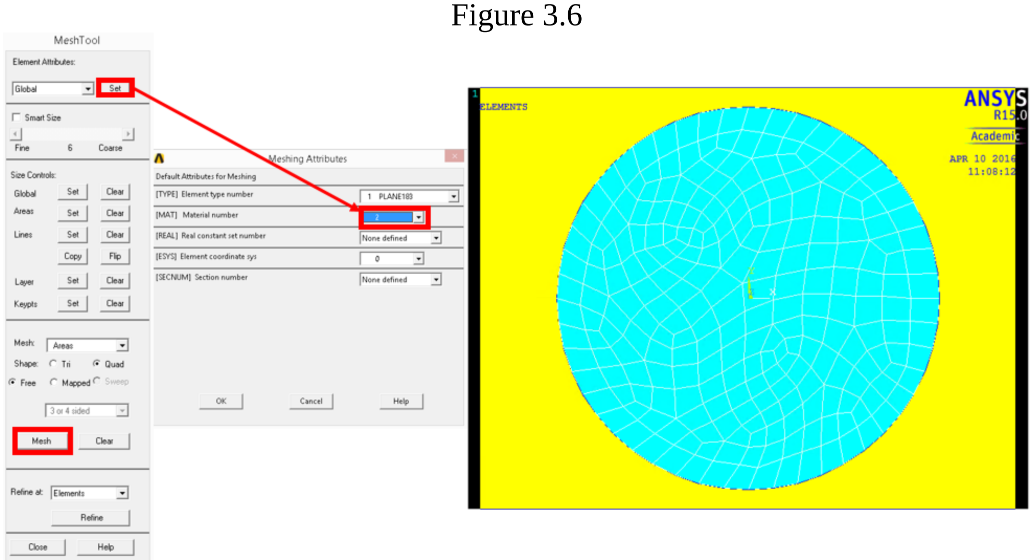


Figure 3.7

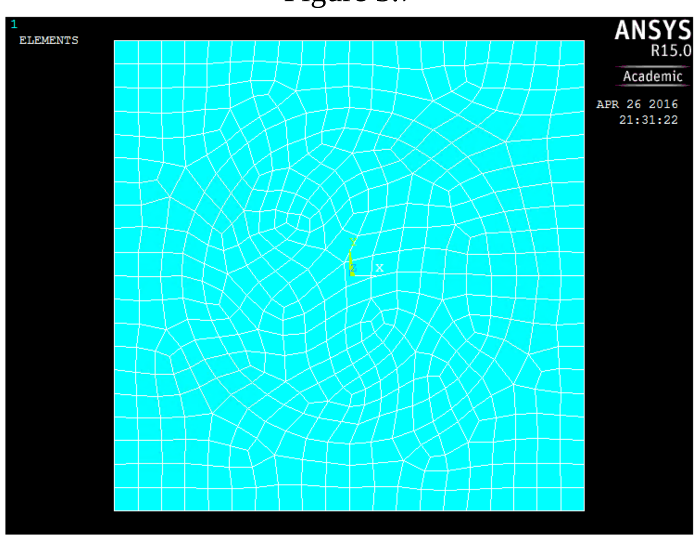


Figure 3.8

The rest procedures are the same as the previous chapter for generating common SG model:
Preprocessor → Solution → Homogenization → Solid Model.
Preprocessor → Solution → Dehomogenization → Solid Model.

3.2 Spherical Inclusions Microstructure (3D SG)

Define Materials

This step is the same as defining materials for 2D common SG model in Chapter Two.

Generate User-defined Model Geometry

First, choose Isometric view.

Preprocessor → Modeling → Create → Volumes → Block → By Dimensions.
Input the two corner coordinates of the block, i.e. x1=0.5, y1=0.5, z1=0.5, x2=-0.5, y2=-0.5, z2=-0.5 (see Figure 3.9). These coordinates are used for defining the SG boundary.

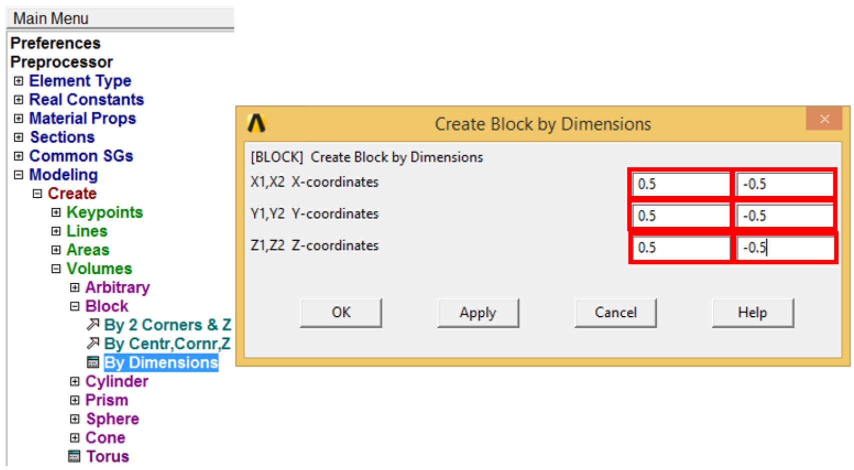


Figure 3.9

Preprocessor → Modeling → Create → Volumes → Sphere → By Dimensions.
Input the outer radius of the sphere, i.e. RAD1=0.457. This parameter is used for defining the sphere boundary, and needs to be calculated first according to the volume fraction of sphere (see Figure 3.10).

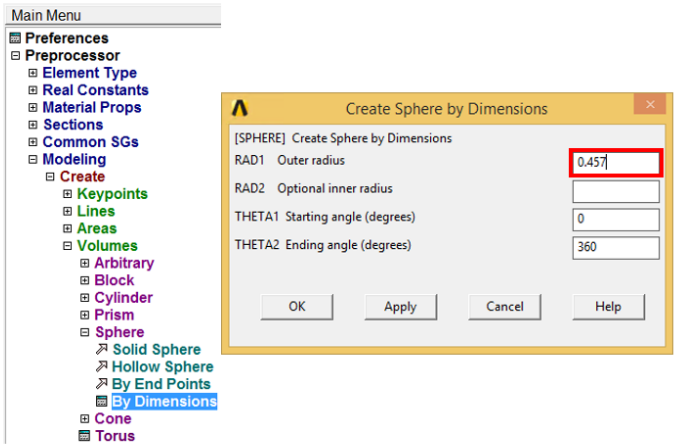


Figure 3.10

Preprocessor → Modeling → Operate → Booleans → Overlap → Volumes.
Click 'Pick All' in the Overlap Volumes dialog box (see Figure 3.11). This operation will create two separated volumes for mesh. And this is the final step for creating geometry of the model. The geometry of the model should look like Figure 3.11.

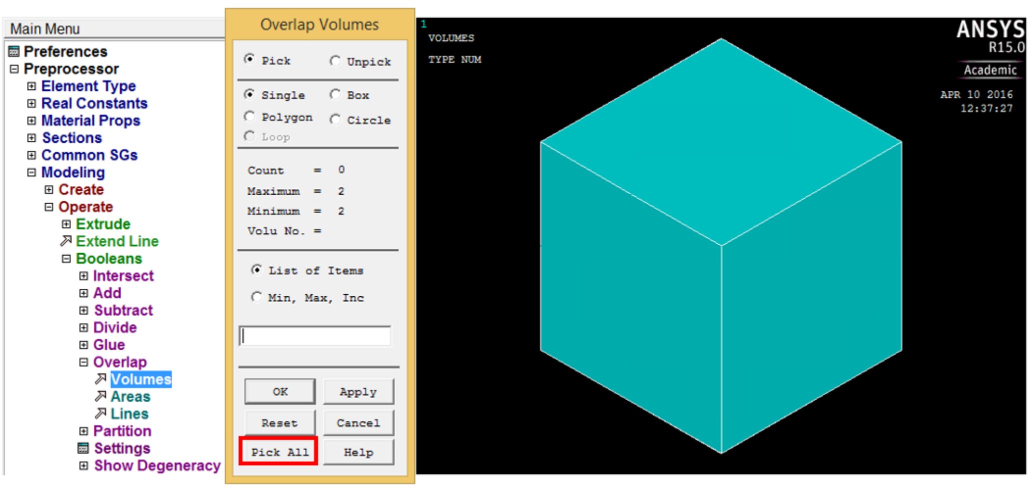


Figure 3.11

Generate User-defined Model Mesh

Preprocessor → Element Type → Add/Edit/Delete → Add…→ Solid → 10 node 187.
Preprocessor → Element Type → Add/Edit/Delete → Add…→ Shell → 8 node 281.
This step is to choose element type for mesh. We choose tetrahedron 10 nodes 187 element for the model (see Figure 3.12). We also add a shell 8 node 281 element to help create a mapped mesh on the boundary of the model for periodic nodes needed for periodic materials.

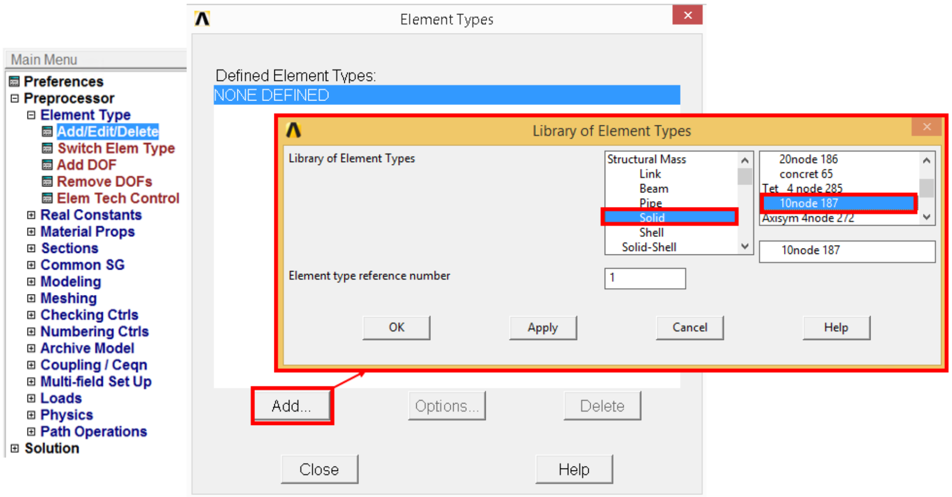


Figure 3.12

Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size.
Input element edge length size, i.e. 1/10. This element size controls how many elements at the boundary of our model, which creates a periodic node distribution at the boundary needed for periodic materials (see Figure 3.13).

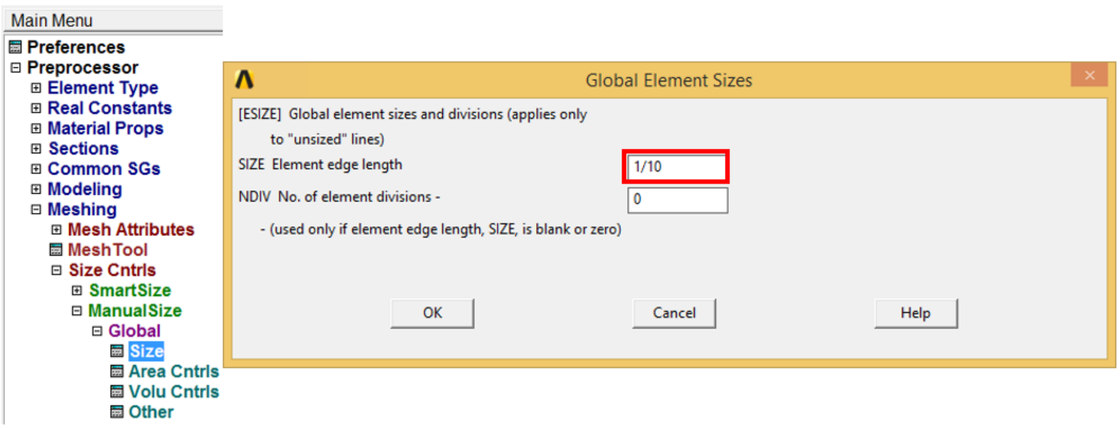


Figure 3.13

Preprocessor → Meshing → MeshTool → Set (change element type number) → Mesh (choose areas).
First, use shell element to create a mapped mesh at the boundary of the model. Click Mesh. Choose the six surfaces of the model (see Figure 3.14). Click Ok. Then, the model should look like Figure 3.15.

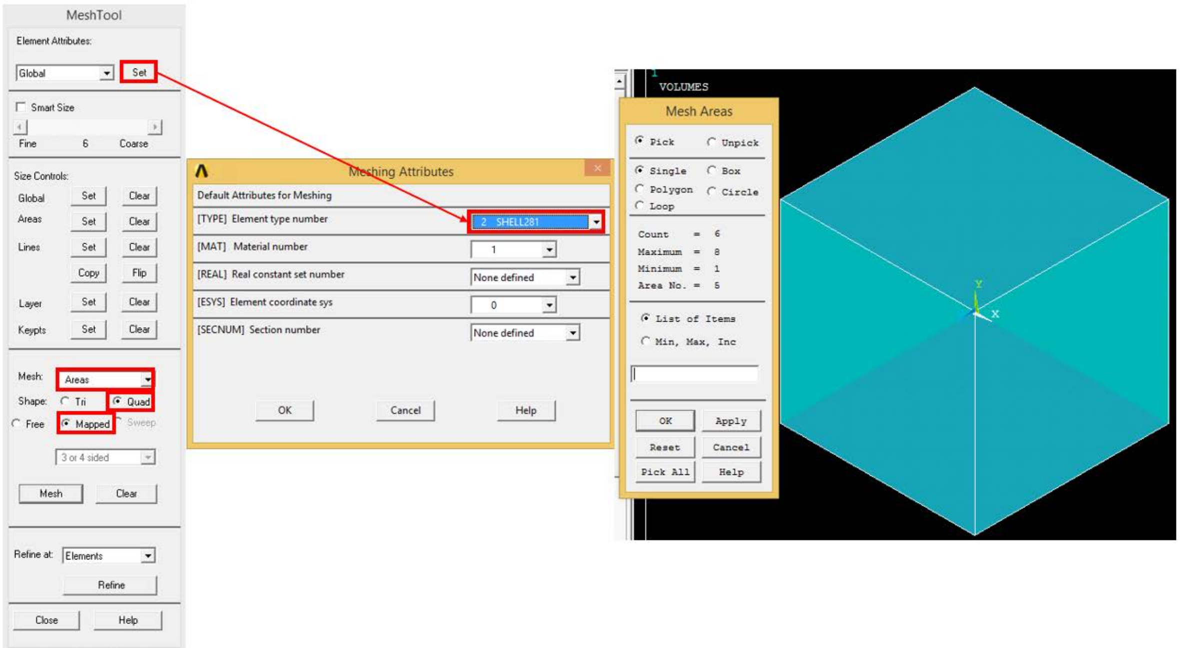


Figure 3.14

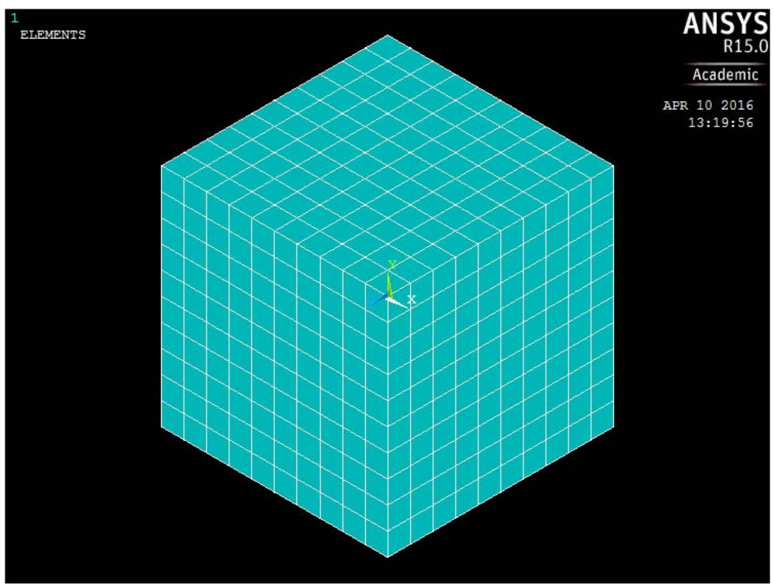


Figure 3.15

Preprocessor → Meshing → MeshTool → Set (change element type number) → Mesh.
Mesh sphere (volume at the center of the model) with Material Model Number 1 (default) (see Figure 3.16).

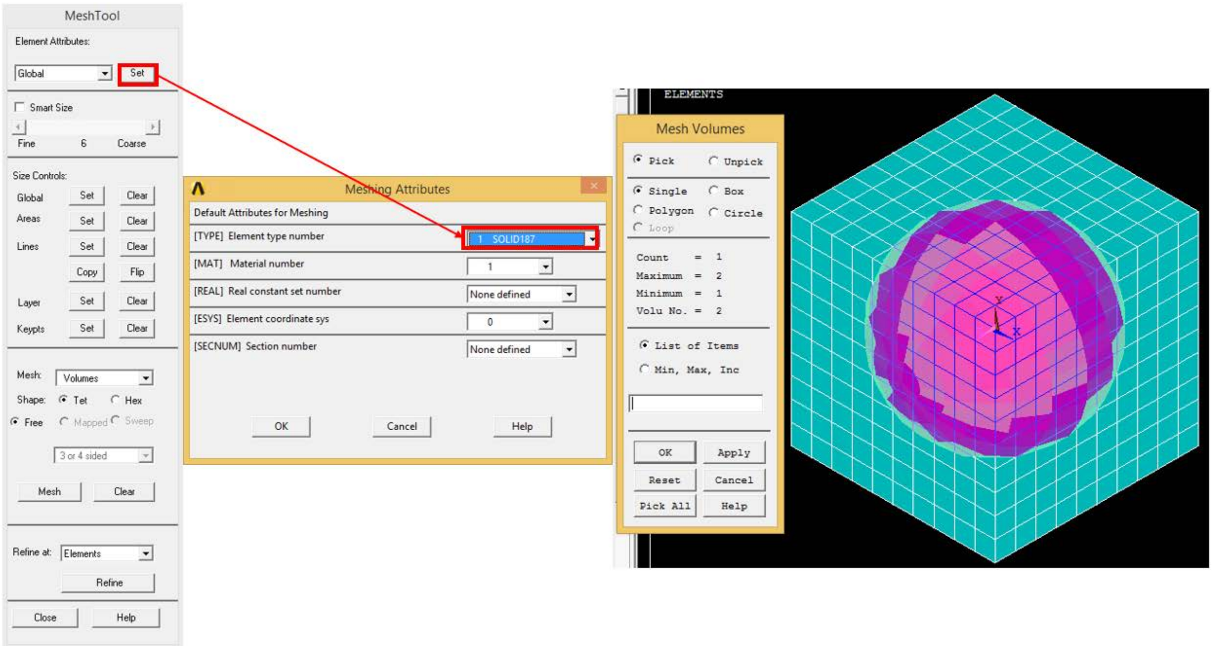


Figure 3.16

Preprocessor → Meshing → MeshTool → Set (change material number) → Mesh.
Mesh matrix with Material Model Number 2. Choose the volume that surrounds sphere (see Figure 3.17).

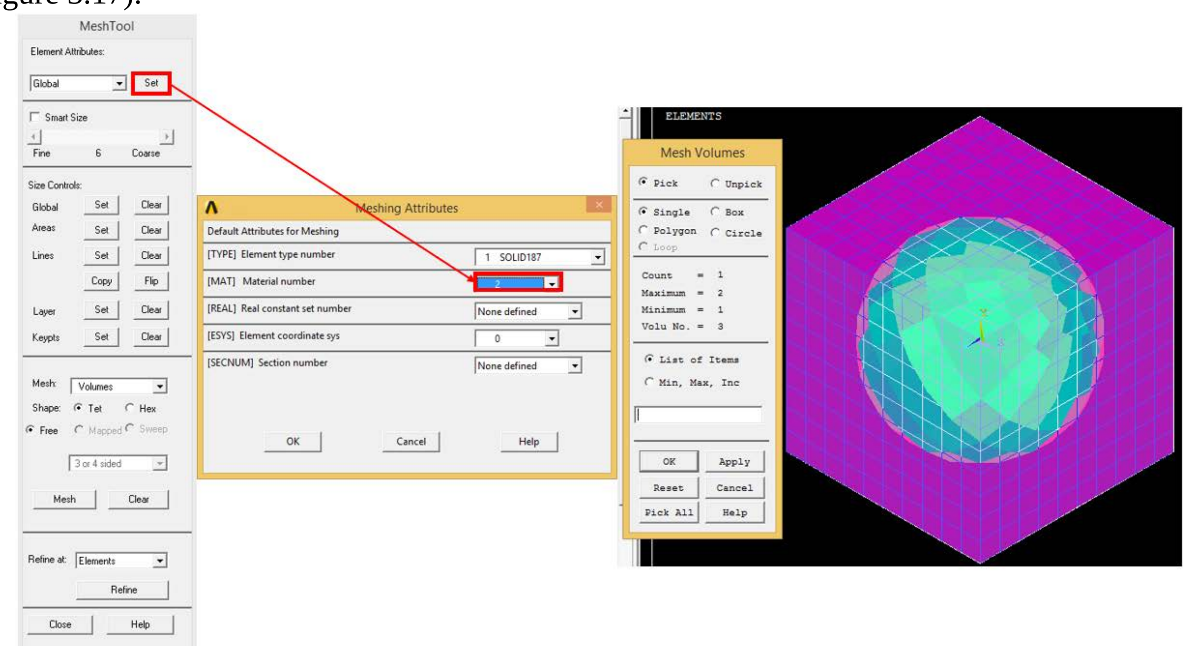


Figure 3.17

Preprocessor → Meshing → Clear → Areas.
Delete the shell element. Click Pick All. Then, choose Isometric view. This is the final step for meshing the model. After this step, the model should look like Figure 3.18.

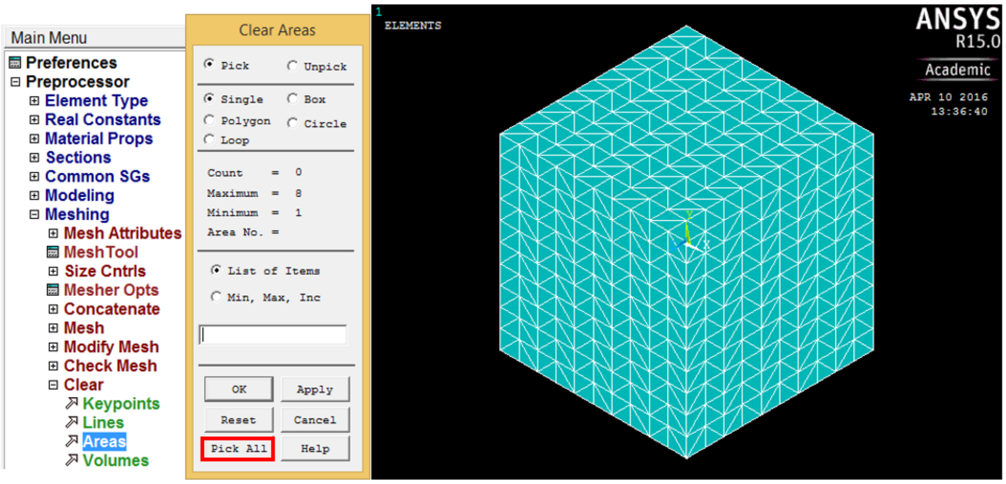


Figure 3.18

The rest procedures are the same as the previous chapter for generating common SG model:

Preprocessor → Solution → Homogenization → Solid Model.

Preprocessor → Solution → Dehomogenization → Solid Model.

3.3 Arbitrary Shape Inclusions Microstructure (2D SG)

One more user-defined model is shown here except for the two common SGs which can also gbe generated by Common SGs function. Here we use a rectangle SG with two arbitrary inclusions. Users can understand how to create complex shape in ANSYS-SwiftComp GUI, and also know the capability of SwiftComp™ to calculate such models. Since this is not a common model in ANSYS-SwiftComp GUI, all the steps needed for homogenization and dehomogenization will be provided below.

Define material

Preprocessor → Material Props → Material Models → Structural → Linear → Elastic.

Choose materials type (e.g. Isotropic, Orthotropic or Anisotropic).

In this example, assume that inclusions and matrix are both isotropic materials (Material 1: E = 379.3 GPa, υ = 0.1; Material 2: E = 279.3 GPa, υ = 0.1; Material 3: E = 68.3 GPa, υ = 0.3).

Generate User-defined Model Geometry

Preprocessor → Modeling → Create → Keypoints → On Active CS (input value).

Create Keypoints. Input the coordinates of four points (-0.5, -1, 0), (-0.5, 1, 0), (0.5, 1, 0) and (0.5, -1, 0). These coordinates are used for defining the 2D SG boundary.

Then the points inside the 2D SG are given here: (-0.2, 0.8, 0), (-0.3, 0.7, 0,), (-0.3, 0.4, 0), (-0.1, 0.6, 0), (0, 0.8, 0), (0, 0, 0), (-0.1, 0, 0), (-0.2, -0.3, 0), (0.1, -0.5, 0), (0.1, -0.3, 0) and (0.2, -0.1, 0) (see Figure 3.19).

Note: User does not have to input Keypoints number. ANSYS will automatically increase Keypoints number as user click OK or Apply.

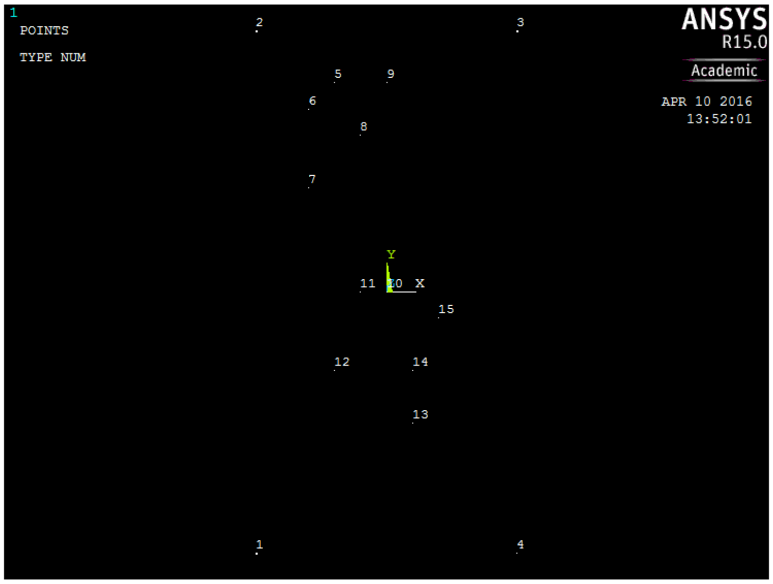


Figure 3.19

Main Menu → Preprocessor → Modeling → Create → Lines → Lines → Straight Line → (Choose KP) OK.

Create Line. Line 1 {1 4} Line 2 {4 3}, Line 3 {3 2}, Line 4 {2 1} (see Figure 3.20).

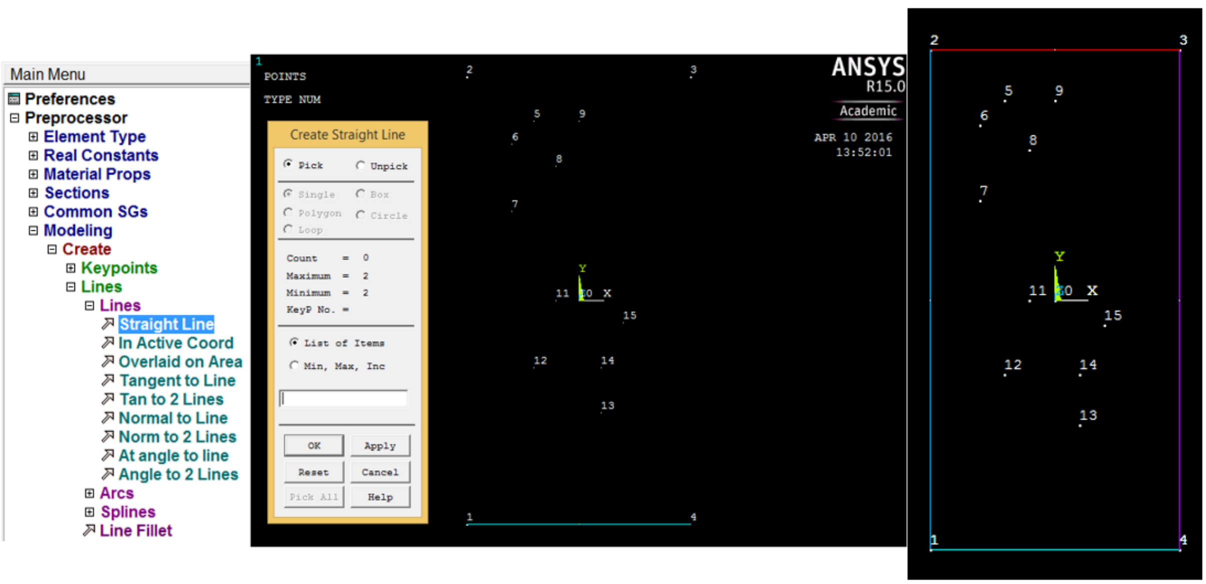


Figure 3.20

Preprocessor → Modeling → Create → Lines → Splines → Segmented Spline (Choose KP).

Create spline line. For the first one, pick KP 5, 6, 7, 8, 9 and 5. For the second one, pick 10, 11, 12, 13, 14, 15 and 10 (see Figure 3.21).

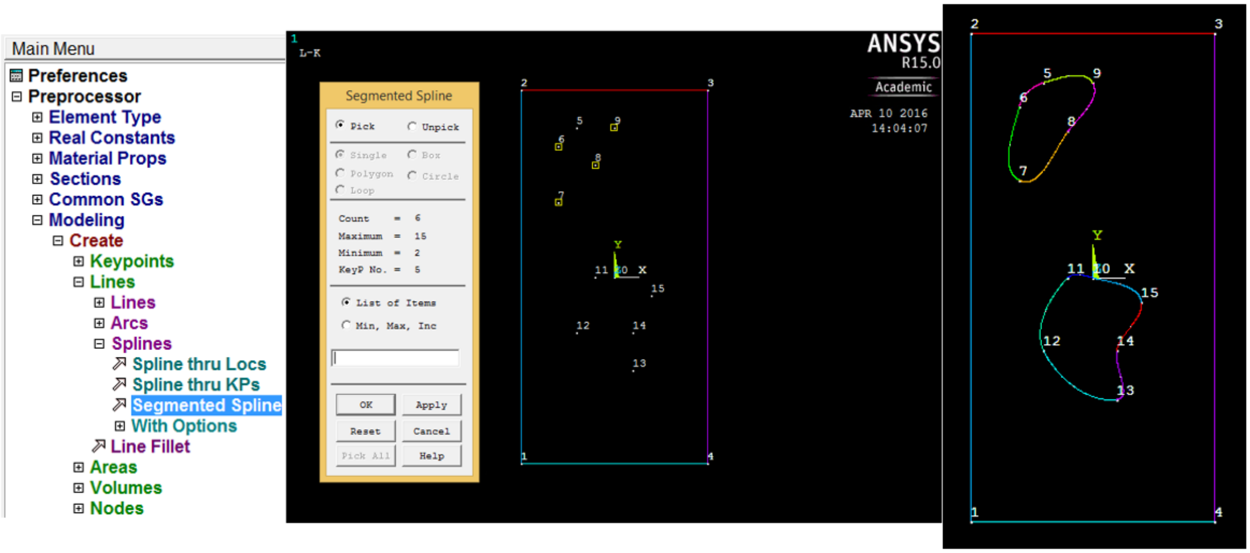


Figure 3.21

Preprocessor → Modeling → Create → Areas → Arbitrary → By Lines (Choose lines).
Create area. First choose the spline line for the upper one. Then choose the spline line for the lower one. Finally choose all line (see Figure 3.22).

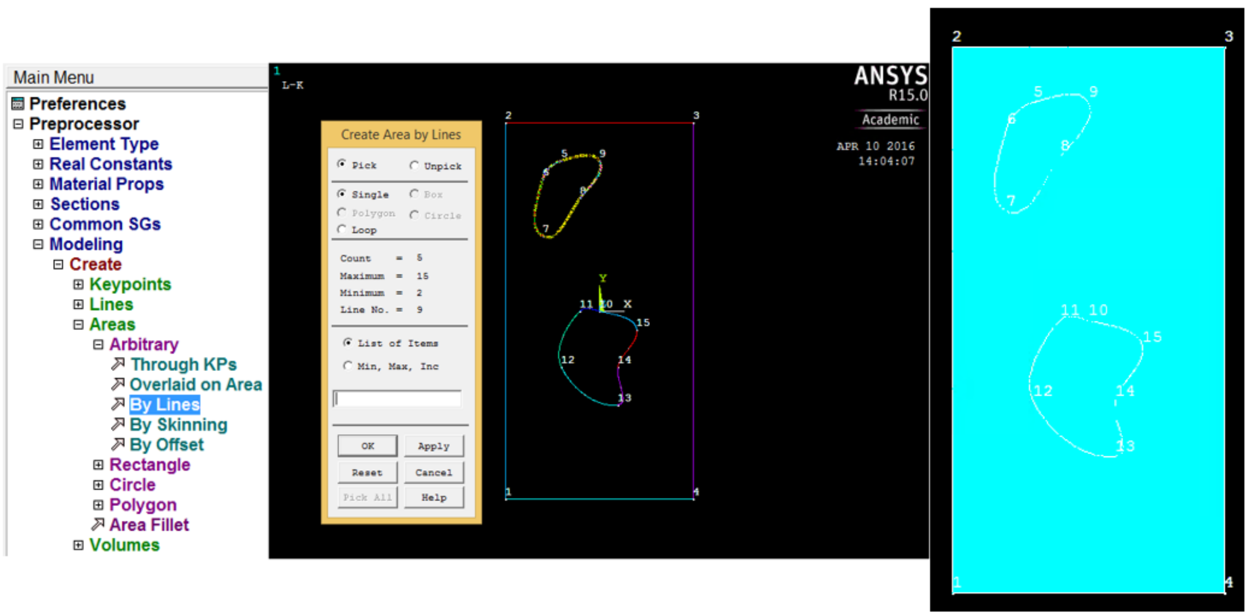


Figure 3.22

Preprocessor → Modeling → Operate → Booleans→ Glue → Areas.
Finally, glue all of the areas. This is the last step for creating the geometry of the model. The model should look like Figure 3.23.

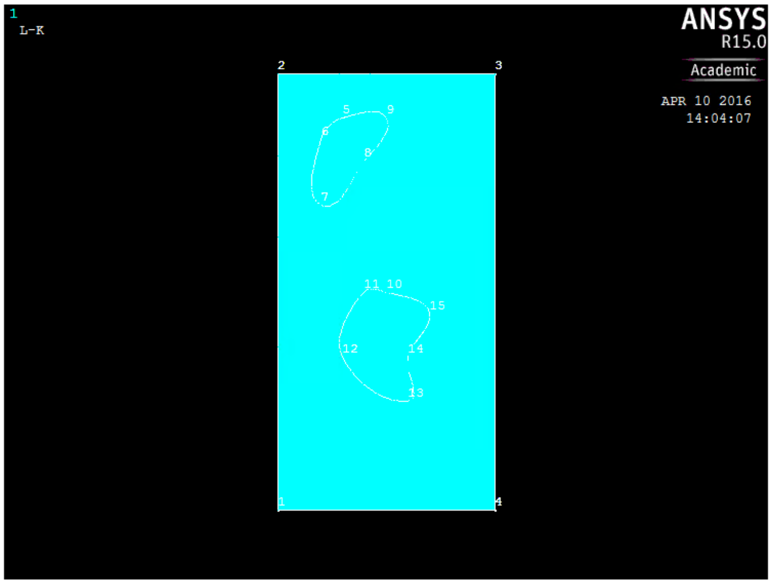


Figure 3.23

Generate User-defined Model Mesh

Preprocessor → Element Type → Add/Edit/Delete → Add…→ Solid → 8 node 183.
This step is to choose element type for mesh. We choose quadrilateral 8 nodes 183 element (see Figure 3.24).

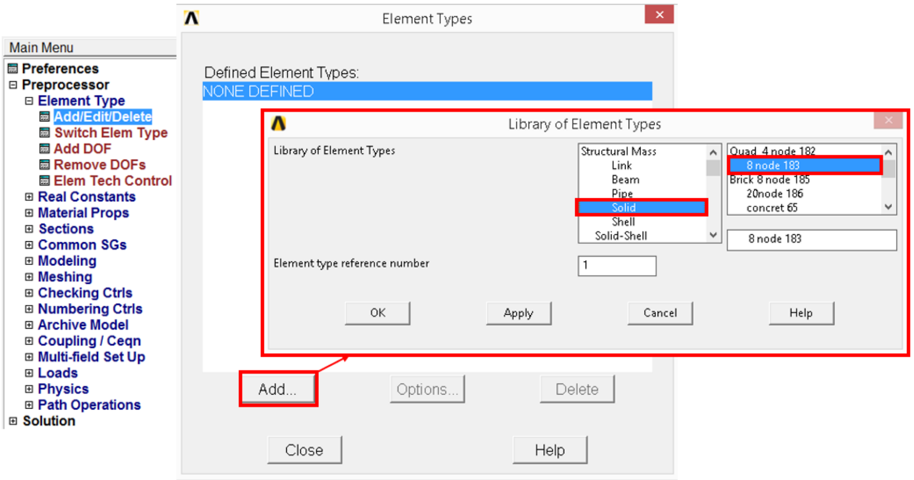


Figure 3.24

Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size.
Input element edge length size, i.e. 1/20. This element size controls how many elements at the boundary of our model, which creates a periodic node distribution at the boundary needed for periodic materials (see Figure 3.25).

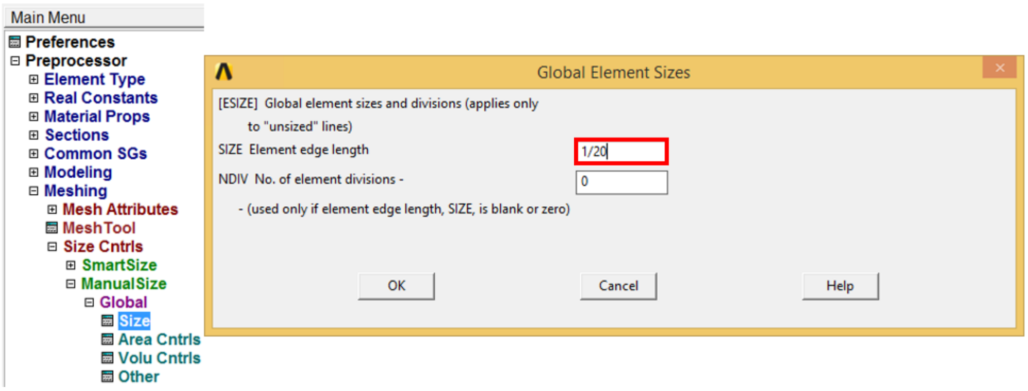


Figure 3.25

Preprocessor → Meshing → MeshTool → Mesh (choose areas).
Mesh upper inclusion (Area at the top left corner of the model) with Material Model Number 1 (default) (see Figure 3.26).

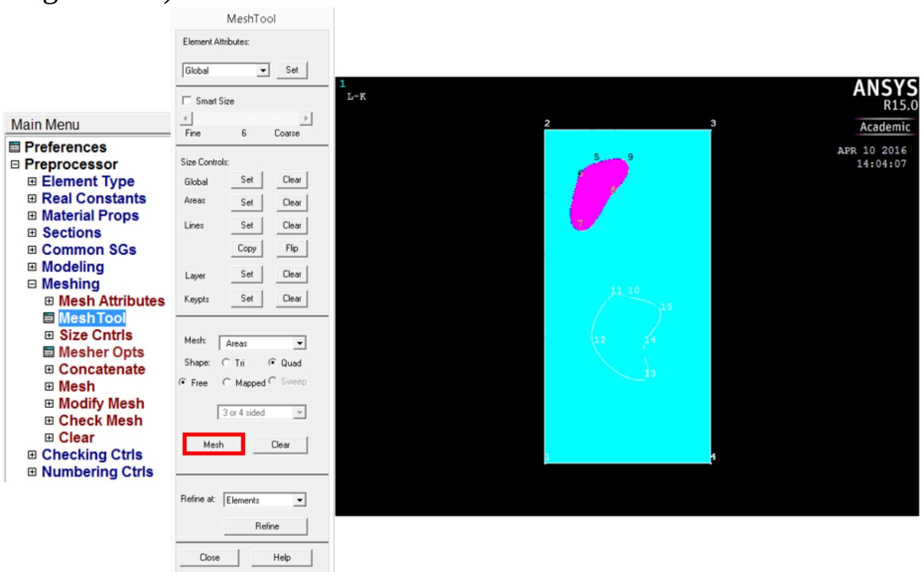


Figure 3.26

Note: To see the whole model, choose Multi-Plots in the Plot menu bar.

Preprocessor → Meshing → MeshTool → Set (change material number) → Mesh (choose areas).
Mesh bottom inclusion with Material Model Number 2 (Area at the bottom right corner of the model) (see Figure 3.27).

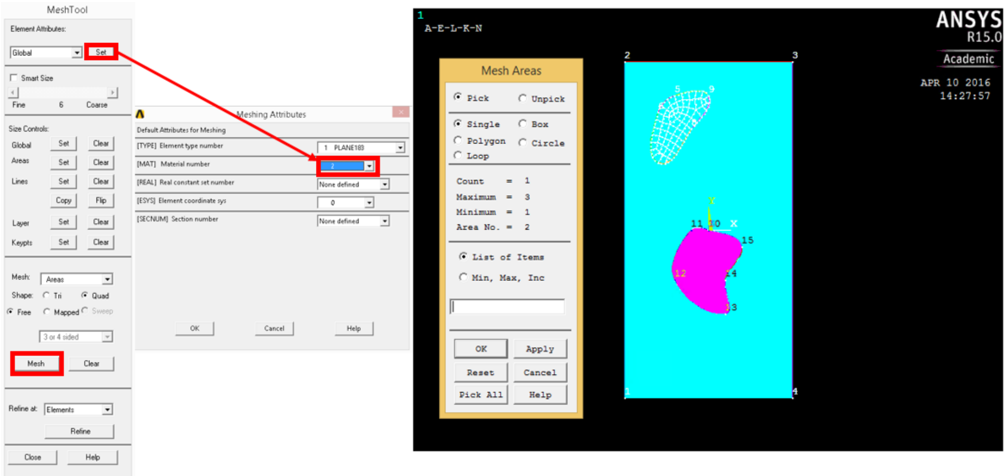


Figure 3.27

Preprocessor → Meshing → MeshTool → Set (change material number) → Mesh (choose areas).

Mesh matrix with Material Model Number 3 (area that surrounds the two inclusions) (see Figure 3.28). This is the final step for meshing the model. After this step, the model should look like Figure 3.29.

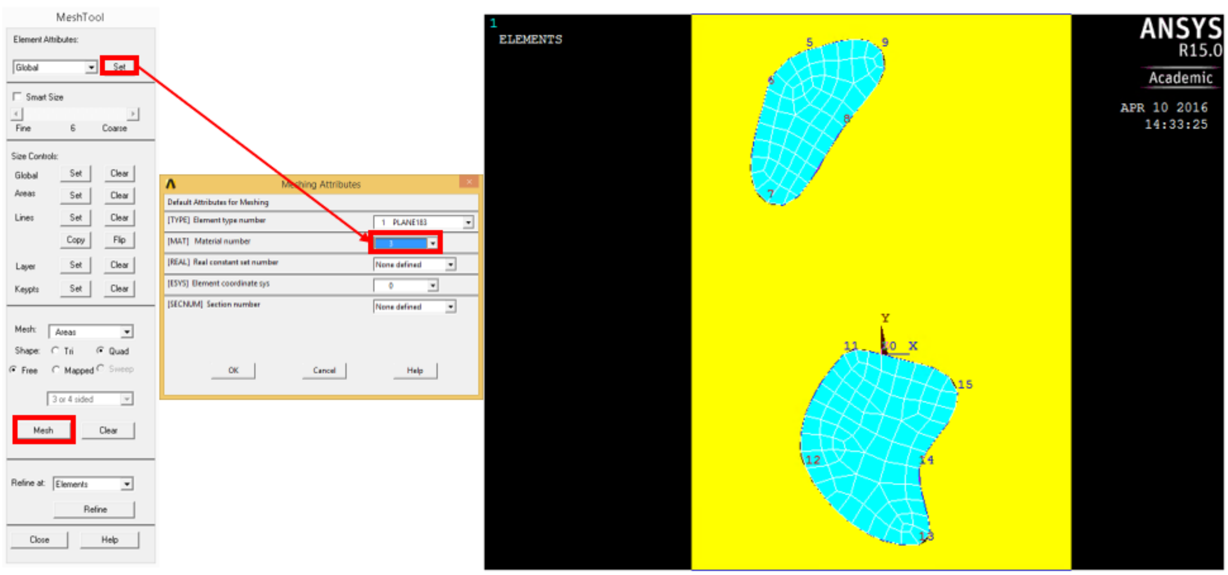


Figure 3.28

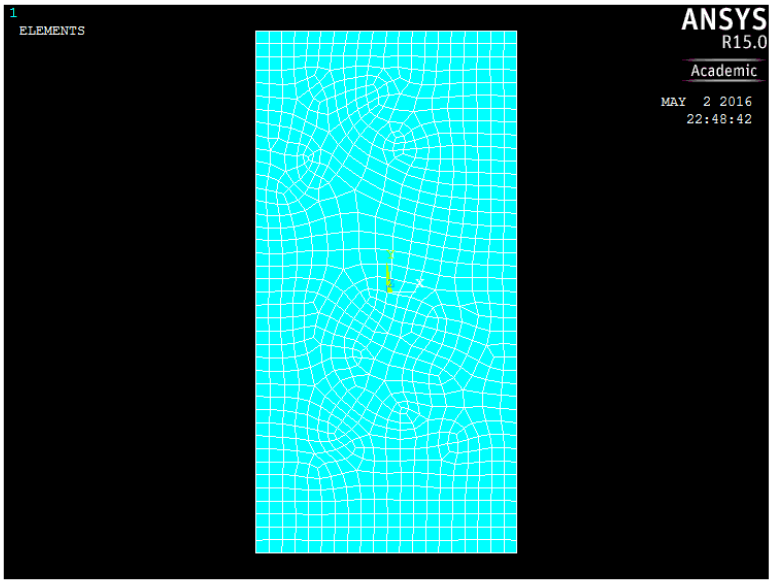


Figure 3.29

Homogenization

Preprocessor → Solution → Homogenization → Solid Model.

Please read the manual of SwiftComp™ to see the meaning of each parameters.

In this example, we consider our model is Solid Model, keep all default settings, click Ok (see Figure 3.30).

Wait for preparing the input file of SwiftComp and SwiftComp™ to finish the computation. Note that if the SG contains a large number of elements, it will take some time to prepare the input file and finish the computation. Then the effective properties will pop up automatically (see Figure 3.31).

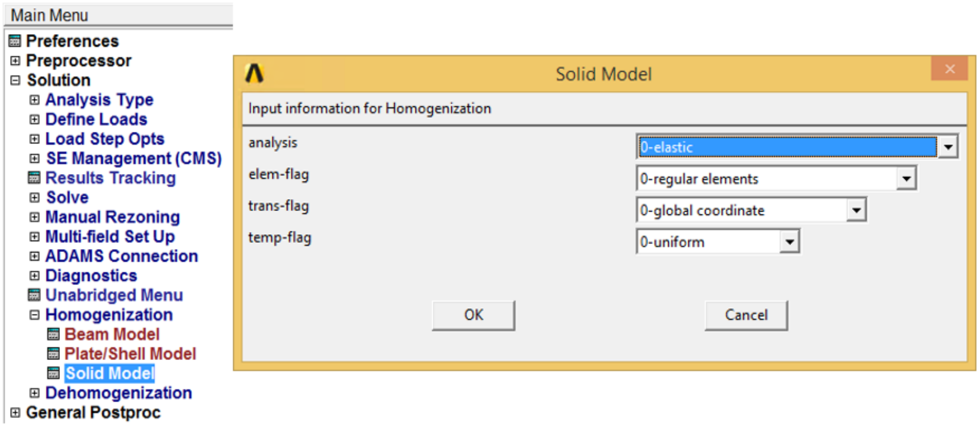
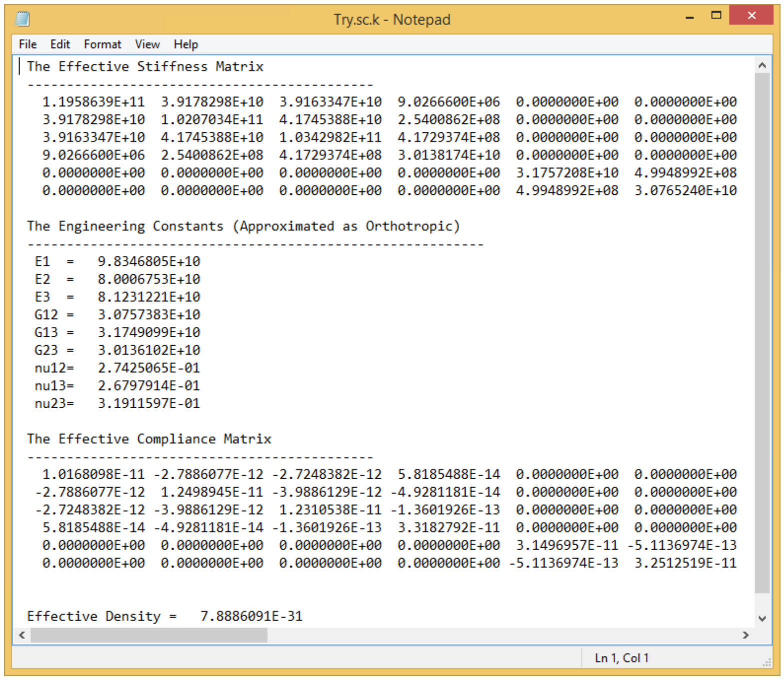


Figure 3.30




```
The Effective Stiffness Matrix
-----------------------------------------
  1.1958639E+11  3.9178298E+10  3.9163347E+10  9.0266600E+06  0.0000000E+00  0.0000000E+00
  3.9178298E+10  1.0207034E+11  4.1745388E+10  2.5400862E+08  0.0000000E+00  0.0000000E+00
  3.9163347E+10  4.1745388E+10  1.0342982E+11  4.1729374E+08  0.0000000E+00  0.0000000E+00
  9.0266600E+06  2.5400862E+08  4.1729374E+08  3.0138174E+10  0.0000000E+00  0.0000000E+00
  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  3.1757208E+10  4.9948992E+08
  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  4.9948992E+08  3.0765240E+10

The Engineering Constants (Approximated as Orthotropic)
----------------------------------------------------------
 E1  =    9.8346805E+10
 E2  =    8.0006753E+10
 E3  =    8.1231221E+10
 G12 =    3.0757383E+10
 G13 =    3.1749099E+10
 G23 =    3.0136102E+10
 nu12=    2.7425065E-01
 nu13=    2.6797914E-01
 nu23=    3.1911597E-01

The Effective Compliance Matrix
-----------------------------------------
  1.0168098E-11 -2.7886077E-12 -2.7248382E-12  5.8185488E-14  0.0000000E+00  0.0000000E+00
 -2.7886077E-12  1.2498945E-11 -3.9886129E-12 -4.9281181E-14  0.0000000E+00  0.0000000E+00
 -2.7248382E-12 -3.9886129E-12  1.2310538E-11 -1.3601926E-13  0.0000000E+00  0.0000000E+00
  5.8185488E-14 -4.9281181E-14 -1.3601926E-13  3.3182792E-11  0.0000000E+00  0.0000000E+00
  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  3.1496957E-11 -5.1136974E-13
  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00 -5.1136974E-13  3.2512519E-11


Effective Density =   7.8886091E-31
```

Figure 3.31

Dehomogenization

Preprocessor → Solution → Dehomogenization → Solid Model.

Input the global behavior from the macroscopic structural analysis. Please refer to SwiftComp™ manual for meaning of the global behavior parameters. Click Ok (see Figure 3.32).

Wait for SwiftComp™ to finish the computation. The post-processing results will be automatically loaded. The default value is the magnitude of displacement as in Figure 3.33.

Contour plots are available for all local fields including three displacement components and its magnitude, six strain components and six stress components. General Postproc → Plot Results → Contour Plot.

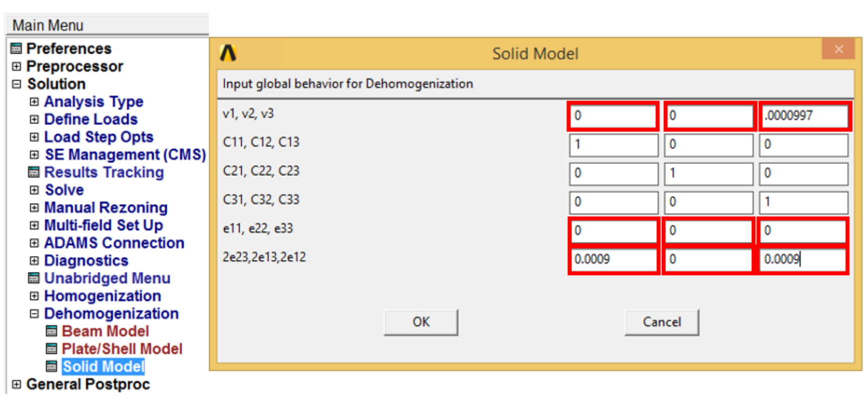


Figure 3.32

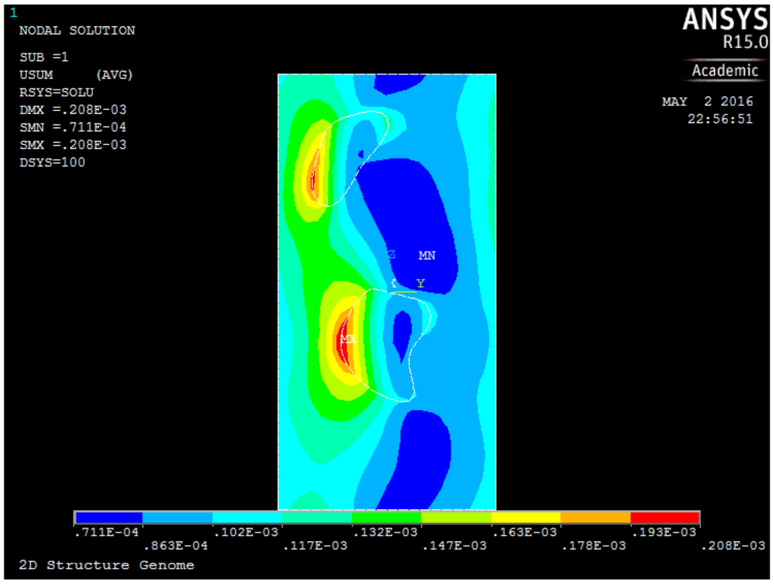


Figure 3.33

3.4 Summary

ANSYS-SwiftComp GUI could be used to create user-defined models according to analysis needs. In general, users need to complete the flowing steps to create the user-defined SG model:

1. Define materials
2. Create geometry (areas or volumes)
3. Define element type
4. Define mesh size
5. Mesh
6. Run SwiftComp.