

Elastic analysis of a nacelle modeled using Abaqus and its constituent composite plate homogenized using Gmsh4SC

Elastic analysis of a nacelle modeled using Abaqus and its constituent composite plate homogenized using Gmsh4SC

This tutorial shows how to calculate cross-sectional properties of a composite plate with elastic matrix and unidirectional transversely isotropic elastic fibers for given orientation and material properties using Gmsh4SC. These properties are used to model a nacelle of a wind-turbine in Abaqus, as a homogeneous solid body. Hence the nacelle component can be subjected to structural analysis, with reduced computational effort.

The orientation ie layup of the plates are (0/30/45/90/0/0/90/-45/-30/0) and the materials used are M30S fibers and PMT-F7 epoxy resin with the material properties given below.

The material properties of the PMT-F7 epoxy are, $E = 2990 \text{ MPa}$ and $\nu = 0.36$.

Properties	M30S Fiber
Longitudinal Stiffness, E_1 (MPa)	294,000
Transverse Stiffness, E_2 (MPa)	29148
Shear Stiffness, G_{12} (MPa)	11310
In-plane Shear Stiffness, G_{23} (MPa)	10000
Poisson's Ratio, $\nu_{12}=\nu_{13}$	0.2
Poisson's Ratio, ν_{23}	0.46

Fiber properties of

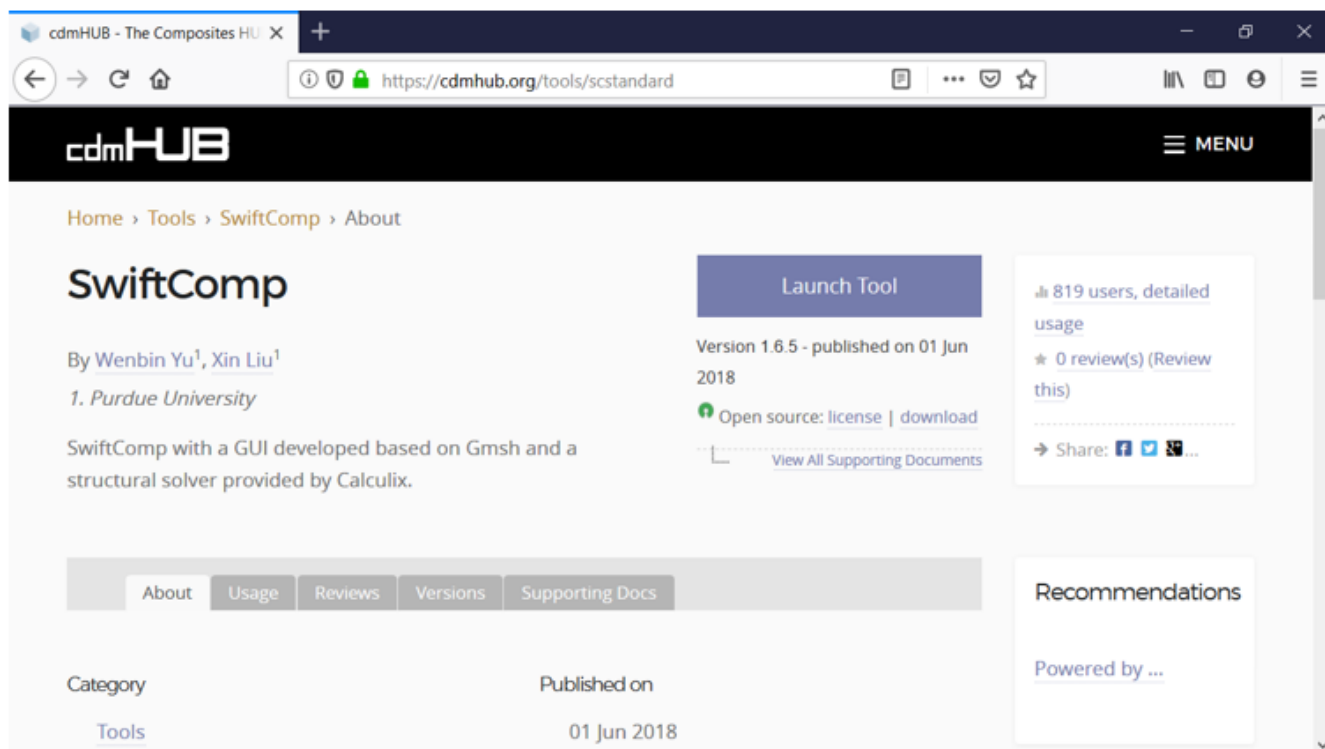
M30S

Solution The linear elastic MSG-based modeling of plate is solved in three steps. 1) Elastic homogenization of fiber and matrix at microscale, using Gmsh4SC. 2) Elastic homogenization of the layup (stack of oriented plates), using Gmsh4SC. 3) Modeling of the Nacelle component in Abaqus as a homogeneous body.

1) Elastic homogenization of fiber and matrix at microscale, using Gmsh4SC.

Gmsh4SC is available at cdmHUB.org. Go to <https://cdmhub.org/tools/scstandard> and launch

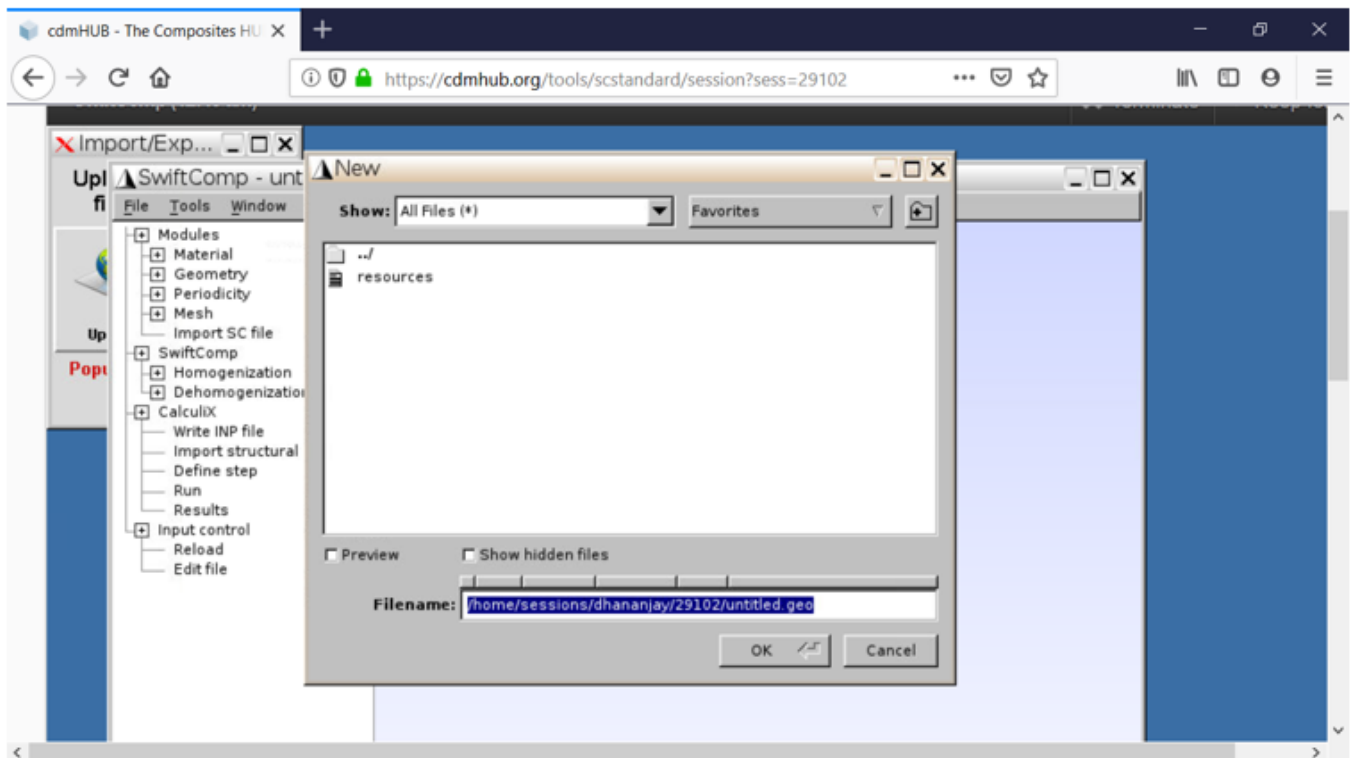
the Gmsh4SC tool.



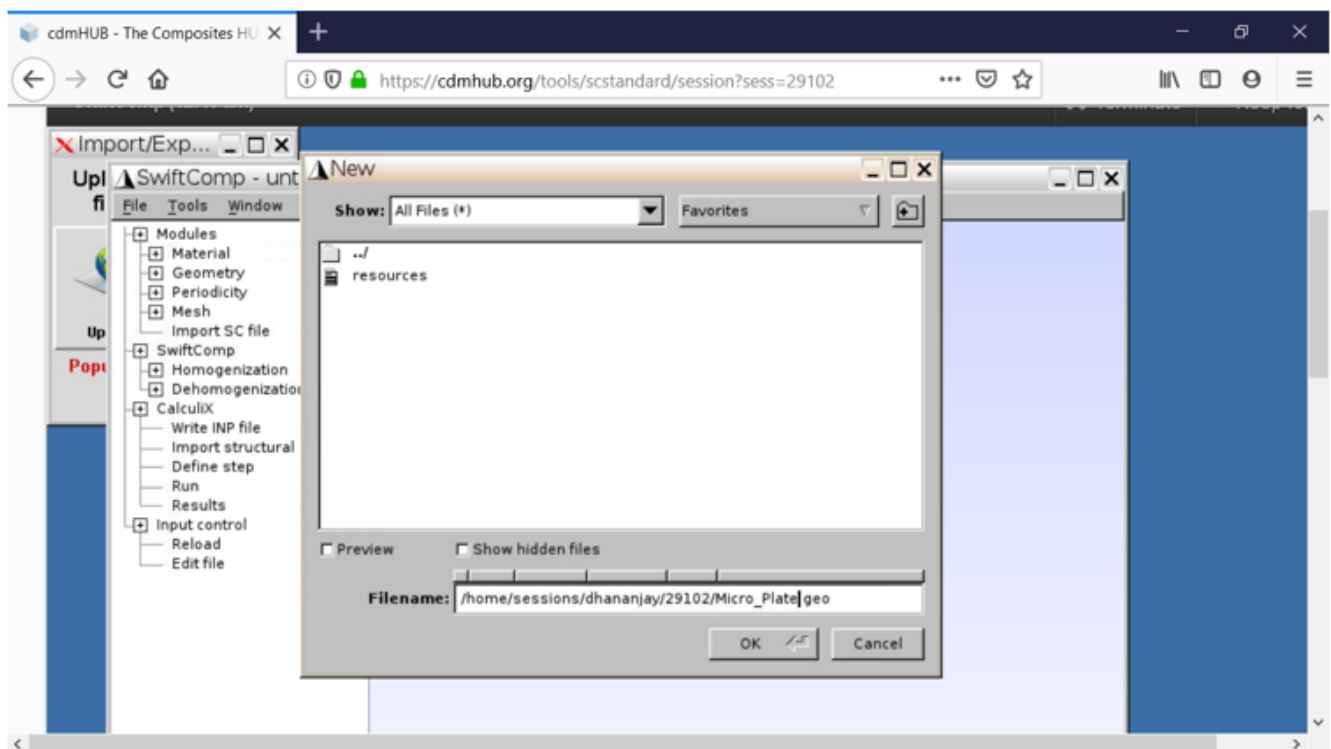
Gmsh4SC

1.1 Creating a new model

Go to File select New and save the model as shown in the figures below.



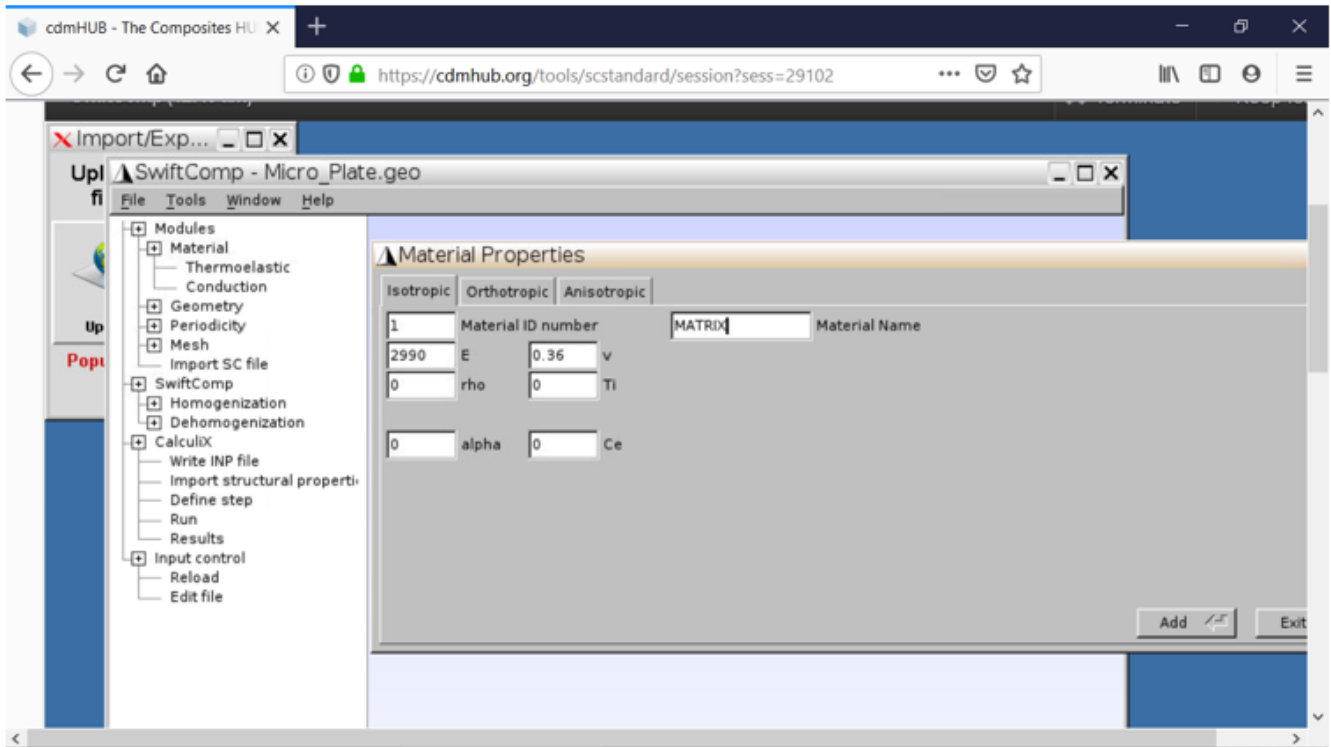
Saving the new model



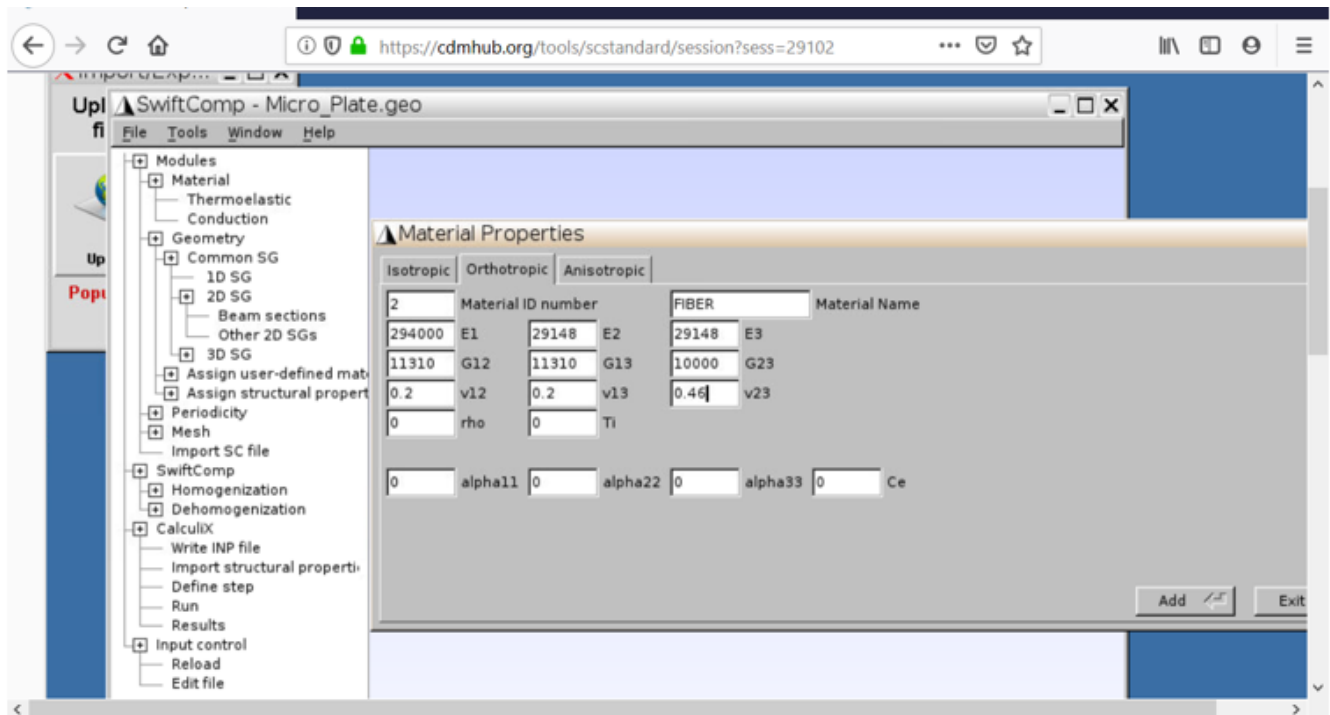
Saving the new model

1.2 Assigning the material properties:

Go to Material select Thermoelastic and in the pop up window choose the material type (Isotropic, Orthotropic or Anisotropic) and set the material constants and material name, then select Add. Repeat this for all the materials required for the model and close the material properties window. For this tutorial we add the isotropic matrix and transversely isotropic elastic fiber properties under appropriate names.



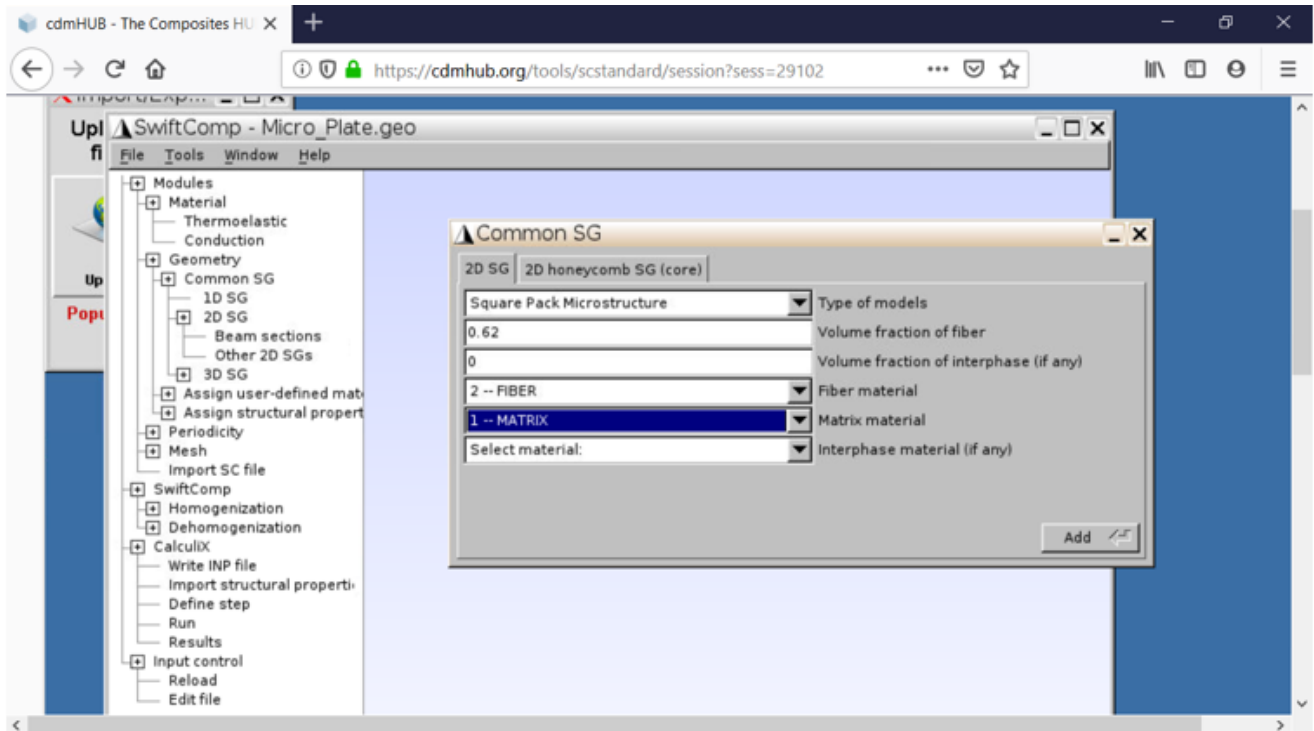
Assigning matrix material properties



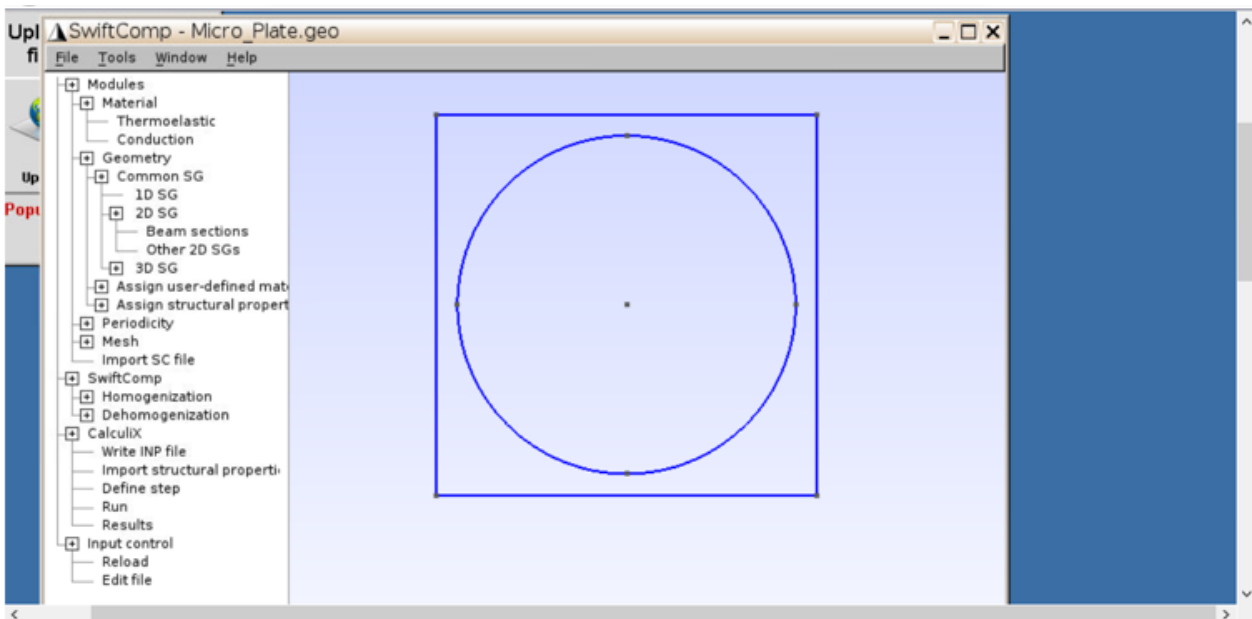
Assigning fiber material properties

1.3 Creating geometry

Go to Geometry select 2D SG and choose Other 2D SGs and in the pop up window, set the Type of models as Square Pack Microstructure. Modify the Volume fraction of the fiber accordingly and set the Fiber material and Matrix material as required and Add. Close the Window.



Creating geometry

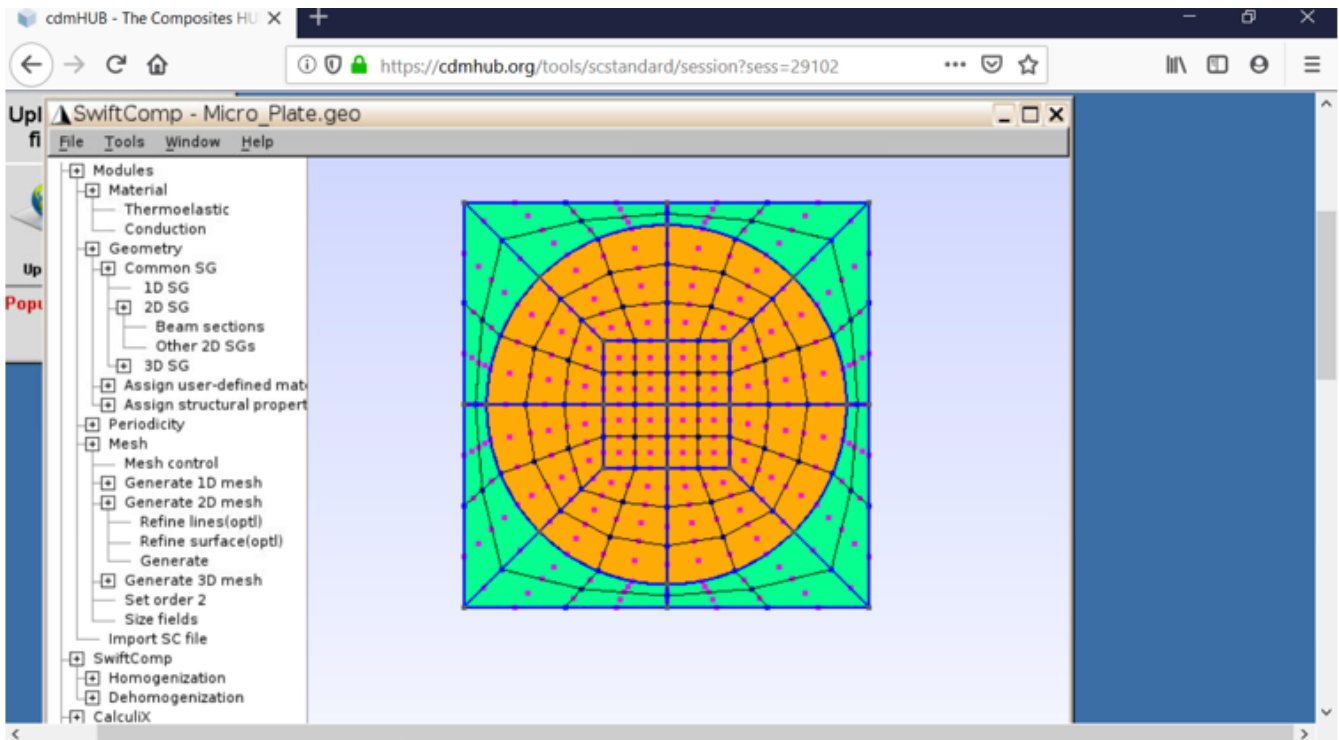


Creating geometry

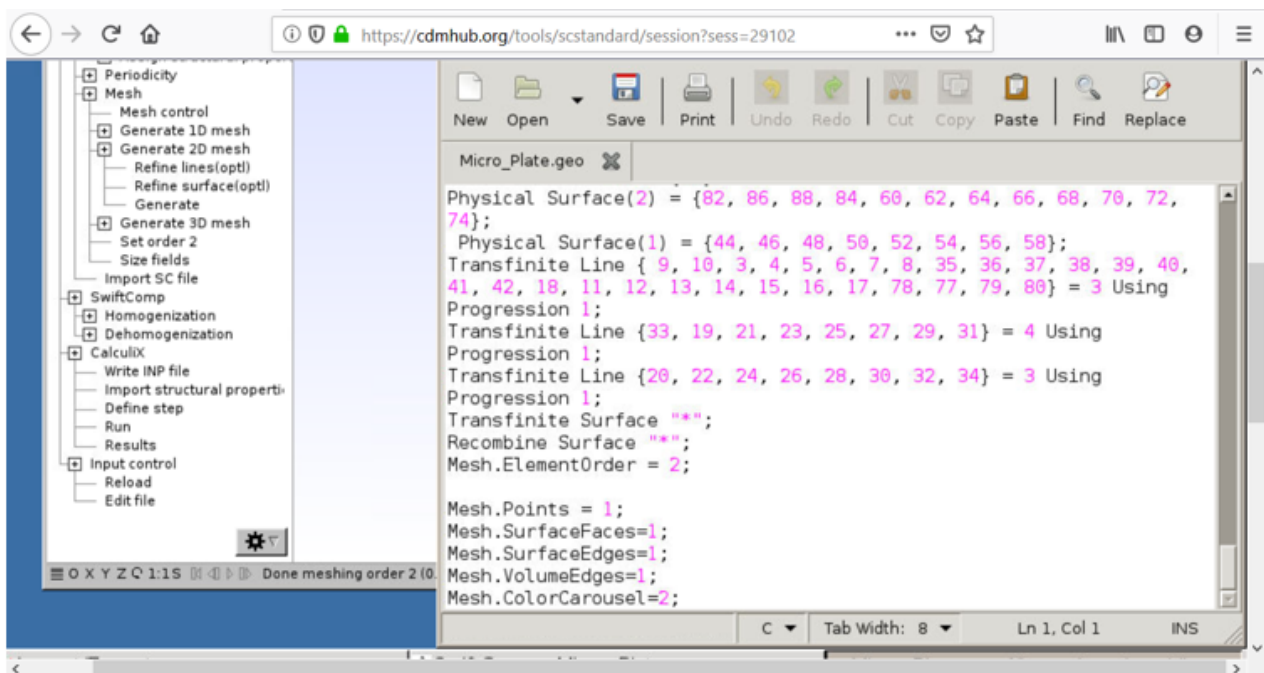
1.4 Setting the mesh

Go to Mesh select Generate 2D mesh and choose Generate. To get a finer mesh, go to Input control, click on Edit file and in the pop up window and scroll down to the three Transfinite line commands and increase their value by the factor you wish to increase your mesh. In this

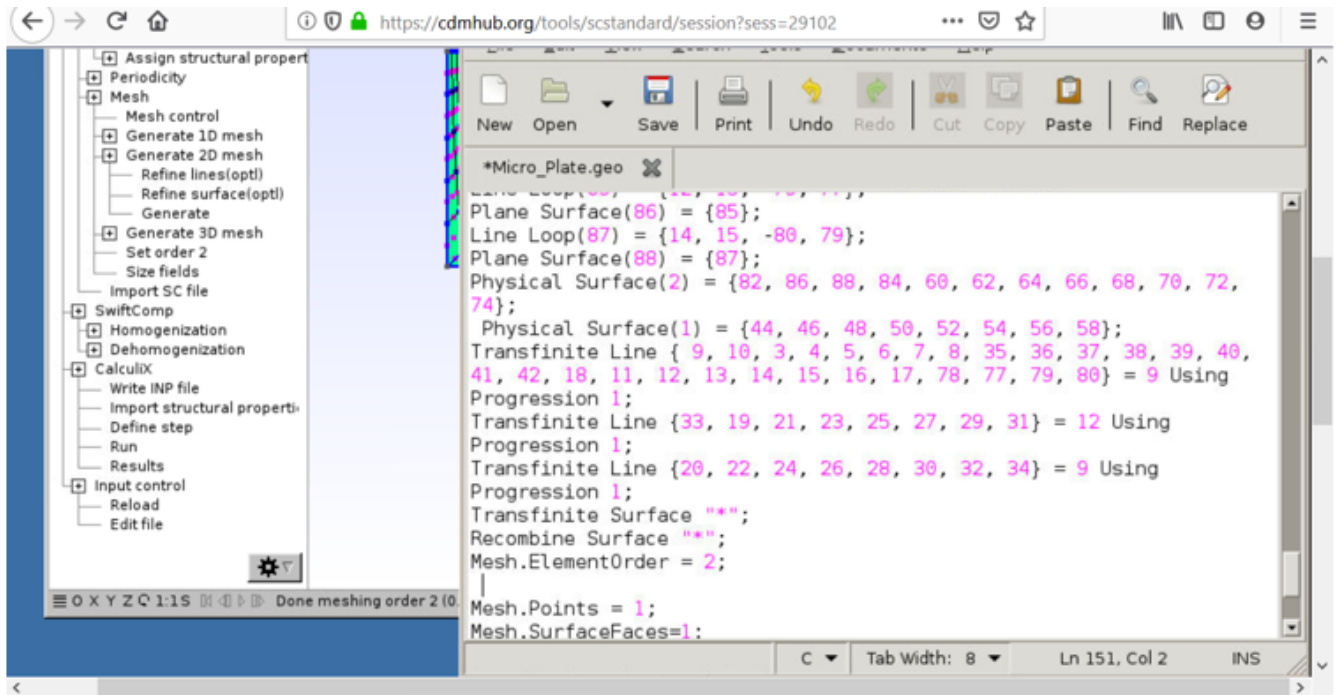
example the mesh is increased by a factor of three. Then save the changes in the window and exit it. Generate the mesh again.



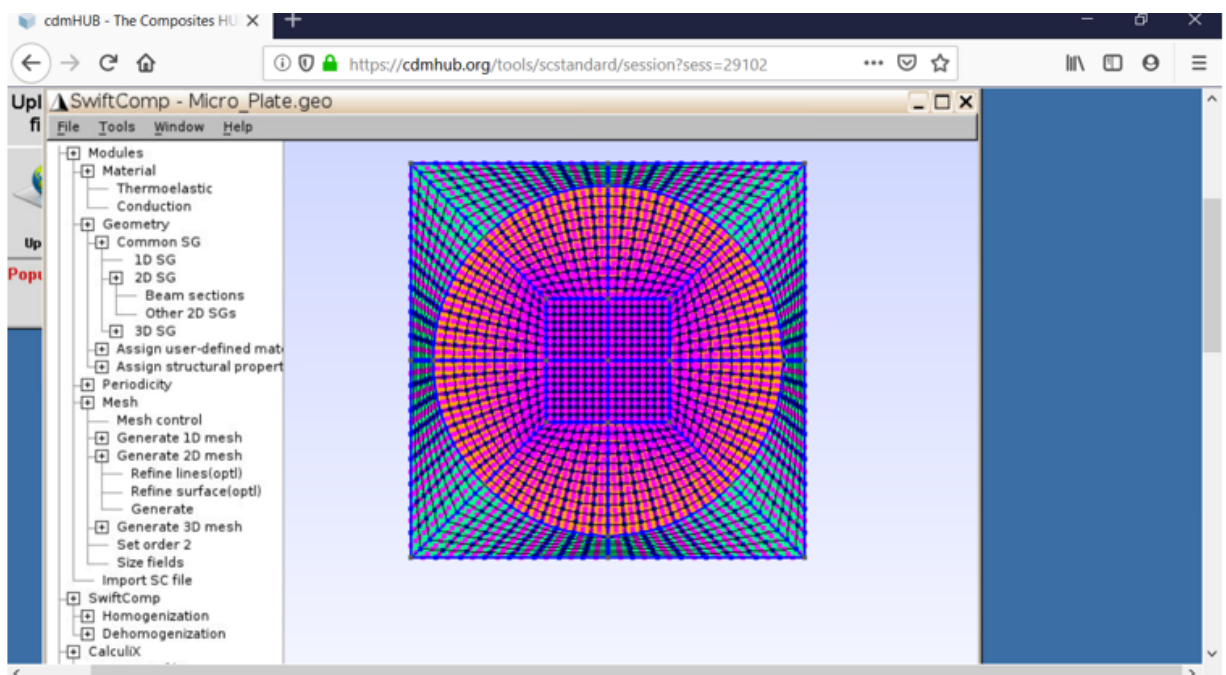
Meshing the model



Meshing the model



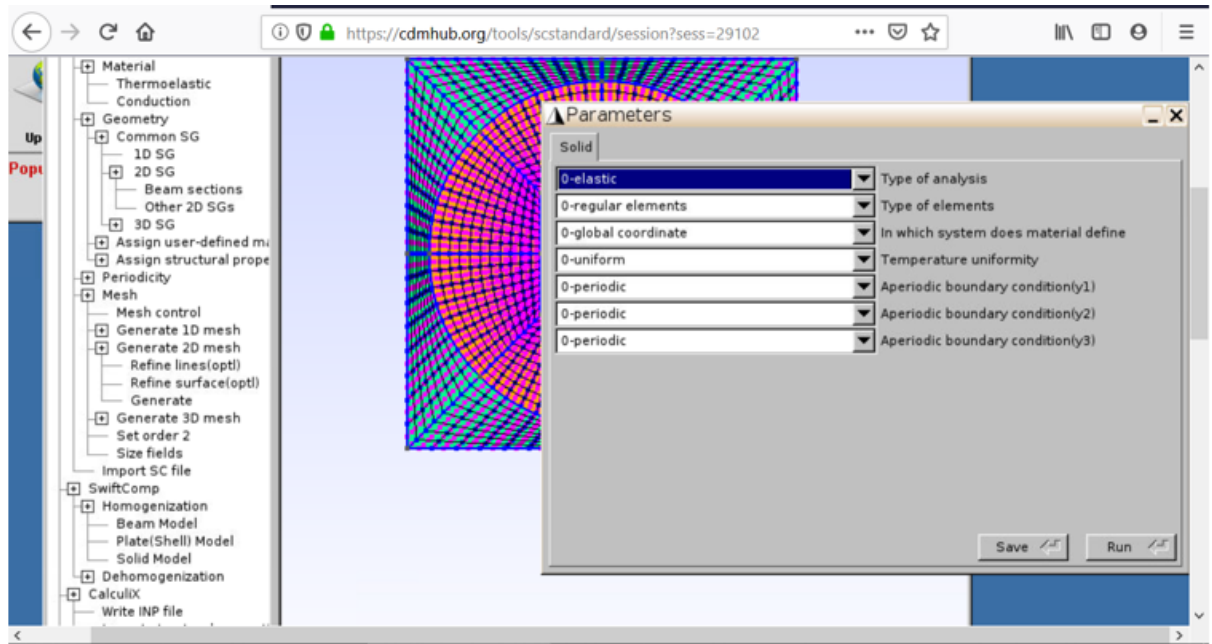
Meshing the model



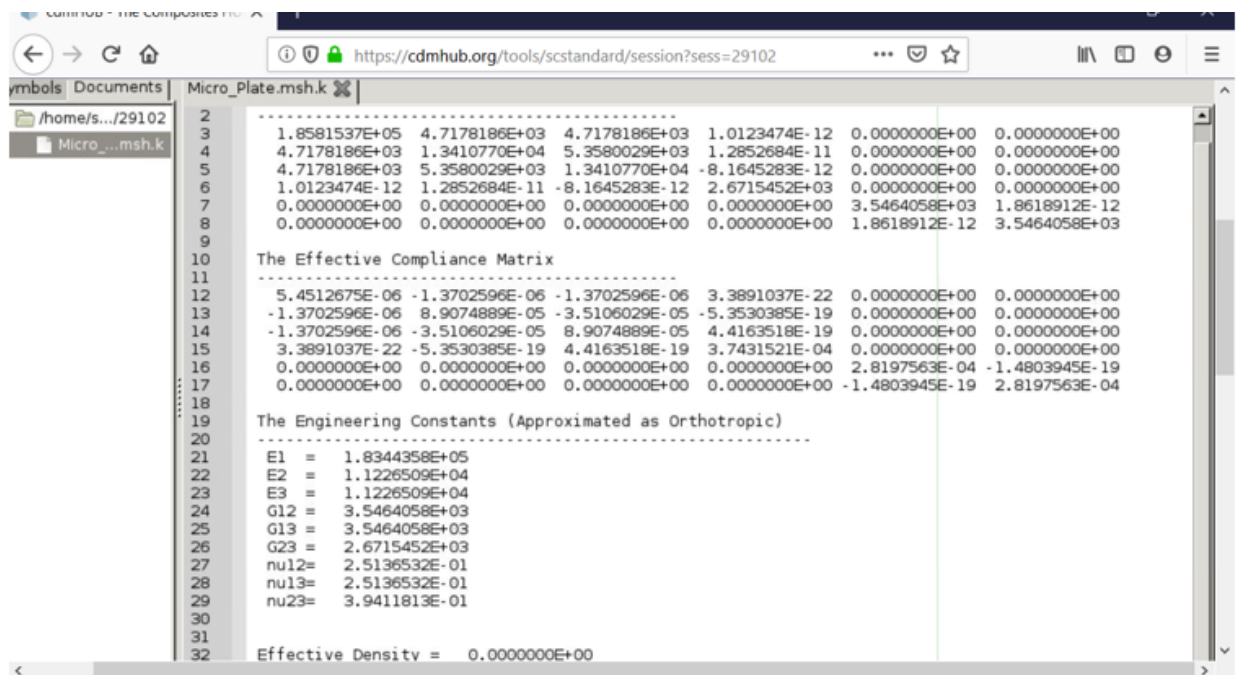
Meshing the model

1.5 Homogenization

Go to [SwiftComp](#) and select Homogenization, and in the window keep the preset options, save the model and run it. The homogenized results will be displayed in a new pop up window.



Homogenization



Microscale Results

1.6 Save the results using an ftp app or note the for the next step.

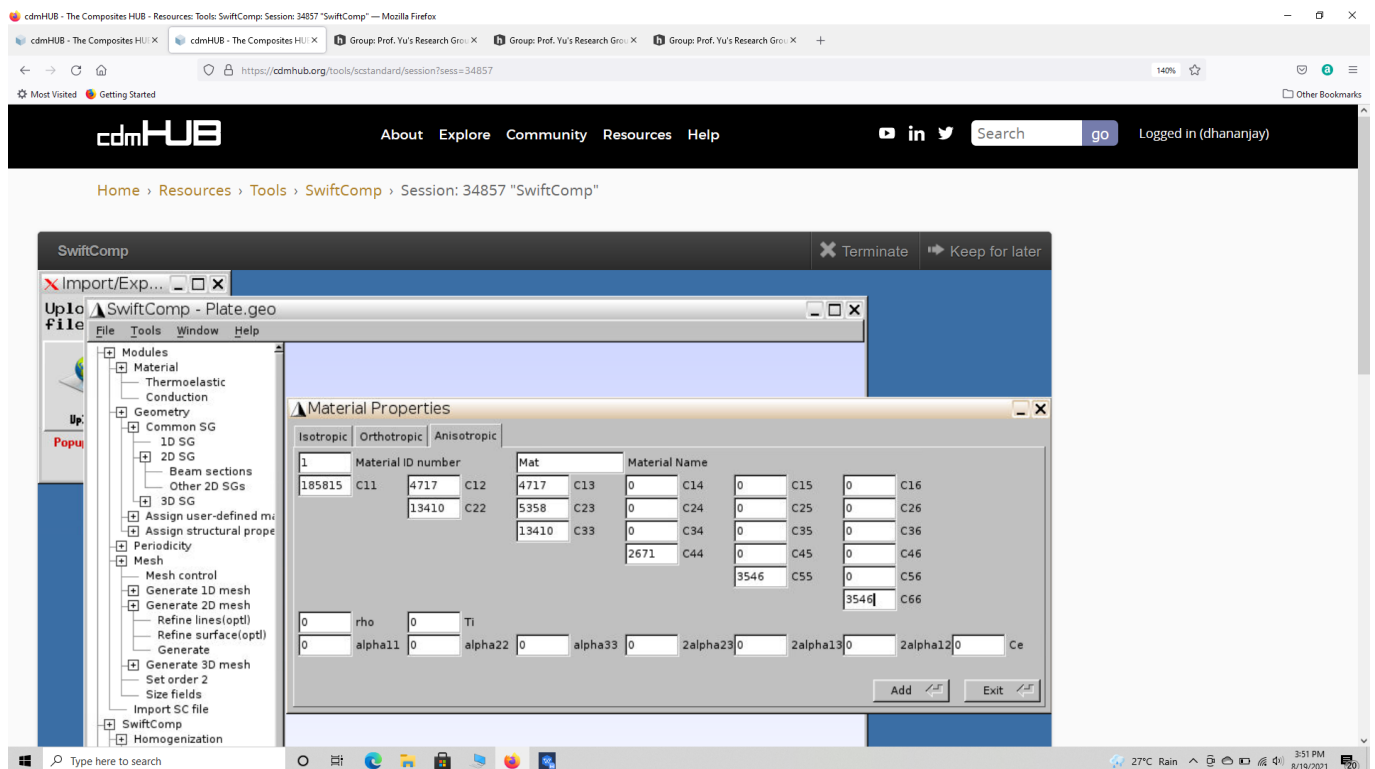
2) Elastic homogenization of the layup (stack of oriented plates), using Gmsh4SC.

2.1 Open a new model

Go to File select New and save the model as shown previously.

2.2 Assigning the material properties:

Go to Material select Thermoelastic and in the pop up window choose the material type. Use the approximated Orthotropic engineering constants or use the stiffness matrix (more accurate) from the microscale analysis to enter the material properties, then select Add.

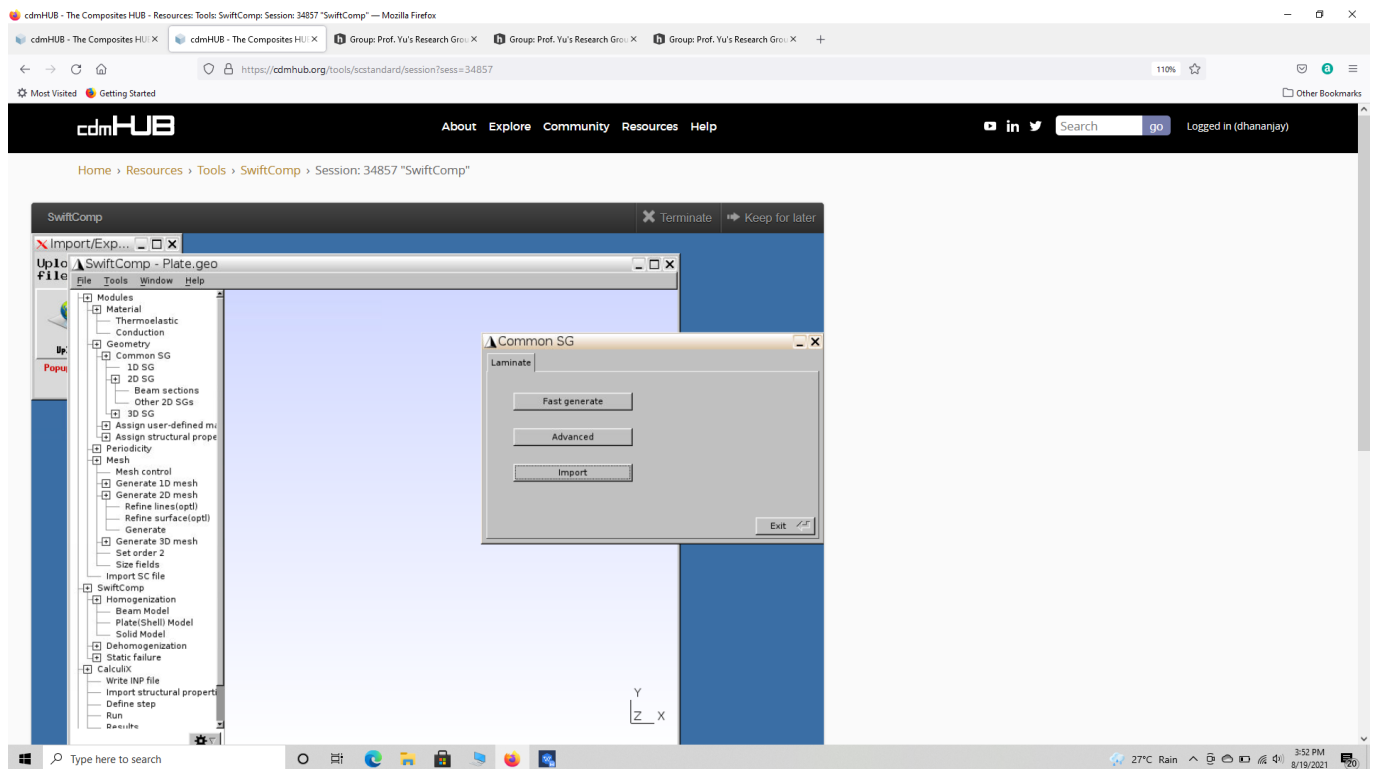


Assigning matrix material properties

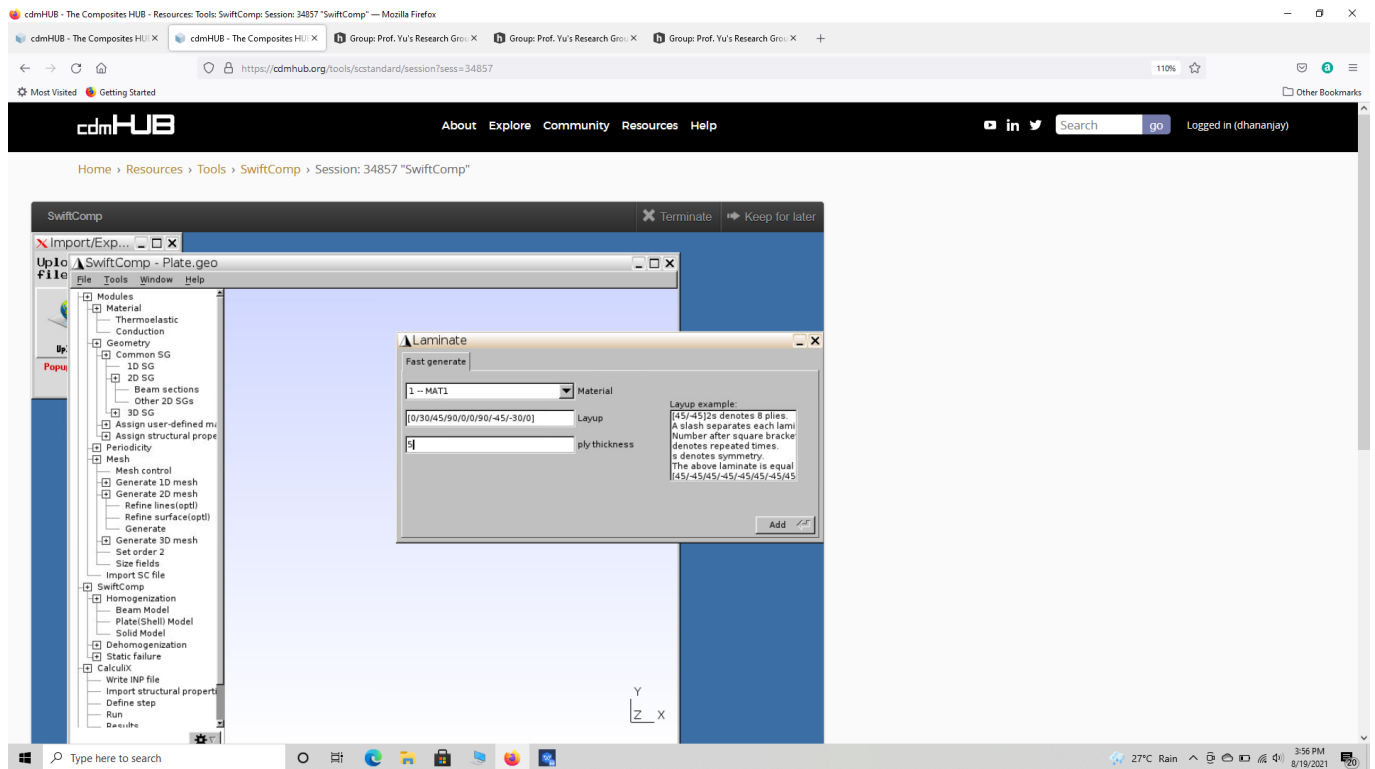
2.3 Creating geometry

Go to Geometry select 1D SG, choose fast generate and enter the layup as (0/30/45/90/0/0/90/-45/30/0), but use the square bracket to enter the layups.

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL

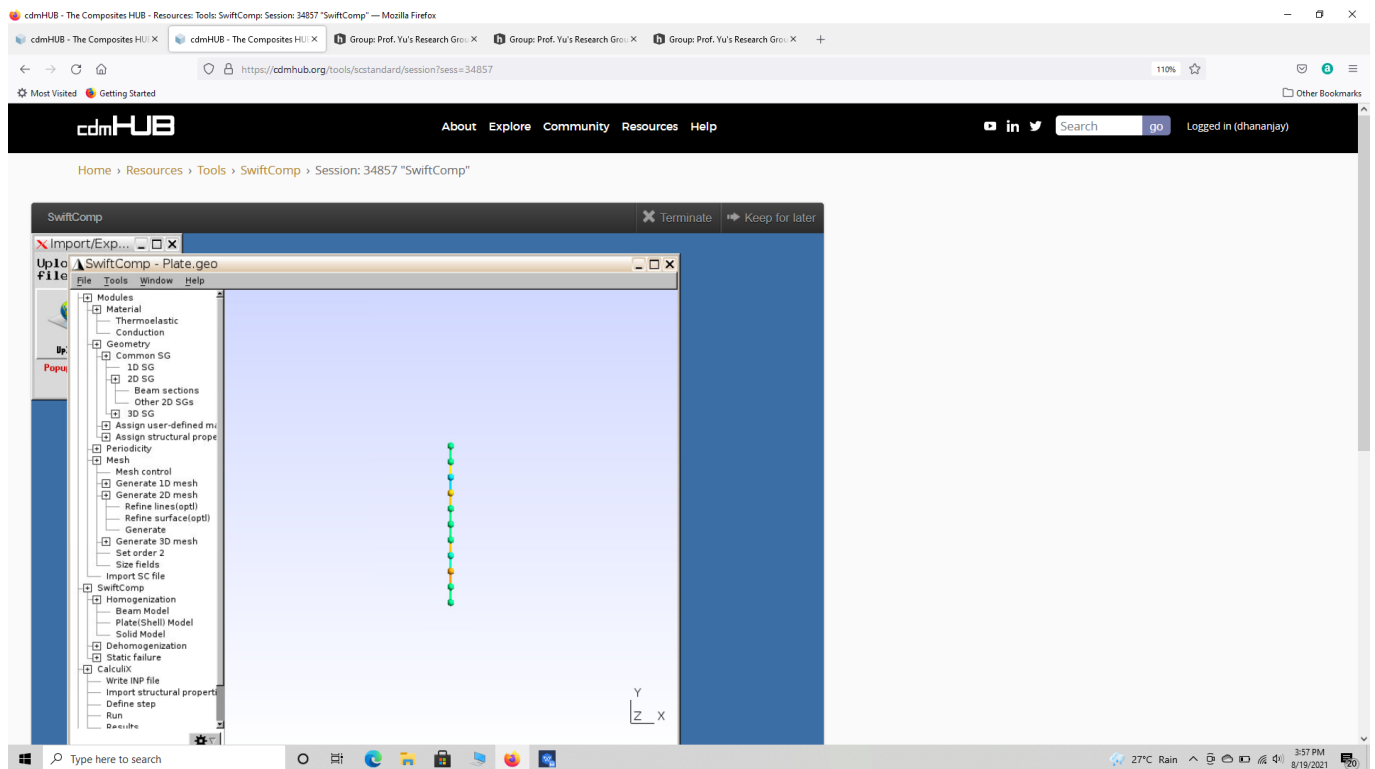


Creating plate SG geometry



Creating plate SG geometry

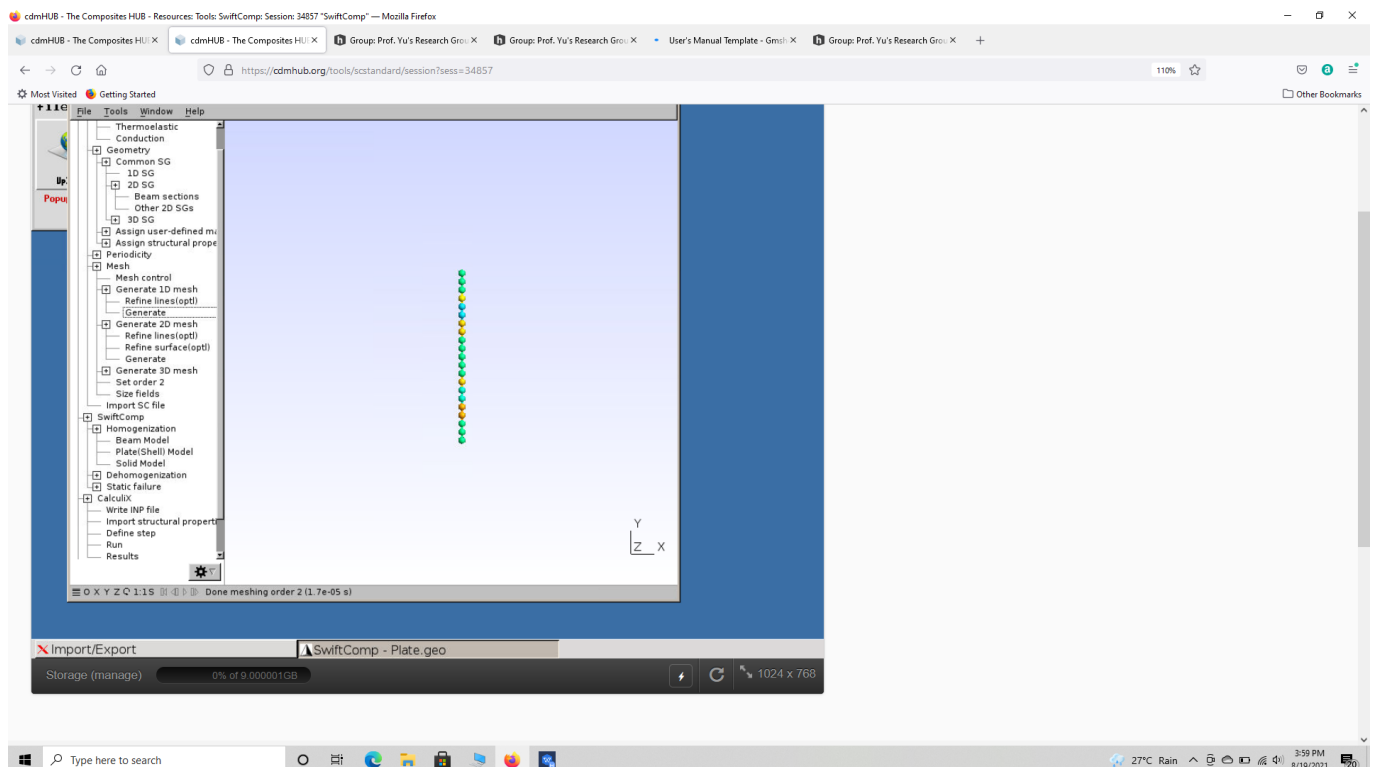
ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL



Creating layout SG geometry

2.4 Setting the mesh

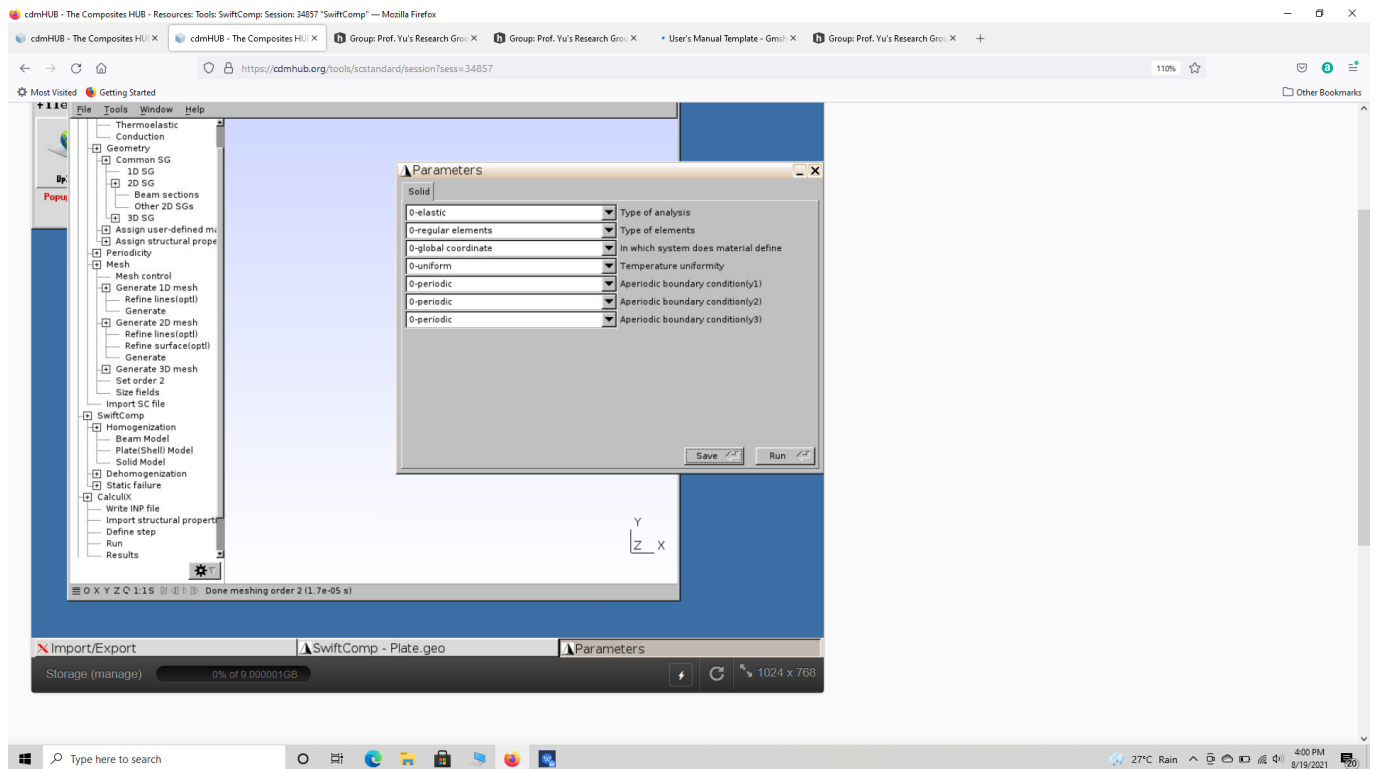
Go to Mesh select Generate 1D mesh and choose Generate.



Creating layup SG mesh

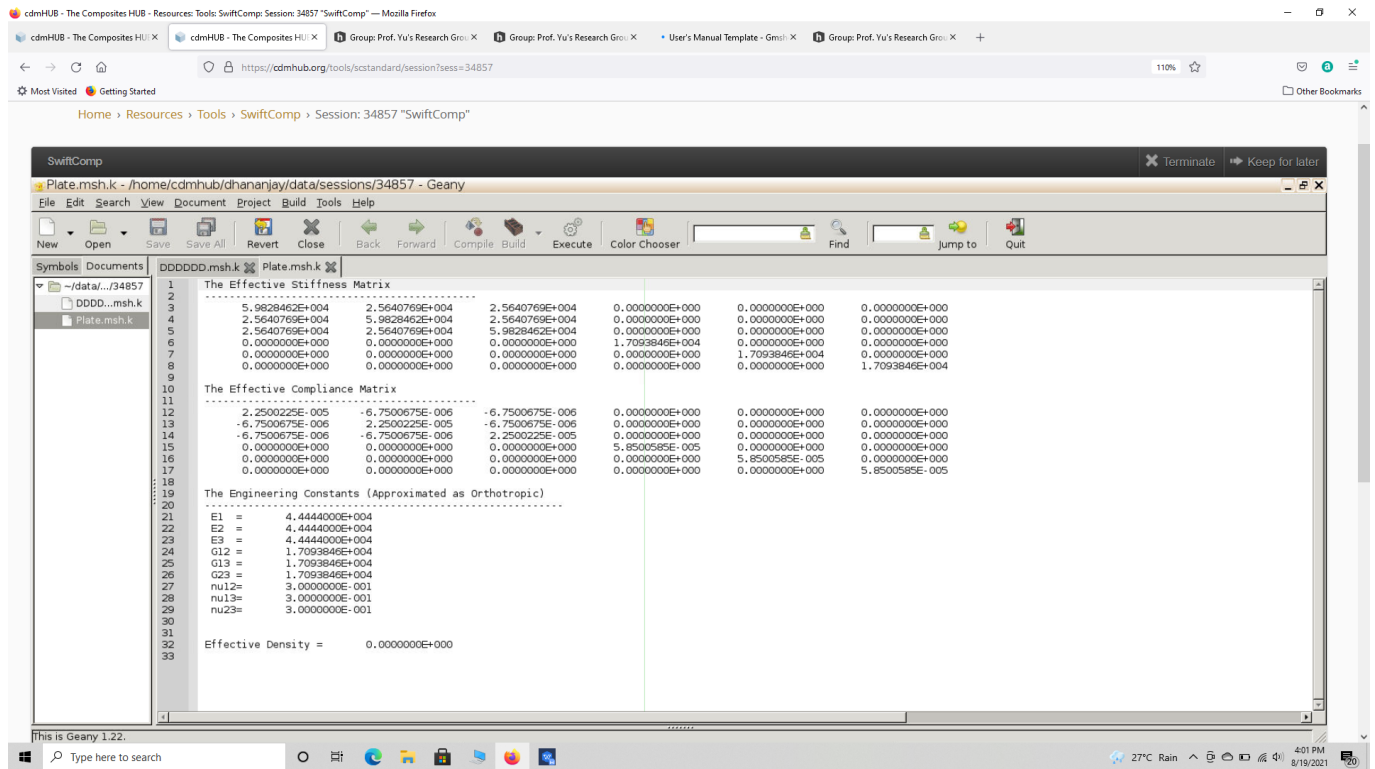
2.5 Homogenization

Go to [SwiftComp](#) and select Homogenization, and in the window keep the preset options, save the model and run it. The homogenized results will be displayed in a new pop up window.



Homogenization of the layup

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL

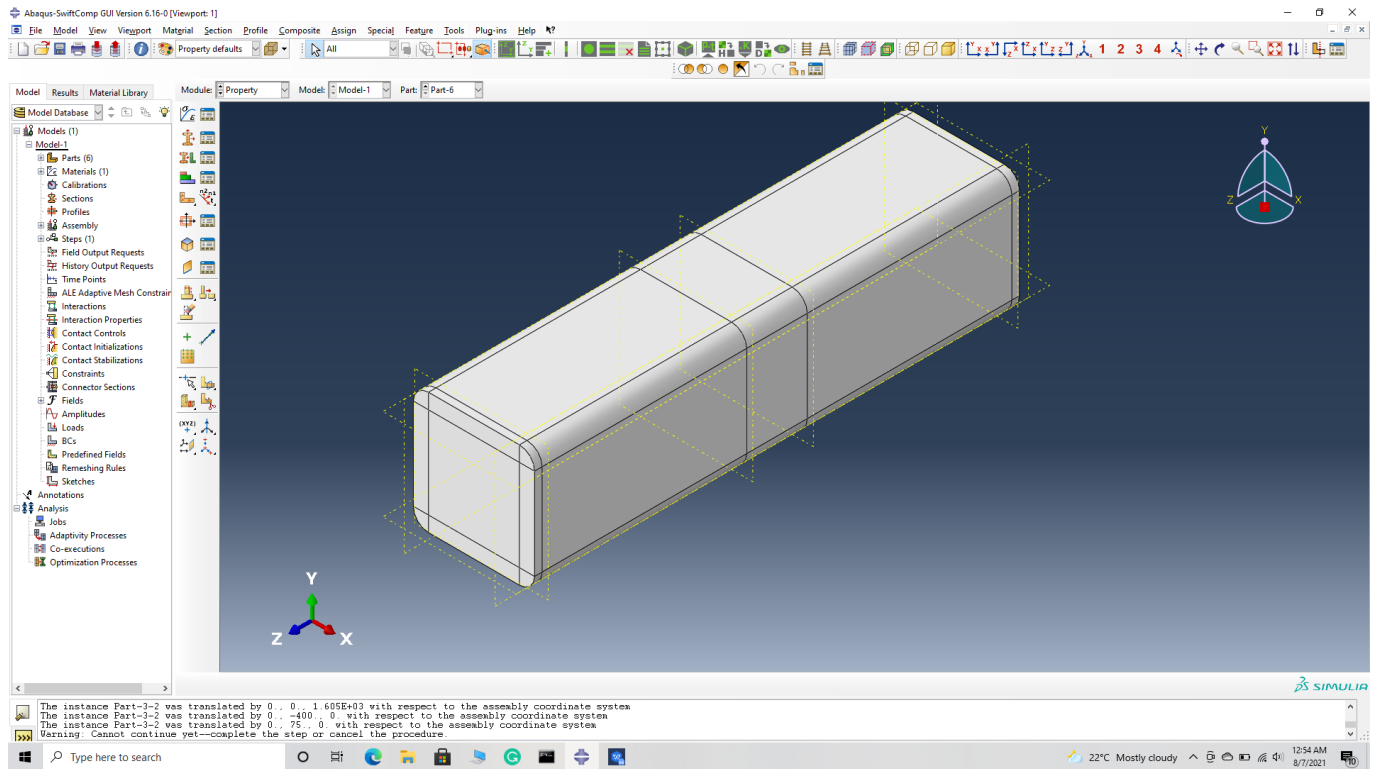


Results of layup homogenization

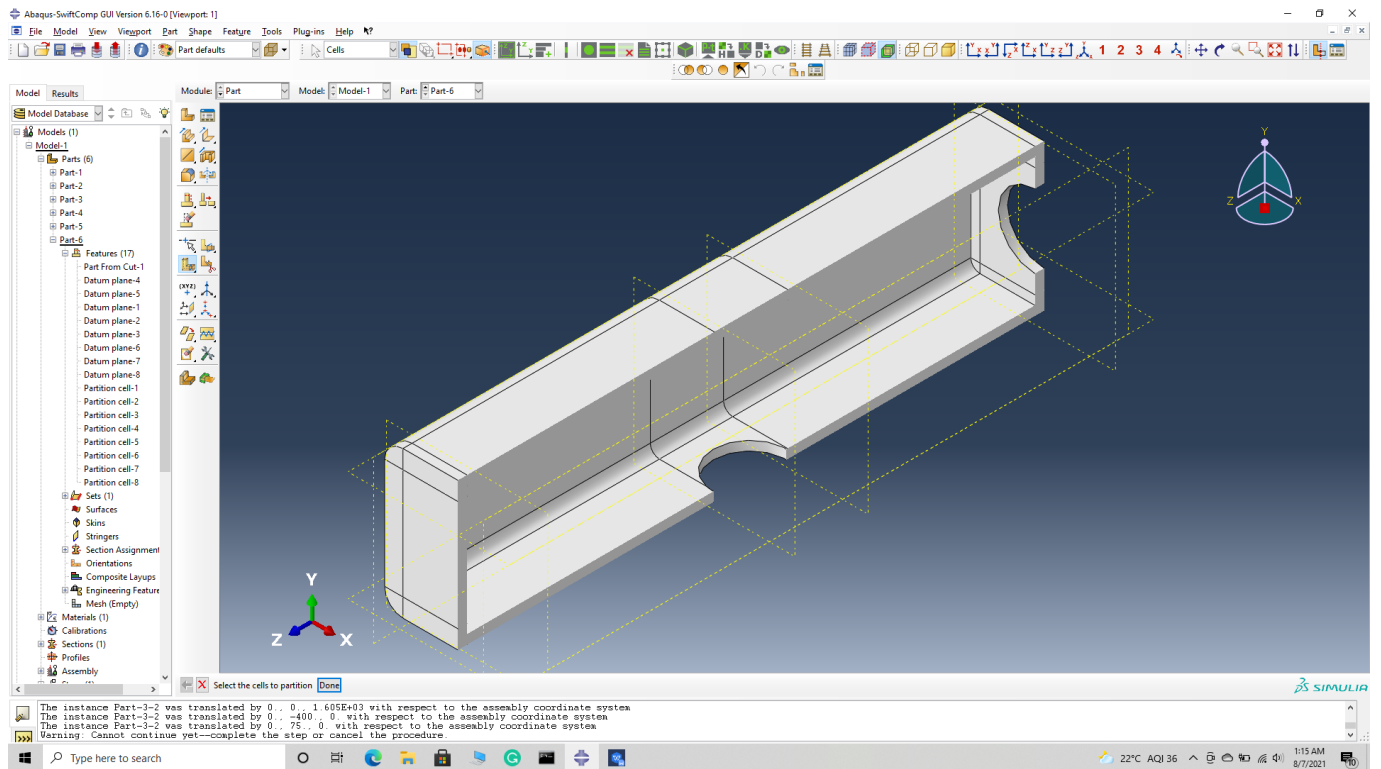
3) Modeling of the Nacelle component in Abaqus as a homogeneous body.

Step 3.1. Create the 3d part shown below using conventional abaqus methodology. Notice we model the nacelle as a 3D homogeneous part. Ensure the part is partitioned as shown to add the material properties later. The half section of the part is also shown.

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL



Nacelle shell



Nacelle shell half section

Step 3.2.' Enter the material properties for the model. First we need to choose the material

properties from the results of the computed effective elastic properties in the previous part. Then we need to convert the Constitutive relations provided as SwiftComp's results into Abaqus's Constitutive relations. This can be done by switching the 4th and 6th rows for the relation and also switching the 4th and 6th column of the stiffness matrix. The relations are provided below. Within the Materials section of Abaqus CAE, we create a material called "Material-1" and add the corresponding properties. We can also import the homogenized properties from the toolbar if we have downloaded the property file from the previous step..

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} = \begin{bmatrix} D_{1111} & D_{1122} & D_{1133} & D_{1112} & D_{1113} & D_{1123} \\ & D_{2222} & D_{2233} & D_{2212} & D_{2213} & D_{2223} \\ & & D_{3333} & D_{3312} & D_{3313} & D_{3323} \\ & & & D_{1212} & D_{1213} & D_{1223} \\ & sym & & & D_{1313} & D_{1323} \\ & & & & & D_{2323} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix}$$

Abaqus's

Constitutive relations

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

SwiftComp's

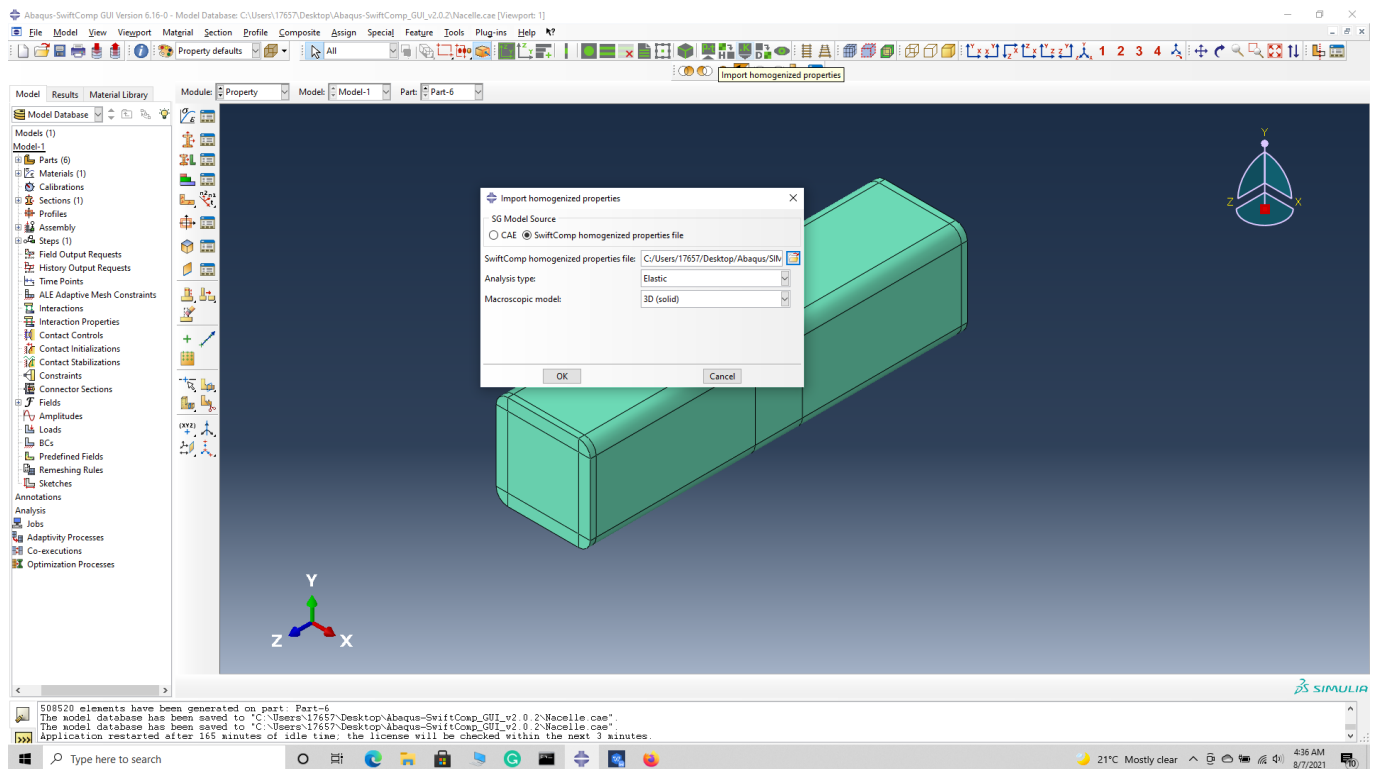
Constitutive relations

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

SwiftComp's

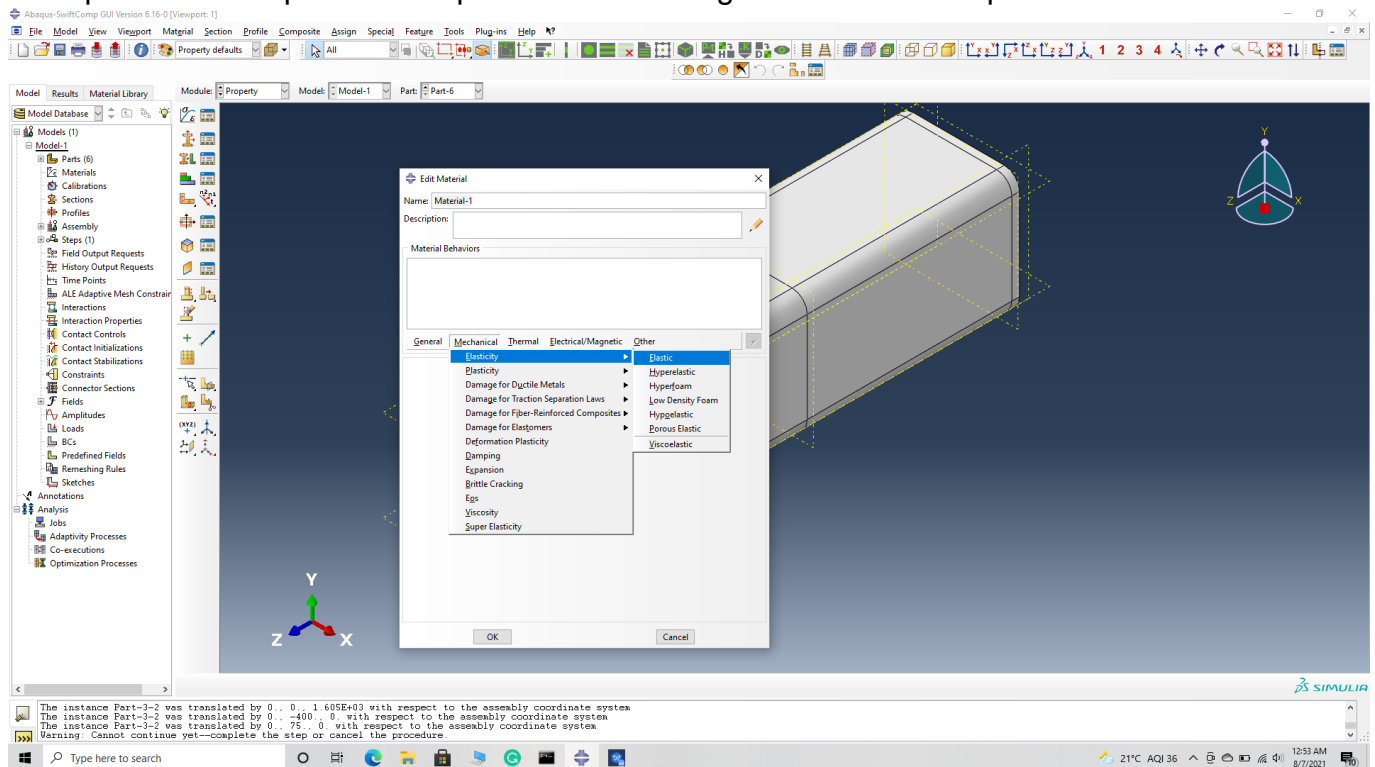
output Constitutive relations converted into Abaqus's input Constitutive relations

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL



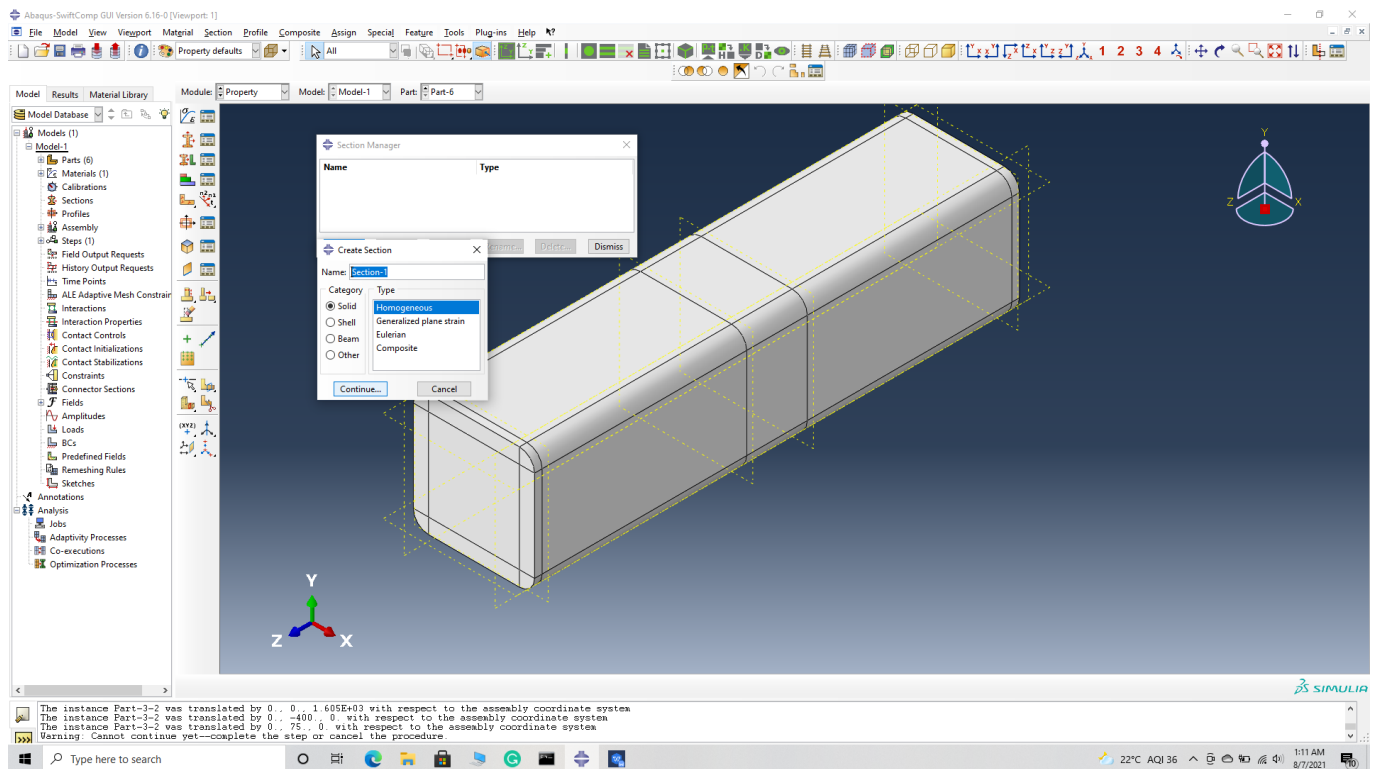
Importing Material properties

Step 3.3.' Create an anisotropic elastic material and add the homogenized material properties as explained in the previous step. Create and assign a section to the part.

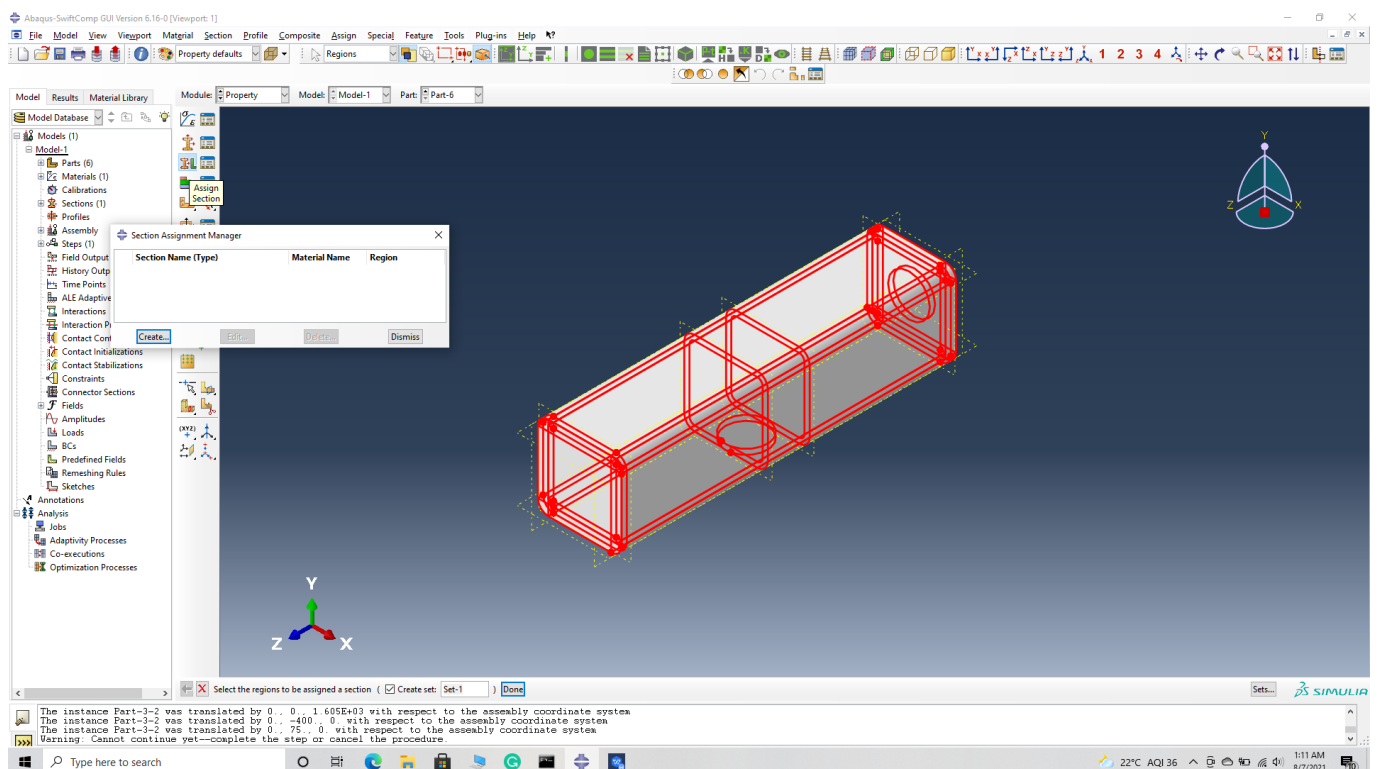


Create an anisotropic elastic material

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL



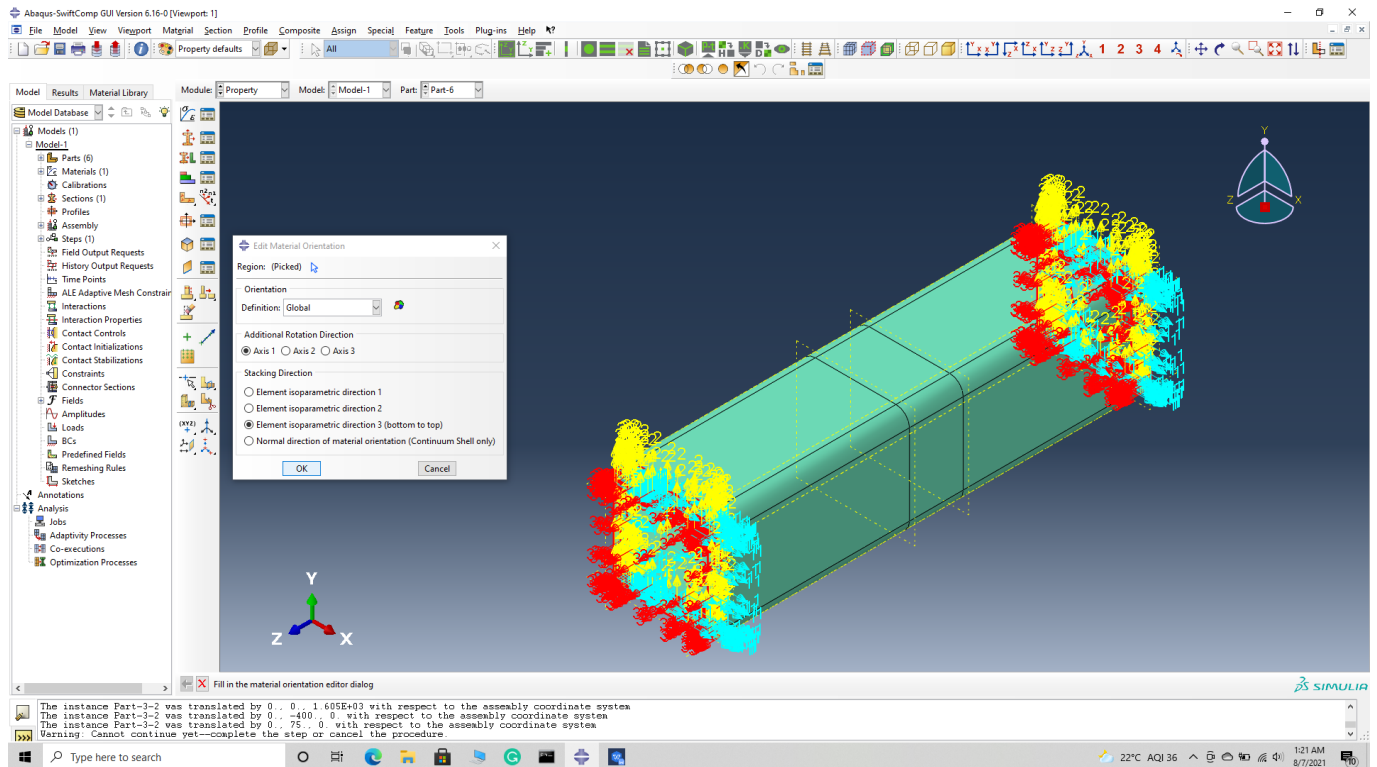
Create a section



assign the section

' # Step 3.4. Now we assign the material orientation for the part. Go to Assign material orientation -> select the end sections of the part and -> Done -> Select a CSYS (use default

orientation or other method) -> set the global orientation to both as shown. They will have opposite orientation if the plies are symmetric. But we are not using symmetric model for the nacelle. -> Done –

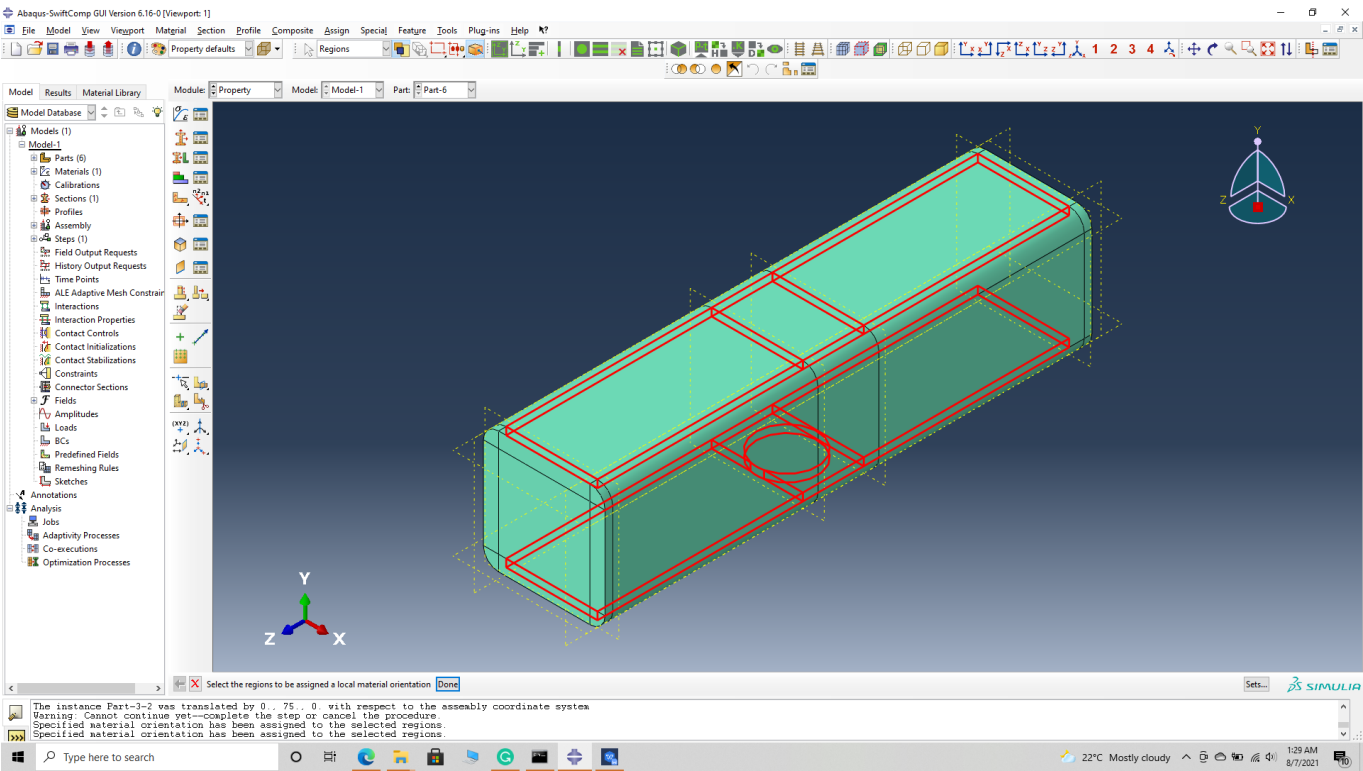


Assign material orientation

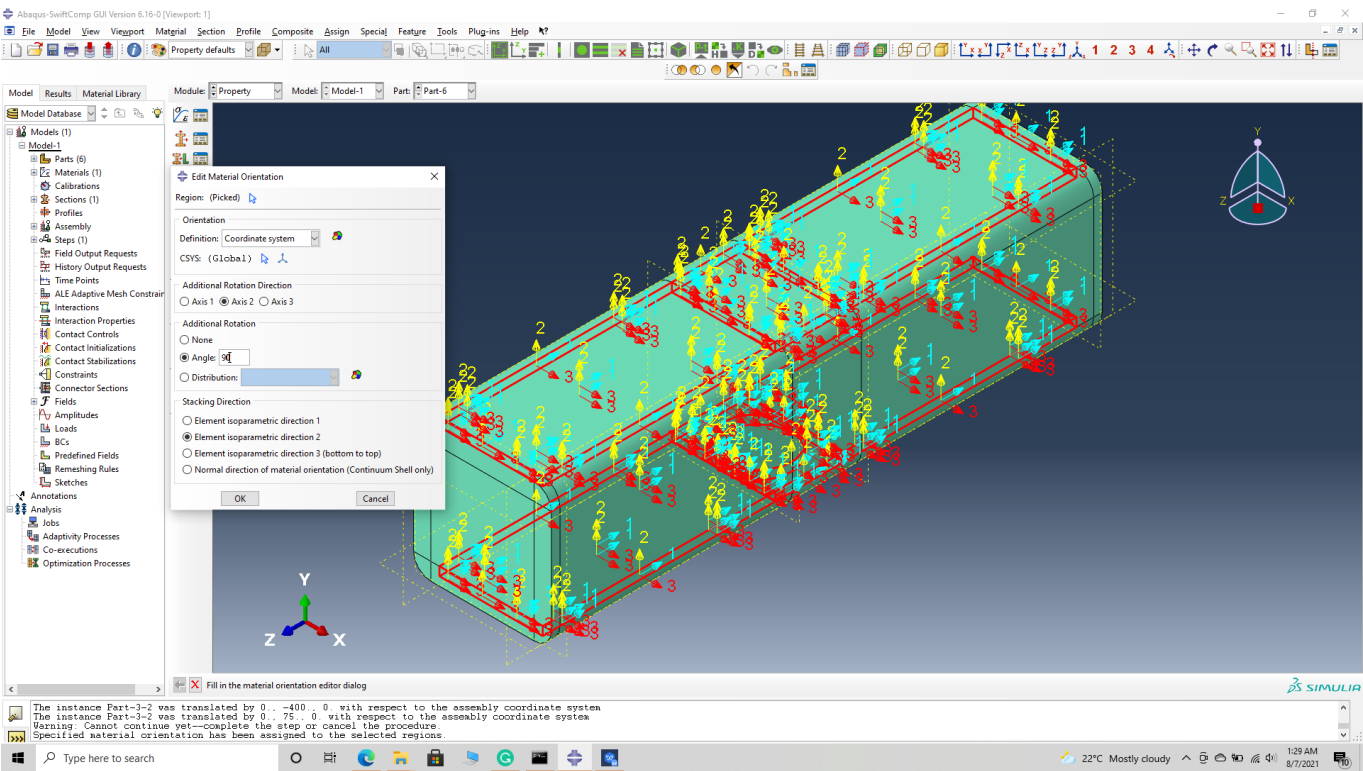
(Image(N3.42.png, desc="Assign material orientation") failed - File not found)

‘ # Step 3.5. Select the top and bottom sections of the part and -> Done -> Select a CSYS (use default orientation or other method) -> set the Coordinate system orientation to both as shown. -> Define -> addition axis of rotation -> axis 2 as 90 degree -> Continue -> OK -> Done –

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL

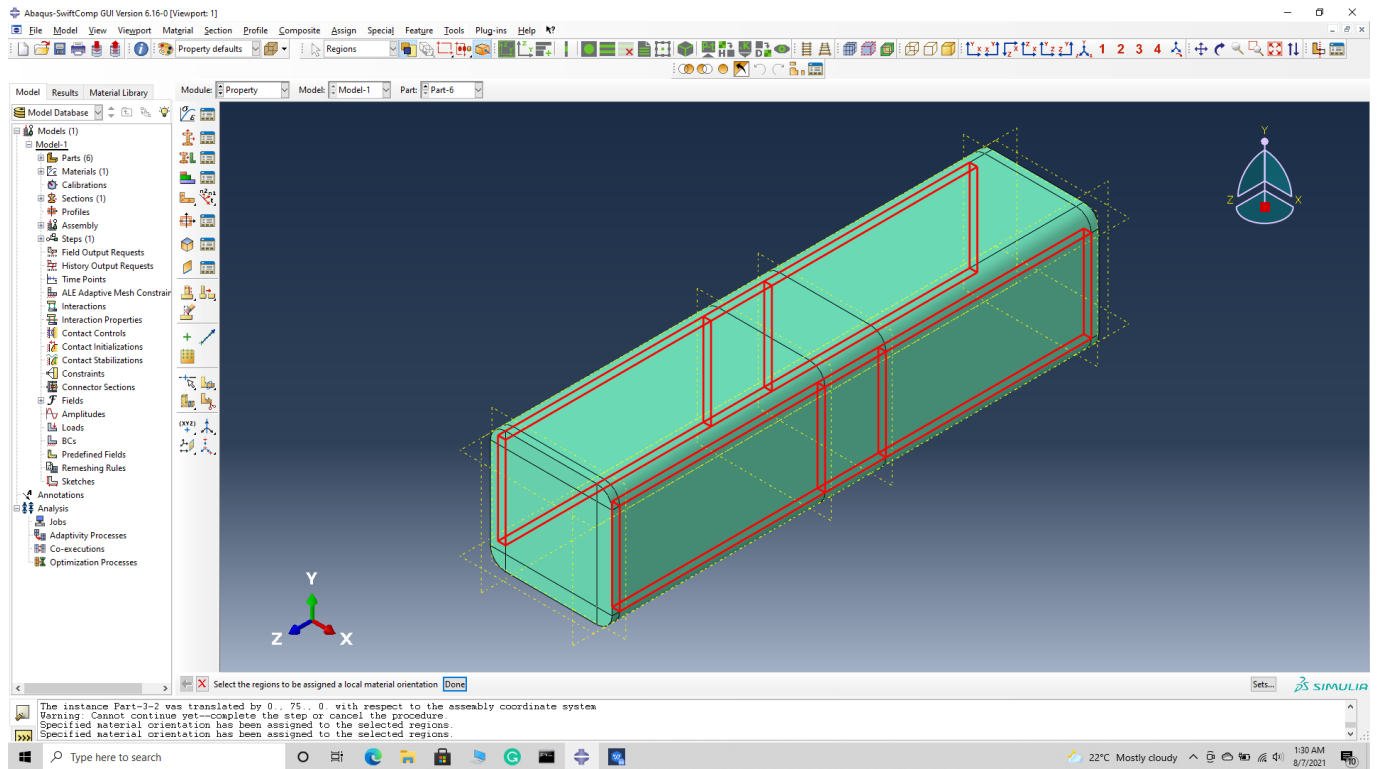


Assign material orientation

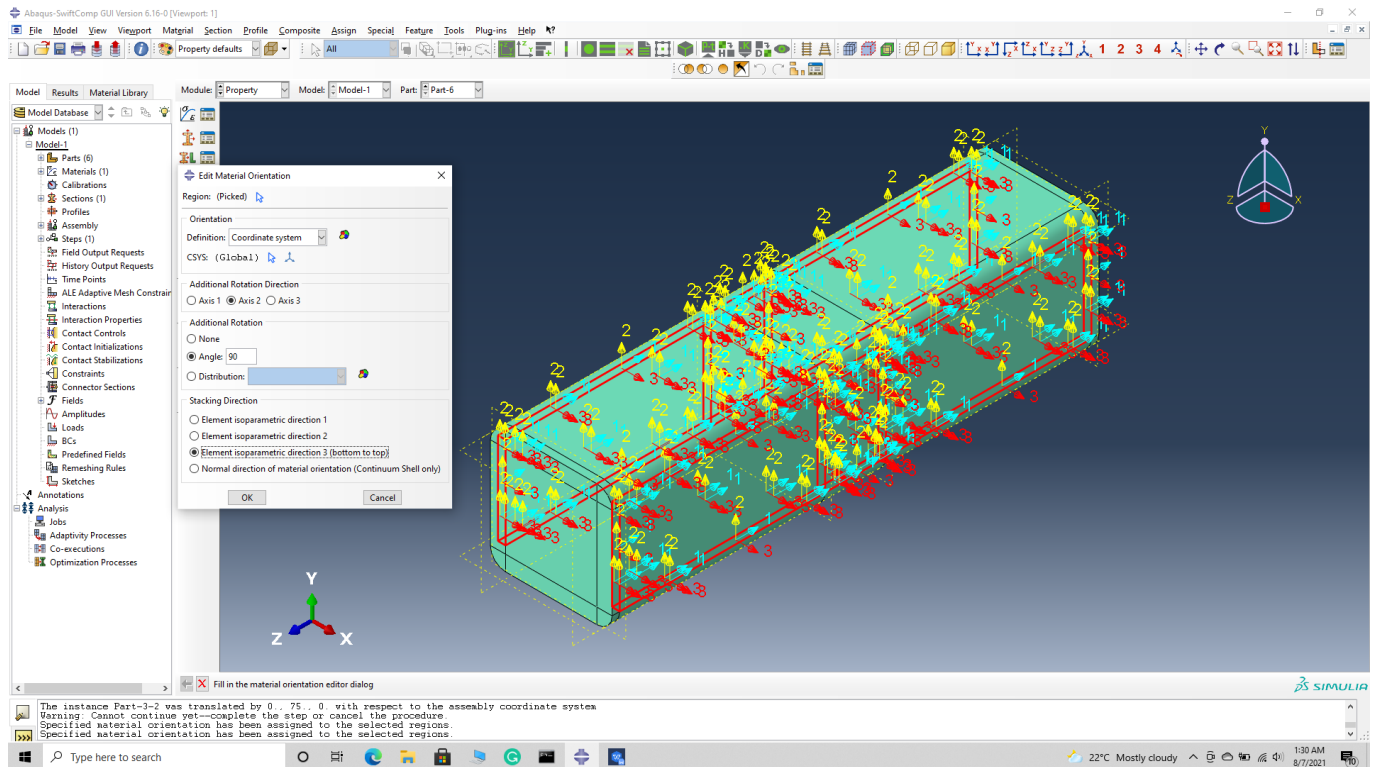


Assign material orientation

Step 3.6. Select the side sections of the part and -> Done -> Select a CSYS (use default orientation or other method) -> set the Coordinate system orientation to both as shown. -> Define -> addition axis of rotation -> axis 2 as 90 degree -> Continue -> OK . -> Done –



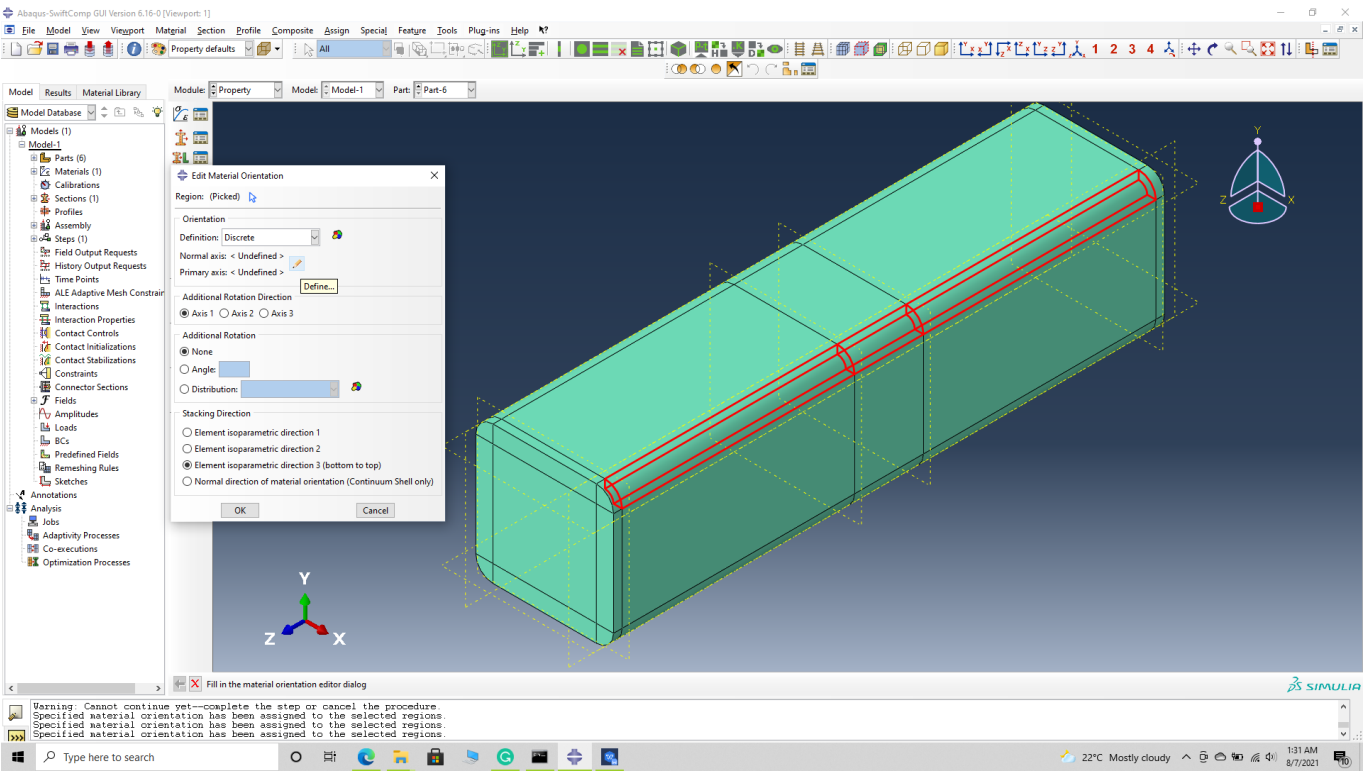
Assign material orientation



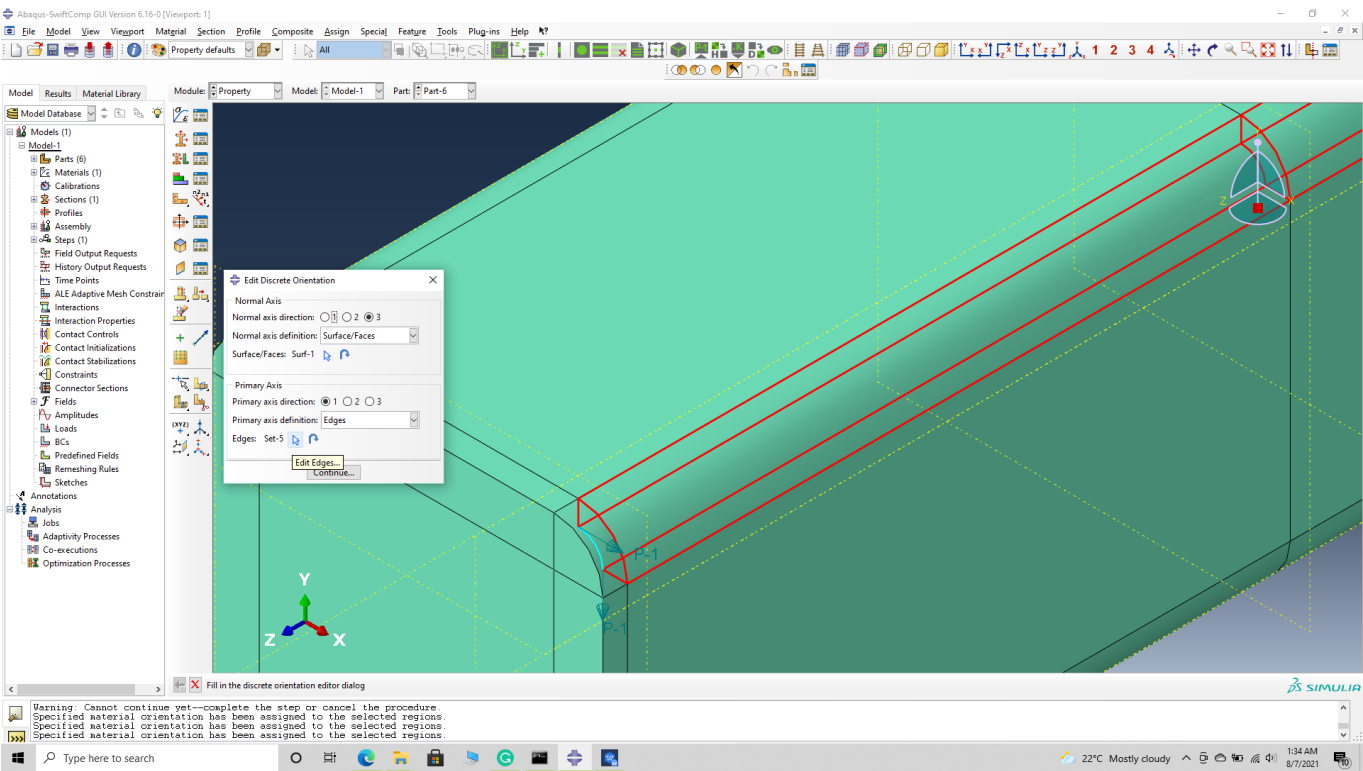
Assign material orientation

Step 3.7. Select the top right curved sections of the part and -> Done -> Select a CSYS (use default orientation or other method) -> set the discrete orientation as shown. -> Define -> Primary axis orientation -> choose edge and flip direction if needed to make the axis point towards a clockwise direction -> Choose the surfaces for the normal axis definition -> Continue -> OK -> Done –

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL

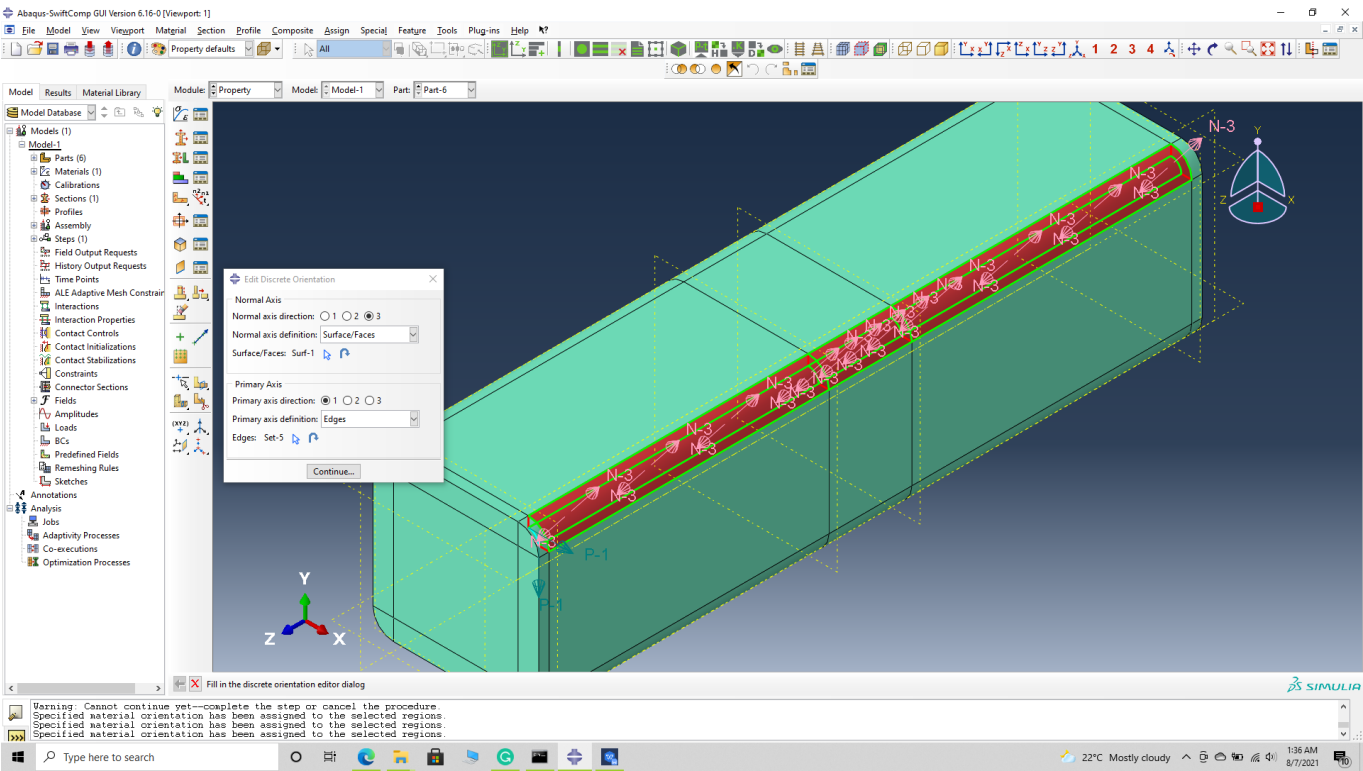


Assign material orientation

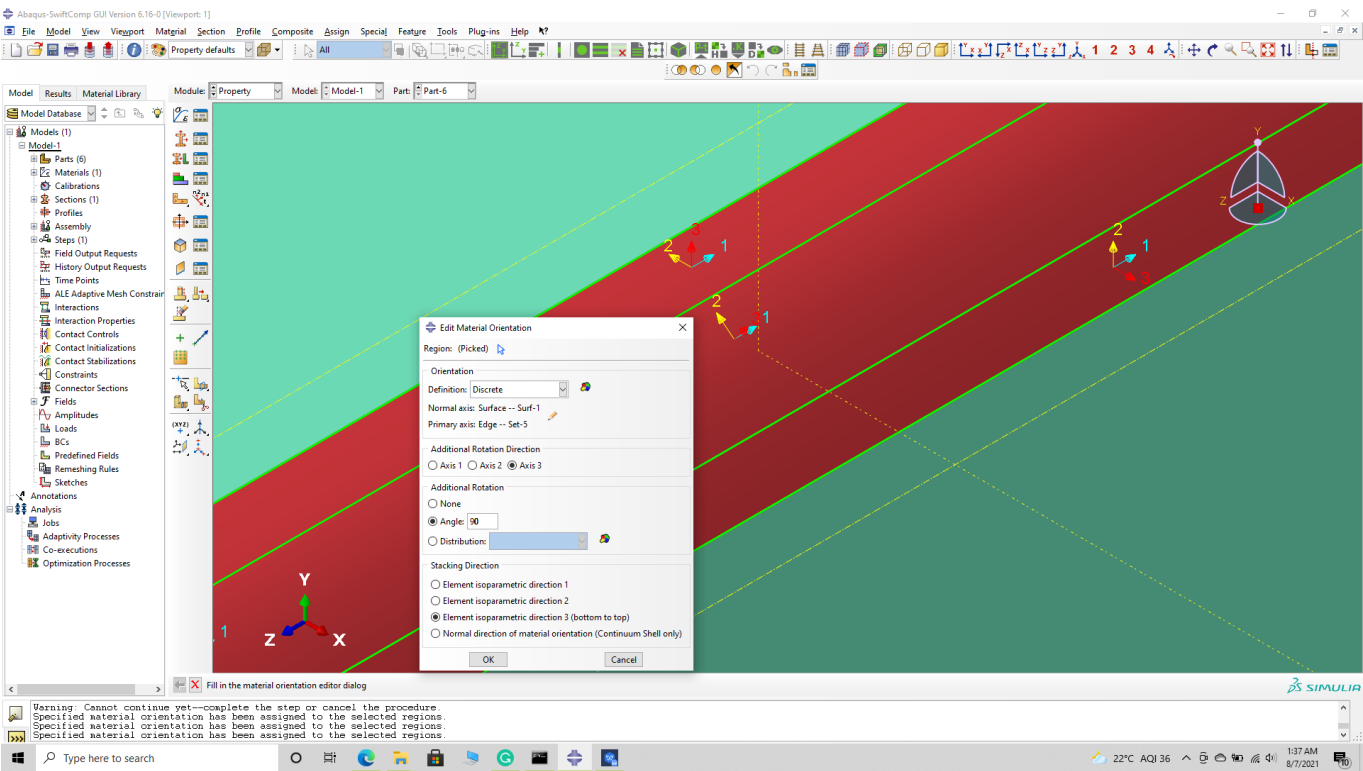


Assign material orientation

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL

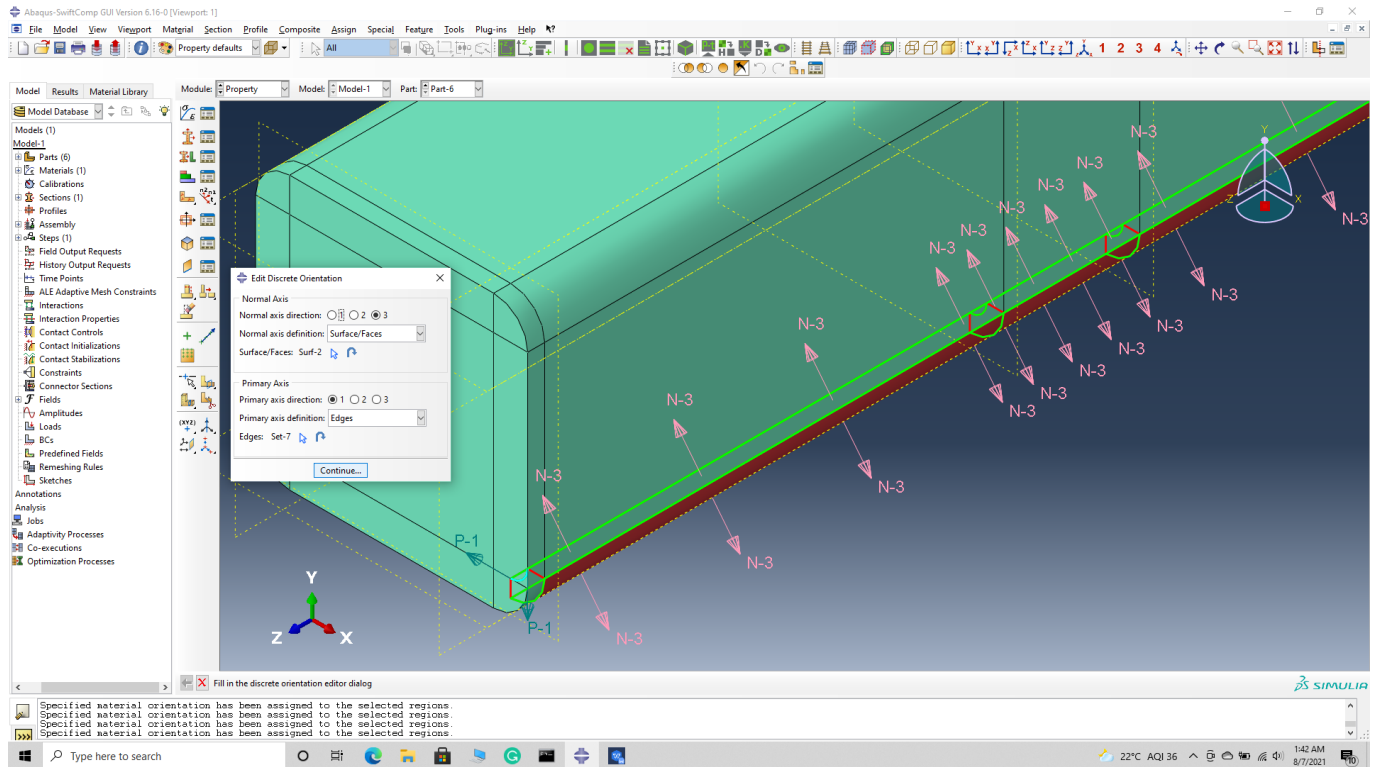


Assign material orientation



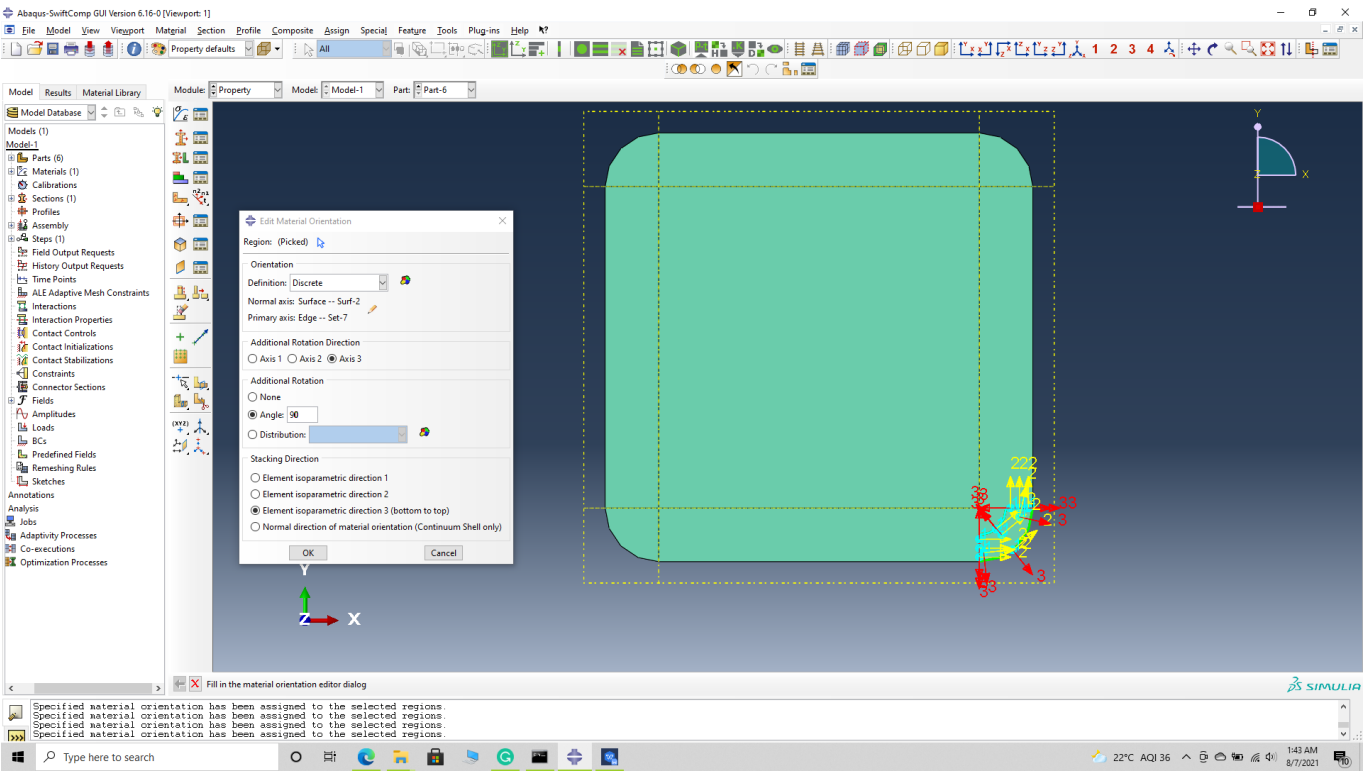
Assign material orientation

Step 3.8. Select the bottom right curved sections of the part and -> Done -> Select a CSYS (use default orientation or other method) -> set the discrete orientation as shown. -> Define -> Primary axis orientation -> choose edge and flip direction if needed to make the axis point towards a clockwise direction -> Choose the surfaces for the normal axis definition -> Continue -> OK -> Done –

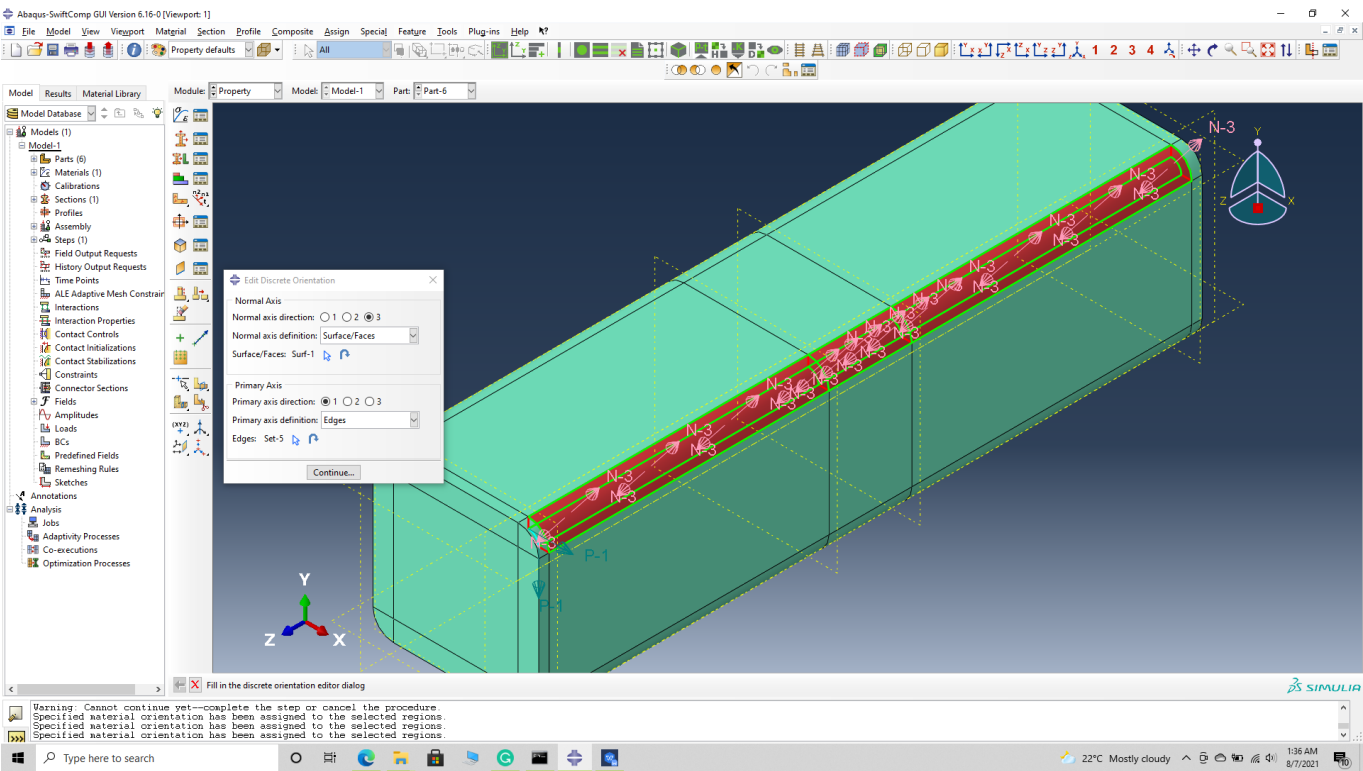


Assign material orientation

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL

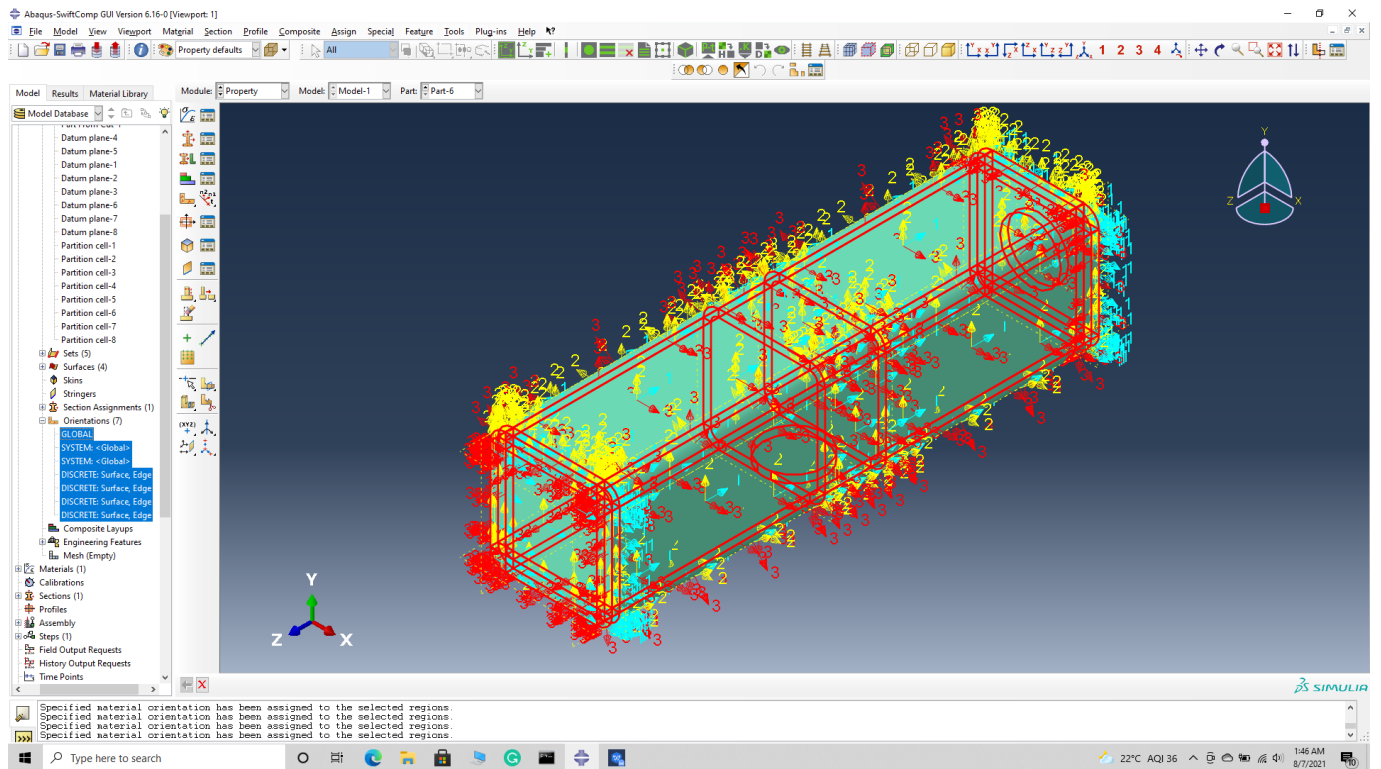


Assign material orientation



Assign material orientation

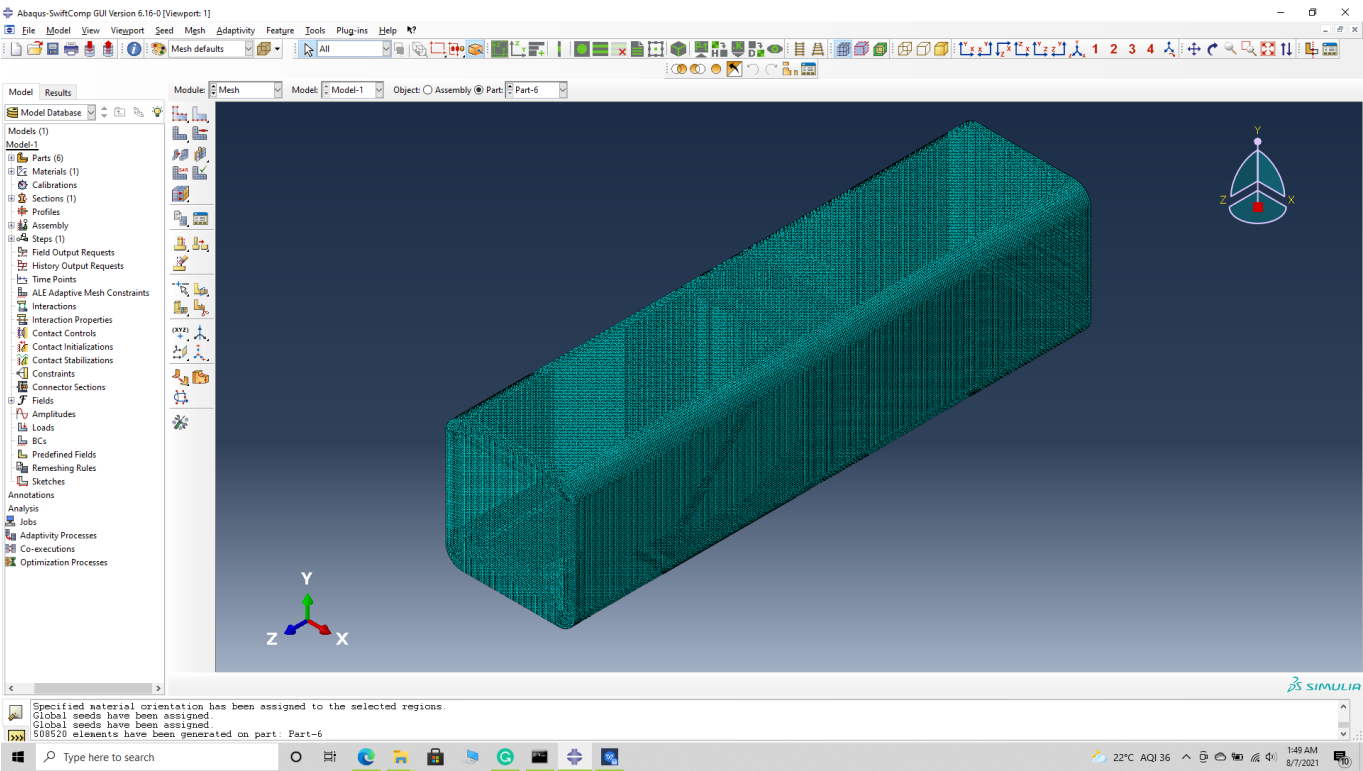
Step 3.9. The final orientation should look like this



Assign material orientation

Step 3.10. Mesh the part and continue with the traditional structural analysis with reduced mesh elements.

ELASTIC ANALYSIS OF A NACELLE MODELED USING ABAQUS AND ITS CONSTITUENT COMPOSITE PL



Assign material orientation