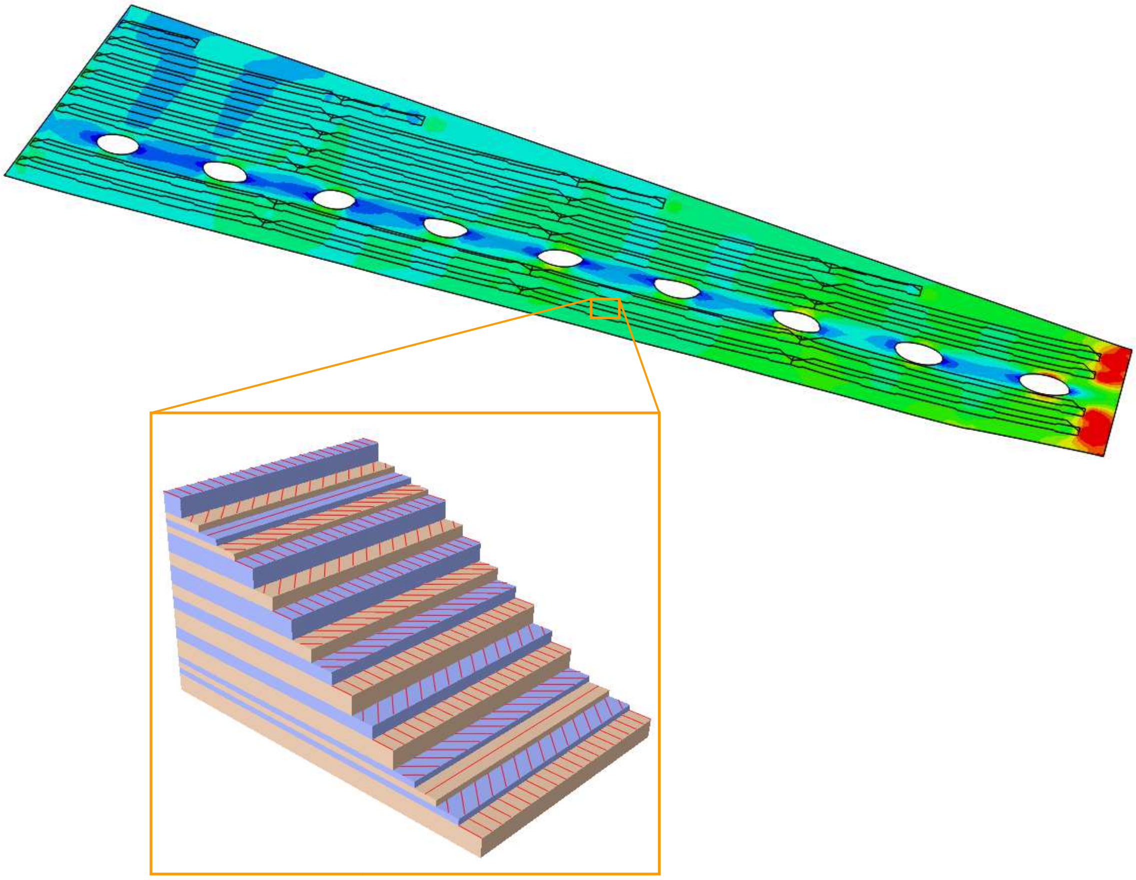
Tutorial 10: Composites, Modelling composite structures

Stephanie Miot



Strategic Simulation & Analysis Ltd
Southill Barn, Southill Business Park, Cornbury Park, Charlbury, Oxfordshire, OX7 3EW
T. 01608 811777 F. 01608811770 E.info@ssanalysis.co.uk W. www.ssanalysis.co.uk

1. Introduction

In this tutorial, you will modify a structural model of an aircraft wing to define the material properties and the stacking sequence of the laminated structures. You will then perform a static analysis and visualize the results of the simulation with Abaqus/Viewer.

When you complete this tutorial, you will be able to:

- Define the material properties of a composite ply
- Create and modify a composite lay-up
- Define the stacking direction
- Use the visualization module to create ply stack plots and contour plots on different plies

Preliminaries

The wing model is composed of:

- a cover
    - Dimensions: 19.5 m x 5 m, 6 mm thick
    - Lay-up: $(45_2, -45_2, 90_2, 0_2, 90_2, 0_2)_S$
    - Material: UD carbon / epoxy T300/M18

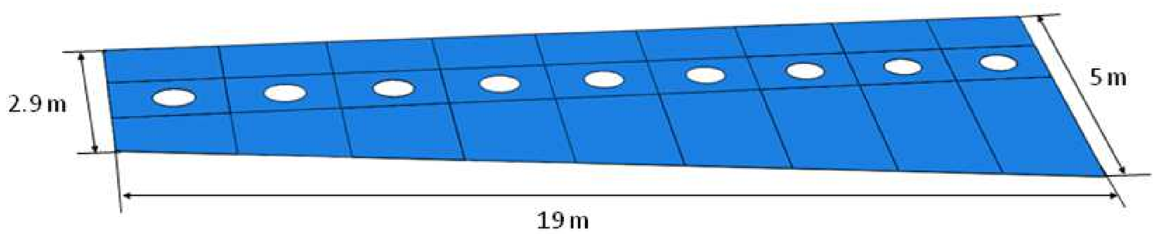


Figure 1: Geometry of the cover

- short stiffeners (SRO1)
  - Dimensions: 0.2 m x 0.14 m, 1.6 m long, 4 mm thick
  - Lay-up: $(0_3, 90, -45_2, 45_2)_S$
  - Material: UD carbon / epoxy T300/M18

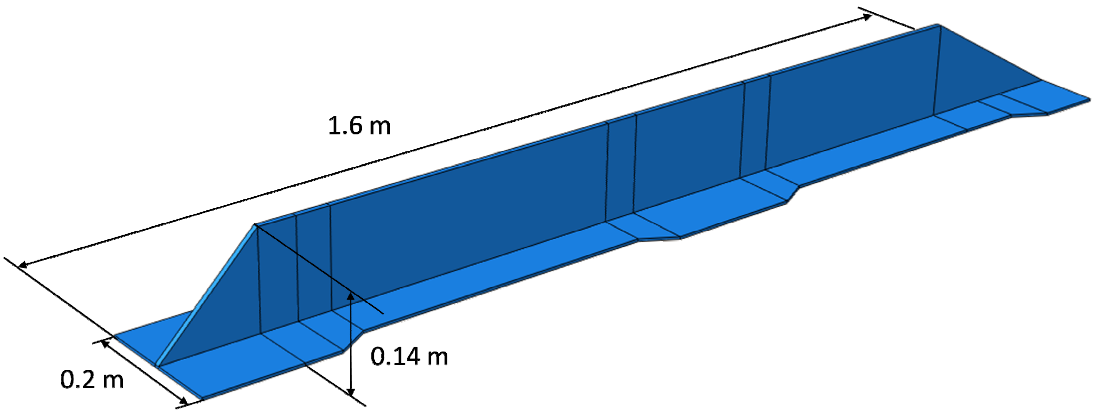


Figure 2: Geometry of the stiffener SRO1

- long stiffeners (SRO2)
  - Dimensions: 0.2 m x 0.2 m, 4.75 m long, 4 mm thick
  - Lay-up: $(0_3, -45, 90, 45, 0_3, -45_2, 0_3, 45_2)_S$
  - Material: UD carbon / epoxy T800/M18

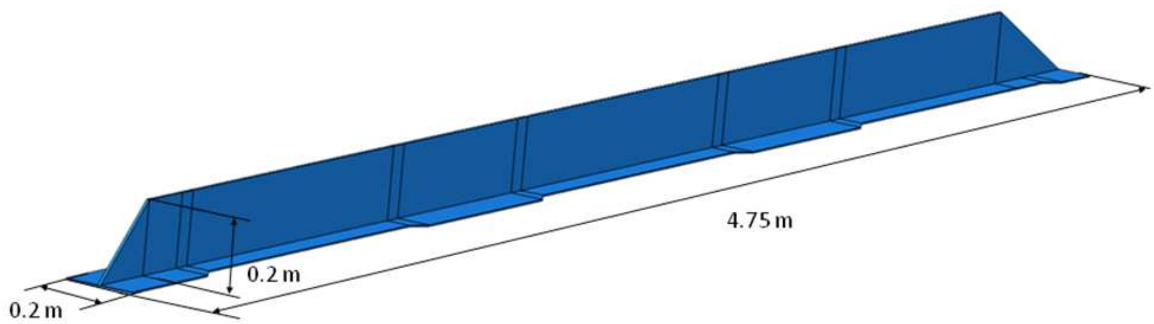


Figure 3: Geometry of the stiffener SRO2

2. Setting up the model

Open the model **Tutorial10.cae**.

This file contains the geometry of the different components, the assembly, the boundary conditions and the loading. The units are: m and Pa.

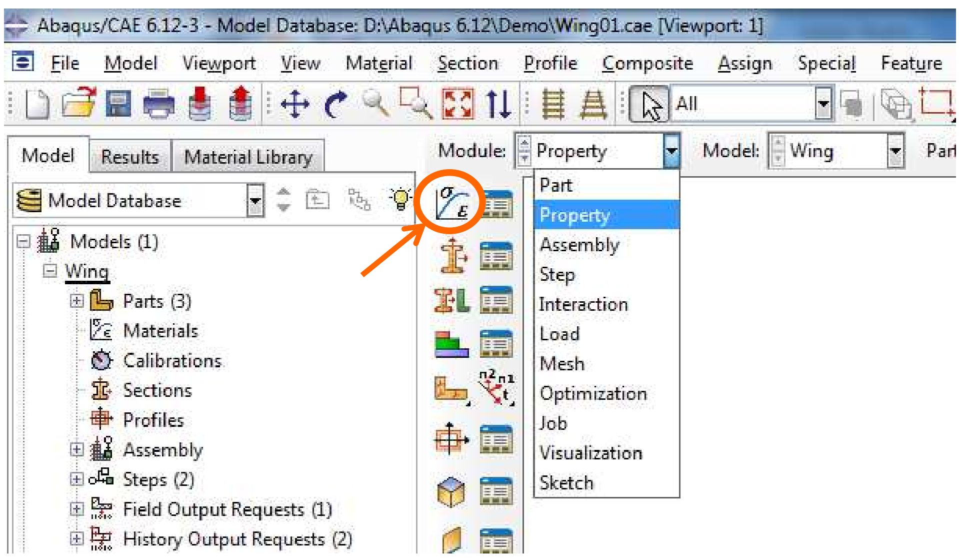
In this tutorial, you will define the material properties, the composite lay-up and the mesh for the cover and the stiffener SRO1. The lay-up and the mesh have been predefined for the stiffener SRO2. You will then run a static analysis and use the visualization module to post-process the results of the simulation.

3. Material and section properties

1. Define the orthotropic elastic behaviour of the UD ply with the following material properties:

- E_1 = 170 GPa
- E_2 = 9 GPa
- ν_{12} = 0.34
- $G_{12} = G_{13}$ = 4.8 GPa
- G_{23} = 4.5 GPa

a. Go into the **Property Module** and click the **Create Material** icon 



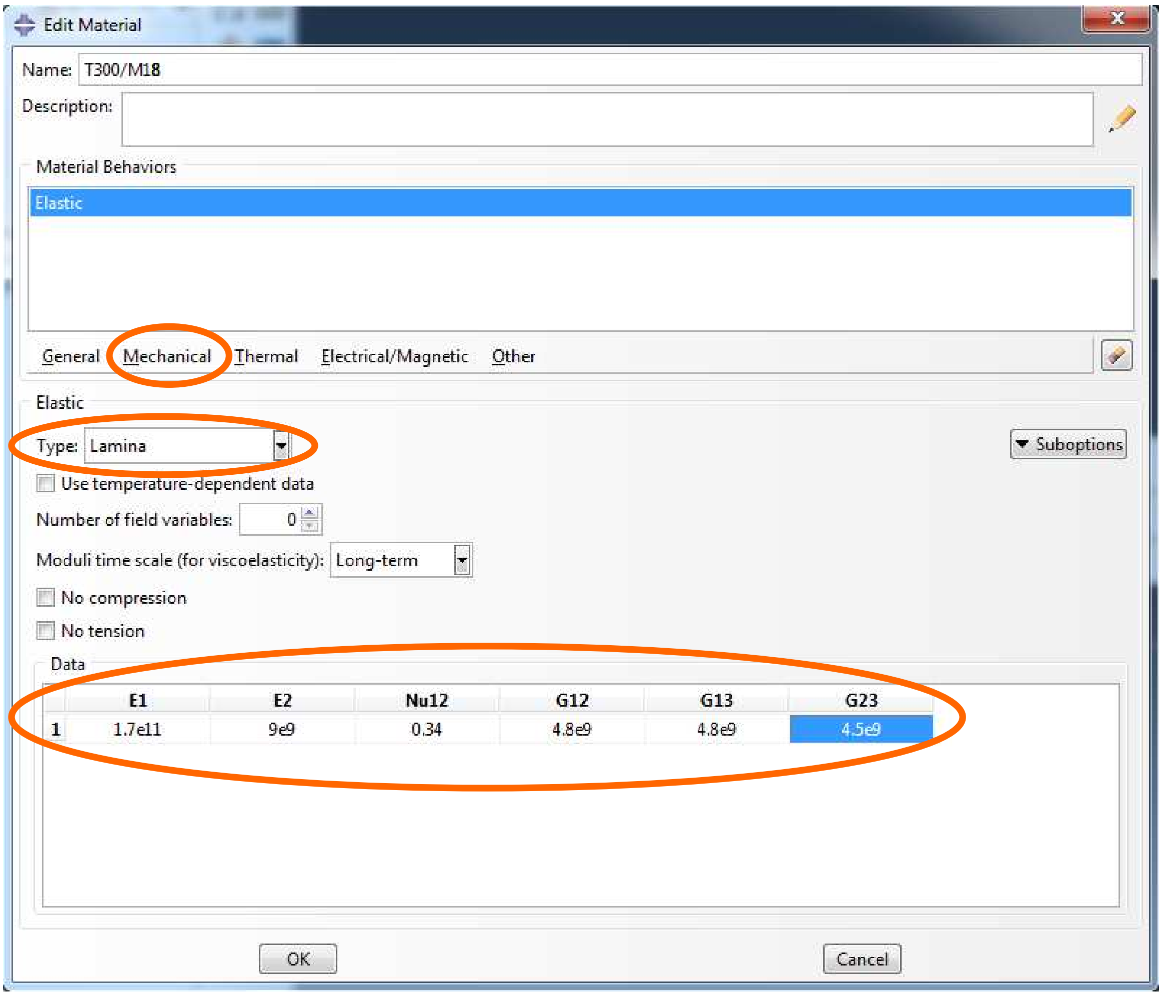
b. In the **Edit Material** dialog box, name the material *T300/M18*.

c. From the material editor's menu bar, select **Mechanical → Elasticity → Elastic**

d. Select **Type: Lamina**

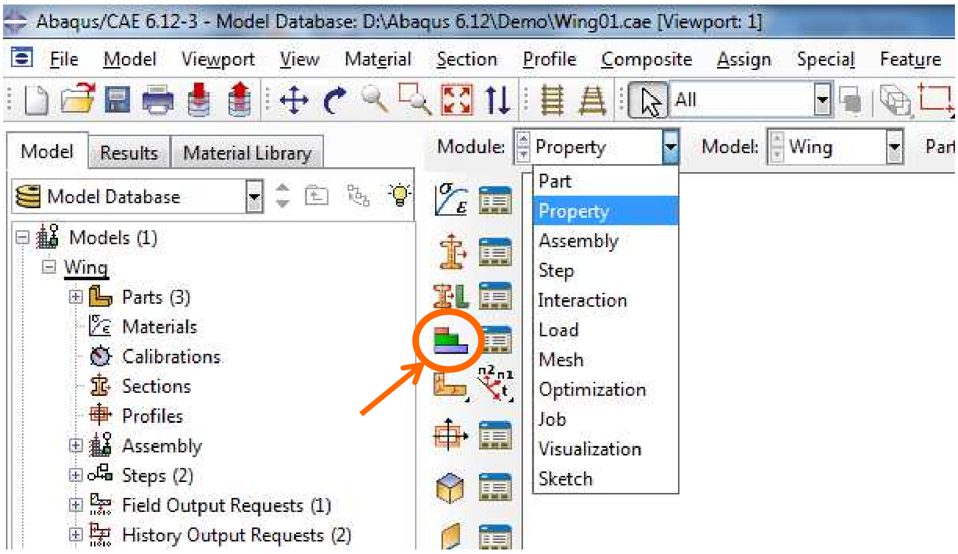
e. Enter the material data as defined above.

f. Click **OK** to exit the material editor.

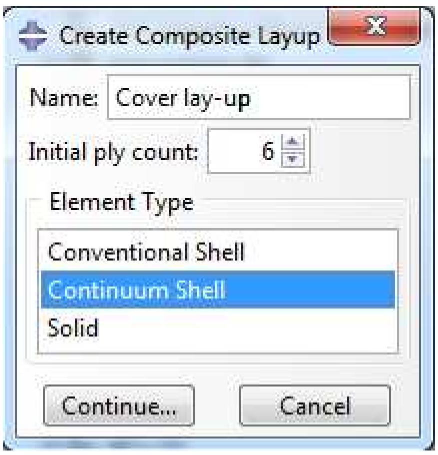


2. Define the lay-up of the cover. The cover is made of 24 plies. The stacking sequence is defined as: $(45_2, -45_2, 90_2, 0_2, 90_2, 0_2)_S$.

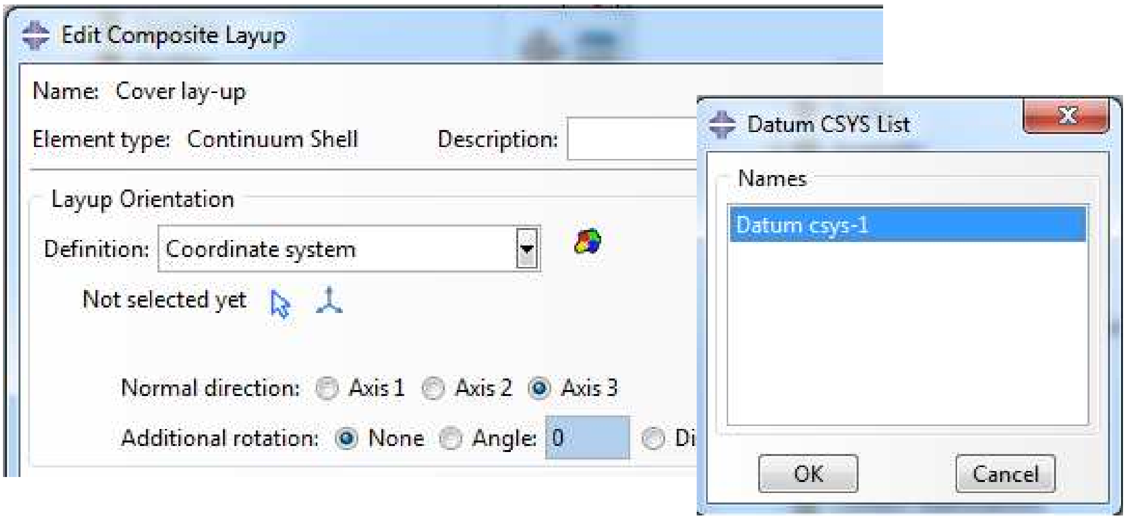
a. Click the **Create Composite Layup** icon 



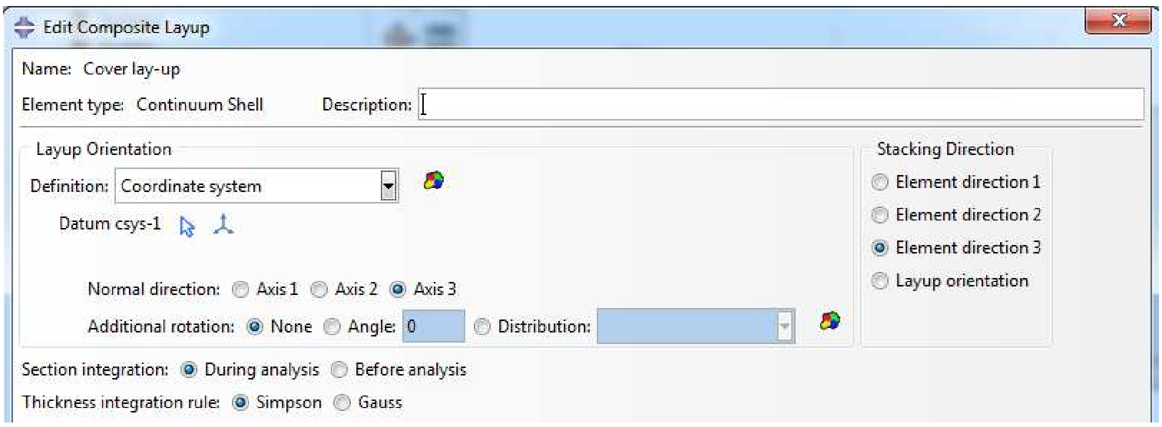
b. In the **Create Composite Layup** dialog box, name the lay-up *Cover lay-up*. Set the **Initial ply count** at 6 and select the **Element Type: Continuum Shell**. Click **Continue...**



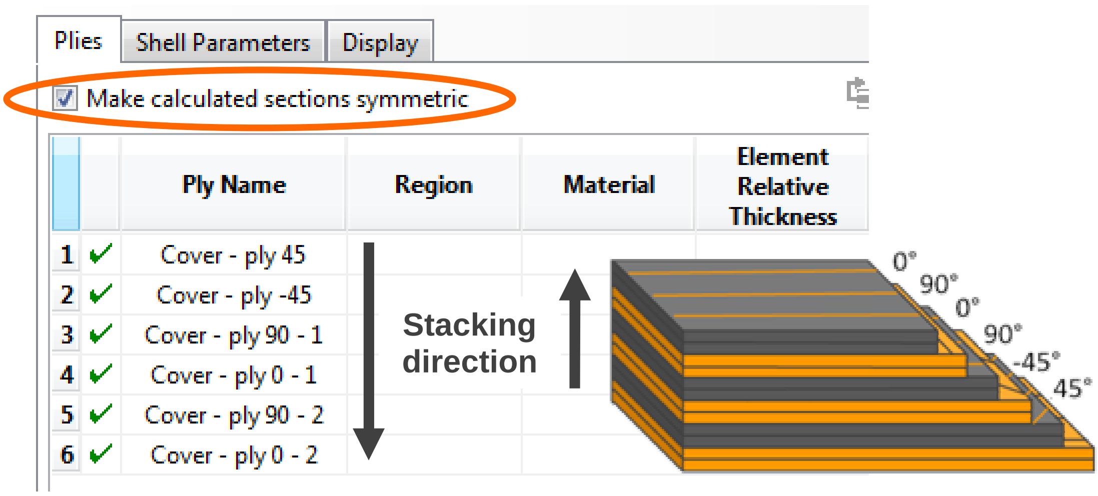
c. In the **Edit Composite Layup** dialog box, define the Layup orientation. Select **Definition: Coordinate system**. Click the **Select CSYS** icon to select an existing coordinate system. Click the **Datum CSYS List**... icon Datum CSYS List... in the bottom right corner. In the **Datum CSYS List**, select **Datum csys-1**. Click **OK** to go back to the composite lay-up editor.



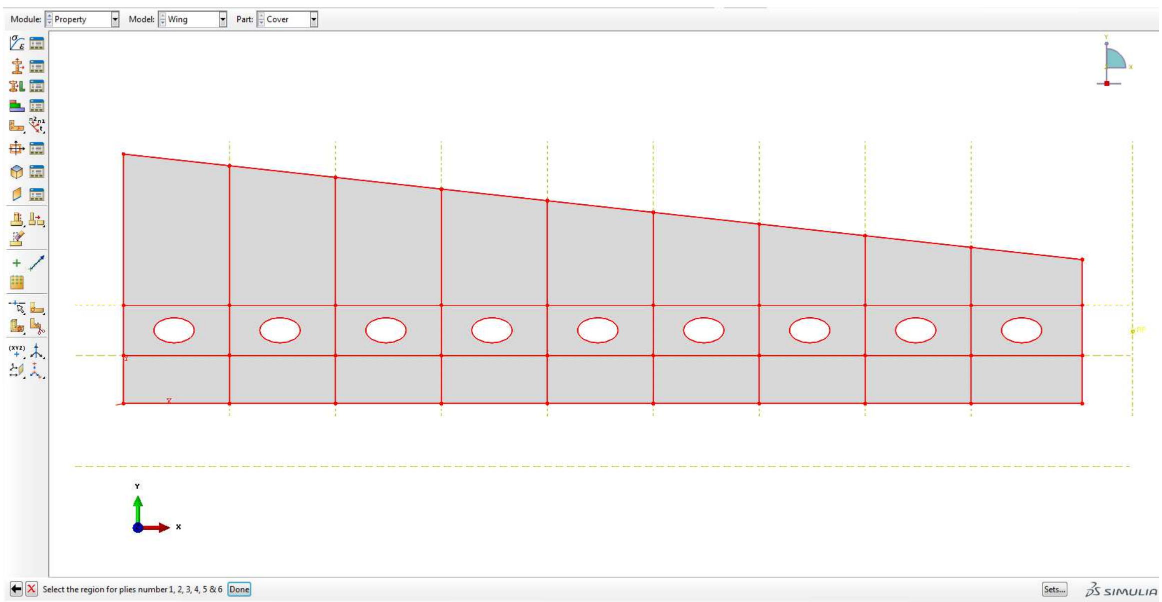
d. Accept the default selection for the **Normal direction: Axis 3** and the **Stacking Direction: Element direction 3**.



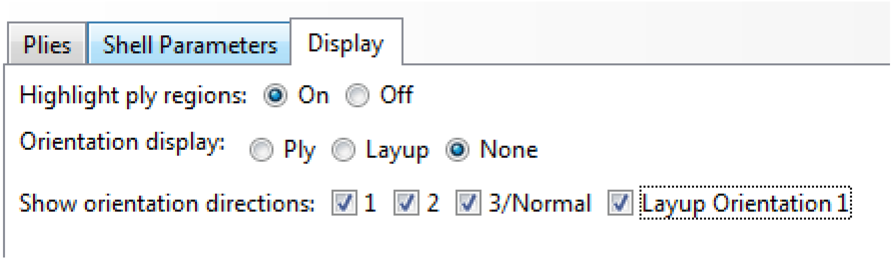
e. In the **Plies** tab, toggle on **Make calculated sections symmetric**. Rename the plies as shown below. Note that the first ply in the table is the first ply in the stacking sequence.



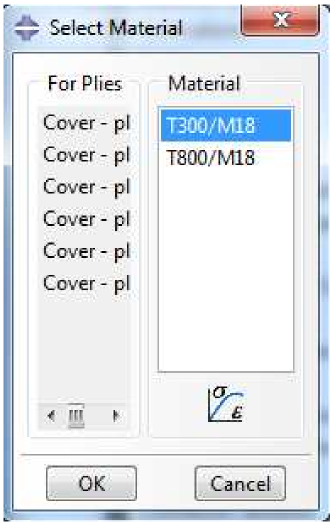
f. Double-click the **Region** button and select the entire part by dragging a rectangle around it. Click **mouse button 2** or **Done**.



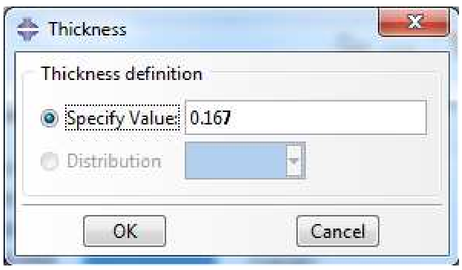
g. In the **Display** tab, select **Highlight ply regions: On** and **Orientation display: None**. You can check if the selection is correct.



h. In the **Plies** tab, double-click the **Material** button. Select the material T300/M18. Click **OK**.



i. Double-click the **Element Relative Thickness** button. Set the value at 0.167. Click **OK**.



j. In the column **Rotation Angle**, define the orientation for each ply as shown below.

k. Double-click the **Integration Points** button and set the number of integration point at 1.

Plies | Shell Parameters | Display

☑ Make calculated sections symmetric

		Ply Name	Region	Material	Element Relative Thickness	CSYS	Rotation Angle	Integration Points
1	✔	Cover - ply 45	(Picked)	T300/M18	0.167	<Layup>	45	1
2	✔	Cover - ply -45	(Picked)	T300/M18	0.167	<Layup>	-45	1
3	✔	Cover - ply 90 - 1	(Picked)	T300/M18	0.167	<Layup>	90	1
4	✔	Cover - ply 0 - 1	(Picked)	T300/M18	0.167	<Layup>	0	1
5	✔	Cover - ply 90 - 2	(Picked)	T300/M18	0.167	<Layup>	90	1
6	✔	Cover - ply 0 - 2	(Picked)	T300/M18	0.167	<Layup>	0	1

l. Use the options available in the **Display** tab to check the orientation of the lay-up. Then click **OK** to create the cover lay-up.

3. Define the lay-up of the stiffener SRO1.

The stiffener SRO1 is made of two co-cured stringers as shown in figure 4. Each stringer is made of 16 plies. The stacking sequence is $(0_3, 90, -45_2, 45_2)_S$. The co-cured stringers do not need to be modelled separately.

The stiffener is composed of two regions:

- seat, 16-ply region
- web, 32-ply region

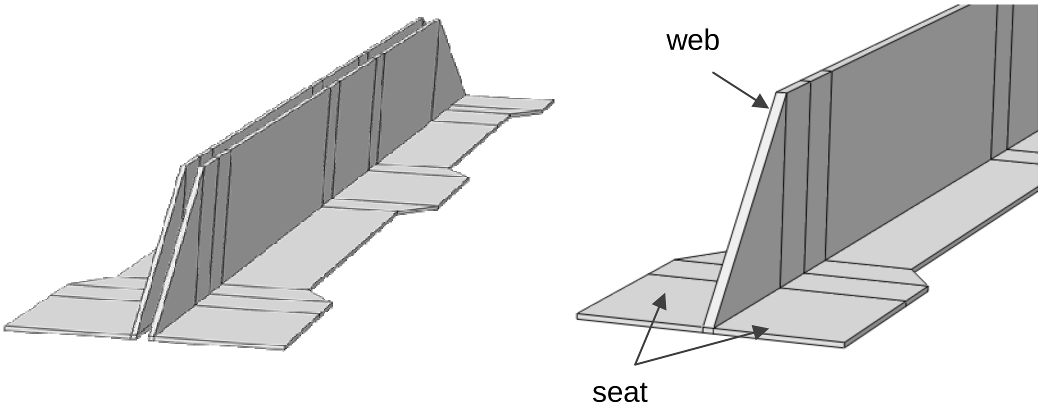


Figure 4: Stiffener SRO1

Two composite lay-ups will be created to define the two regions of the stiffener.

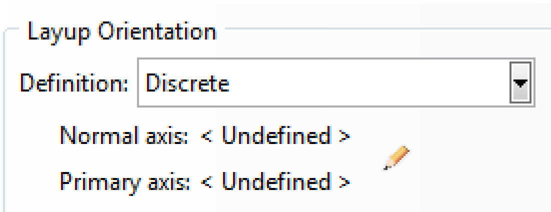
a. In the model tree, display the different parts by clicking the '+' symbol next to the **Part** icon 

   Double-click **SRO1**.

b. Click the **Create Composite Layup** icon 

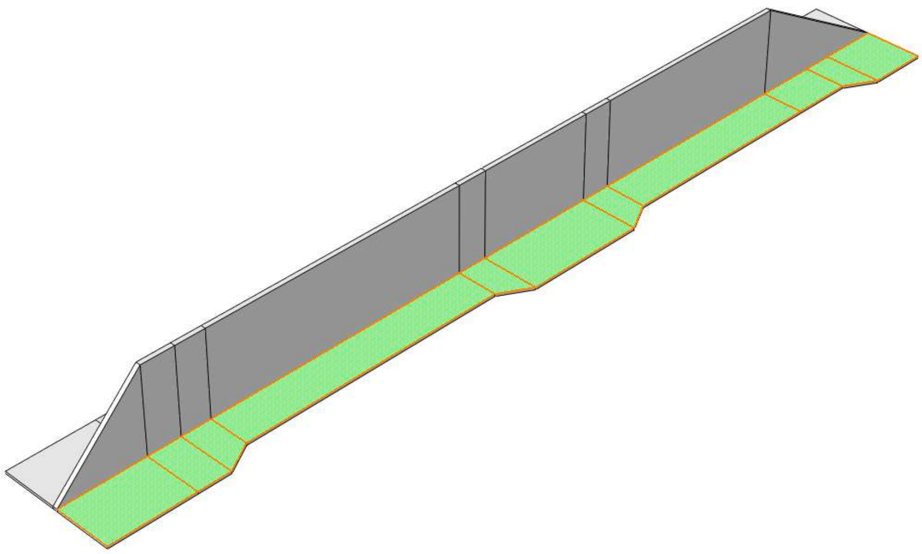
c. In the **Create Composite Layup** dialog box, name the lay-up *SRO1 lay-up*. Set the **Initial ply count** at 4 and select the **Element Type: Continuum Shell**. Click **Continue...**

d. In the **Edit Composite Layup** dialog box, define the Layup orientation. Select **Definition: Discrete**. Click the **Define...** icon to select the normal and the primary axes.



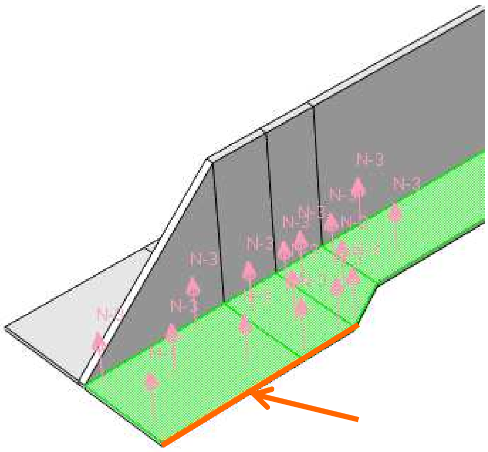
e. In the **Edit Discrete Orientation** dialog box, accept the default selections for the **Normal axis direction: 3** and the **Normal axis definition: Surface/Faces**. Click the **Edit Surface/Faces** icon and select the regions that define the normal axis of the material orientation. Choose selection **by angle** and toggle off **Create surface.**

f. Select the faces as shown below. Click **mouse button 2** or **Done**.

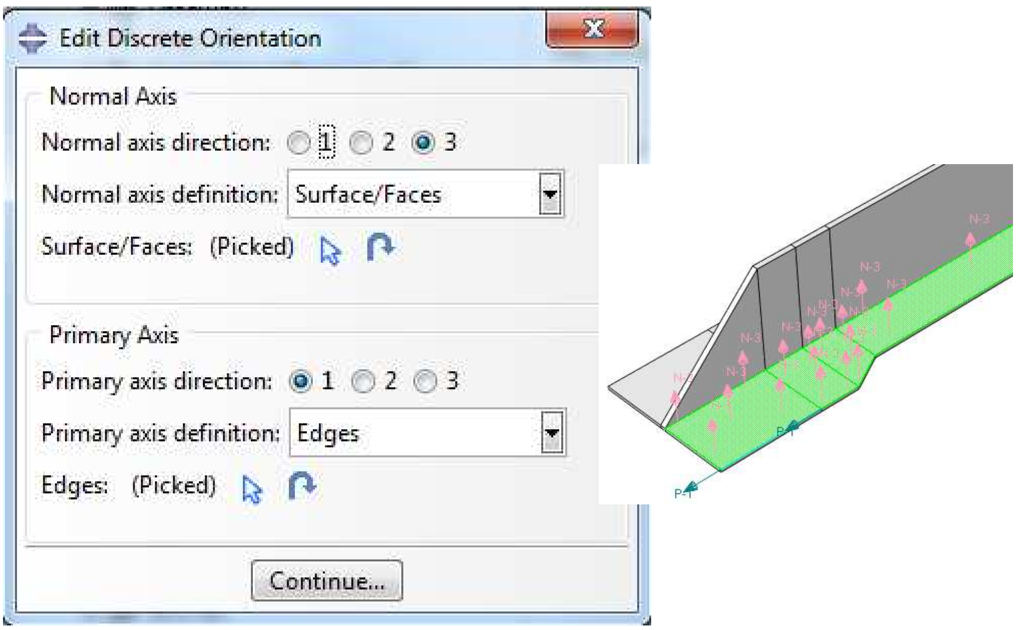


g. In the **Edit Discrete Orientation** dialog box, accept the default selections for the **Primary axis direction: 1** and the **Primary axis definition: Edges**. Click the **Edit Edges** icon and select the edges that define the primary axis of the material orientation.

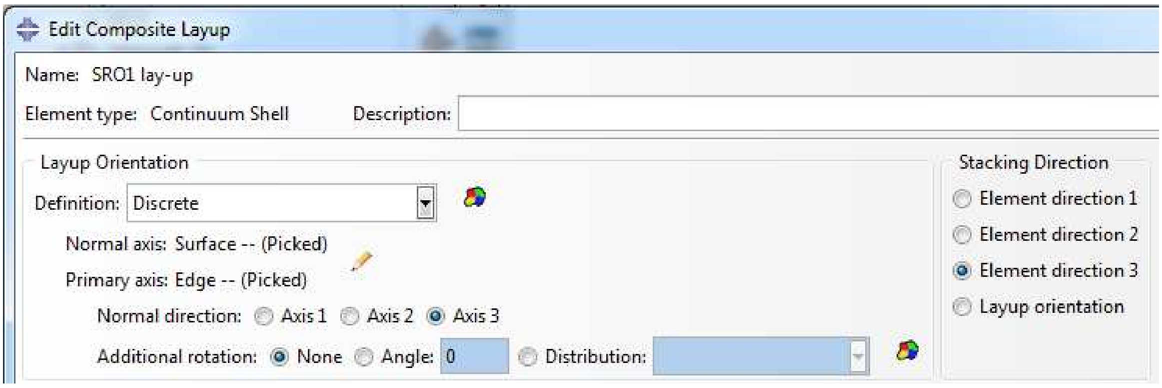
h. Select the edges as shown below. Click **mouse button 2** or **Done**.



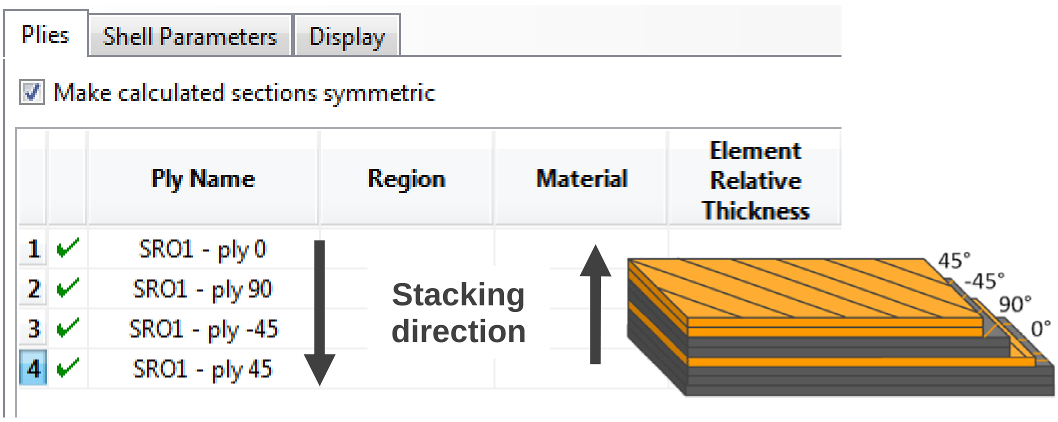
i. In the **Edit Discrete Orientation** dialog box, click **Continue...**



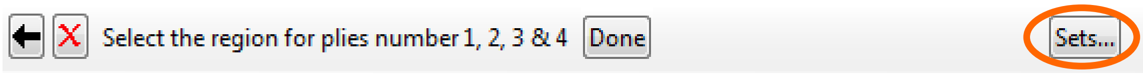
j. In the **Edit Composite Layup** dialog box, accept the default selections for the **Normal direction: 3** and the **Element direction: 3**.



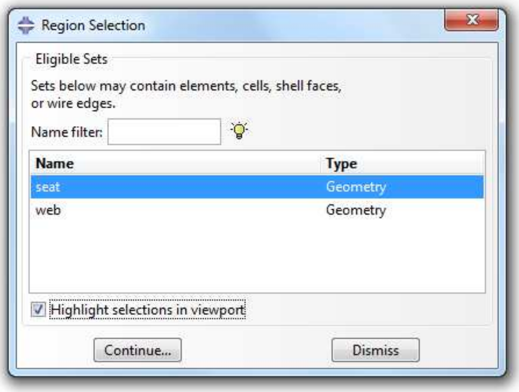
k. In the **Plies** tab, toggle on **Make calculated sections symmetric**. Rename the plies as shown below. Remember that the first ply in the table is the first ply in the stacking sequence.

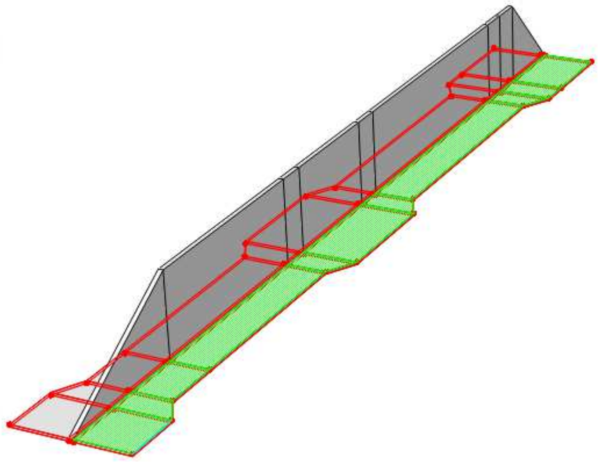


l. Double-click the **Region** button. Click **Sets**... and select the set: **seat**.



m. Toggle on **Highlight selections in viewport** and check if the selection is correct. Click **Continue...**





n. In the **Plies** tab, double-click the **Material** button. Select the material T300/M18. Click **OK**.

o. In the column **Element Relative Thickness**, define the relative thickness of each group of plies and in the column **Rotation Angle**, define the orientation for each ply as shown below.

Plies | Shell Parameters | Display

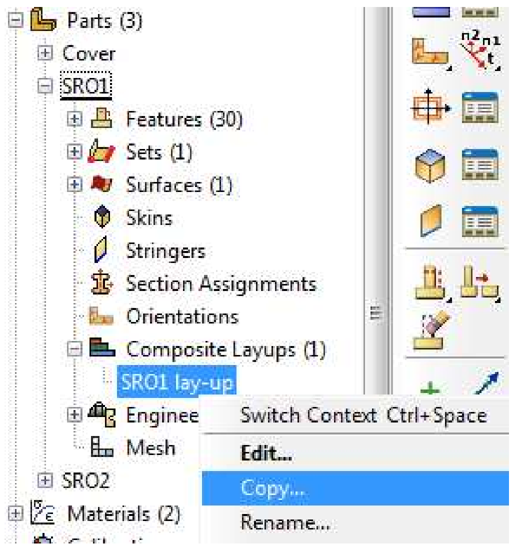
☑ Make calculated sections symmetric

		Ply Name	Region	Material	Element Relative Thickness	CSYS	Rotation Angle
1	✔	SRO1 - ply 0	(Picked)	T300/M18	0.375	<Layup>	0
2	✔	SRO1 - ply 90	(Picked)	T300/M18	0.125	<Layup>	90
3	✔	SRO1 - ply -45	(Picked)	T300/M18	0.25	<Layup>	-45
4	✔	SRO1 - ply 45	(Picked)	T300/M18	0.25	<Layup>	45

p. Double-click the **Integration Points** button and set the number of integration point at 1.

q. Use the options available in the **Display** tab to check the orientation of the different plies. Then click **OK** to create the lay-up.

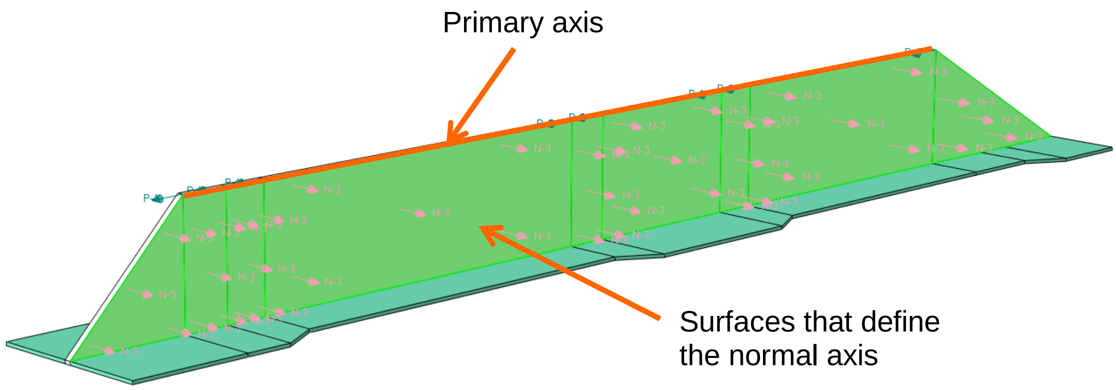
r. In the model tree, expand **SRO1** and **Composite Layups (1)**. Right-click **SRO1 lay-up** and select **Copy...** Rename the new lay-up: *SRO1 lay-up 2*. Click **OK**.



s. In the model tree. Double-click **SRO1 lay-up 2**. Modify the Layup Orientation. Click the **Define...** icon to modify the definition of the normal and primary axes.

t. In the **Edit Discrete Orientation** dialog box, click the **Edit Surface/Faces** icon and select the regions that define the normal axis of the material orientation as shown below (green). Click **mouse button 2** or **Done**.

u. Click the **Edit Edges** icon and select the edges that define the primary axis of the material orientation. Click **mouse button 2** or **Done**.



v. In the **Edit Discrete Orientation** dialog box, click **Continue...**

w. Double-click the **Region** button and select the set: **web** in the Region Selection dialog box. Click **Continue...**

x. Click **OK** to save the modifications made in the **Edit Composite Layup** dialog box.

Plies | Shell Parameters | Display

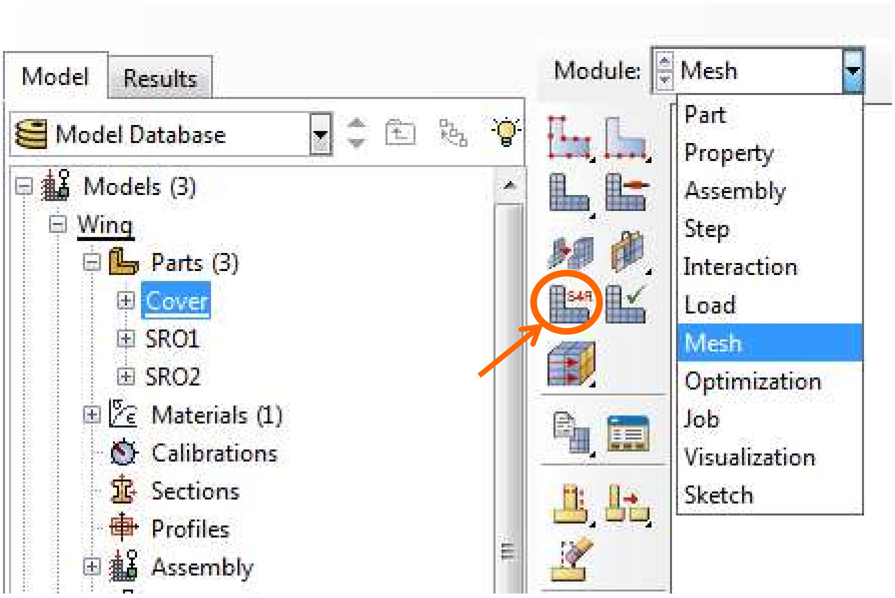
☑ Make calculated sections symmetric

		Ply Name	Region	Material	Element Relative Thickness	CSYS	Rotation Angle	Integration Points
1	✔	SRO1 - ply 0	(Picked)	T300/M18	0.375	<Layup>	0	1
2	✔	SRO1 - ply 90	(Picked)	T300/M18	0.125	<Layup>	90	1
3	✔	SRO1 - ply -45	(Picked)	T300/M18	0.25	<Layup>	-45	1
4	✔	SRO1 - ply 45	(Picked)	T300/M18	0.25	<Layup>	45	1

4. Mesh

1. Create the FE mesh for the cover. Note that the global seeds and mesh controls have been predefined.

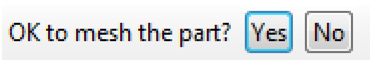
a. Go into the **Mesh Module**. In the model tree, double-click the part: **Cover**. Then click the **Assign Element Type** icon



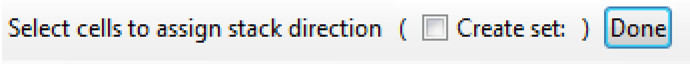
b. Drag a rectangle around the cover to select the entire part and click **mouse button 2** or **Done.**

c. In the **Element Type** dialog box, select **Family: Continuum Shell**. Accept the other default selections and click **OK**.

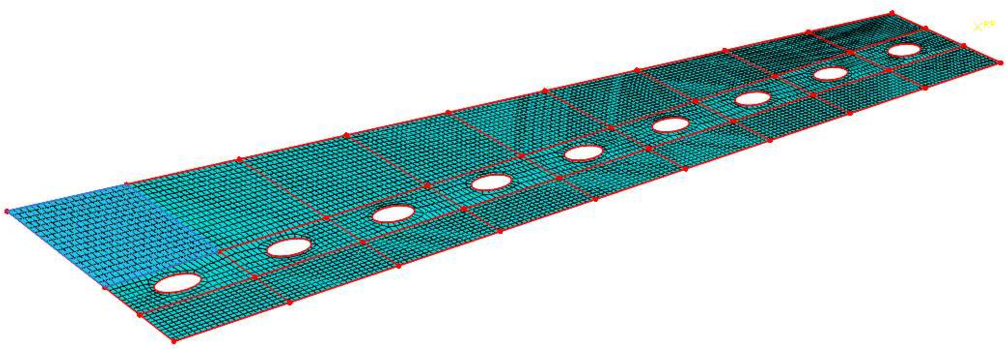
d. Click the **Mesh Part** icon and click **mouse button 2** or **Yes**.



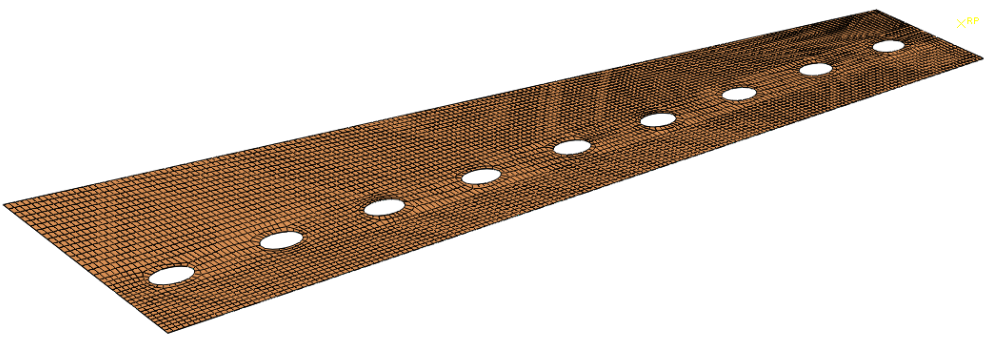
e. Click the **Assign Stack Direction** icon and select the entire part. Make sure that the **Create set** option is not active. Click **mouse button 2** or **Done.**



f. Click a face on top of the cover as shown below to define the reference orientation.



g. Click **mouse button 2** or **Yes** to assign the reference orientation. Top element faces are coloured brown.

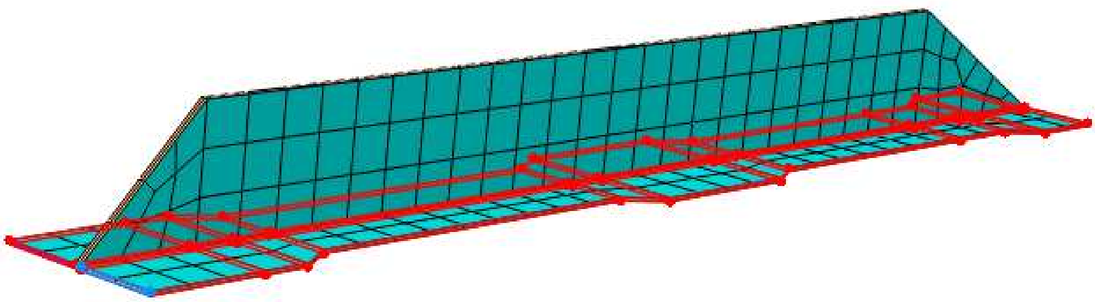


2. Create the FE mesh for the stiffener SRO1. Note that the global seeds and mesh controls have been predefined.

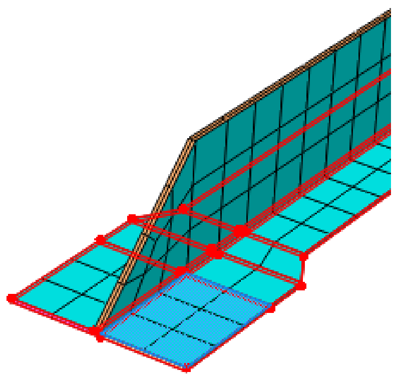
a. In the model tree, double-click the part: **SRO1**. Then click the **Assign Element Type** icon . Select the entire part click **mouse button 2** or **Yes**. In the **Element Type** dialog box, select **Family: Continuum Shell** and click **OK**.

b. Click the **Mesh Part** icon and click **mouse button 2** or **Yes**.

c. Click the **Assign Stack Direction** icon and click the **Sets...** button. Select the set: **seat** and click **Continue...**

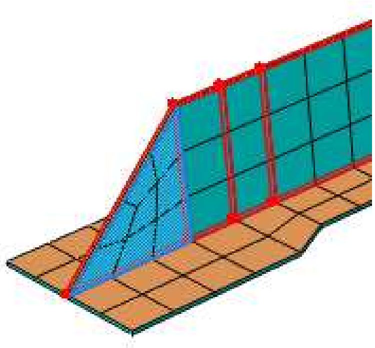


d. Click a face on top of the cells as shown below to define the reference orientation.



e. Click **mouse button 2** or **Yes** to assign the reference orientation.

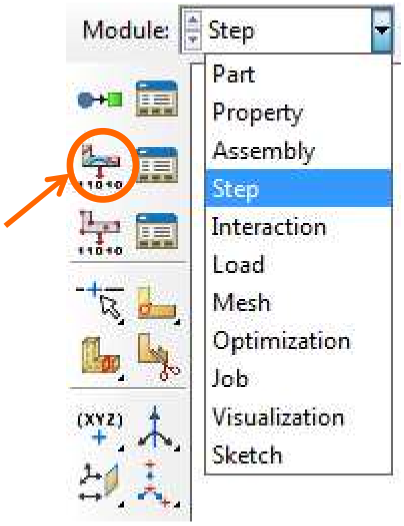
f. In the **Region Selection** dialog box, select the set: **web**. Click **Continue...** Click a top face as shown below and click **mouse button 2** or **Yes** to assign the reference orientation.



5. Static analysis and post-processing

1. Create new output requests. The preselected default output does not include every section point. To visualize the stresses in each ply in the Visualization module, you will write additional field output to the output database file. Note that the output requests have been predefined for the stiffeners.

a. Go into the **Step Module**. Click the **Create Field Output** icon 



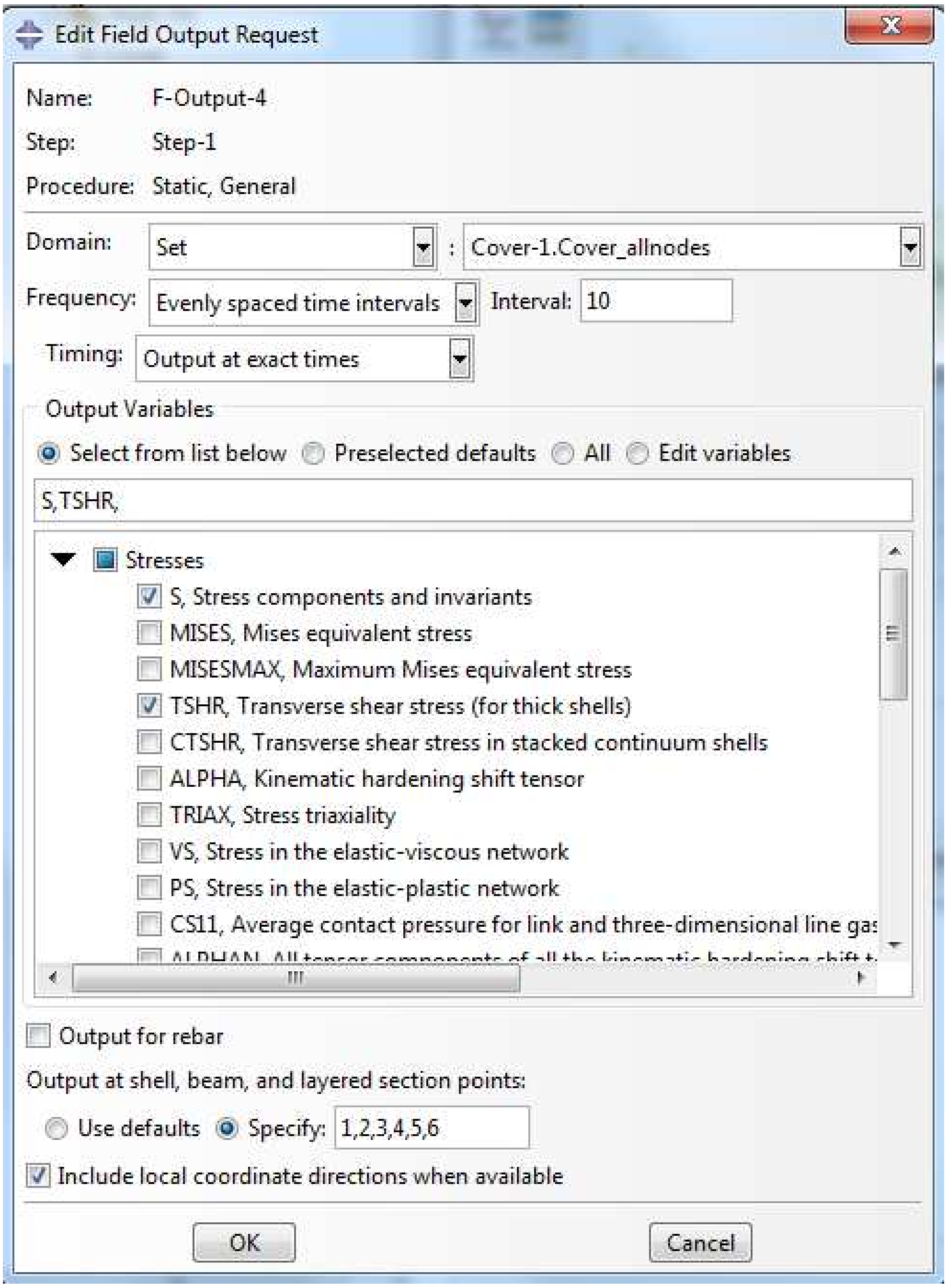
b. In the **Create Field** dialog box, click **Continue...**

c. In the **Edit Field Output Request** dialog box, select **Domain: Set: Cover-1.Cover_allnodes**. Then select **Frequency: Evenly spaced time intervals** and set **Interval: 10**.

d. In the **Output Variables** list, expand the **Stresses** list and toggle on the variables **S** and **TSHR**.

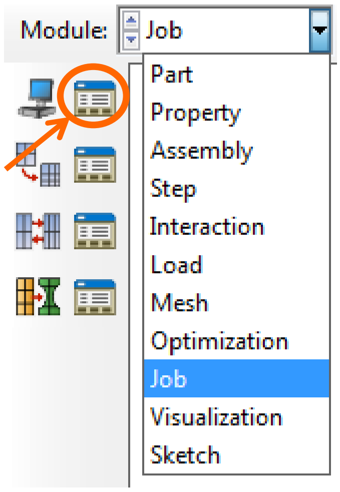
e. **Specify** the output at layered section points: 1, 2, 3, 4, 5, 6.

f. Click **OK**.

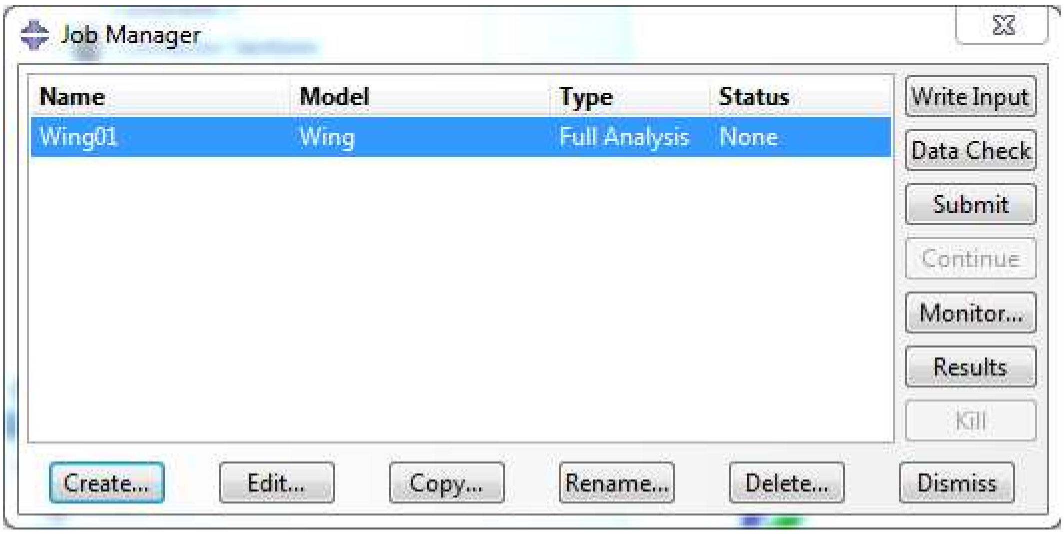

Edit Field Output Request
Name: F-Output-4
Step: Step-1
Procedure: Static, General
Domain: Set : Cover-1.Cover_allnodes
Frequency: Evenly spaced time intervals Interval: 10
Timing: Output at exact times
Output Variables
Select from list below Preselected defaults All Edit variables
S,TSHR,
Stresses
S, Stress components and invariants
MISES, Mises equivalent stress
MISESMAX, Maximum Mises equivalent stress
TSHR, Transverse shear stress (for thick shells)
CTSHR, Transverse shear stress in stacked continuum shells
ALPHA, Kinematic hardening shift tensor
TRIAX, Stress triaxiality
VS, Stress in the elastic-viscous network
PS, Stress in the elastic-plastic network
CS11, Average contact pressure for link and three-dimensional line gas
Output for rebar
Output at shell, beam, and layered section points:
Use defaults Specify: 1,2,3,4,5,6
Include local coordinate directions when available
OK
Cancel

2. Run the job. Note that the job has been created.

g. Go into the **Job Module**. Click the **Job Manager** icon.



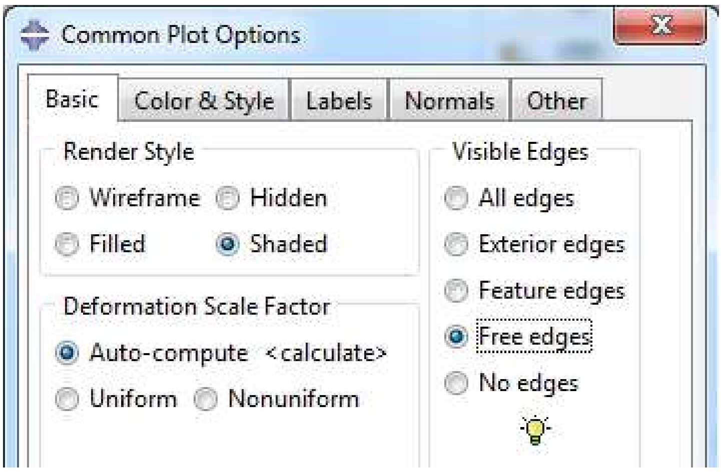
h. In the **Job Manager** dialog box, click **Submit** to run the **Job: Wing01**.



3. Analyse the results of the simulation.

a. When the job submission has been completed, in the **Job Manager** dialog box, click **Results**.
You can also open *Wing01.odb* in the **Visualization Module**.

b. Click the **Common Options** icon and select **Visible Edges: Free edges**. Click **OK**.



c. Click the **Create Display Group** icon or click **Tools → Display Group → Create...**

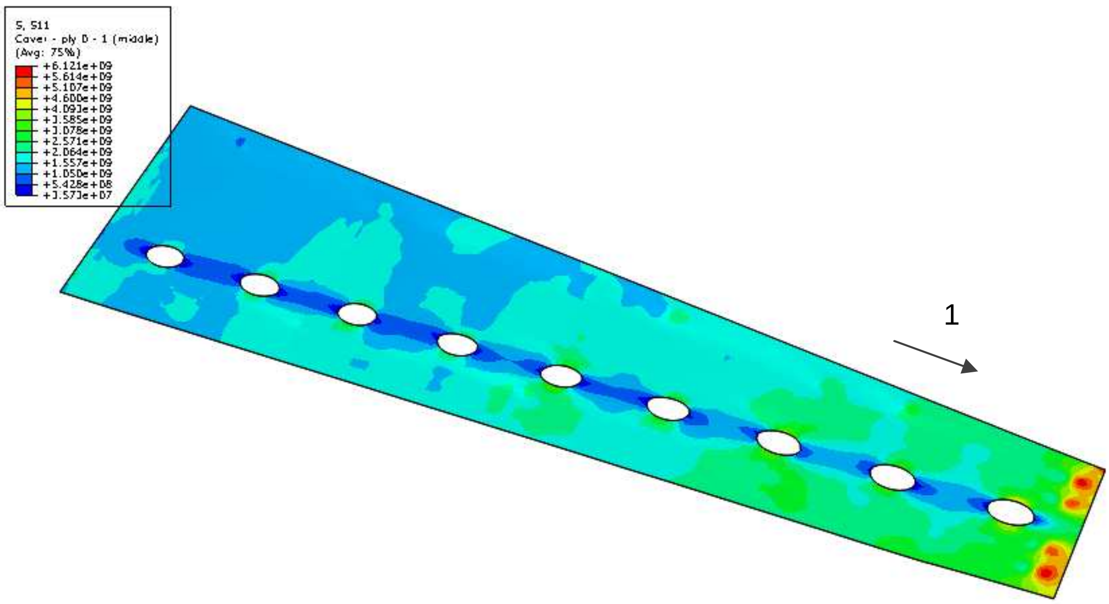
d. In the **Create Display Group** dialog box, select **Part instances: COVER-1**. Click the **Replace** icon then click **Dismiss**.

e. Click the **Plot Contours on Deformed Shape icon**

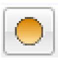
f. Click the **Field Output Dialog** icon or click **Result → Field Output**.

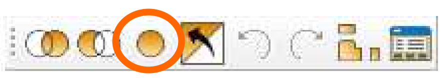
g. In the **Field Output** dialog box, in the **Primary Variable** tab, select the **Output Variable: S** and the **Component: S11**. Then click **Section Points...**

h. In the **Section Points** dialog box, click **Selection method: Plies** and select the ply **Cover - ply 0 – 1**. Click **Apply**.
You can then visualise the stress in the fibre direction in the first 0° ply of the cover.



i. Select other plies to create additional contour plots.

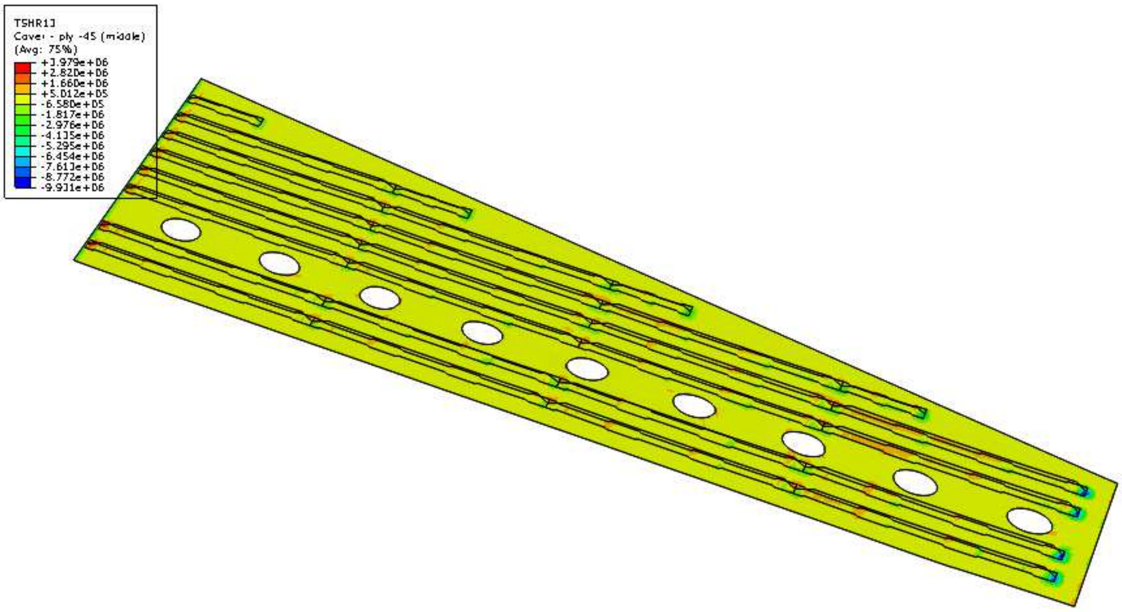
j. Click the **Replace All** icon  .



k. In the **Field Output** dialog box, in the **Primary Variable** tab, select the **Output Variable: TSHR13.** Click **Apply** then click **Section Points...**

l. In the **Section Points** dialog box, click **Selection method: Plies** and select the ply **Cover - ply -45**. Click **Apply**.

You can then visualise the transverse shear stress in the -45° ply of the cover.



m. Click **Tools → Query**. In the **Query** dialog box, select **Visualization Module Queries: Ply stack plot** then select the cover to display the composite lay-up.