

TKT4142 Finite Element Methods in Structural Engineering

WORKSHOP CASE STUDY 3

General tips


- Use consistent units.
- Set your Work Directory (**File – Set Work Directory** in Abaqus CAE) to a specific folder.

1.0 Creating the Model

Open Abaqus CAE 2022 and create a Model Database with a **Standard/Explicit Model**. Rename **Model-1** to e.g., **Axisymm_Spherical**. When we want to perform our mesh study, we can simply copy this model, change the name, and change the mesh seeding. We can also copy the model when we want to make the flat top housing and change the sketch in the Part module.

1.1 Creating the Part


Double-click **Parts** under **Axisymm_Spherical**. Name your part. This will be an **Axisymmetric, Deformable, Shell** part with an **Approximate size 600**. Now we are ready to start drawing the part.

We are only sketching the cross-section of the housing. It is important that we have an axis of revolution and that this axis is placed on the y-axis in the sketch. There we can insert vertical a construction line on this axis (). All of the sketch entities must lie to the right of this axis and must coincide with the axis at the top of the housing.

Try to assign the same dimensions as in the assignment sketch. Then use constraints to make relationships between the lines. For instance, making lines parallel or tangent.

After sketching up the cross-section, click **Done**.

In order to make a more structured mesh, we can partition the part close to the transition between the 4 mm thickness and 10 mm thickness. To partition the face click **Partition**

Face: Sketch (). Select the face you want to partition. Draw a partition line, see Figure 1 1, and click **Done**.

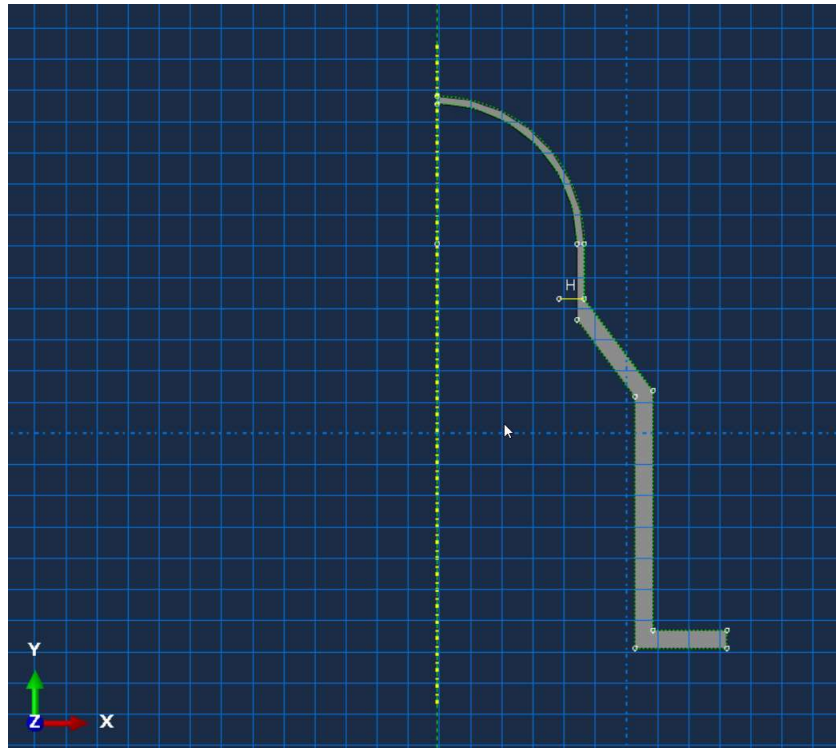


Figure 1: Partitioning face.

To structure the mesh even further, we can make more partitions. These partitions can also be utilized to assign the number of elements over the thickness for the 10 mm section.

As we want to apply a radial boundary condition to the inner edge of the housing, we can partition this edge as well. Choose **Create Datum Point: Offset From Point** (↗️) Pick the point at the bottom of the inner edge and type in **30** in the y-direction, see Figure 2 2. It is also possible to create a face partition 30 mm over the bottom edge. The edge to radially constrain will automatically be partitioned out.

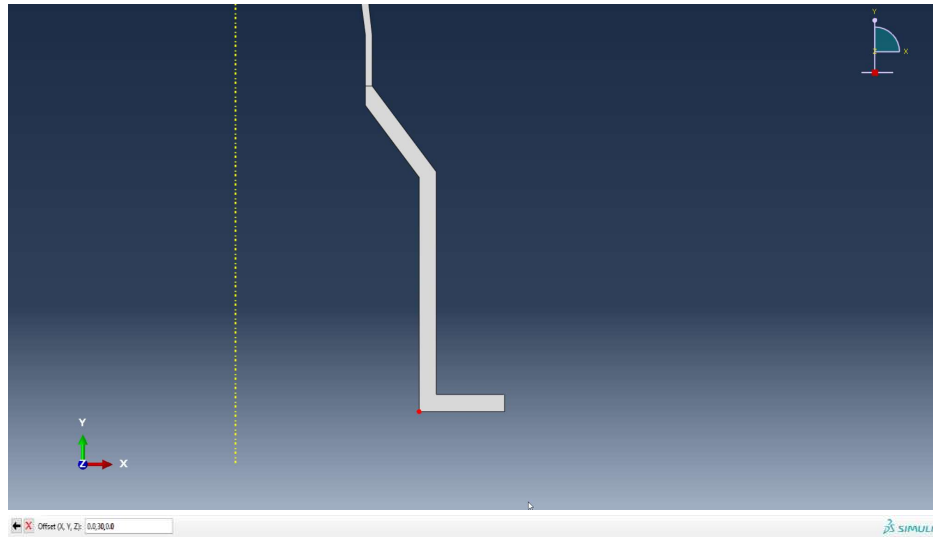


Figure 2: Offsetting from point.

Then click **Partition Edge: Select Midpoint/Datum Point** (with a red dot icon) and choose the datum point just created.

As we want to extract the reaction forces from the bottom flange edge, it is a good idea to create a set for these nodes. Double-click **Sets** and give it an appropriate name. Choose **Type Geometry** and pick the bottom flange edge, see Figure 3.

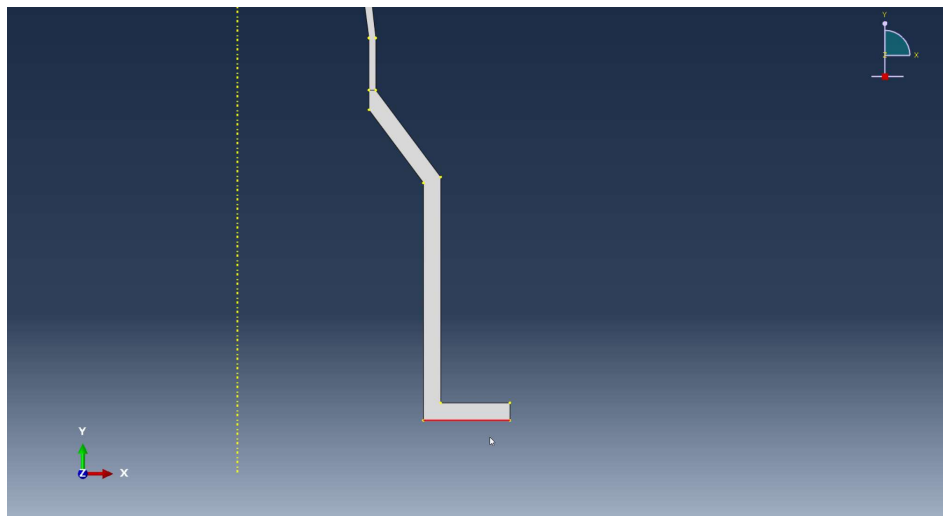


Figure 3: Bottom flange edge.

Use **Partition Face: Sketch** to partition the **Part** according to Figure 4.

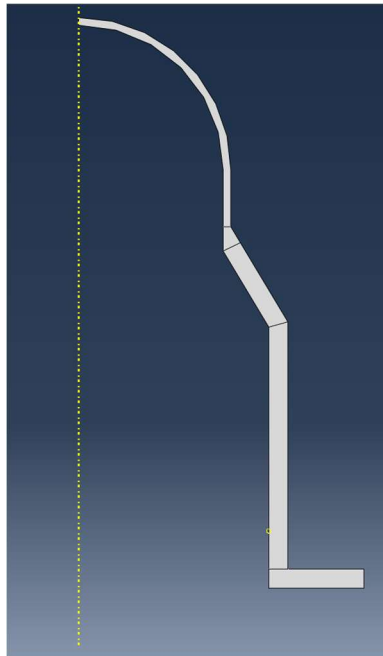


Figure 4: Partitioned part.

1.2 Assigning Properties (Material and Section)

Double-click **Materials** to create a material. Name the material e.g. **Titanium Ti-6Al-4V**. Assign the material behavior from Table 1. Remember to be consistent with the units. Density could be written in tonnes/mm³ in Abaqus to keep units consistent with MPa.

Table 1: Material properties for Ti-6Al-4V

Density [kg/m ³]	Young's Modulus [MPa]	Poisson's ratio [-]
4430	113800	0.342

Double-click **Sections** to create a **Section**. Give the section an appropriate name. This section is **Homogeneous** and **Solid**. Pick the titanium **material** you defined earlier. Now we can assign the section to the part. Double-click **Section Assignments** under **Part**. Pick the geometry in the drawing area and click **Done**. Pick the section we just created.

1.3 Create an Assembly

Double-click **Instances** under **Assembly**. Pick **Create Instance from Parts**, choose the part and pick **Instance Type Independent**.

1.4 Defining Step and Output Data

Double-click **Step** and create a **Static, General** step called **Loadstep** after **Initial**. **Time period** can be set to **1**. Under **Incrementation** we don't need to change the initial increment size from **1** as this is a static analysis and incrementation is not needed.

Double-click **History Output Requests** and give it an appropriate name. Change the **Domain** from **Whole Model** to **Set** and choose your set for the bottom edge of the flange. Select reaction forces **RF2**, see Figure 5.

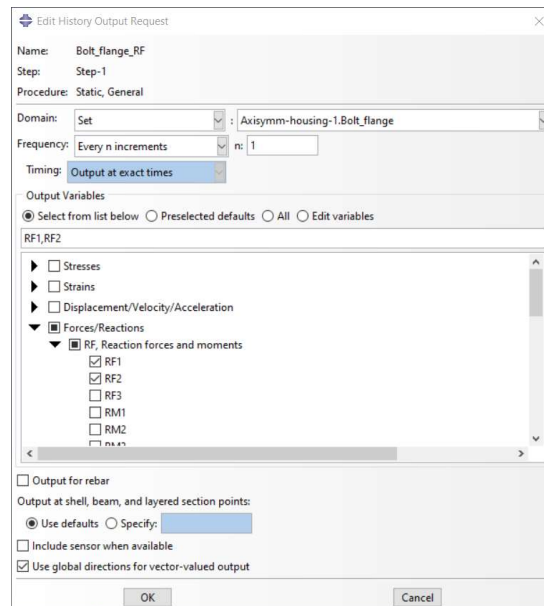


Figure 5: Creating History Output.

1.5 Applying Load and Boundary Conditions to the Model

The load will be applied as a pressure load. Double-click **Loads**. Create a load and name it. Category **Mechanical** and type **Pressure**. Pick the edges of the housing exposed to water, see Figure 6. Hold Shift to choose multiple edges. Remember to use consistent units. Distribution **Uniform**, appropriate **Magnitude** and **Amplitude (Ramp)**.

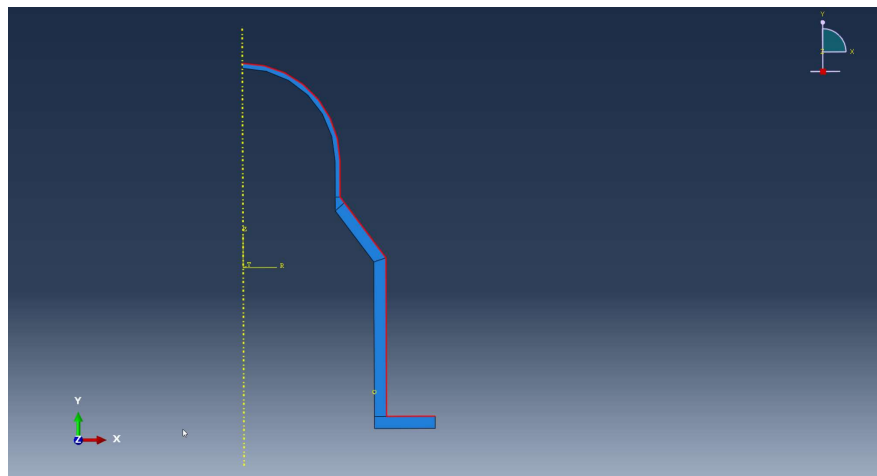


Figure 6: Edges exposed to water.

We are now going to apply boundary conditions. There are two BCs to be applied. Double-click **BCs**. Choose an appropriate name. **Step: Initial, Category: Mechanical, Types for Selected Step: Displacement/Rotation**. Pick the bottom edge and constrain **U1** (radially) and **U2** (axially), see Figure 7. Create a new BC and name it. Pick the inner edge that we partitioned out earlier and constrain (**U1**), see Figure 8.

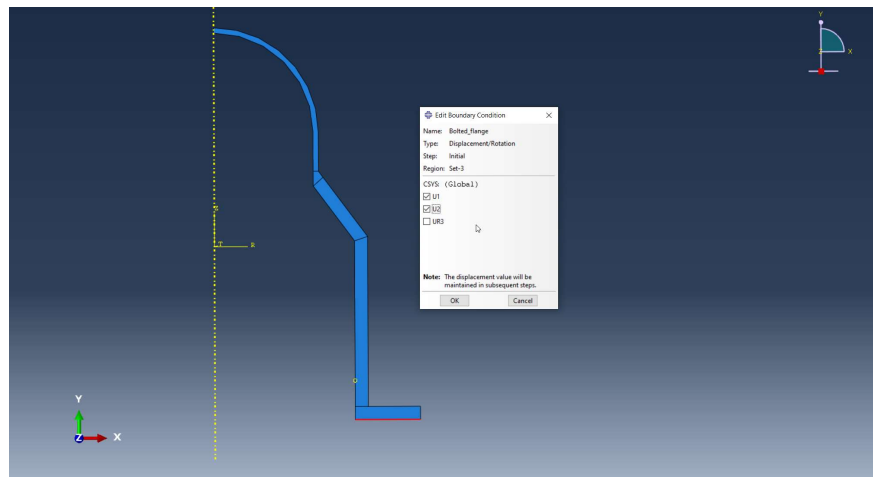


Figure 7: Boundary condition for bolt flange.

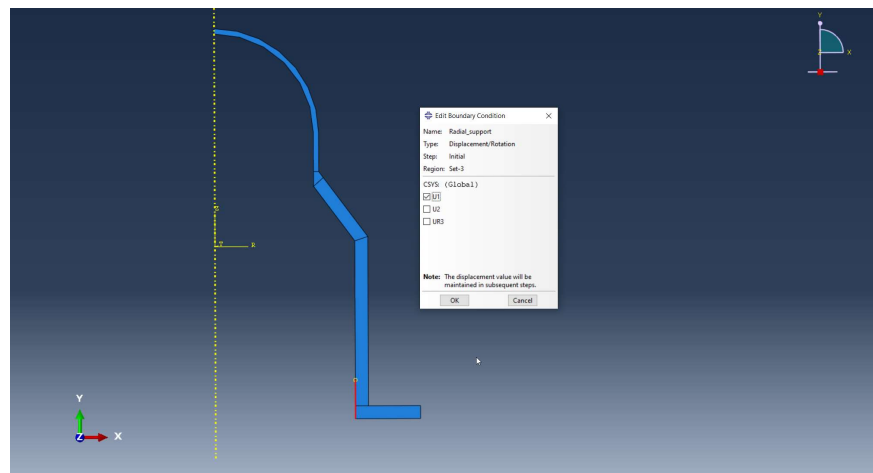


Figure 8: Boundary condition for radial support.

1.6 Meshing the Model

To create a mesh, choose **Mesh** in the **Module**. We can decide how many elements we want over the thickness by assigning mesh seed to the partitions we made earlier. We choose **Seed Edges** and pick the partition edges. Set the **Method** to **By number** and select the number of elements to the desired number over the thickness, see Figure 9. We should also go to **Mesh Controls** and select **Quad** and **Structured** for the entire part.

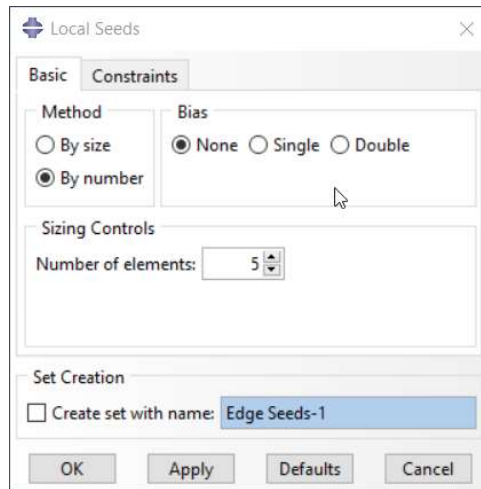


Figure 9: Setting number of elements for the Local Seeds.

Now, we go to **Assign Element Type** and pick all regions. The automatically chosen element type is the one we want, **CAX4R**. We are ready to mesh the part. Click **Mesh Part Instance** and **Yes** at the bottom of the window.

Note: You can play around with the element size/number of elements in the different region on the part using the **Seed Edges** tool. Our mesh should look something like in Figure 10.

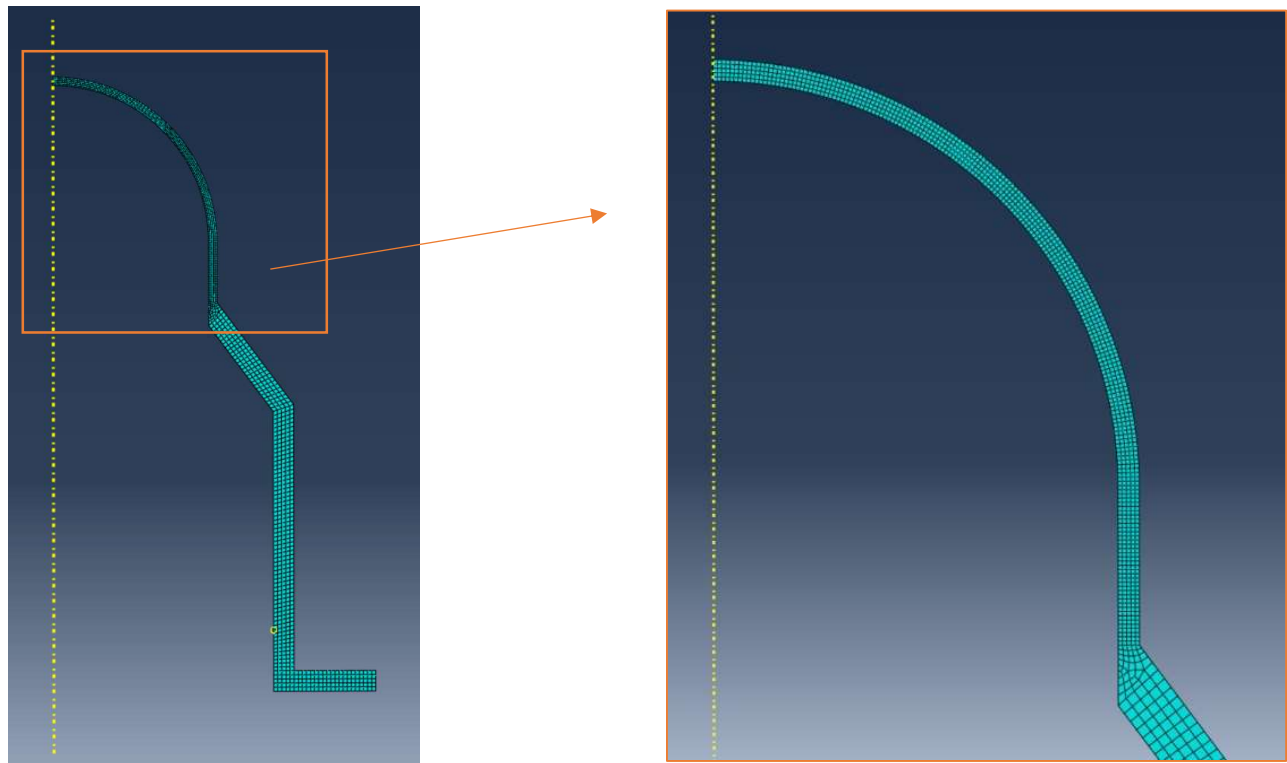


Figure 10: Mesh of spherical housing.

1.7 Creating and Submitting an Analysis

To create a job, double-click **Jobs** under **Analysis**. We can give the job a name or description so that we know what model it is if we come back to check our results another time. We can choose the same name as the model and the mesh size; **Axisymm_Spherical_5**. Now we know that this job has our mesh with 5 elements over the thickness. We do not have to specify anything in the **Edit Job** box.

Our job is now created under **Jobs**. Check that you have set the right **Work Directory**. Right-click the job and **Submit**. We can monitor the analysis progress under **Monitor**.

1.8 Post-processing the Results

The job should be finished within seconds. Right-click the job and press **Results**. Now, we are in the **Visualization Module**. We are ready to report some results.

First, we might want to change the deformation scale factor.

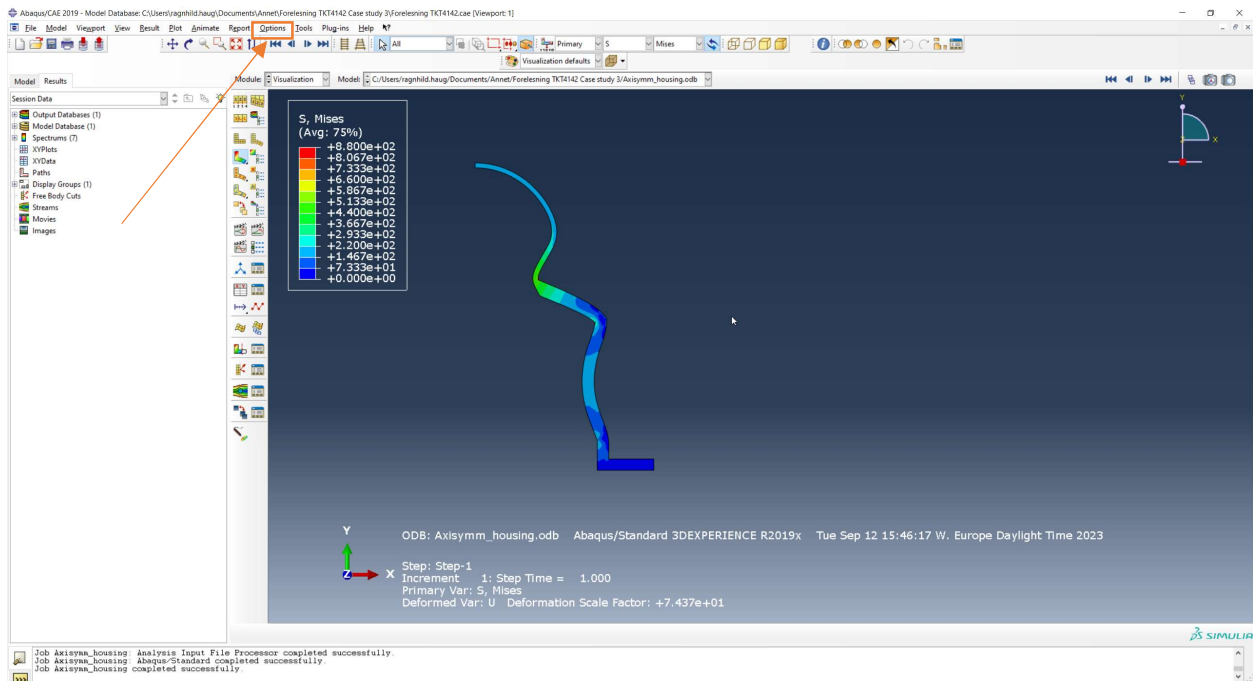


Figure 41: Changing the deformation scale factor.

Go to **Options** in the Menu bar and click **Common**. In the window popping up, see Figure 11, change from **Auto-compute** to **Uniform** and enter **1**, see Figure 11. Now the deformations are to scale.

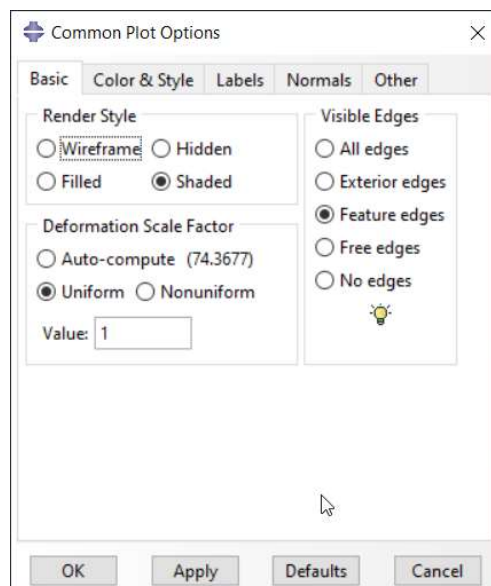


Figure 5: Common Plot Options.

To find the reaction forces from the bottom flange edge go to **XY-data, Create, ODB history output**, select all forces (for all nodes) that should be summed, **Save as, sum((XY,XY,...))**. Read the result from graph: **Tools, query, probe values**, and select the last data point which will be the sum of reaction forces.

We can sweep the 2D results to look at a 3D version of the housing. Go to **View** in the Menu bar and choose **ODB Display Options**. Navigate to **Sweep/Extrude**. Check **Sweep elements** and set the sweep from **0** to **360**. Increase the **Number of segments** to get a smoother sweep.

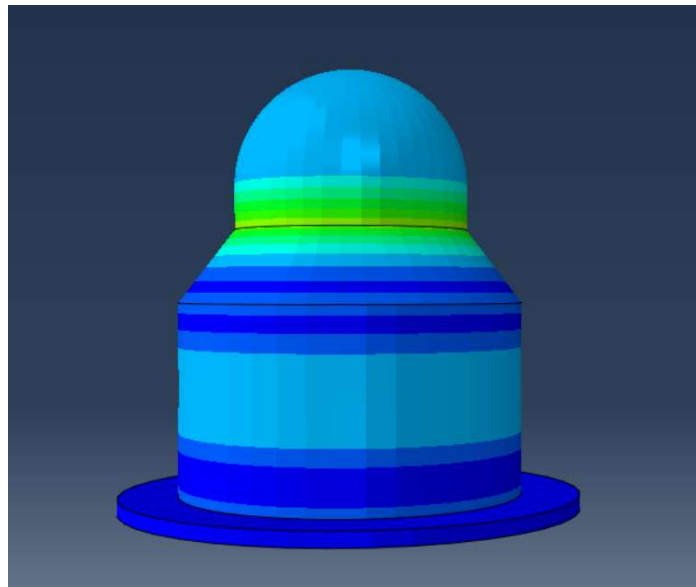


Figure 6: Sweep of housing.

When we want to check the radial (S11), axial (S22) and hoop (S33) stresses we can change the variable of the contour plot by going to the drop-down menu in Figure 14 and choose e.g., S11 instead of Mises.

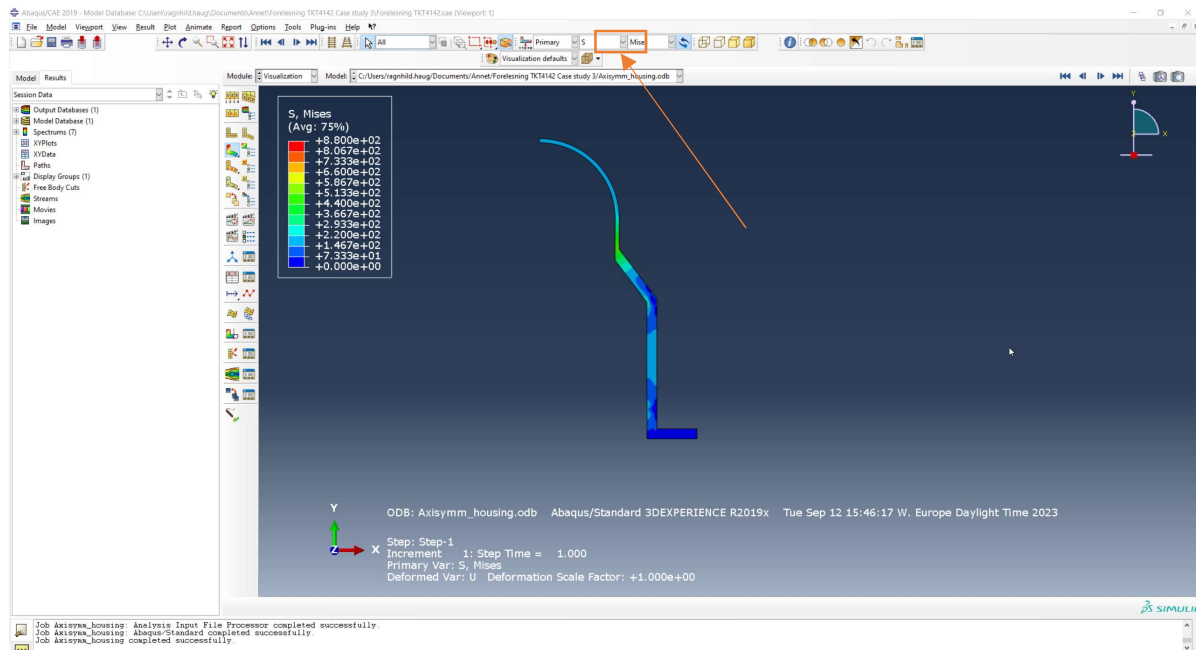


Figure 7: Changing the variable in the contour plot.

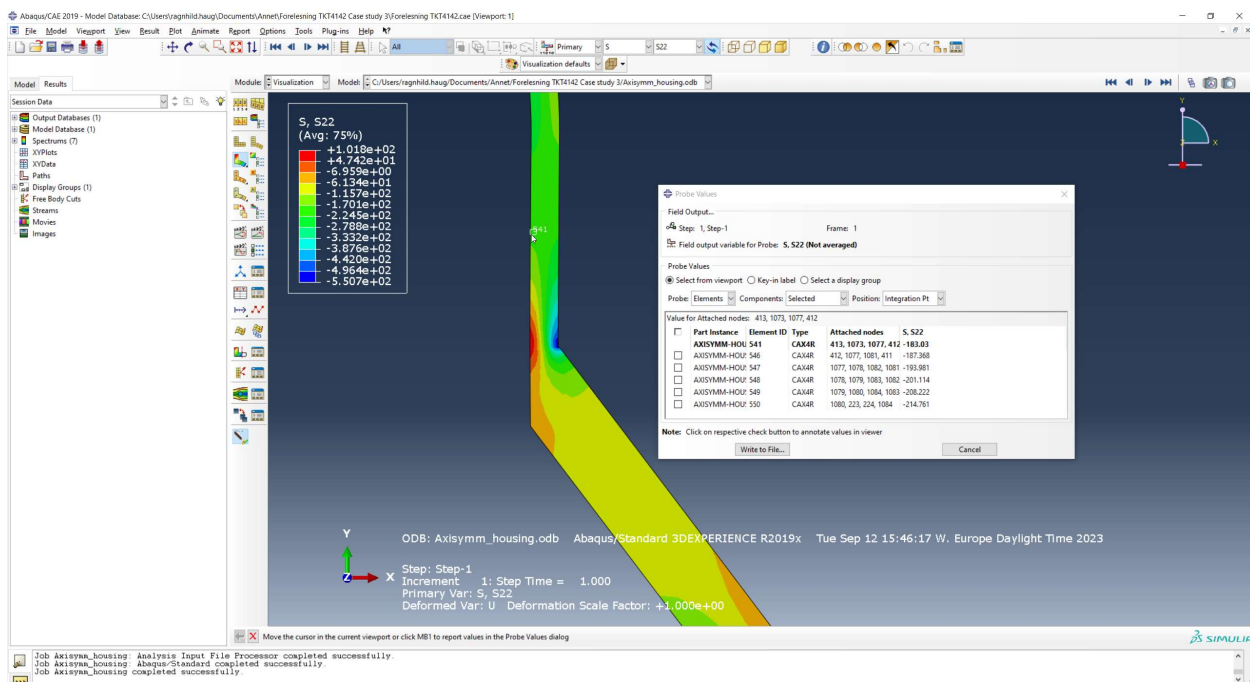



Figure 15: Finding the average stress value across the thickness.

To find the stresses in the cylinder area, we can probe values in the **Contour Plot** with the  button, or from selecting **Tools, query, probe values**. Choose five elements across the

thickness by clicking all of them. The values are displayed to the right in the **Probe Values** window, see Figure 15. Average these values.

2.0 Changing geometry and mesh size

As mentioned earlier we can copy the model when we want to change the housing to flat top or change the number of elements over the thickness. Be sure to check that the right model is the one active, the active model will be underlined. When we want to make the housing flat top, we can go into our sketch under our part and change it. We will have to regenerate the **Shell Planar**. If we edit it and click **OK**, it should update.

If we want to change the number of elements across the thickness, change the mesh seeding. It is also a good idea to change the global seed accordingly to avoid generating elements with bad aspect ratios.