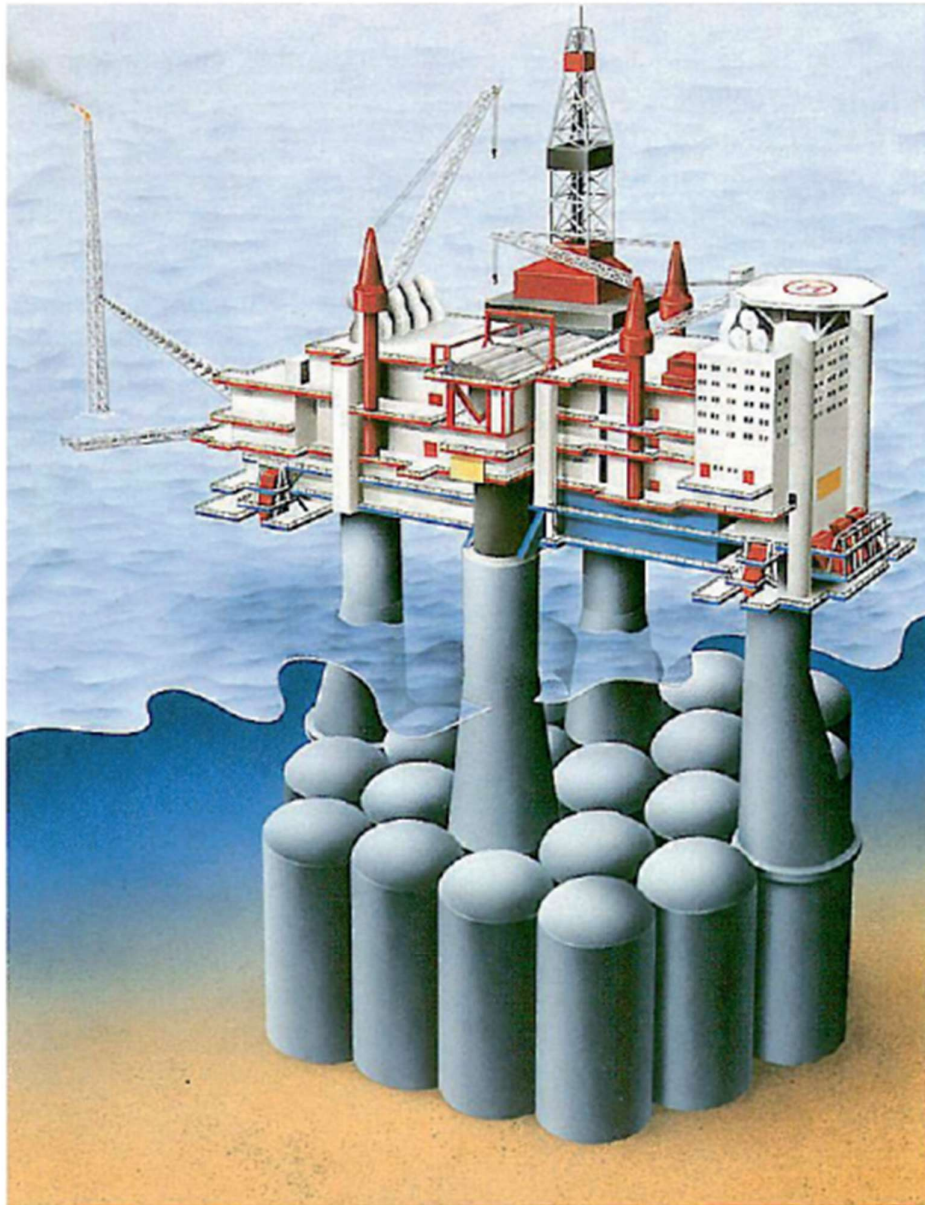


TKT4142 Finite Element Methods in Structural Engineering WORKSHOP CASE STUDY 2

SLEIPNER A - CONDEEP PLATFORM



Sleipner A. Illustrasjon: Norwegian Contractors

- Part: Creating a Part
- Property: defining and assigning material and section
- Assembly: Gather parts and instances. Defining partitions.
- Step: Creating a step.
- Interaction: Contacts etc. Not applicable to this workshop.
- Load: Apply loads and boundary conditions.
- Mesh: Mesh part or assembly.
- Job: Submit, manage, and monitor.
- Visualization: Post-process the results.

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**. Since we are going to investigate three meshes and two element types, we can organize our models according to the following procedure. Rename “**Model-1**” to e.g., “**Mesh-1-CPE4**”. When we want to change the mesh or element type, we can simply copy this model, change the name, and change the mesh/element type. More on this later.

1.1 Creating the Part

Double-click **Parts** under **Mesh-1-CPE4**. Name your part (e.g., **TriCell**). This will be a **2D Planar, Deformable, Shell** part with an **Approximate size 6000**. Now we are ready to start drawing the part.

By exploiting symmetry in load and geometry, it is sufficient to consider a reduced computation *plane* model in which the model can be reduced to 1/6 of the tricell as shown in in Case Study 2.

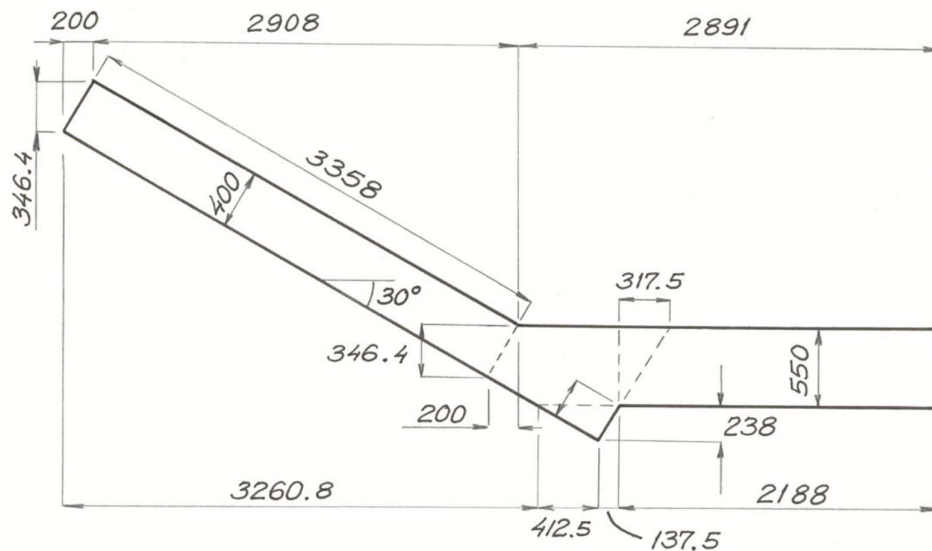


Figure 2: Geometry of the computational model

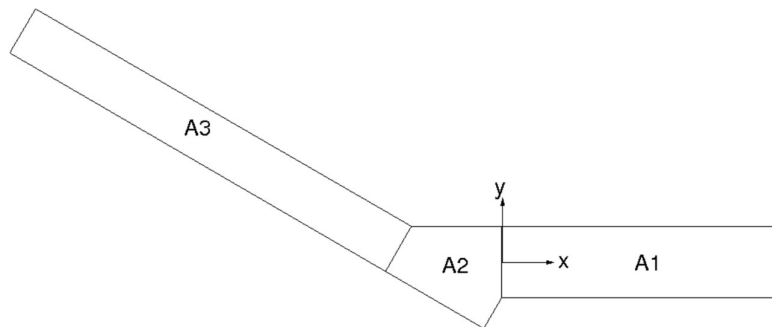


Figure 3: Sketch of the geometry with partitions A1, A2, A3.

With a simple geometrical consideration, we find the points that describe the model geometry in Figure 2. If we consider Figure 3 with the tricell divided into three partitions, we see some additional points that are useful to characterize. We use the coordinate system with point of origin between A1 and A2 in Figure 3.

Use **Create Lines: Connected** to draw the shape using the points in Table 1.

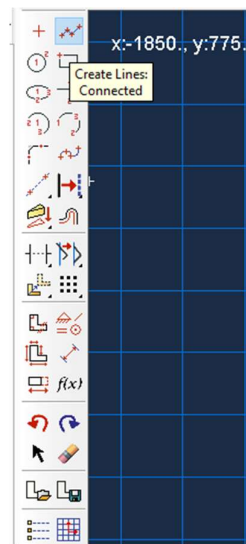


Figure 4

x coordinate	y coordinate
0	275
-703	275
-3610.8	1954
-3810.8	1607.6
-903	-71.4
-550	-275
-137.5	-513
0	-275
2188	-275
2188	275
0	275

Table 1

If you misplace a point, you can use **Drag Entities** to move the point. When you have finished drawing the part, click **Done**.

1.2 Assigning Properties (Material and Section)

For the model of the Sleipner Platform, we will use a simple concrete material. To create the material, double-click **Materials**. Pick an appropriate name for the material and add the material behaviour from Table 2. Remember to use consistent units. Press **OK** to accept.

Density [kg/m ³]	Young's Modulus [MPa]	Poisson's ratio [-]
2500	30000	0.15

Table 2

Double-click **Sections**. Give the section an appropriate name. This section is **Homogeneous** and **Solid**. Click **Continue**. Pick the **Concrete Material** and choose **Plane stress/strain thickness 1**.

Now we can assign the section to the part. Go to **Parts, Tricell**. Double-click **Section Assignments**. Pick the geometry region in the drawing area. Click **Done**. Pick the section we created. Click **OK**.

1.3 Create an Assembly

We need to create an instance in the model. Go to **Model, Assembly** and double-click **Instances**. Pick **Create instance from Parts**, choose the part, and pick **Instance Type Independent**. Click **OK**.

We also want to create the partitions. This will ease our meshing later. Pick **Assembly in Module**:

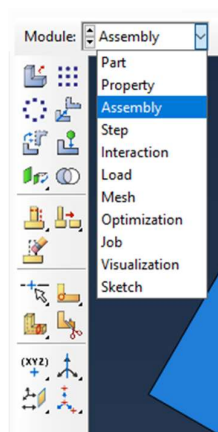


Figure 5

Go to **Tools, Partition** in the top ribbon. Type **Face** and **Sketch**. Use **Create Lines: Connected** and pick two and two appropriate points to divide the part into A1, A2 and A3. Use the **Esc** key when you have drawn a line to finish it. When you have drawn both lines, click **Done** at the bottom of the window. Now the part should look like this:

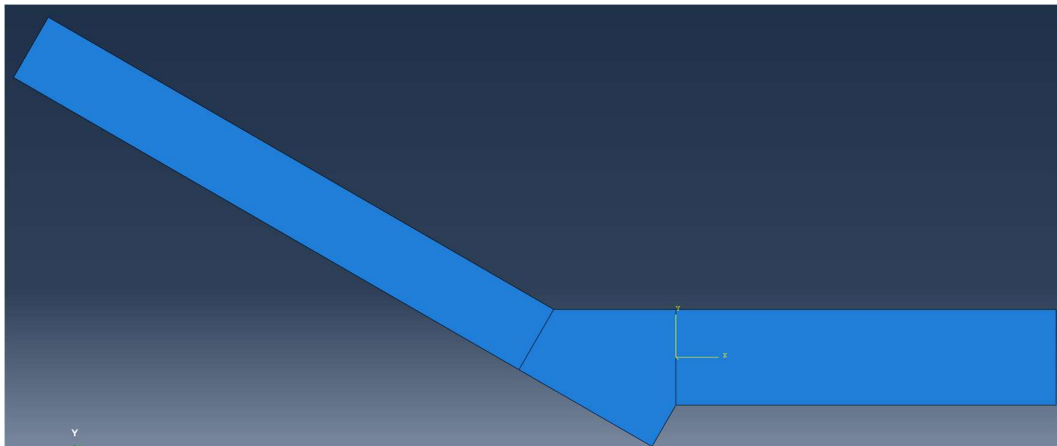


Figure 6

Now, we have to define some more **Partitions** in the model to obtain the force and moment distributions in Task b). This is somewhat cumbersome, so we need to follow the next steps carefully.

We want to find create five **Surfaces** along the tricell wall where we extract the forces and moments. We will define each surface by using **Datum Planes**.

Go to **Assembly**. Click **Create datum plane: Offset from principal plane, YZ Plane**. We now want to create three equidistant Datum planes. In **Offset**:, enter the following distances to offset our datum planes: 547, 1094 and 1647. Press **Esc** when you have placed all four planes. Now, we want to create five **Reference points**. With the **Assembly** module still open, go to the top ribbon and click **Tools, Reference Point** and enter the following five points: (0,0), (547, 0), (1094, 0), (1647, 0) and (2188, 0). Press **Esc** when you have placed all five points. Now, we are ready to **Partition**. With the **Assembly Module** open, click **Partition Face: Use Datum Plane**. Click the tricell wall (A1) in **Select the faces to partition**. **Partition A1** using all the **Datum Planes** we created.

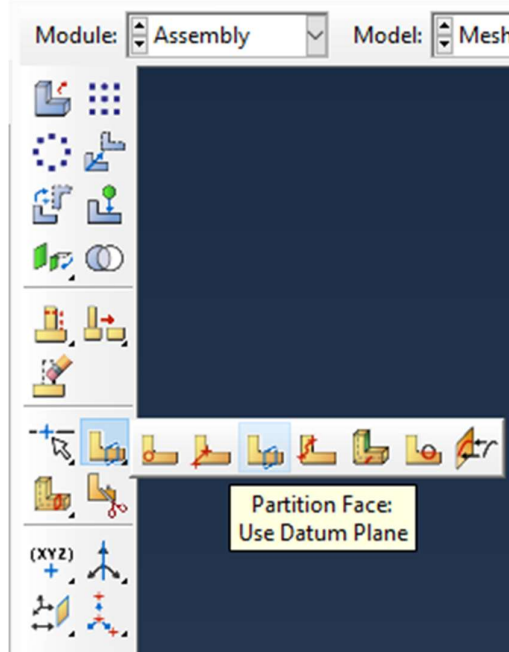


Figure 7

Your **Assembly** should now look something like this:

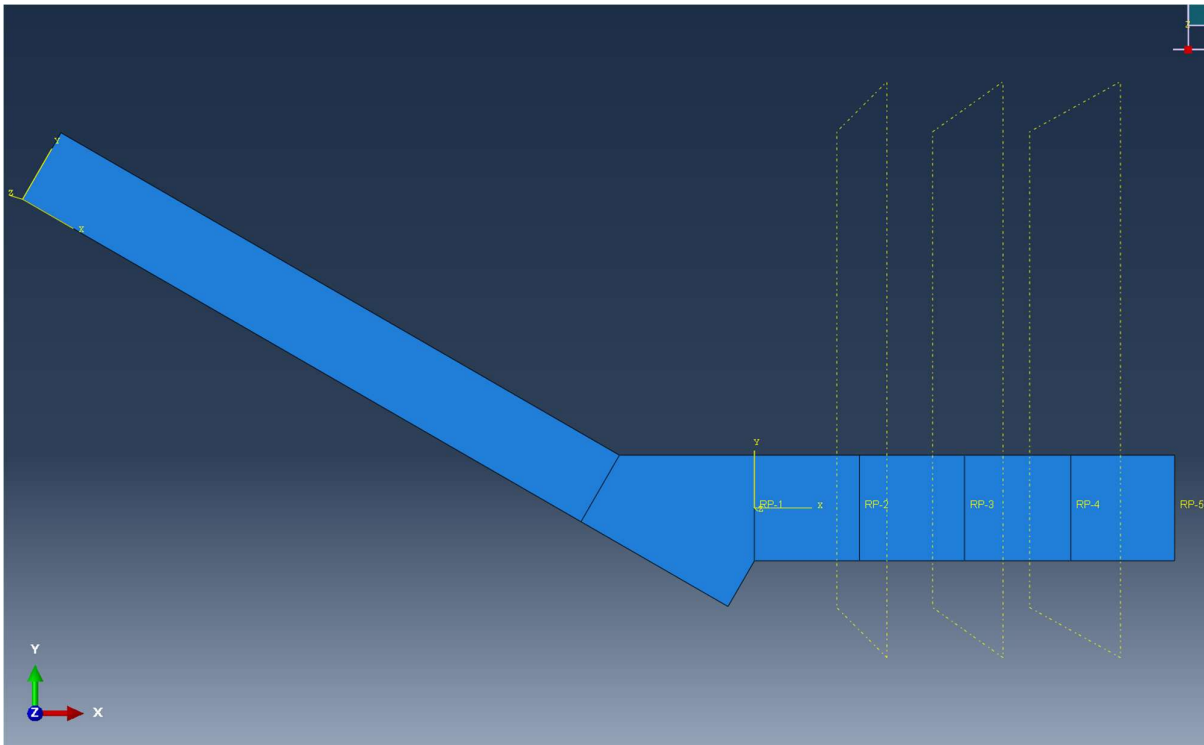


Figure 8: Assembly after partitioning using datum planes

Now, we want to create five surfaces that we can use for our outputs. Double-click **Surfaces** in **Assembly** and create a surface with **Name Surf0000** from **Type: Geometry**. Click **Continue**. Now, click **Select From All Entities** in the top ribbon. You have to select **Select From All Entities** every time you create one of these **Surfaces**.

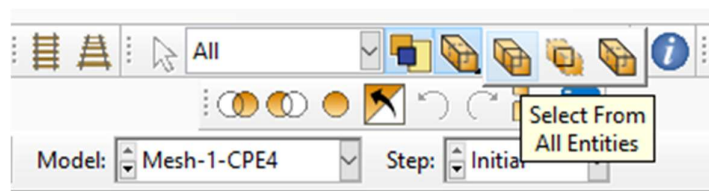


Figure 9

Select the edge at $x = 0$. Click **Done**. When you have the choice between **Magenta** and **Yellow**, click the one that is on the left in the **Viewport**.

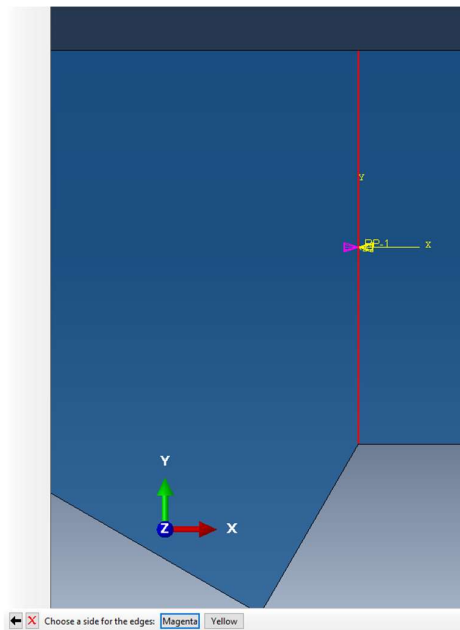


Figure 10: We select **Magenta** here, as it is on the left side.

Repeat this **Surface** creating procedure four times for surfaces called **Surf0547**, **Surf1094**, **Surf1647** and **Surf2188**. You will not get any options for **Yellow** or **Magenta** for **Surf2188**. The numbers after “Surf” are not important for Abaqus, but they are a naming convention we will use to keep track of where we created the surface.

1.4 Defining the Step and Output Requests

Before we apply loads and boundary conditions, we should create a step. Double-click **Steps** and create a **Static, General** step called **LoadStep** after **Initial**. **Time period** can be set to **1**. Under **Incrementation** we can set the initial **Increment size** to **0.01**. Click **OK**.

Abaqus automatically creates two output requests called **F-Output-1** and **H-Output-1** once a step is defined. These two requests contain the most basic information. Check out CS1 for more explanation regarding the basic outputs.

To get the nodal forces, we will request a Field Output. Double-click **Field Output Requests**, **Name: F-Output-2**, **Step: LoadStep**. Click **Continue**. In **Domain**, choose **Whole model**. On **Element output position**, pick **Nodes**. The output we request is under **Forces/Reactions: NFORC, Nodal forces due to element stresses**. Click **OK**.

We also want to report Sum of Forces (SOF) and Sum of Moments (SOM) for our model. To do this, we go to the **Step Module**. In the top ribbon, click **Output, Integrated Output Sections** (we can call this “IOS”) and **Create**. Create an **IOS** called **I-Section-0000**. Click **Continue** and click **Surfaces** in the bottom right corner of the **Viewport**. Pick **Surface0000** and **Continue**. Now, we see the **Edit Integrated Output Section** dialogue box. Click **Anchor at reference point**. Click the mouse cursor symbol and **Dismiss** the **Region Selection** dialogue box. Pick the **Reference point (RP)** in the **Viewport** that is on the **Surface0000**, that is **RP1**. Check **Move point to centre of surface**, with **Point Motion: Average translation and rotation**.

Click **OK**. We have to create four more **IOSes**, one for each of the **Surfaces** with corresponding **Reference Point**. Use the same naming convention for the **IOSes** as we used for the **Surfaces**.

When we have created all five **Integrated Output Sections**, we are ready to create five **outputs requests** for **SOF** and **SOM**. Double-click **History Output Requests, Name: H-Output-Section-0000** in **LoadStep**. **Domain: Integrated output section**, and pick **I-Section-0000**. Check both **SOF** and **SOM** in the list below. Click **OK**. *Repeat this process for the other IOSes.*

1.5 Applying Load and Boundary Conditions to the Model

From the case study document, we have found the explanation and values for p_1 and p_2 .

$$\begin{aligned}
 p_1 &= \rho g h = 0.674 \text{ MPa} \\
 p_2 &= 0.55 p_1 \frac{r}{w} = 5.68 \text{ MPa}
 \end{aligned}
 \quad \text{where} \quad
 \begin{cases}
 \rho = 1025 \text{ kg/m}^3 \text{ (sea water)} \\
 g = 9.81 \text{ m/s}^2 \\
 h = 67 \text{ m} \\
 r = 12.25 \text{ m} \\
 w = 0.8 \text{ m}
 \end{cases}$$

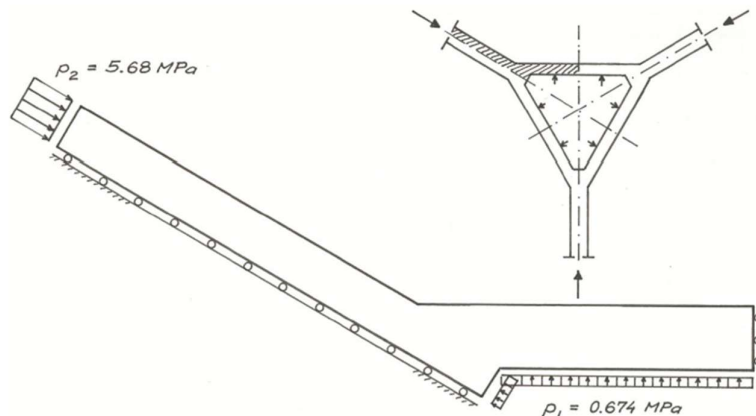


Figure 11

These loads will be applied as pressure loads in our model. Double-click **Loads**. Create a load called **LeftEdgeLoad**. **Category Mechanical** and **type Pressure**. Pick the left edge of the part and apply p_2 . Remember to use consistent units. Distribution **Uniform**, appropriate **Magnitude** and **Amplitude (Ramp)**. Click **OK**. Repeat for p_1 . Choose an appropriate name (e.g., **BottomEdgeLoad**). Remember to pick the appropriate edges (see Figure 11). Hold Shift to select several edges. We observe that pressure loads automatically act perpendicular to the selected edges/surfaces.

We are now going to apply the boundary conditions. We see from Figure 11 that 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 right-side edge with the coordinate system (**CSYS**) **Global** and fix **U1** and **UR3**. The edge is now free to move in the global **U2** (that is y) direction. Click **OK**.

Double-click **BCs** again. The left edge has the same type of BC, however, we should create a local **CSYS**. Simply pick all the appropriate edges (be careful to select all), and next to **CSYS**, choose the axes-symbol:

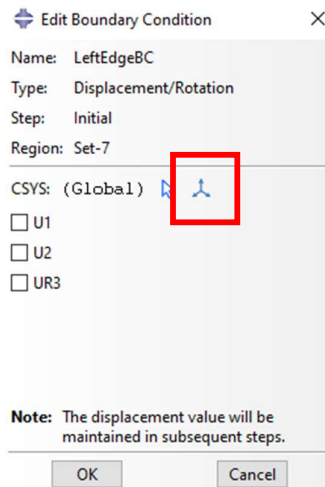


Figure 12

Now, we define the local axes. **Type: Rectangular**. Choose the lowest point on the left edge (with coordinates -3810.8, 1607.6). Next, pick a point to define the **x-axis** (e.g., -550, -275). Next, pick any point on the **X-Y plane**. Now, we have our local axis system defined. Press the mouse symbol in the **Edit Boundary Condition** box and pick the **CSYS**. We fix **U2** and **UR3**.

1.6 Meshing the Model

In the case study, we will investigate three different meshes.

Mesh number	Number of elements over thickness
Mesh 1	1
Mesh 2	2
Mesh 3	4

Table 3

To create a mesh, choose **Mesh** in the **Module**. First, we want **1** element over the thickness in partition **A3**. We can choose **Seed Part Instance** and use an **Approximate global size** of **550**. Click **Apply**.

Now, we go to **Assign Element Type** and pick all three regions. We find the **Plane Strain** elements in **Family**. Find the linear element type **CPE4** by turning off **Reduced integration**. Note that we find **CPE8** if we go to **Quadratic**. Choose element type **CPE4**. Click **OK**.

Go to **Mesh controls** and pick **Quad-dominated, Structured**. Click **OK**.

Now, we are ready to mesh the part. Click **Mesh Part Instance**, and **Yes** at the bottom of the window. We have now created a quite coarse mesh on the part instance. You can find the mesh under **Assembly, Instances, TriCell, Mesh** or under **Mesh** in the **Module**.

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; **Mesh-1-CPE4**. Now we know that this job has our **Mesh 1** and the **CPE4 elements**. We do not have to specify anything in the **Edit Job** box. Click **OK**. 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 job/analysis progress under **Monitor** to see if we get any warnings or errors.

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 investigate some results.

Task a) tells us to plot the distribution of the two stresses over the end section of the tricell wall, at $x = 0$ in Figure 3. To find/plot nodal values of σ_{xx} and σ_{xy} , we can use a **Path**. In the top ribbon, go to **Tools, Path, Create**. We want to create a path from **Node list**. Under **Viewport selections**, choose **Add After**. Now, pick the nodes you want to define your path. A good rule is to be consistent with the direction you choose for the path. Here, we go from the bottom-up, meaning that we choose nodes **2** and **1** (the nodes might be numbered differently in your model). Click **Done**. The path is now listed in the left menu. Double click **XYData** in the **Model tree**, and choose **Path** as **Source**. Choose the appropriate path and choose **S11** (which is σ_{11} or σ_{xx}) as **Field Output**. **Plot** shows the data, while **Save As** allows us to save the data for later plotting. **Save** the data for **S11**, and plot **S12** (which is σ_{12} or σ_{xy}). Save the **S12** data. Press **Cancel**. Right-click the data for **S11** and **S12** in **XYData** and **Add to plot**. You can either export these data to another program to plot them, or you can print the plot using **File** → **Print**.

If you want to copy the data into another program (Python, Matlab, or similar softwares), one (manual) way to do it is to right click e.g., **S11**, **Edit** and copy the data from the table.

Export a picture of the deformation plot from **File** → **Print** in the main menu bar. Change the **Destination** to **File** in the **Print** dialog box and set your destination.

Obtaining numerical values from the Field Output.

Another way to obtain numerical values from the Field Output is to **Probe values**. Select the **Field Output** you want from the **Field Output** menu. Now, use the **Query Information** tool . Click **Probe values**. In the **Probe values** dialog box, **Probe: Nodes**, and either **Select from viewport** or **Key-in label** if you already know which nodes you want to probe. The right-most column in the table shows the value you have probed.

2 Expanding the Model

2.1 Editing Mesh and Element Type

Refining the mesh

We have now created a model with the coarsest mesh and element type **CPE4**. In the case study, we want to investigate finer meshes and elements with more integration points.

One way to do this without losing the results from the first model, is to copy the model and make changes in the copy.

Go to **Models** and right-click **Mesh-1-CPE4** (or **Model-1**). Click **Copy Model**. First, we will investigate a finer mesh. Name the new model **Mesh-2-CPE4**. Since we created the mesh in assembly, we go to **Assembly, Instances, TriCell-1** (the part) and double-click **Mesh**. Now we are ready to refine the mesh.

We can use **Seed Edges** to assign how many elements we want across the thickness. Click **Seed Edges** and pick **4 edges**: the left-most edge, the edge between A2 and A3, the edge between A2 and A1, and the rightmost edge. Hold **Shift** to select more edges. **Done**. We can choose **By number**, and choose **2** as **Number of elements**. **OK**. We should also apply a **Global seed**. Click **Seed Part Instance** and choose **Approximate global size of 275** ($=550/2$). Click **OK**.

Click **Mesh Part Instance**. We now have two elements across the thickness in both A3 and A1. For the finest mesh, we follow the same procedure, but with a global seed of **137.5** and **4 elements** on the **edges**. With the new mesh(es), we can create a new **Job**.

Change element type

Again, it is useful to create a new model when we want to change the element type and compare the results. In the new model, we go to the **Mesh**. Click **Assign Element Type** and pick all regions on the part. We can change the **Geometric Order** to **Quadratic** to find the **CPE8** element. Click **OK, Done**. Now, we can create a new **Job** and run it.

2.2 Finding Internal Nodal Forces

We can use the NFORC output we requested to find the resultants. Click **Primary** and **NFORC1** as shown in Figure 13.

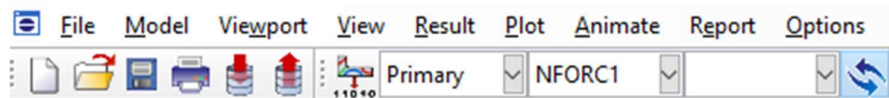


Figure 13

It is easier to see which elements to pick if we highlight their labels in the viewport. Click **Options** and **Common**. Under **Labels** click **Show element labels** and **Show node labels**. This is shown in Figure 14.

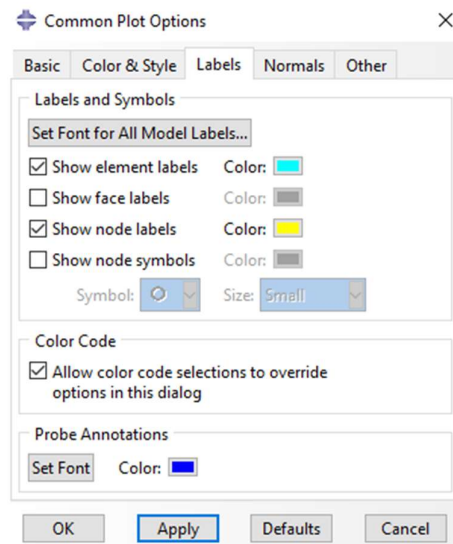


Figure 14

Now, we can use **Probe values** to get the nodal forces. First, turn off averaging in **Results, Options**. Uncheck **Average element output at nodes**. Now, we are ready to find internal nodal forces. Click **Query information** and **Probe values**. **Probe: Elements**, and pick e.g., **element 1** from the viewport. Now, we can read the internal nodal forces in the x-direction in the element's four nodes.

When we want to find the forces in the y-direction, we repeat the procedure in "2.2 Finding Internal Nodal Forces", but we select **NFORC2** instead of **NFORC1**. The Probe Values dialogue box is shown in Figure 15. We see that each node's **NFORC1** is printed in the rightmost column corresponding to the nodes in the column **Attached nodes**.

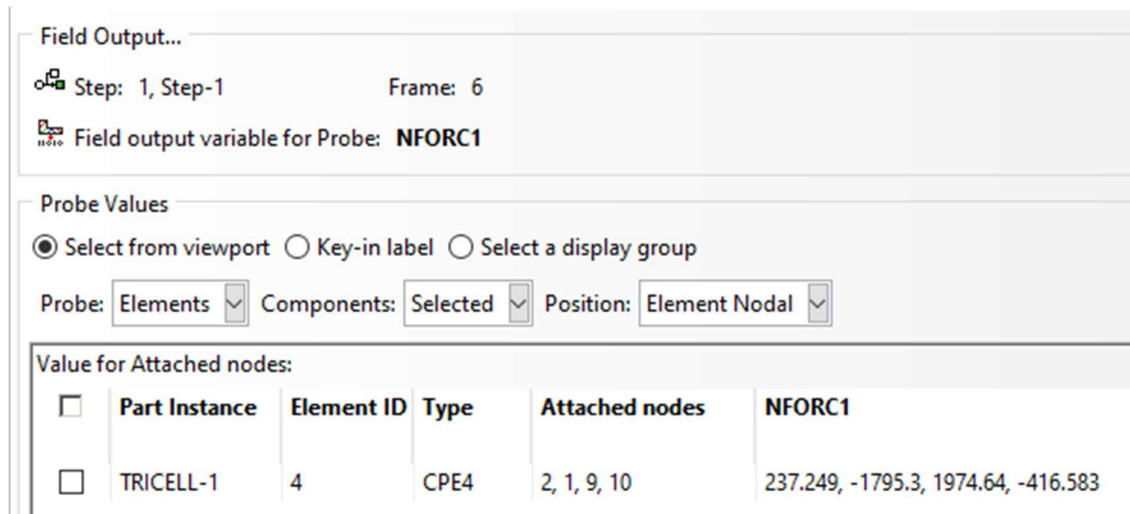


Figure 15: Probing values for NFORC1.

We will now see how we can find values for the shear and moment distribution. Double-click **XYData** and choose **ODB history output** as **Source**. We now see that we have gotten a bunch of outputs for **SOF1**, **SOF2**, **SOF3**, **SOFM**, **SOM1**, **SOM2**, **SOM3** and **SOMM** with their corresponding **Section** and **Surface**. We are interested in the force in the y-direction (**SOF2**)

and the moment about the z-axis (**SOM3**). We can mark all these outputs (Figure 16) and plot them against time, as shown in Figure 17.

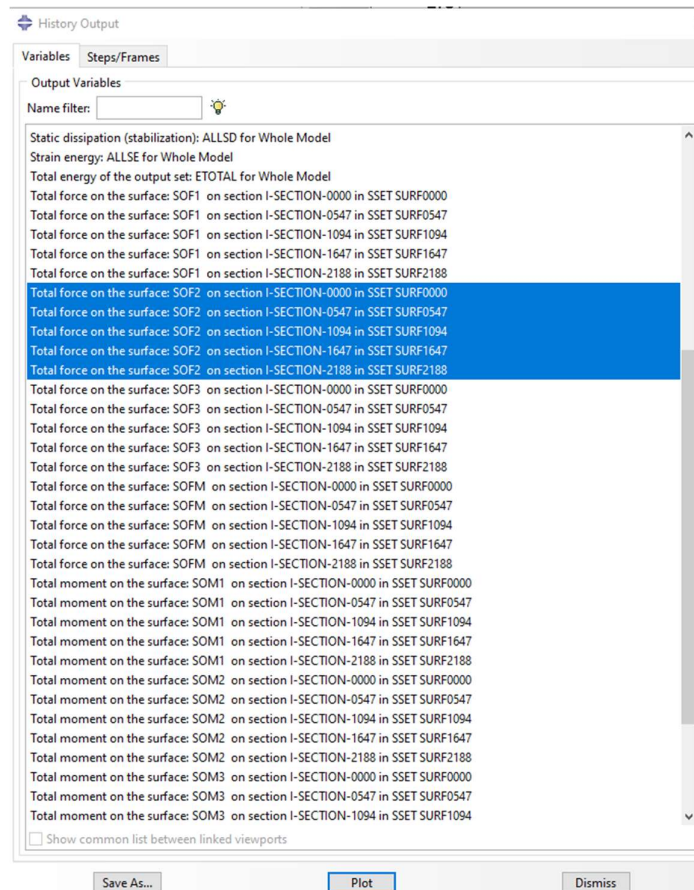



Figure 16: Marking all SOF2 outputs.

By using the **Probe values**  tool in the plot, we can probe the values at **Time= 1**. It is recommended to use another software to plot these values as a function of x in the tricell wall. Figure 17 shows the SOF2 values plotted against time for all our **Surfaces**.

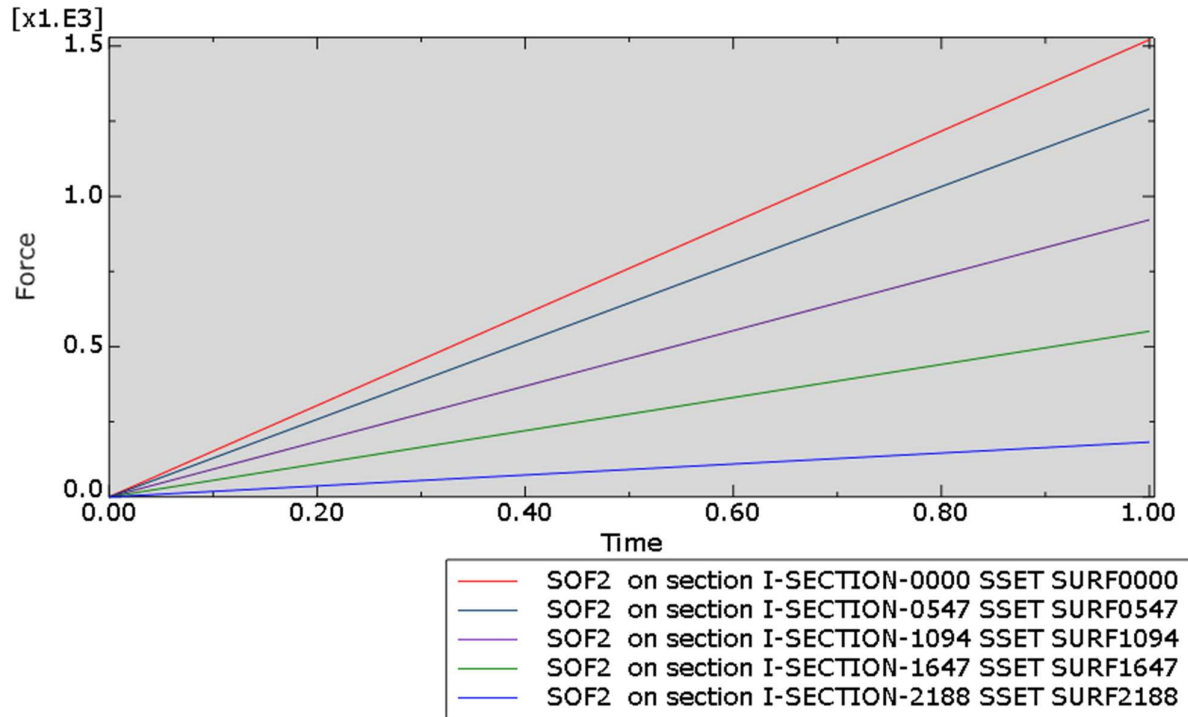


Figure 17: The SOF2 outputs plotted against time. Note: this is not the shear force diagram.