

TKT4142 Finite Element Methods in Structural Engineering

Workshop 5

This workshop will briefly guide you through Case Study 5. Case Study 5 will use a concrete slab to address the modelling of plate problems. perform a finite element analysis (FEA) of the rooftop of a car park when exposed to two different load scenarios.

Section 1 introduces 3D plate/shell elements in Abaqus by modelling the rooftop with a uniform load before the load is changed to point loads in Section 2.

1. Modelling the rooftop when exposed to uniform loading

The rooftop with dimensions and the uniform loading is shown in Figure 1.

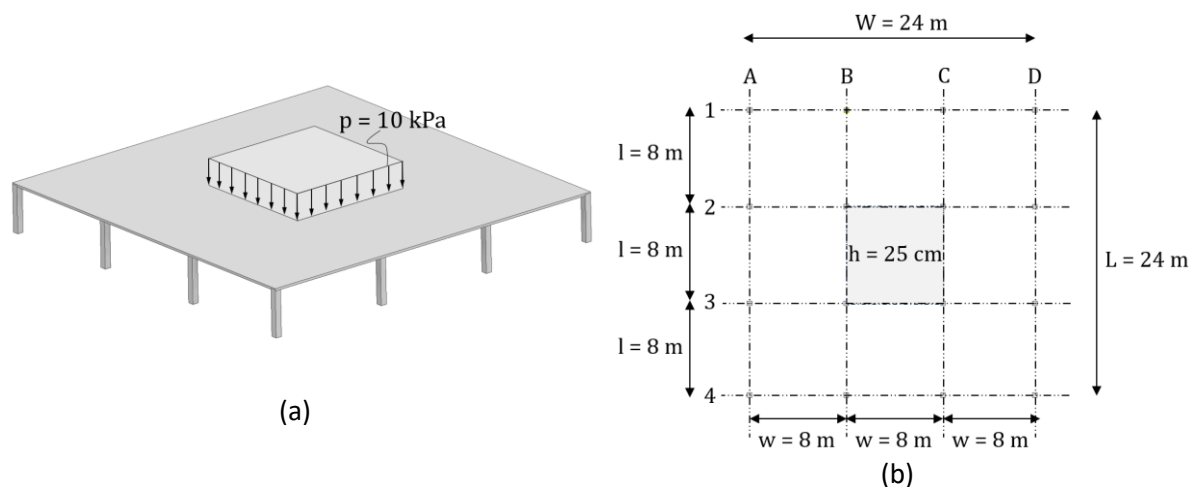


Figure 1 – A rooftop supported by 16 columns and exposed to a uniform load $p = 10 \text{ kPa}$ between axes B, C, 2, and 3.

Load: $p = -0.010 \text{ N/mm}^2$ (downwards)

Material data: $E = 32\,000 \text{ N/mm}^2$, $\nu = 0.20$, $\rho = 2500 \text{ kg/m}^3$, $\sigma_y = 20 \text{ N/mm}^2$


Make sure your work directory is properly set.

Change the model's name by right click on the model (named Model-1) in the **Model Tree** to access the **Models** menu. Select **Rename...** and enter **ROOFTOP** in the **Rename Model** dialog box. Select **OK**.

1.1. Creating a Part

1. Create a new part in **ROOFTOP** by double-click on **Parts** in the **Model Tree**. The **Create Part** dialog box appears.
2. In the **Create Part** dialog box, see Figure 1-1:
 - a. Name the part **Roof**.
 - b. Choose **3D Planar** in the **Modeling Space**.
 - c. Choose **Deformable** as the **Type**.
 - d. Use **Shell** as the **Base Feature** and **Planar** as **Type**.
 - e. **Approximate size** can be set to **10000**.

Click on **Continue**. This brings you into the sketch environment.

- Sketch the geometry in Figure 1. We will use mm as measurements. Use **Create Lines: Rectangle (4 lines)** tool () and create the rectangle representing the area of the rooftop of by specifying the starting points as (-12000,-12000) and the endpoint as (12000,12000).
- Click on **OK** in the prompt area to finish your sketch.

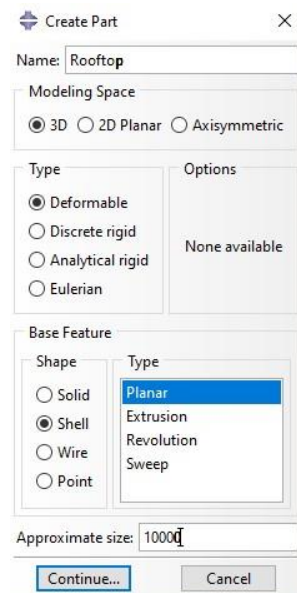


Figure 1-1 – The create part dialog box.

1.2. Assigning properties

- Define a linear **Material** named **Concrete**. The **Young's Modulus** and **Poisson's Ratio** are set to **32 000** and **0.2**, respectively.
- Create a **Homogeneous Shell Section** named **Slab**. Make sure to select **Shell** in **Category**. See Figure 1-2(a). Click on **Continue**.
- In the **Edit Section** dialog box, input a **Shell thickness** value of **250**. See Figure 1-2(b). Click on **OK**.

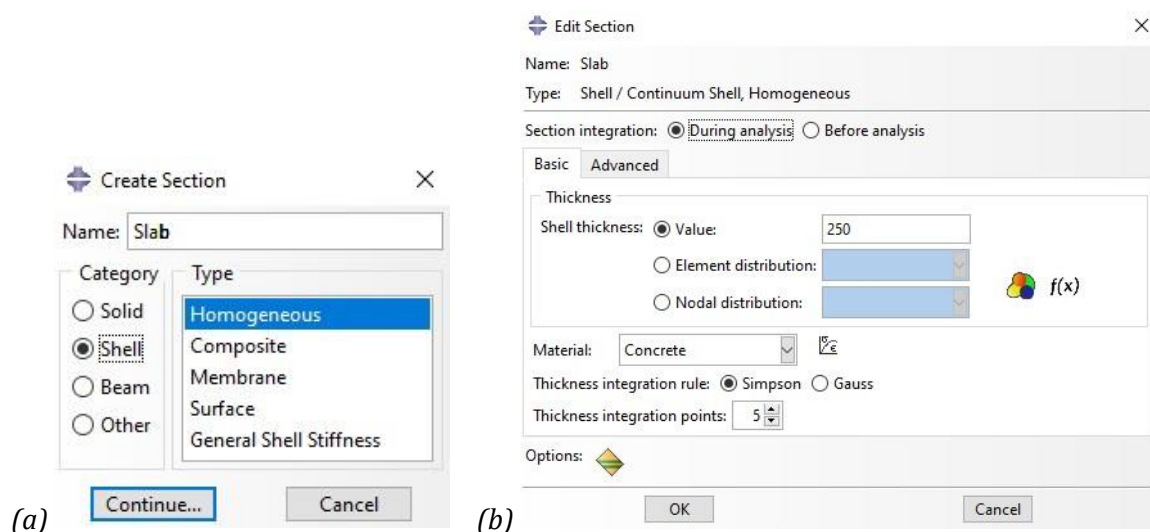


Figure 1-2 – (a) The Create Section dialog box and (b) the Edit Section dialog box.

Next, we must assign the section to our part. This must be done for both the flange and the web.

4. In the **Model Tree**, expand the branch for the part **Rooftop**. Double-click **Section Assignments** to assign a section to the part Rooftop.
5. Select the rooftop (See Figure 1-3(a).) and check **Create set** in the prompt. Call the set **Slab** and click **Done**.
6. In the **Edit Section Assignment** dialog box, select the Slab as the section. Use **From section** as **Thickness** and **Middle surface** as **Shell offset**. See Figure 1-3(b).

Your part should have turned green before you proceed.

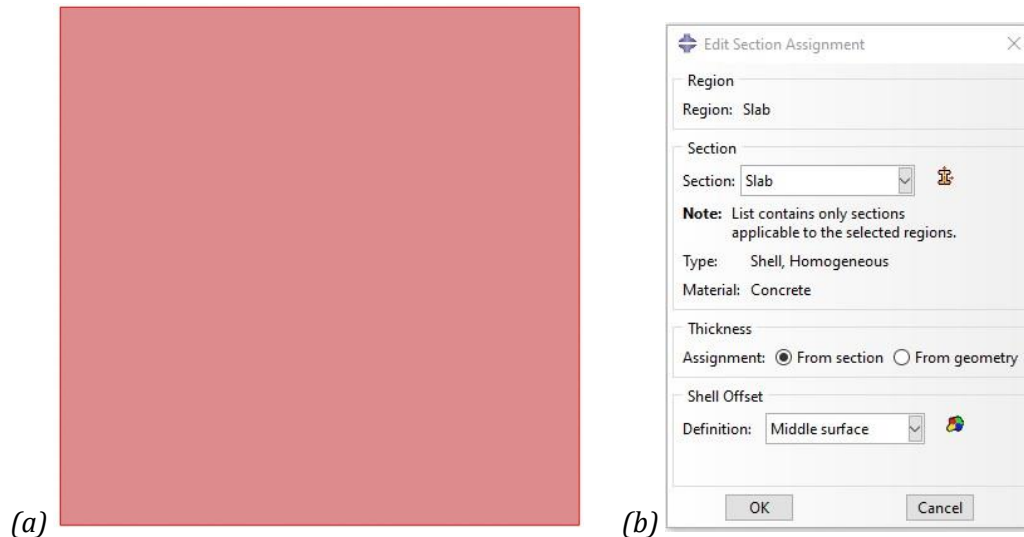


Figure 1-3 – (a) Selecting the rooftop in the viewport and (b) the Edit Section Assignment dialog box.

1.3. Create an Assembly

Create an **Instance**. Choose **Dependent (mesh on part)** as **Instance Type**.

1.4. Defining the Step and Output data

Create a **Step** called **Load**. Choose **Static, General** as **Procedure type**. Set the **Time period** to **1** and **Initial Increment size** to **0.1**.

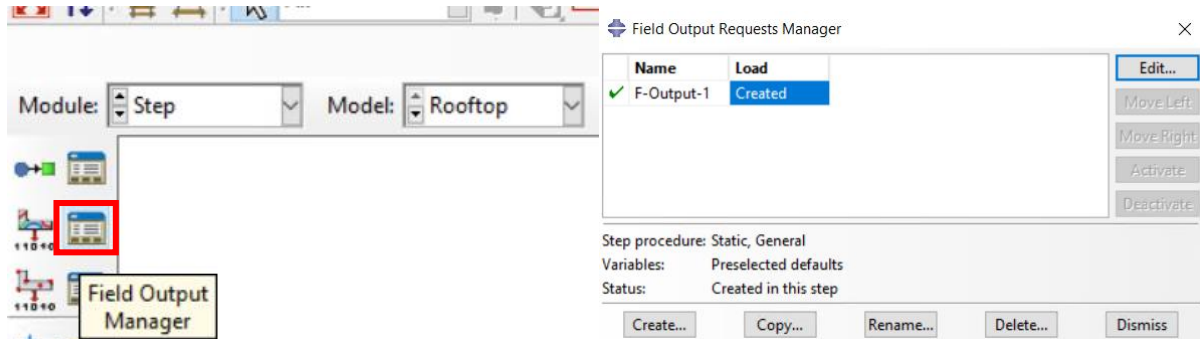
Output Requests

The **Field Output Requests** and the **History Output Requests** in the model tree define which data (i.e. stress, strain, displacement, temperature, etc.) should be included in our result file. Abaqus/CAE 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. They may be modified, but we will keep them as they are. In general:

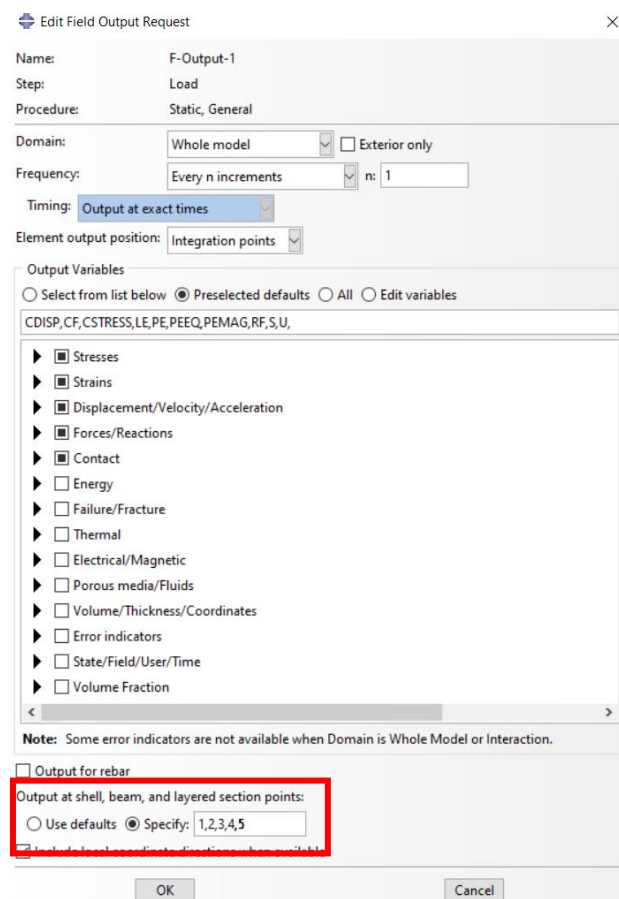
- **Field Output Requests** – The output-data defined here will be included for every point of your model. This allows us to output the data as a contour plot for instance. However, for very large models with many time steps, a field output may become large, resulting in a large output file. In addition, writing the data to the output file can be time-consuming. Thus, only essential data should be included as a field output in large models.
- **History Output Requests** – If we want to output data for only a few points, say the displacement at the end of the model, a history output is normally used. A history output request consumes much less space.

In Tasks 1b and 2b, we are asked to show the stress distribution over the cross section of the critical element. This implies that we need to define the integration points over the thickness in the Element Output. We can then access this information in the postprocessing of the results.

Use the Field Output Manager to Edit the field output requested in F-OutPut-1.



In the **Edit Field Output Request** under **Output at shell, beam, and layered section points**, choose **Specify** and ask for access to the data at all of the 5 integration points (1, 2, 3, 4, and 5) specified in Section 1.2 and Figure 1-2(b).



1.5. Applying Load and Boundary Condition to the Model

You must first apply **boundary conditions** to your model by constraining the model according to the columns supporting the rooftop in the vertical direction.

1. We start by partitioning the rooftop according to the axes A-D and 1-4 in Figure 1b. This can be done by revisiting the part module and sketch lines representing these axes, e.g.,

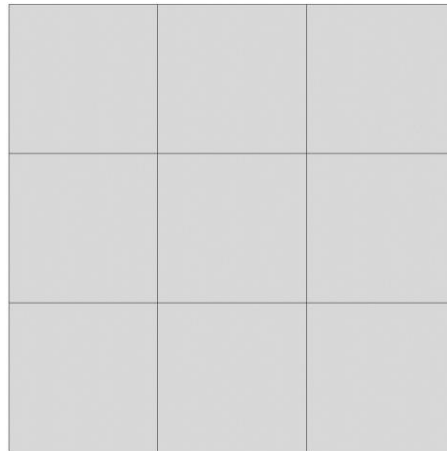


Figure 1-4 – Partitioning the part according to axes A-D and 1-4 in Figure 1b.

2. Create BCs called **Column_support**. In the **Create Boundary Condition** dialog box
 - a. **Step** is set to **Initial**.
 - b. **Types for Selected Step** is set to **Displacement/Rotations**.
3. Select the intersection points of your partitions/axes in the viewport. See red markers in Figure 1-5(a). Click **Done** in the prompt area.
4. Check **U3** in the **Edit Boundary Condition** dialog box. See Figure 1-5(b). This will restrain the rooftop from displacements in the z-direction at the center point of the columns.

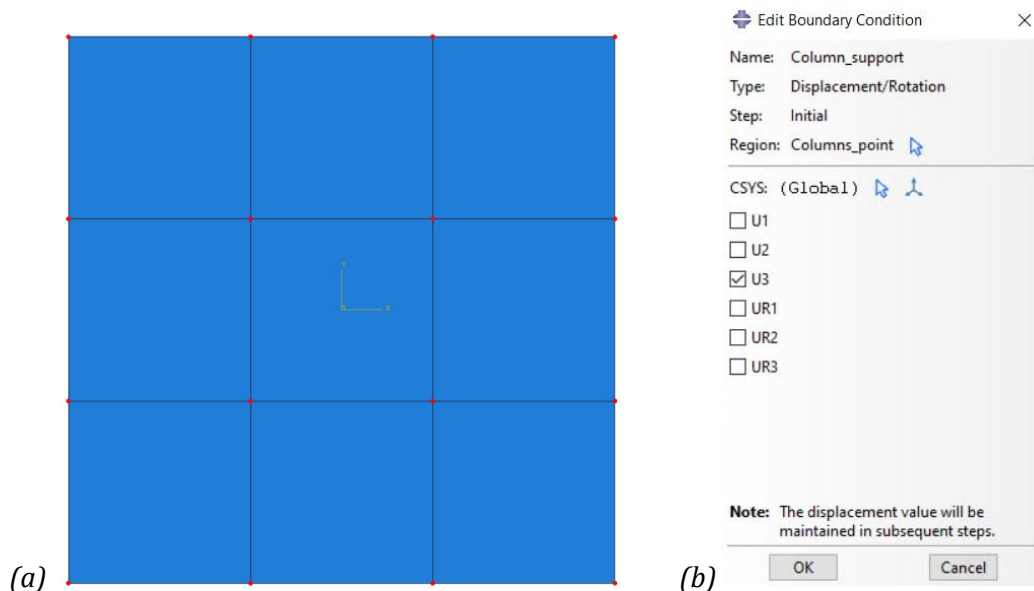


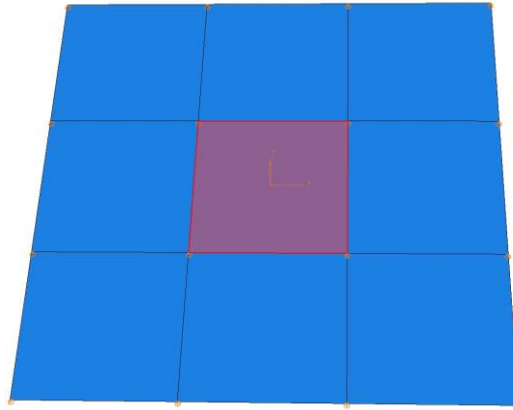
Figure 1-2 – (a) The selected right edges and (b) The Edit Boundary Condition dialog box.

Note #1: We have assumed that the rooftop is free to rotate at the columns.

Note #2: This is very simplified boundary conditions. An alternative will be to calculate the reaction forces in the columns and distribute these reaction forces as a uniform pressure over the cross-section of the columns. However, this is left as an exercise for the interested FEA analyst.

Next, you need to **define the distributed load** between axes B, C, 2, and 3.

1. Create a new **Load** called **UniformPressure**. This opens the **Create Load** dialog box. See Figure 1-3(a). Here
 - a. **Step** is set to **Load**
 - b. **Types for Selected Step** is set to **Pressure**.
 - c. Click **Continue**.
2. Select the region between axes B, C, 2, and 3 in the viewport. Click **Done** in the prompt area.



3. In the **Edit Load** dialog box, see Figure 1-3(b)
 - a. Set **Distribution** to **Uniform**
 - b. Set **Magnitude** to **- 0.01**.
 - c. Use **(Ramp)** as **Amplitude**.

Make sure that the load is applied in the downward direction in your model. This is done by choosing the correct surface for the choice Brown/Purple. If you end up with the wrong direction of the loading, you can also change the sign of your magnitude.

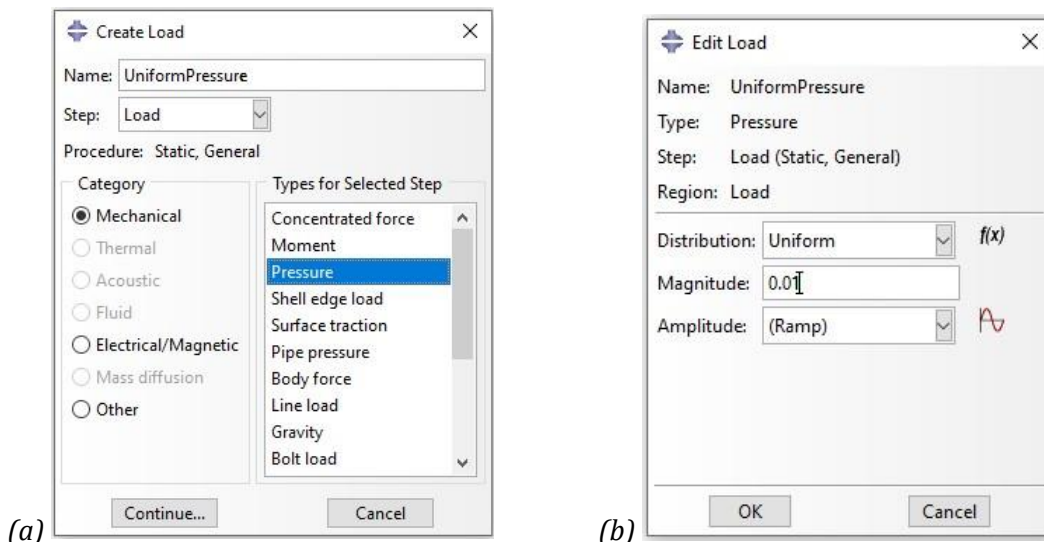


Figure 1-3 – (a) The Create Load dialog box and (b) The Edit Load dialog box.

Your model should look like Figure 1-4.

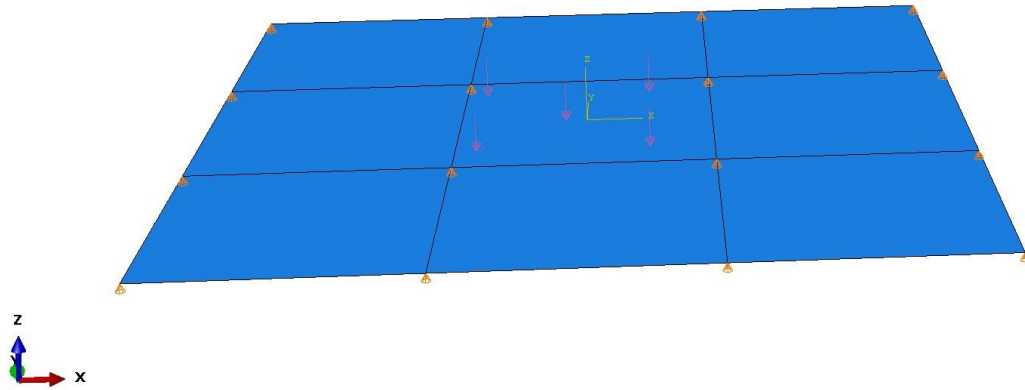


Figure 1-4 – The model with load and BC.

1.6. Meshing the Model

Navigate to the **Mesh module**. Make sure that **Object** is set to **Part**.

- From the tools on the left ribbon, choose **Mesh Controls**. In the **Mesh Controls** dialog box
 - For S4 and S8 elements: Use **Quad** as **Element type** and **Structured** as **Technique**. See Figure 1-8(a).
 - For S3 elements: Use **Tri** as **Element type** and **Structured** as **Technique**.
- From the tools on the left ribbon, choose **Assign Element Type**. See Figure 1-8(b). The following elements can be interesting to investigate:
 - S4: 4-node bi-linear rectangular shell element with full integration
 - S4R: 4-node bi-linear rectangular shell element with reduced integration
 - S8R: 8-node quadratic rectangular thick shell element with reduced integration
 - S3: 3-node linear triangular shell element for large deformations

You can read more about the different types of shell elements in the Abaqus documentation (see Section 29.6 in the Abaqus Analysis User's Guide). **Note** that Abaqus uses shell elements and not plate elements. A shell element is described by superposition of two contributions, i.e., the out-of-plane (bending) action and the in-plane (membrane) action. In general:

S3 and S4 elements could be used on both thin and thick shells, while the rest of the elements are specifically designed for either thick or thin shells. The membrane part of the S4 element is based on assumed strains (ANS = Assumed Natural Strains), which yields an element that is more robust and has higher accuracy when the element is irregularly shaped compared to a displacement-based element. In all of our meshes the elements are rectangular shaped such that we don't benefit from the fact that S4 is more robust.

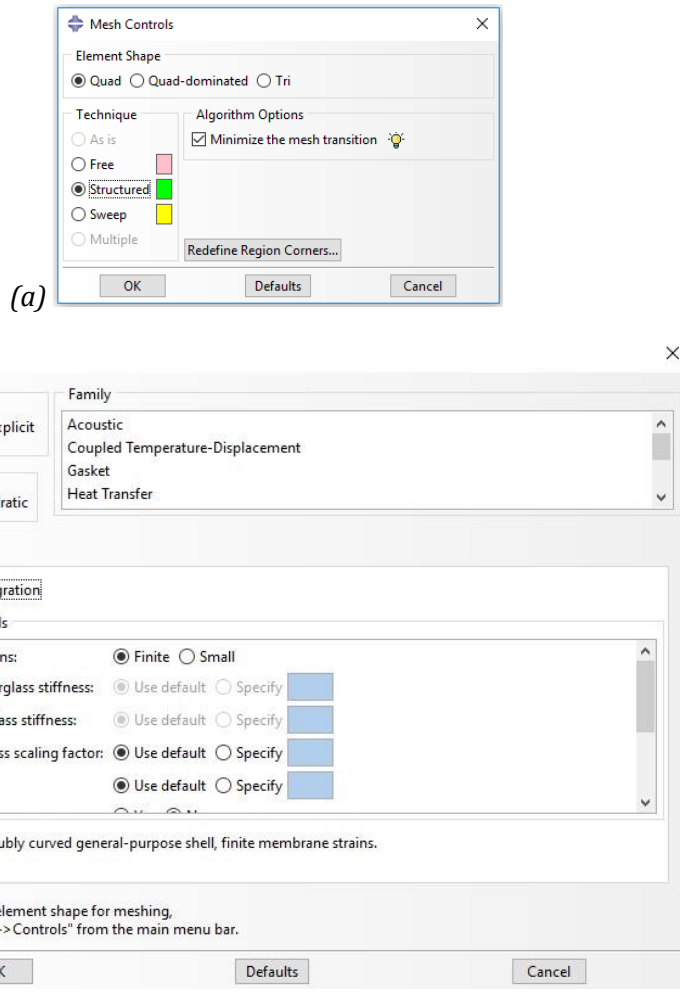


Figure 1-8 – (a) The Mesh Controls and (b) Element Type dialog box.

3. From the tools on the left ribbon, choose **Seed Part**. Use an **Approximate global size of 800**.
4. From the tools in the left ribbon, choose the **Mesh Part**.

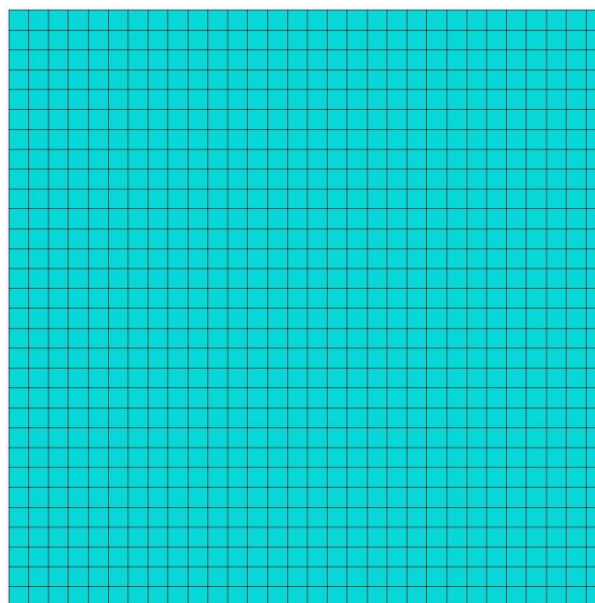


Figure 1-9 – Meshed part with quadratic shell elements.

1.7. Creating and Submitting an Analysis

The definition of the model **ROOFTOP** is now complete. Next, you will create and submit an analysis job to analyze the model.

In the **Create Job** dialog box, name the job **Deform** and select the model **ROOFTOP**. Click **Continue**.

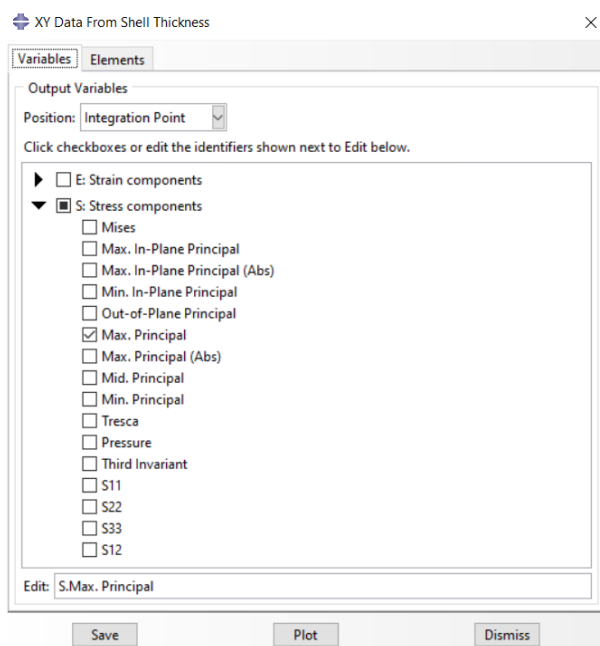
1.8. Post-processing the Results

Open a Result (.odb) File

Abaqus stores the requested output data in an ODB-file (ODB = Output Database). This file is saved in your work directory.

You can access the Field Output defined in Section 1.4 to show the stress distribution over the cross section of the critical element. Select **Tools - XY Data - Create** from the main menu bar, then choose **Thickness**.

The **XY Data from Shell Thickness** dialog box appears. Choose the field output variable(s) you want to show.



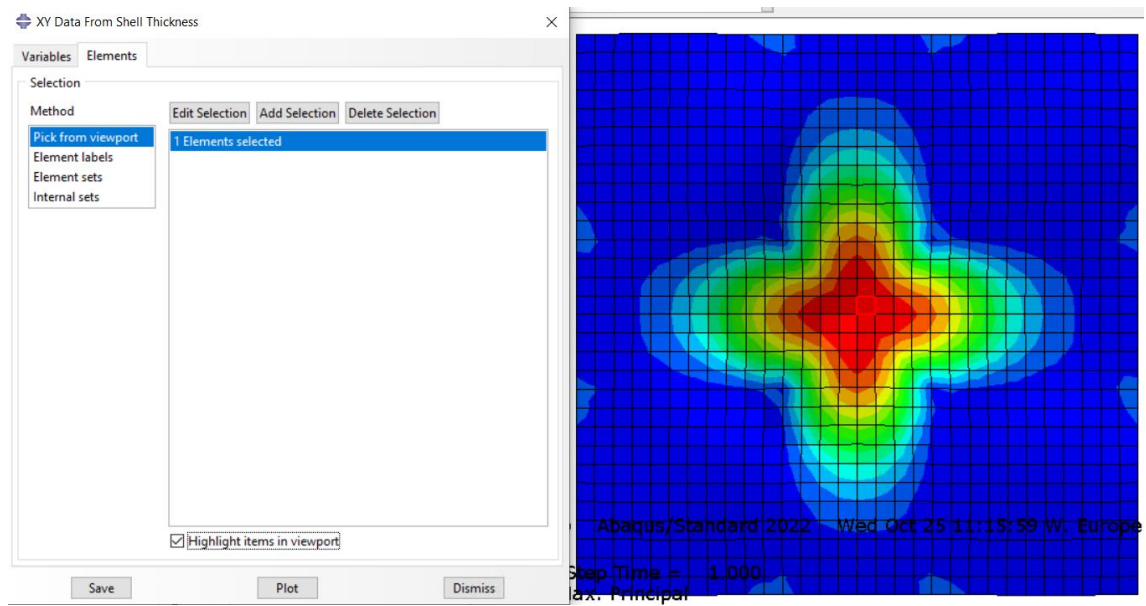
Click the **Elements** tab. The Elements options appear. Inspect critical elements by picking them directly from the viewport. Choose **Pick from viewport** from the **Selection Method** list.

Click on the critical element in the viewport and **Edit Selection**.

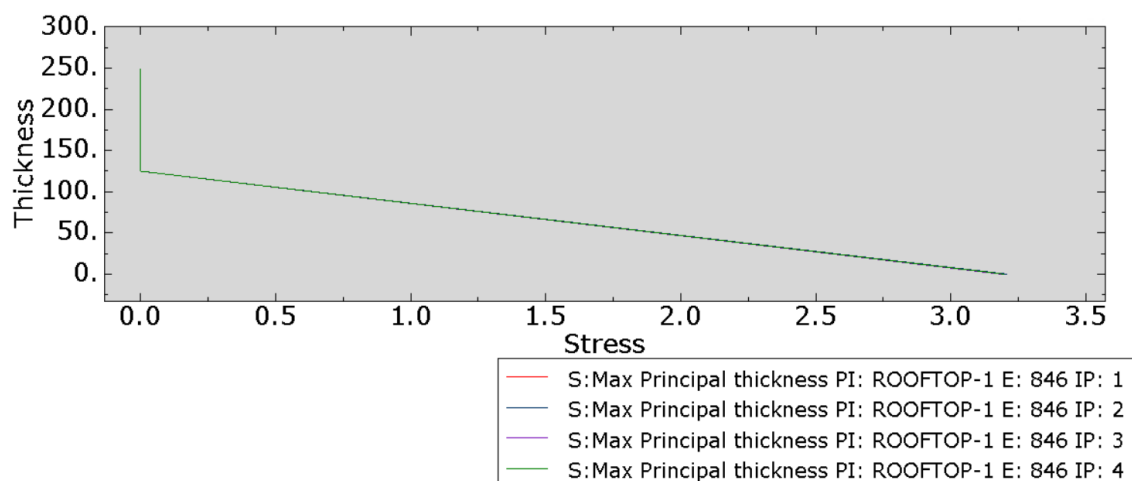
To evaluate and display the data, click **Plot**.

An X-Y plot appears in the current viewport. The plot represents the data you have configured in the dialog box, which Abaqus considers temporary data whether or not you have clicked Save to save it.

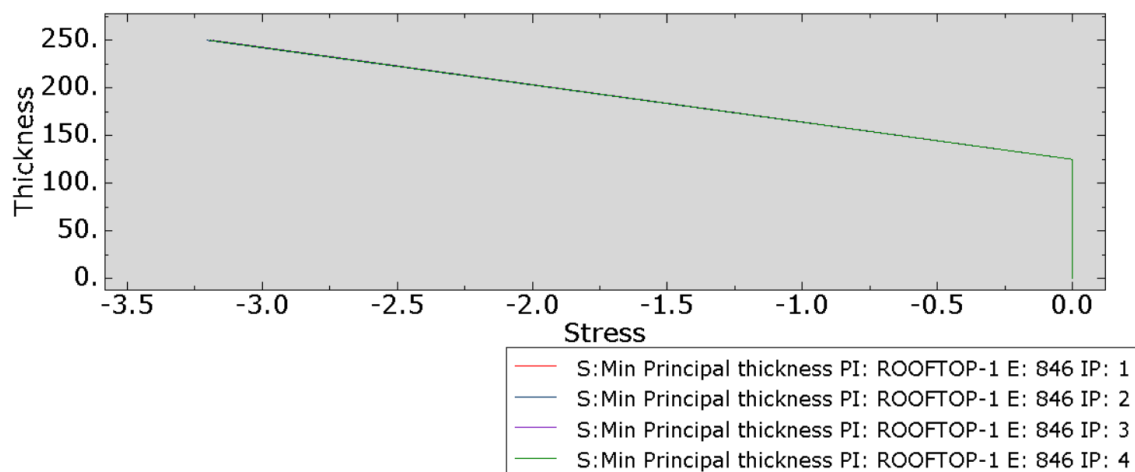
To save the data you have configured, click **Save**.



The stress (Max principal stress = $\max(\sigma_1, \sigma_2, 0)$) distribution over the cross-section in each integration point will look something like this:



Similarly, it will look something like this for the minimum principal stress ($= \min(\sigma_1, \sigma_2, 0)$):



2. Modelling the rooftop when the loading is represented as point loads

We will now replace the uniform load with points loads $P = 17.8 \text{ kN}$ representing the forces imposed on the rooftop by the car tires (see Figure 2).

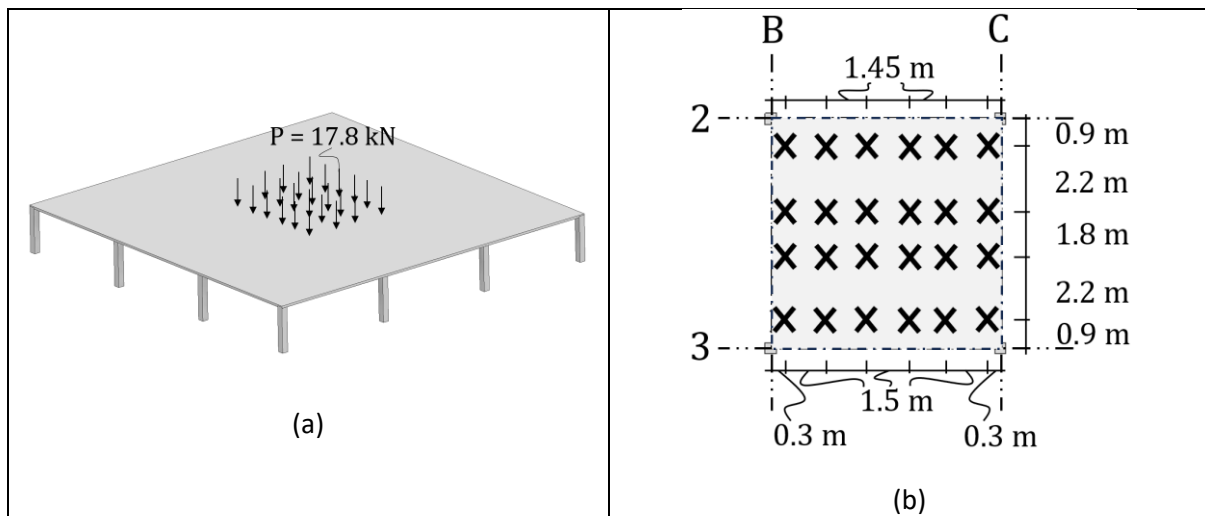


Figure 2 – A rooftop supported by 16 columns and exposed to 24 points loads $P = 17.8 \text{ kN}$ representing the forces imposed on the rooftop by the car tires.

First, create a new model by right-click on the previous **Rooftop** model in the **Model Tree**. Select **Copy Model** to create a new model. Name the model **RooftopPL**. Click **OK**. A new model is added to work **Model Tree**.

Then, we need to delete the **UniformPressure** loading from Task 1 and create the concentrated point loads (see Figure 2). There are at least three ways to approach the modelling of concentrated point loads:

- 1) Create more partition lines on the part Rooftop to enable the assignment of concentrated loads at the intersection of partitioning lines, i.e., we will have nodes corresponding to the location of each point load. This is a good alternative, but it will influence the mesh size and most probably result in a non-uniform mesh.
- 2) Create reference points at the positions of the point loads and apply concentrated loads at these points. You can create multiple reference points on the assembly. However, we then need to embed these reference points in our finite element model because the mesh is dependent on the part.
- 3) Use the reference points to identify nodes (already in our original mesh) closest to the location of the point load.

We choose the last one (alternative 3) because this is a good choice in terms of balancing modelling time and accuracy. However, it may not be the most accurate solution.

1. We start by creating reference points for the coordinates of each point load. In the **Assembly module**, choose **Tools – Reference points**.

Specify the coordinates of each point load in the prompt area. See Figure 2-1(a). If you have the extremities of your rectangle defined by $(-12000, -12000)$ and $(12000, 12000)$ as suggested in Section 1.1, the (x, y, z) coordinates will be given as follows

-3700, 3100, 0	-2200, 3100, 0	-750, 3100, 0	750, 3100, 0	2200, 3100, 0	3700, 3100, 0
-3700, 900, 0	-2200, 900, 0	-750, 900, 0	750, 900, 0	2200, 900, 0	3700, 900, 0
-3700, -900, 0	-2200, -900, 0	-750, -900, 0	750, -900, 0	2200, -900, 0	3700, -900, 0
-3700, -3100, 0	-2200, -3100, 0	-750, -3100, 0	750, -3100, 0	2200, -3100, 0	3700, -3100, 0

This will result in 24 reference points corresponding to the positions of the point loads. See Figure 2-1(b).

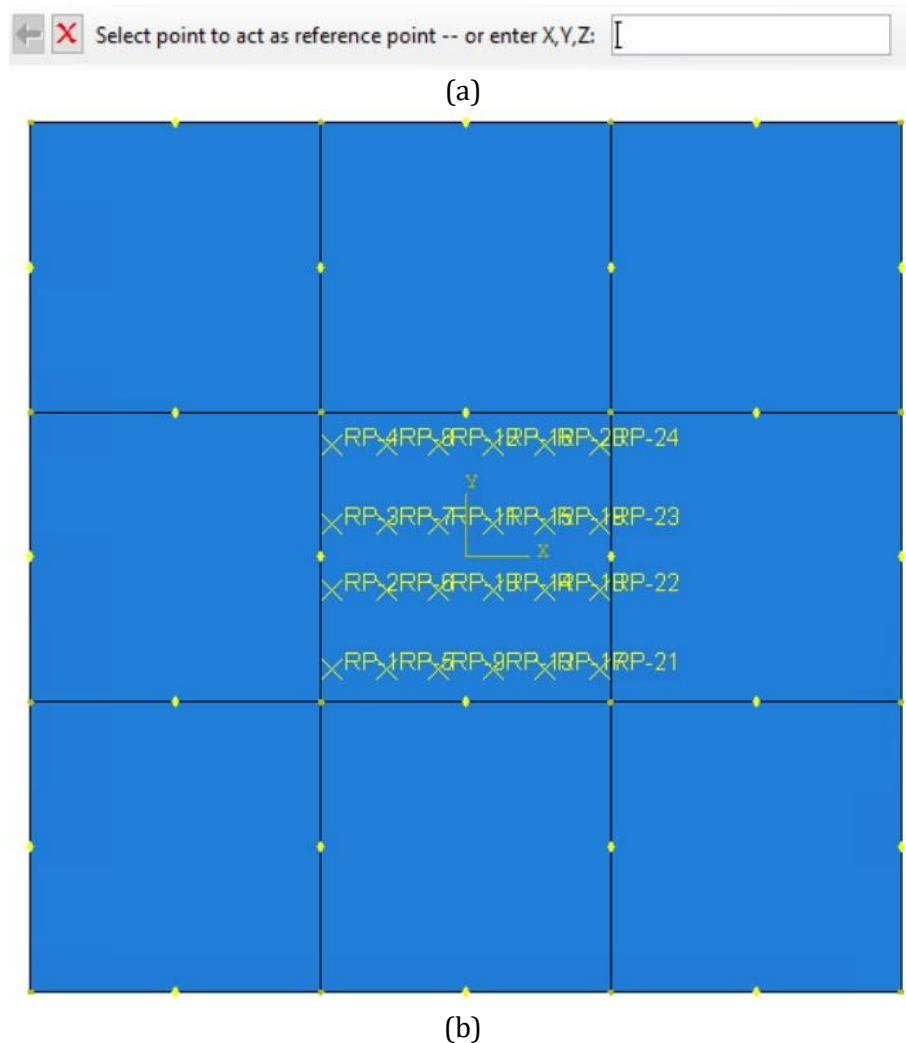


Figure 2-1 – Reference points corresponding to the positions of the point loads.

2. Move to the **Mesh** module and choose **Tools – Set – Create**. See Figure 2-2(a).

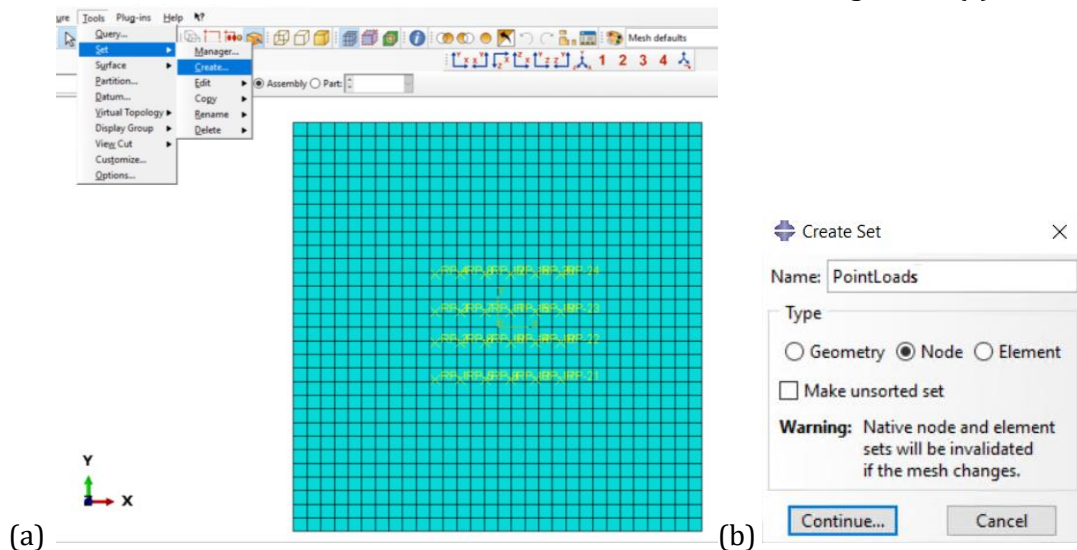


Figure 2-2 – Create new set for nodes representing the position of the point loads.

- Choose all the nodes closest to the reference points and name the set **PointLoads**. See Figure 2-3.
- Use type **Node** (see Figure 2-2(b)) and click **Done** in the prompt area. **NB: Object** needs to be selected as **Assembly** because the reference points are created in the Assembly module.

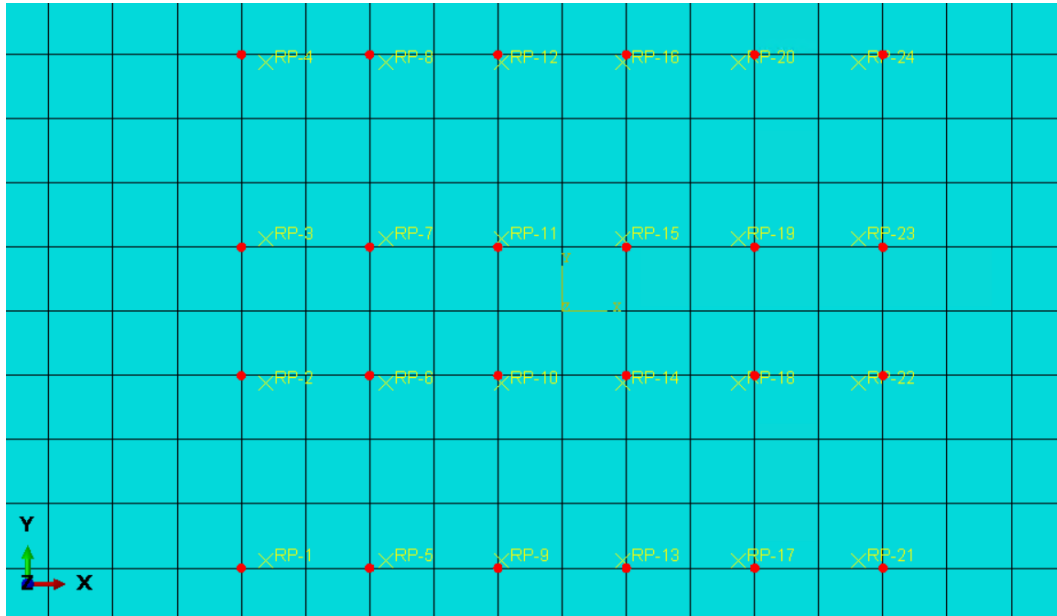


Figure 2-3 – Nodes to be selected (marked in red) closest to the reference points representing the position of the point loads.

- In the **Load Module**, delete the **UniformPressure** loading and create a new load called **PointLoads**.
- When defining the load, use **Types for Selected Step: Concentrated force** in the **Create Load** dialog box. See Figure 2-4.

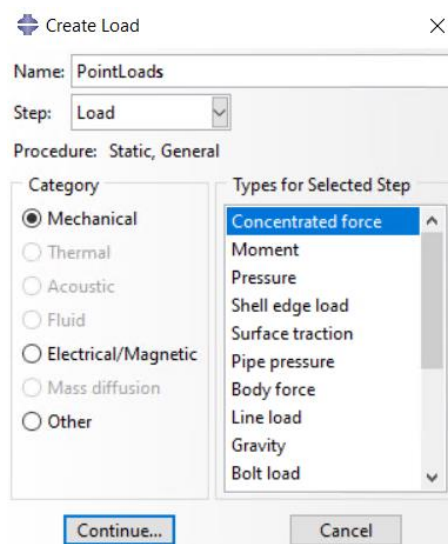


Figure 2-4 – The Create Load dialog box.

7. In the prompt area (see Figure 2-5), open the predefined sets by clicking the button **Sets...**

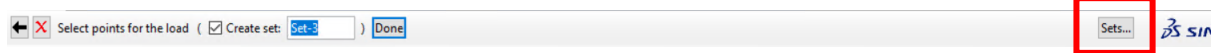


Figure 2-5 – Prompt area and access to predefined sets (button inside red box).

8. Choose the predefined set **PointLoads** and click **Continue**. See Figure 2-6(a).
9. In the **Edit Load** dialog box, define the concentrated load CF3. Use a sign as the load acts in the negative z-direction. See Figure 2-6(b).

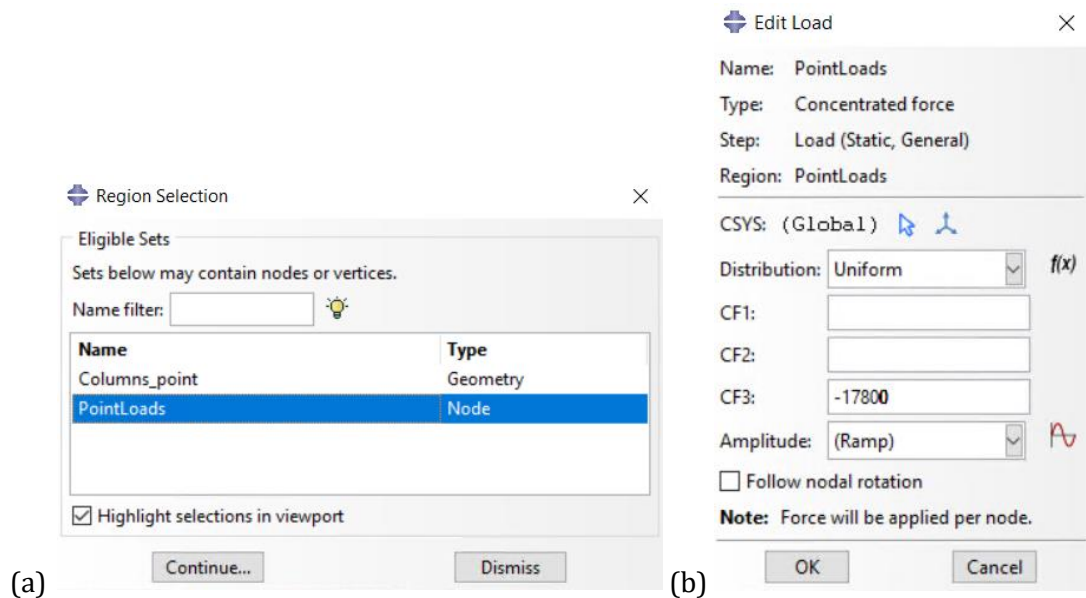


Figure 2-6 – Region Selection (a) and Edit Load (b) dialog box.

10. Your model should look like Figure 2-7 (zoomed view in the x-y plane) and Figure 2-8 (perspective view).

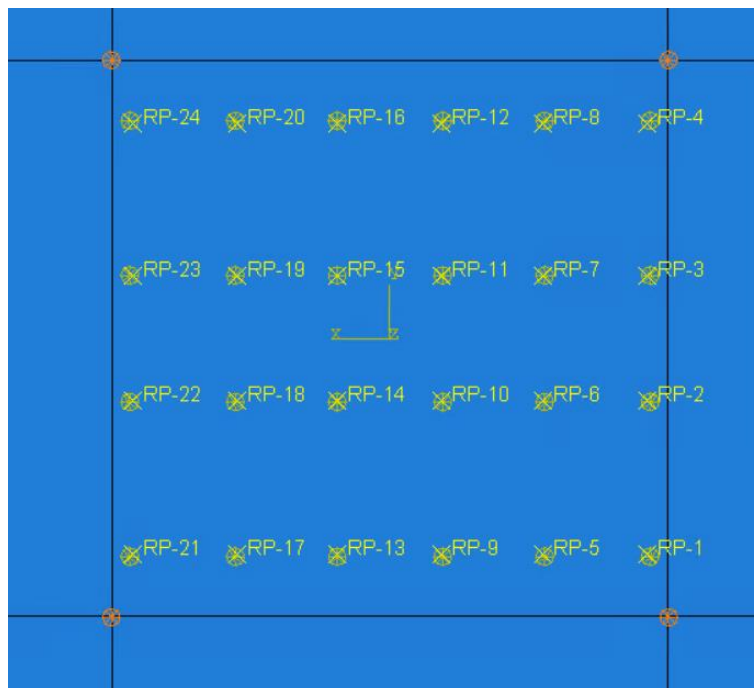


Figure 2-7 – Zoomed view of the model in the x-y plane.

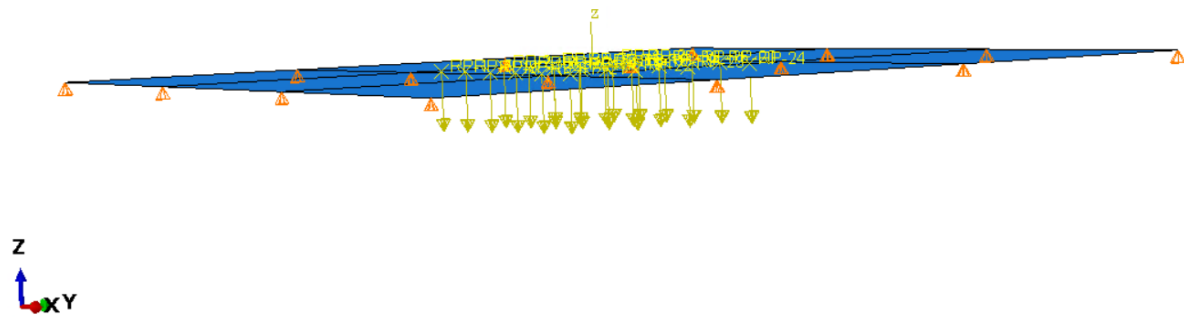


Figure 2-8 – Perspective view of the model.

We see that the concentrated loads are very close to the reference points (Figure 2-7).

11. In the **Job module**, create a new job called **DeformPL**. **Submit** the job.
12. Voilla! You can now postprocess the results.