

TKT4142 Finite Element Methods in Structural Engineering

Workshop 1

The aim of this workshop is to give an introduction to Abaqus and to provide a framework for how you may establish a simple linear static numerical model. A wooden beam will be used for this purpose. This problem is a part of Case Study 1 in the course TKT4142 – Finite Element Methods in Structural Engineering and is based on the finite element software Abaqus 2022.

Section 1 addresses some important information about Abaqus. The beam is first modeled as a cantilever beam in Section 2 using 2D plane stress elements before we increase the complexity of the model to also include the hole at the steel rod support in Section 3.

1. Preliminaries

1.1. Units of Measurements in ABAQUS

There are **no units of measurements** in Abaqus, which implies that the user must be consistent with the inputs of parameters. In Figure 1-1 below, consistent systems of units are listed. We will sketch in millimeters and let the time be in seconds. This is convenient since the force and stress are then measured in N and MPa, respectively.

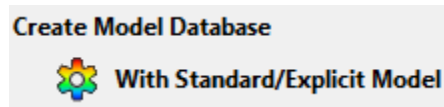
| MASS | LENGTH | TIME | FORCE | STRESS | ENERGY | Steel Density | Steel Modulus | G - Gravity Constant |
|------------------------|--------|------|-------|----------------------|----------|---------------|---------------|----------------------|
| kg | m | s | N | Pa | Joule | 7.83E+03 | 2.07E+11 | 9.81 |
| kg | mm | ms | kN | Gpa | kN-mm | 7.83E-06 | 2.07E+02 | 9.81E-03 |
| g | cm | s | dyne | dyne/cm ² | erg | 7.83E+00 | 2.07E+12 | 9.81E+02 |
| g | cm | us | 1e7N | Mbar | 1e7 N-cm | 7.83E+00 | 2.07E+00 | 9.81E-10 |
| g | mm | s | 1e-6N | Pa | 1e-9 J | 7.83E-03 | 2.07E+11 | 9.81E+03 |
| g | mm | ms | N | Mpa | N-mm | 7.83E-03 | 2.07E+05 | 9.81E-03 |
| ton | mm | s | N | Mpa | N-mm | 7.83E-09 | 2.07E+05 | 9.81E+03 |
| lbf-s ² /in | in | s | lbf | psi | lbf-in | 7.33E-04 | 3.00E+07 | 3.86E+02 |
| slug | ft | s | lbf | psi | lbf-ft | 1.52E+01 | 4.32E+09 | 32.2 |

Figure 1-1 – Units in ABAQUS.

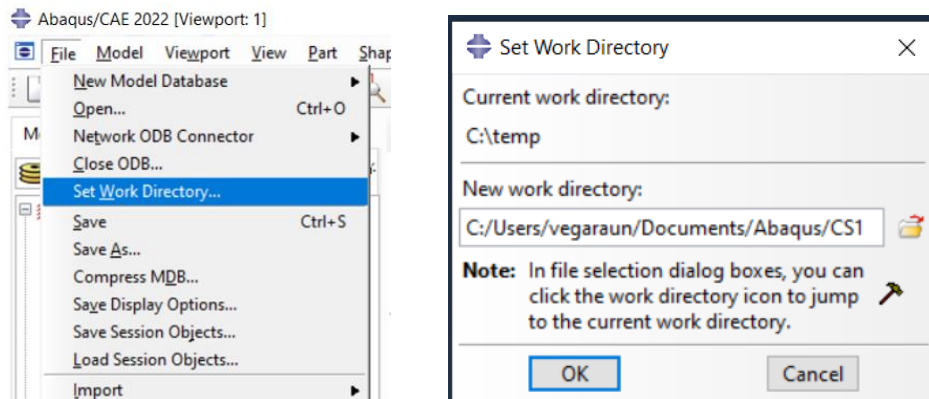
1.2. Set the Work Directory

It is important to define the working directory after opening Abaqus/CAE. An Abaqus analysis generates a lot of different files, and these files will be saved in your working directory. If the working directory is not defined, these files will be saved in a temporary folder located at “C:\temp”. This is not a very convenient folder to save your files. Good FEA practice is to have a separate folder for each of your simulations due to the large number of files generated.

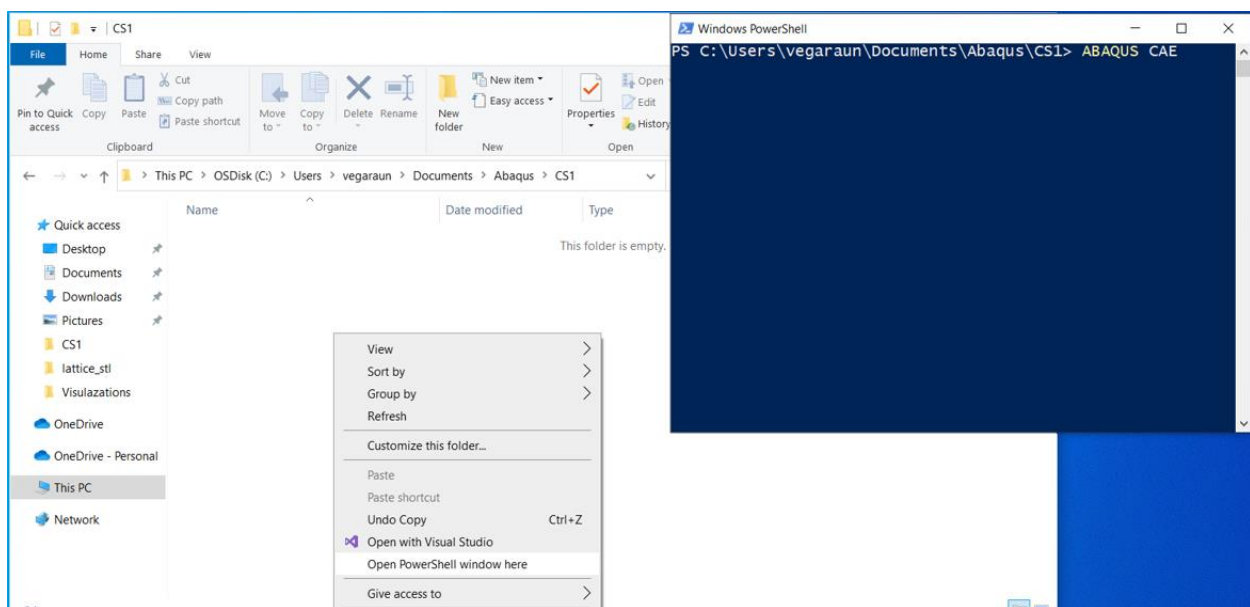
When you open Abaqus/CAE, select **Create Model Database with Standard/Explicit Model** from the **Start Session** dialog box.



Select **File** → **Set Work Directory** from the main menu bar. In the **Set Work Directory** dialog box, enter your **new work directory**. It is possible to search for the directory using the Select (📁) option.



Note: An **alternative way** of launching Abaqus/CAE is to use the **command prompt**. Navigate to your work folder in the File Explorer. Then hit **Shift + Right Click** and select **Open PowerShell window here**. Type **ABAQUS CAE** in the command window and hit **Enter**. This will open Abaqus/CAE and automatically set your work directory.



1.3. Modelling Procedure

Various modules are required for making a numerical model in Abaqus. The required modules for modeling the cantilever beam are listed below. The keywords are presented in chronological order from a modeling perspective of view. This tutorial will follow the same order.

1. Part
 - Create the unique parts you need in your simulation.
 - Define sets for your part.
2. Property
 - Define the material property.
 - Define and assign sections to parts or regions.
3. Assembly
 - Gather your parts as instances and position them in space relative to each other. An instance is essentially a realization of a part: the same part may be included several times as different instances. Notice that we only deal with one part in this workshop.
 - Define sets for your assembly.
4. Step
 - Define the analysis steps and output requests.
5. Interaction
 - Create interactions/contacts and constraints between parts. This module is not applicable for this workshop.
6. Load
 - Apply loads and boundary conditions to regions or named sets and assign them to steps in the analysis history.
7. Mesh
 - Mesh your part or assembly.
8. Job
 - Submit, manage, and monitor analysis jobs.
9. Visualization
 - Post-process the results. Visualize the results and export data.

You may notice that the same modules are listed in the **Module drop-down menu**. See Figure 1-2. This makes it easy to navigate between the different modules. By following the bullet points successively, you never have to go back. You can access what you have done in the **Model Tree**.

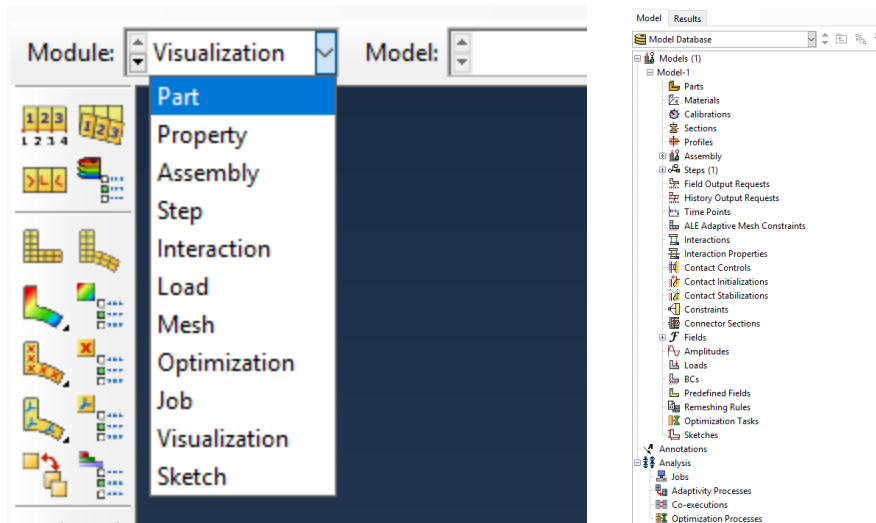






Figure 1-2 –Module drop-down menu and Model Tree.

1.4. Save Your Model

To save the model database, select **File** → **Save As** from the main menu bar and type the **File Name** in the **Save Model Database As** dialog box. Click **OK**. The **.cae** extension is added to the file name automatically.

1.5. Some Other Tips and Tricks

The documentation of the software (**Help** → **Search and browse guides**) contains large amounts of information and theory and is highly recommended to use actively. To rotate, pan and magnify your model:

- Rotate  : Ctrl + Alt + Left Mouse Button
- Move Sideways (Pan)  : Ctrl + Alt + Middle Mouse Button
- Magnify  : Scroll the wheel
- Perspective on/off 

The **view** toolbar is useful when navigating in the model view. To access it, go to **View** → **Toolbars** → **Views**.

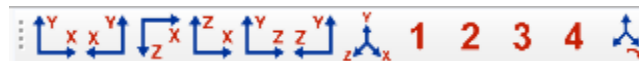


Figure 1-3 – The view toolbar.

2. Modelling the Cantilever Beam Using 2D Plane Stress Elements

The cantilever beam with dimensions and applied load is shown in Figure 2-1.

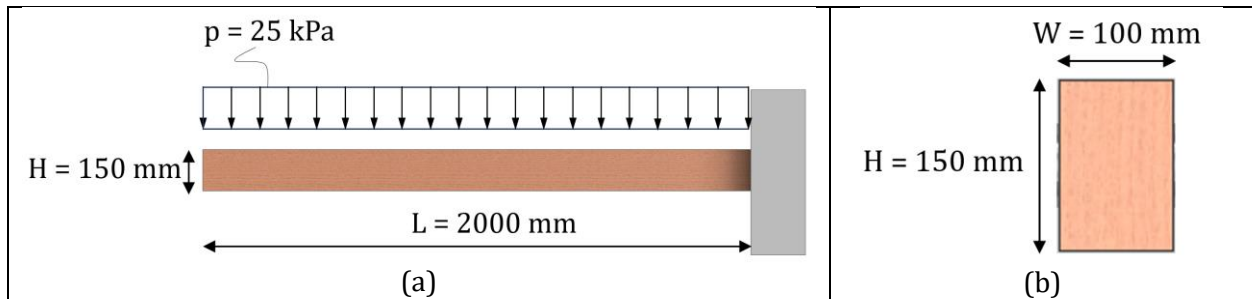


Figure 2-1 – Cantilever beam.

Load: $p = -0.025 \text{ N/mm}^2$ (Downwards)

Material data: $E = 10\,000 \text{ N/mm}^2$, $\nu = 0.30$, $\rho = 500 \text{ kg/m}^3$, $\sigma_y = 20 \text{ N/mm}^2$

Make sure your work directory is properly set as explained in Section 1.2.

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

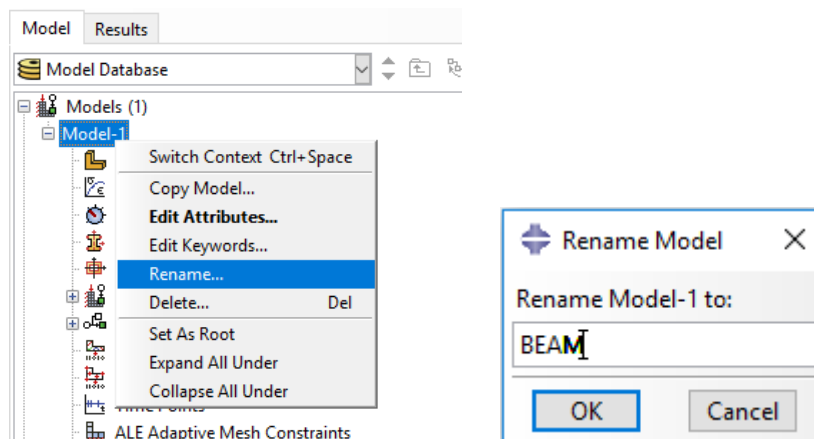


Figure 2-2 – Rename the model.

2.1. Creating a Part

In this section, you will create a 2D model of the beam by sketching the two-dimensional profile of the beam (a rectangle) as shown in Figure 2-1(a).

1. Create a new part in **BEAM_2D** by double-click on **Parts** in the **Model Tree**. The **Create Part** dialog box appears.
2. In the **Create Part** dialog box:
 - a. Name the part **Beam**.
 - b. Choose **2D Planar** in the **Modeling Space**.
 - c. Choose **Deformable** as the **Type**.

- d. Use **Shell** as the **Base Feature**.
- e. **Approximate size** can be set to **4000**.

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

Abaqus/CAE displays text in the **prompt area** near the bottom of the window to guide you through the procedure, as shown in Figure 2-3. Click the **cancel** button to cancel the current task; click the **backup** button to cancel the current step in the task and return to the previous step.

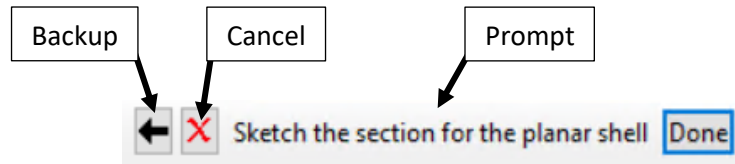


Figure 2-3 – Prompt area

- 3. Sketch a 2000 by 150 rectangle by following the next steps.
- 4. To sketch the profile of the cantilever beam, you need to select the **rectangle drawing tool**, as shown in Figure 2-4.

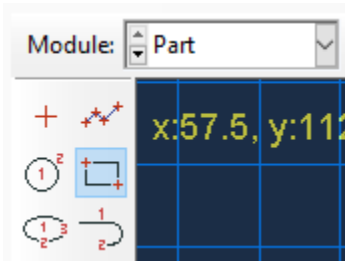


Figure 2-4 – Rectangle sketch tool.

- 5. In the viewport, sketch the rectangle using the following steps:
 - a. You will first sketch a rough approximation of the beam and then use constraints and dimensions to refine the sketch. Select any two points as the opposite corners of the rectangle.
 - b. Click the mouse **Esc** or **Cancel** (X) in the prompt area to exit the rectangle tool.
 - c. The Sketcher automatically adds constraints to the sketch (in this case the four corners of the rectangle are assigned perpendicular constraints and one edge is designated as horizontal).
 - d. Use the dimension tool () to dimension the top and left edges of the rectangle. The top edge should have a horizontal dimension of **2000** mm, and the left edge should have a vertical dimension of **150** mm. See Figure 2-1. When dimensioning each edge, simply select the line, Left-click to position the dimension text, and then enter the new dimension in the prompt area.
 - e. The final sketch is shown in Figure 2-5.

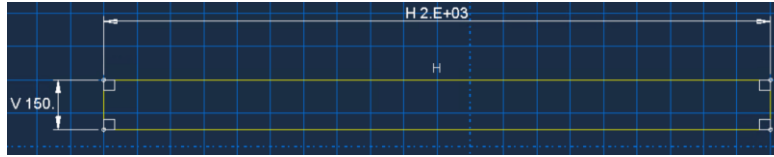


Figure 2-5 – Sketch of the rectangle.

6. Click **Done** in the prompt area to exit the sketcher.
7. Abaqus/CAE displays an isometric view of the new part, as shown in Figure 2-6(a). The Part **Beam** is now added to the **Model Tree** in Figure 2-6(b). If you want to make any changes to the sketch, double-click on **Section Sketch** (See Figure 2-6(b)).

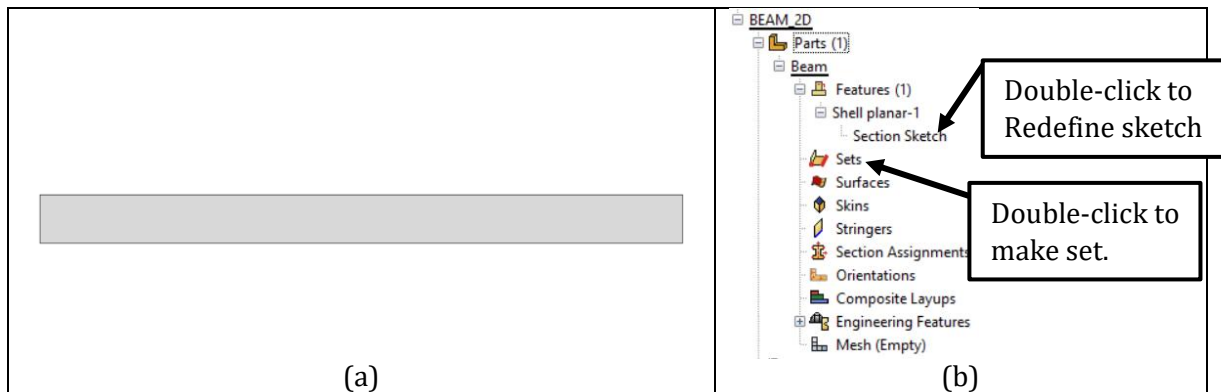


Figure 2-6 – Planar part and Model Tree.

8. It is often convenient to define particular regions of the model into different sets. Loads and BC can then be added to these sets. It is also possible to extract values (Force/Displacement/etc.) from a set using History Output.
9. In the **Model Tree** under BEAM → Parts → Beam, double-click on **Sets** (See Figure 2-6(b)). Name this set **Fixed** and click **Continue...** in the **Create Set** dialog box. Select the right edge of the model, as shown in Figure 2-7. Use the **Rotate View** tool (🔄) in order to rotate the model.

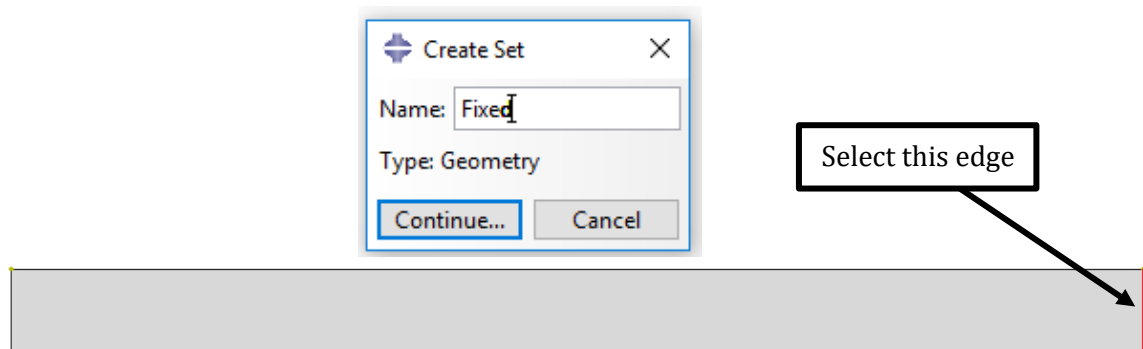


Figure 2-7 – Create Set dialog box and selected edge for the set "Fixed".

2.2. Assigning Properties (Material and Section) to the Model

We will create a single elastic material with Young's modulus of 10 000 MPa and Poisson's ratio of 0.3. Next, we will create a homogenous solid section and assign it to the beam.

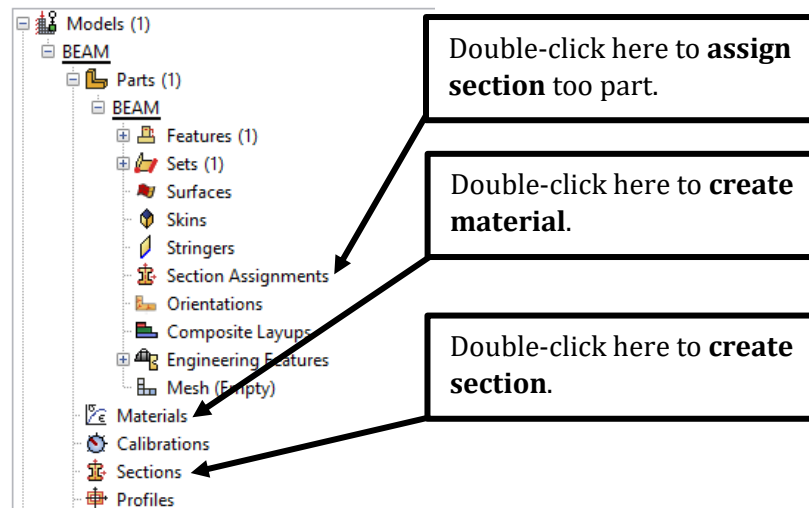


Figure 2-8 – The Model Tree.

Creating a Material

1. In the **Model Tree**, double-click **Materials** to create a new material in the model BEAM. See Figure 2-8. Notice that Abaqus/CAE switches to the Property module and the **Edit Material** dialog box appears.
2. In the **Edit Material** dialog box, name the material **Wood**. Notice the various options available in this dialog box.
3. From the material editor's menu bar, select **Mechanical** → **Elasticity** → **Elastic**, as shown in Figure 2-9(a).
Abaqus/CAE displays the **Elastic** data form.
4. Enter a value of **10000** for **Young's Modulus** and a value of **0.3** for **Poisson's Ratio** in the respective fields, as shown in Figure 2-9(b).
5. Click **OK** to exit the material editor.

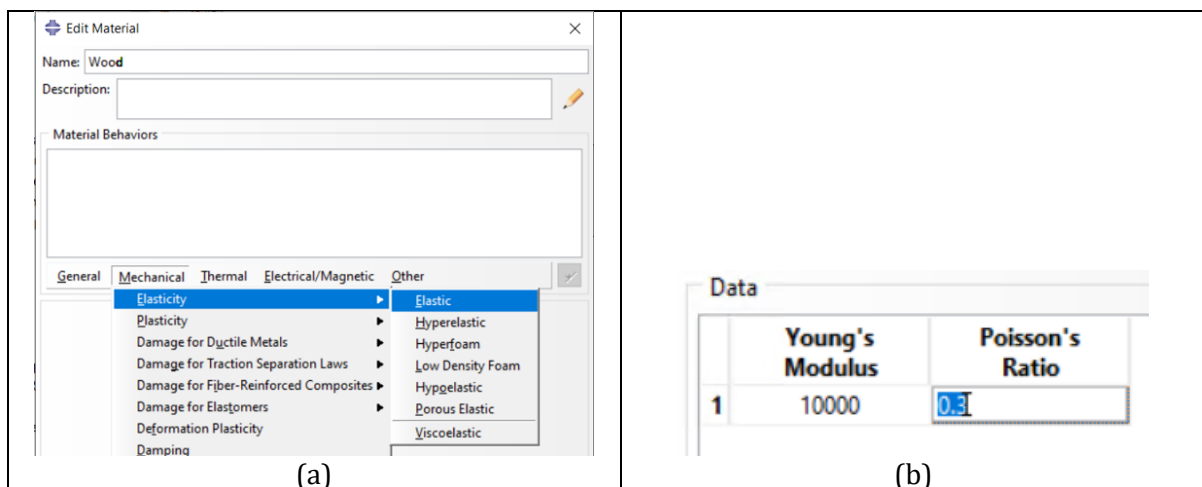


Figure 2-9 – (a) The Edit Material dialog box and (b) material editor.

Creating a Homogeneous Solid Section

1. In the Model Tree, double-click **Sections** to create a new section in the model BEAM. See Figure 2-8.
The **Create Section** dialog box appears, shown in Figure 2-9(a).
2. In the **Create Section** dialog box:
 - a. Name the section **BeamSection**.
 - b. Accept the default category **Solid** and the default type **Homogeneous**.
 - c. Select **Continue**.
 The **Edit Section** dialog box appears, shown in Figure 2-9(b).
3. In the **Edit Section** dialog box:
 - a. Accept the default selection of **Wood** for the **Material** associated with the section.
 - b. Set the **Plane stress/strain thickness** to **100**. See Figure 2-12(b).
Note: This value is the width of the beam (see Figure 2-1(b)).
 - c. Click **OK**.

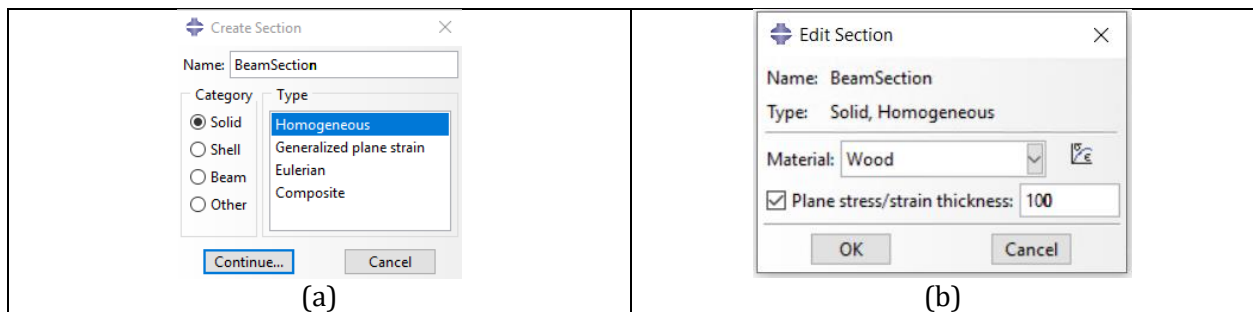


Figure 2-10 – (a) the Create Section dialog box and (b) the Edit Section dialog box.

Assign the Section to the Part

1. In the Model Tree, expand the branch for the part **Beam**. Double-click **Section Assignments** to assign a section to the part Beam, as shown in Figure 2-8. Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.
2. Click anywhere on the beam to select the entire part as the region to which the section will be assigned, as shown in Figure 2-11(a).
3. In the prompts, check **Create set**. See Figure 2-11(b). Call this set **Whole** and click **Done** in the prompt area to accept the selected geometry. This will make a new set called **Whole** and contains the whole part. This set can be found in the **Model Tree** together with the set made in Section 2.1.
The **Edit Section Assignment** dialog box appears, as shown in Figure 2-12(a).
4. In the **Edit Section Assignment** dialog box, accept the default selection of **BeamSection** as the section definition, and click **OK**. Make sure that **Thickness Assignment** is set to **From section**, Figure 2-14(a).
Abaqus/CAE colors the beam green to indicate that the section has been assigned, as shown in Figure 2-12(b).

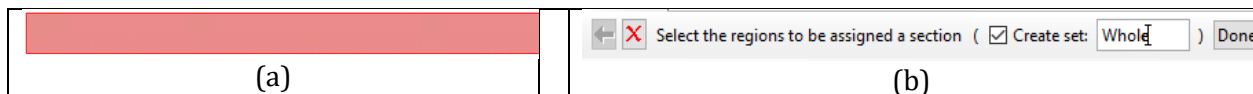


Figure 2-11 – (a) The selected geometry and (b) the prompt area.

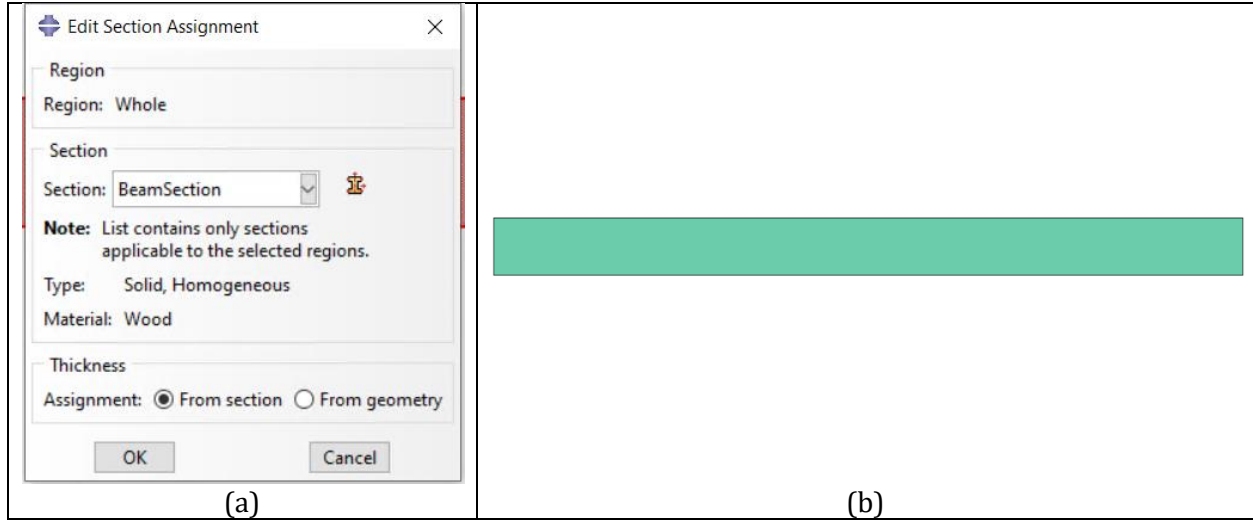


Figure 2-12 – (a) The Edit Section Assignment dialog box and (b) the part after a section is assigned.

2.3. Create an Assembly

The assembly for this analysis consists of a single instance of the part **Beam**.

Assemble the Model:

1. In the **Model Tree**, expand the branch for the **Assembly** of the model **BEAM** and double-click **Instances** to create a new part instance as shown in Figure 2-13(b).
Abaqus/CAE switches to the Assembly module, and the **Create Instance** dialog box appears. See Figure 2-13(a).
2. In the **Create Instance** dialog box, select **Beam**. Choose the instance type **Dependent (mesh on part)** and click **OK**.
Abaqus/CAE displays the new part instance in the viewport.

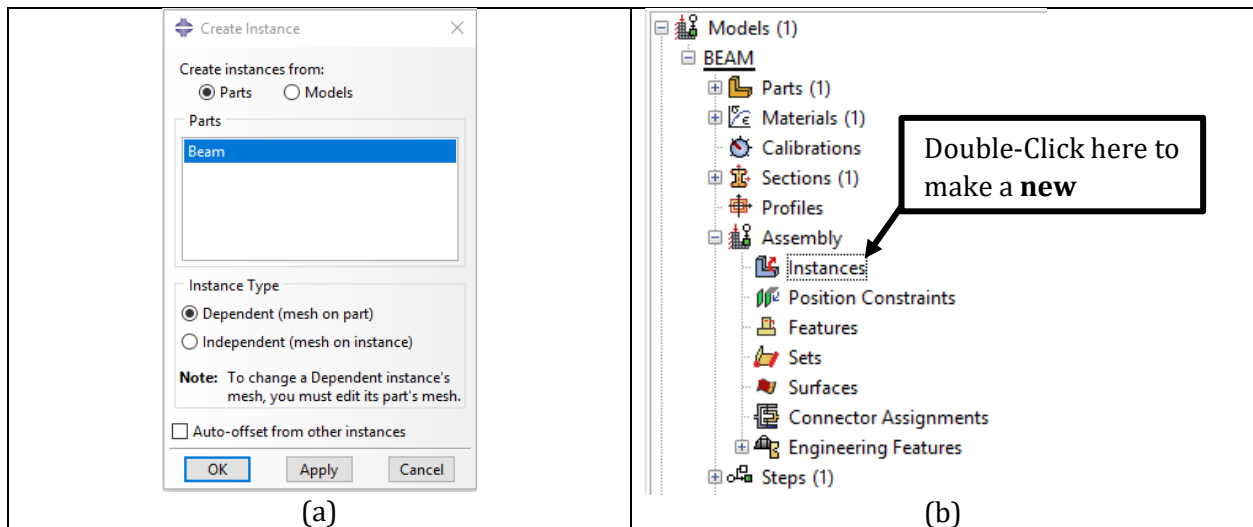


Figure 2-13 – (a) The Create Instance dialog box and (b) The Model Tree.

2.4. Defining the Step and Output data

In this simulation, we are interested in the static response of the cantilever beam to a pressure load applied over the beam top. This is a single event, so only a single analysis step is needed for the simulation. Consequently, this model will consist of two steps:

- An initial step, in which you will apply a boundary condition that constrains one end of the cantilever beam.
- A general, static analysis step, in which you will apply a pressure load to the top edge of the beam.

Abaqus/CAE generates the initial step automatically, but you must create the analysis step yourself.

Create a General, Static Analysis Step

1. In the **Model Tree**, double-click **Steps** to create a new step in the model **BEAM**. See Figure 2-14 (b).

Abaqus/CAE switches to the Step module, and the **Create Step** dialog box appears, shown in Figure 2-14(a).

2. In the **Create Step** dialog box:
 - a. Name the step **BeamLoad**.
 - b. From the list of available general procedures in the **Create Step** dialog box, select **Static, General** if it is not already selected.
 - c. Click **Continue**.

The **Edit Step** dialog box appears. See Figure 2-15.

3. In the **Edit Step** dialog box:
 - a. In the **Time period** field of the **Basic** tab page, enter **1**. Let **Nlgeom** (Non-linear geometry) be turned off. See Figure 2-15(a).
 - b. Click the **Incrementation** tab, and delete the value of **1** that appears in the **Initial** text field. Type a value of **0.1** for the initial increment size. See Figure 2-15(b).
 - c. You can accept the default values provided in the **Other** tab.
 - d. Click **OK** to create the step and to exit the step editor.

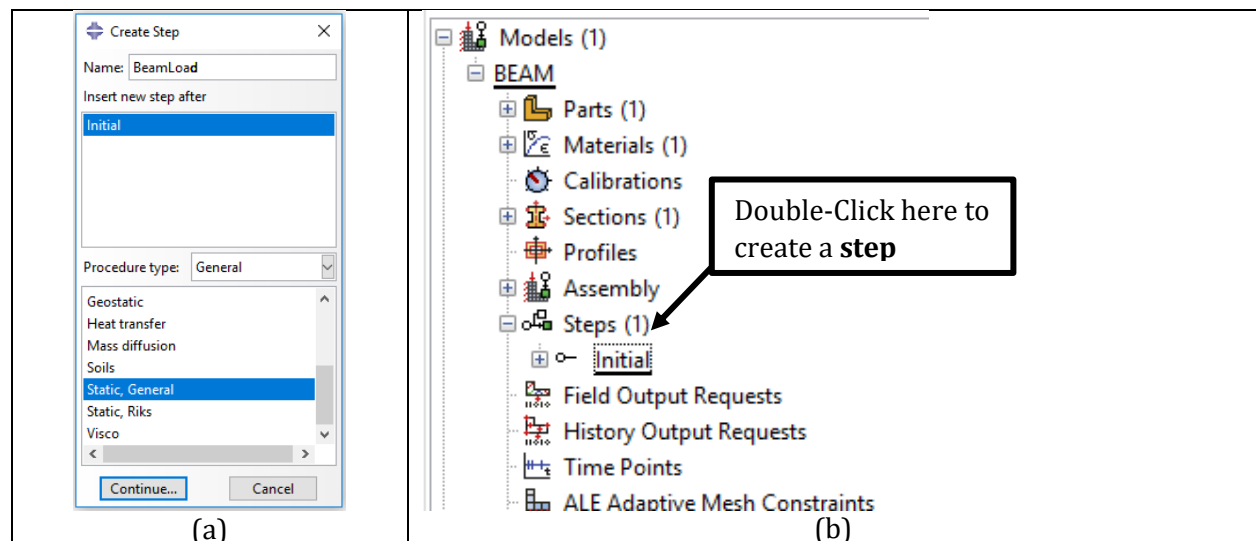


Figure 2-14 – (a) The Create Step dialog box and (b) the Model Tree.

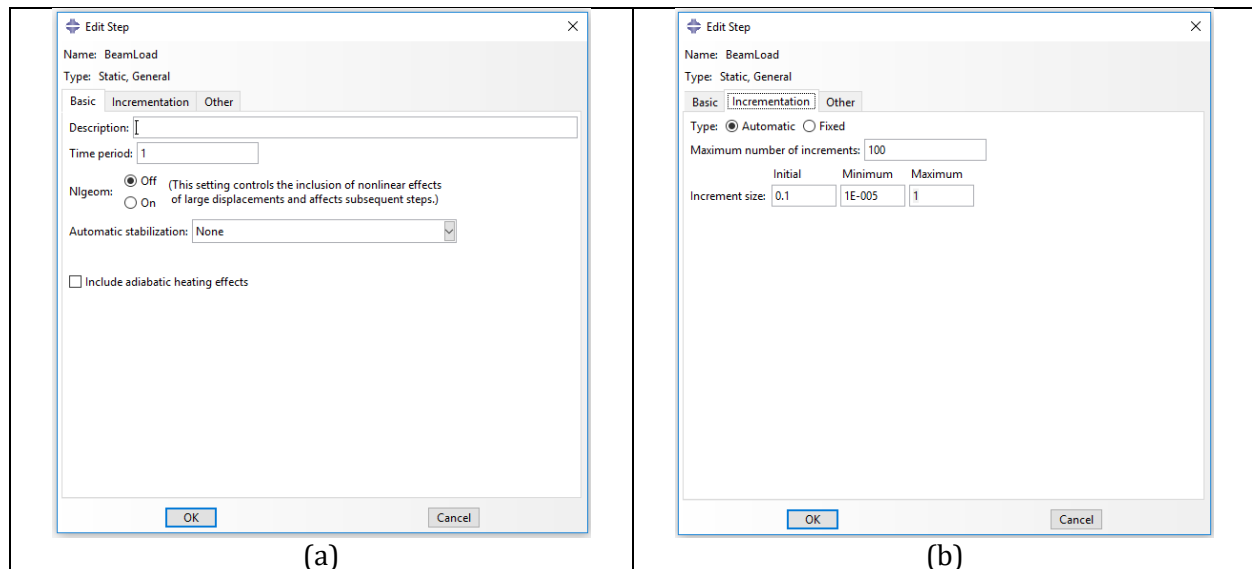


Figure 2-15 – The Edit Step dialog box. (a) Basic tab and (b) increment tab.

Output Requests

The **Field Output Requests** and the **History Output Requests** in the model tree defines which data (i.e., stress, strain, displacement, temperature, etc.) that 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.

2.5. Applying Load and Boundary Condition to the Model

Next, you will define the boundary condition and loading that will be active during the **BeamLoad** step.

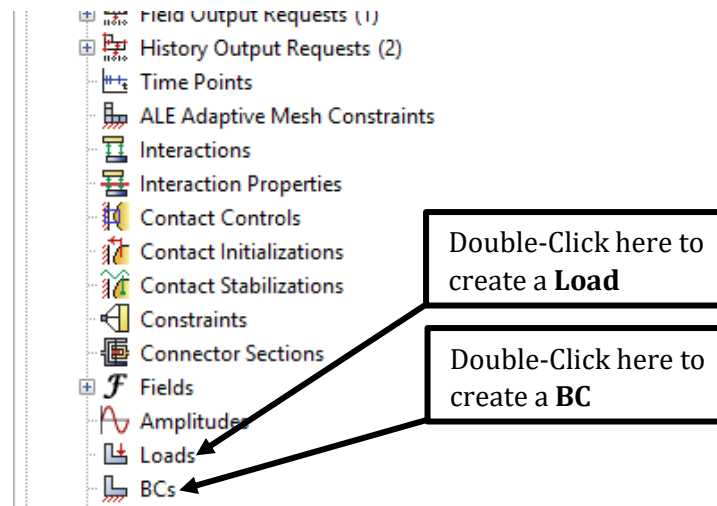


Figure 2-16 – The Model Tree.

Apply Boundary Condition to One End of the Cantilever Beam

1. In the **Model Tree**, double-click BCs to create a new boundary condition in the model **BEAM**. Abaqus/CAE switches to the Load module, and the **Create Boundary Condition** dialog box appears, see Figure 2-17(a).
2. In the **Create Boundary Condition** dialog box:
 - a. Name the boundary condition **Fixed**.
 - b. Select **Initial** as the step in which the boundary condition will be activated.
 - c. In the **Category** list, accept the default category selection **Mechanical**.
 - d. In the **Types for Selected Step** list, select **Displacement/Rotation** as the type.
 - e. Click **Continue**.

The **Region Selection** dialog box will appear, Figure 2-17(b). If not, click on **Sets** (Sets...) in the prompt area to access this dialog box.

3. Select the set **Beam-1.Fixed** in the **Region Selection** dialog box. This set is the same as the set created in Section 2.1. Click **Continue**.

The **Edit Boundary Condition** dialog box appears, as shown in Figure 2-18.

4. In the **Edit Boundary Condition** dialog box:
 - a. Toggle on **U1** and **U2** since only the translational degrees of freedom need to be constrained (the beam will be meshed with 2D elements later, which has no rotational degrees of freedom at its nodes.). This will fix the displacement components at the end of the beam.
 - b. Click **OK** to create the boundary condition definition and to exit the editor.

Abaqus/CAE displays arrows at each corner and midpoint on the selected face to indicate the constrained degrees of freedom.

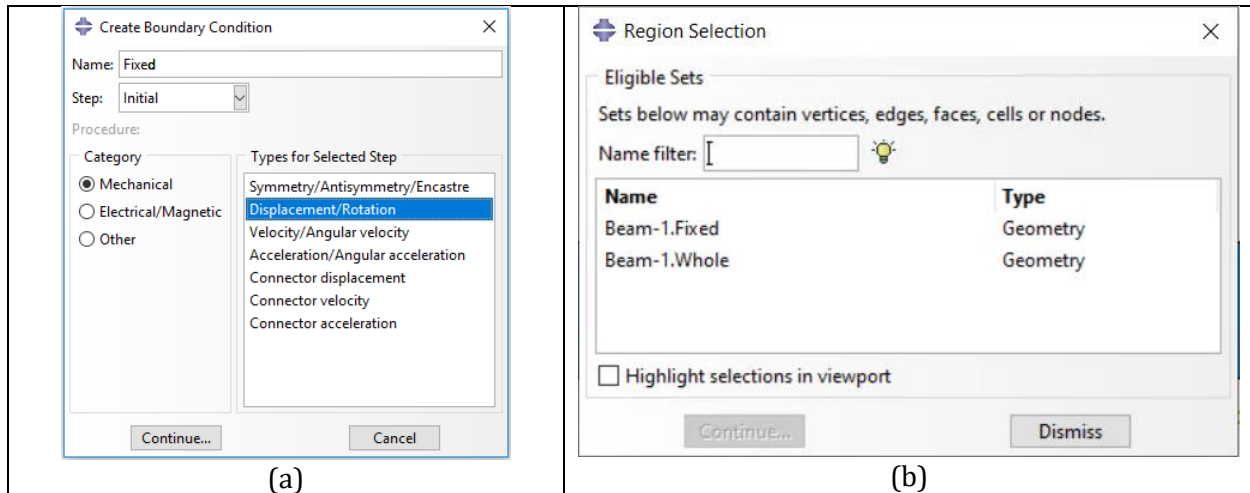


Figure 2-17 – (a) The Create Boundary Condition dialog box and (b) the Region Selection dialog box.

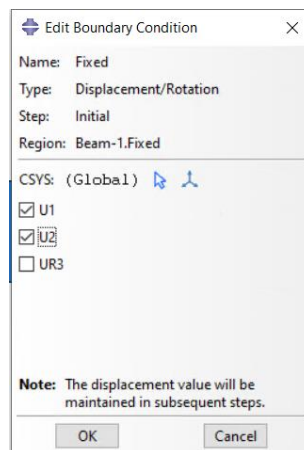


Figure 2-18 – The Edit Boundary Condition dialog box.

Apply a Load to the Top of the Cantilever Beam

1. In the **Model Tree**, double-click **Loads** to create a new load in the model **BEAM**. The **Create Load** dialog box appears, Figure 2-19(a).
2. In the **Create Load** dialog box:
 - a. Name the load **Pressure**.
 - b. Select **BeamLoad** as the step in which the load will be applied.
 - c. In the **Category** list, accept the default category selection **Mechanical**.
 - d. In the **Types for Selected Step** list, select **Pressure**.
 - e. Click **Continue**.

Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.
3. In the viewport, select the top edge of the beam as the surface to which the load will be applied. The desired edge is highlighted in Figure 2-19(b). Check the **BeamCreate Surface** and name it **BeamLoad**. Click **Done** in the prompt area to indicate that you have finished selecting regions.

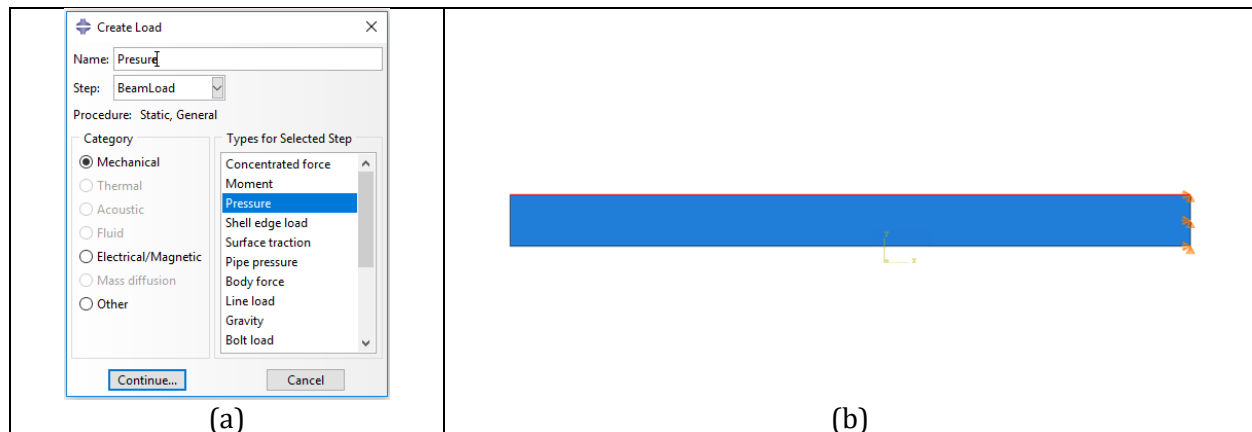


Figure 2-19 – (a) The Create Load dialog box and (b) the prompt area.

The **Edit Load** dialog box appears, Figure 2-20.

4. In the **Edit Load** dialog box:
 - a. Enter a magnitude of **0.025** for the load (see Figure 2-1(a)).
 - b. Accept the default **Amplitude** selection (**Ramp**) and the default **Distribution** (**Uniform**).
 - c. Click **OK** to create the load definition and to exit the editor.

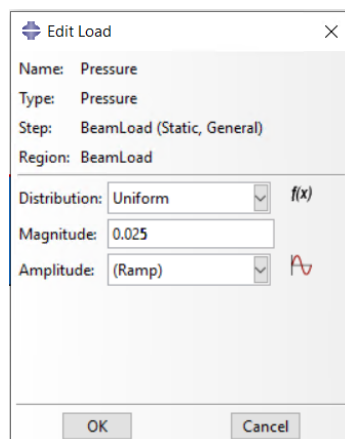


Figure 2-20 – The Edit Load dialog box.

2.6. Meshing the Model

You use the Mesh module to generate the finite element mesh. You can choose the meshing technique that Abaqus/CAE will use to create the mesh, the element shape, and the element type. Abaqus/CAE uses several different meshing techniques. The default meshing technique assigned to the model is indicated by the color of the model when you enter the Mesh module; if Abaqus/CAE displays the model in orange, it cannot be meshed without assistance from the user.

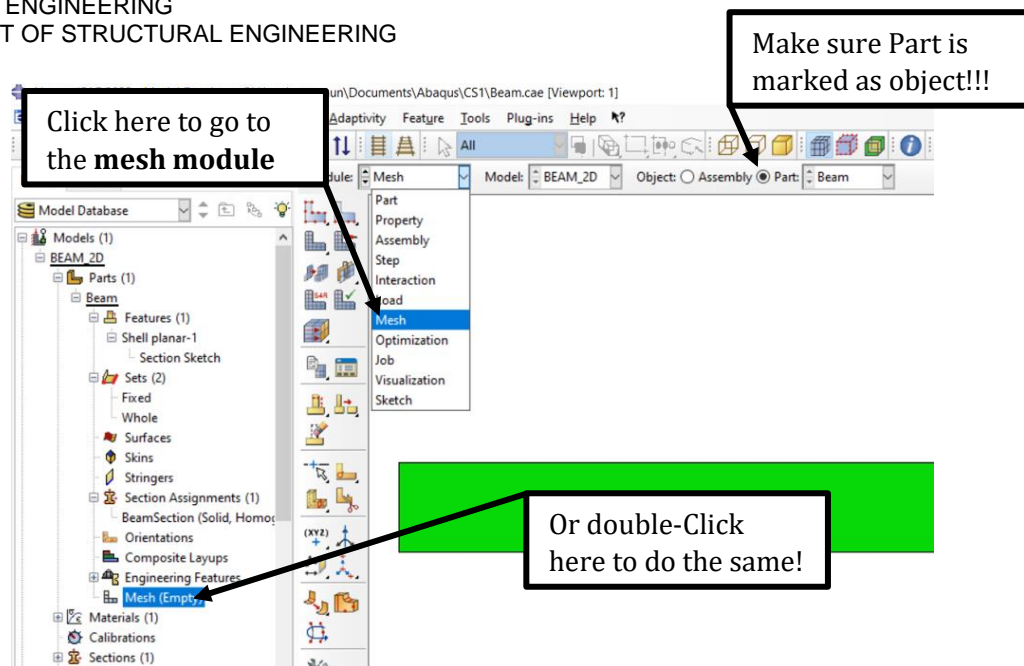


Figure 2-21 – How to access the meshing environment.

Assign the Mesh Controls:

1. In the Module drop-down menu, choose **Mesh** to access the meshing environment. You may alternately double-click on **Mesh** in the branch for the part **Beam** in the **Model Tree**. See Figure 2-21.
Abaqus/CAE switches to the Mesh module and displays the part **Beam**. **Make sure Part is selected as Object!**
2. From the tools on the left ribbon, choose the **Mesh Controls**. See (1) in Figure 2-22(b).
3. In the **Mesh Controls** dialog box, set **Quad** as the **Element Shape** selection.
4. Set **Structured** as the default **Technique** selection.
5. Click **OK** to assign the mesh controls and to close the dialog box.

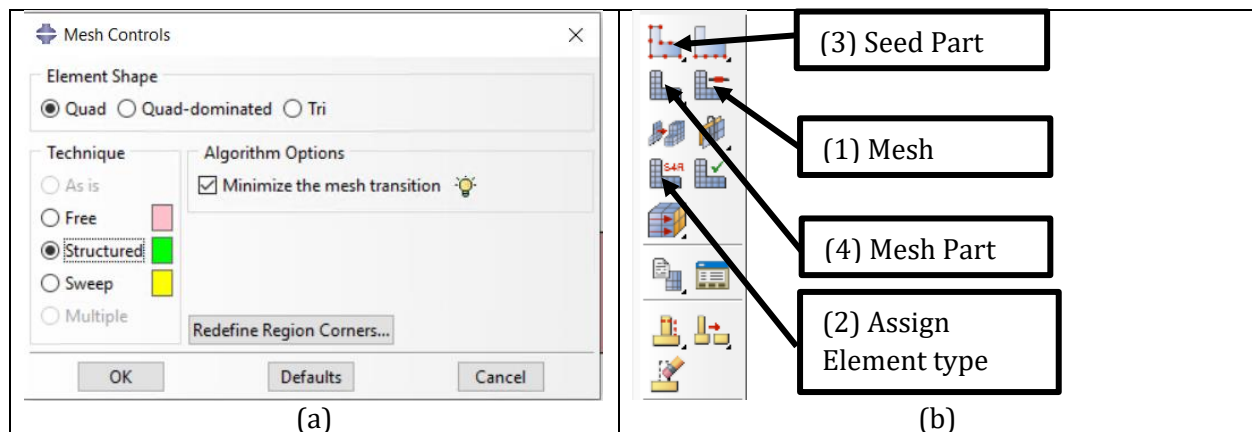


Figure 2-22 – (a) The Mesh Control dialog box and (b) the different tools for meshing a part in this workshop.

Assign an Abaqus Element Type:

1. From the tools in the left ribbon, choose the **Assign Element type**. See (2) in Figure 2-22(b).

2. In the **Element Type** dialog box, as shown in Figure 2-23. Accept the following default selections that control the elements that are available for selection:
 - **Standard** is the default **Element Library** selection.
 - **Linear** is the default **Geometric Order**.
 - **Plane Stress** is the default **Family** of elements.
 3. In the lower portion of the dialog box, examine the element shape options. A brief description of the default element selection is available at the bottom of each tabbed page.
 4. In the **Quad** tabbed page, unselect **Reduced integration**. No modification is necessary in the **Tri** tab.
- A description of the element type CPS4 appears at the bottom of the dialog box.
 Abaqus/CAE will now mesh the part with QPS4 elements.
 Note: **CPS** stands for constant plane stress and **4** implies 4 nodes. Incompatible modes¹ should probably have been used, but we will leave it up to you to test it out. If incompatible modes are selected the name will get an additional I at the end.
5. Click **OK** to assign the element type and to close the dialog box.

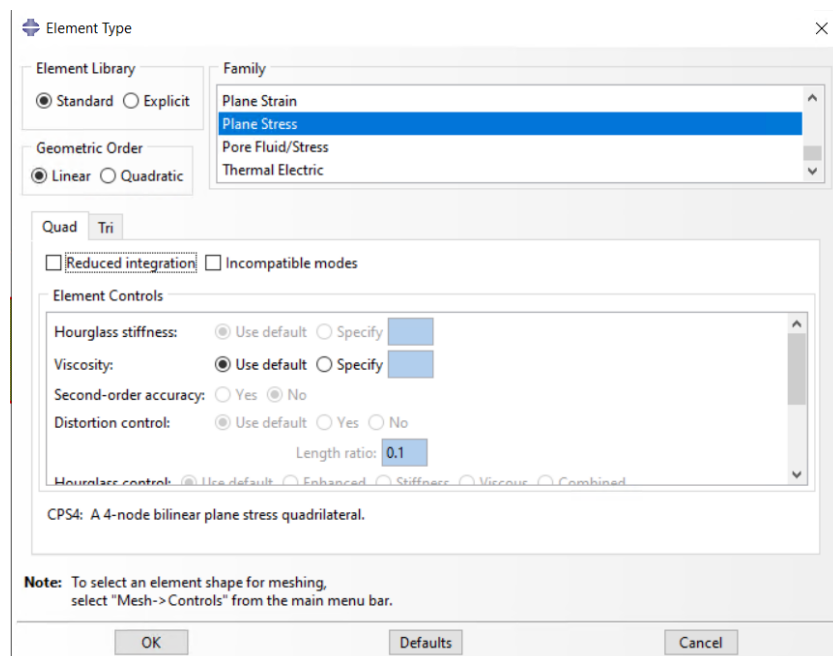


Figure 2-23 – The Element Type dialog box.

¹ The incompatible element will be further explained in the lectures. In general, a linear element experiences shear locking when subjected to bending. Thus, linear elements are not recommended for bending problem, especially if you use a coarse mesh. Quadratic elements are an alternative but is more computationally expensive. The incompatible element is therefore often preferred in such problems if the elements are not too distorted since the accuracy declines rapidly with increasing distortion for these kinds of elements.

Mesh the Model:

1. From the tools in the left ribbon, choose the **Seed Part**. See (3) in Figure 2-22(b).
2. The **Global Seeds** dialog box appears, Figure 2-24. The default global element size is based on the size of the part.
3. In the **Global Seeds** dialog box, enter an approximate global size of **50** (mm) and click **OK**. Abaqus/CAE applies the seeds to the part, as shown in Figure 2-25(a).
4. From the tools in the left ribbon, choose the **Mesh Part**. See (4) in Figure 2-22(b).
5. Click **Yes** in the prompt area to confirm that you want to mesh the part instance. Abaqus/CAE meshes the part instance and displays the resulting mesh, as shown in Figure 2-25(b).

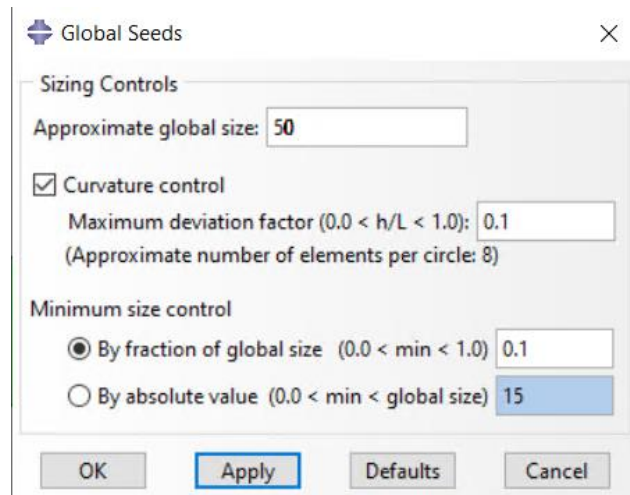


Figure 2-24 – The Global Seeds dialog box.

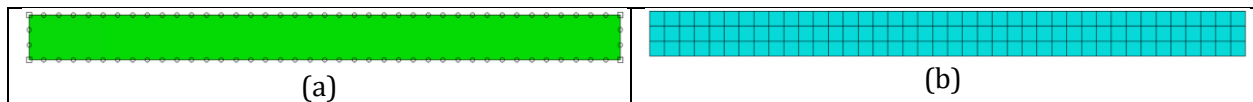


Figure 2-25 – (a) The Seeded part instance and (b) part instance showing the resulting mesh.

2.7. Creating and Submitting an Analysis

The definition of the model **BEAM** is now complete. Next, you will create and submit an analysis job to analyze the model. Before submitting a job, **ALWAYS check if the work directory is set**. See Section 1.2 for more information.

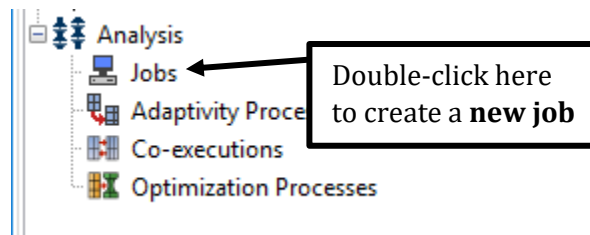


Figure 2-26 – The analysis part of the Model Tree

Create and Submit an Analysis Job

1. In the Model Tree, double-click **Jobs** to create a new analysis job.
Abaqus/CAE switches to the Job module, and the **Create Job** dialog box appears, see Figure 2-27(a).
2. In the **Create Job** dialog box, name the job **Deform** and select the model **BEAM_2D**. Click **Continue**.
The **Edit Job** dialog box appears.
3. In the **Parallelization** tab, Figure 2-27(b), check the **Use multiple processors** in order to use more than one CPU, allowing faster simulations. Click **OK** to accept the default job settings.

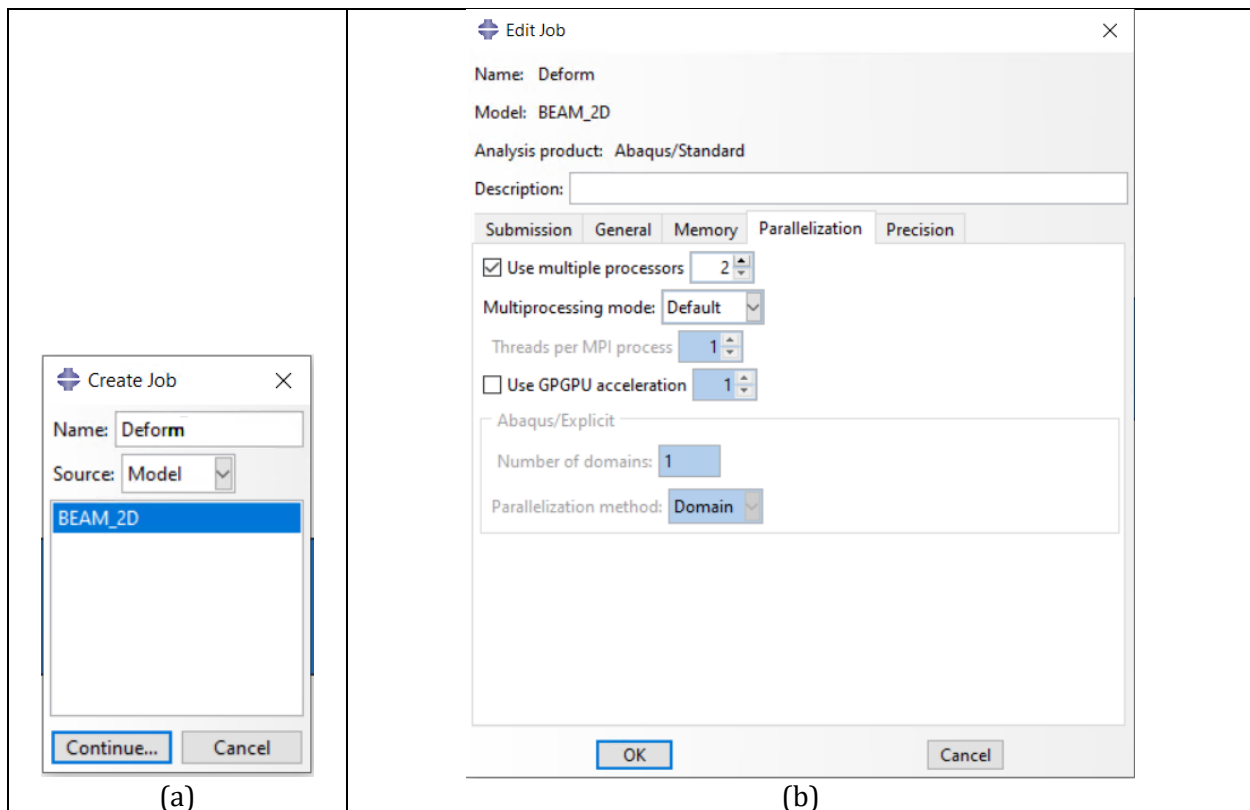


Figure 2-27 – (a) The Create Job dialog box and (b) the Edit Job dialog box.

4. In the **Model Tree**, expand the **Jobs** container and right-click on the job **Deform**.
The menu for the job **Deform** appears, as shown in Figure 2-28(a).
5. From the job menu, select **Submit**. Abaqus/CAE will now send the job to a solver, which performs the analysis.
The icon for the job will change to indicate the status of the job in parenthesis after the job name. As the job runs the status **Running** will be shown in the Model Tree. When the job completes successfully, the status will change to **Completed**, as shown in Figure 2-28(b). Notice that Abaqus has generated many new files to your work directory. The **.odb** file contains the results from your analysis.

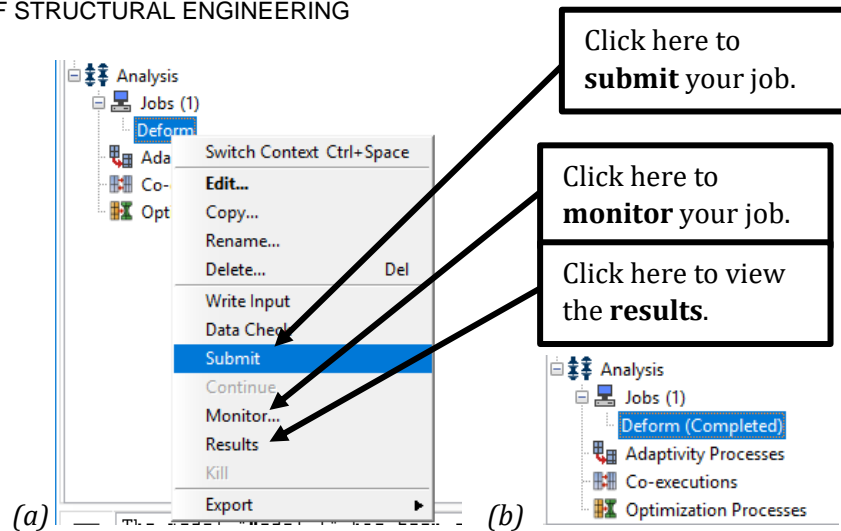


Figure 2-28 – (a) The menu for the Deform job and (b) status when the analysis has completed.

6. In the menu for the job **Deform**, Figure 2-28(a), click on **Monitor** to access the **Monitor** dialog box. Here the analysis status may be checked for **Errors** and **Warnings**. **This should be checked for every analysis.** The same information can be found in the **.msg** and **.sta** files in your working directory.

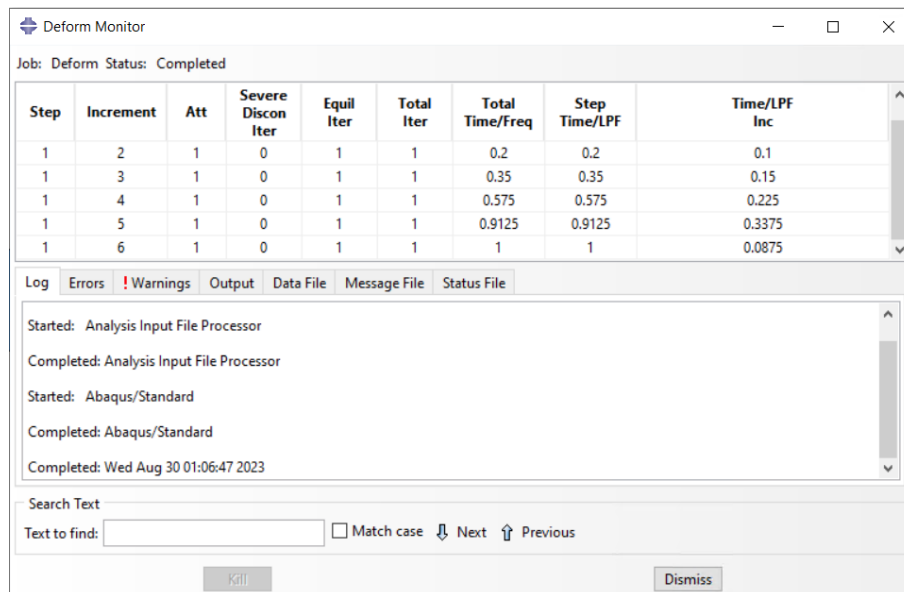


Figure 2-29 – The Monitor dialog box.

2.8. 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 open this file two ways

1. Right-click on your job-file in the **Model Tree** and choose **Results**. See Figure 2-28.

2. In the main menu bar, select **File** → **Open**. The **Open Database** dialog box appears. Make sure to set the **File Filter** to **Output Database (*.odb)**.


Abaqus/CAE switches to the Visualization module, opens the output database created by the job (**Deform.odb**), and displays the undeformed shape of the model, as shown in Figure 2-30.



Figure 2-30 – Un-deformed model shape.

Plotting Deformation and Contour Plots

The data saved in the **Field Output Requests** can be viewed as a deformation and contour plot in the viewport.

1. In the toolbox, click  (or select **Plot** → **Deformed Shape** from the main menu bar) to view a deformed model shape plot, as shown in Figure 2-31.
In order to plot the un-deformed beam in the same plot, go to **Plot** → **Allow Multiple Plot State**.

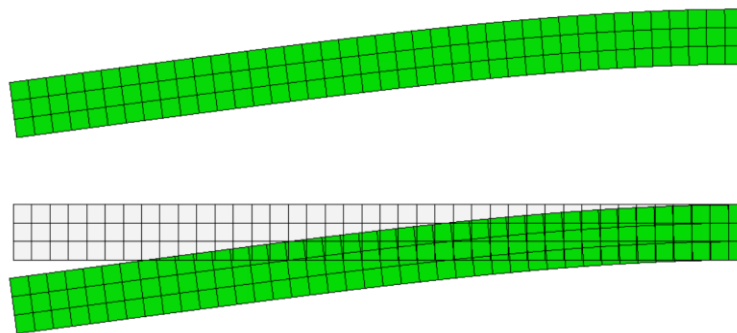



Figure 2-31 – Deformed model shape.

You may need to use the **Auto-Fit View** tool  to rescale the figure in the viewport.

2. In the toolbox, click  (or select **Plot** → **Contours** → **on Deformed Shape** from the main menu bar) to view a contour plot of the von Mises stress, as shown in Figure 2-32.

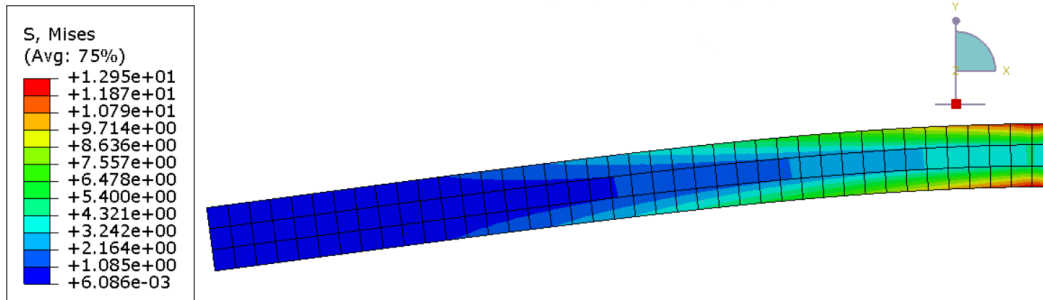


Figure 2-32 – von Mises contour plot.

3. You can change the contour plot from the **Field Output** menu, as shown in Figure 2-33. You may choose from the output data defined in the **History Output Request**, see Section 2.4.

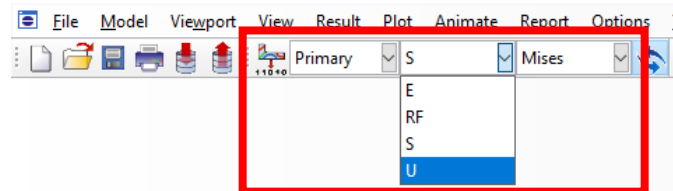


Figure 2-33 – The Field Output menu.

4. You can change the color in the background of the viewport by selecting **View** → **Graphics Options** from the main menu bar. Here, change the **Viewport Background** option.
5. The annotations can be removed by selecting **Viewport** → **Viewport Annotation Options** from the main menu bar. Uncheck **Show compass** and **show triad**. Select **OK**.
6. 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. Use PNG as format. See Figure 2-34.

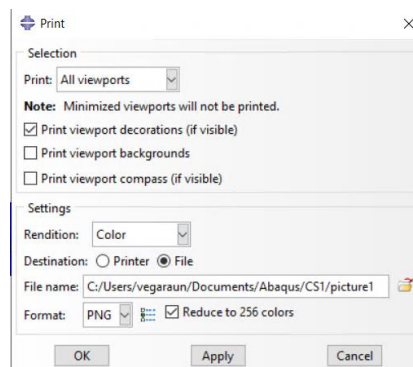



Figure 2-34 –The Print dialog box

Obtain Numerical Values From the Field Output

You can obtain numerical values from the Field Output. The simplest way is to use the **Query information** tool.

1. In the upper toolbox, select use the **Query information** tool . The **Query** dialog box opens.

- In the **Query** dialog box, use **Node** as the **General Queries**. Input the node in the prompt area or select a node in the viewport. The numerical value for the displacement is displayed in the **Kernel Command Line Interface**, see Figure 2-35(b).

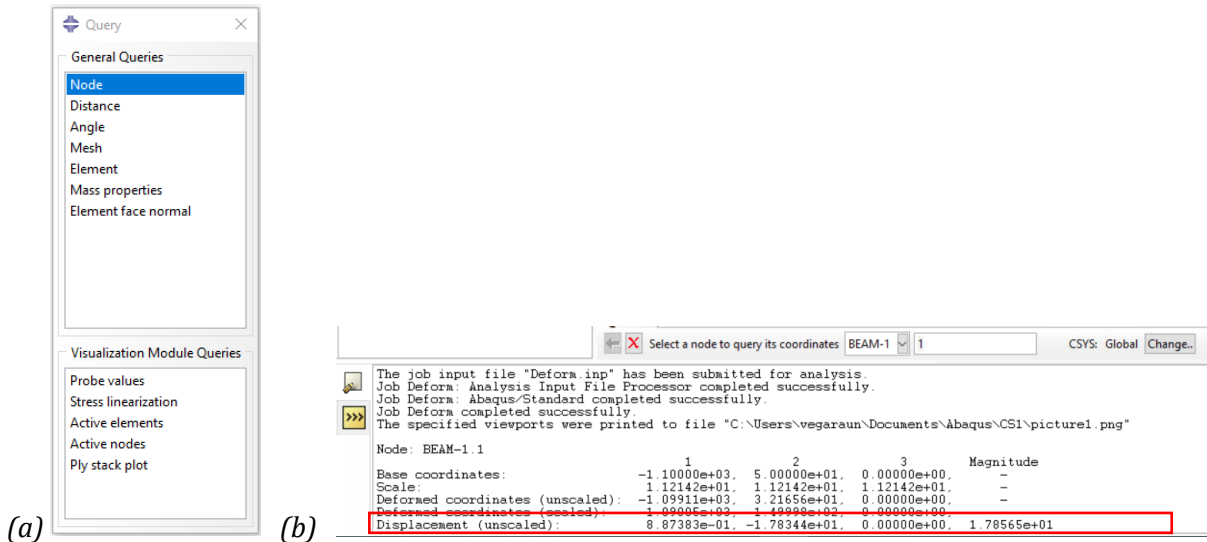


Figure 2-35 – (a) The Query dialog box and (b) the Kernel Command Line Interface.

You can also use the **Probe values** tool in **Query** dialog box. This opens the **Probe Values** dialog box. Select the last **Frame** and **Field output variable for Probe**. In the **Probe Values**, select **Key-in label** and set the **Probe** to **Nodes**. See Figure Figure 2-36. Input the **Node labels**.

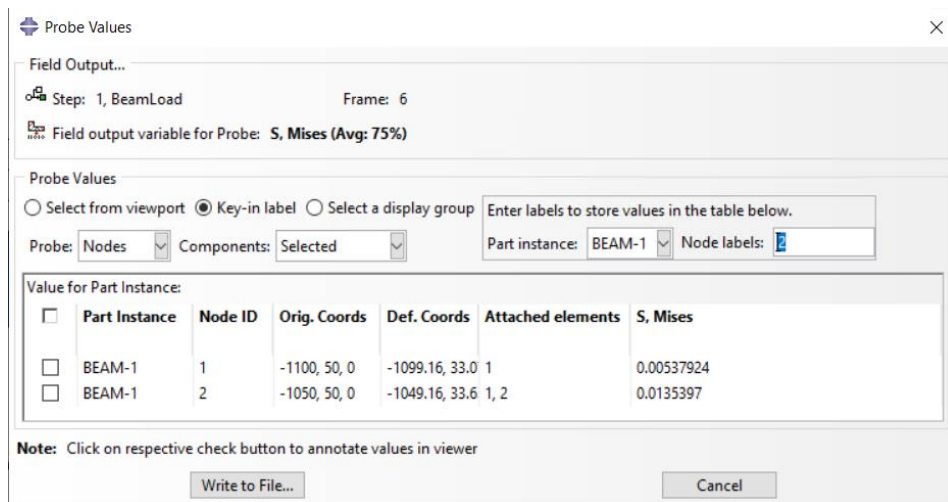


Figure 2-36 – The Probe Values dialog box.

3. Modelling the Beam with a Hole Using 2D Plane Stress Elements

We will now increase the complexity of the model to also include the hole at the steel rod support. See Figure 3-1. The process of modeling the beam with a hole is rather similar as done in Section 2. This section will only highlight the main differences. The reader is referred to Section 2 for a more in-depth description of the modeling process.

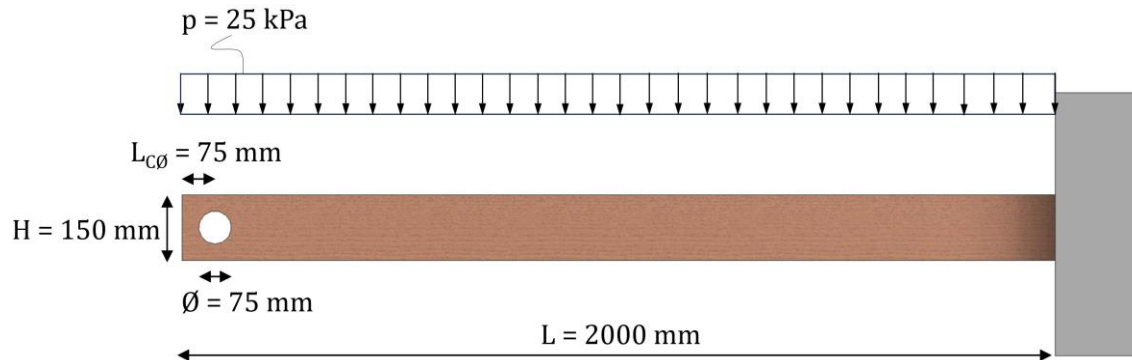


Figure 3-1 – Side view of the beam including the hole at the end of the beam.

1. First, create a new model by right-click on the previous **BEAM_2D** model in the **Model Tree**. Select **Copy Model** to create a new model. Name the model **BEAM_2D_HOLE**. Click **OK**. A new model is added to work **Model Tree**.

Right-click here to copy the previous model

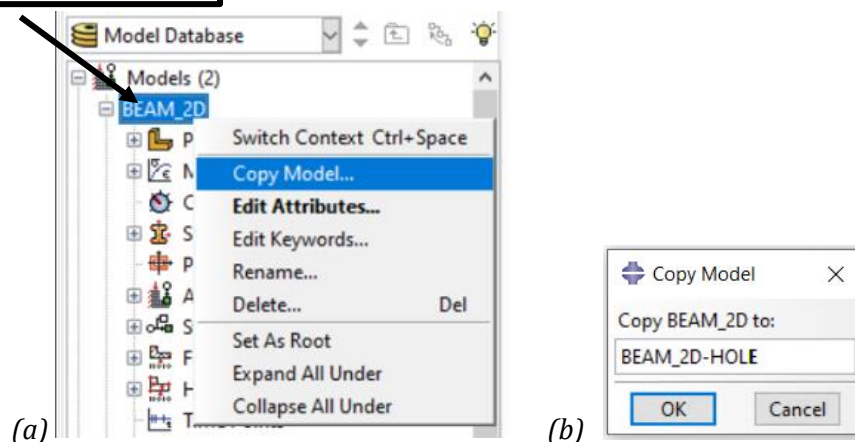


Figure 3-2 – (a) The Model Tree and (b) the Edit Model Attributes dialog box.

You can switch between your models using the Model drop-down menu as shown in Figure 3-3.

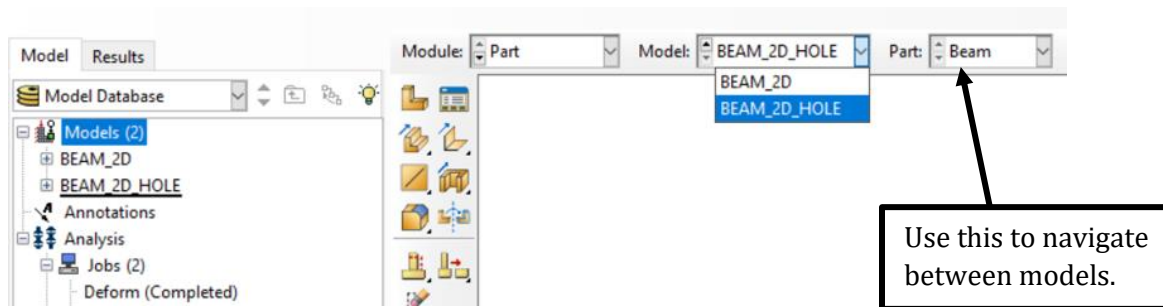


Figure 3-3 – The Model drop-down menu.

2. You can make any changes to the sketch by right-clicking on **Section Sketch** and choose **Edit**. See Figure 3-4. See also 7 in Section 2.1.

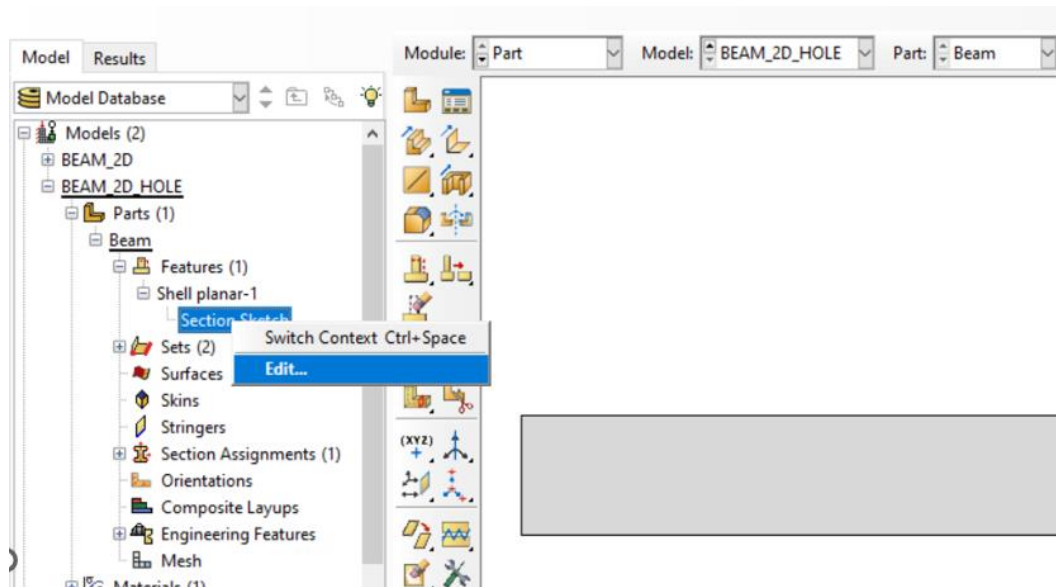


Figure 3-4 – Edit section sketch to make a hole in the beam.

3. To sketch the hole, you need to select the **circular drawing** tool, as shown in Figure 3-5.

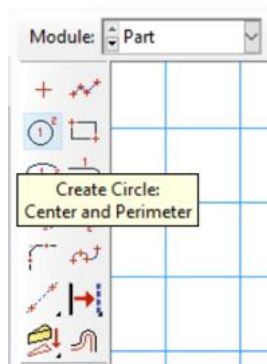


Figure 3-5 – Use the create circle drawing tool.

4. In the viewport, sketch the circle using the following steps:
 - a. You will first sketch a rough approximation of the circle and then use constraints and dimensions to refine the sketch. Select any two points for the center point and perimeter point, respectively.
 - b. Click the mouse **Esc** or **Cancel** (✖) in the prompt area to exit the create circle tool.
 - c. The Sketcher automatically adds the center point and perimeter point to the sketch. See Figure 3-6.
 - d. Use the dimension tool (📏) to dimension the radius (**37.5 mm**) and to position the center point of the circle. The position of the center point should be **75 mm** from the bottom edge and **75 mm** from the left end. When dimensioning each point, simply select the point and its reference point, and then enter the new dimension in the prompt area.
 - e. The final sketch is shown in Figure 3-7.
5. Click **Done** in the prompt area to exit the sketcher.

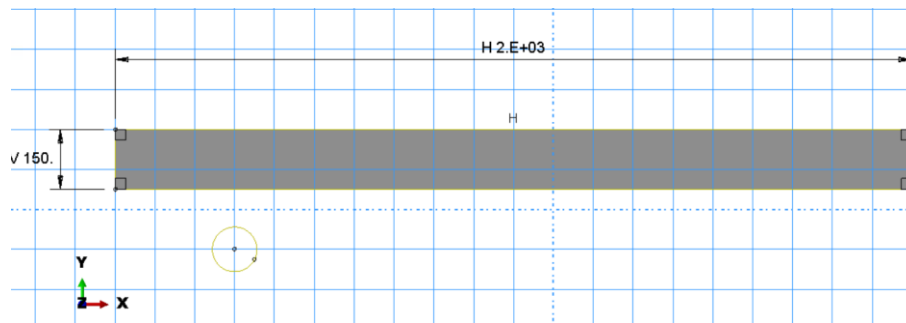


Figure 3-6 – Sketch a rough approximation of the circle.

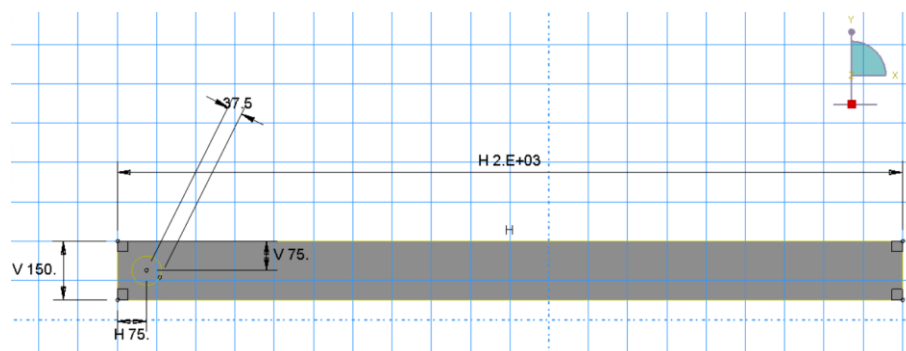
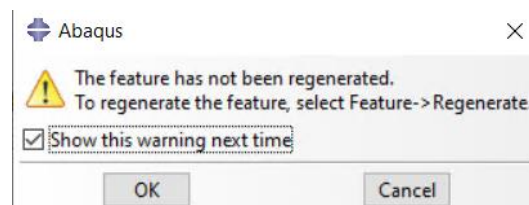


Figure 3-7 – Use the create circle drawing tool.

The following warning shows up.



6. Click **OK**. Then click **Feature - Regenerate** to allow changes in the part geometry.

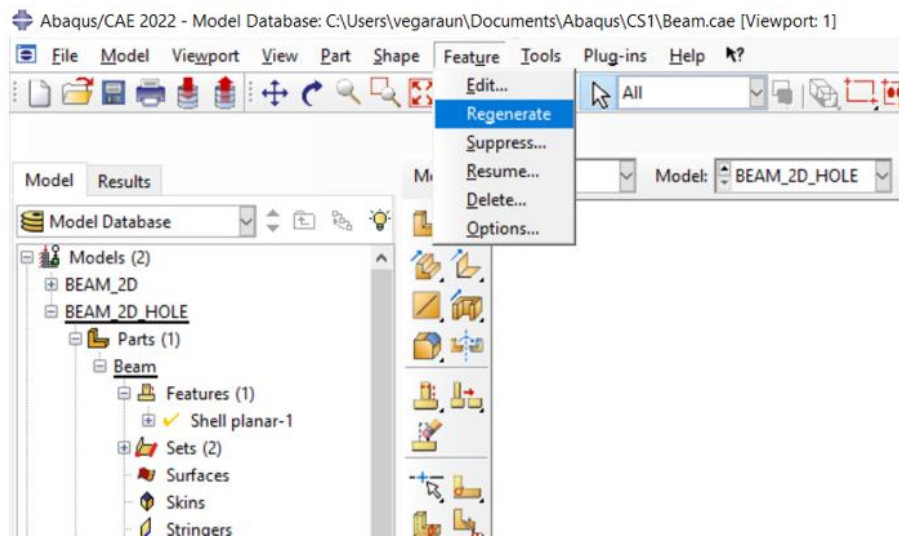


Figure 3-8 – Select Feature – Regenerate to allow for the hole to be a part of the geometry.

7. We now have a hole at one end of the beam. See Figure 3-9.



Figure 3-9 – Geometry of the new part.

8. Before meshing the new part, it is convenient to partition the area around the hole to better control the mesh process. Click in the **Partition Face: Sketch** tool. See Figure 3-10.

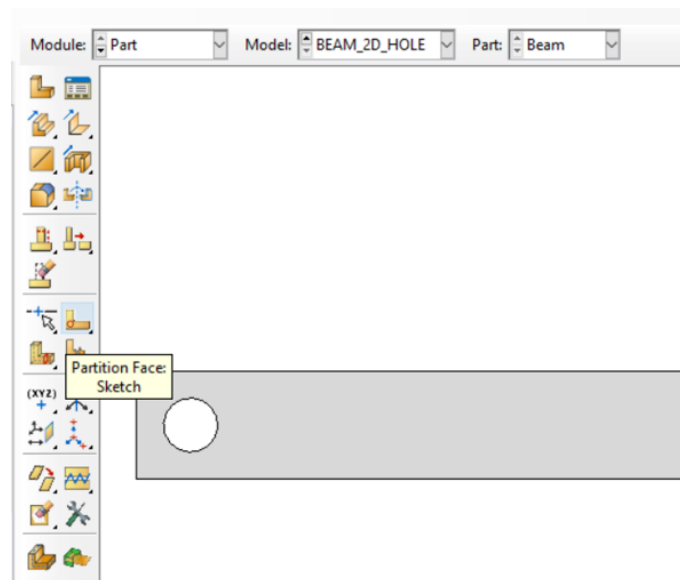



Figure 3-10 – Sketch partition face tool.

9. Draw a rectangle around the hole and ensure that the dimensions are **150 mm x 150 mm** by using the constraint tool (). See Figure 3-11.

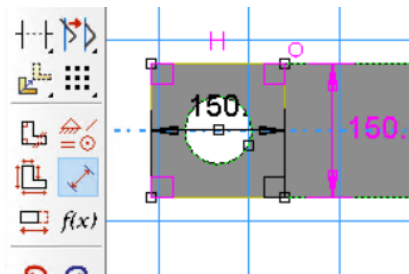



Figure 3-11 – Sketch rectangular partition face around the hole.

10. Use the Create Lines tool () to draw lines along the horizontal, vertical and diagonal of the rectangle. See Figure 3-12.

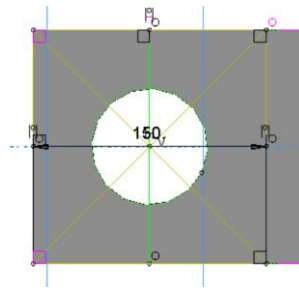



Figure 3-12 – Sketch of final partition face around the hole.

11. Click **Done** in the prompt area. If you get an error/warning related to the dimensioning of the partition and that this will not allow you to complete the sketching operation. Then, delete () the 150 mm x 150 mm dimensions and try again.
12. Move to the Mesh module. See Figure 3-13(a). The part will now turn pink. See Figure 3-13(b).

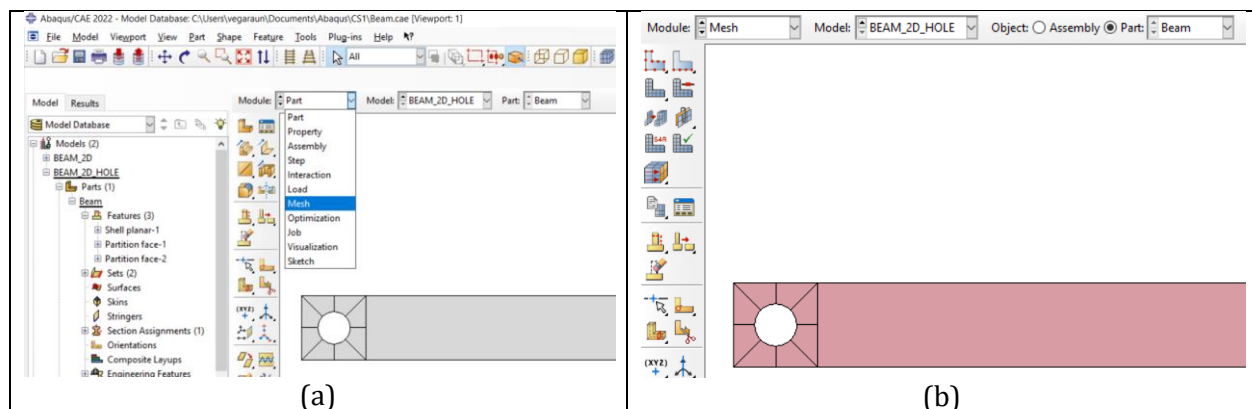


Figure 3-13 – Move to the mesh module.

13. Assign mesh controls and mesh the part by following the same procedure as in Section 2.6. Use an approximate global seed size of **12.5 mm**. The final mesh is shown in Figure 3-14.

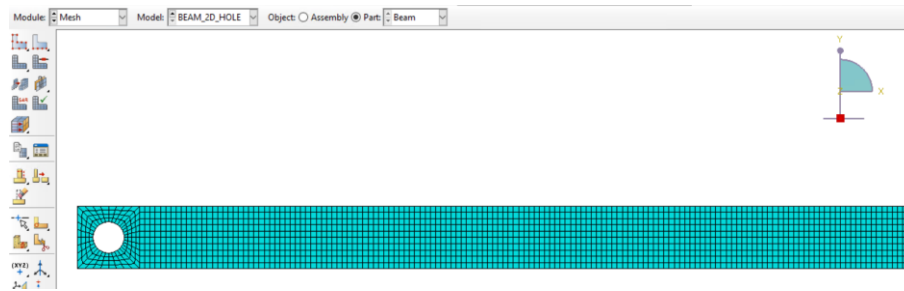


Figure 3-14 – Mesh of the new part.

14. We now need to impose the boundary conditions around the hole due to the support of the steel rod (see the problem description of Case Study 1). We will assume that the rod is in contact with the beam at parts of the perimeter and that the stiffness of the steel rod is high. The beam will therefore be restricted for horizontal and vertical displacements at the perimeter of the hole.

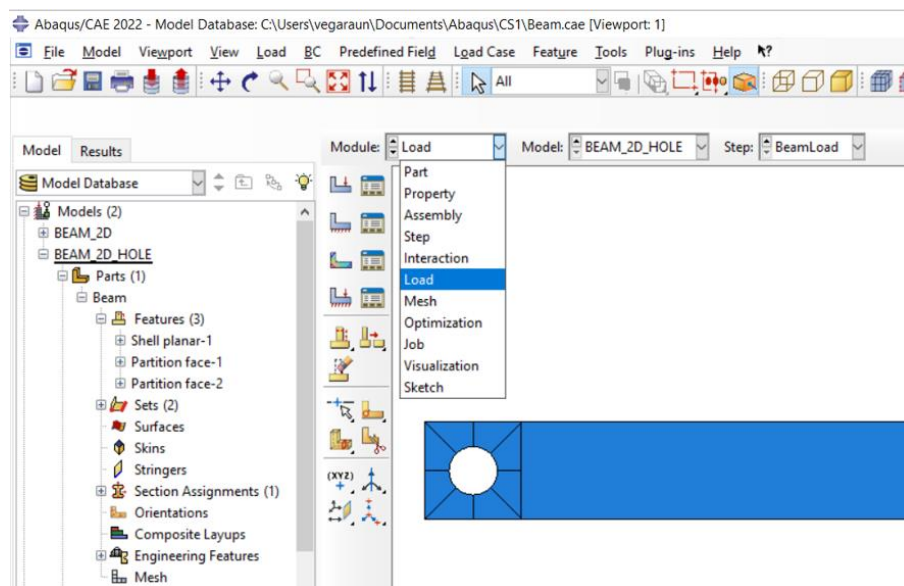



Figure 3-15 – Move to the Load module.

15. Click on the Create Boundary Condition tool (), give it the name **VerticalConstraint**, select **Initial** under step, use category **Mechanical** and type **Displacement/Rotation**. Click on **Continue** and use the viewport to select the edges corresponding to the upper half of the perimeter. These edges are indicated by red color in Figure 3-16. In the prompt area, assign the name **UpperHalf** for the set and click **Done**.

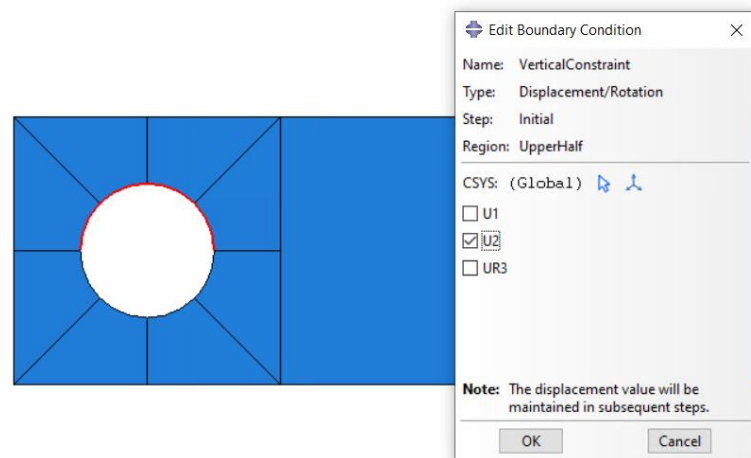


Figure 3-16 – Assigning boundary conditions restricting vertical movement of the beam at the hole.

16. In the **Edit Boundary Condition** dialog box, toggle on **U2** since only the vertical translational degrees of freedom need to be constrained in this boundary condition. See Figure 3-16. Click **OK**.
17. Repeat 15 for the horizontal constraint. Select the edges of the left perimeter of the hole. Assign appropriate names to the boundary condition and the new set. See Figure 3-17.

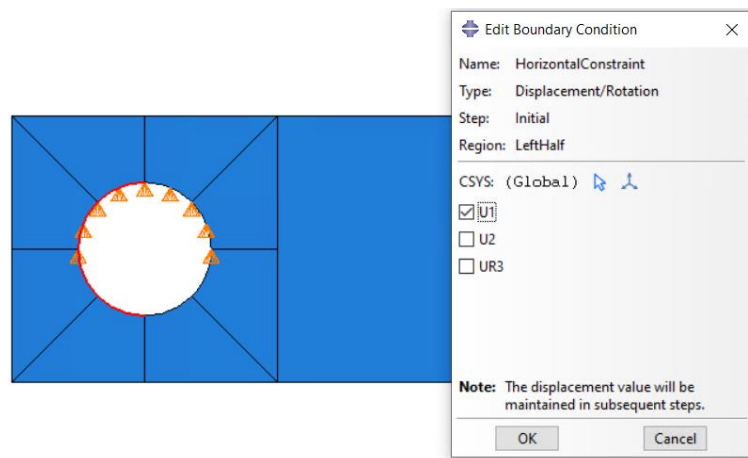


Figure 3-17 – Assigning boundary conditions restricting horizontal movement of the beam at the hole.

18. In the **Edit Boundary Condition** dialog box, toggle on **U1** since only the horizontal translational degrees of freedom need to be constrained in this boundary condition. See Figure 3-17. Click **OK**.
19. In the **Load module**, open the **Load Manager** as shown in Figure 3-18. Choose **Edit** in the Load Manager and click on the arrow next to Region. This will allow you to edit the loaded region according to the new geometry and partitions.

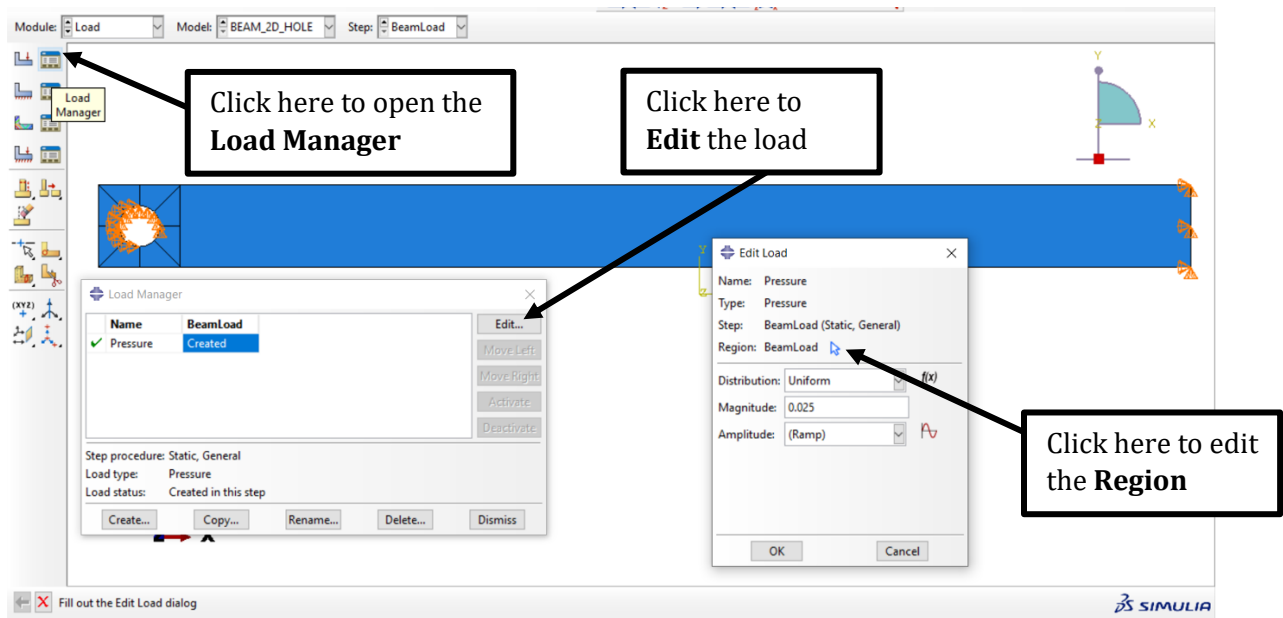


Figure 3-18 – Edit the load by updating the region according to the new geometry and partitions.

20. Reselect the edges for the load and assign the name **NewLoad** in the prompt area. Click **Done**.



Figure 3-19 – Selected edges for the updated loading regions.

21. Accept the updates by clicking **OK** in the **Edit Load** box. The loading should now be applied to the updated regions. See Figure 3-19.

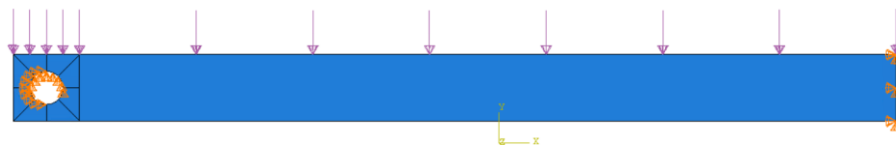


Figure 3-19 – The load after editing the regions.

22. Save the model. Make sure you have the correct Working Directory (see Section 1.2).
23. Follow the steps in Section 2.7. for Creating and Submitting an Analysis
24. Finally, post-processing the results following the steps according to what you are asked to report in Case Study 1.

