

Torsional-lateral buckling – bifurcation analysis with a simple beam using Abaqus 6.10

This document contains an Abaqus tutorial for performing a buckling analysis using the finite element program Abaqus/Standard, version 6.10. The analysis is performed on a simply supported beam with end moments. The beam is modelled using shell elements.

Many different types of analyses are available with Abaqus. However, only one single type of analysis is covered by the guide, namely a so-called buckle analysis used to examine instability. This type of analysis solves the eigenvalue problem associated with the structure with given loads, in order to assess bifurcation load values.

Abaqus has two different ways of providing input: Via a graphical user interface (the so-called CAE) or directly via an input text file. Only the first method will be described in the following. Advanced users may prefer the second option as it provides some freedom compared with the CAE.

Getting started

Run Abaqus CAE and choose **Create Model Database With Standard/Explicit Model**. You should now see the screen visualised in Figure 1. On the left you have the project tree and on the right you have the working area. The field right above the working area is denoted the context bar.

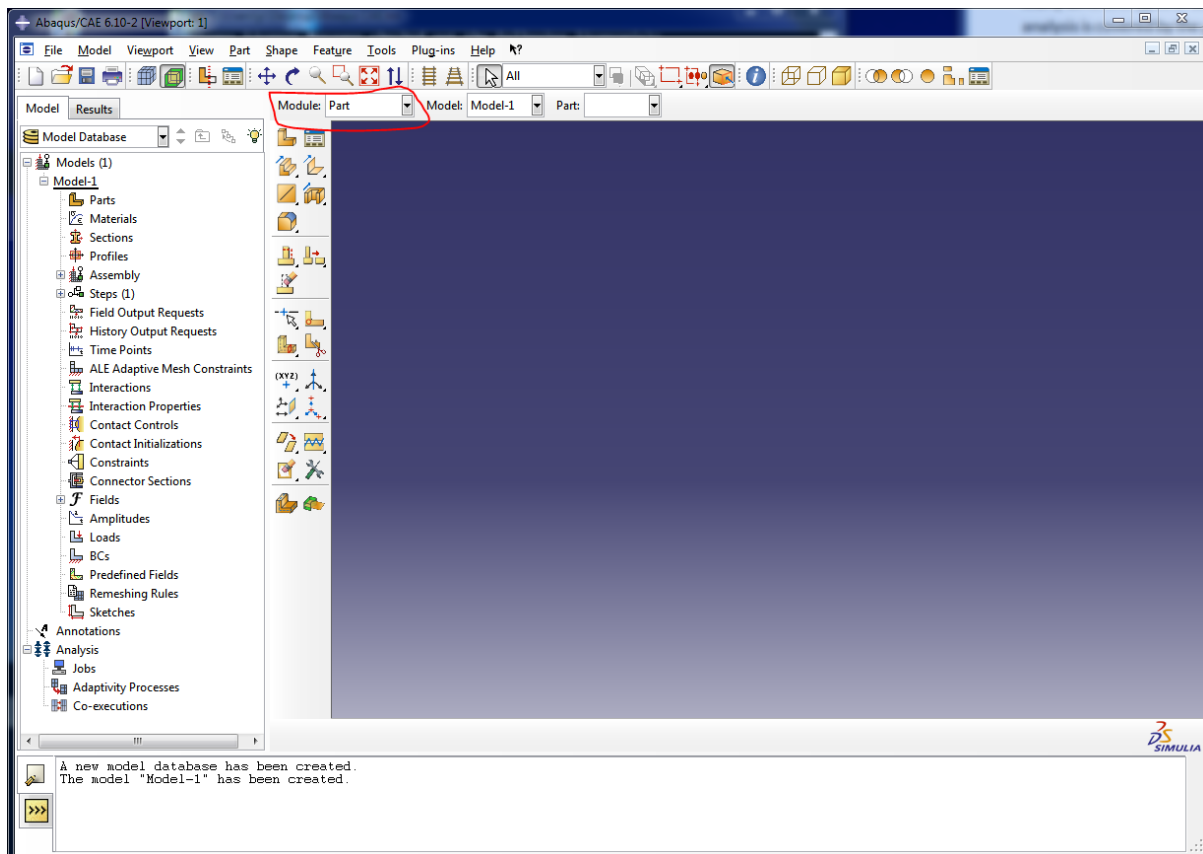


Figure 1 Start-up screen in Abaqus 6.10-2. The Module drop-down menu is encircled in red.

Abaqus CAE has a number of so-called modules. Each module is utilized to define some part of the model, e.g. the geometry or the boundary conditions. You work in one module at a time and can change module in the drop-down menu **Module** which can be found in the context bar as indicated by the red rectangle in Figure 1.

The modules are:

1. **Part** – defines the geometry of a structural element or model to be used in the analysis.
2. **Property** – defines materials and cross sections.
3. **Assembly** – assembles a number of parts to form the global geometry of a model.
4. **Step** – defines the different analyses to be carried out.
5. **Interaction** – defines connections and interface conditions between different parts.
6. **Load** – defines the boundary conditions of the model.
7. **Mesh** – provides the discretization of the model into finite elements.
8. **Job** – defines the jobs to be carried out by the analysis program.
9. **Visualization** – is utilized for viewing and post processing the results.
10. **Sketch** – can be used as a simple CAD programme for making additional drawings.

As indicated in Figure 1, you will start out in the module Part.

Defining the geometry

The first step is to define the geometry of the model. In this case, we would like to analyse a simply supported beam with an I-profile. Our strategy is to establish a shell finite-element model from a solid part using the following approach.

1. In the Model Tree, double-click the **Parts** container to create a new part. From the **Create Part** dialog box that appears, name the part Beam, and accept the default settings of a three-dimensional, deformable body and a solid, extruded base feature. In the **Approximate size** text field, type 0.40. This value is twice the largest dimension of the part. Click **Continue** to exit the **Create Part** dialog box.
 - a. Create a rectangle using the **Create Lines: Rectangle** tool. The rectangle should have the dimensions 0.08 x 0.32 as shown in Figure 2. This can be achieved by typing 0,0 as the starting point and 0.08,0.32 as the end point in the input box below the window.
 - b. Hit the Esc-key and click **Done** in the prompt area.
 - c. The **Edit Base Extrusion** dialog box appears. To complete the part definition, you must specify the distance over which the profile will be extruded.
 - d. In the dialog box, enter an extrusion distance of 8.0 m.
 - e. Abaqus/CAE exits the Sketcher and displays the part.

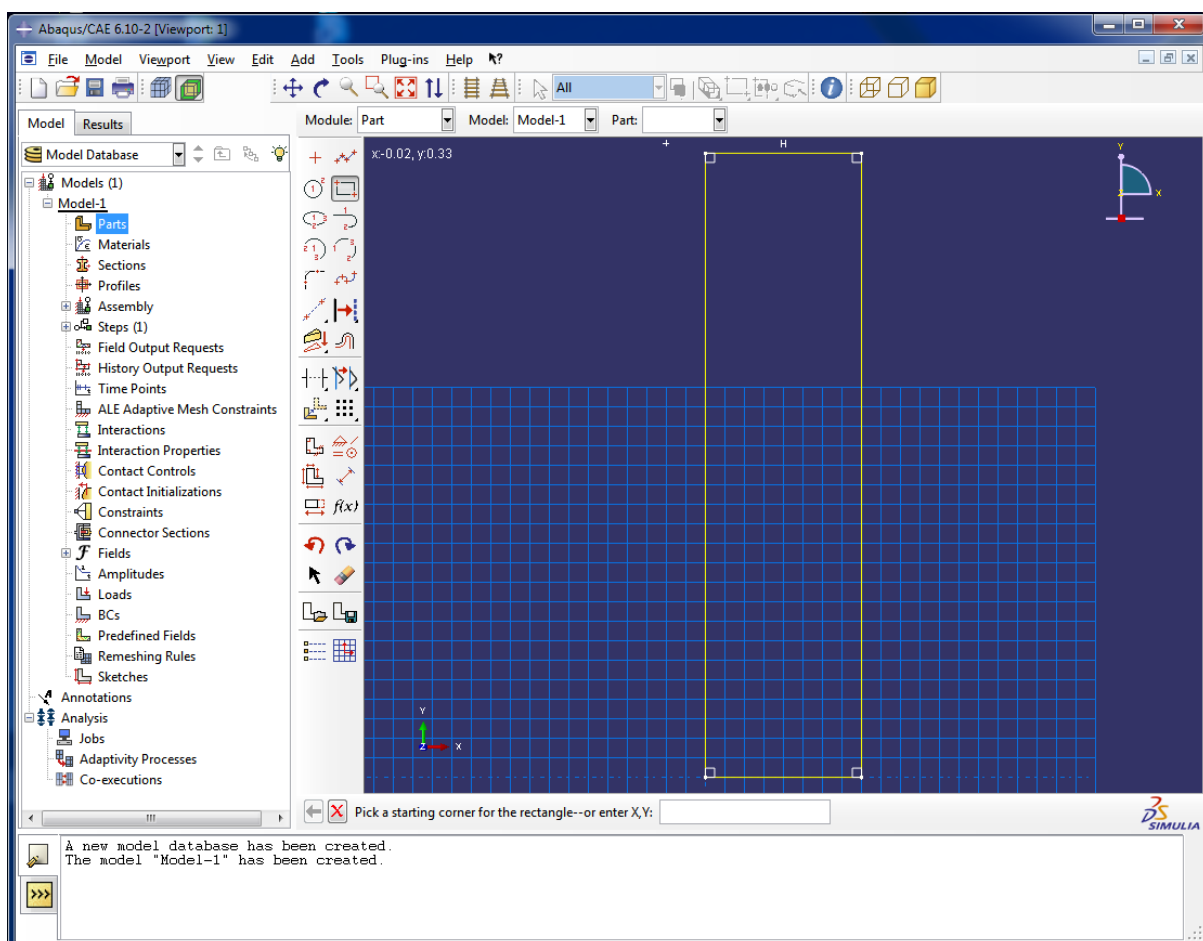


Figure 2 Sketch of the rectangular cross section of the solid profile.

2. Create a cut through the part by using the **Partition Cell: Define Cutting Plane** tool. Select **Normal To Edge**.
 - a. Choose the bottom edge (the short edge) at one end of the solid rectangular beam.
 - b. Select the midpoint of the chosen edge.
 - c. Press **Create Partition**, then click **Done** in the prompt area.
 - d. The solid is now divided into two halves along the symmetry plane.
3. Convert the solid part to a shell model by the **Create Shell: From Solid** tool. The tool can be found by pressing the small black triangle marked by a red circle in Figure 3.
 - a. Select both of the solids. This can be done by holding down the shift key. Then click done twice.

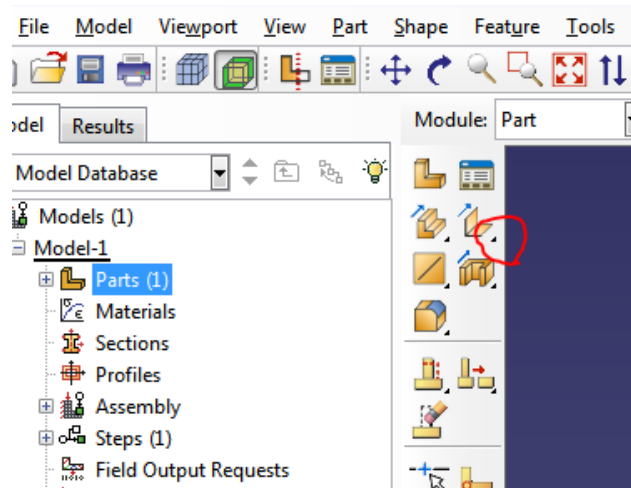


Figure 3. The small black triangle needed to find the **Create Shell: From Solid** tool

4. To remove the two shell faces click **Remove Faces**.
 - a. Select the large surface that is the beam side and press done. The face is now removed.
 - b. Now hold down Ctrl+Alt and use Mouse Key 1 to rotate the model so that the opposite side surface becomes visible. Repeat item 4.a. for this surface.
5. Skip to the Assembly module to assemble the model.
 - a. Use the **Instance Part** tool to include the part (Beam) you just created in the model.
 - b. Keep the Instance Type as **Dependent**
 - c. Click **OK** to finish the Create Instance box.
 - d. The geometry has now been defined – except the shell thickness. The model should appear as shown in Figure 4. Note that the beam axis is parallel to the z-axis.
6. If you have not yet saved the model, this is certainly the time to do so!

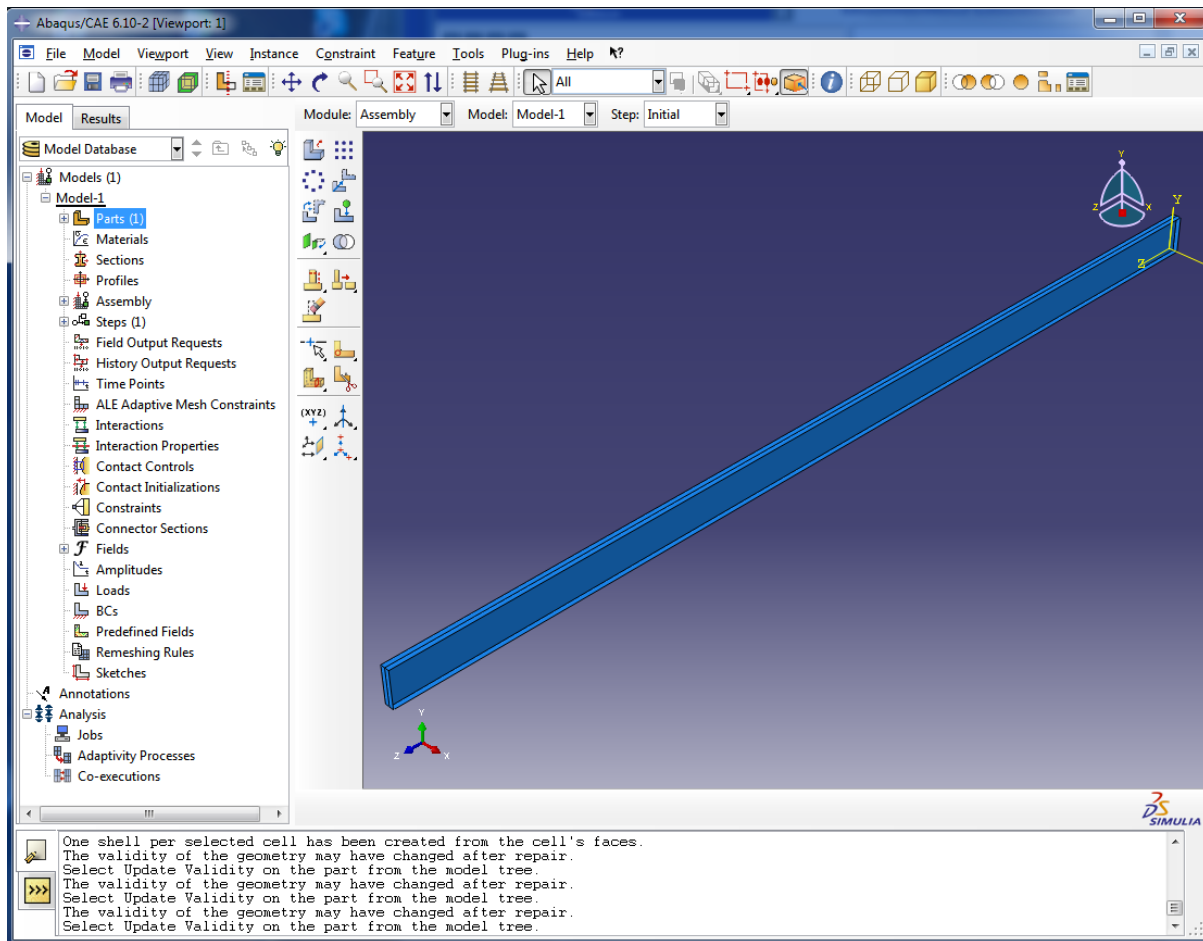


Figure 4 Sketch of the I-beam

Defining the material and cross-sectional properties

Now, go back to the **Property** module in which the properties of the beam will be defined.

1. Click the **Create Material** tool to define a new material
 - a. Change the name to Steel.
 - b. From the menu in the Material Behaviors field, select Mechanical/Elasticity/Elastic.
 - c. Put Young's Modulus as 2.1×10^{11} and Poisson's Ratio as 0.3 and change nothing else.
 - d. Click **OK** to close the **Edit Material** box.
2. Next, use the **Create Section** tool to define a shell cross section.
 - a. Change the name to Plate
 - b. Change **Category** to **Shell** and keep **Type** as **Homogeneous**. Then click **Continue ...**
 - c. Change **Section integration** to **Before analysis**.
 - d. Put the Shell thickness as 0.008.
 - e. The **Material** should automatically be put as Steel.
 - f. Click **OK** to close the **Edit Section** box.
3. Use the **Assign Section** tool to assign the shell data to the model.
 - a. Select the entire model by dragging a rectangle with Mouse Key 1 pressed.
 - b. Click **Done** in the prompt area.
 - c. The **Section** should automatically appear as Plate.
 - d. Push **OK** to close the Edit Section Assignment box.

Setting up the analysis

It is now time to define the steps in the analysis. This is done in the **Step** module.

1. Use the **Create Step** tool to define a new analysis step.
 - a. Change the name to Instability.
 - b. Change **Procedure type** to **Linear perturbation**.
 - c. Select Buckle in the sub-menu and press **Continue ...**
 - d. In the Edit Step box, change **Eigensolver** to **Lanczos** and put the requested number of eigenvalues as 4. Then click **OK** to close the box.

Defining the boundary conditions

Boundary conditions in Abaqus are defined in the **Load** module. The various boundary conditions are generated in a given Step and may be transferred to subsequent steps.

1. Use the **Create Boundary Condition** tool to define supports for the beam in the x-direction.
 - a. Change **Name** to Fix-x.
 - b. Change **Step** to Initial.
 - c. Keep **Category** as Mechanical and put **Type for Selected Step** as Displacement/Rotation. Then click **Continue ...**
 - d. Select the midlines at the ends of the beam. More lines may be selected simultaneously by holding down Shift. You may use the **Rotate View** tool (or alternatively hold down Ctrl+Alt+Mouse Key 1) to rotate the model.
 - e. When the midlines at both ends have been highlighted as indicated in Figure 5, click **Done** in the prompt area.
 - f. In the **Edit Boundary Condition** box, check U1. Then click **OK** to exit the box.

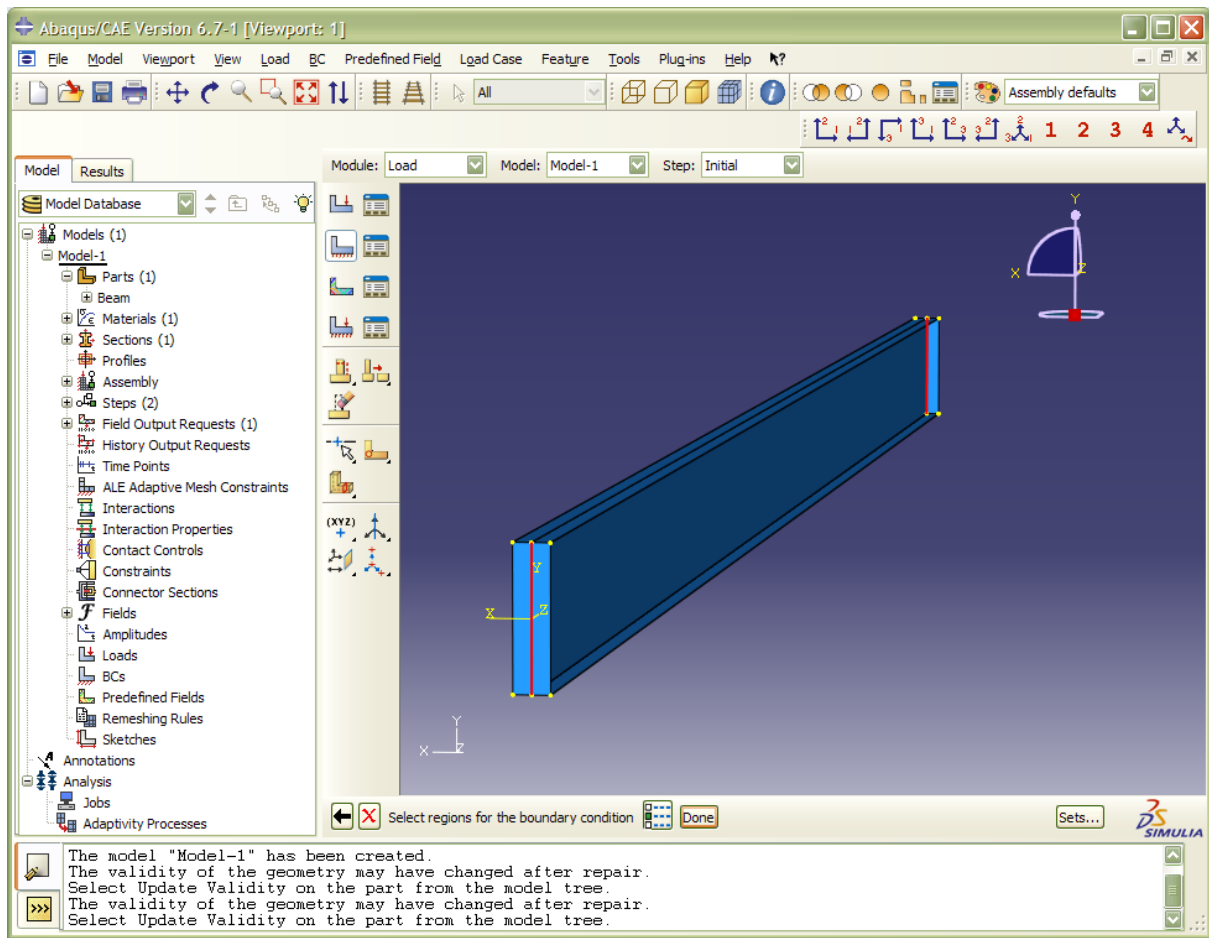


Figure 5 The midlines at both ends have been selected for application of supports in the x-direction.

2. Use the **Create Boundary Condition** tool to define supports for the beam in the y-direction.
 - a. Change **Name** to Fix-y.
 - b. Keep **Step** as Initial.
 - c. Keep **Category** as Mechanical and **Type for Selected Step** as Displacement/Rotation. Then click **Continue ...**
 - d. Select the bottom edges at both ends of the beam and click **Done** in the prompt area. Note that the bottom edge at either end is composed of two lines. The boundary conditions should be applied along the entire edge.
 - e. In the **Edit Boundary Condition** box, check U2. Then click **OK** to exit the box.
3. Use the **Create Boundary Condition** tool to define supports for the beam in the z-direction.
 - a. Change **Name** to Fix-z.
 - b. Keep **Step** as Initial.
 - c. Keep **Category** as Mechanical and put **Type for Selected Step** as Displacement/Rotation. Then click **Continue ...**
 - d. Select the bottom edge at one end of the beam and click **Done** in the prompt area.
 - e. In the **Edit Boundary Condition** box, check U3. Then click **OK** to exit the box.
4. Now, apply the **Create Load** tool to define a moment at one end for the buckle analysis.
 - a. Change **Name** to Moment 1.
 - b. Change **Step** to Instability.
 - c. Keep **Category** as Mechanical and change **Type for Selected Step** to Moment. Then click **Continue ...**

- d. Select the mid-points at the bottom edge and the top edge at one end of the beam. Again, use Shift to mark more points at a time. When the visualization of the model looks like the illustration on Figure 6, click **Done** in the prompt area.
 - e. In the Edit Load box, set **CM1** to -0.5. This produces a bending moment with a total magnitude of 1 around the global x-axis and leading to tension in the bottom flange.
 - f. The moment is indicated by two-headed arrows pointing in the negative x-direction.
5. Again, apply the Create Load tool to define a moment at the other end of the beam.
 - a. Change **Name** to Moment 2.
 - b. Keep **Step** as Instability, **Category** as Mechanical and **Type for Selected Step** as Moment. Then click **Continue ...**
 - c. Select the mid-points at the bottom edge and the top edge at the other end of the beam. Then click **Done** in the prompt area.
 - d. In the Edit Load box, set **CM1** to 0.5. This produces a bending moment with a total magnitude of 1 around the global x-axis and leading to tension in the bottom flange.
 - e. The moment is indicated by two-headed arrows pointing in the positive x-direction.

The boundary conditions defined for the model correspond to a simply supported beam which has been forked at both ends, i.e. torsional rotation is not possible at the ends of the beam. In the buckle analysis, the bending moments applied at the ends lead to a homogeneous moment along the beam. This corresponds to the boundary conditions in the classical torsional–lateral instability problem.

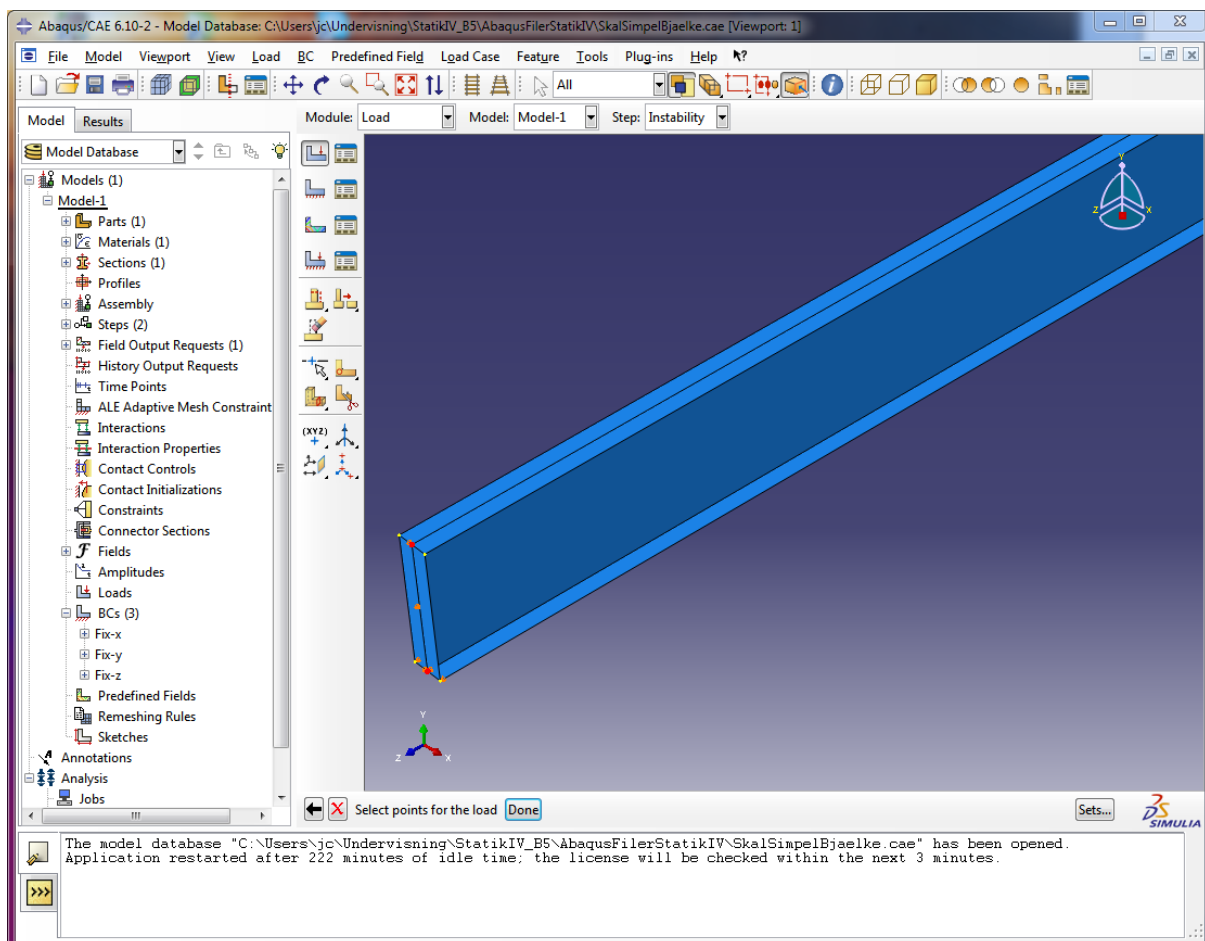


Figure 6 Points at which the moment is applied at one end of the beam.

Discretization into Finite Elements

Proceed to the **Mesh** module to create the finite element model of the simply supported beam.

1. In the context bar, change **Object** from Assembly to Part. The part is automatically identified as Beam. The context bar is located just right of the Module Menu.
2. Use the **Seed Part** tool to define nodes along the edges of the beam model.
 - a. Change **Approximate global size** to 0.32 and click **Apply**.
 - b. There should now be 26 nodes along the beam and three nodes over the width. Click **OK**.
3. Apply the **Assign Mesh Controls** tool to set the meshing technique.
 - a. Select the entire model using the mouse. Then click **Done**.
 - b. In the Mesh Controls box, set **Element Shape** to Quad.
 - c. Set **Technique** to Structured and click **OK** to close the box.
 - d. The beam will now change colour to green.
 - e. Click **Done** in the prompt area.
4. Utilize the **Assign Element Type** tool to select a shell finite element for the model.
 - a. Select the entire model using the mouse. Then click **Done**.
 - b. Keep **Element Library** as Standard and **Family** as Shell.
 - c. Change Geometric order to Quadratic.
 - d. **DOF per node** should be 6 and the remaining **Element Controls** should all be set to the default values.
 - e. Click **OK** to close the box.
5. Use the **Mesh Part** tool to form the finite element model.
 - a. Click Yes in the prompt area to apply the mesh.
 - b. The model now changes colour to cyan and the elements should be visible.
6. Save the model before you continue.

Running the analysis

Go to the Job module to submit the analysis to Abaqus/Standard. This is the solver, which provides an output database that may later be accessed from Abaqus/CAE.

1. Use the Create Job tool to define a new analysis.
 - a. Change **Name** to I-beam-01.
 - b. Click **Continue ...**
 - c. In the Edit Job box, simply click **OK** to close the box.
2. Open the **Job Manager** from the tools menu.
 - a. Only a single job should be defined at this stage, i.e. I-beam-01. This job is automatically selected.
 - b. Click **Submit** to run the analysis. The status will change to Submitted.
 - c. After a few seconds, the status will change to Completed, provided that there were any analysis tokens available from the license server at the time.
 - d. When the job has been completed, click **Results**. This will take you to the **Visualization** module and automatically open the output database I-beam-01.odb generated by Abaqus/Standard during the analysis.

Visualisation and post processing of the results

Abaqus CAE can plot the displacements and other field quantities in numerous ways. This takes place in the **Visualization** module.

1. Try the **Plot Contours on Deformed Shape** tool. This will provide a plot as illustrated in Figure 6.
2. In the analysis, the first four instability modes were identified. You can skip through the different modes by clicking the **Next** button (or **Previous** button) in the context bar.
3. As indicated in Figure 7, the EigenValue corresponding to the first instability mode, i.e. Mode 1, is 15663. This means that the beam can sustain a moment of 15663 before instability occurs.
4. The EigenValue corresponding to Mode 2 is -18681. The deformation pattern is the opposite situation in comparison to Mode 1. Hence, the beam can sustain a slightly higher bending moment of this kind before instability occurs. This difference is due to the asymmetric placement of the fixities with respect to the centre axis of the beam.

