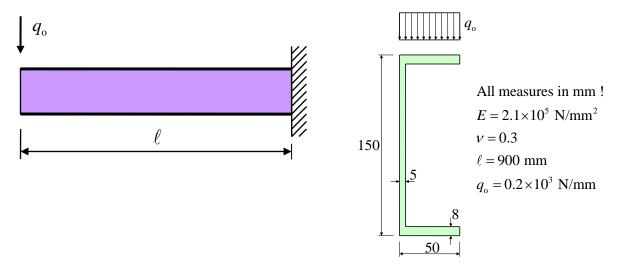# TKT4142 Finite Element Methods in Structural Engineering
Workshop 6

This workshop will briefly guide you through Case Study 6. Case Study 6 will use a thin-walled canal cross-section to address the modelling of shell problems (i.e., h/L < 1/10).

In Section 1, shell elements will be used to model the cantilevered beam, whereas Section 2 deals with 1D beam elements.

1. Shell elements

Figure 1-1 shows a cantilevered beam with a thin-walled canal cross section. The beam is fixed in one end and a line load is applied to the top flange in the other end.

All measures in mm !
$E = 2.1 \times 10^5 \text{ N/mm}^2$
$v = 0.3$
$\ell = 900 \text{ mm}$
$q_o = 0.2 \times 10^3 \text{ N/mm}$

Figure 1-1 – The model of a thin-walled canal cantilevered beam.

First, make sure your **work directory** is properly set in Abaqus/CAE.

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

1.1. Creating a Part

1. Create a new part in **BEAM_SHELL** by double-clicking on **Parts** in the **Model Tree.**
   The **Create Part** dialog box appears.
2. In the **Create Part** dialog box, see Figure 1-2(a):
   a. Name the part **Beam**.
   b. Choose **3D** in the **Modeling Space.**
   c. Choose **Deformable** as the **Type**.
   d. Use **Shell** as the **Base Feature** and **Extrusion** as **Type**.
   e. **Approximate size** can be set to **200**.
   Click on **Continue**. This brings you into the sketch environment.

3. Sketch the geometry in Figure 1-1. We will use mm as measurements. Use **Create Lines: Connected** tool (📐) and create the "⊏-shape" of the center of the geometry. A sketch of the part is shown in Figure 1-2(b).
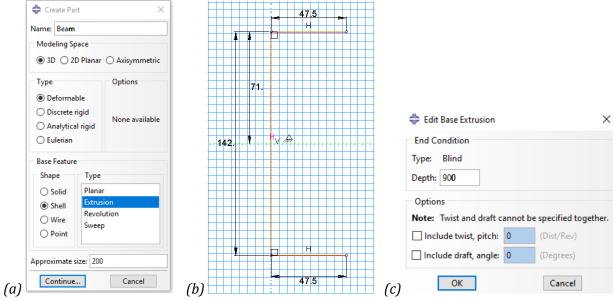4. In the **Edit Feature** dialog box, use a **Depth** of **900**. Click on **OK** to extrude your sketch.

(a)   *(b)*   *(c)*

Figure 1-2 – (a) The create part dialog box, (b) the sketch, and (c) the Edit Base Extrusion dialog box.

1.2.   Assigning properties

1. Define a linear **Material** named **Steel**. The **Young's Modulus** and **Poisson's Ratio** are set to **210 000** and **0.3**, respectively.
2. Create a **Homogeneous Shell Section** named **Web**. Make sure to select **Shell** in **Category**. See Figure 1-3(a). Click on **Continue**.
3. In the **Edit Section** dialog box, input a **Shell thickness** value of **5**. See Figure 1-3(b). Click on **OK**.
4. Repeat step 2 and 3, making a new **Section** named **Flange** with a **thickness** value of **8**.
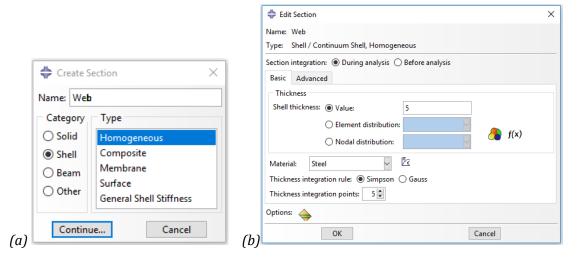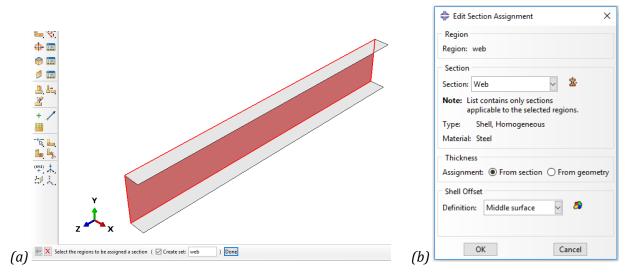

(a)   *(b)*

Figure 1-3 – (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.

5. In the **Model Tree**, expand the branch for the part **Beam**. Double-click **Section Assignments** to assign a section to the part Beam.
6. Select the web of the beam and check **Create set** in the prompt. See Figure 1-4(a). Call the set **web** and click **Done**.
7. In the **Edit Section Assignment** dialog box, select the Web as the section. Use **From section** as **Thickness** and **Middle surface** as **Shell offset**. See Figure 1-4(b).
8. Repeat steps 5-7 for the flanges.

Your part should have turned green before you proceed.

(a)   *(b)*

Figure 1-4 – (a) Selecting the web 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** as done in Section 2.4. Choose **Static, General** as **Procedure type**. Set the **Time period** to **1** and **Initial Increment size** to **0.1**.

1.5. Applying Load and Boundary Condition to the Model
You must first apply **boundary conditions** to your model by fixing the right edge.

1. Create a **BCs** called **Fixed.** In the **Create Boundary Condition** dialog box
   a. **Step** is set to **Initial**.
   b. **Types for Selected Step** is set to **Displacement/Rotations**.
2. Select the **right edges** in the viewport. See Figure 1-5(a). Click **Done** in the prompt area.

3. Check **U1**, **U2, U3, UR1, UR2** and **UR3** in the **Edit Boundary Condition** dialog box. See Figure 1-5(b). This will lock the left edge from displacements and rotations. Note that we must constrain the rotations as well since the shell element also has rotational dofs in the nodes.
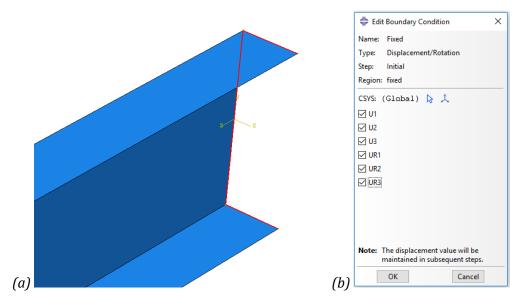

(a)   (b)

Figure 1-5 – (a) The selected right edges and (b) The Edit Boundary Condition dialog box.

Next, you need to **define the distributed load** at the left top flange.

1. Create a new **Load** called **Load**. This opens the **Create Load** dialog box. See Figure 1-6(a). Here
   a. **Step** is set to **Load.**
   b. **Types for Selected Step** is set to **Shell Edge Load.**
   c. Click **Continue.**
2. Select the **left-upper edge** in the viewport. Click **Done** in the prompt area.
3. In the **Edit Load** dialog box, see Figure 1-6(b)
   a. Set **Distribution** to **Uniform**
   b. Set **Traction** to **Transverse**.
   c. Set **Magnitude** to **- 215.5**[1].
   d. Use **(Ramp)** as **Amplitude**.
   Make sure that the load is applied in the downward direction in your model. If not, change the sign for the magnitude.

---

[1] Note that $P = q_0 b = \tilde{q}_0 \tilde{b} \Rightarrow \tilde{q}_0 = \frac{q_0 b}{\tilde{b}} = 200 \cdot \frac{50}{47.5} \text{N/mm} = 210.5 \text{ N/mm}$.
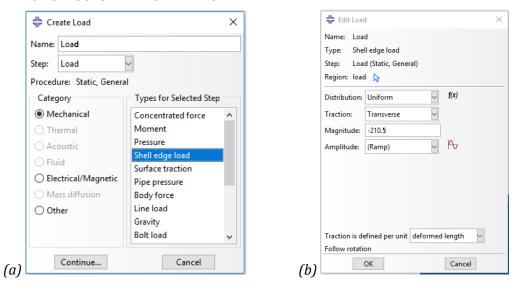
(a)   *(b)*

Figure 1-6 – (a) The Create Load dialog box and (b) The Edit Load dialog box.
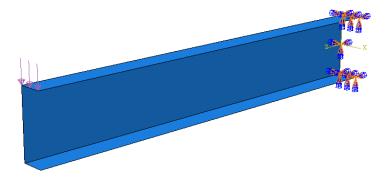
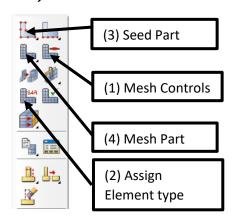Your model should look like Figure 1-7.

Figure 1-7 – The model with load and BC.

1.6.   Meshing the Model

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

Figure 1-8 – The different tools for meshing a part in this workshop.

1. From the tools on the left ribbon, choose **Mesh Controls**. See (1) in Figure 1-8. In the **Mesh Controls** dialog box
   - For S4 and S8 elements: Use **Quad** as **Element type** and **Structured** as **Technique**. See Figure 1-9(a).
   - For S3, STRI3 and STRI65 elements: Use **Tri** as **Element type** and **Structured** as **Technique**.
2. From the tools on the left ribbon, choose **Assign Element Type**. See (2) in Figure 1-8. 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
   - STRI3:   3-node linear triangular thin shell element for small deformations
   - STRI65: 6-node quadratic triangular thin shell element

   You can read more about the different types of shell elements in the Abaqus documentation. 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. The accuracy of the membrane part of S4 is similar to a QM6 element and significantly better than a fully integrated or selectively reduced integrated Q4 element.
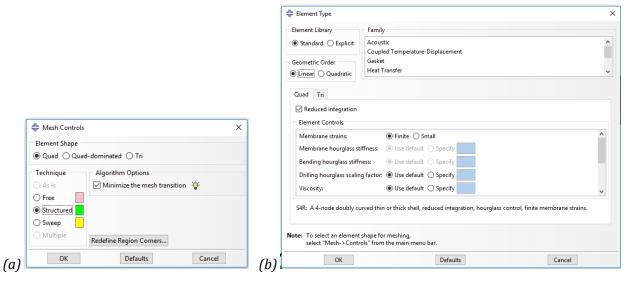

Figure 1-9 – (a) The Mesh Controls and (b) Element Type dialog box.

3. From the tools on the left ribbon, choose **Seed Part**. See (3) in Figure 1-8. Use an **Approximate global size** of **30.**
4. From the tools in the left ribbon, choose the **Mesh Part**. See (4) in Figure 1-8(b).
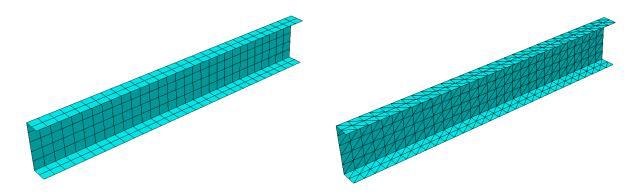
Figure 1-10 – Meshed part with quadratic and triangular shell elements.

1.7.   Creating and Submitting an Analysis

Create and submit your job.

2. Beam elements

We will now model the beam using 3D beam elements, but it is also possible to do the same in a 2D modeling space.

First, create a new model by double-click on **Models** in the **Model Tree**. The **Edit Model Attributes** dialog box appears. Name the model **BEAM_1D**.

2.1.   Creating a Part

1. Create a new part in **BEAM_1D** by double-click on **Parts** in the **Model Tree.**
   The **Create Part** dialog box appears. See Figure 2-1.
2. In the **Create Part** dialog box:
   a. Name the part **Beam**.
   b. Choose **3D** in the **Modeling Space.**
   c. Choose **Deformable** as the type.
   d. Use **Wire** as the **Base Feature**.
   e. **Approximate size** can be set to **2000**.
   Click on **Continue**. This brings you into the sketch environment.
3. Sketch a line using the **Create Lines; connected** tool ( ), and give it a total length of **900**.
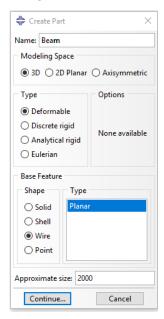   See Figure 2-2. Click on **Done** in the prompt area.

Figure 2-1 – The Create Part dialog box
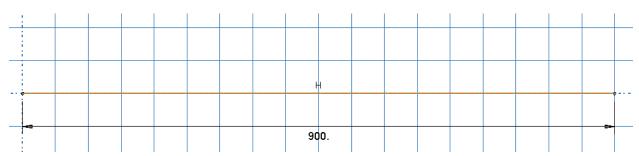

Figure 2-2 – The sketch

2.2.   Assigning properties

1. Define a linear **Material** named **Steel**. The **Young's Modulus** and **Poisson's Ratio** are set to **210 000** and **0.3**, respectively.
2. Define a profile by double-click on **Profiles** in the **Model Tree**.
   The **Create Profile** dialog box appears. See Figure 2-3(a).
3. Name the profile **Beam** and select **Arbitrary** as the shape. Click **Continue.**
   The **Edit Profile** dialog box appears.
4. Input the geometry and thickness, as shown in Figure 2-3(b).

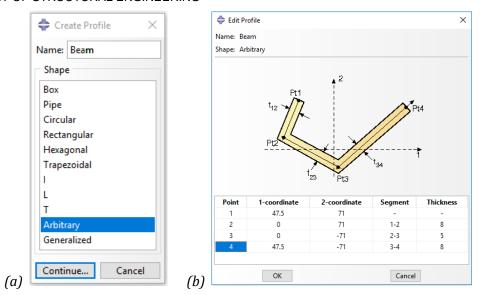| Point	1-coordinate	2-coordinate	Segment	Thickness
1	47.5	71	-	-
2	0	71	1-2	8
3	0	-71	2-3	5
4	47.5	-71	3-4	8

Figure 2-3 – (a) the Create Profile dialog box and (b) the Edit Profile dialog box.

5. Create a Section. In the **Create Section** dialog box, see Figure 2-4(a)
   a. Name the Section **BeamSection**.
   b. Use **Beam** as **Category.**
   c. Use **Beam** as **Type.**
   d. Click on **Continue** to continue.
   The **Edit Beam Section** dialog box appears. See Figure 2-4(b)
6. In the **Edit Beam Section** dialog box:
   e. Set the profile **Beam** as the **Profile Name**.
   f. Set **Steel** as the material.
   g. **Section Poisson's ratio** can be set to **0**.
   h. The other settings are accepted as default. Click on **OK**.
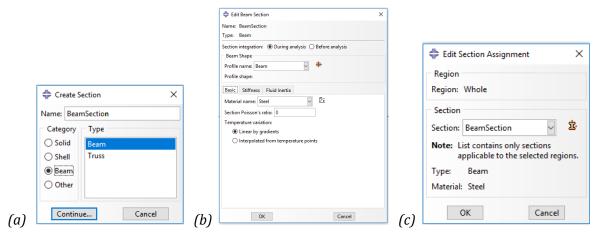7. **Assign** this section to your part as done previously.

Figure 2-4 – (a) The Create Section, (b) the Edit Beam Section and (c) the Edit Section Assignment dialog box.

9

8. Next, we must define the **beam orientation**. In the main menu bar, select **Assign → Beam Section Orientation**. See Figure 2-5.
Select your beam and click on **Done** in the prompt area, Figure 2-6(a).
Define the **tangent vector** as **0.0, 0.0, -1.0**, Figure 2-6(b).
The **1** and **2** directions will be displayed on you part, as shown in Figure 2-6(c). These directions correspond to the direction of the profile as shown in Figure 2-3(b).
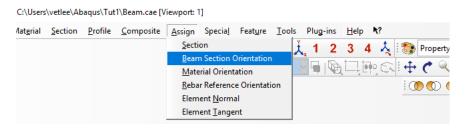

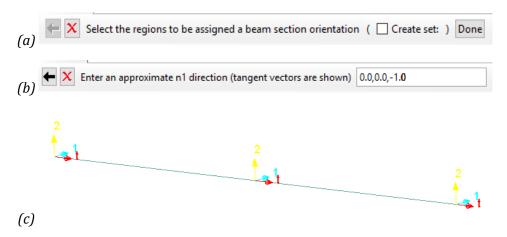*Figure 2-5 – The Beam Section Orientation tool.*

(a)

(b)

(c)

Figure 2-6 – (a)/(b) The prompt area and (c) the beam orientation in the viewport.

9. If you want to render the profile in your preview, select **View → Part Display Options** from the main menu bar.
The **Part Display Options** dialog box appears, Figure 2-7. Check the **Render Beam Profile** box. Your profile is now displayed in the viewport.
NB: Note that once you enter the assembly module, your profile will not be rendered. If you work with an assembly, the same can be achieved from **View → Assembly Display Options**.
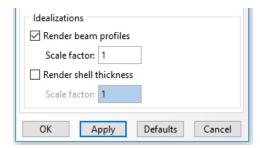

Figure 2-7 – Some of the Part Display Option dialog box.

2.3.   Create an Assembly

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

2.4.   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**.

2.5.   Applying Load and Boundary Condition to the Model

Apply **a fixed boundary condition** to the right-most node called. Use the **Initial** step. We will use **Symmetry/Antisymmetry/Encastre** as **Type** this time. Select **ENCASTRE** in the Edit Boundary Condition dialog box as shown in Figure 2-8(b). Notice that this is the same as locking all degrees of freedoms as done in Section 1.5.
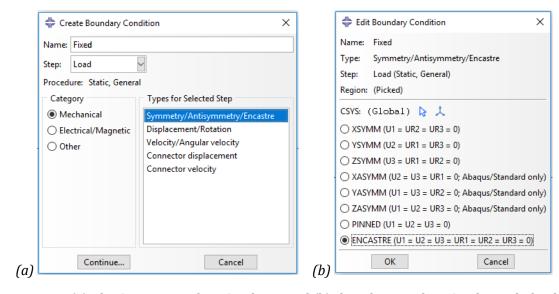

(a)   *(b)*

Figure 2-8 – (a) The Create Boundary Condition and (b) the Edit Boundary Condition dialog box.

We must define a load at the left-most node. Since the force is acting on the flange, both a concentrated force as well as a concentrated torque must be applied. These are equal to

$$P = -q_0 b = -200 \times 50\text{N} = -10000\text{N}$$

$$M = \frac{P\tilde{b}}{2} = -\frac{10000 \times 47.5}{2}\text{Nmm} = -237500\text{Nmm}$$

First, assign a **Concentrated force** to the left-most node of the beam. Use **Load** as **Step** and **Concentrated force** as **Type**. See Figure 2-9(a). In the Edit Load dialog box, Figure 2-9(b), set **CF2** to -10000.
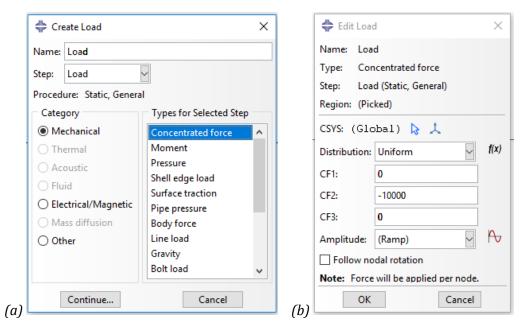

(a)   (b)

Figure 2-9 – (a) The Create Boundary Condition and (b) the Edit Boundary Condition dialog box.

Next, assign a **Moment** to the left-most node of the beam. Use **Load** as **Step** and **Moment** as **Type**. See Figure 2-10(a). In the Edit Load dialog box, Figure 2-10 (b), set **CM1** to -237500.
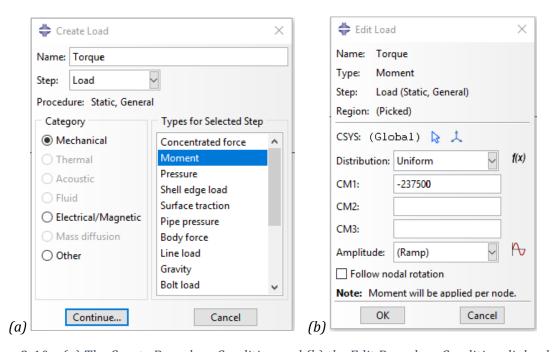

(a)   (b)

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

Figure 2-11 shows how the model with BC and loads.

Figure 2-11 – Model with BC and loads.

2.6.   Meshing the Model

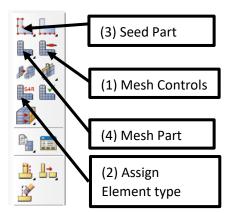Navigate to the **Mesh module**. Make sure that **Object** is set to **Part**.

Figure 2-12 – The different tools for meshing a part in this workshop.

1. From the tools on the left ribbon, choose **Assign Element Type**. See (2) in Figure 2-12. The following elements can be interesting to investigate
   - B33 – A 2-node cubic beam in space. Euler-Bernoulli theory.
   - B31 – A 2-node linear beam in space. Timoshenko theory.
   - B32 – A 3-node quadratic beam in space. Timoshenko theory.
   - B31OS – A 2-node linear open-section beam in space. Timoshenko theory.
   - B32OS – A 3-node quadratic open-section beam in space. Timoshenko theory.

You can read more about the different types of shell elements in the Abaqus documentation. Note that you are not able to select the B2* elements since the beam is defined in 3D modeling space in Section 2.1. In general:

In ABAQUS one can choose between elements based on Euler-Bernoulli and Timoshenko beam theory. Euler-Bernoulli is often referred to as elementary beam theory, assumes that the deformation occurs according to Navier's hypothesis; plane cross sections that are normal to the beam axis remain plane and normal to the beam axis after bending. In ABAQUS is Euler-Bernoulli beam elements (B23 and B33) based on cubic interpolation polynomials of both axial and transverse displacements. B23 and B33 are C^1-elements, that in addition to bending with regards to the beam's two main axis also describe twisting or torsion.

Beam elements based on Timoshenko beam theory (B21, B22, B31 and B32) are C^0-elements, that in addition to bending of the two main axis also describe torsion and transverse shear strains for both «thin» and «thick» beams. The C^0-elements in ABAQUS are available as both 2-

node linear (B21 and B31) and 3-node quadratic (B22 and B32) elements. All Timoshenko elements in ABAQUS are formulated with reduced integration, 1-point for linear and 2-point Gauss integration for the quadratic element. This is done to avoid issues with shear locking.

As opposed to the C^1-elements that only can describe small strains the C^0-elements are formulated to describe large membrane (axial) and bending strains, while transverse and torsional strains are assumed to be moderate («small»). Even if C^0-elements are more general than the C^1-elements, the C^1-elements are much better suited for slim beam structures where the shear deformations are of lesser importance. It is for such cases much more accurate, and a higher convergence rate especially in terms of displacements, $e_u = o(h^{p+1})$, while for strains/stresses, $e_\varepsilon = e_\sigma = O(h^{p+1-m}) \Rightarrow e_\sigma^{EB} = O(h^2), e_\sigma^{T1} = O(h^1)$ and $e_\sigma^{T2} = O(h^1)$, where e_σ^{EB} express the convergence rate for Euler-Bernoulli elements, while e_σ^{T1} and e_σ^{T2} express the convergence rate for linear and quadratic Timoshenko elements respectively.

For thin-walled open cross section, composed of thin segments it may occur warping of the cross section. If warping is restricted at the fixed end, there will be axial stresses in the flanges due to bending of the flanges in their own plane. The C^0-elements B31OS and B32OS in ABAQUS are linear and quadratic 3D beam elements for modeling of "Open Section". The formulation requires that warping deformation is small compared with the other strains.
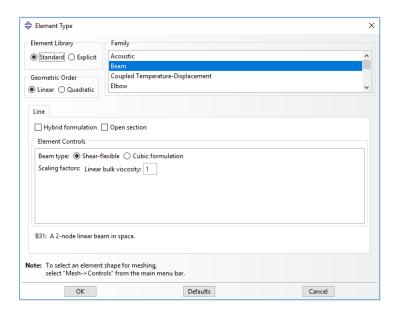

Figure 2-13 – The Element Type dialog box.

2. From the tools on the left ribbon, choose **Seed Part**. See (3) in Figure 2-12. Use an **Approximate global size** of **30.**
3. From the tools in the left ribbon, choose the **Mesh Part**. See (4) in Figure 2-12.

2.7.   Creating and Submitting an Analysis

Create and submit your job.