

Cantilever Beam Bending Analysis

Type of Solver: ABAQUS CAE/Standard

TLP: Bending and Torsion of Beams -http://www.doitpoms.ac.uk/tlplib/beam_bending/index.php

Continuum Mechanics – Beam Bending

Problem Description:

Consider the cantilever beam shown below. The beam is made from aluminium, which has a Young's modulus of $E = 70$ GPa, a shear modulus of $G = 25$ GPa, and a Poisson's ratio of $\nu = 0.33$. The beam is 1 m in length ($L = 1$) and has a square section with $a = b = 0.025$ m. When a transverse load is applied at some distance (x) along the beam length, a bending moment, M , is generated, where:

$$M = EI \frac{d^2 y}{dx^2} = F(L - x) \quad (1)$$

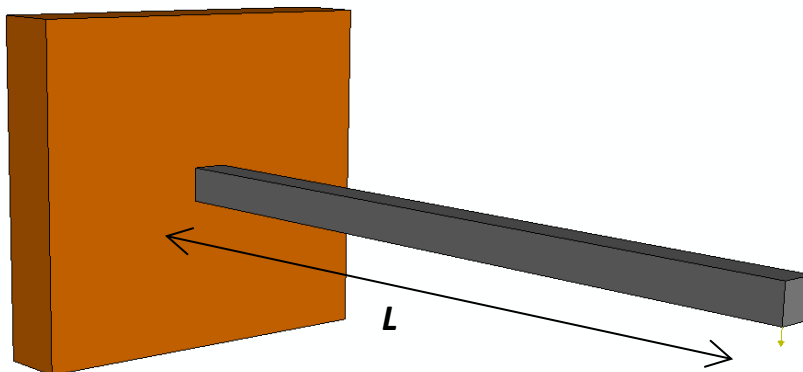
The deflection of the beam is given by:

$$\delta = \frac{Fx^2(3L - x)}{6EI} \quad (2)$$

I is the second moment of area. For a square cross section:

$$I = \frac{a^4}{12} \quad (3)$$

X is the distance from the clamped end!!



- (a) Using a 1-dimensional finite element model, compute the deflection of a cantilever beam loaded at its end with a force of 80 N. Compare the FEM predicted deflections with those predicted by ordinary beam bending theory. Assume that the beam is made from aluminium, is homogenous and isotropic, and that it behaves in a linear elastic fashion.
- (b) Using a 3-dimensional finite element model, compute the deflection of a cantilever beam loaded at its end with a force of 80 N. Compare the FEM predicted deflections, with those predicted by ordinary beam bending theory. Assume that the beam is made from aluminium, is homogenous and isotropic, and that it behaves in a linear elastic fashion.
- (c) Using the 3-dimensional FE model, investigate the effect of **mesh density** on the predicted FEM deflections. Re-mesh the cantilever beam with 5000, C3D8R elements, and re-run the analysis. Compare the subsequent FEM predicted deflections with those of ordinary beam bending theory.
- (d) Investigate the effect of **element type** on the predicted FEM deflections. Re-mesh the cantilever beam with 5000, C3D20R elements, and re-run the analysis. Compare the subsequent FEM predicted deflections with those of ordinary beam bending theory. Why might these elements be more accurate?
- (e) Using the 3-D FE model (5000 C3D20R elements) plot the distribution of stress through the section of the model at $x = 0.1$.
- (f) Compare the predicted stress at $x = 0.1$ for $y = 0.004$ and $y = 0.0125$ (where y is the distance from the neutral axis) with the stress at those positions predicted by ordinary beam bending theory. Plot these predictions on the same graph.

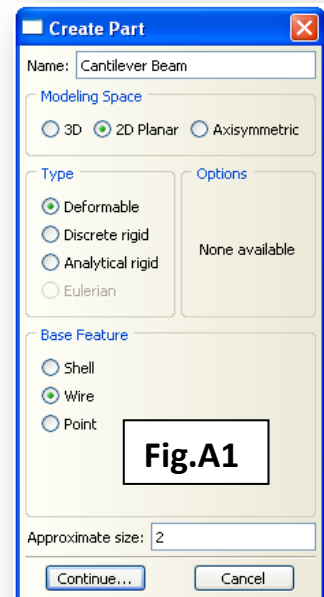
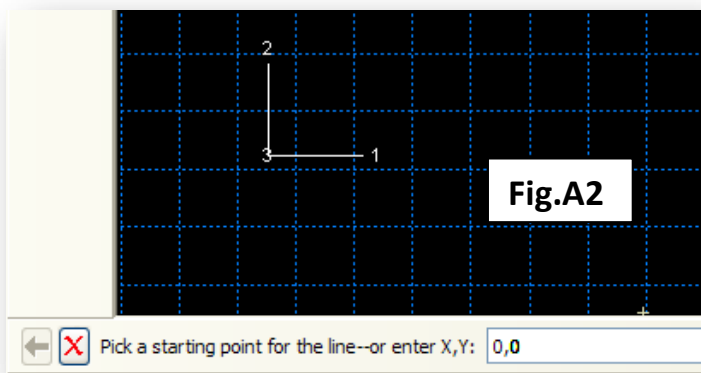
Solution (a):

- Start ABAQUS/CAE. At the **Start Session** dialogue box, click **Create Model Database with Standard/Explicit Model**.
- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialogue box appears; name the model `Cantilever_1D`

A. MODULE→PART

Under the Part module, we will construct the beam (1-D)

1. From the main menu bar, select **Part→Create**
2. The **Create Part** Dialogue box appears. Name the part `Cantilever Beam` and fill in the options as shown in **Fig.A1**. Click **Continue** to create the part.
3. From the main menu bar, select **Add→Line→Connected Line**
 - (a) Select the co-ordinates $(0, 0)$ for the first vertex (**enter**) as shown in **Fig.A2**.
 - (b) Select the co-ordinates $(1, 0)$ for the second vertex (**enter**).
 - (c) Click **X** in the prompt area.
 - (d) Click **Done** in the prompt area

**Fig.A1****Fig.A2**

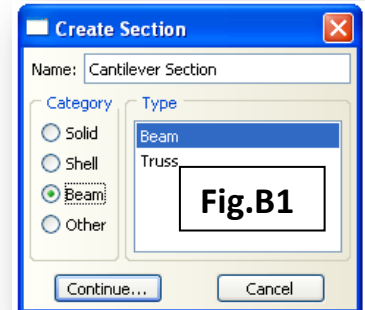
The part, `Cantilever Beam` will now appear in the **Viewer** window, as a 1-dimensional beam. The following tasks must be completed:

- The beam section geometry must be defined.
- The beam material properties must be defined.
- The boundary conditions (constraints and loads) must be defined.
- A mesh must be assigned.

B. MODULE→PROPERTY

In this module (property), you will define the beam geometry (width and height), you will define the beam material properties (E , G and ν) and you will assign these material properties to the beam.

1. From the main menu bar, select **Section→Create**. The **Create Section Dialogue** box will open as shown in **Fig.B1**. Name the section `Cantilever Section`.

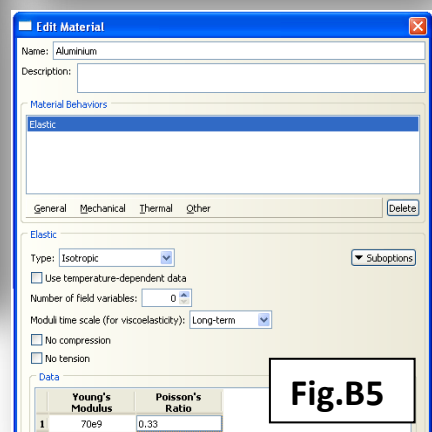
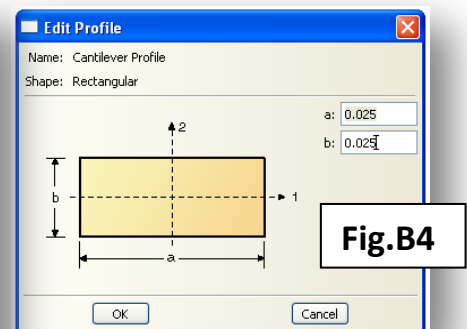
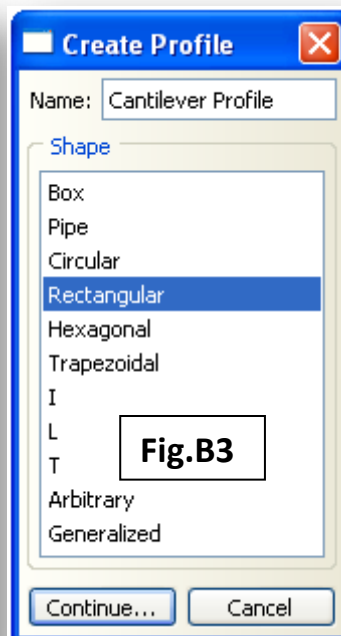
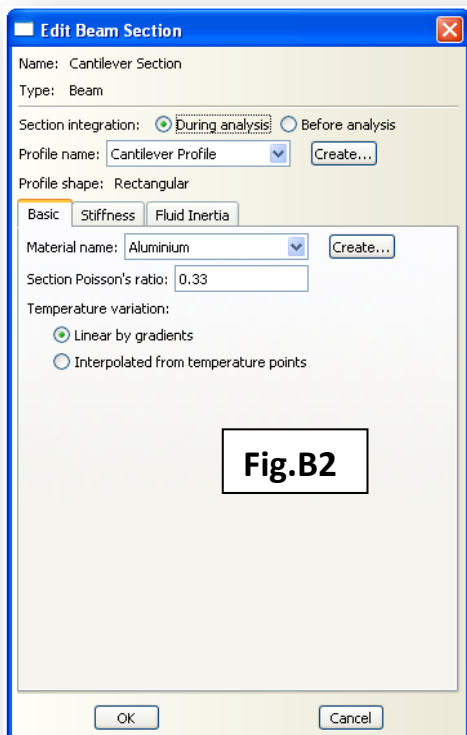


(a) Under **Category**, choose **Beam**.

(b) Under **Type**, choose **Beam**.

(c) Click **Continue**.

2. The **Edit Beam Section** dialogue box will open (**Fig.B2**). Next to **Profile Name**, click **Create** and the **Create Profile** dialogue box will open (**Fig.B3**). Name the profile `Cantilever Profile`. Select **Rectangular** and click **Continue**. The **Edit Profile** dialogue box will open (**Fig.B4**). Enter the cantilever cross sectional dimensions ($a = b = 0.025$ m) and click **OK**.



4. From the main menu bar select **Assign→Section**. Use the mouse cursor to select the Cantilever Beam part and select **Done** in the prompt area. The **Edit Section Assignments** dialogue box will open as shown in **Fig.B6**.

- (a) Check that Cantilever Section is selected under the **Section** options.
- (b) Check that the **Type** is **Beam**.
- (c) Click **OK**.

It is now necessary to define a beam orientation (*this is important for the second moment of area (I) calculation, particularly if the beam has geometry where $a \neq b$. In this case, $a = b$, and the moment of area is independent of the loading direction; however, by default, ABAQUS requires a beam orientation to be defined for all beam sections).

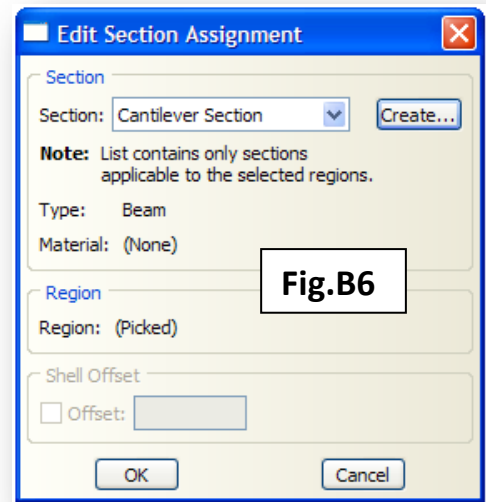


Fig.B6

5. From the main menu bar select **Assign→Assign Beam Section Orientation**. Use the mouse cursor to select the part in the **Viewer** and click **Done** in the prompt area. The default orientation can be selected by pressing **Enter**, clicking **OK** and then clicking **Done**.

C. MODULE→ASSEMBLY

In the assembly module, multiple parts can be 'assembled' into an assembly of parts. This is done by creating 'instances' of each part. In this case, we have only one part (Cantilever Beam). ABAQUS still requires, however, that an instance of this part is created. (FYI multiple instances of a single part can be created if required.)

1. From the main menu bar select **Instance→Create**. The **Create Instance** dialogue box will open (**Fig.C1**).

- (a) Under Instance **Type**, select **Independent**.

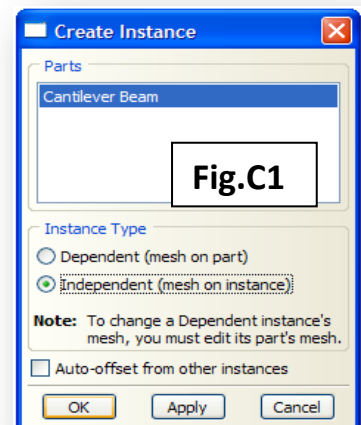


Fig.C1

D. MODULE→STEP

In the step module, you will define the type of analysis that is to be undertaken (static in this case).

1. From the main menu bar, select **Step→Create**. The **Create Step** dialogue box will appear (**Fig.D1**). Name the step Load Step.

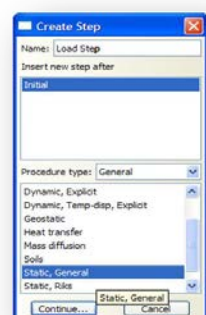


Fig.D1

2. Select **General** from the **Procedure Type** options.
3. Select **Static, General** from the list of analysis types. Click **Continue** and **OK**.

E. MODULE→INTERACTION

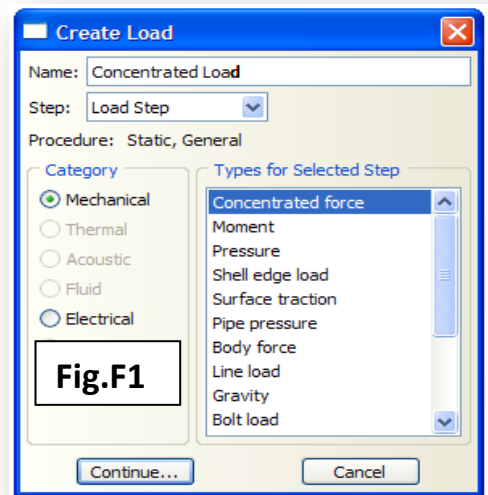
There are no interactions in this analysis.

F. MODULE→LOAD

In the load module you define the boundary conditions (constraints and loads). You will constrain one end of the cantilever beam to be fixed (zero displacements) and you will define an 80 N load at the free end of the beam.

1. From the main menu bar, select **Load→Create**. The **Create Load** Dialogue box will open (**Fig.F1**).

- (a) Name the load **Concentrated Load**.
- (b) Choose **Load Step** as the **Step** option.
- (c) Choose **Mechanical** for the **Category**.
- (d) Choose **Concentrated Force** for the **Type**.
- (e) Click **Continue**.
- (f) Using the mouse cursor, select the second vertex (node)
- (g) Click **Done** in the prompt area.

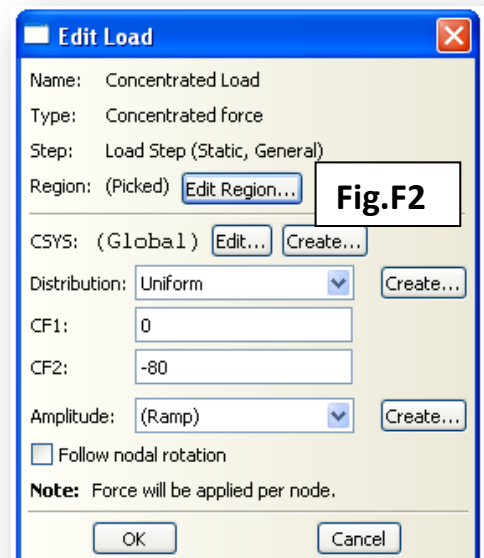


2. The **Edit Load** dialogue box will open, as shown in **Fig.F2**. You now need to specify a transverse load of 80 N.

- (a) Input $CF1 = 0$ (load in the x direction)
- (b) Input $CF2 = -80$ (load in the y direction)
- (c) Toggle off **Follow Nodal Rotation**

This ensures that the load is continuously applied in the y direction and not in a direction normal to the tangent of the node.

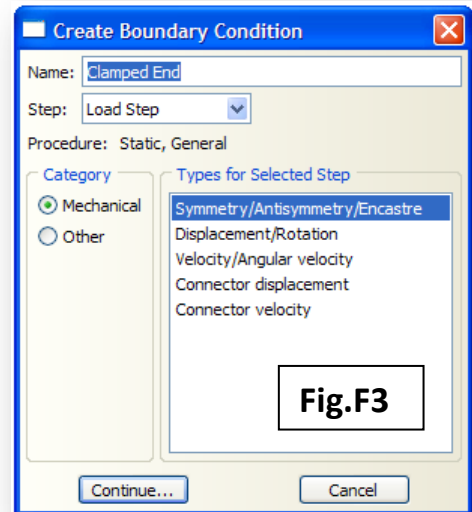
- (d) Click **OK**.



It is now necessary to define the constraint boundary conditions (i.e. to fix the opposite end of the beam so that it cannot move).

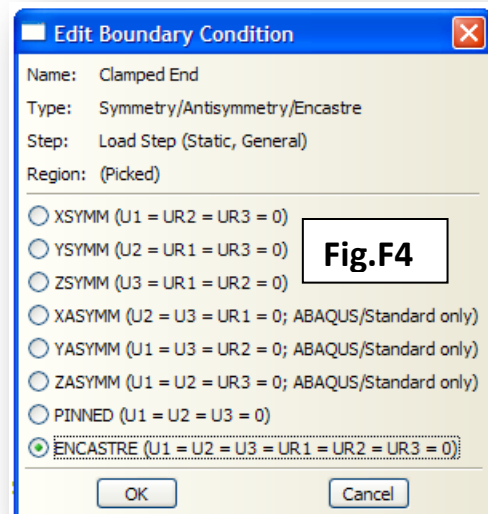
3. From the main menu bar, select **BC→Create**. The **Create Boundary Condition** dialogue box will appear, **Fig.F3**.

- Name the boundary condition, **Clamped End**.
- Choose **Load Step**, for the **Step** type.
- Choose **Mechanical** for the **Category**.
- Choose **Symmetry/Antisymmetry/Encastre** for the **Types for Selected Step** option.
- Click **Continue**.
- Use the mouse cursor to select the first vertex (node) which is going to be clamped.
- Click **Done** in the prompt area.



The Edit Boundary Condition dialogue box will open as seen in **Fig.F4**.

- Select **Encastre** as the boundary condition.
- Click **OK**.



G. MODULE→MESH

In this mesh module, you will mesh the **Cantilever Beam** instance, by assigning seeds (nodal positions), mesh controls and element types.

- From the main menu bar, select **Seed→Edges**.
- Use the mouse cursor and select the **Cantilever Beam** instance and click **Done** in the prompt area.

In the **Local Seeds** dialogue box that appears, change the seeding **method** to **By Number**

3. Type **20** for the number of nodes along the beam length.

4. Click **OK**

5. From the main menu bar, select **Mesh→Element Type**. Using the cursor select the part instance and click **Done** in the prompt area. The **Element Type** dialogue box will appear as shown in **Fig.G1**.

6. Choose **Standard** from the **Element Library**.

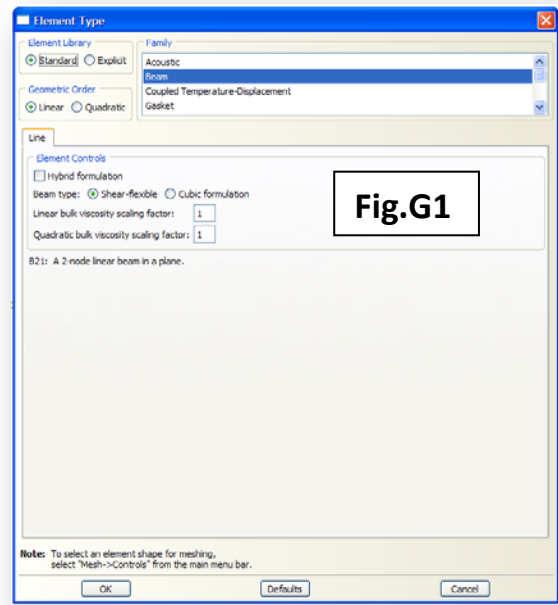
7. Choose **Linear** for the **Geometric Order**.

8. Choose **Beam** for the **Family**.

9. Click **OK**.

10. From the main menu bar, select **Mesh→Instance**

11. Click **Yes** in the prompt area.



You have so far built the geometry, prescribed the beam section geometry, the beam material properties and the beam section orientation. You have created an instance of the **Cantilever Beam** part, defined a constraint boundary condition and a loading boundary condition. You have meshed the instance with 20, two-node, linear B21 beam elements. All that is now required is for the job to be submitted to the solver.

H. MODULE→JOB

In this module, you will submit the job to the solver for analysis.

1. From the main menu bar select **Job→Create**. The **Create Job** dialogue box will open.

(a) Name the job **Cantilever_1D**.

(b) Click **Continue**.

2. The **Edit Job** dialogue box will open:

(a) In the **Description Field**, type **1D Cantilever Beam Bending**.

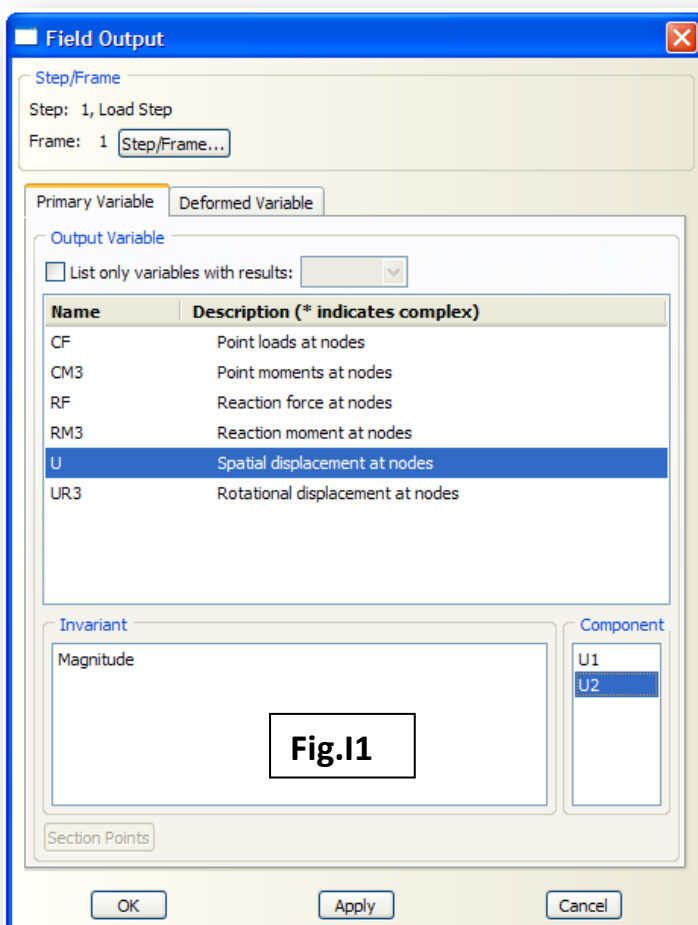
(b) Click **OK**.

3. From the main menu bar, select **Job→Submit** and choose the `Cantilever_1D` job.
4. You can monitor the job progress by selecting **Job →Monitor** from the main menu bar. When the job is complete, you can view the results in the visualisation module.

I. MODULE→VISUALISATION

In this module you can view the results of your analysis, output xy data, operate on data and export images and movies.

1. From the main menu bar select **File→Open→Cantilever_1D.odb** (C:/Temp directory).
2. From the main menu bar select **Results→Field Output**. The **Field Output** dialogue box will appear as shown in **Fig.I1**.
 - (a) From the **Output Variables** list select **U** (displacement)
 - (b) Under **Component**, select **U2** (vertical displacements)
 - (c) Click **OK**.
3. From the main menu bar select **Plot→Contours→Deformed Shape**.



The deformed cantilever beam will now be displayed and coloured according to the contours of displacement. From the contours, you should be able to see that at the free end, the beam has deflected by 11.7 mm (**Fig.I3**).

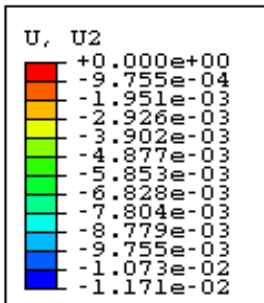


Fig.I3



Our interest now is in the full deflected profile. We'd like to compare the FEM predicted deflections with those from ordinary beam bending theory. To obtain the full FEM predicted profile:

4. From the main menu bar select **Plot→Contours→On Undeformed Shape**.

5. From the main menu bar select **Tools→Path→Create**. The **Create Path** Dialogue box will open (**Fig.I4**).

- Name the path, Cantilever Path.
- Select **Edge List** as the **Type**.
- Click **Continue**.

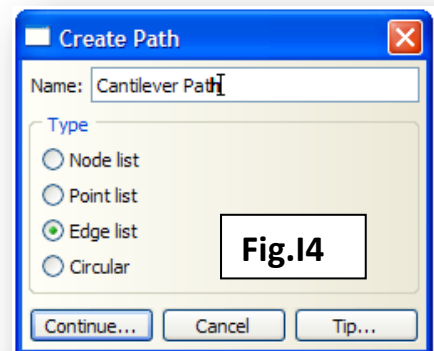


Fig.I4

The **Edit Edge List Path** dialogue box will open (**Fig.I5**)

- Select **Add Before** from the **Viewport Selection** options.
- In the prompt area, choose **Feature Edge** (**Fig.I6**)
- Using the mouse cursor, select the first element of the beam in the **Viewer**.

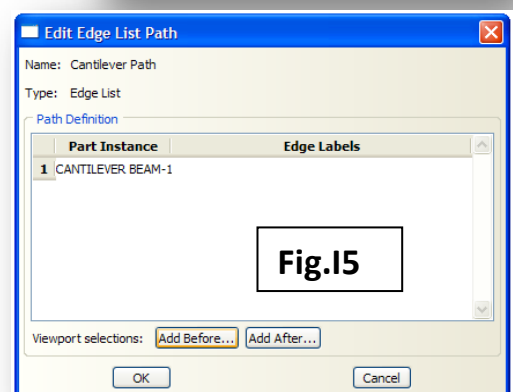


Fig.I5

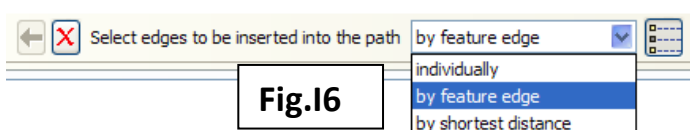


Fig.I6

(d) Click **Done** in the prompt area and **OK** in the **Edit Edge List Path** dialogue box.

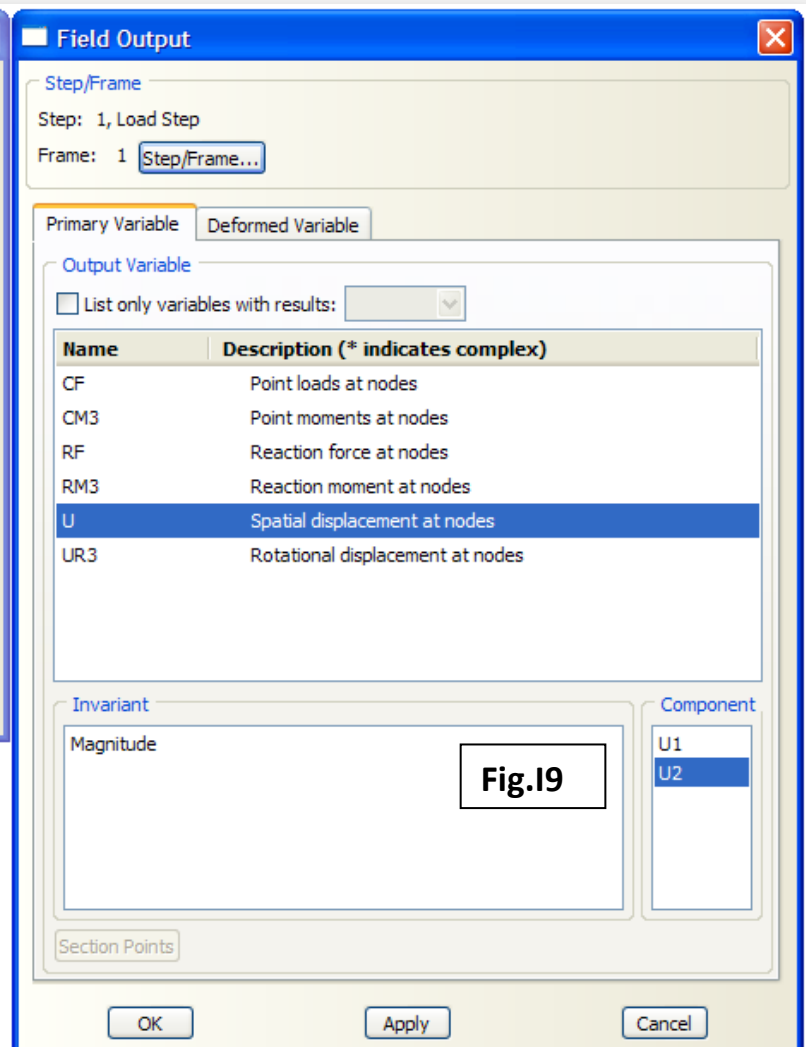
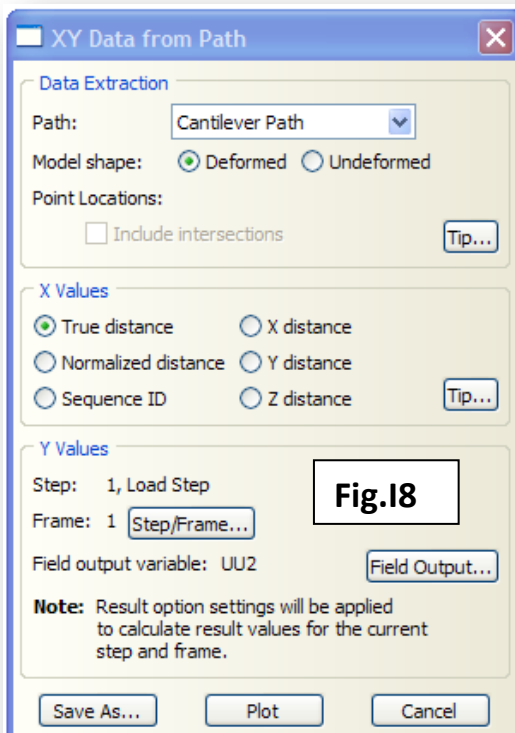
The nodes that form the path will be identified as shown in **Fig.I7**.

Fig.I7

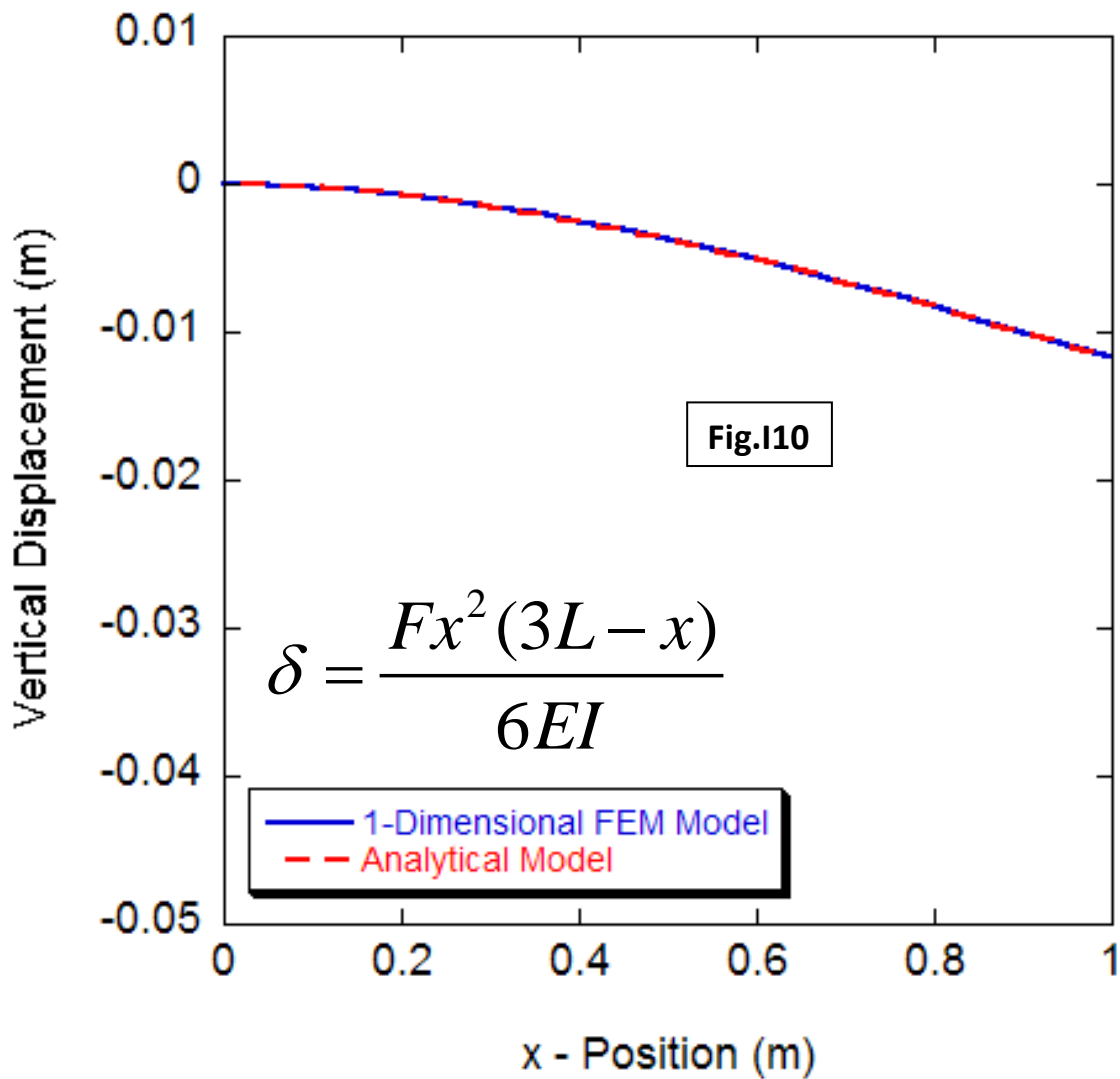


6. From the main menu bar select **Tools→XY Data→Create**. The **Create XY Data** dialogue box will open. Select **Path** from the list of options and click **Continue**. The **XY Data from Path** dialogue box will open (**Fig.I8**).

- Select **Cantilever Path** from the **Path** options
- Select **Deformed** from the **Model Plot** options.
- Select **True Distance** as shown.
- Clicking the **Field Output** icon will bring up the **Field Output** dialogue box (**Fig.I9**).
- Select **U** as the primary variable and **U2** as the **Component**. Click **OK**.
- To view the path plot, click **Plot**.
- Save the data by clicking **Save As...**, and name the data **AI_Deflection_Elastic_1D**.



7. To access the data so that you can compare the predictions to those of ordinary beam bending theory, select **Tools**→**XY Data**→**Edit**→**AI_Deflection_Elastic_1D**. Copy and paste the data into Excel. The results should look as shown in **Fig.I10**.




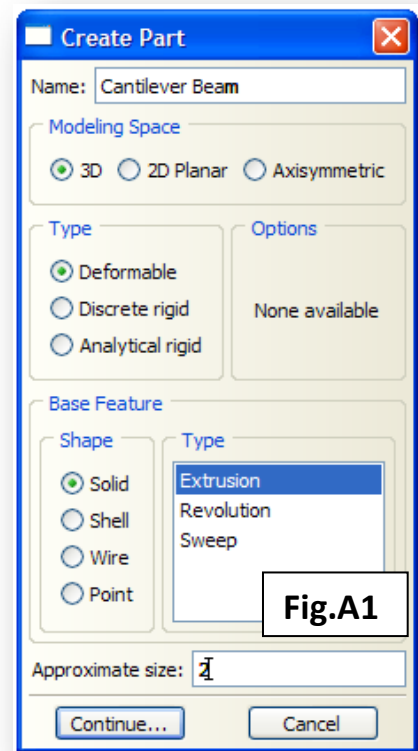
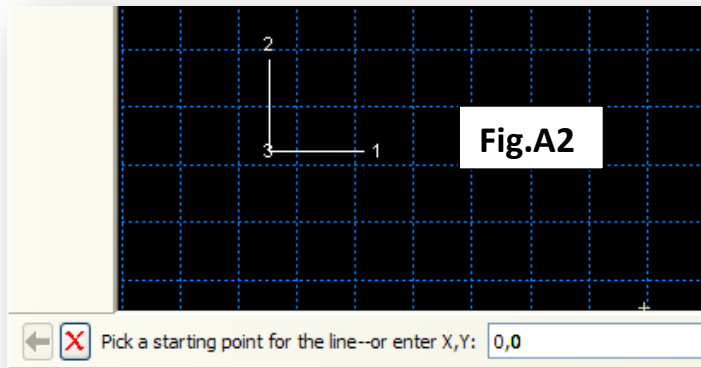
Solution (b):

- Start ABAQUS/CAE. At the **Start Session** dialogue box, click **Create Model Database**.
- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialogue box appears; name the model `Cantilever_3D`.

A. MODULE→PART

Under the Part module, we will construct the beam (3-D)

1. From the main menu bar, select **Part→Create**
2. The **Create Part** Dialogue box appears. Name the part `Cantilever Beam` and fill in the options as shown in **Fig.A1**. Click Continue to create the part.
3. From the main menu bar, select **Add→Line→Rectangle**
 - (a) Select the co-ordinates $(0, 0)$ for the first vertex (**enter**) as shown in **Fig.A2**.
 - (b) Select the co-ordinates $(1, 0.025)$ for the second vertex (**enter**).
 - (c) Click  in the prompt area.
 - (d) Click **Done** in the prompt area.



4. The **Edit Base Extrusion** dialogue box will open. In the **Depth Field**, type **0.025** and click **OK**. The part, `Cantilever Beam` will now appear in the **Viewer** window, as a 3-dimensional beam. The following tasks must be completed:

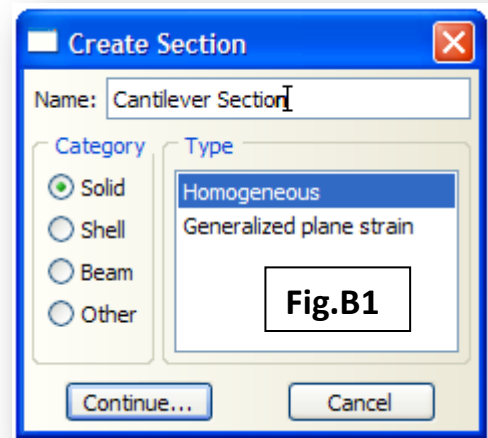
- The beam material properties must be defined.
- The boundary conditions (constraints and loads) must be defined.
- A mesh must be assigned.

B. MODULE→PROPERTY

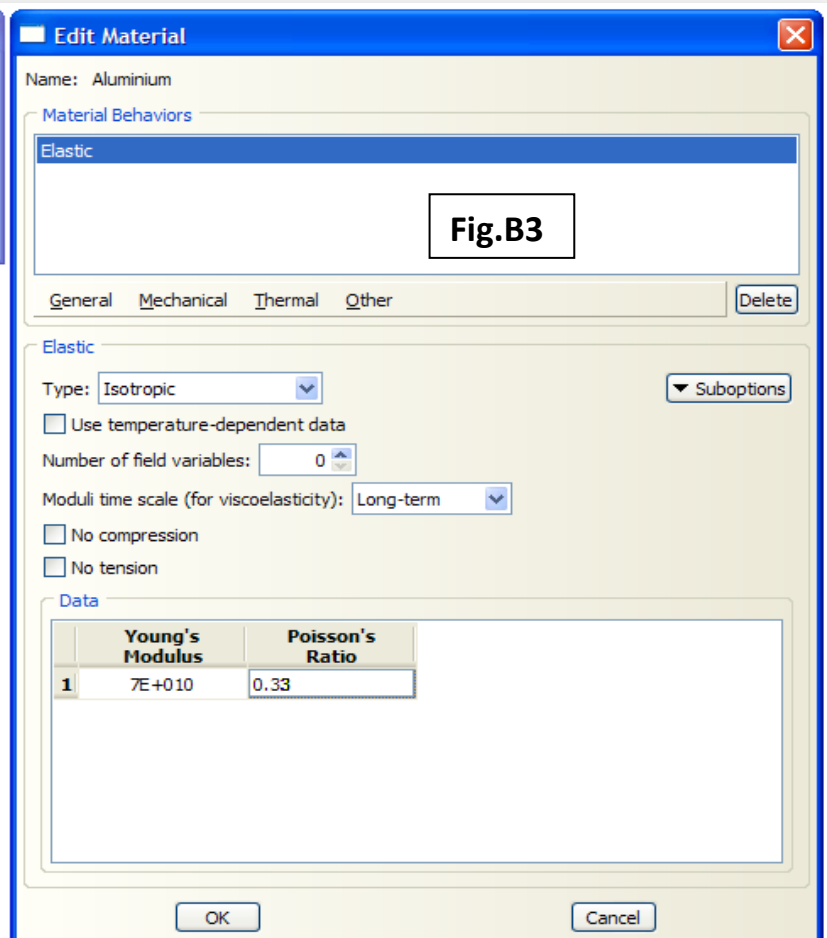
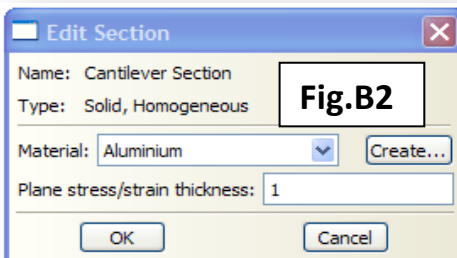
In this module (property), you will define the beam material properties (E and ν) and you will assign these material properties to the beam.

1. From the main menu bar, select **Section→Create**. The **Create Section Dialogue** box will open as shown in **Fig.B1**. Name the section Cantilever Section.

- (a) Under **Category**, choose **Solid**.
- (b) Under **Type**, choose **Homogenous**.
- (c) Click **Continue**.



2. The **Edit Section** dialogue box will open (**Fig.B2**). Click **Create** and the **Create Material** dialogue box will open (**Fig.B3**). Name the material Aluminium. Enter a Young's Modulus of $E = 70 \text{ GPa}$, and a Poisson's ratio of $\nu = 0.33$ and click **OK**.



You now need to assign the `Cantilever` Section and the `Cantilever` Material to the 3-D beam part that you have created.

3. From the main menu bar select **Assign→Section**. Using the mouse cursor, select the part in the **Viewer** and click **Done**. The **Edit Section Assignment** dialogue box will open. Check that the **Section** that is chosen is the `Cantilever` Section that you created (it should be there by default!). Click **OK**. The part will change colour, which is an acknowledgement that that section has been assigned to the material.

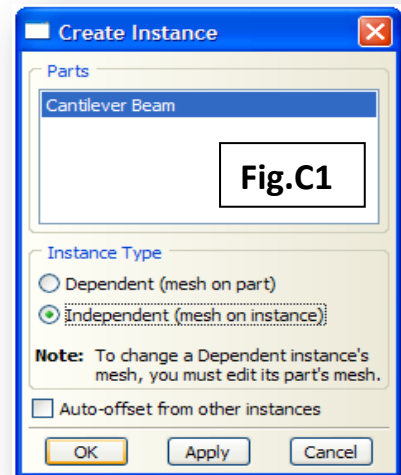
C. MODULE→ASSEMBLY

In the assembly module, multiple parts can be ‘assembled’ into an assembly of parts. This is done by creating ‘instances’ of each part. In this case, we have only one part (`Cantilever Beam`). ABAQUS still requires, however, that an instance of this part is created. (FYI multiple instances of a single part can be created if required.)

1. From the main menu bar select **Instance→Create**. The **Create Instance** dialogue box will open (**Fig.C1**).

(a) Under Instance Type, select **Independent**.

(b) Click **OK**.



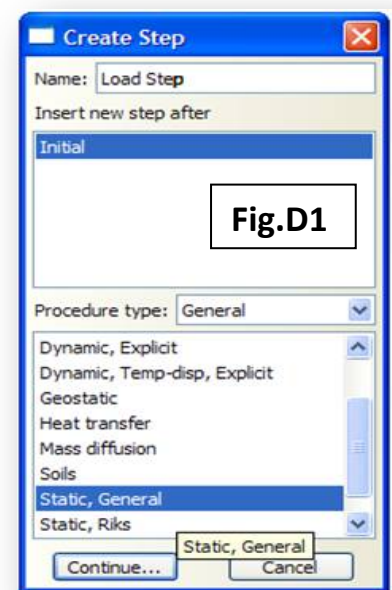
D. MODULE→STEP

In the step module, you will define the type of analysis that is to be undertaken (static in this case).

1. From the main menu bar, select **Step→Create**. The **Create Step** dialogue box will appear (**Fig.D1**). Name the step `Load Step`.

2. Select **General** from the **Procedure Type** options.

3. Select **Static, General** from the list of analysis types. Click **Continue**.



E. MODULE→INTERACTION

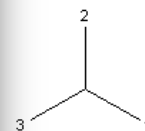
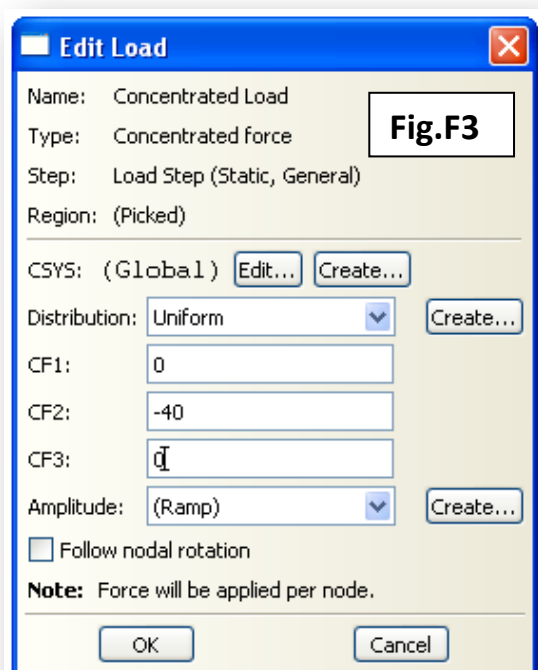
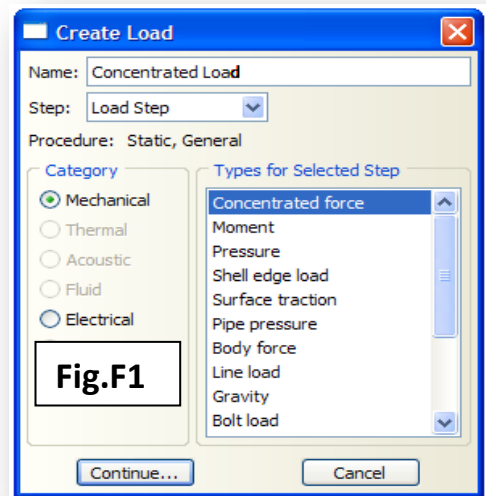
There are no interactions in this analysis.

F. MODULE→LOAD

In the load module you define the boundary conditions (constraints and loads). You will constrain one end of the cantilever beam to be fixed (zero displacements) and you will define an 80 N load at the free end of the beam.

1. From the main menu bar, select **Load→Create**. The **Create Load** Dialogue box will open (**Fig.F1**).

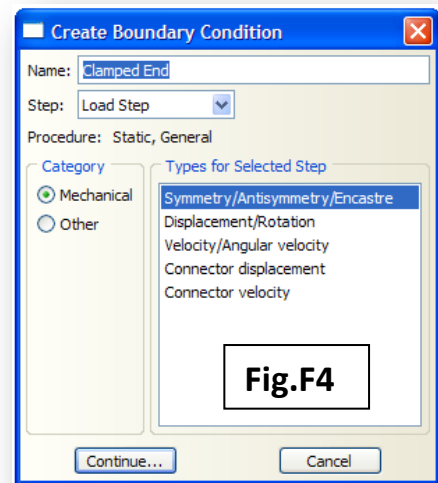
- Name the load **Concentrated Load**.
- Choose **Load Step** as the **Step** option.
- Choose **Mechanical** for the **Category**.
- Choose **Concentrated Force** for the **Type**.
- Click **Continue**.
- Using the mouse cursor, select the two nodes as shown in **Fig.F2**.
- Click **Done** in the prompt area and the **Edit Load** dialogue box will open (**Fig.F3**).
- Type **0** for **CF1** and **CF3**. Type **-40** for **CF2** (i.e. 40 N load at each node).



- Toggle off **Follow Nodal Rotation**.
- Click **OK**.

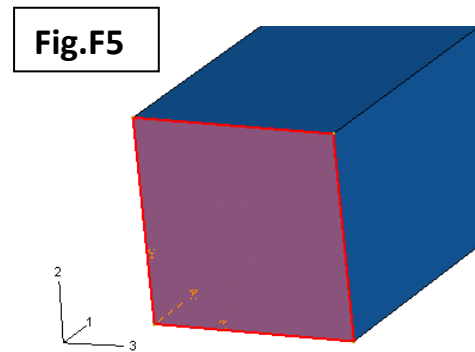
2. From the main menu bar select **BC→Create**. The **Create Boundary Condition** dialogue box will open as shown in **Fig.F4**.

- Name the boundary condition **Clamped End**.
- Choose **Load Step** for the step.
- Choose **Mechanical** for the **Category**.
- Choose **Symmetry/Antisymmetry/Encastre** for the **Types for Selected Step**.
- Click **Continue**.
- Using the mouse cursor, select the opposite end of the bar to which the loading condition was specified (**Fig.F5**). The selected face will turn purple/red.



3. The Edit Boundary Condition dialogue box will appear.

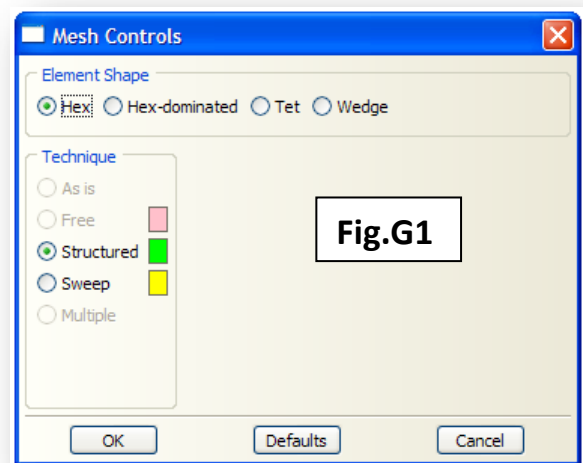
- Choose **Encastre** as the boundary condition.
- Click **OK**.



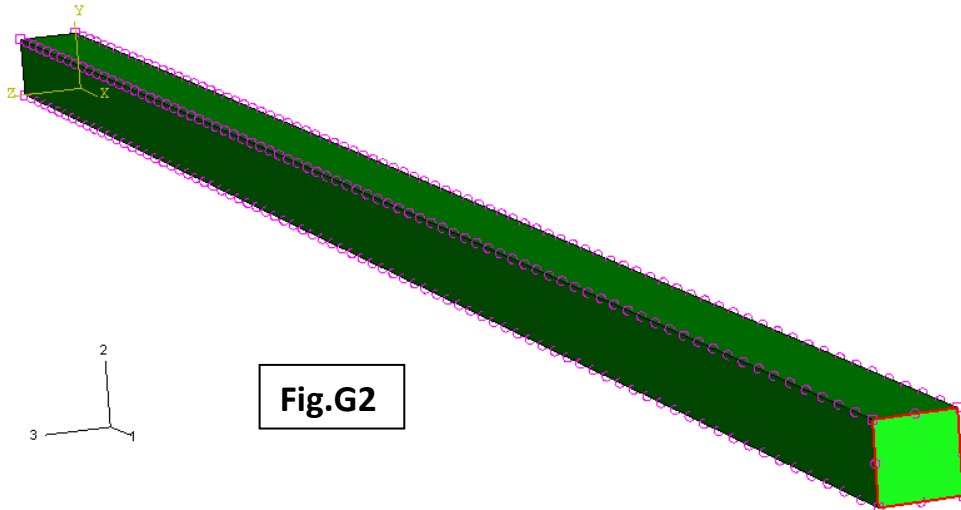
G. MODULE→MESH

1. From the main menu bar select **Mesh→Controls**. The **Mesh Controls** dialogue box will appear as shown in **Fig.G1**.

- Choose **Hex** as the **Element Shape**.
- Select **Structured** for the **Technique**.
- Click **OK**.



2. From the main menu bar select **Seed**→**Seed Edge by Number**. Using the mouse cursor (holding shift), select the end faces. Click **Done** in the prompt area. Type **2** for the number of elements along those face edges. Click **Done** in the prompt area. Repeat the operation for the side edges, choosing 80 seeds along each edge as seen in **Fig.G2**.

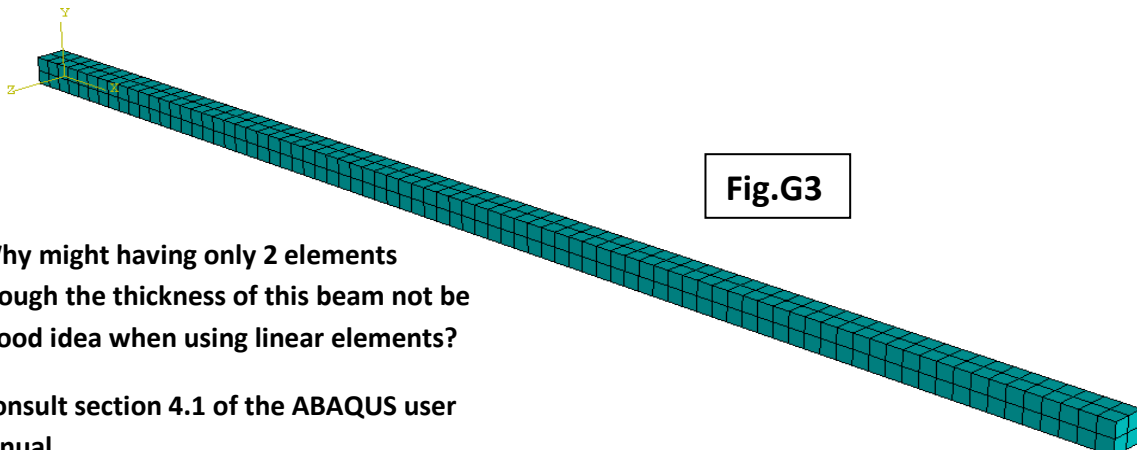


3. From the main menu bar select **Mesh**→**Element Type**. The Element Type dialogue box will appear.

- (a) Choose **Standard** from the **Element Library**.
- (b) Choose **Linear** for the **Geometric Order**.
- (c) Choose **3D Stress** for the **Family**.
- (d) Click **OK** to choose **C3D8R** elements.

4. From the main menu select **Mesh**→**Instance**.

5. Click **Yes** in the prompt area to generate 320 elements on your part instance (**Fig.G3**).



***Why might having only 2 elements through the thickness of this beam not be a good idea when using linear elements?**

***Consult section 4.1 of the ABAQUS user manual**

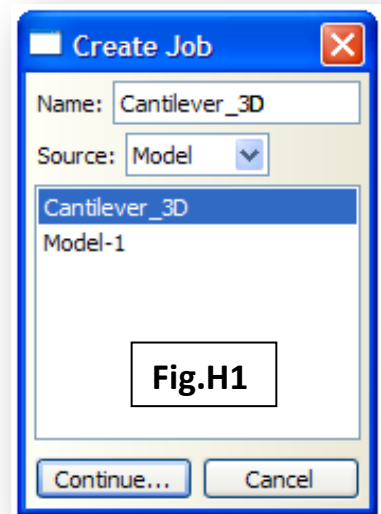
H. MODULE→JOB

1. From the main menu bar select **Job→Create**. The **Create Job** dialogue box will open as seen in **Fig.H1**.

- Name the job **Cantilever_3D**.
- Select the **Cantilever_3D Model**.
- Click **Continue**.

2. The Edit Job dialogue box will open.

- In the **Description** field, type **3D Cantilever Bending Analysis**.
- Click **OK**.



3. From the main menu bar select **Job→Submit→Cantilever_3D**. You can monitor the progress of your job by selecting **Job→Job Manager** from the main menu bar and **Monitor** from the **Job Manager** dialogue box. The analysis will take approximately 30 seconds (although this will depend on the CPU spec.). By default, the results will be saved in C:\Temp.

I. MODULE→VISUALISATION

1. From the main menu bar select **View→Toolbars→Views**. Choose plane 1-2 from the **Views** dialogue box (**Fig.I1**).



2. From the main menu bar select **Results→Field Output**. The **Field Output** options box will open. From the list of options, select **U** from the primary variables and **U2** as the component (**Fig.I2**).

- Click **OK**.
- Select **Contour** from the **Select Plot State** dialogue box and Click **OK**.
- The deformed cantilever beam will be displayed.

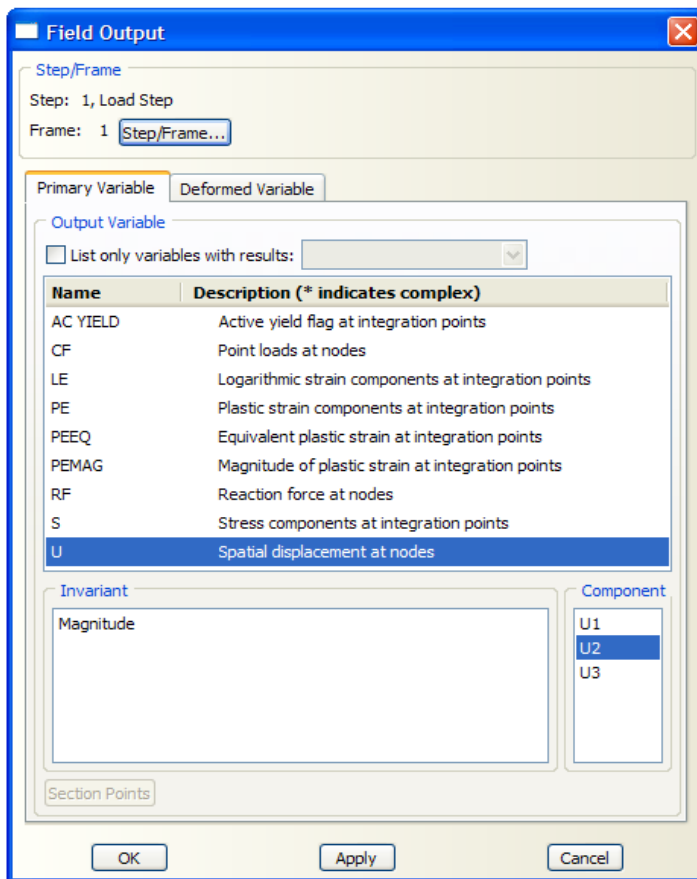
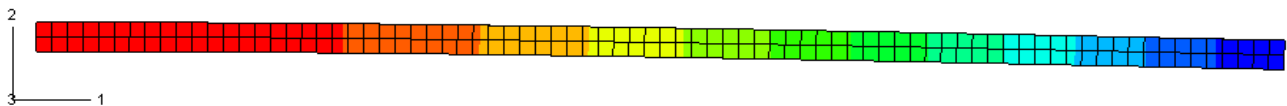
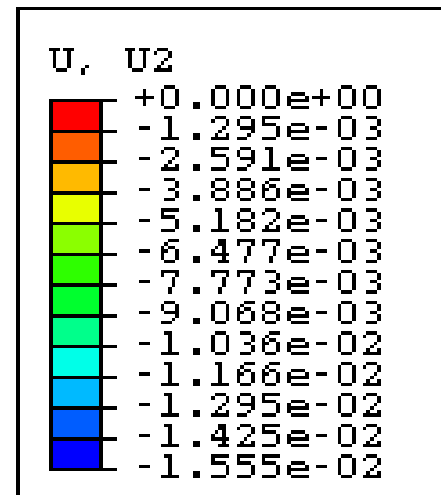


Fig.I2



Our interest now is in the full deflected profile. To obtain this information:

3. From the main menu bar select **Plot→Contours→On Undeformed Shape**.
4. From the main menu bar select **Tools→Path→Create**. The Create Path dialogue box will open.
 - (a) Choose **Edge list** as the **Type** (Fig.I3)
 - (b) Name the path **Cantilever Path**.
 - (c) Click **Continue**.
5. The Edit Edge List Path dialogue box will open.
 - (a) Select **Add After** and **OK**.
 - (b) In the prompt area select **By Feature Edge**.

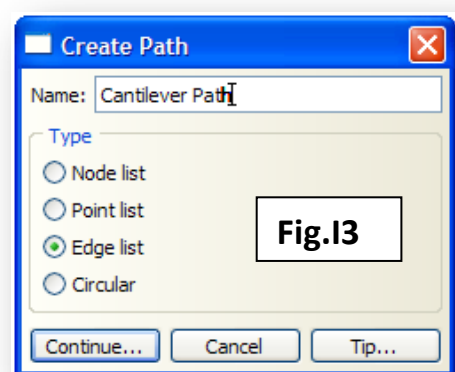
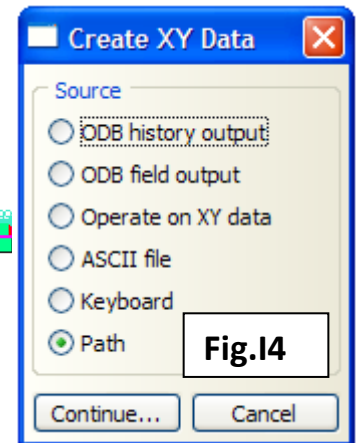
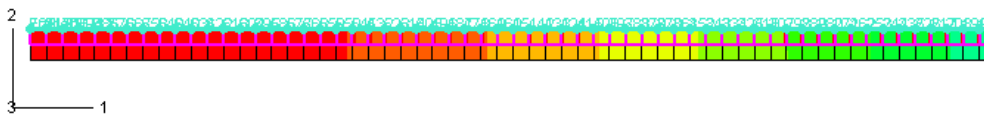


Fig.I3

6. Using the mouse cursor, select the first element edge.

(a) Click **Flip** and then Click **Done** in the prompt area.

7. From the main menu bar select **Tools→XY Data→Create**. The **Create XY Data** dialogue box will open. Select **Path (Fig.I4)** and click **Continue**.



8. The **XY Data from Path** dialogue box will open (**Fig.I5**).

(a) Select **Cantilever Path** from the **Path** options.

(b) Select **Deformed** from the **Model Shape** options.

(c) Select **True Distance** from the **X Values** option.

9. Select the **Field Output** icon and the **Field Output** dialogue box will open (**Fig.I6**).

(a) Select **U** as the primary variable and **U2** as the component.

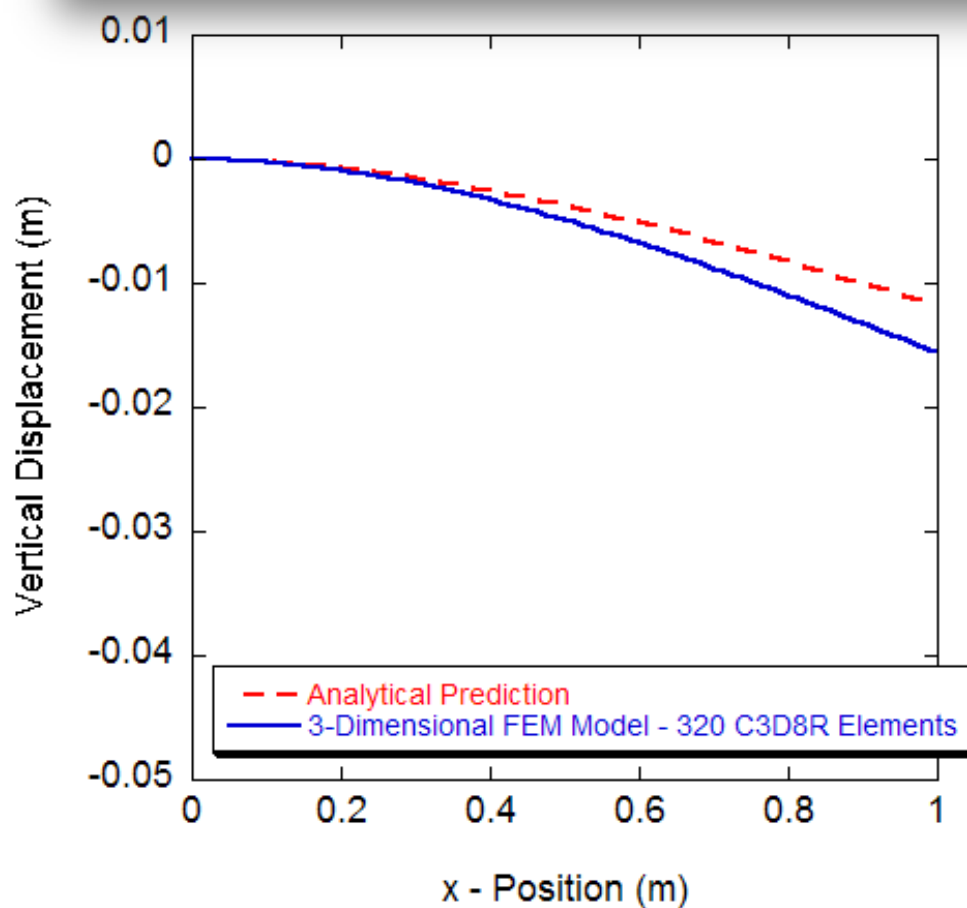
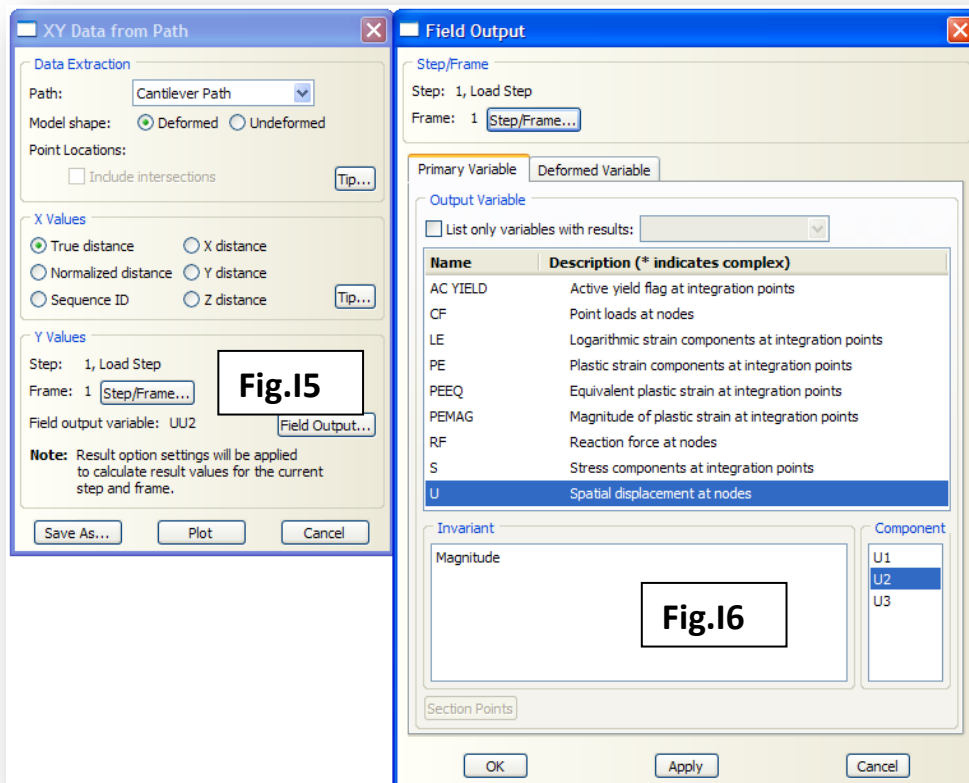
(b) Click **OK** and then click **Plot** in the **XY Data from Path** dialogue box.

(c) Click **Save As...** and name the file **A1_Deflection_Elastic_3D**. Click **OK**.

10. To access the data, from the main menu bar select :

XY Data→Edit→A1_Deflection_Elastic_3D.

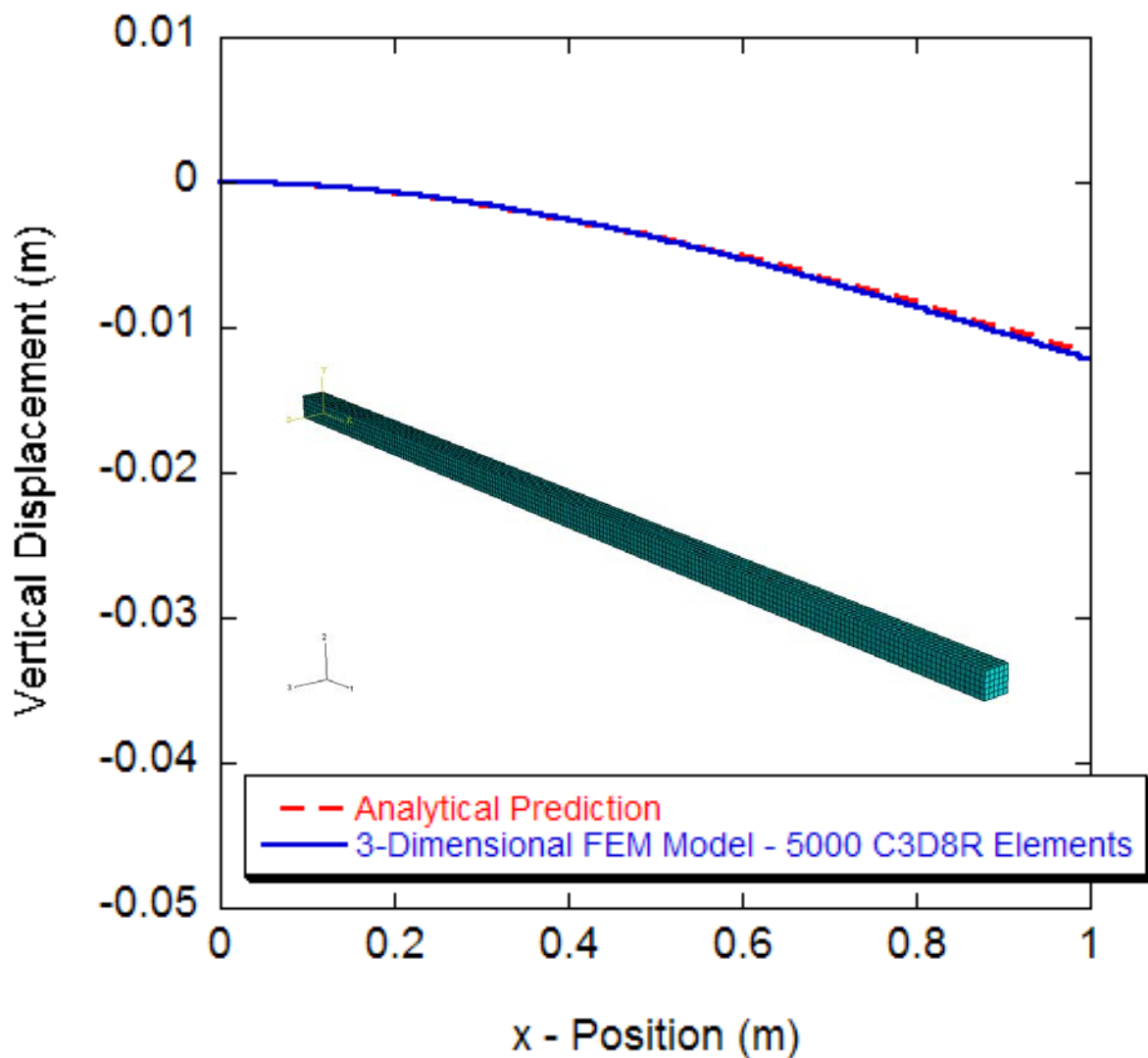
Compare these predictions with those from ordinary beam bending theory.



11. Note that in this plot, the FEM predicted deflection at the end of the bar is 15.5 mm. Ordinary beam bending theory predicts the deflection to be 11.7 mm. There is, therefore an apparent error of 24.5%. Why might this be?

Solution (c):

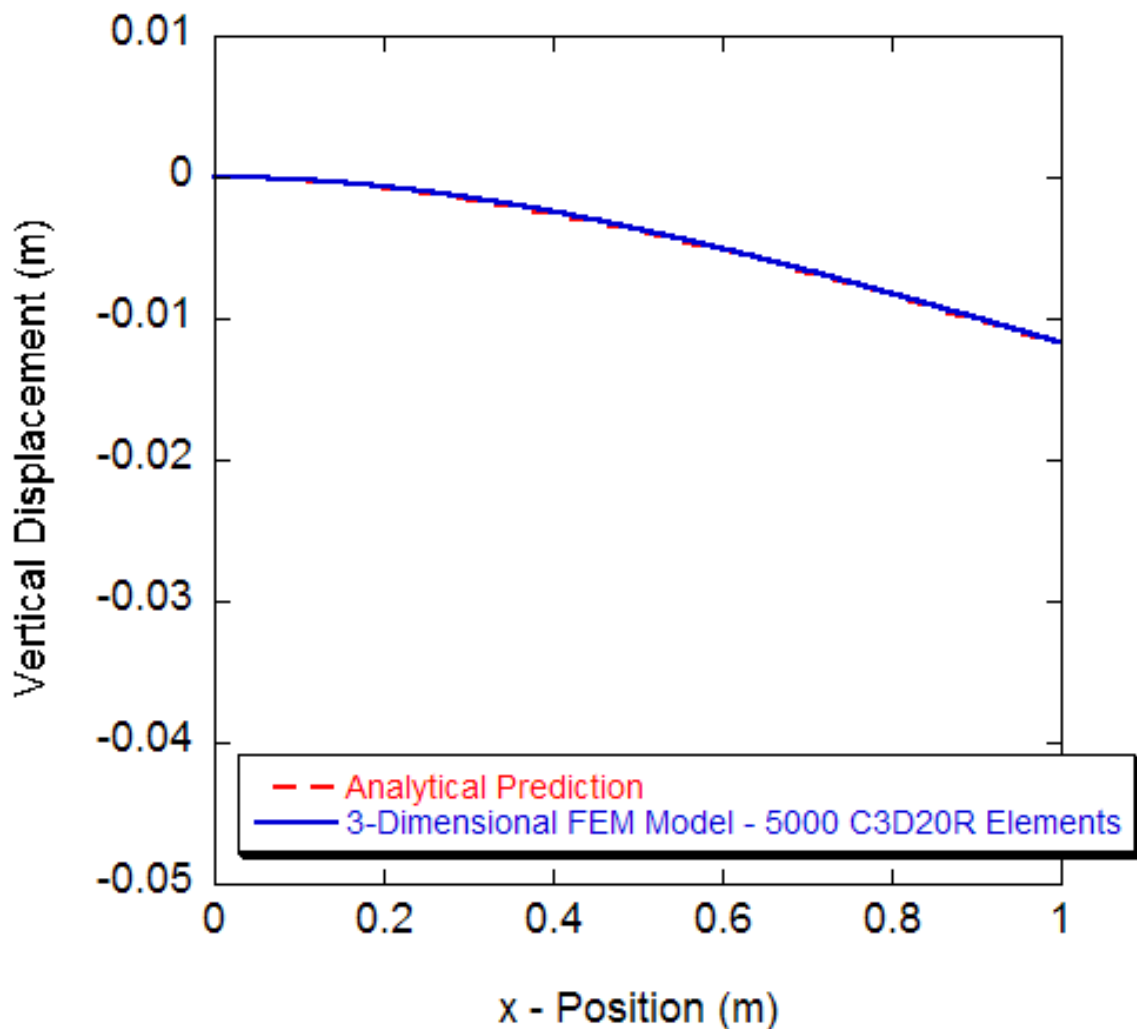
- Return to the **Mesh** module and re-seed the 3-D beam (200 seeds along the long edges and 5 seeds along the short edges).
- Re-mesh the beam; **Mesh**→**Instance**.
- Re-run the analysis; **Job**→**Job Manager**→**Submit**.
- Obtain the deflected profile; **Tools**→**XY Data**→**Create**. The **Create XY Data** dialogue box will open. Select **Path**.....
- Compare the FEM predictions with those from beam bending theory.



It is clear to see from these two analyses that the mesh density is important. However, there still remains a small discrepancy between the FEM predicted deflection at the end of the cantilever beam and that predicted by ordinary beam bending theory (7.8%).

Solution (d):

- Return to the **Mesh** module and change the element type; **Mesh**→**Element Type**. Select **Quadratic** from the **Geometric Order**. Choose **C3D20R** elements.
- Re-mesh the beam; **Mesh**→**Instance**.
- Re-run the analysis; **Job**→**Job Manager**→**Submit**.
- Obtain the deflected profile; **Tools**→**XY Data**→**Create**. The **Create XY Data** dialogue box will open. Select **Path**.....
- Compare the FEM predictions with those from beam bending theory.



The error now falls to 0.17%.

Solution (e)

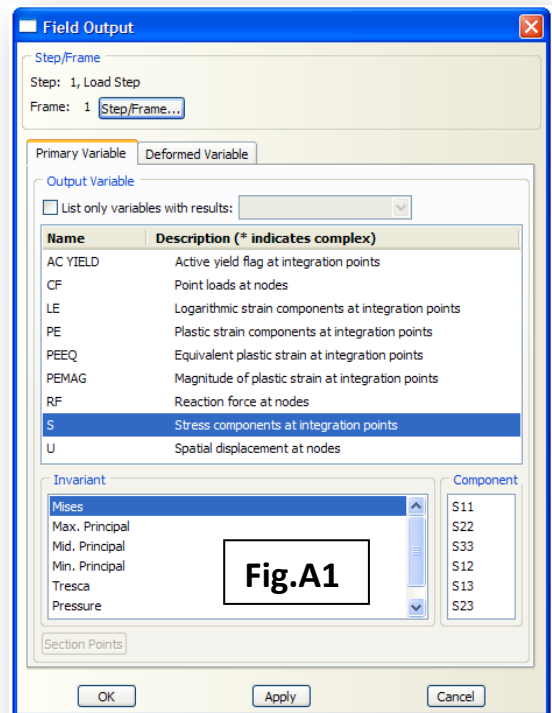
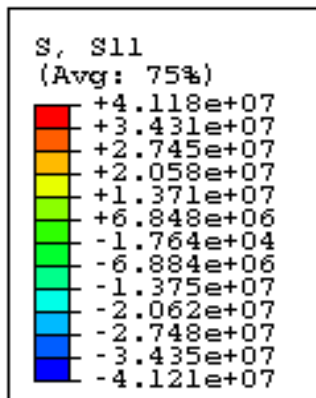
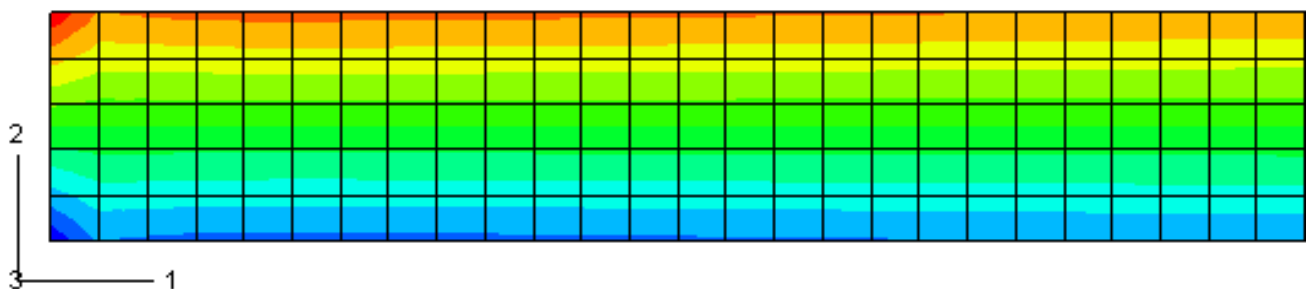
- Return to the **Mesh** module and change the element type; **Mesh**→**Element Type**. Select **Quadratic** from the **Geometric Order**. Choose **C3D20R** elements.

A. MODULE→VISUALISATION

1. From the main menu bar select **Results**→**Field Output**. The **Field Output** dialogue box will open (**Fig.A1**).

- Select **S** as the primary variable.
- Select **S11** as the stress component (axial stress)
- Click **OK**.

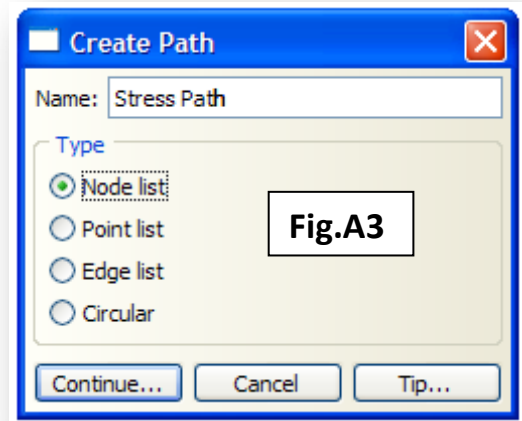
2. The stress distribution will appear as shown in Fig.A2. There is a tensile state of axial stress at the top of the beam and a compressive axial stress state at the bottom. The peak tensile and compressive stresses are ~41 MPa .

**Fig.A1****Fig.A2**

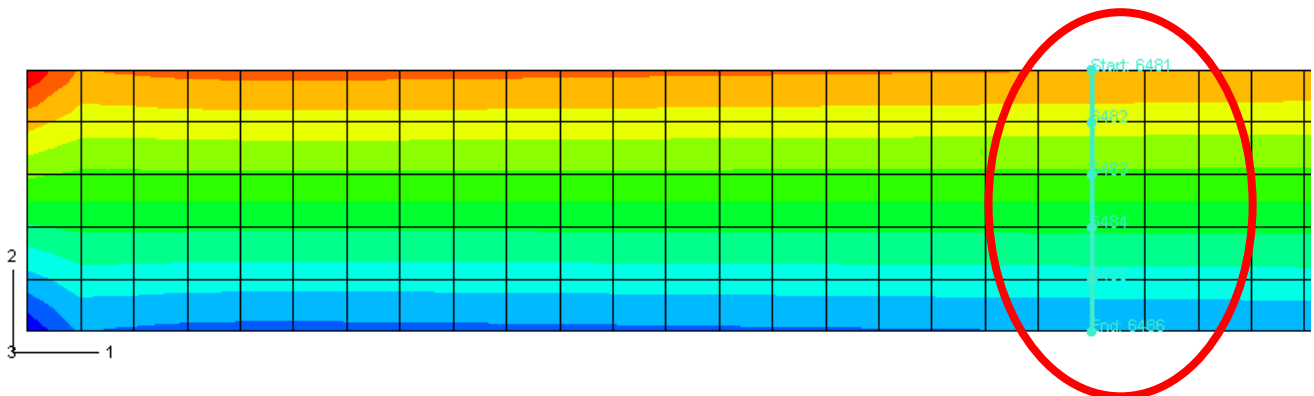
3. To plot the distribution of stress through the beam at $x = 0.1$, complete the following set of tasks:

4. From the main menu bar select **Tools→Path→Create**. The **Create Path** dialogue box will open as in **Fig.A3**.

- (a) Name the path `Stress Path`.
- (b) Choose **Node List** for the **Type**.
- (c) Click **Continue**.
- (d) From the **Edit Path List** dialogue box that opens select **Add Before**.



5. The length of the beam is 1 m. Since the beam has been meshed with 200 elements along the edge, the position $x = 0.1$, is equal to 20 elements along the beam. Using the cursor, select the nodes across the beam at $x = 0.1$ as shown. Click **OK** in the **Edit Path List** dialogue box.

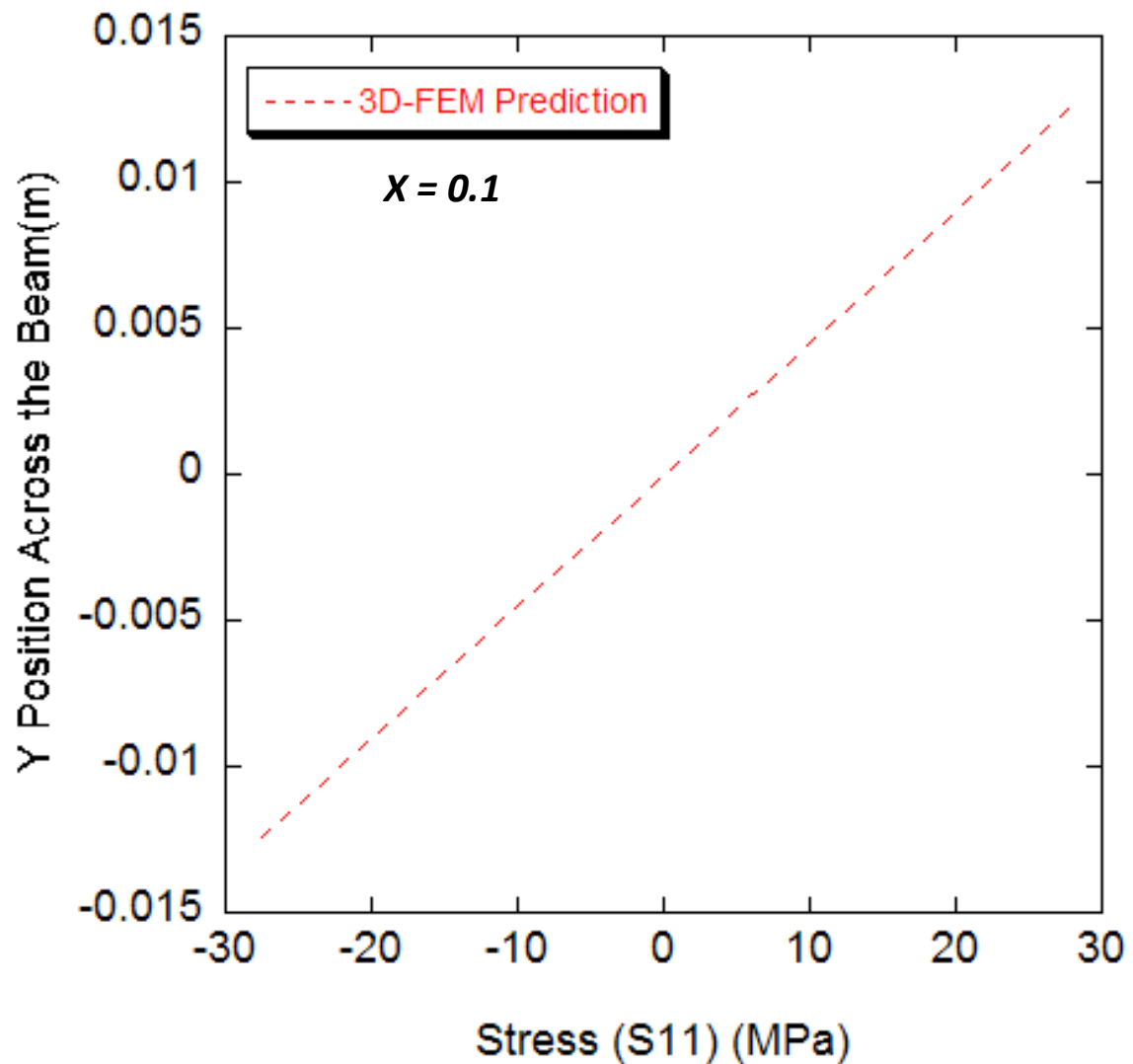


6. From the main menu bar select **Tools→XY Data→Create**. The **Create XY Data** dialogue box will open.

- (a) Select **Path** and click **Continue**.
- (b) Choose `Stress Path` as the path from the **XY Data from Path** dialogue box.
- (c) Choose **Deformed** for the **Model Shape**.
- (d) Under **X Values**. Toggle on **Y Distance**.
- (e) Click the **Field Output** icon and the **Field Output** dialogue box will open.
- (f) Select **S** as the primary variable and **S11** from the **Component** options.

(g) Click Plot.

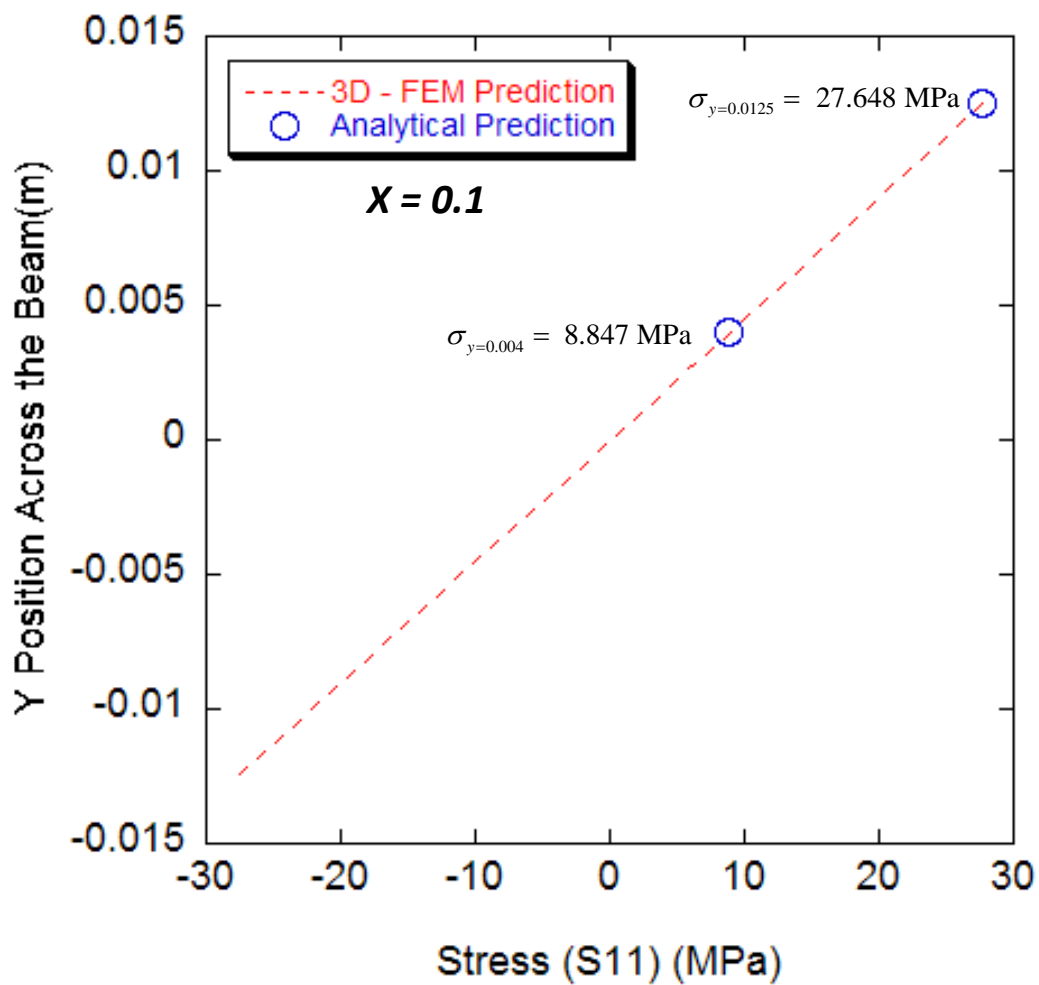
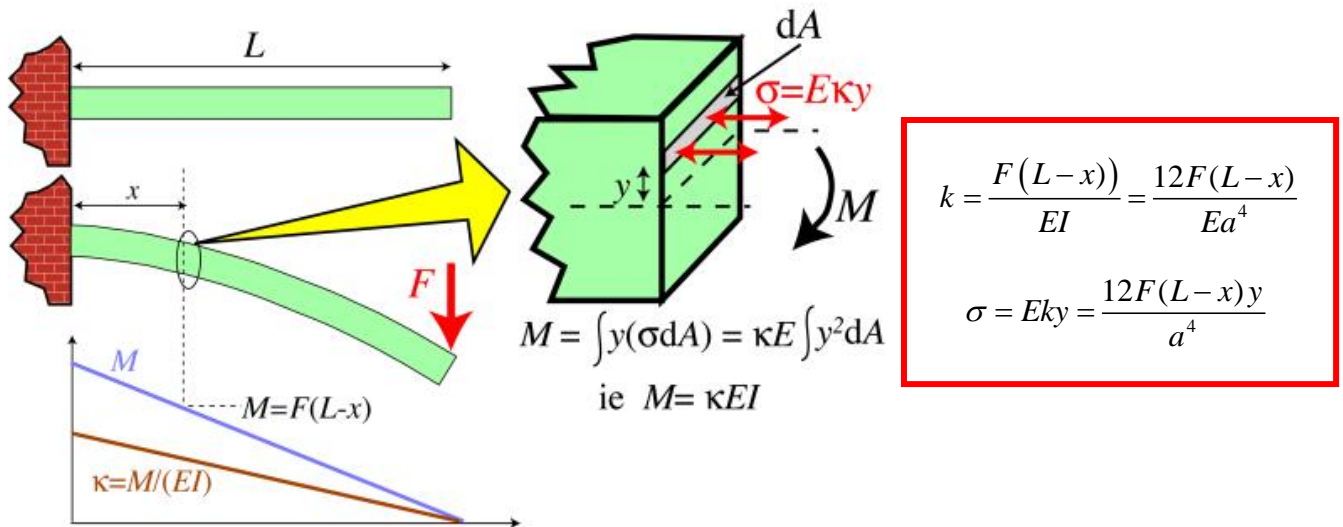
(h) Save the data. To access the data, from the main menu bar select **Tools**→**XY Data**→**Edit**.



Is there an easier way to obtain these data? If you have time, think about creating a node set at $x = 0.1$.

Solution (f)

- Compare these stress data with those from ordinary beam bending theory at $x = 0.1$ for $y = 0.004$ and $y = 0.0125$.



OPTIONAL TASKS

1. Assume now that the material behaves in an elastic – perfectly plastic fashion, and that the yield stress is 150 MPa. Using the 3-dimensional finite element model, compute the deflection of the cantilever beam when loaded at its end with a force of 600 N. (*Hint; you will have to define this plastic material behaviour in your material model.)
2. Using the 3-dimensional finite element model, and assuming elastic - perfectly plastic behaviour, plot the **residual curvature** of the cantilever beam after it has been loaded at its end with a force of 600 N. Compare the deflected profile from Question 1 with this residual curvature. (*Hint; Think about creating a second step in which the load from the Load Step is not **Propagated!**)
3. Assume now that the aluminium exhibits linear work hardening behaviour, with a work hardening rate $d\sigma/d\varepsilon$ of 300 MPa. Compute the deflection of the beam when loaded at its end with a force of 600 N. Compute also the deflection of the beam when the load is removed. Compare the results with those of the perfectly plastic case.

(*Hint; $\sigma_Y = 150$ MPa at $\varepsilon_{\text{plastic}} = 0$ and $\sigma_Y = 450$ MPa at $\varepsilon_{\text{plastic}} = 1$)
4. Assume now that the aluminium hardens according to a power-law relationship of the following kind: $\sigma = \sigma_y + k\varepsilon_{\text{plastic}}^n$, where the yield stress is 150 MPa, k is a constant equal to 300 MPa and the hardening exponent, n , is 0.4. Calculate σ for $0 < \varepsilon_{\text{plastic}} < 1$. Using this data, compute the deflection of the cantilever beam, loaded at its end with a force of 600 N. Find also the residual curvature of the beam. Compare these data with the linear work-hardening case.