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 v = 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:

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

The deflection of the beam is given by:

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

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

$$I = \frac{a^4}{12} \tag{3}$$

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



Create Part		
Name: Cantilever Beam		
Modeling Space		
🔘 3D 💿 2D Planar	O Axisymmetric	
Туре	Options	
💿 Deformable		
🔘 Discrete rigid	Mono ausilable	
Analytical rigid	None available	
🔿 Eulerian		
Base Feature		
🔘 Shell		
💽 Wire		
O Point	- 11	
FIE	g.AI	
Approximate size: 2		
Continue	Cancel	

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.

Create Section

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.

3. In the Edit Beam Section dialogue box, next to Material name, click Create, and the Edit Material Dialogue box will open (Fig.B5). Name the material Aluminium and then Click Mechanical \rightarrow Elasticity \rightarrow Elastic. Specify the elastic material properties, *E* and *v* and click OK. Specify the section Poisson's ratio to be 0.33 in the Edit Beam Section dialogue box. Click OK.

Edit Beam Section	Create Drofile	Edit Profile
Name: Cantilever Section		Name: Cantilever Profile
Type: Beam	Name: Cantilever Profile	Shape: Rectangular
Section integration: 💿 During analysis 🔘 Before analysis		≜2 a: 0.025
Profile name: Cantilever Profile 🔽 Create	Shape	b: 0.025
Profile shape: Rectangular	Box	h
Basic Stiffness Fluid Inertia	Pipe	
Material name: Aluminium 💽 Create	Circular	Fig.B4
Section Poisson's ratio: 0.33	Destasedan	
Temperature variation:	Rectangular	OK Cancel
O Linear by gradients	Hexagonal	
 Interpolated from temperature points 	Trapezoidal	Edit Material
	I	Name: Aluminium
		Description:
Fig B2	Fig.B3	Material Behaviors
118.02		Elastic
	Arbitrary	
	Generalized	Conseral Machanical Thermal Other
		Celestic
	Continue Cancel	Type: Isotropic 💌 🔍
		Use temperature-dependent data
		Number of hield variables: 0
		No compression
OK Cancel		Data
		Young's Poisson's Fig.B5
		1 70e9 0.33

Module 1

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).

Edit S	ection Assignn	nent 🛛 🛛
Section		
Section:	Cantilever Section	Create
Note: Li a	ist contains only se pplicable to the sel	ctions ected regions.
Type:	Beam	
Material:	(None)	
Region		Fig.B6
Region:	(Picked)	
- Shell Of	fset	
Offse	t:	
C	ОК	Cancel

5. From the main menu bar select $Assign \rightarrow Assign Beam Section Orientation$. 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 \rightarrow Create. The Create Instance dialogue box will open (Fig.C1).

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

🗖 Create Instance 🛛 🛛 🔀		
Parts		
Cantilever Beam		
Fig.C1		
Instance Type		
 Dependent (mesh on part) Independent (mesh on instance) 		
Note: To change a Dependent instance's mesh, you must edit its part's mesh.		
Auto-offset from other instances		
OK Apply Cancel		

D. MODULE \rightarrow 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 \rightarrow 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 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.

🔲 Edit Lo	ad	
Name: Cor	ncentrated Load	
Type: Co	ncentrated force	
Step: Loa	d Step (Static, General)
Region: (Pic	ked) Edit Region	Fig.F2
CSYS: (G1	obal) Edit Cre	ate
Distribution:	Uniform	Create
CF1:	0	
CF2:	-80	
Amplitude:	(Ramp)	Create
E Follow no	dal rotation	
Note: Force will be applied per node.		
	ж	Cancel

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**.

- (a) Name the boundary condition, Clamped End.
- (b) Choose Load Step, for the Step type.
- (c) Choose Mechanical for the Category.
- (d) Choose **Symmetry/Antisymmetry/Encastre** for the **Types for Selected Step option**.
- (e) Click **Continue**.

(f) Use the mouse cursor to select the first vertex (node) which is going to be clamped.

(g) Click **Done** in the prompt area.

Create Bour	ndary Condition	
Name: Clamped E	ind	
Step: Load Step	×	
Procedure: Static	, General	
Category	Types for Selected Step	
 Mechanical Other 	Symmetry/Antisymmetry/Encastre Displacement/Rotation Velocity/Angular velocity Connector displacement Connector velocity	
	Fig.F3	
Continue	Cancel	
Continue		

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

- (a) Select Encastre as the boundary condition.
- (b) 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.

1. From the main menu bar, select **Seed** \rightarrow **Edges**.

2. 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** \rightarrow **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.

Liement Library	Family		
Standard O Explicit	Acoustic		^
Geometric Order	Coupled Temperature-Displacement		
🖲 Linear 🔘 Quadratic	Gasket		×
line			
Element Controls			
Hybrid formulation			
Beam type: () Shear-fi	exible 🔘 Cubic formulation	F '- C 4	
Linear bulk viscosity scal	ng factor: 1	Fig.G1	
Quadratic bulk viscosity	caling factor: 1	–	
ote: To select an element select 'Mesh->Cont	shape for meshing, visif from the main menu bar.		

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 \rightarrow 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** \rightarrow **Monitor** from the main menu bar. When the job is complete, you can view the results in the visualisation module.

I. MODULE \rightarrow 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**.

Step/Frame		
Step: 1, Load Step		
Frame: 1 Sten/F	rame	
(<i>p</i> /-		
Primary Variable	Deformed Variable	
C Output Variable		
List only varia	ables with results:	
Name	Description (* indicates complex)	
CF	Point loads at nodes	
CM3	Point moments at nodes	
RF	Reaction force at nodes	
RM3	Reaction moment at nodes	
U	Spatial displacement at nodes	
UR3	Rotational displacement at nodes	
Invariant Magnitude	Fig.I1	Component U1 U2
ОК	Apply	Cancel

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**).





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** \rightarrow **Path** \rightarrow **Create**. The **Create Path** Dialogue box will open (Fig.I4).

- (a) Name the path, Cantilever Path.
- (b) Select Edge List as the Type.
- (c) Click Continue.

- 1

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

(a) Select **Add Before** from the **Viewport Selection** options.

(b) In the prompt area, choose **Feature Edge** (**Fig.I6**)

(c) Using the mouse cursor, select the first element of the beam in the **Viewer**.

Elect edges to be i	inserted into the path	by feature edge 🛛 😽	B
		individually	
	Fig.16	by feature edge	
		by shortest distance	

Fornt list Edge list Circular Fig.14
Continue Cancel Tip
Edit Edge List Path
Name: Cantilever Path Type: Edge List Path Definition
Part Instance Edge Labels
Fig.I5
Viewport selections: Add Before Add After
OK Cancel

Create Path

Type

Node list

Name: Cantilever Path

(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

End: 21

6. From the main menu bar select **Tools** \rightarrow **XY Data** \rightarrow **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.18**).

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

🗆 XY Data from Path 🛛 🛛 🗙	Field Output
XY Data from Path X Data Extraction Path: Cantilever Path Path: Cantilever Path Model shape: Model shape: Deformed Undeformed Point Locations: Include intersections Tip X Values Y distance X distance Normalized distance Y distance Tip Y Values Z distance Tip Y Values Z distance Tip Y Values Fig.18 Fig.18 Frame: 1 Step/Frame Field Output variable: UU2 Field Output Note: Result option settings will be applied to calculate result values for the current step and frame. Save As Save As Plot Cancel	Field Output Image: Step/Frame Step/Frame Step/Frame Step: 1, Load Step Frame: 1 Step/Frame Primary Variable Deformed Variable Output Variable Image: Step/Frame Uist only variables with results: Image: Step/Frame Name Description (* indicates complex) CF Point loads at nodes CM3 Point moments at nodes RF Reaction force at nodes RM3 Reaction moment at nodes UR3 Rotational displacement at nodes UR3 Rotational displacement at nodes Invariant Component Magnitude Image: Ulticate step (Step
	Section Points OK Apply Cancel

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.

Create Section		
Name: Cantilever Section		
Category Type	1	
Solid Homogeneous		
Shell Generalized plane strain		
Beam		
Other Fig.B1		
	J	
Continue Cancel		

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 GPa, and a Poisson's ratio of v = 0.33 and click OK.

Edit Section	Edit Material
Name: Cantilever Section	Name: Aluminium
Name: Cantilever Section Type: Solid, Homogeneous Material: Aluminium Image: Create Plane stress/strain thickness: OK Cancel	Name: Aluminium Material Behaviors Elastic General Mechanical Thermal Other Elastic Type: Isotropic Value temperature-dependent data Number of field variables: O Moduli time scale (for viscoelasticity): Long-term No tension Data Young's Youss Poisson's No dullus Poisson's Notension
	1 7E+010 0.33 OK Cancel

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).

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**.





Create Instance

Parts

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 two nodes as shown in **Fig.F2**.

(g) Click **Done** in the prompt area and the **Edit Load** dialogue box will open (**Fig.F3**).

(h) Type **0** for **CF1** and **CF3**. Type **-40** for **CF2** (i.e. 40 N load at each node).

🗖 Edit Loa	ad 🛛 🔀
Name: Cor Type: Cor	ncentrated Load Incentrated force Fig.F3
Step: Loa Region: (Pic	d Step (Static, General) ked)
CSYS: (G1	obal) Edit) Create)
Distribution:	Uniform 🛛 🔽 Create
CF1:	0
CF2:	-40
CF3:	đ
Amplitude:	(Ramp) 🔽 Create
Follow no	dal rotation
Note: Force	e will be applied per node.
	K Cancel





- (i) Toggle off Follow Nodal Rotation.
- (j) Click OK.

Module 1

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

- (a) Name the boundary condition Clamped End.
- (b) Choose Load Step for the step.
- (c) Choose Mechanical for the Category.
- (d) Choose **Symmetry/Antisymmetry/Encastre** for the **Types for Selected Step**.
- (e) Click Continue.

(f) 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.
 - (a) Choose Encastre as the boundary condition.
 - (b) 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**.

- (a) Choose Hex as the Element Shape.
- (b) Select Structured for the Technique.
- (c) Click OK.



30

2. From the main menu bar select **Seed** \rightarrow **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).



H. MODULE→JOB

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

- (a) Name the job Cantilever_3D.
- (b) Select the Cantilever_3D Model.
- (c) Click Continue.
- 2. The Edit Job dialogue box will open.

(a) In the Description field, type 3D Cantilever Bending Analysis.

(b) Click OK

3.	From the main menu bar select Job-Submit-Cantilever_3D . You can monitor the
pro	gress 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
dep	pend on the CPU spec.). By default, the results will be saved in C:\Temp.

I. MODULE \rightarrow VISUALISATION

1. From the main menu bar select View -> Toolbars -> Views. Choose plane 1-2 from the Views dialogue box (Fig.l1).

Views					×	
$\downarrow x x \downarrow z \downarrow $	1	2	3	4	Å	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.l2).

- (a) Click OK.
- (b) Select Contour from the Select Plot State dialogue box and Click OK.
- (c) The deformed cantilever beam will be displayed.

Cantilever_3D	
Model-1	
Fig.H1	
Continue Cancel	
Nou can monitor the	
nu bar and Monitor from	

Create Job

Source: Model

Name: Cantilever_3D

C Step/Frame Step: 1, Load Step	Fig 12
Frame: 1 Step/Frame	.
Primary Variable Deformed Variable Output Variable	
Name Description (* indicates complex)	
AC YIELD Active yield flag at integration points	
CF Point loads at nodes	
LE Logarithmic strain components at integration points	
PE Plastic strain components at integration points	
PEEQ Equivalent plastic strain at integration points	0,02
PEMAG Magnitude of plastic strain at integration points	👝 +0.000e+00
RF Reaction force at nodes	<mark></mark> -1.295e-03
S Stress components at integration points	
U Spatial displacement at nodes	<mark>—</mark> -3.88бе-ОЗ
Component	<mark> </mark> -5.182e-03
Magnitude U1 U2 U3 U3 Section Points OK Anny Carrel	-6.477e-03 -7.773e-03 -9.068e-03 -1.036e-02 -1.166e-02 -1.295e-02 -1.425e-02 -1.555e-02

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 \rightarrow Path \rightarrow 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**.

Create Path	
Name: Cantilever Path	
С Туре	
○ Node list	
O Point list	Fig. 12
 Edge list 	Fig.13
O Circular	
Continue Cance	el Tip

- 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 Al_Deflection_Elastic_3D. Click OK.
- 10. To access the data, from the main menu bar select :

XY Data \rightarrow Edit \rightarrow Al_Deflection_Elastic_3D.

Compare these predictions with those from ordinary beam bending theory.

Module 1



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**).

- (a) Select **S** as the primary variable.
- (b) Select **S11** as the stress component (axial stress)
- (c) 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 .

Field Output		
Step/Frame		
Step: 1, Load Step	,	
Frame: 1 Step/F	rame	
Primary Variable	Deformed Variable	
 Output Variable 		
List only varia	bles with results:	
Name	Description (* indicates complex)	
AC YIELD	Active yield flag at integration points	
CF	Point loads at nodes	
LE	Logarithmic strain components at integra	ition points
PE	Plastic strain components at integration p	points
PEEQ	Equivalent plastic strain at integration po	bints
PEMAG	Magnitude of plastic strain at integration	points
RF	Reaction force at nodes	
S	Stress components at integration points	
U	Spatial displacement at nodes	
Invariant		Component
Mises		▲ \$11
Max. Principal		S22
Mid. Principal		S33
Min. Principal	Fig.A1	S12
Iresca	1.8.7.1	513
Leressure		S23
Section Points		
ОК	Apply	Cancel



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

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

(a) Name me pair scress Pach	(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**.

Create Path	\mathbf{X}
Name: Stress Path	
Type Node list Point list Edge list Circular	
Continue Cancel Tip	

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 $Tools \rightarrow XY$ Data $\rightarrow Edit$.



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; σ_{Y} = 150 MPa at $\epsilon_{plastic}$ = 0 and σ_{Y} = 450 MPa at $\epsilon_{plastic}$ = 1)

4. Assume now that the aluminium hardens according to a power-law relationship of the following kind: $\sigma = \sigma_y + k \varepsilon_{plastic}^n$, 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 $0 < \varepsilon_{plastic} < 1$. 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.