

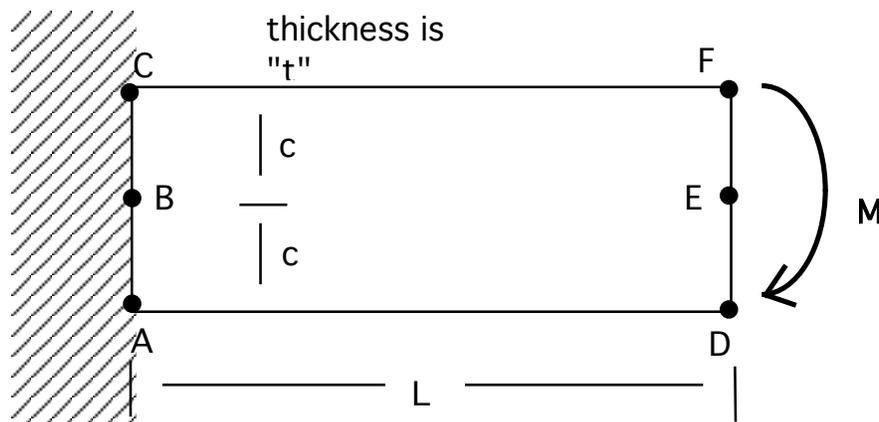
ME309**Homework #2****OBJECTIVES:**

- Introduction to convergence issues and modeling approaches
- Postprocessing procedures
- Element selection
- Experience with shape functions and stiffness terms

NOTES: Problems 1 and 2 do not require a FEM solver (i.e. ANSYS). Problems 3 and 4 do require a FEM solver.

1.) (30%) Element type and shape function.

1.1) The cantilever beam shown in Figure 1 is tip-loaded by a moment M .

**Figure 1**

Assume that Poisson's ratio is zero. Using beam theory to compute the displacement components of points D, E, and F (Eqns. 2.1.4A and 2.1.4B of Module #2) the displacement of each point should be as follows;

Table 1

	Displacement in X (U_x)	Displacement in Y (U_y)
Point D	$\frac{-3ML}{2Etc^2}$	$\frac{-3ML^2}{4Etc^3}$
Point E	0	$\frac{-3ML^2}{4Etc^3}$
Point F	$\frac{3ML}{2Etc^2}$	$\frac{-3ML^2}{4Etc^3}$

Regarding these results as nodal displacements, use them to compute stresses (in terms of M , L , c and thickness t) in elements defined as:

- (a) A CST element whose nodes are A, E, and C.
- (b) A Q4 element whose nodes are A, D, F, and C.

In setting up the solution for each of the two cases, clearly present

- your numbering scheme for the element
- the shape functions for the element. Also show that these shape functions meet the requirement of taking on a non-zero value at only 1 of the nodes.
- the displacement fields for the element based on these shape functions
- the relationship between displacement and strain
- the relationship between strain and stress

1.2) If the structure of 1A was modeled with Q4 elements (4 in the x-direction, and 6 elements in the y-direction), suggest how you would apply the bending moment M to the model. (Do not actually build the model)

2.) (10%) Determining a stiffness matrix

Consider the uniform bar element in shown below in Figure 2 with a given axial displacement field u :

$$u = [1 \quad s^2] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

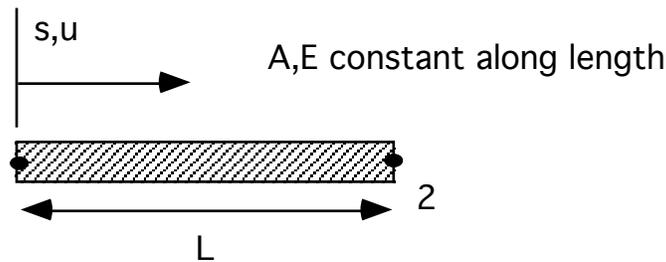


Figure 2

- 2.1)** Determine the strain-displacement matrix $[B]$.
- 2.2)** Determine the element stiffness matrix $[k]$, in terms of A , E , and L .
- 2.3)** Comment on any problems or anomalies you see with the result.

3.) (30%) The influence of mesh layout, element type and load application.

Explore the effect of mesh density and configuration on results. Find the deflection and stress at the locations indicated in the models described below. Summarize your results in the table. Also comment on the results by comparing various mesh densities, comparing FEA results with theory, and comparing the application of loads to FEA models.

Table 2

Model No.	$(\sigma_c/\sigma_{bt})^*$	$\sigma_{eqv, a}^{**}$	$(v_a/v_{bt})^{***}$
I			
II			
III			
IV			
V			

* σ_c is the normal stress σ_{xx} at point C found from FEA, σ_{bt} is the stress σ_{xx} at point C from beam theory

** $\sigma_{eqv, a}$ is the Von Mises equivalent stress σ_{vm} at point A

*** v_a is the vertical deflection at point A found from FEA, v_{bt} is the vertical deflection at point A from beam theory (neglect shear effects).

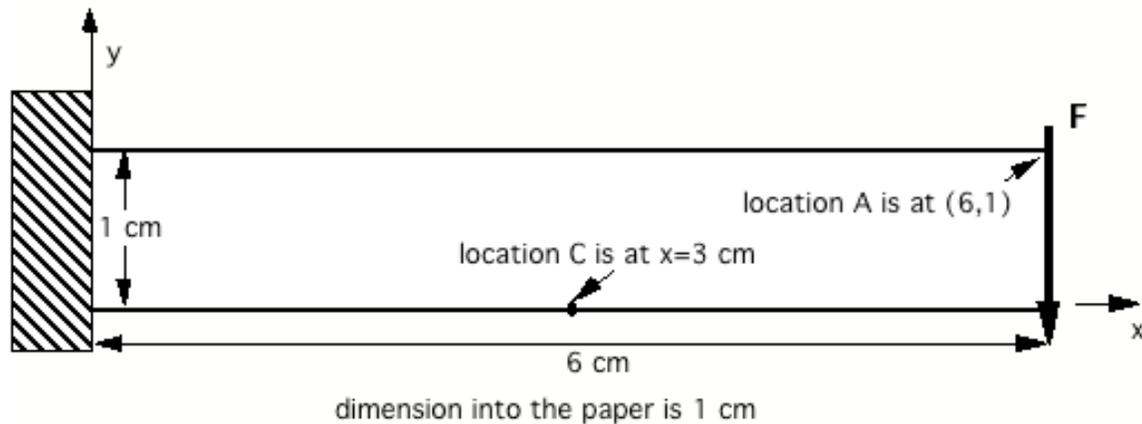


Figure 3

Model I: 2 elements in the x-direction, 1 in the y-direction. Use an element type with corner nodes only. Have the extra shape function turned ON with the appropriate KEYOPT.

Model II: 6 elements in the x-direction, 1 in the y-direction. Use an element type with corner nodes only. Have the extra shape function turned ON (Keyopt(2))

Model III: 6 elements in the x-direction, 1 in the y-direction. Use an element type with corner nodes only. Have the extra shape function turned OFF (Keyopt(2))

Model IV: 2 elements in the x-direction, 2 in the y-direction. Use an element type with mid-side nodes.

Model V: Model with beam elements. Use # of elements for adequate convergence.

Notes:

$F=1$ N, $E=204$ GPa, $\nu=0.3$

Models I-III element possibilities are: PLANE42, SOLID45, SHELL63

Use PLANE82 for Model IV, beam element(s) for Model V.

4.) (30%) A study of modeling approaches.

Consider a shaft that is 0.4 m in length with a rectangular cross-section that measures 0.04 x 0.02 m, as shown below in Figure 4. One end of the shaft is fixed. At the other end a torque T is applied.

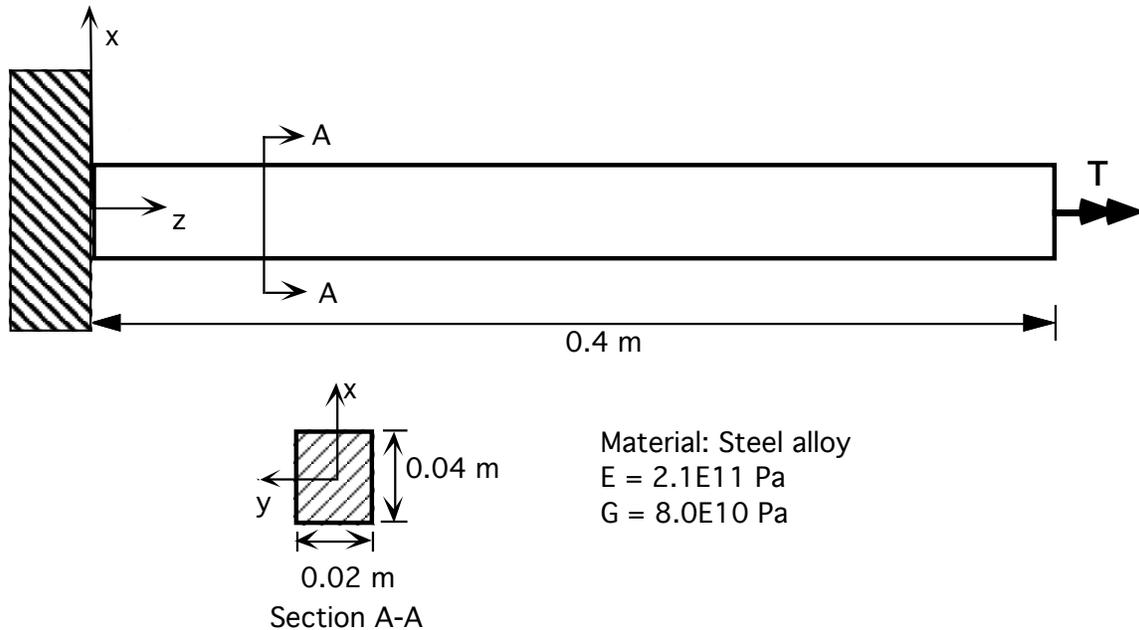


Figure 4

4.1) Use the theoretical solution on the attached page to determine the maximum value of shear stress (τ_{zx}) if T is 1 N-m. With a sketch, clearly indicate where in the shaft this stress is located. Also, what is the angle of rotation (in radians) of the unfixed end of the shaft ($z = 0.4$ m) if the shaft is made out of a steel alloy?

4.2) Now use ANSYS (or another finite element solver) to predict maximum shear stress and the angle of rotation. Begin by describing at least 3 ways in which the torque at $z = 0.4$ m might actually be applied to the model. Choose one of these approaches for use in the rest of the problem.

*HINT: As you work through 4.3) - 4.6), work to keep files organized by removing unwanted files. ANSYS creates a number of temporary files that can use up your disk space quickly. See, "Some Notes on ANSYS Files" below.

4.3)* Before entering ANSYS, read in the log file **beam1.log** posted on coursework. **beam1.log** is the log file for a model of the shaft that was created with a block primitive and free meshing. In ANSYS use "File >> Read Input from...." and double click on **beam1.log**. DO NOT use "File >> resume from...", as with database files.

Now fix the end at $z = 0$, apply the torque as you described in **4.2)**, then run the model. Turn in:

- the name of the element type used, and number of elements in the model
- an isometric contour plot of τ_{zx}
- a printout with the maximum value of τ_{zx} and some indication of this stress location
- calculations showing the rotational angle predicted by the model
- comparison (in percentage form) of the theoretical and predicted values of the maximum shear stress and angle of rotation.

4.4)* Read the log file **beam3.log** posted on coursework. **beam3.log** is the log file for a model of the shaft that was created with a block primitive that has been mapped meshed. Fix the end at $z = 0$, apply the torque as you described in **4.2)**, then run the model. Turn in:

- the name of the element type used, number of elements in the model, and a description of the mesh density (i.e., number of elements in the x-, y- and z-directions)
- an isometric contour plot of τ_{zx}
- a printout with the maximum value of τ_{zx} and some indication of this stress location
- calculations showing the rotational angle predicted by the model
- comparison (in percentage form) of the theoretical and predicted values of the maximum shear stress and angle of rotation.

4.5)* Read the log file **beam2.log** posted on coursework. **beam2.log** is the log file for a model of the shaft that was created with a planar mapped mesh that is extruded. Comment on whether the resultant mesh is like beam1 or beam 3.

4.6)* Choose one of the beam models (beam1, beam2 or beam3) to refine until both the maximum shear stress and angle of rotation are within 5% of the theoretical solution. For the refined model, turn in:

- an isometric plot that shows the mesh layout.
- an isometric contour plot of τ_{zx}
- a printout with the maximum value of τ_{zx} and some indication of this stress location
- calculations showing the rotational angle predicted by the model
- comparison (in percentage form) of the theoretical and predicted values of the maximum shear stress and angle of rotation.

** For students not using ANSYS, replace 4.3) - 4.6) with:

non-ANSYS 4.3) Create a model of the shaft that has 4 solid elements (8-node bricks) in the x-direction, 2 elements in the y-direction, and 40 elements in the z-direction. Now fix the end at $z = 0$, apply the torque as you described in 4.2), then run the model. Turn in:

- an isometric contour plot of τ_{zx}
- a printout with the maximum value of τ_{zx} and some indication where this stress location
- calculations showing the rotational angle predicted by the model
- comparison (in percentage form) of the theoretical and predicted values of the maximum shear stress and angle of rotation.

non-ANSYS 4.4) Refine the model you have created above until both the maximum shear stress and angle of rotation are within 5% of the theoretical solution. For this refined model, turn in:

- an isometric plot that shows the mesh layout.
- an isometric contour plot of τ_{zx}
- a printout with the maximum value of τ_{zx} and some indication where this stress is
- calculations showing the rotational angle predicted by the model
- comparison (in percentage form) of the theoretical and predicted values of the maximum shear stress and angle of rotation.

non-ANSYS 4.5) Briefly describe the meshing tools available in the finite element package that you are using.

Solution for shear stress and rotation of non-circular member in torsion from "Advanced Mechanics of Materials," 3rd edition, by Boresi, Sidebottom, Seely and Smith, John Wiley Publisher, 1978.

Rectangular Cross Sections / The indirect method outlined above fails for rectangular cross sections. Special methods,¹ which are beyond the scope of this book, are required to obtain the torsion solution for rectangular cross sections. We merely summarize some of the results here. Consider the rectangular cross section shown in Fig. 5-3.3. Relations between the cross-sectional dimensions, T , τ_{\max} , and θ take the form

$$\theta = \frac{T}{k_1 G (2b)(2h)^3} \quad \tau_{\max} = \frac{T}{k_2 (2b)(2h)^2} \quad (5-3.14)$$

where τ_{\max} is the maximum shearing stress at the center of the long side at the boundary. Values of the parameters k_1 and k_2 are tabulated in Table 5-3.1 for several values of the ratio b/h . In Eq. (5-3.14), the factor $k_1 G (2b)(2h)^3$ is the torsional rigidity of the section.

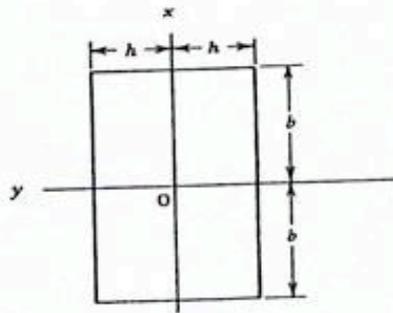


Fig. 5-3.3/Rectangle.

Table 5-3.1

b/h	1	1.5	2	2.5	3	4	6	10	∞
k_1	0.141	0.196	0.229	0.249	0.263	0.281	0.299	0.312	0.333
k_2	0.208	0.231	0.246	0.256	0.267	0.282	0.299	0.312	0.333

Other Cross Sections / There are many torsion members whose cross sections are so complex that exact analytical solutions are difficult to obtain. However, approximate solutions may be obtained by Prandtl's membrane analogy (see Art. 5-4). An important class of torsion members are those with thin walls. Included in the class of thin-walled torsion members are rolled sections and box sections. Approximate solutions for these types of section are obtained in Arts. 5-5 and 5-6 by means of the Prandtl membrane analogy.

Some Notes on ANSYS Files

FILE MANAGEMENT

ANSYS generates a lot of files as it assembles and solves a model. Most of these files are unreadable by you, and unnecessary for you save in your directory. As ANSYS runs, it will name all the files it generates with the current job name (e.g. *beam.db*, *beam.err*, *beam.rst*). You specify this job name in the initial start-up menu under “*jobname*”. All the files will be saved in the “*working directory*” specified in the start-up menu. Therefore, it is easiest to keep track of your files if you start ANSYS in whatever directory you want the files in.

Warning: As the models get complicated, it will be very easy to run out of memory in your directory.

What files do you need to save?

1. *****.db**: This is the file saves your model geometry, material properties, mesh, loads, etc. You can only read it through the graphical user interface of ANSYS (not as a text file). This is the guts of you model – SAVE IT! Because it is the guts, it can get quite large. It may help to zip it (*gzip jobname.db*) before saving it to your directory.

Note: It is possible to have a different job name and *****.db** name. In order to keep your files straight, every time you exit one **.db** and change to a different **.db** make sure to change your job name. Otherwise you will be in your new **.db**, but writing over the files of the old job name.

To start a new **.db**:

- Utility Menu>>File>>Clear Start New...
 - pay attention to the warnings
 - Utility Menu>>File>>Change Job Name ...
 - change the job name to the new **.db** name
 - indicate that you want to start new **.log** and **.err** files
 - Utility Menu>>File>>Resume **.db** from...
 - select the new **.db** file.
2. *****.rst** and *****.rth**: These are your result files (structural and thermal respectively). You can read in the results without having to solve the model again.
 3. *****.log**: This is a text version of **every** command you have entered (including changing viewing angle, etc.). If something happens to your **.db** file, this can save you!

It is sometimes helpful to have a cleaner copy of the `***.log` file, without all the extraneous commands. You can create a `***.lgw` file by:

- Utility Menu>>File>>Write DB Log File
- chose option to write only the essential commands
- save it as a `***.lgw` file

It is helpful to be able to read the text version of the commands. Get used to identifying commands in the `.log` or `.lgw` files.

SESSION EDITOR – the Undo Button

The closest thing ANSYS has to an “undo” button is the Session Editor. It is found on the Analysis Main Menu.

Clicking on it reveals all the commands you have entered since the last `Save.db`. To “undo” a command, simply delete it from the text and click OK. This will run all the commands in the window again (minus the one you deleted).

Beware! The session editor will list **all** the commands you have entered since this last save and run them **all**. This means creating and deleting geometry, creating and deleting meshes, etc. It can take a while if you haven't saved in for a while. Word to the wise: save often, especially right before you do something that you think may not work.

log files for Problem #4 that you can download from the coursework website

! ME309 beam1: Beam created by block approach and free meshed

```

/BATCH
/FILNAM, beam1
/Title, beam1: Beam created by block approach and free meshed
/PREP7

! assign material properties for steel in metric units
UIMP,1,EX, , ,2.1e11,
UIMP,1,NUXY, , ,.3,

! create solid model
BLOCK,0,.04,0,.02,0,.4,

! choose element type
et,1,solid92

! free mesh model
! adjust the number 10 to change mesh density (10=coarse, 1=fine)
smrt,10
mshape,1,3d
mshkey,0
vmesh,all

```

! ME309 beam2.log: Beam created by extrusion of mapped area mesh

```

/BATCH
/FILNAM, beam2
/Title, beam2: Beam created by extrusion approach
/PREP7

! assign area and volume element types
ET,1,PLANE42
ET,2,SOLID45

! assign material properties for steel in metric units
UIMP,1,EX, , ,2.1e11,
UIMP,1,NUXY, , ,.3,

!create area model

```

```
RECTNG,0,0.04,0,0.02,
```

```
! create mapped mesh on area model
```

```
FLST,5,1,4,ORDE,1
```

```
FITEM,5,3
```

```
CM,_Y,LINE
```

```
LSEL, , , ,P51X
```

```
CM,_Y1,LINE
```

```
CMSEL, ,_Y
```

```
! adjust the number 4 to change the number of elements in x-direction
```

```
LESIZE,_Y1, , ,4,1,
```

```
CMDEL,_Y
```

```
CMDEL,_Y1
```

```
FLST,5,1,4,ORDE,1
```

```
FITEM,5,4
```

```
CM,_Y,LINE
```

```
LSEL, , , ,P51X
```

```
CM,_Y1,LINE
```

```
CMSEL, ,_Y
```

```
! adjust the number 2 to change the number of elements in y-direction
```

```
LESIZE,_Y1, , ,2,1,
```

```
CMDEL,_Y
```

```
CMDEL,_Y1
```

```
MSHAPE,0,2D
```

```
MSHKEY,1
```

```
CM,_Y,AREA
```

```
ASEL, , , , 1
```

```
CM,_Y1,AREA
```

```
CHKMSH,'AREA'
```

```
CMSEL,S,_Y
```

```
! mesh the area
```

```
AMESH,_Y1
```

```
CMDEL,_Y
```

```
CMDEL,_Y1
```

```
CMDEL,_Y2
```

```
/UI,MESH,OFF
```

```
TYPE, 2
```

```
! setup mesh for extrusion
```

```
! adjust the number 40 to change the number of elements in z-direction
```

```
EXTOPT,ESIZE,40,1,
```

```
EXTOPT,ACLEAR,0
```

```

EXTOPT,ATTR,0,0,0
MAT,1
REAL,_Z4
ESYS,0

```

```

! extrude area
VOFFST,1,0.4, ,

```

! ME309 beam3.log: Beam created with block approach and mapped mesh

```

/BATCH
/FILNAM, beam3
/Title, beam3: Beam created with block approach and mapped mesh
/PREP7

```

```

! assign element type
ET,1,45

```

```

! assign material properties for steel in metric units
UIMP,1,EX, , ,2.1e11,
UIMP,1,NUXY, , ,.3,

```

```

! create solid model
BLOCK,0,0.04,0,0.02,0,0.4,

```

```

! create mapped mesh
FLST,5,1,4,ORDE,1
FITEM,5,12
CM,_Y,LINE
LSEL, , , ,P51X

```

```

CM,_Y1,LINE
CMSEL,,_Y

```

```

! adjust the number 40 to change the number of elements in z-direction
LESIZE,_Y1, , ,40,1,
CMDEL,_Y
CMDEL,_Y1

```

```

FLST,5,1,4,ORDE,1
FITEM,5,2
CM,_Y,LINE
LSEL, , , ,P51X

```

```
CM,_Y1,LINE  
CMSEL,_,_Y
```

```
! adjust the number 4 to change the number of elements in x-direction
```

```
LESIZE,_Y1, , ,4,1,  
CMDEL,_Y  
CMDEL,_Y1
```

```
FLST,5,1,4,ORDE,1  
FITEM,5,3  
CM,_Y,LINE  
LSEL, , , ,P51X
```

```
CM,_Y1,LINE  
CMSEL,_,_Y
```

```
! adjust the number 2 to change the number of elements in y-direction
```

```
LESIZE,_Y1, , ,2,1,  
CMDEL,_Y  
CMDEL,_Y1
```

```
MSHAPE,0,3D  
MSHKEY,1
```

```
CM,_Y,VOLU  
VSEL, , , , 1  
CM,_Y1,VOLU  
CHKMSH,'VOLU'  
CMSEL,S,_Y
```

```
VMESH,_Y1
```

```
CMDEL,_Y  
CMDEL,_Y1  
CMDEL,_Y2
```

```
/UI,MESH,OFF
```