





|| Jai Sri Gurudev ||

Sri Adichunchanagiri Shikshana Trust®

**BGS INSTITUTE OF TECHNOLOGY**

B G Nagara – 571448

**Department of Mechanical Engineering**

**SUBJECT : MODELING & ANALYSIS LAB**  
**NAME : MANUAL**

**SUBJECT : 17MEL68**  
**CODE**

**SEMESTER : VI**

---

# **CONTENTS**

<b>Sl No</b>	<b>Title</b>	<b>Page no</b>
1.	Getting Started with ANSYS 10	03
2.	General Steps	07
3.	Simply Supported Beam	08
4.	Cantilever Beam	10
5.	Simply Supported Beam with Uniformly distributed load	12
6.	Beam with angular loads, one end hinged and at other end roller support	14
7.	Beam with moment and overhung	16
8.	Simply Supported Beam with Uniformly varying load	18
9.	Bars of Constant Cross-section Area	20
10.	Stepped Bar	22
11.	Bars of Tapered Cross section Area	24
12.	Trusses	26
13.	Stress analysis of a rectangular plate with a circular hole	30
14.	Corner angle bracket	32
15.	Spanner under plane stress	34
16.	Thermal Analysis	37
17.	Modal Analysis of Cantilever beam for natural Frequency determination	41
18.	Harmonic Analysis of Cantilever beam	42
19.	Dynamic analysis of bar subjected to forcing function	44
20.	Laminar Flow Analyses in a 2-D Duct	46

---



## COMPUTER AIDED MODELING AND ANALYSIS LABORATORY

Subject Code	: 17MEL68	IA Marks	: 25
No. of Practical Hrs/Week	: 03	Exam Hours	: 03
Total No. of Practical Hrs.	: 42	Exam Marks	: 50

---

### PART-A

#### Study of a FEA package and modeling stress analysis of

- a. Bars of constant cross section area, tapered cross section area and stepped bar  
**6 Hours**
- b. Trusses- (Minimum 2 exercises)  
**3 Hours**
- c. Beams - Simply supported, cantilever. beams with UDL, beams with varying load.etc (Minimum 6 exercises)  
**12 Hours**

### PART -B

- a) Stress analysis of a rectangular plate with a circular hole  
**3 Hours**
- b) Thermal Analysis - 2D problem with conduction and convection boundary conditions (Minimum 2 exercises)  
**6 Hours**
- c) Fluid flow Analysis - Potential distribution in the 2 - D bodies  
**3 Hours**
- d) Dynamic Analysis
  - 1) Fixed- fixed beam for natural frequency determination
  - 2) Bar subjected to forcing function
  - 3) Fixed- fixed beam subjected to forcing function**9 Hours**

#### REFERENCE BOOKS:

1. A first course in the Finite element method by Daryl L Logan, Thomason, Third Edition
2. Fundamentals of FEM by Hutton- McGraw Hill, 2004
3. Finite Element Analysis by George R. Buchanan, Schaum Series

#### Scheme for Examination:

One Question from Part A	-	20Marks (05 Write up + 15)
One Question from Part B	-	20Marks (05 Write up + 15)
Viva-Voce	-	10 Marks
<b>Total</b>	-	<b>50 Marks</b>

# Getting Started with ANSYS 10

---

## Performing a Typical ANSYS Analysis

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analyses for different engineering disciplines.

A typical ANSYS analysis has three distinct steps:

- Build the model.
- Apply loads and obtain the solution.
- Review the results.

## Building a Model

Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First, you specify a jobname and analysis title. Then, you use the PREP7 preprocessor to define the element types, element real constants, material properties, and the model geometry.

### Specifying a Jobname and Analysis Title

This task is not required for an analysis, but is *recommended*.

### Defining the Jobname

The *jobname* is a name that identifies the ANSYS job. When you define a jobname for an analysis, the jobname becomes the first part of the name of all files the analysis creates. (The extension or suffix for these files' names is a file identifier such as .DB.) By using a jobname for each analysis, you insure that no files are overwritten.

If you do not specify a jobname, all files receive the name *FILE* or *file*, depending on the operating system.

Command(s): **/FILNAME**

GUI: **Utility Menu>File>Change Jobname**

### Defining Element Types

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category: BEAM4, PLANE77, SOLID96, etc. The following element categories are available:

BEAM	PLANE
COMBINation	SHELL
CONtACT	
FLUID	SOLID
	SOURCe
HYPERelastic	SURFace
INFINite	
LINK	TARGEt
MASS	USER
MATRIX	INTERface
PIPE	VISCOelastic (or viscoplastic)

The element type determines, among other things:

The degree-of-freedom set (which in turn implies the discipline-structural, thermal, magnetic, electric, quadrilateral, brick, etc.)  
Whether the element lies in two-dimensional or three-dimensional space.

For example, BEAM4, has six structural degrees of freedom (UX, UY, UZ, ROTX, ROTY, ROTZ), is a line element, and can be modeled in 3-D space. PLANE77 has a thermal degree of freedom (TEMP), is an eight-node quadrilateral element, and can be modeled only in 2-D space.

### Defining Element Real Constants

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. For example, real constants for BEAM3, the 2-D beam element, are area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant (SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS). Not all element types require real constants, and different elements of the same type may have different real constant values.

As with element types, each set of real constants has a reference number, and the table of reference number versus real constant set is called the *real constant table*. While defining the elements, you point to the appropriate real constant reference number using the **REAL** command (**Main Menu**> **Preprocessor**>**Create**>**Elements**>**Elem Attributes**).

### Defining Material Properties

Most element types require material properties. Depending on the application, material properties may be:

Linear or nonlinear  
Isotropic, orthotropic, or anisotropic  
Constant temperature or temperature-dependent.

As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the *material table*. Within one analysis, you may have multiple material property sets (to correspond with multiple materials used in the model). ANSYS identifies each set with a unique reference number.

**Main Menu> Preprocessor> Material Props> Material Models.**

### **Creating the Model Geometry**

Once you have defined material properties, the next step in an analysis is generating a finite element model-nodes and elements-that adequately describes the model geometry.

There are two methods to create the finite element model: solid modeling and direct generation. With *solid modeling*, you describe the geometric shape of your model, then instruct the ANSYS program to automatically *mesh* the geometry with nodes and elements. You can control the size and shape of the elements that the program creates. With *direct generation*, you "manually" define the location of each node and the connectivity of each element. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc. are available.

### **Apply Loads and Obtain the Solution**

In this step, you use the SOLUTION processor to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution. You also can apply loads using the PREP7 preprocessor.

#### **Applying Loads**

The word *loads* as used in this manual includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally and internally applied loads. Loads in the ANSYS program are divided into six categories:

- DOF Constraints
- Forces
- Surface Loads
- Body Loads
- Inertia Loads
- Coupled-field Loads

You can apply most of these loads either on the solid model (keypoints, lines, and areas) or the finite element model (nodes and elements).

Two important load-related terms you need to know are load step and substep. A *load step* is simply a configuration of loads for which you obtain a solution. In a structural analysis, for example, you may apply wind loads in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.



*Substeps* are incremental steps taken within a load step. You use them mainly for accuracy and convergence purposes in transient and nonlinear analyses. Substeps are also known as *time steps* taken over a period of time.

### **Initiating the Solution**

To initiate solution calculations, use either of the following:

Command(s): **SOLVE**

GUI: **Main Menu>Solution>Current LS**

When you issue this command, the ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file (*Jobname.RST*, *Jobname.RTH*, *Jobname.RMG*, or *Jobname.RFL*) and also to the database. The only difference is that only one set of results can reside in the database at one time, while you can write all sets of results (for all substeps) to the results file.

### **Review the Results**

Once the solution has been calculated, you can use the ANSYS postprocessors to review the results.

# General Steps

## Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok  
File – change job name – enter new job name – xxxx – ok  
File – change title – enter new title – yyy – ok

## Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

## Step 3: Preprocessor

**Element type** – select type of element from the table and the required options

**Real constants** – give the details such as thickness, areas, moment of inertia, etc.  
required depending on the nature of the problem.

**Material Properties** – give the details such as Young's modulus, Poisson's ratio etc.  
depending on the nature of the problem.

**Step 4: Modeling** – create the required geometry such as nodes elements, area, volume by using  
the appropriate options.

**Step 5: Generate** – Elements/ nodes using Mesh Tool if necessary (in 2D and 3D problems)

**Step 6: Apply boundary conditions/loads** such as DOF constraints, Force/Momentum,  
Pressure etc.

**Step 7: Solution** – Solve the problem

**Step 8: General Post Processor** – plot / list the required results.

**Step 9: Plot ctrls – animate – deformed shape – def+undeformed-ok**

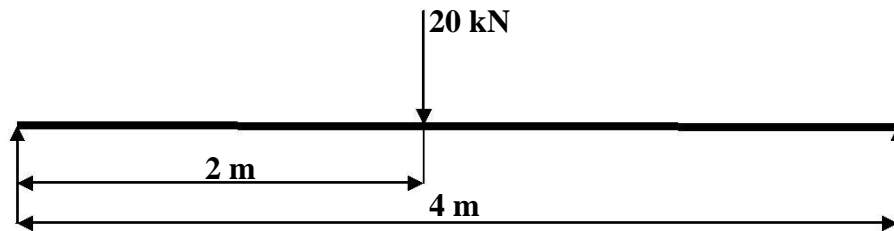
**Step 10:** to save the solution  
ansys tool bar- save,,,,model

# PART A

## BEAMS

### 1. Simply Supported Beam

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m \* 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



#### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

#### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

#### Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

**Real constants** – Add – ok – real constant set no – 1 – c/s area – 0.2\*0.3 moment of inertia –  $0.2*0.3**3/12$  – total beam height – 0.3 – ok.

**Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – PRXY – 0.27 – ok – close.

#### Step 4: Preprocessor

**Modeling** – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 2 (x value w.r.t first node) – apply (second node is created) – 4 (x value w.r.t first node) – ok (third node is created).

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 apply – pick 2 & 3 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 3 – apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – -20000 (-ve value) – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

### Step 7: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.

Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC –

SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num –

SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

**NOTE:** For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

### Step 8: General Post Processor

Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).

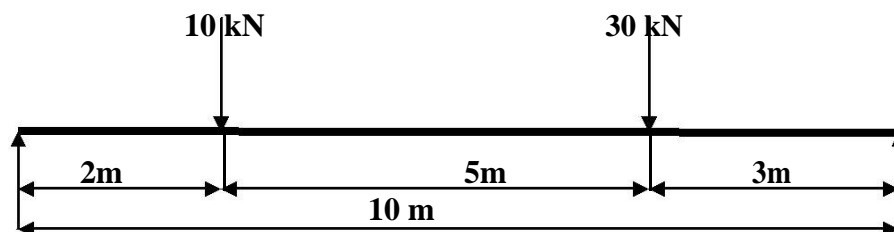
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

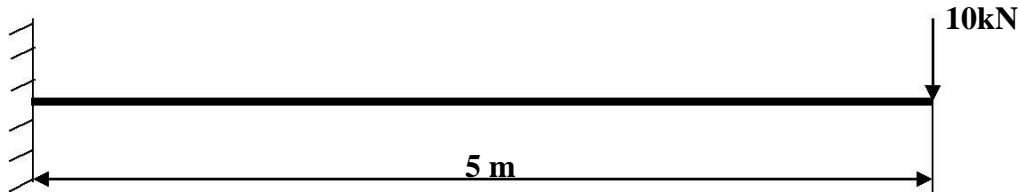
**Step 9: PlotCtrls** – Animate – Deformed results – DOF solution – USUM – ok.

### ASSIGNMENT



## 2. Cantilever Beam

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m \* 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.2\*0.3 moment of inertia –  $0.2*0.3**3/12$  – total beam height – 0.3 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – PRXY – 0.27 –ok – close.

### Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 5 (x value w.r.t first node) – ok (second node is created).

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – ALL DOF – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value - -10000 (-ve value) – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

### Step 7: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.

Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC –

SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num –

SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

**NOTE:** For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

### Step 8: General Post Processor

Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).

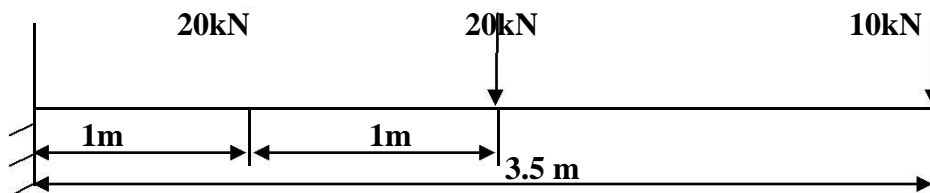
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

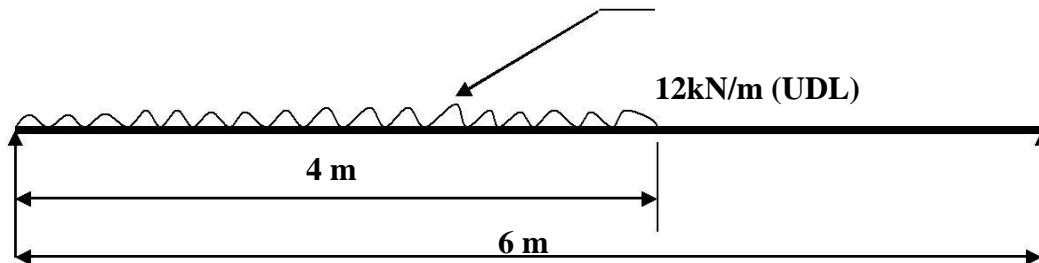
**Step 9: PlotCtrls** – Animate – Deformed results – DOF solution – USUM – ok.

### ASSIGNMENT



### 3. Simply Supported Beam with Uniformly distributed load.

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m \* 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



#### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

#### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

#### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.2\*0.3 moment of inertia –  $0.2*0.3**3/12$  – total beam height – 0.3 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – PRXY – 0.27 –ok – close.

#### Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 4 (x value w.r.t first node) – apply (second node is created) – 6 (x value w.r.t first node) – ok (third node is created).

Create – Nodes – Fill between Nds – pick 1 & 2 – apply – number of nodes to fill 7 – starting node no – 4 – ok.

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 4 apply– pick 4 & 5 apply– pick 5 & 6 apply– pick 6 & 7 apply– pick 7 & 8 apply– pick 8 & 9 apply – pick 9 & 10 apply– pick 10 & 2 apply – pick 2 & 3 – ok (elements are created through nodes).

#### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 3 – apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Pressure – on Beams – pick all elements between nodes 1 & 2 – apply – pressure value at node I – 12000 – pressure value at node J – 12000 – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

### Step 7: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.

Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC –

SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

**NOTE:** For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

### Step 8: General Post Processor

Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).

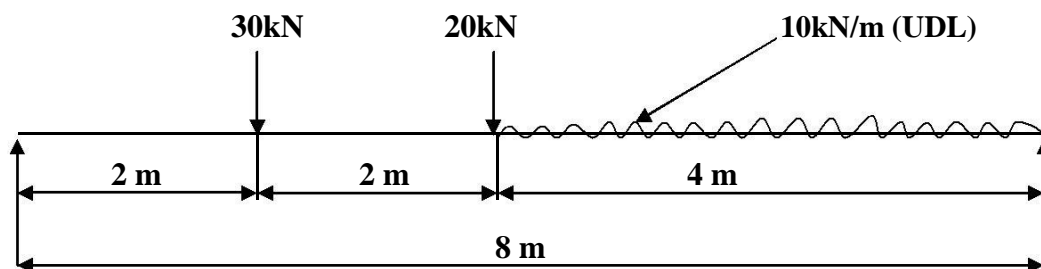
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

**Step 9: PlotCtrls** – Animate – Deformed results – DOF solution – USUM – ok.

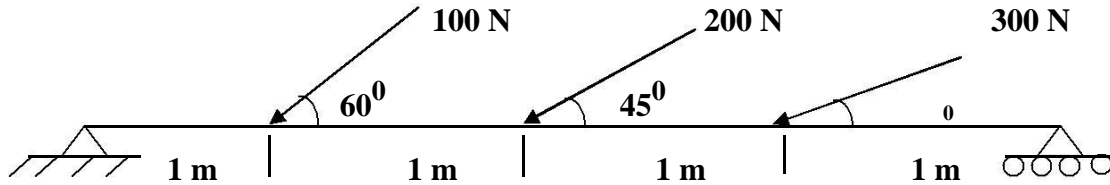
## ASSIGNMENT





#### 4. Beam with angular loads, one end hinged and at the other end roller support.

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m \* 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



#### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

#### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

#### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.2\*0.3 moment of inertia –  $0.2*0.3**3/12$  – total beam height – 0.3 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – PRXY – 0.27 –ok – close.

#### Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 1 (x value w.r.t first node) – apply (second node is created) – 2 (x value w.r.t first node) – apply (third node is created) – 3 (x value w.r.t first node) – apply (forth node is created) – 4 (x value w.r.t first node) – ok (fifth node is created).

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 4 – apply – pick 4 & 5 – ok (elements are created through nodes).

Create – Nodes – Rotate nodes CS – by angles – pick node 2 – apply – about nodal z-axis – 60 – apply – pick node 3 – apply about nodal z- axis – 45 – apply – pick node 4 – apply – about nodal z – axis – 30 – ok.

#### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – UX & UY – apply – pick node 5 – apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value - -100 (-ve value) – apply – pick node 3 – apply – direction of For/Mom – FX – Force/Moment value - -200 (-ve value) – apply – pick node 4 – apply – direction of For/Mom – FX – Force/Moment value - -300 (-ve value) – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

### Step 7: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.  
Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC –

SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

**NOTE:** For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

### Step 8: General Post Processor

Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).

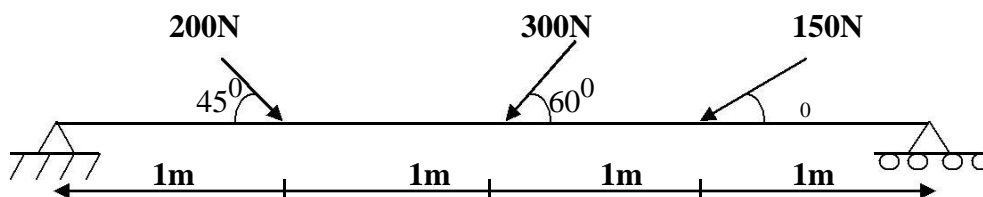
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

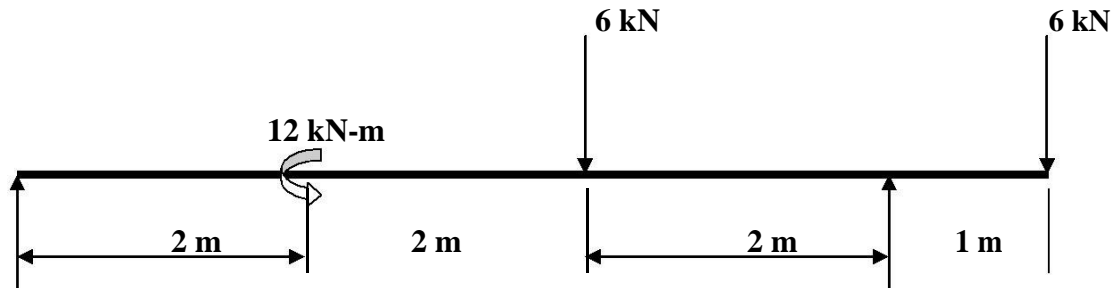
**Step 9: PlotCtrls** – Animate – Deformed results – DOF solution – USUM – ok.

### ASSIGNMENT



## 5. Beam with moment and overhung

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of  $0.2 \times 0.3$  m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area –  $0.2 \times 0.3$  moment of inertia –  $0.2 \times 0.3^3 / 12$  – total beam height – 0.3 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX –  $210 \times 10^9$  – PRXY – 0.27 – ok – close.

### Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 2 (x value w.r.t first node) – apply (second node is created) – 4 (x value w.r.t first node) – apply (third node is created) – 6 (x value w.r.t first node) – apply (fourth node is created) – 7 (x value w.r.t first node) – ok (fifth node is created).

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 4 – apply – pick 4 & 5 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – MZ – Force/Moment value - 12000 (anticlockwise, +ve value) – apply –

pick node 3 – apply – direction of For/Mom – FY – Force/Moment value - -6000 (-ve value) – apply – pick node 5 – apply – direction of For/Mom – FY – Force/Moment value - -6000 (-ve value) – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

### Step 7: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.  
Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC –

SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

**NOTE:** For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

### Step 8: General Post Processor

Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).

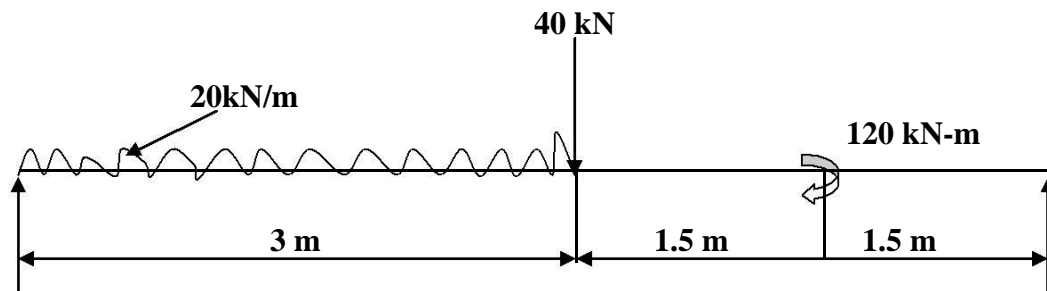
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

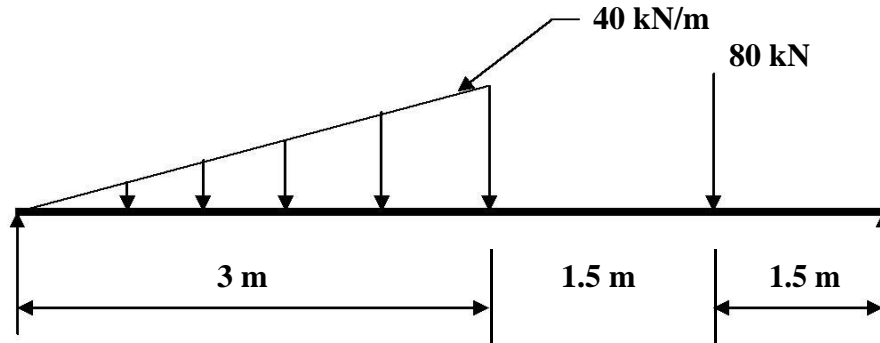
**Step 9: PlotCtrls** – Animate – Deformed results – DOF solution – USUM – ok.

## ASSIGNMENT



## 6. Simply Supported Beam with Uniformly varying load.

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m \* 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area –  $0.2 \times 0.3$  moment of inertia –  $0.2 \times 0.3^3 / 12$  – total beam height – 0.3 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX –  $210 \times 10^9$  – PRXY – 0.27 – ok – close.

### Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 3 (x value w.r.t first node) – apply (second node is created) – 4.5 (x value w.r.t first node) – apply (third node is created) – 6 (x value w.r.t first node) – ok (fourth node is created).

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 4 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Pressure – on Beams – pick element between nodes 1 & 2 – apply – pressure value at node I – 0 (value) – pressure value at node J – 40000 – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value - -80000 (-ve value) – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

### Step 7: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.

Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC –

SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num –

SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

**NOTE:** For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

### Step 8: General Post Processor

Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).

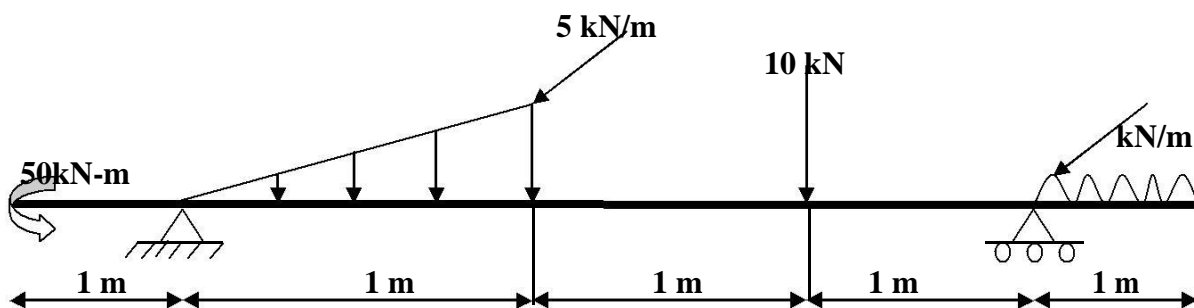
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

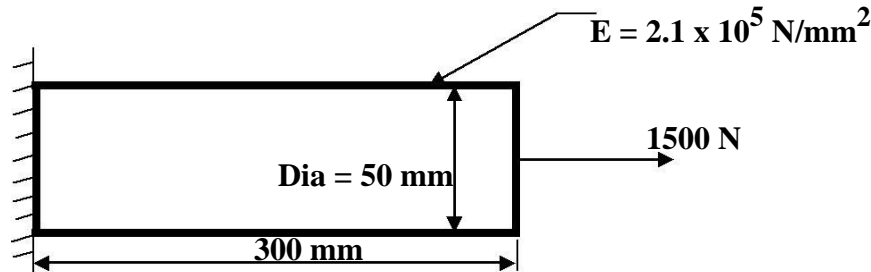
**Step 9: PlotCtrls** – Animate – Deformed results – DOF solution – USUM – ok.

### ASSIGNMENT



## Bars of Constant Cross-section Area

Consider the bar shown in figure below. Determine the Nodal Displacement, Stress in each element, Reaction forces.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area –  $22/7 * 50^{**2} / 4$  – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX –  $2.1e5$  – PRXY – 0.27 – ok – close.

### Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 300 (x value w.r.t first node) – ok (second node is created).

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – All DOF – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 1500 (+ve value) – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

**Step 7: General Post Processor**

Element table – Define table – Add – ‘ Results data item’ – By Sequence num – LS – LS1 – ok.

**Step 8: General Post Processor**

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

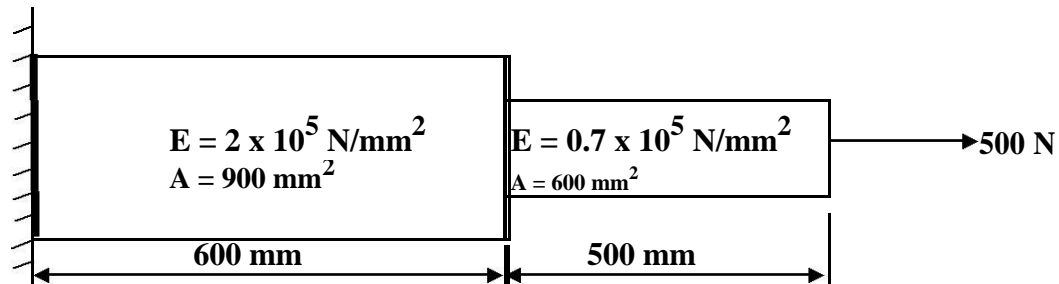
List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

**Step 9: PlotCtrls – Animate – Deformed shape – def+undeformed-ok**



## Stepped Bars

Consider the stepped bar shown in figure below. Determine the Nodal Displacement, Stress in each element, Reaction forces.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 900 – apply – real constant set no – 2 – c/s area – 600 – ok – close.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX –  $2e5$  – ok, – Material – New model – Define material ID – 2 – ok – Structural – Linear – Elastic – Isotropic – EX –  $0.7e5$  – ok – close.

### Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 600 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 1100 (x value w.r.t first node) – ok (third node is created).

Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 1 – ok Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Create – Elements – Elem Attributes – Material number – 2 – Real constant set number – 2 – ok Auto numbered – Thru Nodes – pick 2 & 3 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – All DOF – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FX – Force/Moment value – 500 (+ve value) – ok.

**Step 6: Solution**

Solve – current LS – ok (Solution is done is displayed) – close.

**Step 7: General Post Processor**

Element table – Define table – Add – ‘ Results data item’ – By Sequence num – LS – LS1 – ok.

**Step 8: General Post Processor**

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

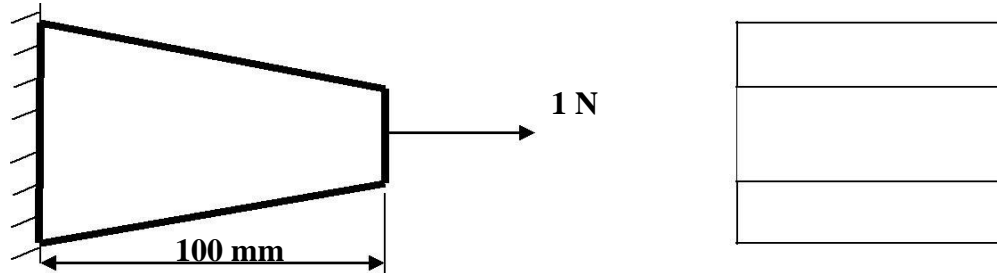
List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

**Step 9: PlotCtrls – Animate – Deformed shape – def+undeformed-ok**

## Bars of Tapered Cross section Area

Consider the Tapered bar shown in figure below. Determine the Nodal Displacement, Stress in each element, Reaction forces.



$E = 2 \times 10^5 \text{ N/mm}^2$ , Area at root =  $20 \times 20 = 400 \text{ mm}^2$ , Area at the end =  $20 \times 10 = 200 \text{ mm}^2$ .

### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – tapered 54 – ok- close.

Real constants – Add – ok – real constant set no – 1 – cross-sectional AREA1 – 400 – moment of inertia about Z IZ1 –  $20 \times 20^3 / 12$  – cross-sectional AREA2 – 200 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX –  $2e5$  – PRXY – 0.27 – ok – close.

### Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 100 (x value w.r.t first node) – ok (second node is created).

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – ALL DOF – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 1 (+ve value) – ok.

**Step 6: Solution**

Solve – current LS – ok (Solution is done is displayed) – close.

**Step 7: General Post Processor**

Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC – SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

Element table – define table – add – ‘Results data item’ – By Sequence num – NMISC – NMISC, 1 – apply, ‘results data item’ – By Sequence num – NMISC – NMISC, 3 – ok.

**Step 8: General Post Processor**

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).

Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

**NOTE:** For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12. For Maximum Stress diagram use the combination NMISC 1 & NMISC 3.

Plot results – contour plot – Line Element Results – Elem table item at node I – NMIS1 – Elem table item at node J – NMIS3 – ok (the maximum stress value will be displayed).

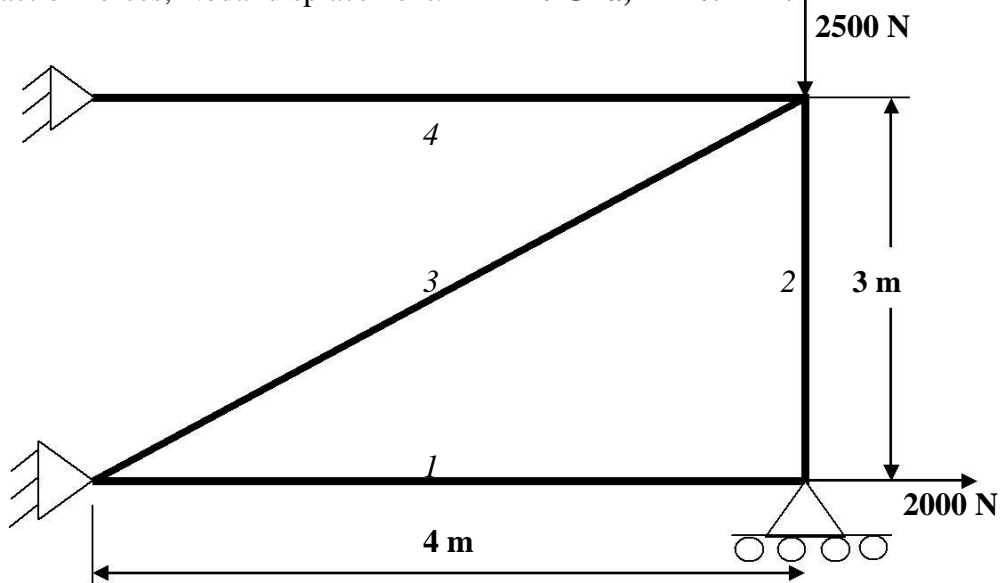
List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

**Step 9: PlotCtrls** – Animate – Deformed shape – def+undeformed-ok.

## TRUSSES

**Prob. 1.** Consider the four bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement.  $E = 210 \text{ GPa}$ ,  $A = 0.1 \text{ m}^2$ .



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

**Real constants** – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.

**Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – ok – close.

### Step 4: Preprocessor

**Modeling** – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 4 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 4, 3 (x, y value w.r.t first node) – apply (third node is created) – 0, 3 (x, y value w.r.t first node) – ok (fourth node is created).

Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 1 – ok  
Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 1 – apply – pick 3 & 4 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 & 4 – apply – DOFs to be constrained – All DOF – ok – on Nodes – pick node 2 – apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

### Step 7: General Post Processor

Element table – Define table – Add – ‘Results data item’ – By Sequence num – LS – LS1 – ok.

### Step 8: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

Plot results – contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

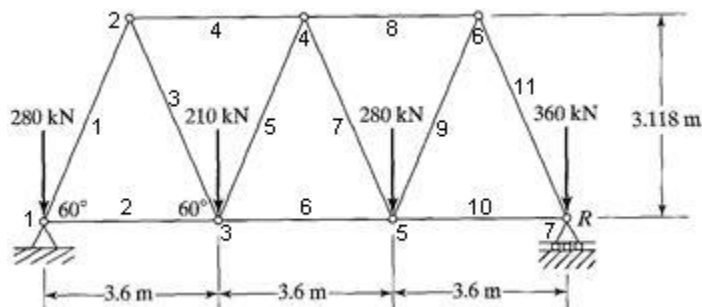
List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

**Step 9: PlotCtrls** – Animate – Deformed shape – def+undeformed-ok

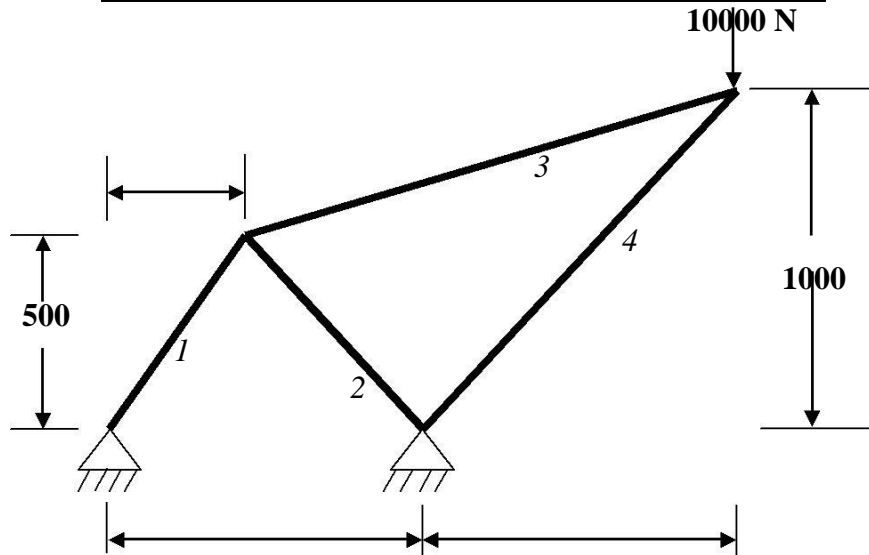
## ASSIGNMENT

Determine the nodal deflections, reaction forces, and stress for the truss system shown below ( $E = 200\text{GPa}$ ,  $A = 3250\text{mm}^2$ ).



**Prob. 2.** For the given data, find internal stresses developed, Nodal displacement in the planar truss shown in figure when a vertically downward load of 10000 N is applied as shown.

Member	C/s area mm <sup>2</sup>	E N/mm <sup>2</sup>
1	200	2 x 10 <sup>5</sup>
2	200	
3	100	
4	100	



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

**Real constants** – Add – ok – real constant set no – 1 – c/s area – 200 – apply – real constant set no – 2 – c/s area – 100 – ok – close.

**Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 –

PRXY – 0.27 – ok – close.

### Step 4: Preprocessor

**Modeling** – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 1000 (x value w.r.t first node) – apply (second node is created) – 500, 500 (x, y value w.r.t first node) – apply (third node is created) – 2000, 1000 (x, y value w.r.t first node) – ok (fourth node is created).

Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 1 – ok  
– Auto numbered – Thru Nodes – pick 1 & 3 – apply – pick 2 & 3 – ok – Elem Attributes –  
Material number – 1 – Real constant set number – 2 – ok – Auto numbered – Thru Nodes – pick  
3 & 4 – apply – pick 2 & 4 – ok (elements are created through nodes).

### Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 & 2 – apply  
– DOFs to be constrained – All DOF – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 4 – apply –  
direction of For/Mom – FY – Force/Moment value – -10000 (-ve value) – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

### Step 7: General Post Processor

Element table – Define table – Add – ‘Results data item’ – By Sequence num – LS – LS1 – ok.

### Step 8: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table  
item at node J – LS1 – ok (Line Stress diagram will be displayed).

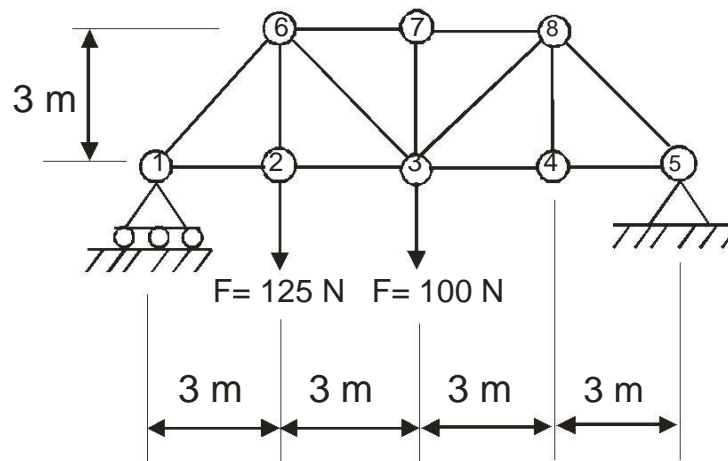
Plot results – contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be  
displayed with the node numbers).

**Step 9: PlotCtrls** – Animate – Deformed shape – def+undeformed-ok

### ASSIGNMENT

Note: Cross-sectional area of truss members =  $3.0E-4 \text{ m}^2$ ; Modulus of Elasticity =  $2.07E11 \text{ N/m}^2$ .  
Circled numbers shown are node numbers.



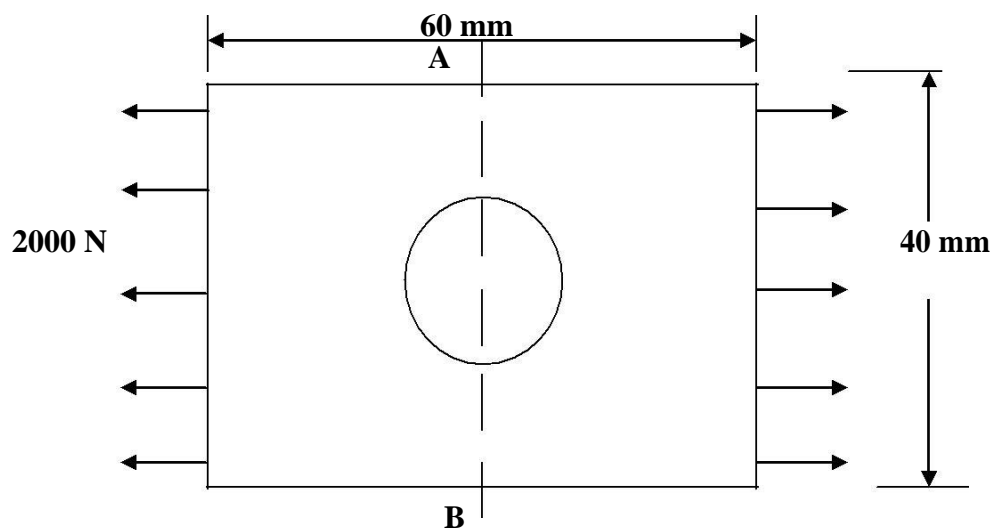


## PART B

### Stress analysis of a rectangular plate with a circular hole

\*\*\*\* For 2D and 3D problems, after the geometry has been created meshing is to be done (elements/ nodes are created) \*\*\*\*

**Problem 1.** In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A-B (you may use  $t = 1$  mm).  $E = 210$  GPa,  $t = 1$  mm, Poisson's ratio = 0.3, Dia of the circle = 10 mm, Analysis assumption – plane stress with thickness is used.



#### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

#### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

#### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 42 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

Real constants – Add – ok – real constant set no – 1 – Thickness – 1 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX –  $2.1e5$  – PRXY – 0.3 – ok – close.

#### **Step 4: Preprocessor**

**Modeling** – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 60, 0, 40 – ok.  
Create – Area – Circle – solid circle – X, Y, radius – 30, 20, 5 – ok.

Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (rectangle) – apply  
– pick area which is to be deleted (circle) – ok.

Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine  
– pick all – Level of refinement – 3 – ok.

#### **Step 5: Preprocessor**

Loads – Define loads – apply – Structural – Displacement – on Nodes – select box – drag the left  
side of the area – apply – DOFs to be constrained – ALL DOF – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes – select box – drag the  
right side of the area – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve  
value) – ok.

#### **Step 6: Solution**

Solve – current LS – ok (Solution is done is displayed) – close.

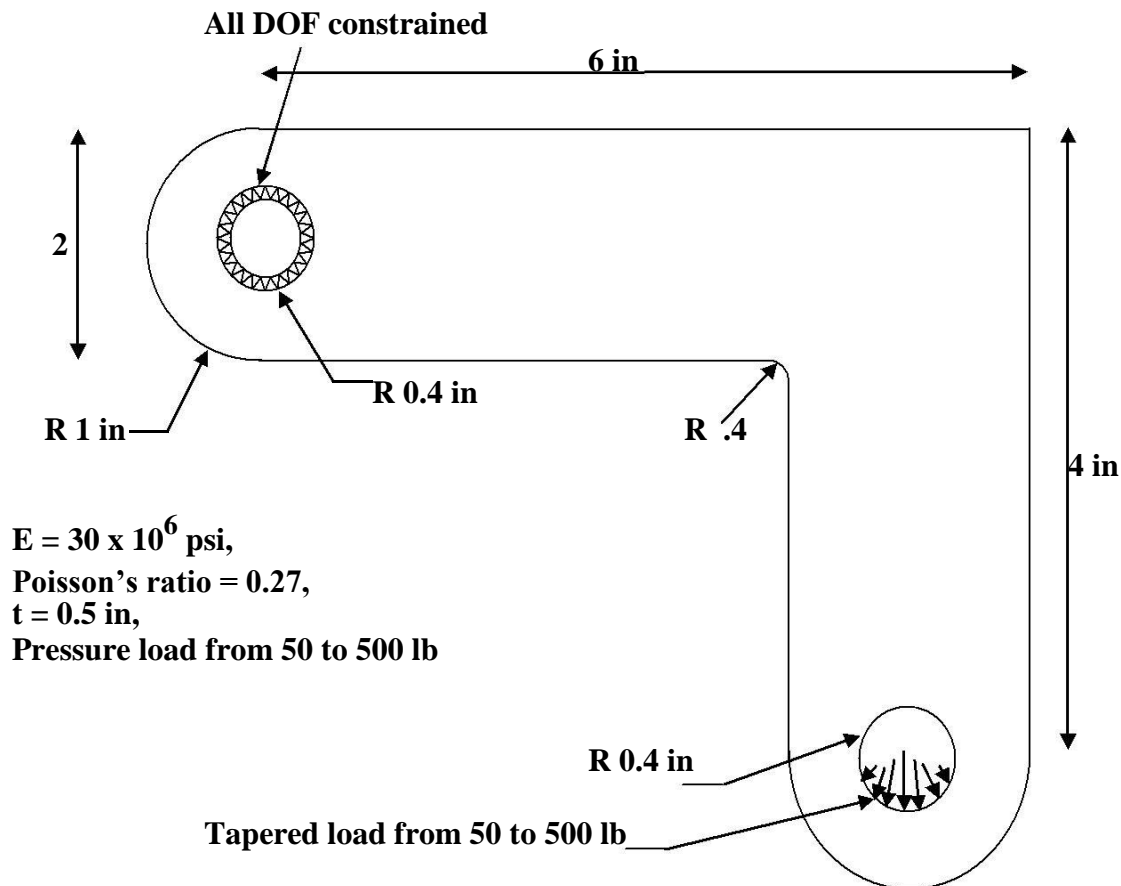
#### **Step 7: General Post Processor**

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress  
distribution diagram will be displayed).

**Step 8: PlotCtrls** – Animate – Deformed shape – def+undeformed-ok

**Problem 2.** The corner angle bracket is shown below. The upper left hand pin-hole is constrained around its entire circumference and a tapered pressure load is applied to the bottom of lower right hand pin-hole. Compute Maximum displacement, Von-Mises stress.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – Solid – Quad 8 node – 82 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

**Real constants** – Add – ok – real constant set no – 1 – Thickness – 0.5 – ok.

**Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 30e6 – PRXY – 0.27 – ok – close.

#### **Step 4: Preprocessor**

**Modeling** – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 6, 0, 2 – apply – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 4, 6, -2, 2 – ok. Create – Area – Circle – solid circle – X, Y, radius – 0, 1, 1 – apply – X, Y, radius – 5, -2, 1 – ok.

Operate – Booleans – Add – Areas – pick all.

Create – Lines – Line fillet – pick the two lines where fillet is required – apply – fillet radius – 0.4 – ok. Create – Areas – Arbitrary – by lines – pick filleted lines – ok. Operate – Booleans – Add – Areas – pick all. Create – Area – Circle – solid circle – X, Y, radius – 0, 1, 0.4 – apply – X, Y, radius – 5, -2, 0.4 – ok.

Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (bracket) – apply – pick areas which is to be deleted (pick two circles) – ok.

Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

#### **Step 5: Preprocessor**

Loads – Define loads – apply – Structural – Displacement – on Lines – select the inner lines of the upper circle – apply – DOFs to be constrained – ALL DOF – ok.

Loads – Define loads – apply – Structural – Pressure – on Lines – *Pick line defining bottom left part of the circle* – apply – load PRES value – 50 – optional PRES value – 500 – ok. Structural – Pressure – on Lines – *Pick line defining bottom right part of the circle* – apply – load PRES value – 500 – optional PRES value – 50 – ok.

#### **Step 6: Solution**

Solve – current LS – ok (Solution is done is displayed) – close.

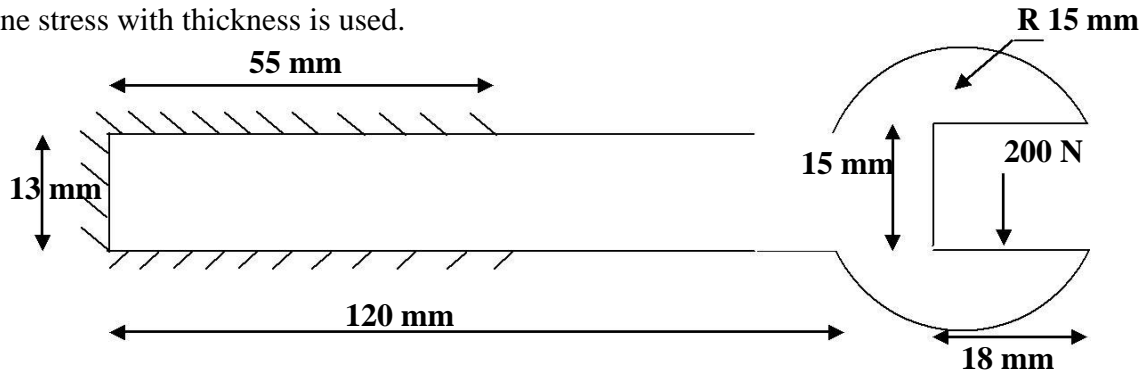
#### **Step 7: General Post Processor**

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

**Step 8: PlotCtrls** – Animate – Deformed shape – def+undeformed-ok

**Problem 3.** In the Spanner under plane stress, find deformed shape and determine the maximum stress distribution.  $E = 2 \times 10^5 \text{ N/mm}^2$ ,  $t = 3 \text{ mm}$ , Poisson's ratio = 0.27, Analysis assumption – plane stress with thickness is used.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – Solid – Quad 8 node – 82 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

**Real constants** – Add – ok – real constant set no – 1 – Thickness – 3 – ok.

**Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX –  $2e5$  – PRXY – 0.27 – ok – close.

### Step 4: Preprocessor

**Modeling** – Create – Areas – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 120, 0, 13 – ok.

Create – Areas – Circle – solid circle – X, Y, radius – 130, 6.5, 15 – ok.

Operate – Booleans – Add – Areas – Pick all – ok.

Create – Areas – Rectangle – By 2 Corners – X, Y, Width, Height – 125, 14, 25, -15 – ok. Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (Spanner body) – apply – pick area which is to be deleted (Rectangle) – ok.

**Ansys Utility Menu** – WorkPlane – Offset WP by Increments – X,Y,Z offsets – 55,0,0 – Degrees – XY,YZ,ZX – 0,0,90 – ok.

Operate – Booleans – Divide – Areas by Workplane – pick the area – ok.

**Meshing** – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

### **Step 5: Preprocessor**

Loads – Define loads – apply – Structural – Displacement – on Areas – select the Spanner end – apply – DOFs to be constrained – ALL DOF – ok.

**Ansys Utility Menu** – Select – Entities – Lines – by Num/Pick – Select all – apply (select the line where load is applied – ok – Nodes – Attached to – lines, all – select all – ok.

Plot – nodes (only nodes attached to the lines will be displayed).

**Preprocessor** – Loads – Define loads – apply – Structural – Force/Moment – on Nodes – select box – drag all the nodes (note the no of nodes) – apply – direction of For/Mom – FY – Force/Moment value – -200/no of nodes – ok.

### **Step 6: Solution**

Solve – current LS – ok (Solution is done is displayed) – close.

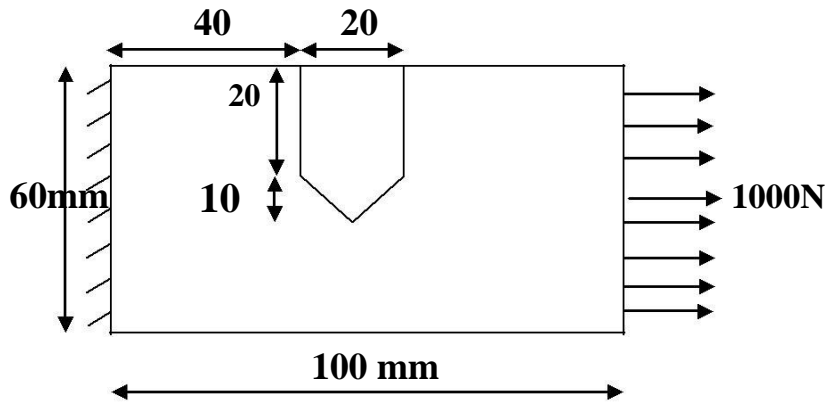
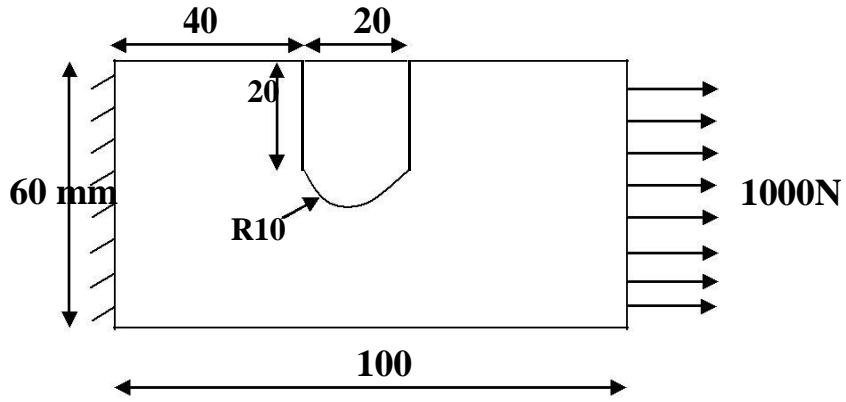
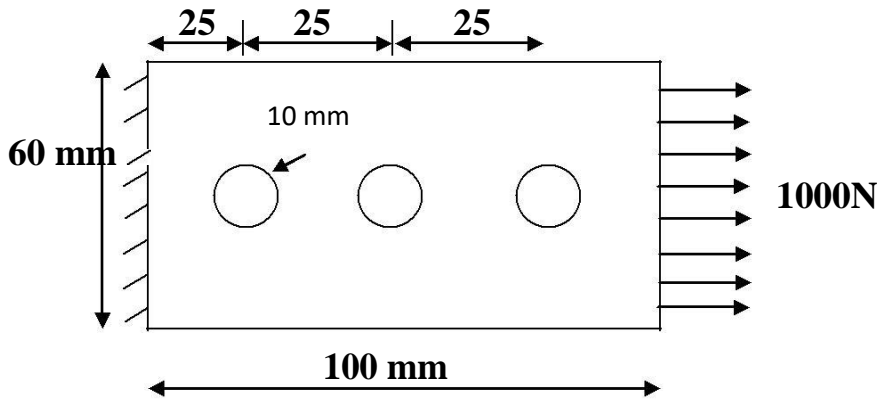
### **Step 7: General Post Processor**

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

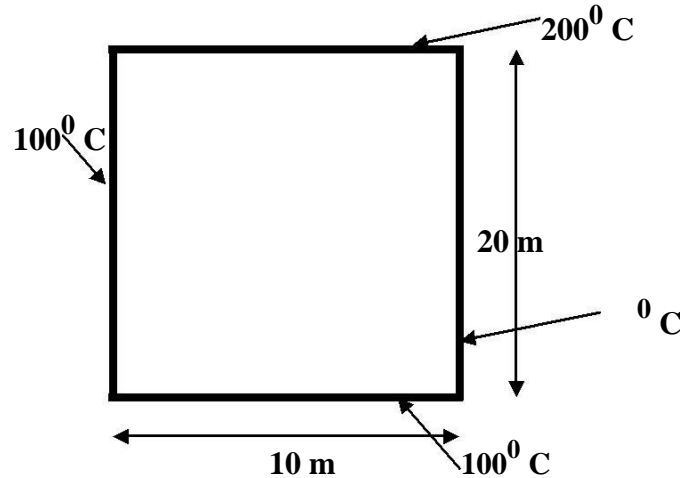
**Step 8: PlotCtrls** – Animate – Deformed shape – def+undeformed-ok

**ASSIGNMENTS**



## THERMAL ANALYSIS

**Problem 1.** Solve the 2-D heat conduction problem for the temperature distribution within the rectangular plate. Thermal conductivity of the plate,  $K_{XX}=401$  W/(m-K).



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – THERMAL - ok

### Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – Solid – Quad 4 node – 55 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

**Material Properties** – material models – Thermal – Conductivity – Isotropic –  $K_{XX}$  – 401.

### Step 4: Preprocessor

**Modeling** – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 10, 0, 20 – ok.

**Meshing** – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

### Step 5: Preprocessor

**Loads** – Define loads – apply – Thermal – Temperature – on Lines – select 100°C lines – apply – DOFs to be constrained – TEMP – Temp value – 100°C – ok.

**Loads** – Define loads – apply – Thermal – Temperature – on Lines – select 100°C lines – apply – DOFs to be constrained – TEMP – Temp value – 200°C – ok.

### Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.



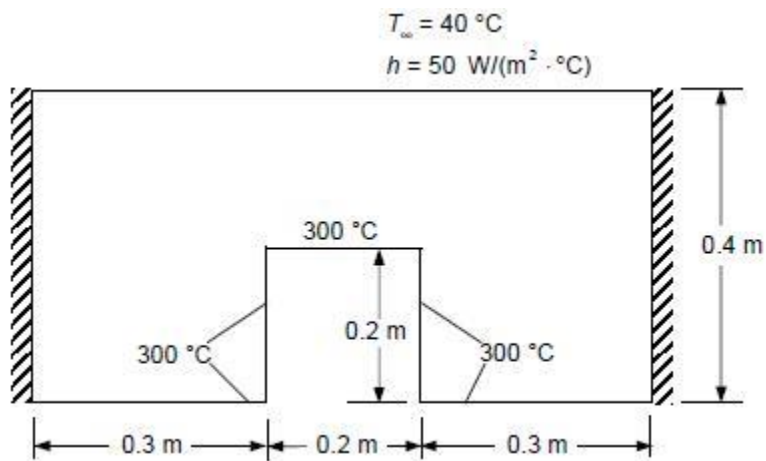
### Step 7: General Post Processor

Plot results – contour plot – Nodal solu – DOF solu – Nodal Temperature — ok

Step 8: PlotCtrls – Animate – Deformed results – DOF Solution – Temperature – ok.

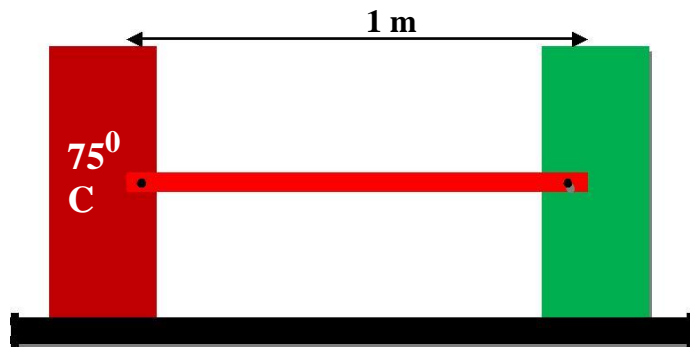
### ASSIGNMENT

**Problem:** For the two-dimensional stainless-steel shown below, determine the temperature distribution. The left and right sides are insulated. The top surface is subjected to heat transfer by convection. The bottom and internal portion surfaces are maintained at 300 °C. Thermal conductivity of stainless steel = 16 W/m.K)



**Problem 2.** A steel link, with no internal stresses, is pinned between two solid structures at a reference temperature of  $0^{\circ}\text{C}$  (273 K). One of the solid structures is heated to a temperature of  $75^{\circ}\text{C}$  (348 K). As heat is transferred from the solid structure into the link (Area = 2cm x 2cm), the link will attempt to expand. However, since it is pinned this cannot occur and as such, stress is created in the link. A steady-state solution of the resulting stress will be found to simplify the analysis.

Loads will not be applied to the link, only a temperature change of 75 degrees Celsius. The link is steel with a modulus of elasticity of 200 GPa, a thermal conductivity of 60.5 W/(m-K) and a thermal expansion coefficient of  $12\text{e-}6$  /K.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok

File – change job name – enter new job name – TSA – ok

File – change title – enter new title – yyy – ok

### Step 2: Ansys Main Menu – Preferences

select – THERMAL - ok

### Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – Link – 3D conduction 33 – ok – close.

**Real Constants** – Add/Edit/Delete – Add – ok – AREA –  $4\text{e-}4$  – ok – close.

**Material Properties** – material models – Thermal – Conductivity – Isotropic – KXX – 60.5.

### Step 4: Preprocessor

**Modeling** – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations – 1,0 – ok (Keypoints created).

Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.

**Meshing** – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

### **Step 5: Preprocessor**

Physics – Environment – Write – enter the TITLE – Thermal – ok. Physics – Environment – Clear – ok.

**Preprocessor** – Element Type – Switch Elem Type – Thermal to Struc – ok.

**Material Properties** – Material models – Structural – Linear – Elastic – Isotropic – EX – 200e9 – PRXY – 0.3. Material Models – Structural – Thermal Expansion Coef – Secant Coefficient – Isotropic – ALPX – 12e-6 – ok.

Physics – Environment – Write – enter the TITLE – Struct – ok.

### **Step 6: Solution**

Solution – Analysis Type – New Analysis – Static – ok.

Solution – Physics – Environment – Read – Thermal – ok (If the Physics option is not available under Solution, click **Unabridged Menu** at the bottom of the Solution menu. This should make it visible).

Solution – Define Loads – Apply – Thermal – Temperature – On Keypoints – pick first keypoint – apply – TEMP – value – 348 – ok.

Solve – current LS – ok (Solution is done is displayed) – close.

Main Menu > Finish

### **Step 7: Solution**

Solution – Physics – Environment – Read – Struct – ok.

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (1 – ALLDOF, 2- UX).

Solution – Define Loads – Apply – Structural – Temperature – from Therm Analy – enter Name of result file (or Browse...) – TSA.rth – ok.

**Step 8: Preprocessor** – Loads – Define Loads – Settings – Reference Temp – 273 – ok.

### **Step 9: Solution**

Solve – current LS – ok (Solution is done is displayed) – close.

### **Step 10: General Post Processor**

Element table – Define table – Add – ‘Results data item’ – By Sequence num – LS – LS1 – ok.

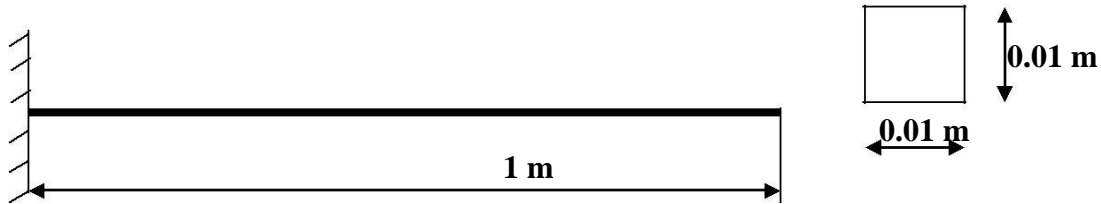
Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

Element table – List element table – LS1 – ok.

(Note the stress in each element: -0.180e9 Pa, or 180 MPa in compression as expected.)

## DYNAMIC ANALYSIS

**Modal Analysis of Cantilever beam** for natural frequency determination. Modulus of elasticity = 200GPa, Density = 7800 Kg/m<sup>3</sup>



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

**Real constants** – Add – ok – real constant set no – 1 – c/s area – 0.01\*0.01 moment of inertia – 0.01\*0.01\*\*3/12 – total beam height – 0.01 – ok.

**Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 200e9 – PRXY – 0.27 – Density – 7800 – ok – close.

### Step 4: Preprocessor

**Modeling** – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations – 1,0 – ok (Keypoints created).

Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.

**Meshing** – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

### Step 5: Solution

Solution – Analysis Type – New Analysis – Modal – ok.

Solution – Analysis Type – Subspace – Analysis options – no of modes to extract – 5 – no of modes to expand – 5 – ok – (use default values) – ok.

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick first keypoint – apply – DOFs to be constrained – ALL DOF – ok.

Solve – current LS – ok (Solution is done is displayed) – close.

## Step 7: General Post Processor

Result Summary

## Step 8: General Post Processor

Read Results – First Set

Plot Results – Deformed Shape – def+undeformed – ok.

**PlotCtrls** – Animate – Deformed shape – def+undeformed-ok.

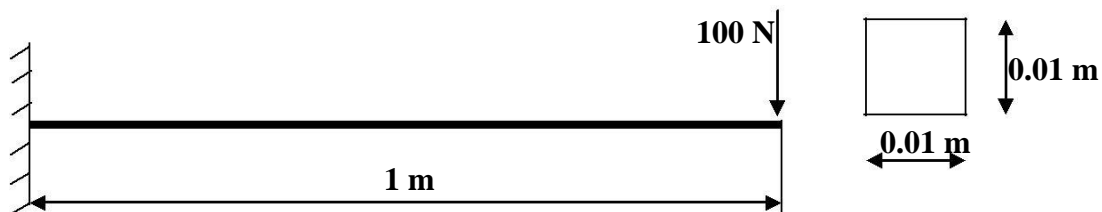
Read Results – Next Set

Plot Results – Deformed Shape – def+undeformed – ok.

**PlotCtrls** – Animate – Deformed shape – def+undeformed-ok.

## Fixed- fixed beam subjected to forcing function

Conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1 - 100 Hz. Modulus of elasticity = 200GPa, Poisson's ratio = 0.3, Density = 7800 Kg/m<sup>3</sup>.



## Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

## Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

## Step 3: Preprocessor

**Element type** – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok – close.

**Real constants** – Add – ok – real constant set no – 1 – c/s area – 0.01\*0.01 moment of inertia – 0.01\*0.01\*\*3/12 – total beam height – 0.01 – ok.

**Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 200e9 – PRXY – 0.3 – Density – 7800 – ok.

## Step 4: Preprocessor

**Modeling** – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations – 1,0 – ok (Keypoints created).

Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.

**Meshing** – Size Cntrl – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

### **Step 5: Solution**

Solution – Analysis Type – New Analysis – Harmonic – ok.

Solution – Analysis Type – Subspace – Analysis options – Solution method – FULL – DOF printout format – Real + imaginary – ok – (use default values) – ok.

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick first keypoint – apply – DOFs to be constrained – ALL DOF – ok.

Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – Pick second node – apply – direction of force/mom – FY – Real part of force/mom – 100 – imaginary part of force/mom – 0 – ok.

Solution – Load Step Opts – Time/Frequency – Freq and Substps... – Harmonic frequency range – 0 – 100 – number of substeps – 100 – B.C – stepped – ok.

Solve – current LS – ok (Solution is done is displayed) – close.

### **Step 6: TimeHist Postpro**

Select 'Add' (the green '+' sign in the upper left corner) from this window – Nodal solution - DOF solution – Y component of Displacement – ok. Graphically select node 2 – ok.

Select 'List Data' (3 buttons to the left of 'Add') from the window.

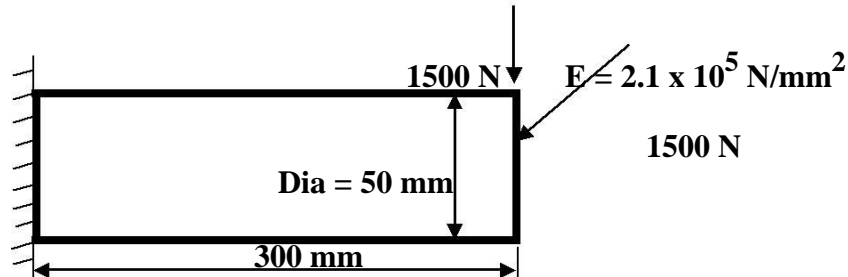
'Time History Variables' window click the 'Plot' button, (2 buttons to the left of 'Add')

**Step 7:** Utility Menu – PlotCtrls – Style – Graphs – Modify Axis – Y axis scale – Logarithmic – ok. Utility Menu – Plot – Replot.

This is the response at node 2 for the cyclic load applied at this node from 0 - 100 Hz.

## Bar subjected to forcing function

Consider the bar shown in figure below. Conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the bar. The frequency of the load will be varied from 1 - 100 Hz. Modulus of elasticity = 200GPa, Poisson's ratio = 0.3, Density = 7800 Kg/m<sup>3</sup>.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

### Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area –  $22/7 * 50^2 * 2/4$  – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX –  $2.1e5$  – PRXY – 0.27 – Density –  $7.8e-6$  – ok – close.

### Step 4: Preprocessor

Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations – 300,0 – ok (Keypoints created).

Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.

Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

### Step 5: Solution

Solution – Analysis Type – New Analysis – Harmonic – ok.

Solution – Analysis Type – Subspace – Analysis options – Solution method – FULL – DOF printout format – Real + imaginary – ok – (use default values) – ok.

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick first keypoint – apply – DOFs to be constrained – ALL DOF – ok.

Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – Pick second node – apply – direction of force/mom – FY – Real part of force/mom – 1500 – imaginary part of force/mom – 0 – ok.

Solution – Load Step Opts – Time/Frequency – Freq and Substps... – Harmonic frequency range – 0 – 100 – number of substeps – 100 – B.C – stepped – ok.

Solve – current LS – ok (Solution is done is displayed) – close.

### **Step 6: TimeHist Postpro**

Select 'Add' (the green '+' sign in the upper left corner) from this window – Nodal solution - DOF solution – Y component of Displacement – ok. Graphically select node 2 – ok.

Select 'List Data' (3 buttons to the left of 'Add') from the window.

'Time History Variables' window click the 'Plot' button, (2 buttons to the left of 'Add')

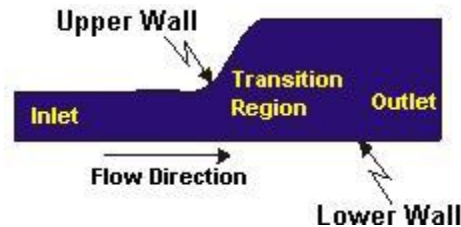
**Step 7:** Utility Menu – PlotCtrls – Style – Graphs – Modify Axis – Y axis scale – Logarithmic – ok. Utility Menu – Plot – Replot.

This is the response at node 2 for the cyclic load applied at this node from 0 - 100 Hz.



## Laminar Flow Analyses in a 2-D Duct

Inlet length = 4 in, Inlet height = 1 in, Transition length = 2 in, outlet height = 2.5 in, outlet length = 4 in, Air Density =  $1.21 \times 10^{-7}$  lb<sub>f</sub>-s<sup>2</sup>/in<sup>4</sup>, Air Viscosity =  $2.642 \times 10^{-9}$  lb<sub>f</sub>-s/in<sup>2</sup>, Inlet Velocity = 1 in/sec. Reynolds number of 90.



### Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok  
File – change job name – enter new job name – FFA – ok  
File – change title – enter new title – yyy – ok

### Step 2: Ansys Main Menu – Preferences

select – FLOTRAN CFD – ok.

### Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – FLOTRAN CFD – 2D FLOTRAN 141 – ok – close.

### Step 4: Preprocessor

Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 4, 0, 1 – apply –  
Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 6, 10, 0, 2.5 – ok.

Create – Lines – Lines – Tan to 2 lines – Pick upper line of left rectangle – ok – Pick the tangency end of the first line (upper right corner) – ok – Pick upper line of right rectangle – ok –  
Pick the tangency end of the first line (upper left corner) – ok – cancel.

Create – Area – Arbitrary – Through KPs – Pick 4 corners in counterclockwise order – ok.

### Utility Menu – Plot – Lines

### Step 5: Preprocessor

Meshing – Mesh Tool – Lines – set – (pick lines in flow direction along the inlet) – apply – enter 15 as No. of element divisions – enter -2 as Spacing ratio – apply – Pick the top and bottom lines of center area – apply – enter 12 as No. of element divisions – enter 1 as Spacing ratio – apply –  
Pick the top and bottom lines of outer region – apply – enter 15 as No. of element divisions – enter 3 as Spacing ratio – ok.

Meshing – Mesh Tool – Lines – Flip – pick the upper line of Outer region – ok.

Meshing – Mesh Tool – Lines – set – Pick the 4 transverse direction lines (vertical lines) – ok – enter 10 as No. of element divisions – enter -2 as Spacing ratio – ok.

Meshing – **Mesh Tool** – Mesh Areas – Quad – Mapped – Mesh – pick all – ok.

### Utility Menu – Plot – Lines

#### Step 6: Preprocessor

Main Menu – Preprocessor – Loads – Define Loads – Apply – Fluid/CFD – Velocity – On Lines – Pick the inlet line (the vertical line at the far left) – ok – VX – 1(value) – VY – 0 – ok.

Main Menu – Preprocessor – Loads – Define Loads – Apply – Fluid/CFD – Velocity – On Lines – Pick the six lines on the top and bottom – ok – VX – 0(value) – VY – 0 – ok.

Main Menu – Preprocessor – Loads – Define Loads – Apply – Fluid/CFD – Pressure DOF – On Lines – Pick the outlet line (vertical line on the far right) – ok – Pressure value – 0 – ok.

#### Step 7: Solution

**FLOTRAN Set Up** – Fluid Properties – Density – AIR IN – Viscosity AIR IN – ok – (Use default values) – ok.

**FLOTRAN Set Up** – Execution Ctrl – Global Iterations – 40 – ok.

**FLOTRAN Set Up** – Flow Environment – Ref Conditions – reference pressure – 14.7 – nominal, stagnation, and reference temperatures – 70 – temperature offset from absolute – 460 – ok.

**RUN FLOTRAN** – solution done – close.

#### Step 8: General Postproc

Read Results – last set

#### Step 9: General Postproc

Plot results – vector plot – predefined – DOF Soln – Velocity V – ok.

#### Step 10: General Postproc

Plot results – contour plot – nodal soln – other FLOTRAN quantities – total stagnation pressure – ok.

#### Step 11: General Postproc

Plot Results – Flow Trace – Defi Trace Pt – Pick three or four points around the inlet region and two or three points in the recirculation region (along the upper wall of the transition region) – ok.

**Utility Menu – PlotCtrls – Animate – Particle Flow** – DOF Solution – Velocity VX – ok.

Path Operations – Define Path – By Nodes – Pick the lowest and then the highest point of the outlet – ok – enter Path Name – V – ok – close.

Path Operations – Map onto Path – user label for Item – VELOCITY – DOF Solution – Velocity VX – ok.

Path Operations – Plot path Item – on Graph – VELOCITY – ok.