## CONTENTS

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

<table>
<thead>
<tr>
<th>Subject Code</th>
<th>IA Marks</th>
<th>No. of Practical Hrs/Week</th>
<th>Exam Hours</th>
<th>Total No. of Practical Hrs.</th>
<th>Exam Marks</th>
</tr>
</thead>
<tbody>
<tr>
<td>06MEL67</td>
<td>25</td>
<td>03</td>
<td>03</td>
<td>42</td>
<td>50</td>
</tr>
</tbody>
</table>

### 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. Fundaments 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
- **Total**  -  **50 Marks**
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:
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—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.

![Beam diagram]

20 kN
2 m
4 m

Step 1: Ansys Utility Menu


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor


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.


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**

**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.

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).


**ASSIGNMENT**

![Assignment Diagram](image-url)
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.

\[ \text{Beam Image} \]

**Step 1: Ansys Utility Menu**


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

select – STRUCTURAL - ok

**Step 3: Preprocessor**


**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.


**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**


**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.

**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).


**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.

![Beam Diagram](image)

**Step 1: Ansys Utility Menu**


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

select – STRUCTURAL - ok

**Step 3: Preprocessor**


**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.


**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.


**Step 5: Preprocessor**


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.

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).


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.

![Beam Diagram](image)

**Step 1: Ansys Utility Menu**


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

select – STRUCTURAL - ok

**Step 3: Preprocessor**


**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.


**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).


**Step 5: Preprocessor**


**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.

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).


**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 m * 0.3 m, Young’s modulus of 210 GPa, Poisson’s ratio 0.27.

Step 1: Ansys Utility Menu


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor


**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.


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 (forth node is created) – 7 (x value w.r.t first node) – ok (fifth node is created).


Step 5: Preprocessor


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).


**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


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor


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.


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 (forth node is created).


Step 5: Preprocessor


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.

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).


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


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.


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.

Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.
Step 7: General Post Processor

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.

![Stepped Bar Diagram](image)

**Step 1: Ansys Utility Menu**


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

select – STRUCTURAL - ok

**Step 3: Preprocessor**

**Element type** – Add>Edit>Delete – Add – Link – 2D spar 1 – 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.

**Step 6: Solution**
Solve – current LS – ok (Solution is done is displayed) – close.

**Step 7: General Post Processor**

**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.

![Diagram of a tapered bar](Image)

$E = 2 \times 10^5 \text{ N/mm}^2$, 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


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor


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


Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor

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).

**TRUSSES**

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

\[ \text{Step 1: Ansys Utility Menu} \]


\[ \text{Step 2: Ansys Main Menu – Preferences} \]

select – STRUCTURAL - ok

\[ \text{Step 3: Preprocessor} \]

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.


\[ \text{Step 4: Preprocessor} \]

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 (forth node is created).

Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 1 – ok

**Step 5: Preprocessor**


**Step 6: Solution**

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

**Step 7: General Post Processor**


**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

**ASSIGNMENT**

Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm2).
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.

<table>
<thead>
<tr>
<th>Member</th>
<th>C/s area mm²</th>
<th>E N/mm²</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>200</td>
<td>2 x 10⁵</td>
</tr>
<tr>
<td>2</td>
<td>200</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>100</td>
<td></td>
</tr>
</tbody>
</table>

Step 1: Ansys Utility Menu


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – 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 (forth node is created).
Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 1 – ok
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**
– DOFs to be constrained – All DOF – ok.
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**

**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).

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

**ASSIGNMENT**
Note: Cross-sectional area of truss members = 3.0E-4 m2; Modulus of Elasticity = 2.07E11
N/m2. 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 = 210GPa, 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


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor


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

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.
– 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.

All DOF constrained

6 in

2 in

R 1 in

R 0.4 in

R 0.4

4 in

E = 30 x 10^6 psi,
Poisson’s ratio = 0.27,
t = 0.5 in,
Pressure load from 50 to 500 lb

Tapered load from 50 to 500 lb

Step 1: Ansys Utility Menu


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor


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

**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.


**Step 5: Preprocessor**


**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


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor


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.

**Step 5: Preprocessor**

**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).


**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

1. A structure with dimensions 60 mm x 100 mm, with three circular cutouts of diameter 10 mm, and 1000 N loads applied at the edges.

2. A structure with dimensions 60 mm x 100 mm, with a radius R10 cutout, 20 mm vertical cutout, and 1000 N loads applied at the edges.

3. A structure with dimensions 60 mm x 100 mm, with a radius R10 cutout, 20 mm vertical cutout, and 1000 N loads applied at the edges.
**THERMAL ANALYSIS**

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

![Diagram of a rectangular plate with temperature distribution](image)

**Step 1: Ansys Utility Menu**


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

select – THERMAL - ok

**Step 3: Preprocessor**


**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.


**Step 5: Preprocessor**

Loads – Define loads – apply – Thermal – Temperature – on Lines – select \( 100^\circ \text{C} \) lines – apply – DOFs to be constrained – TEMP – Temp value – \( 100^\circ \text{C} \) – ok.

Loads – Define loads – apply – Thermal – Temperature – on Lines – select \( 200^\circ \text{C} \) lines – apply – DOFs to be constrained – TEMP – Temp value – \( 200^\circ \text{C} \) – 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 $0^\circ C$ (273 K). One of the solid structures is heated to a temperature of $75^\circ C$ (348 K). As heat is transferred from the solid structure into the link (Area = 2cm × 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 12e-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**
- **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.
**Step 5: Preprocessor**
**Preprocessor** – Element Type – Switch Elem Type – Thermal to Struc – 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).
Solve – current LS – ok (Solution is done is displayed) – close.
Main Menu > Finish

**Step 7: Solution**
Solution – Define Loads – Apply – Structural – Temperature – from Therm Analy – enter Name of result file (or Browse…) – TSA.rth – ok.

**Step 9: Solution**
Solve – current LS – ok (Solution is done is displayed) – close.

**Step 10: General Post Processor**
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$^3$

Step 1: Ansys Utility Menu


Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor


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.


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.


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.


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/m³.

Step 1: Ansys Utility Menu

Step 2: Ansys Main Menu – Preferences
select – STRUCTURAL - ok

Step 3: Preprocessor
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.

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.

**Step 5: Solution**
Solution – Analysis Type – New Analysis – Harmonic – 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')


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$^3$.

![Diagram of bar](image)

**Step 1: Ansys Utility Menu**


**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.


**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.


**Step 5: Solution**

Solution – Analysis Type – New Analysis – Harmonic – 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')


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·s$^2$/ft$^4$, Air Viscosity = $2.642 \times 10^{-9}$ lb·s/ft$^2$, 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

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


Utility Menu – Plot – Lines

Step 6: Preprocessor

Step 7: Solution
RUN FLOTRAN – solution done – close.

Step 8: General Postproc
Read Results – last set

Step 9: General Postproc

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.

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