Concordia University
Department of Mechanical & Industrial Engineering

FINITE ELEMENT ANALYSIS

MECH 460

LABORATORY MANUAL

ANSYS

Course Instructor: Dr. R.Ganesan

Prepared by

A. Zabihollah

Approved by

Dr. R. Ganesan

Winter 2007

Table of Contents

1. INTRODUCTION

ANSYS is a powerful general purpose finite element modeling package to numerically solve a wide variety of mechanical, structural and non-structural problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems [1]. In general, a finite element solution may be classified into the following three stages. This is a general guideline that can be used for setting up any finite element analysis.

Preprocessing: defining the problem

The major steps in preprocessing are given below:

- Define key points/lines/areas/volumes
- Define element types and material/geometrical properties
- Mesh lines/areas/volumes as required

Solution: assigning loads, constraints and solving

In the solution level, the loading conditions such as point load or pressure and constraints or boundary conditions are specified and finally the resulting set of equations are solved.

Post processing: further processing and viewing of the results

This stage provides different tools to view the results including:

- Lists of nodal displacements
- Element forces and moments
- Deflection plots
- Stress contour diagrams

In this level our job as an engineer is to interpret the output results and verify their accuracy. Typically with increasing number of elements (using finer mesh), we should expect to more accurate results.

1.1 Starting up ANSYS:

To start ANSYS in Windows environment simply follow the path:
Start Menu >Programs >ANSYS 7.1 >ANSYS Classic
The first window you may see is the main window as shown in Figure 1.

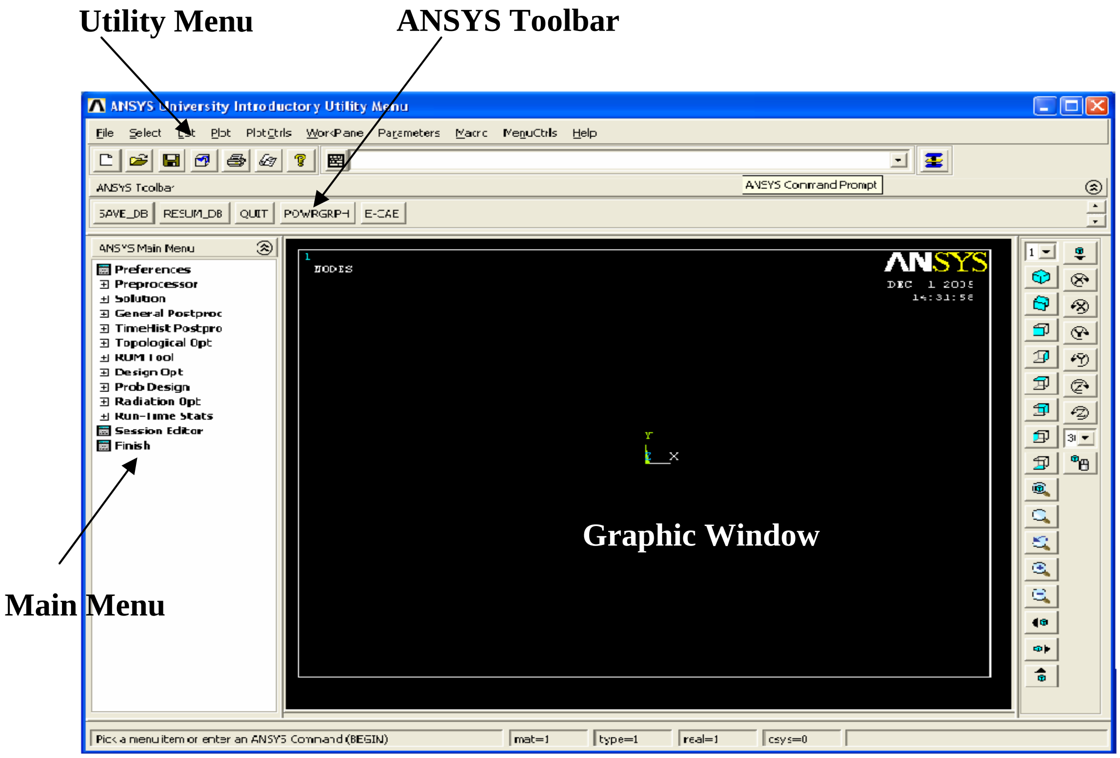


Figure 1: ANSYS main Window

1.2 ANSYS Interface

There are two methods to use ANSYS; Graphical Interface and Command File Coding. The graphical user interface or GUI follows the conventions of Windows based programs. This method is probably the best approach for new users. The command approach is used by professional users. It has the advantage that an entire analysis can be described in a small text file, typically in less than 50 lines of commands. This approach enables easy model modifications and minimal file space requirements. In this lab manual we mainly use the GUI. However, in some cases both the GUI and command code are used to show that how command code could be easy to use.

1.3 ANSYS Files

When run even a simple program by ANSYS, a large number of files are created. Each file has specific purposes. The most important files are described briefly as follows:

FILE.db: Database file (binary). This file stores the geometry, boundary conditions and any solutions.
FILE.dbb: Backup of the database file (binary).
FILE.err: Error file (text). Listing of all error and warning messages.
FILE.out: Output of all ANSYS operations (text). This is what normally scrolls in the output window during an ANSYS session.
FILE.log: Logfile or listing of ANSYS commands (text). Listing of all equivalent ANSYS command line commands used during the current session.

Note that if you started ANSYS without specifying a jobname, the name of all the files created will be `FILE.*`. If you specified a jobname, then all files will be saved by the jobname.

When you are using the GUI, then you only require to save the `.db` file. This file stores the geometry, boundary conditions and any solutions. When you start ANSYS you only need to “Resume from…” and call your saved file.db

1.4 Plotting of Figures

To Plot your created model or output results such as deformed shape, you may follow the following path.

Utility menu bar>PlotCtrls>Hard Copy>Graphics window>Monochrome>Reverse Video>Landscape>Save to>choose a name>OK.

2. STATIC ANALYSIS: TRUSS AND FRAME STRUCTURES

One-dimensional elements can be used in a variety of engineering applications. The simplest 1-D element is the link element which is a line with two nodes at the ends as shown in Figure 2. Degrees of freedom at each node may consider as displacements in length, width and even thickness directions depending upon the application. This element can be used as a truss, a link, a spring, etc. Note that in this element no bending is considered.

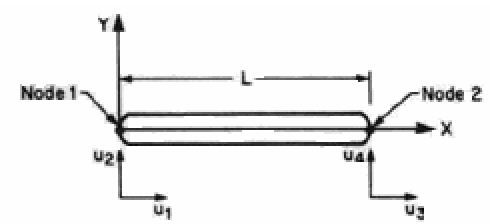


Figure 2: Link element

2.1 Truss Analysis

You may recall that a truss is a structural element that experiences loading only in the axial direction.

2-D Truss

As the first application of link element, consider a truss structure shown in Figure 3. Determine the nodal deflections, reaction forces, and stress for this structure. (E = 200GPa, A = 5000mm^2).

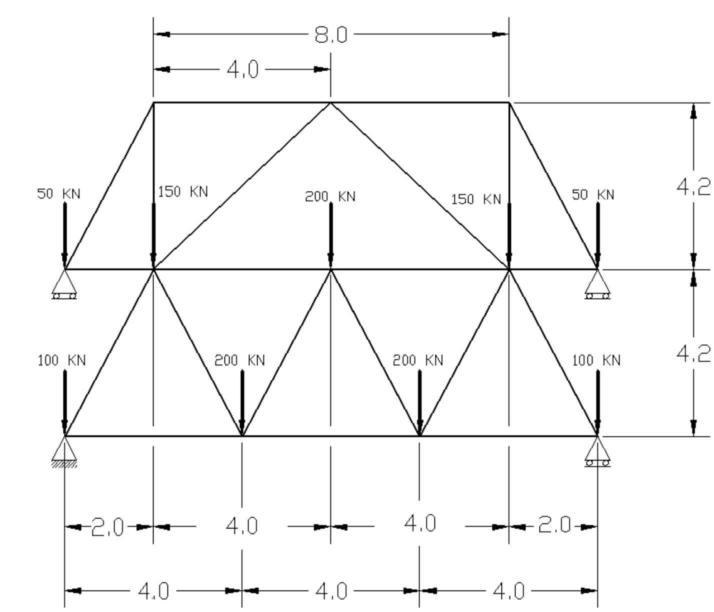


Figure 3: 2-D Truss Structure

Preprocessing: Defining the Problem

1. Select Title
In the Utility menu bar select File > Change Title:

2. Create geometry**: Nodes**
Preprocessor > Modeling > Create > Keypoints > In Active CS> Use the following entries

	Keypoints											
	1	2	3	4	5	6	7	8	9	10	11	12
X	0	4.0	8.0	12.0	12.0	10.0	6.0	2.0	0	2.0	6.0	10.0
Y	0	0	0	0	4.2	4.2	4.2	4.2	4.2	8.4	8.4	8.4

In the main menu select: **Preprocessor > Modeling > Create > Lines > Lines > In Active Coord**.

3. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**.
Use **LINK1** as element type

4. Define **Geometric Properties**
In the Preprocessor menu, select **Real Constants > Add/Edit/Delete**
Enter the area

5. Element Material Properties
In the 'Preprocessor' menu select **Material Props > Material Models**
Double click on **Structural > Linear > Elastic > Isotropic**

6. Meshing > Size Cntrls > ManualSize > Lines > All Lines
In the size 'NDIV' field, enter the desired number of divisions per line. For this example we want only 1 division per line, therefore, enter '1' and then click 'OK'. Note that we have not yet meshed the geometry; we have simply defined the element sizes.
In the 'Preprocessor' menu select **Meshing > Mesh > Lines** and click 'Pick All' in the 'Mesh Lines' Window. Now you should have the layout shown in Figure 4.

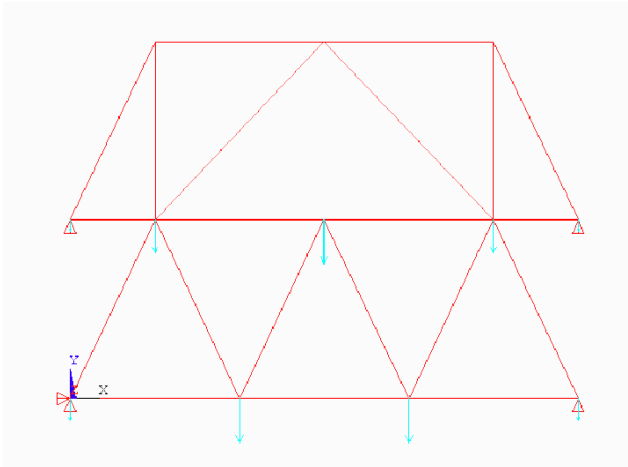


Figure 4: 2-D Truss

7. From the **Utility Menu** (top of screen) select **PlotCtrls** > **Numbering...**

8. Saving Your Work
On the **Utility Menu** select **File** > **Save as....**

Solution: Assigning Loads and Solving

9. From the **Solution** Menu, select **Analysis Type** > **New Analysis**>
Select **Static** solution

10. Apply Constraints
In the **Solution** menu, select **Define Loads** > **Apply** > **Structural** > **Displacement** > **On Nodes**

11. Apply Loads
Select **Define Loads** > **Apply** > **Structural** > **Force/Moment** > **on Nodes**.

12. Solving the System
In the 'Solution' menu select **Solve** > **Current LS**
- Reaction Forces

Post processing

From the Main Menu select **General Postproc** > **List Results** > **Reaction Solu**. (See Figure 5.a and 5.b)

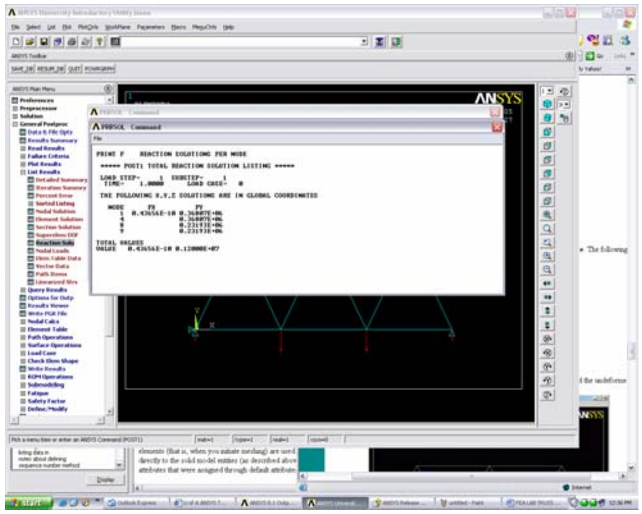


Figure 5.a: 2-D Truss –Reaction solutions

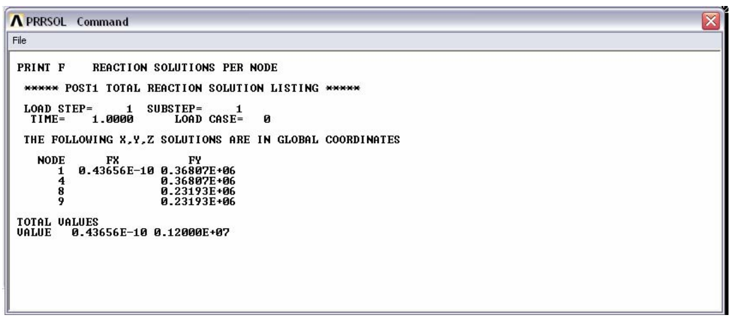


Figure 5.b: 2-D Truss –list results

Select **Plot Results > Deformed Shape** (see Figure 6)

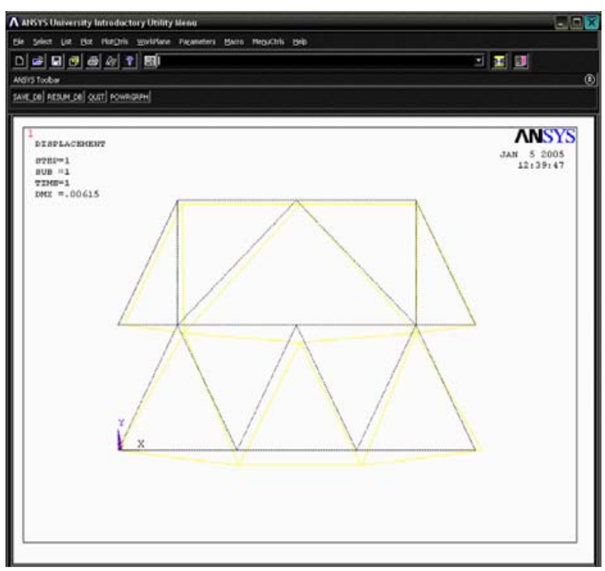


Figure 6: 2-D Truss –deformed shape

From the 'General Postproc' menu select **Plot results > Contour Plot > Nodal Solution** (see Figure 7)

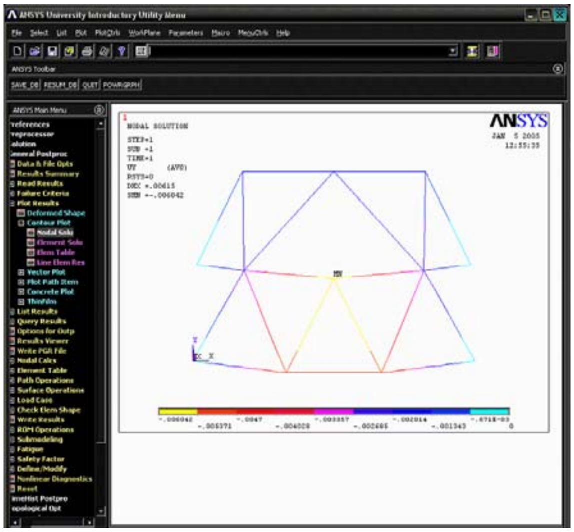


Figure 7: 2-D Truss –nodal solutions

2.2 3-D Truss Structure

Consider the 3-D truss structure shown in Figure 8. Cross sectional area of all the links are $A=1.56e\text{-}3m^2$. Material is Aluminum E=75GPa. The structure is fixed in x, y and z direction at the bottom three corner. The loading is as shown in the figure. (a) Determine deflection at each joint. (b) Determine stress in each member. (c) Determine reaction forces at the base [2].

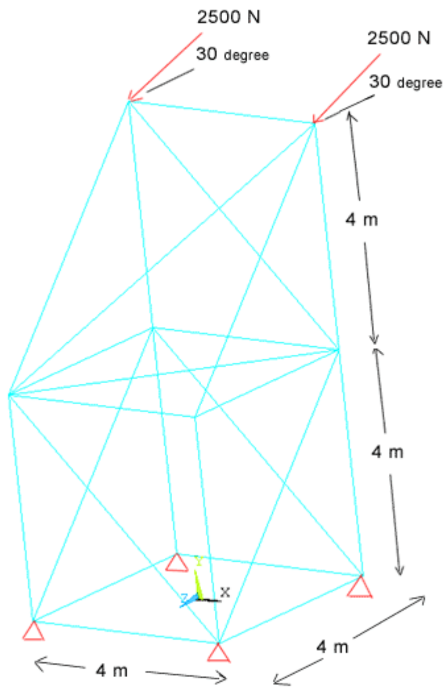


Figure 8: 3-D Truss structure

Preprocessing: Defining the Problem

1. Create key points: **Preprocessor>Modeling>Create>Keypoints>In active CS>** enter the coordinate of key points
To see the structure in 3-D: **Utility Menu>PlotCntrls>Pan Zoom Rotate**

2. Create lines: **Preprocessor>Modeling>Create>Lines>Lines>In Active Coord**.
Create lines between the key points.

3. Choose materials properties: **Preprocessor>Material Props>Material Models**.
Structural>Linear>Elastic>Isotropic.

4. Select element type: **Preprocessor>Element Type>Add/Edit/Delete...**

5. Mesh the structure: **Preprocessor>Meshing>Size Controls>Manual Size>Lines>All Lines**.
Preprocessor>Meshing>Mesh>Lines.

Solution

6. Apply boundary conditions: **Preprocessor>Loads>Define**
Apply loading: **Loads>Apply>Structural>Displacement>On Keypoints**
Preprocessor>Loads>Define Loads>Apply>Structural>Forces/Moment>On Nodes.
Solve the problem: **Main Menu>Solution>Analysis Type>New Analysis**.

7. Solution>Solve>Current LS.

Post processing

8. General Postprocessing>List Results>Nodal Solution.

9. General Postprocessing>List Results>Element Solution.

10. General Postprocessing>Plot Results>Contour Plot>Element Solution.

2.3 Beam Analysis

Beams are the most common type of structural component, particularly in Civil and Mechanical Engineering. A beam is a bar-like structural member as shown in Figure 9 whose primary function is to support transverse loading and carry it to the supports.

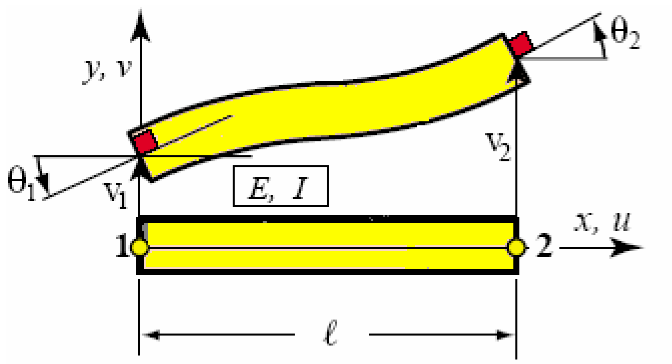


Figure 9: Beam element

A beam resists transverse loads mainly through bending action. Bending produces compressive longitudinal stresses in one side of the beam and tensile stresses in the other. The two regions are separated by a neutral surface of zero stress. The combination of tensile and compressive stresses produces an internal bending moment. This moment is the primary mechanism that transports loads to the supports.

Analysis of beam structures is very similar to the procedure applied for truss structures except:

Element Type > Add/Edit/Delete> Use **BEAM 3** as element type
In the Preprocessor menu, select **Real Constants > Add/Edit/Delete>** enter moment of inertia I and Height, and cross sectional area A.

2.4 Lab Assignment 1

1. Determine the nodal deflections, reaction forces, and stress for the truss system shown in Figure 10 (E = 200GPa, A = 3500mm^2).

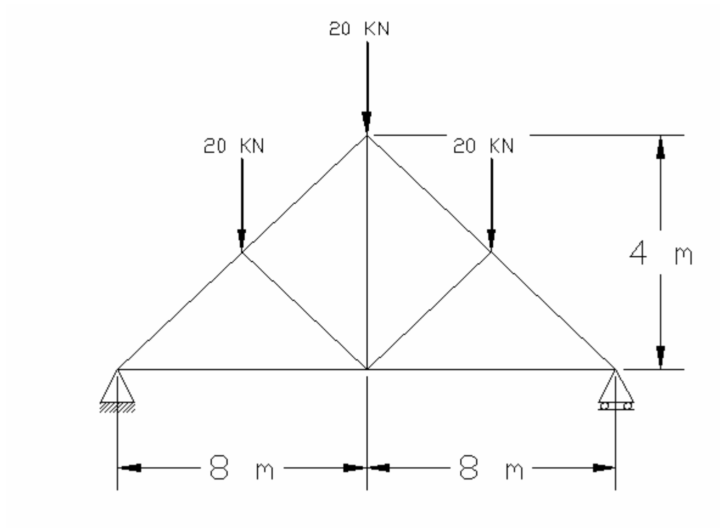


Figure 10: Truss structure- assignment 1

2. Determine the nodal deflections, reaction forces, and stress for the truss system shown in Figure 11 (E = 200GPa, A = 3500mm^2).

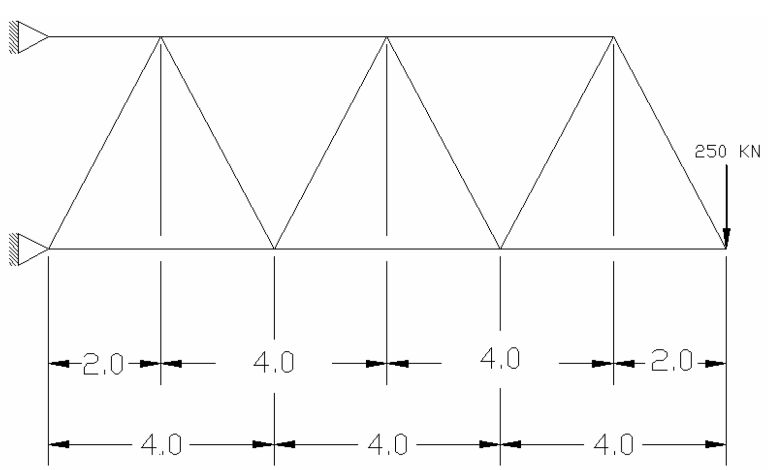


Figure 11: Truss structure- assignment 2

3. TWO-DIMENSIONAL PROBLEMS

Plate element

Two dimensional elements are reduced from 3D element by neglecting stress through the thickness direction (plane stress). The simplest plane element is a three node triangular element shown in Figure 12. In this element each node has two degree of freedom.

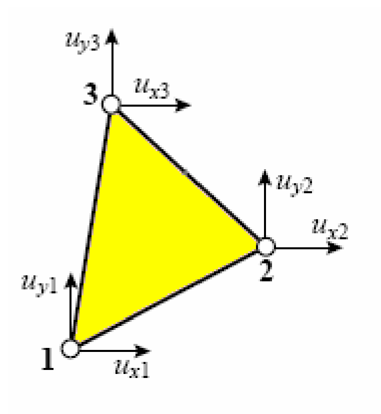


Figure 12: Plane element

3.1 2-D Structure with Different Loadings

Determine the nodal deflections, and von Mises stress for the structure shown in Figure 13. E = 75GPa, Thickness = 0.01m and Poisson Ratio is 0.2.

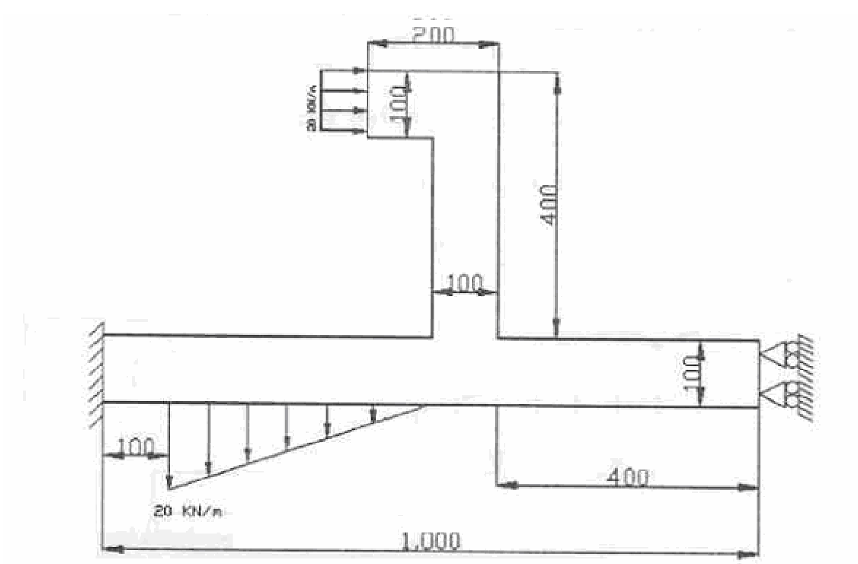


Figure 13: Plate structure of with different loadings

<u>Preprocessing: Defining the Problem</u>

1. Select Title

In the Utility menu bar select File > Change Title: Plate 1

2. Create geometry**: Area**
Preprocessor > Modeling > Create > Areas > Rectangle>By Dimensions> Use the following entries

Rectangle	X1	X2	Y1	Y2
1	0	0.1	0	0.1
2	0.1	0.5	0	0.1
3	0.5	1	0	0.1
4	0.5	0.6	0.1	0.5
5	0.4	0.5	0.4	0.5

Assemble the parts.
Preprocessor > Modeling > Operate> Booleans> Glue > Areas

3. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**.
Use **Plane42** as element type select option plane str w/thk

4. Define **Geometric Properties**
In the Preprocessor menu, select **Real Constants > Add/Edit/Delete**
Enter the thickness

5. Element Material Properties
In the 'Preprocessor' menu select **Material Props > Material Models**
Double click on **Structural > Linear > Elastic > Isotropic**

6. Meshing > Meshtools > Pick All
7. From the **Utility Menu** (top of screen) select **PlotCtrls > Numbering...**

8. Saving Your Work
On the **Utility Menu** select **File > Save as...**.

Solution: Assigning Loads and Solving

9. From the **Solution** Menu, select **Analysis Type > New Analysis**.
Select **Static** solution

10. Apply Constraints
In the **Solution** menu, select **Define Loads > Apply > Structural > Displacement > On Nodes**

11. Apply Loads
Select **Define Loads > Apply > Structural > Pressure > on Lines**.

12. Solving the System

In the 'Solution' menu select **Solve > Current LS**

- Select **Plot Results > Deformed Shape** (see Figure 14)

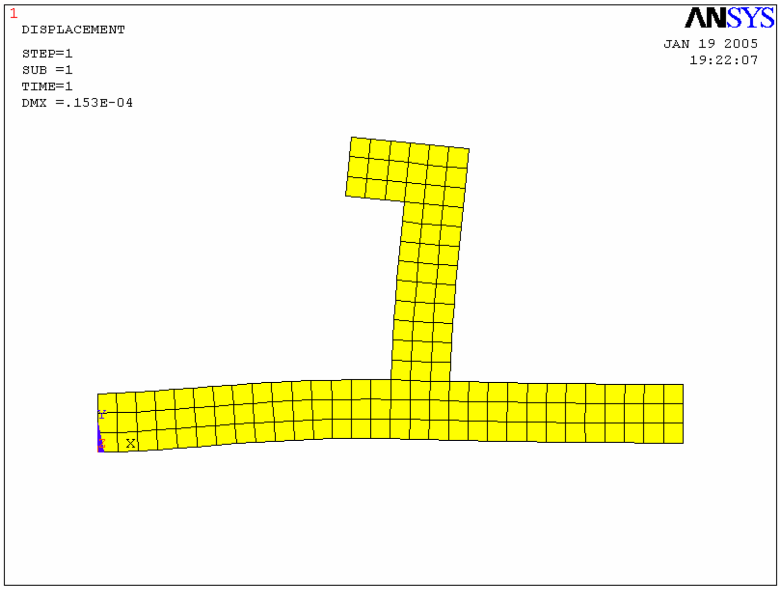


Figure 14: Example 3.1-deformed shape

From the Main Menu select **General Postproc > Plot Results> Contour Solu > Stress>von Mises**. (see Figure 15)

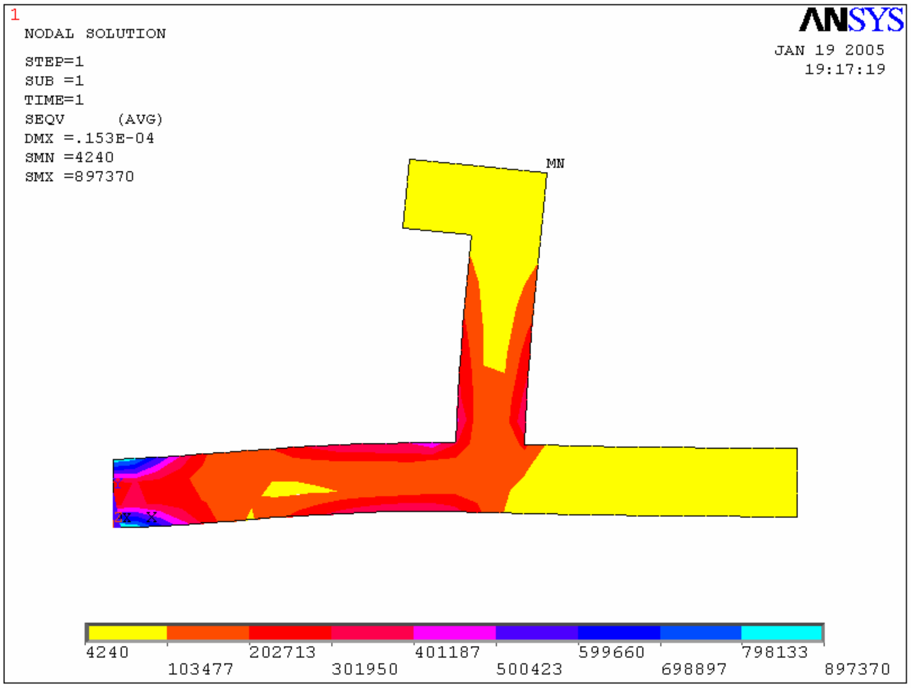


Figure 15: Example 3.1-Stress results

3.2 2-D Structure with Different Materials and Cross Sections
Determine the nodal deflections, and von Mises stress for the beam shown in Figure 16

E (CU) = 120GPa, GXY (CU)=44GPa ; E (AL) = 73GPa, GXY (AL)=26GPa

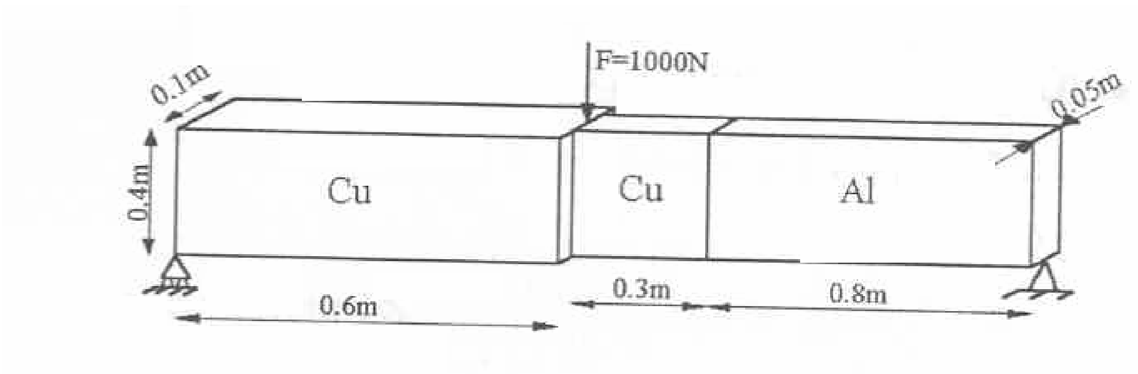


Figure 16: 2D structures with different materials

Preprocessing: Defining the Problem

1. Title
In the Utility menu bar select File > Change Title: Beam 2

2. Create geometry**: Area**
Preprocessor > Modeling > Create > Areas > Rectangle>By Dimensions

Use the following entries

Rectangle	X1	X2	Y1	Y2
1	0	0.6	0	0.4
2	0.6	0.9	0	0.4
3	0.9	1.7	0	0.4

3. Preprocessor > Modeling > Operate> Booleans> Glue > Areas

4. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**.
Use **Plane 82** as element type, select option plane str w/thk

5. Define **Geometric Properties**
In the Preprocessor menu, select **Real Constants > Add/Edit/Delete**
Enter the thickness of CU > Apply > Enter the thickness of AL>OK

6. Element Material Properties
In the 'Preprocessor' menu select **Material Props > Material Models**

Double click on **Structural** > **Linear** > **Elastic** > **Isotropic**
Enter AL properties
Add new models
Enter CU properties

7. Preprocessor > > Operate> Booleans> Glue > Areas

8. Meshing > Mesh Attribute > Pick Area>

9. Meshing > Meshtools (Smart Size=4) > Mesh> Pick All

10. From the **Utility Menu** (top of screen) select **PlotCtrls > Numbering...**

11. Saving Your Work
On the **Utility Menu** select **File > Save as....**

Solution: Assigning Loads and Solving

12. From the **Solution** Menu, select **Analysis Type** > **New Analysis**.
Select **Static** solution

13. Apply Constraints
In the **Solution** menu, select **Define Loads > Apply > Structural > Displacement > On Nodes**

14. Apply Loads
Select **Define Loads > Apply > Structural > Pressure > on Lines**.

15. Solving the System
In the 'Solution' menu select **Solve > Current LS**
Select **Plot Results > Deformed Shape** (See Figure 17)

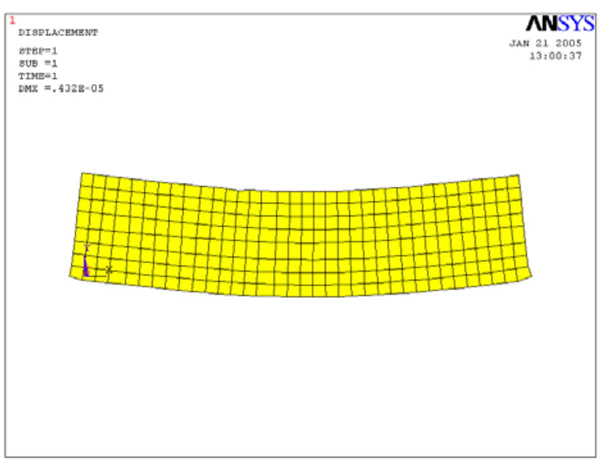


Figure 17: 2-D structure with different materials_ Deformed shape

From the Main Menu select **General Postproc** > **Plot Results**> **Contour Solu** > **Stress>von Mises** (See Figure 18)

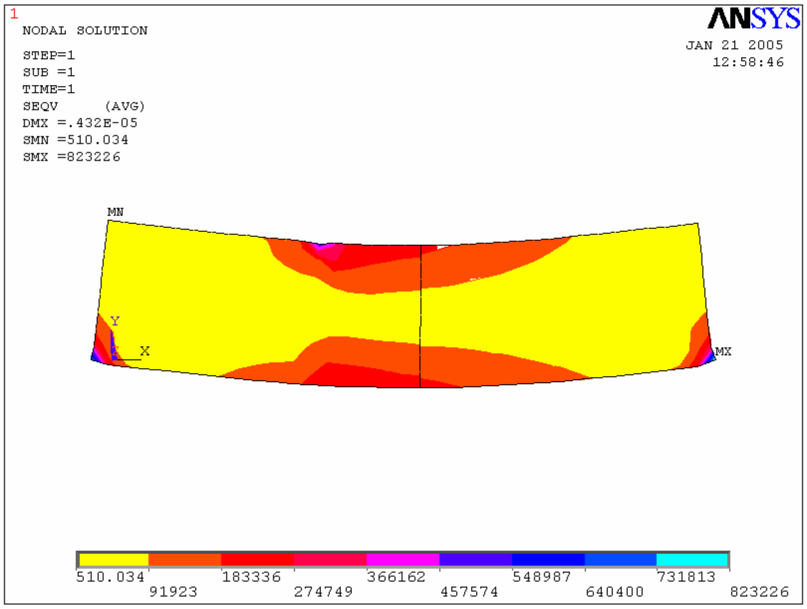


Figure 18: 2-D structure with different materials _stress results

3.3 Plate with Hole

Determine the stress concentration factor in the plate shown in Figure 19 (a) E x= 207GPa, GXY =77GPa

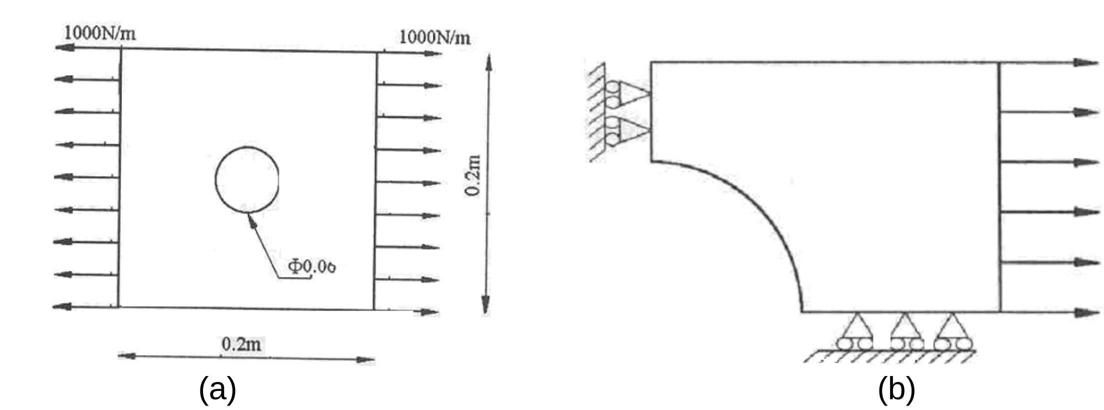


Figure 19: Plate with a hole

Since the shape is symmetric we may consider only a half of the shape for the analysis as shown in Fig.19 (b)

Preprocessing**:** Defining the Problem

1. Title

In the Utility menu bar select File > Change Title: Plate 2

2. Create geometry**: Area**

Preprocessor > Modeling > Create > Areas > Rectangle>By Dimensions
X1=0 X2=0.05 Y1=0, Y2=0.05
Preprocessor > Modeling > Create > Areas >Circle > Solid Circle
X, Y =0, 0, R=0.025
Preprocessor > Modeling > Operate> Subtract> Areas

3. Meshing > Meshtools (Smart Size=4) > Mesh> Pick All

4. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**.
Use **Plane2** as element type select option plane str w/thk

5. Define **Geometric Properties**
In the Preprocessor menu, select **Real Constants > Add/Edit/Delete**

6. Element Material Properties
In the 'Preprocessor' menu select **Material Props > Material Models**
Double click on **Structural > Linear > Elastic > Isotropic**
Enter Material properties

7. Saving Your Work
On the **Utility Menu** select **File > Save as...**.

<u>Solution:</u> Assigning Loads and Solving

8. From the **Solution** Menu, select **Analysis Type > New Analysis**.
Select **Static** solution

9. Apply Constraints
In the **Solution** menu, select **Define Loads > Apply > Structural > Displacement > Symmetric B.C > On Line**

10. Apply Loads
Select **Define Loads > Apply > Structural > Pressure > on Line**.
Enter -1000

11. Solving the System
In the 'Solution' menu select **Solve > Current LS**

12. General Postproc> Plot results>Nodal Solution> von Mises (See Figure 20).

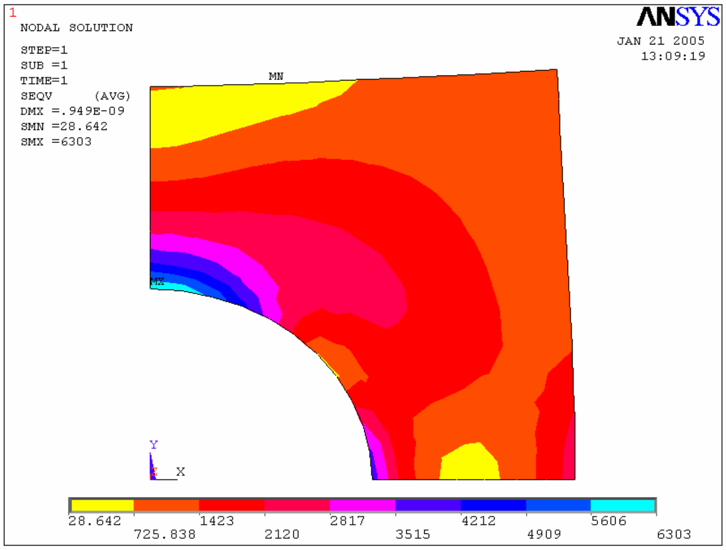


Figure 20: Example 2.3_stress contour

13. General Postproc> Plot results>Query Results> Subgrid Solution Select von Mises (See Figure 21).

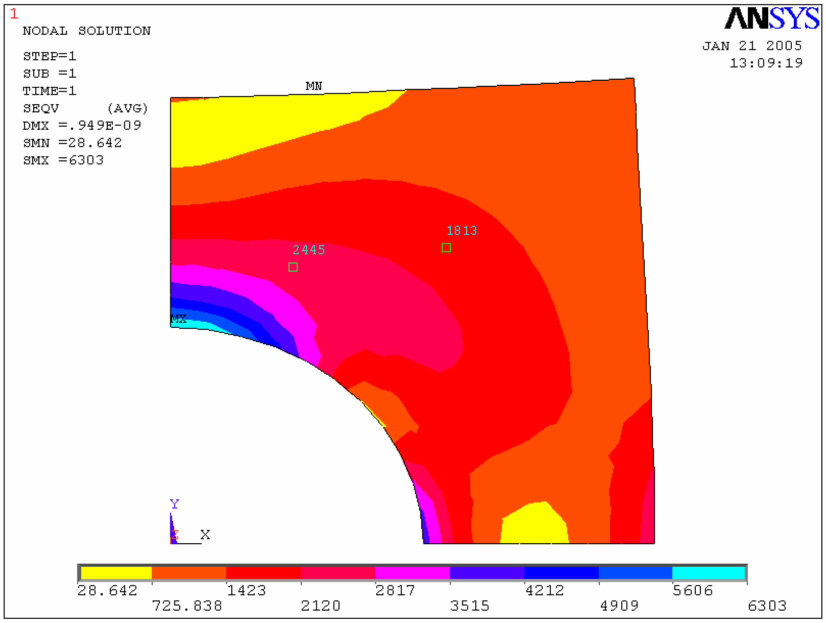


Figure 21: Example 3.3_stress results

3.4 Lab Assignment 2

1. Determine the nodal deflections, reaction forces, and stress for the beam shown in Figure 22 (E = 200GPa, I=0.000925m^4). Hint: Use Beam element. b = 0.02 ft

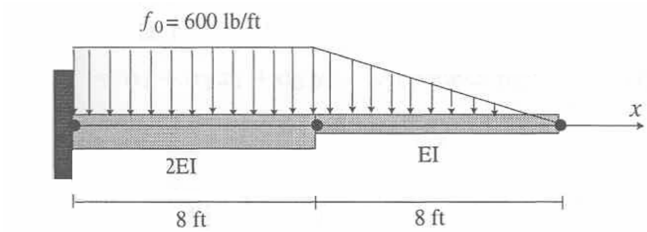


Figure 22: 2-D structure_ assignment 2.1

2. Determine the nodal deflections, reaction forces, and stress for the truss system shown in Figure 23 (E = 200GPa, I=0.000925m^4). Hint: Use Beam element. L = 8 m, b=0.2 m

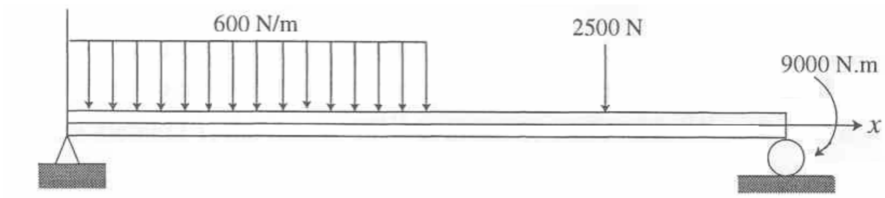


Figure 23: 2-D structure_ assignment 2.2

3. Consider the square plate shown in Figure 24 of uniform thickness with a circular hole with dimensions shown in the figure below. The thickness of the plate is 1 mm. The Young's modulus E =10^7 MPa and the Poisson ratio is 0.3. A uniform pressure p=1 MPa acts on the boundary of the hole. Assume that plane stress conditions prevail. Determine the stress and displacement fields

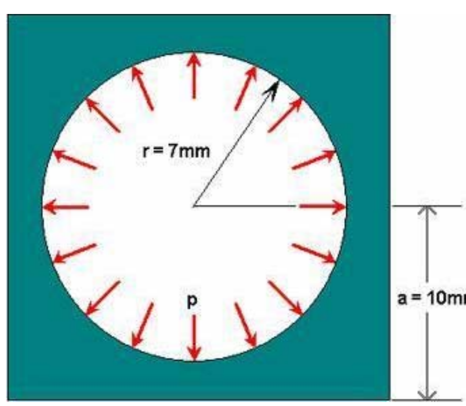


Figure 24: 2-D structure_ assignment 2.3

4. DYNAMIC ANALYSIS

4.1 Free Vibration: Solid Structure (Modal Analysis)

Determine the vibration responses of the work table show in Figure 25.

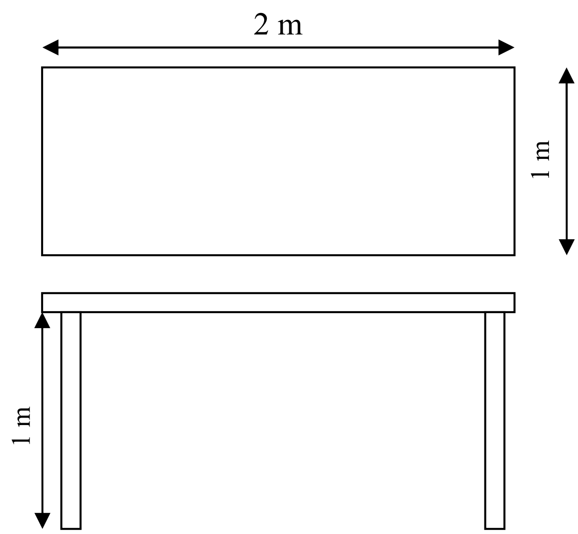


Figure 25: Solid structure

Preprocessing: Defining the Problem

1. Title
In the Utility menu bar select File > Change Title: Work Table

2. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**.
Select **solid45**

3. Element Material Properties
In the 'Preprocessor' menu select **Material Props > Material Models**
Double click on **Structural > Linear > Elastic > Orthotropic**
Enter density = 415
Ex=1e10, GXY=5e8

4. Create geometry**: Area**

Preprocessor > Modeling > Create > Blocks > By Dimensions>Use the following entries

Block	X1	X2	Y1	Y2	Z1	Z2
1	0	2	0	1	0	0.05
2	0.04	0.09	0.04	0.09	0	-1
3	1.91	1.96	0.91	0.96	0	-1
4	1.91	1.96	0.04	0.09	0	-1
5	0.04	0.09	0.91	0.96	0	-1

Plotctrl/zoom/ pan/rotate
Preprocessor > Modeling > Opreate > ADD > volumes

5. Meshing > Meshtools (Smart Size=4) > Mesh

6. Saving Your Work
On the **Utility Menu** select **File > Save as...**.

Solution: Assigning Loads and Solving

7. From the **Solution** Menu, select **Analysis Type > New Analysis**.
Select **modal** solution>analysis option> select 5 modes

8. Apply Constraints
Select>entities>areas/by num/pic> select bottom of legs
Select entities> nodes>Attached to> areas all >ok
Select> everything
In the **Solution** menu, select **Define Loads > Apply > Structural > Displacement > On Nodes> pick all**

9. Solving the System
In the 'Solution' menu select **Solve > Current LS**
General post proc> Result summary
General post proc> Read Result > First set> Plot result> Deformed shape> Plotctrl> animate
The third mode shape is shown in Figure 26.

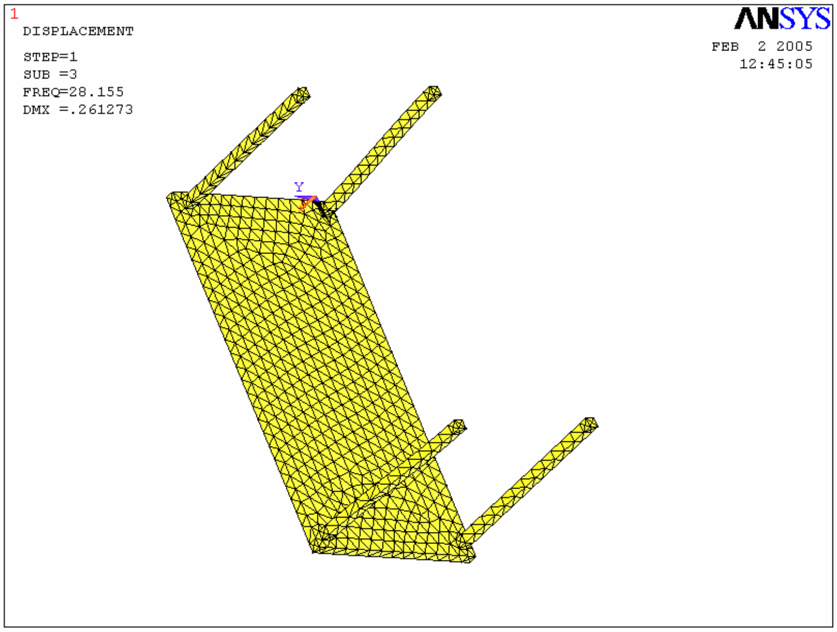


Figure 26: mode shapes of solid structure

4.2 Transient Analysis: Spring-Mass System
Determine transient response of mass 2 of the system shown in Figure 27.

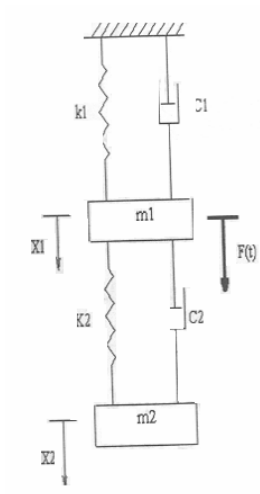


Figure 27: transient analysis _spring-mass system

Preprocessing: Defining the Problem

1. Title
In the Utility menu bar select File > Change Title: mass-spring

2. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**.

Use **Combin 14** for spring and dampers
Use **Mass 21** for mass uses option 2D without rotation

3. Define **Geometric Properties**
In the Preprocessor menu, select **Real Constants > Add/Edit/Delete**
Select mass 21 > mass=10 kg
Select **Combin 14** >K=100, CV1=20

4. Create geometry**: Nodes**
Preprocessor > Modeling > Create > Nodes

Use the following entries
N, 1 0, 0
N, 2, 0,-1
N, 3, 0,-2

Create element for the masses
Set the element attribute> define elements> select mass element> select real set
Create element> auto numbered> through nodes > select node 2> select node 3
Note: The E commands may be used alternatively as shown below.

Type, 2
Real, 1
E, 2; E, 3

Create element for the spring and dampers
Change the element attribute to spring element > select real set 2
Create element> auto numbered> through nodes > select nodes 1, 2> select nodes 2, 3

Type,1
Real,2
E,1,2; E,2,3

5. Saving Your Work
On the **Utility Menu** select **File > Save as...**.

<u>Solution</u>: Assigning Loads and Solving

6. From the **Solution** Menu, select **Analysis Type > New Analysis**>Transient

7. Apply Constraints
In the **Solution** menu, select **Define Loads > Apply > Structural > Displacement > On Nodes**
Select Nodes 1 > all DOF

8. Apply Loads
Switch unabridged to abridge menu
Select **Define Loads > Apply > Structural > Force > on Node**> **Nodes 2 > Fy=-5**
Select **Solution > load step option>Time-Frequency/Time& Substep**
Time at the end of load step =1
Number of substeps =10
KBC: Ramped
Select **Solution > Output ctrls /DB-Results file**
Select every load substeps
Output conrl> Time frequency> time integration> amplitude decay> change Gama to 0.1
Note: Gama may be entered alternatively as shown below.
Use: TITP, 0.1 (for gama)

9. Solving the System
In the 'Solution' menu select **Solve > Current LS**

<u>Preprocessor</u>

- Time Hist. /define variables./add >select modal DOF results
- Enter: DOF:UY, Ref: 2 Node: 3 label: Y3
- Time Hist post Proce>variable viewer >Graph
- PLVAR,2 (see Figure 28)

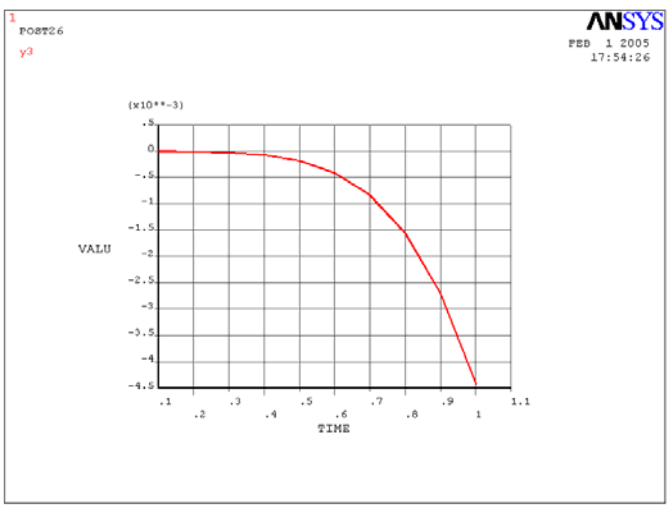


Figure 28: spring-mass system- transient response

The modal solution may be animated using the following sequence.
General post > Top menu> pltctrl> animate

4.3 Lab Assignment 3

1. Determine the first five natural frequencies and modes shapes of Guitar string with the length of 700 mm and diameter of 0.25 mm. the string is made of steel with material properties as follows: E = 190GPa, density = 7920 kg/m^3, A =196e^{-9} I = 1.53 e^{-12} .

Hint: Use Beam 3 element, assume zero displacement at both ends as boundary conditions.

2. Perform the modal analysis on the shell plate clamped in all the edges as shown in Figure 29. The dimensions of the plate is H=1, W=2 with the thickness of 0.01 [m]. (E = 200GPa, Density = 7800 kg/m^3).

Hint: Use Shell63 with uniform thickness.

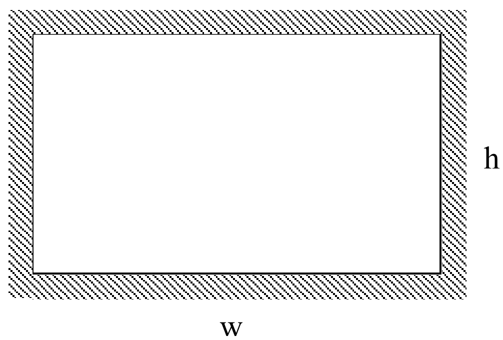
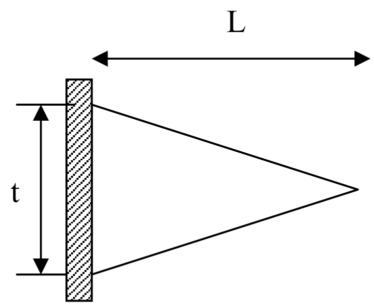


Figure 29: clamped-clamped plate_ assignment 3.2

5. NON-STRUCTURAL PROBLEMS

5.1 Steady State Heat Transfer

Determine temperature distribution in the triangular fine shown in Figure 30 with constant base temperature of 100 C^0:



h = 80 W/m^2 K
t = 0.01m
L = 0.025 m
K = 173 W/mK

Figure 30: steady state heat transfer

Preprocessing: Defining the Problem

1. Title
In the Utility menu bar select File > Change Title: Fine

2. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete> Plane55**.
In the 'Preprocessor' menu select **Material Props > Material Models**
Double click on **Thermal> Conductivity > Isotropic**> KXX =173

3. Create Geometry**: Keypoints**
Preprocessor > Modeling > Create > Key points > In Active CS
Enter the following: X1=0, Y1=0, X2=0.025, Y2=0, X3=0, Y3=0.005
Preprocessor > Modeling > Create > Lines > Straight Line
Preprocessor > Modeling > Area-Arbitrary > By Lines >

4. Meshing > Meshtools > Smart Size> 4

5. Saving Your Work
On the **Utility Menu** select **File > Save as...**.

Solution: Assigning Loads and Solving

6. From the **Solution** Menu, select **Analysis Type > New Analysis**>Steady-state

7. Apply Loads
Select **Define Loads > Apply > Thermal > Temprature > on Node**> **All left Nodes >** 100

Select **Define Loads > Apply > Convention > on Lines**> **Select upper line**

8. Solution> **Solve > Current LS**

<u>Postprocessor</u>

9. Post Pro
Select **Plot Results > Nodal Solution>**
Select **Plot Results > Vector Plot-Predefined>**

5.2 Transient Heat Transfer

A semi-infinite plane is held at 0 C. Suddenly contact with a fluid with 100 C^0 and $h = 50\ W/m^2\ K$. Determine the heat distribution in the plane after 2 Seconds. Material and geometric properties are given bellow.

Material Properties:	Geometric properties:	Loading:
$K = 54\ W/m\ C^0$	$a = 1$ m	$T_0 = 0$ C
$\rho = 7833\ kg/\ m^3$	$b = 0.1$ m	$T_e = 1000C$
$c = 0.465\ kJ/\ Kg\text{-}C^0$		$h = 50\ W/m^2K$.

<u>Preprocessing:</u> Defining the Problem

1. Title
In the Utility menu bar select File > **Change Title**: Transient heat

2. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**>**Plane55**

3. Element Material Properties
In the 'Preprocessor' menu select **Material Props > Material Models**
Double click on **Thermal > Isotropic**

4. Create Geometry**: Area**
Preprocessor > Modeling > Create > Rectangle > By 2 Corners

5. Meshing > Meshtools (Smart Size = 2) > Mesh

6. Saving Your Work
On the **Utility Menu** select **File > Save as...**.

<u>Solution:</u> Assigning Loads and Solving

7. From the **Solution** Menu, select **Analysis Type > New Analysis**> **Transient**

8. Apply Loads
Select **Define Loads > Apply > Thermal > Uniform temp >**
Select **Define Loads > Apply > Convention > on Lines**> **Select upper line**
Enter film coefficients = 50
Bulk tem = 1000
Time-Freq/Time-Timesteps
Time = 2
Deltime = 0.1
Stepped load
Select **Solution > Output Ctrls /DB-Results file>Every Substep**

9. Solving the System
In the 'Solution' menu select **Solve > Current LS**

Postprocessor

10. Post Pro
General post proc> Read Result > Last set
General post proc> Read Result > Nodal solution
- Time Hist. /Define Variables/Add >
- Enter: DOF: Temp, Ref: 2 Node: 1 , label: Temp (N,0,1,0)
- Time Hist post Proce>Variable Viewer >Graph
- PLVAR, 2

5.3 Fluid Analysis

5.3-1 *Cylindrical container*
Obtain the shape of the free surface of the liquid in cylindrical container filled with water when rotating with the rotational velocity of Ω = 25 rad/s. The height of water inside the container is 4 m. The radius of cylinder is 3 m. density of water is 1000 kg/m^3

Preprocessing: Defining the Problem

1. Title
In the Utility menu bar select File > Change Title: Swirl

2. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete> Fluid 79**.
Select Axisymmetric from option menu
In the 'Preprocessor' menu select **Material Props > Material Models>structural>isotropic** >EX=1e20,
Material Props > Material Models>structural>density > 1000

3. Create Geometry**: Area**
Preprocessor > Modeling > Create > Area > Rectangle> by dimension

Enter the following: X1=0, Y1=0, X2=3, Y2=4

4. Meshing > Meshtools > Smart Size> 4

5. Saving Your Work
On the **Utility Menu** select **File > Save as...**.

Solution: Assigning Loads and Solving

6. From the **Solution** Menu, select **Analysis Type > New Analysis**>static

7. Apply Loads
Select **Define Loads > Apply > displacement > on Node**> **All right Nodes >** UX=0
Select **Define Loads > Apply > displacement > on Node**> **All left Nodes >** UY=0
Select **Define Loads > Apply > Inertia>Gravity > Y direction>**9.81
Select **Define Loads > Apply > Inertia > Angular velocity> Global>**Y **direction>**25

Solution

8. Solution> **Solve > Current LS**

Post Processor

9. Select **Plot Results > Deformed shape>**

5.3-2 *Diffusion problems*
Determine the moisture diffusion inside the wooden sphere.

Preprocessor

1. Title
In the Utility menu bar select File > **Change Title**: Diffusion

2. Define **Element**
From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**>**Plane35**
Option > **Axisymmetic**

3. Element Material Properties
In the 'Preprocessor' menu select **Material Props > Material Models**
Double click on **Thermal > Conductivity>Isotropic**>KXX=4.3e-6
Thermal>Density>DENS=1
Thermal>Specific Heat>C=1

4. Create Geometry**: Area**
Preprocessor > Modeling > Create > Areas>Circle>Partial Annulus

WPX = 0
WPY = 0
Rad_1 = 0
Teta_1 = -90
Rad_2 = 0.1
Teta_2 = 90

5. Meshing > Meshtools >Global set>20> Mesh

6. Analysis Type > New Analysis> Transient

7. Define Loads > Apply > Thermal > Temperature>Uniform temp > 70

8. Utility>Select>Entities>Lines>By Num or Pick> Select curved line

9. Utility>Select>Entities>Node>Attached to>Lines-all

10. Solution>Define Loads>Apply>Thermal>Temperature>On Nodes>Pick All> 5

11. Entities>Select>Everything

<u>Solution</u>

12. Solution>Time/Frequency>Time-Time Step>TIME = 200**, DELTIME =** 10**, KBC = Stepped Loading**

13. Solving the System
In the 'Solution' menu select **Solve > Current LS**

<u>Postprocessor</u>

14. General post proc> Read Result > Last set
General post proc> Read Result > Nodal solution

- Time Hist. /Define Variables/Add >
- Enter: DOF: Temp, Ref: 2   Node: 1 , label: Temp (N,0,1,0)
- Time Hist post Proce>Variable Viewer >Graph
- PLVAR, 2

5.4 Lab Assignment 4

Problem 1

A long thick-walled cylinder shown in Figure 31 is maintained at a temperature T_i on the inner surface and T_o on the outer surface. Determine the temperature distribution through the wall thickness [3].

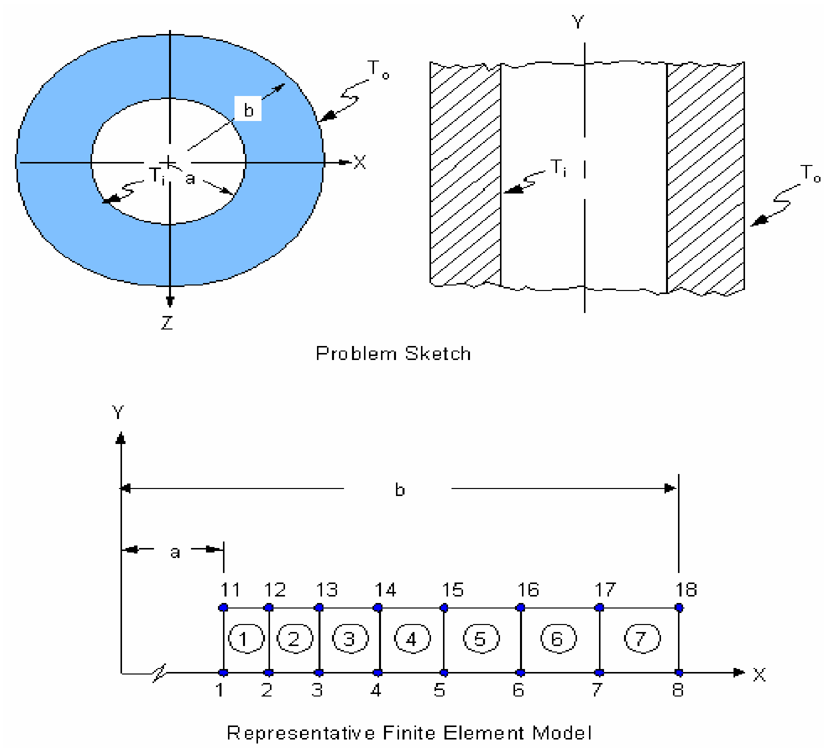


Figure 31: Thermal analysis_ assignment 4.1

Material Properties	Geometric Properties	Loading
$E = 30 \times 10^6$ psi	a = 0.1875 inches	T_i = -1°C
$\alpha = 1.435 \times 10^{-5}$ in/in-°C	b = 0.625 inches	T_o = 0°C
$\upsilon = 0.3$		
k = 3Btu/hr-in-°C		

Problem 2

A tapered rectangular stainless steel cooling fin shown in Figure 32 dissipates heat from an air-cooled cylinder wall. The wall temperature is T_W, the air temperature is T_a, and the convection coefficient between the fin and the air is h. Determine the temperature distribution along the fin and the heat dissipation rate *q*. Use 2-D Thermal Solid Elements (PLANE55) [3].

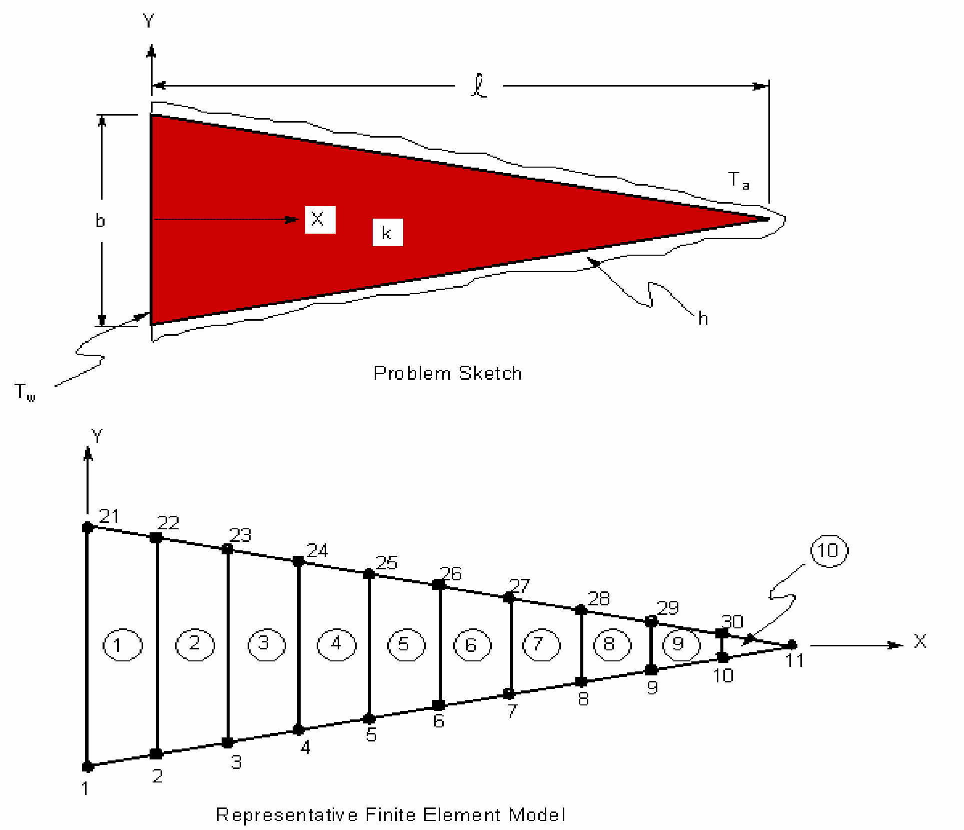


Figure 32: Thermal analysis_ assignment 4.2

Material Properties	**Geometric Properties**	**Loading**
k = 15 Btu/hr-ft-°F	b = 1 in = (1/12) ft	T_W = 1100°F
h = 15 Btu/hr-ft^2-°F		T_a = 100°F
	= 4 in = (4/12) ft	

6. INTRODUCTION TO CATIA

CATIA is very powerful finite element based software with excellent capabilities in design and analysis. Design and modeling with CATIA itself need lots of practice and skills which is not in the scope of this tutorial. In this section, in order to show the capability of CATIA in finite element analysis a simple beam with distributed loads has been introduced.

To start CATIA in Windows environment simply follow the path:
Start Menu >Programs >CATIA >CATIA V5R10.

The first window you may see is the main window shown in Figure 33.

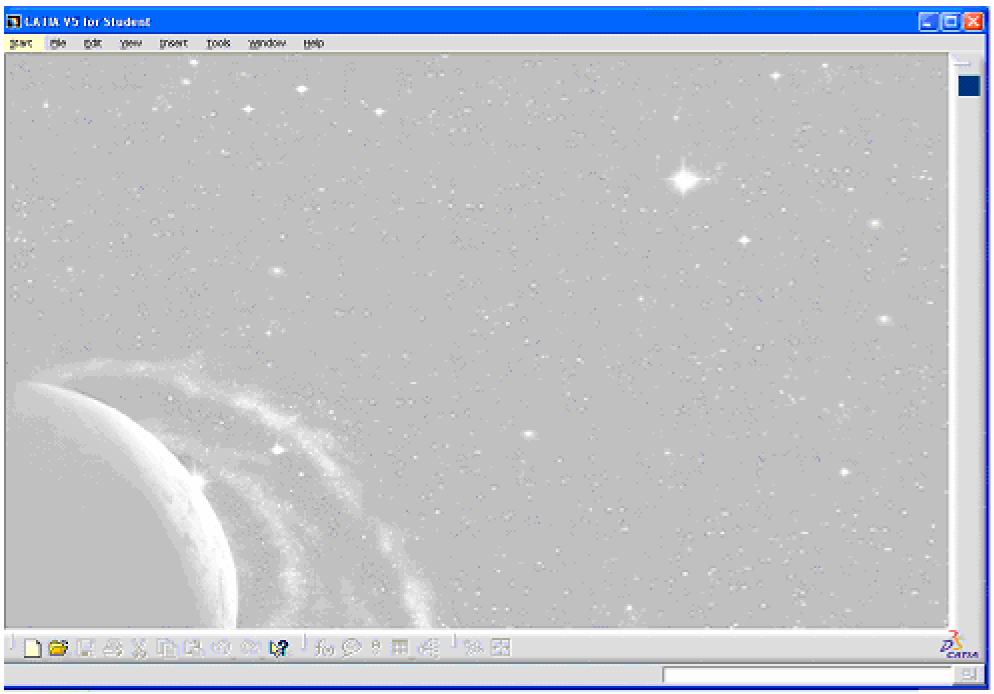


Figure 33: Main Window of CATIA V5

1. To create the model: **Start> Mechanical Design> Sketcher (**See Figure 34)

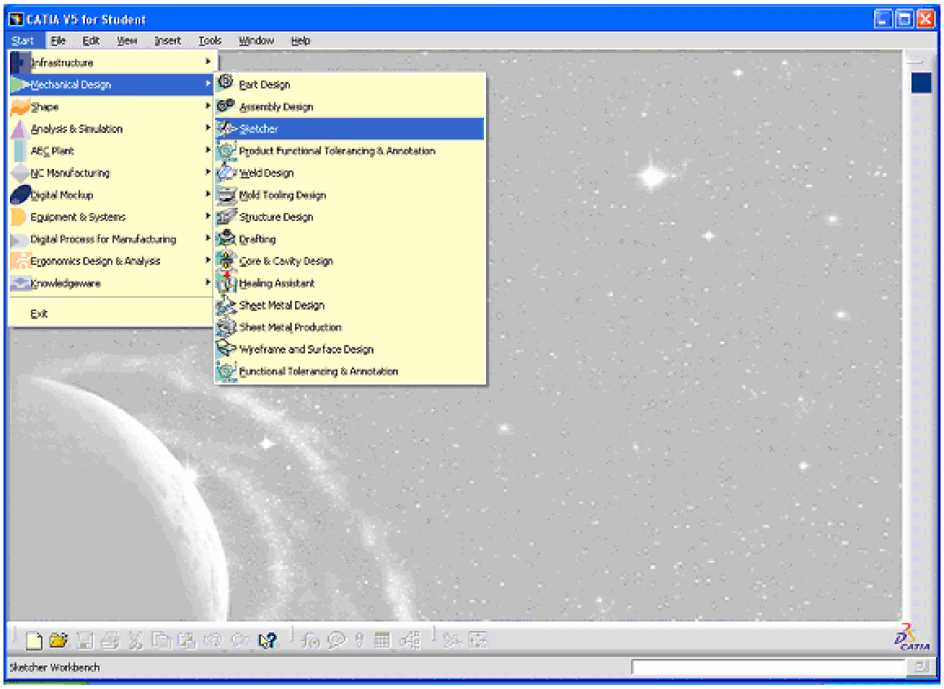


Figure 34: Main Window of CATIA V5

2. Now you should have a window as Figure 35.

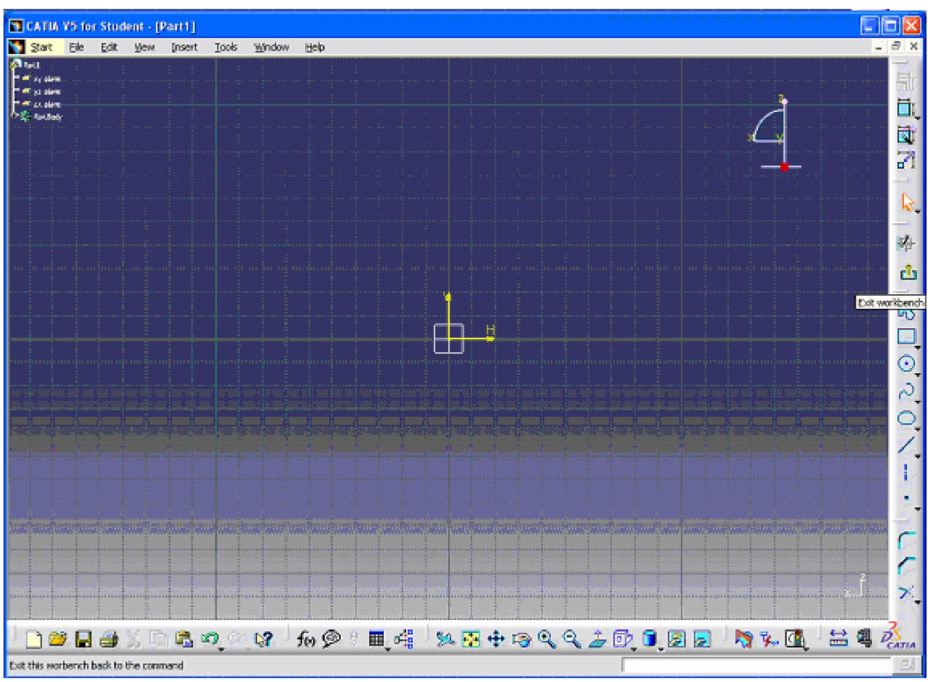


Figure 35: Mechanical Design Sketcher

3. Use drawing tools in the right bar to create your model. For example in Figure 36, we created an arbitrary rectangle. To add dimension to the line, simply click on Constraint option ( ) > select line> double click on the measure line> in the pop up window enter the value.

For symmetric axis, select upper line+ctrl > select lower line> click on constraints key .

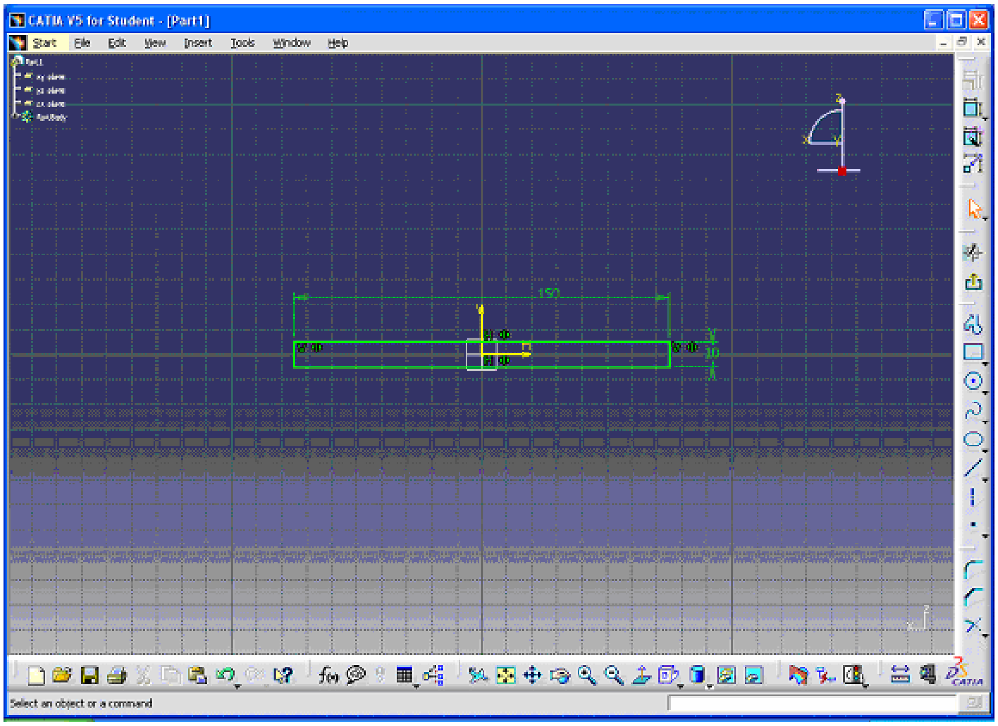
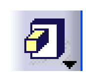


Figure 36: Create an arbitrary rectangle

4. To make a three dimensional design, click on  > click on  > in the pop window select the width.

5. To select materials click on  to have a variety of materials to select as shown in Figure 37. in this example we select steel.

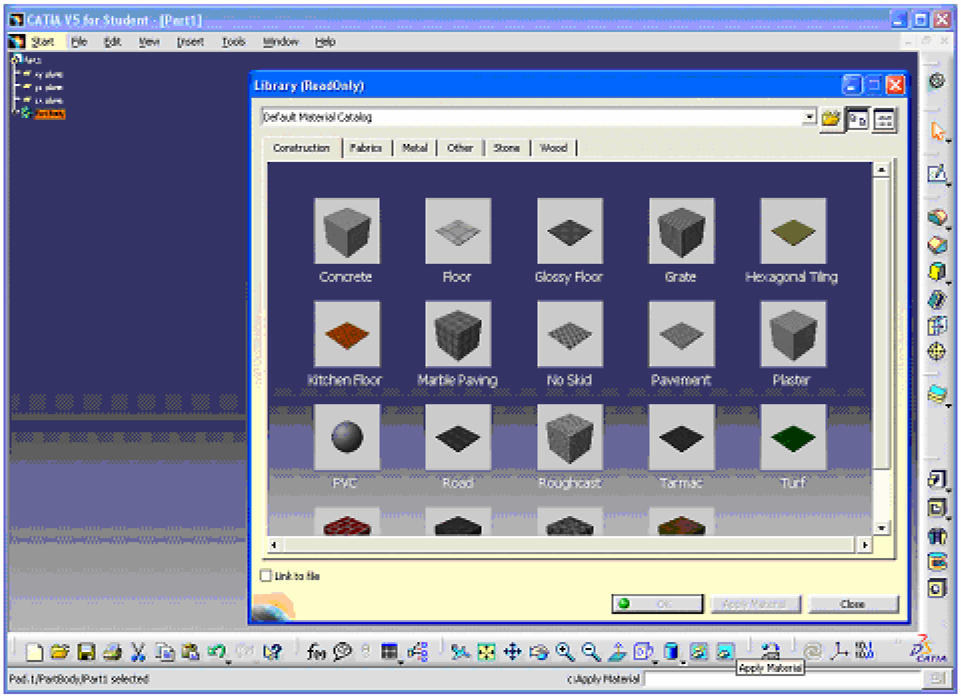


Figure 37: Select material properties

At this level, modeling session is done.

6. To start analysis: **Start> Analysis & Simulation> Generative Structural Analysis>** (See Figure 38).

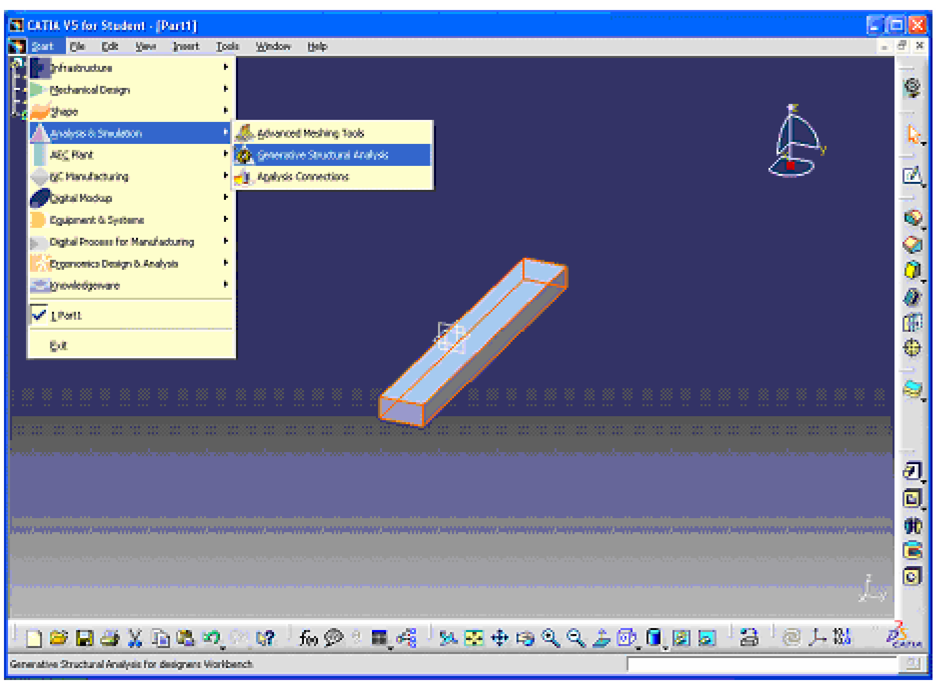


Figure 38: Structural Analysis

In the pop up window select analysis type. In this example we select static analysis. (See Figure 39)

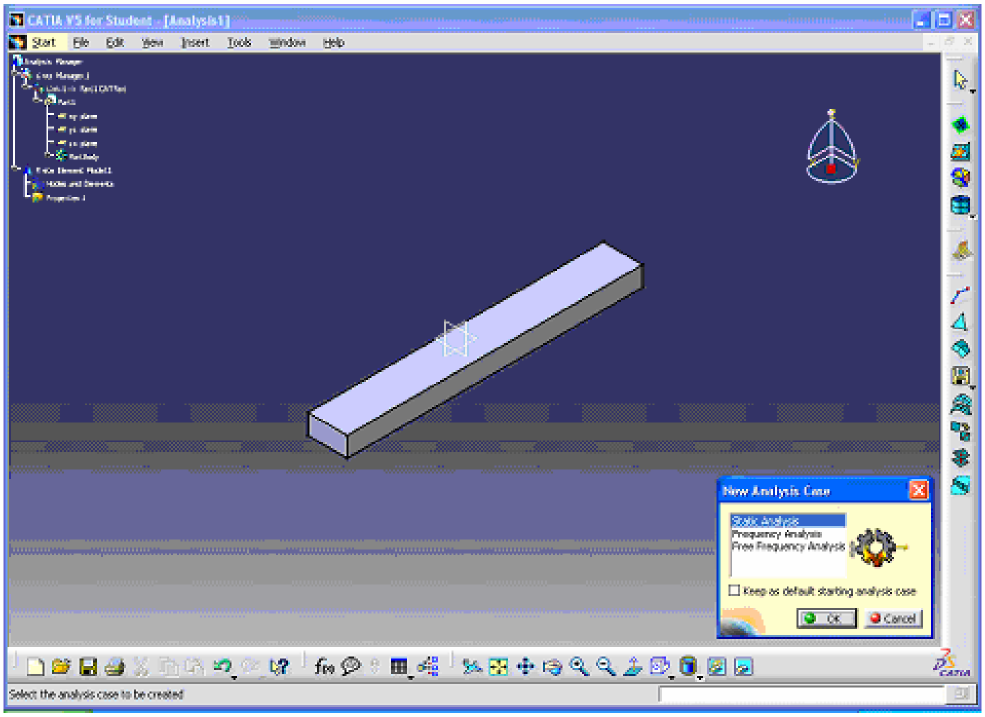


Figure 39: Static Analysis

7. To apply boundary conditions, select left side> click on for clamped-support**.**
8. To apply load select upper right line > click on > enter the value and direction of load
9. To solve the problem: click on .
10. To observe the results: click on to see the nodal displacements (See Figure 40).

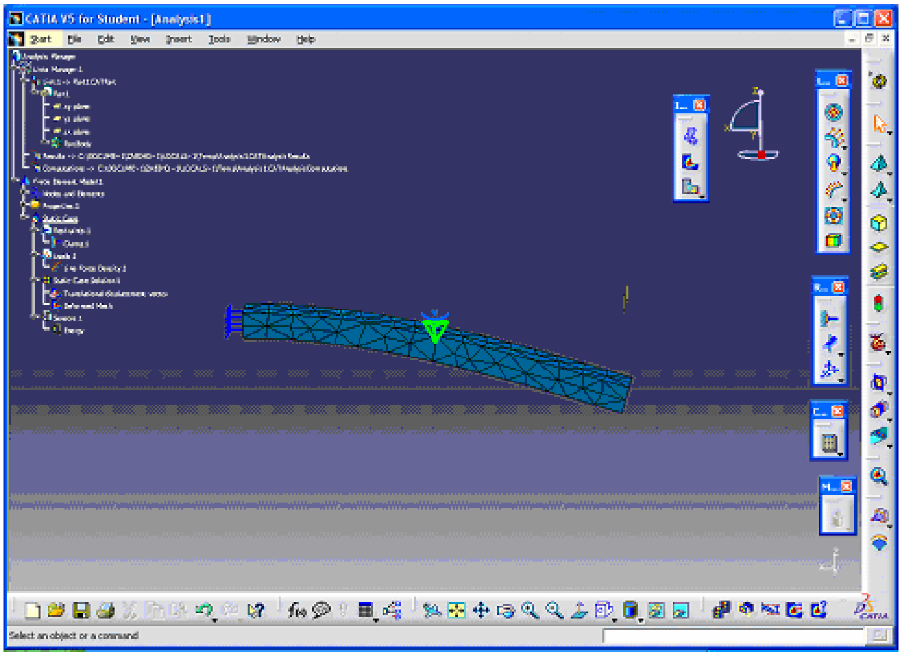


Figure 40: Nodal displacements

11. Click on  to see the Von mises stress (See Figure 41).

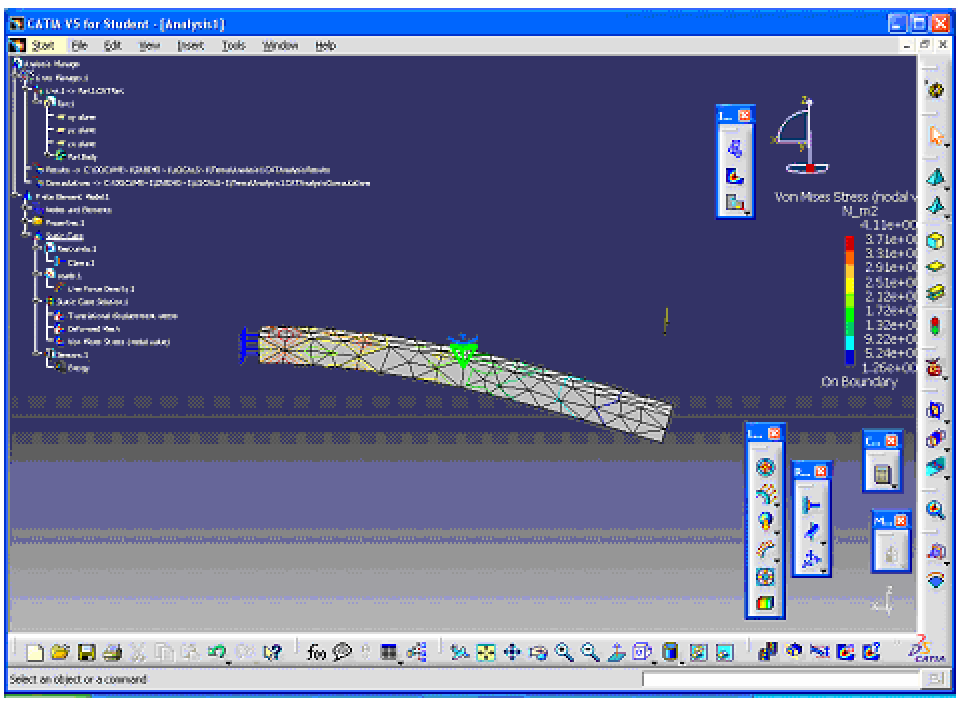


Figure 41: Von Mises stress

7. REFERENCES

[1] www.mece.ualberta.ca/tutorials/ansys/
[2] http://www.andrew.cmu.edu
[3] ANSYS Help manual