

**ME/AE 5212**  
**Introduction to Finite Element Analysis**



**Tutorial**

**Instructor: Dr. K. Chandrashekhara**  
**Computer Assistants: Z. Huo and X. Wang**

**Copyright © 2015**  
**Department of Mechanical and Aerospace Engineering**  
**Missouri University of Science and Technology**  
**Rolla, MO 65409**



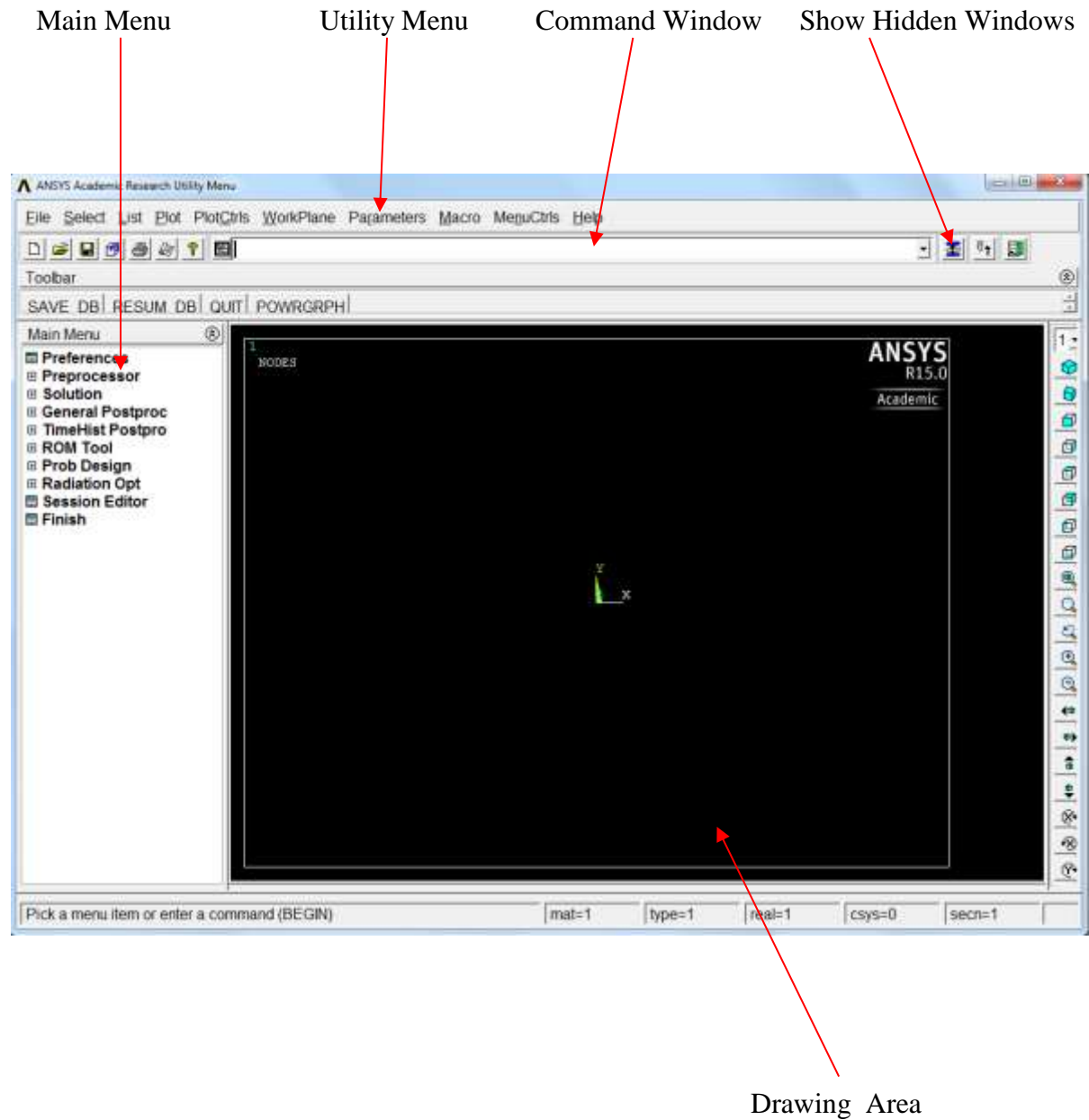
# **ANSYS TUTORIAL**

- **GETTING STARTED WITH ANSYS 15.0**
- **SOLID MECHANICS**
  - 1.1. BEAM**
  - 1.2. CIRCULAR PLATE**
- **HEAT TRANSFER**
- **FLUID MECHANICS**

# INTRODUCTION TO ANSYS 15.0

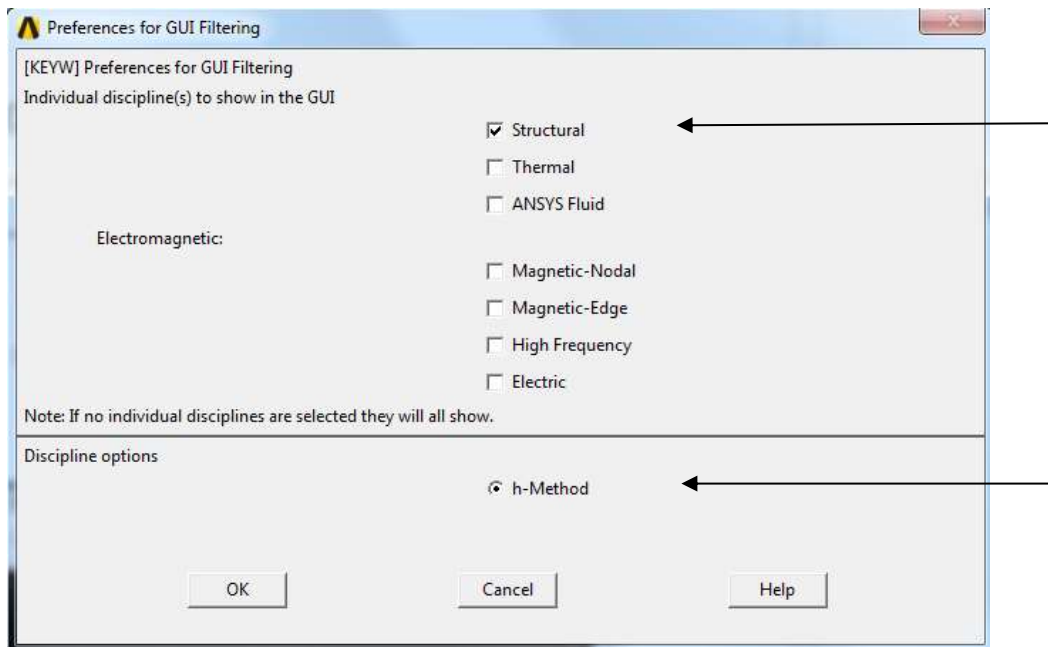
In a windows machine start ANSYS by clicking on

Start → All Programs → ANSYS 15.0 → Mechanical APDL 15.0



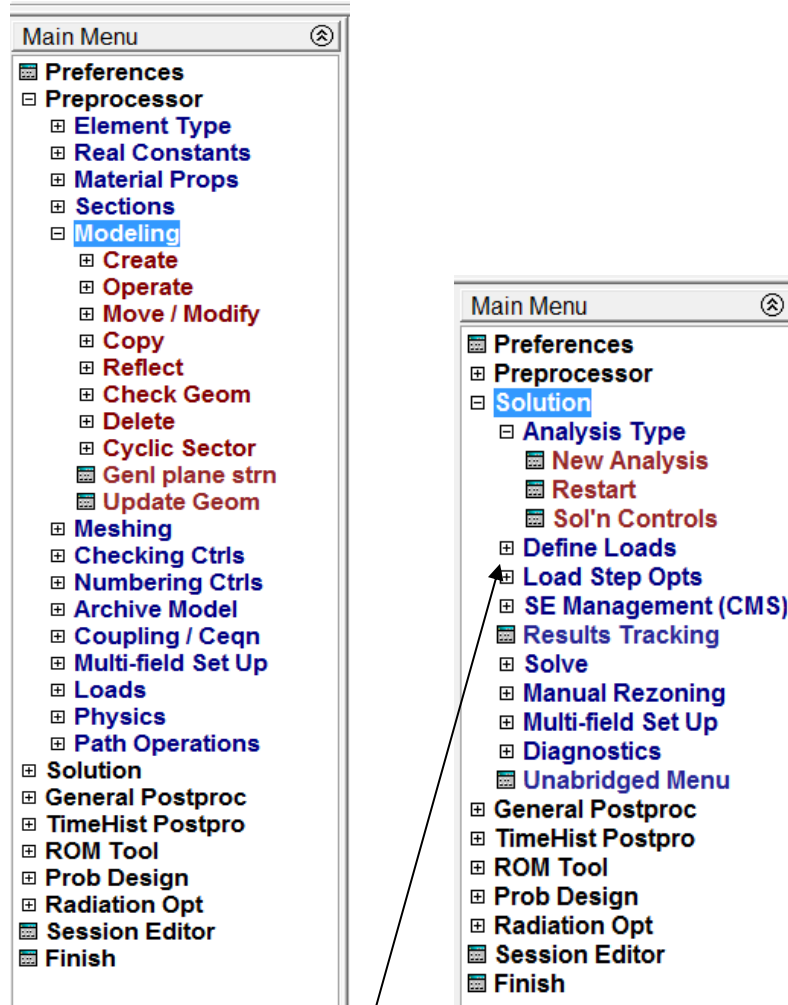
# GETTING START WITH ANSYS 15.0

**Step1:** Click on Preferences and choose the type of problem (Structural in this case). Leave the Discipline Options as default (h-method).



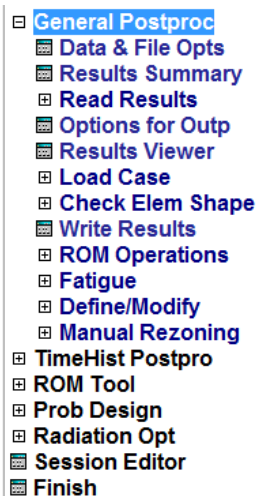
**Step 2:** Enter Preprocessor to establish the Finite Element Model.

- There are two approaches to build a FE model
  - From Top To Bottom
  - From Bottom to Top
- The general steps to follow in the analysis are:
  1. Choose the Element Type for the Analysis
  2. Set the Real constant Values for the element (if necessary)
  3. Define the Material Properties (Modulus, Poisson's Ratio etc)
  4. Create the Finite Element Model using the available tools in the create menu



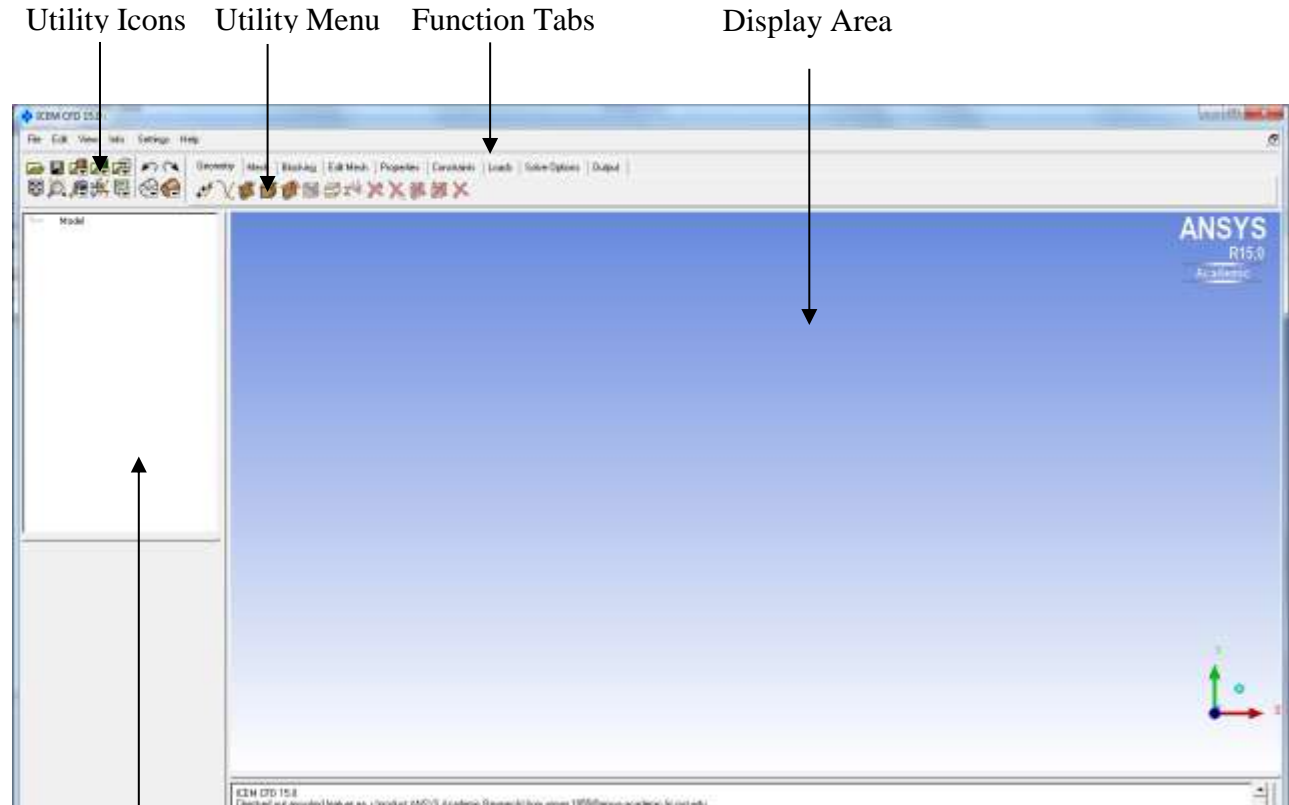
**Step 3:** Enter Solution Menu to Apply the Boundary Conditions and Loads and then solve the Problem

**Step 4:** Once the solution is done, enter the General Postprocessor to List and View the Results



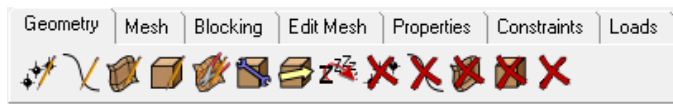
In this manual, ICEM (Integrated Computer-Aided Engineering and Manufacturing) is used to create and mesh the computational domain for fluid mechanics problems. After creation of the mesh, ANSYS CFX will be used to solve the flow problem and analyze the solution.

Start → All Programs → ANSYS 15.0 → Meshing → ICEM CFD 15.0



Model Tree

**Geometry:**



**Mesh:**



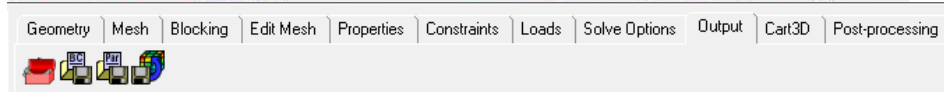
**Blocking:**



**Edit Mesh:**



**Output:**

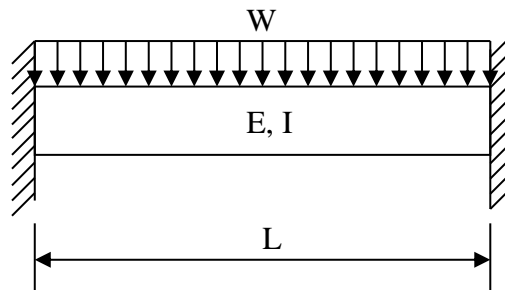


# SOLID MECHANICS

## 1.1 Analysis of a Beam

Determine the mid-span deflection and reactions for the fixed-fixed beam subjected to a uniformly distributed load as shown in the figure.

Take  $W=500 \text{ N/m}$ ,  $E = 200\text{GPa}$ , Poisson's ratio=0.3,  $L = 10 \text{ m}$ ,  $B=H=0.124467\text{m}$ ,  $I=2 \times 10^{-5} \text{ m}^4$



### STEP 1:

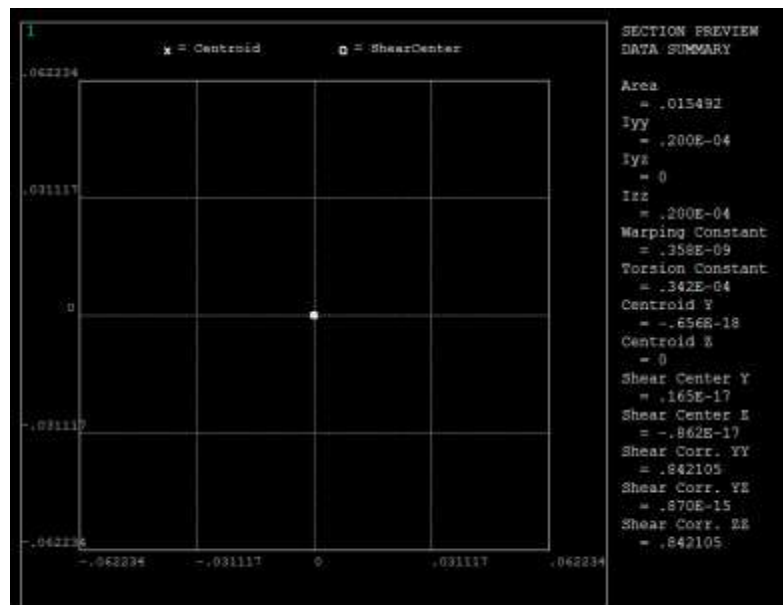
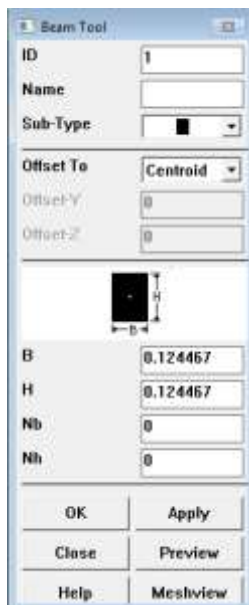
Preferences  $\longrightarrow$  Structural  $\longrightarrow$  OK

### STEP 2: Define Element Type

Preprocessor  $\longrightarrow$  Element Type  $\longrightarrow$  Add/Edit/Delete  $\longrightarrow$  Add...  $\longrightarrow$  Beam  
 $\longrightarrow$  2 node 188  $\longrightarrow$  OK  $\longrightarrow$  Close *Element Types* dialog box

### STEP 3: Define the Section

Preprocessor  $\longrightarrow$  Sections  $\longrightarrow$  Beam  $\longrightarrow$  Common Sections  $\longrightarrow$  Define the section as figure below  $\longrightarrow$  Apply  $\longrightarrow$  Preview, the figure below pop up



#### **STEP 4: Define Material Properties**

Preprocessor → Material Props → Material Models → Structural →  
Linear → Elastic → Isotropic

Enter EX = 200e9 and Poisson's Ratio = 0.3 → OK

#### **STEP 5: Create the Keypoints**

*Create keypoints:*

Preprocessor → Modeling → Create → Keypoints → In Active CS

Enter Keypoint number and X, Y, Z Location in active CS

Keypoint Number	X	Y	Z	
1	0.0	0.0	0.0	→ Apply
2	5.0	0.0	0.0	→ Apply
3	10.0	0.0	0.0	→ OK

The Keypoint numbers appear on the Graphics Window.

*Create Lines:*

Preprocessor → Modeling → Create → Lines → Lines → Straight Line

Select Keypoint 1 and 2

Select Keypoint 2 and 3 → Close the dialogue box

#### **STEP 6 Meshing**

Preprocessor → Meshing → Mesh Tools

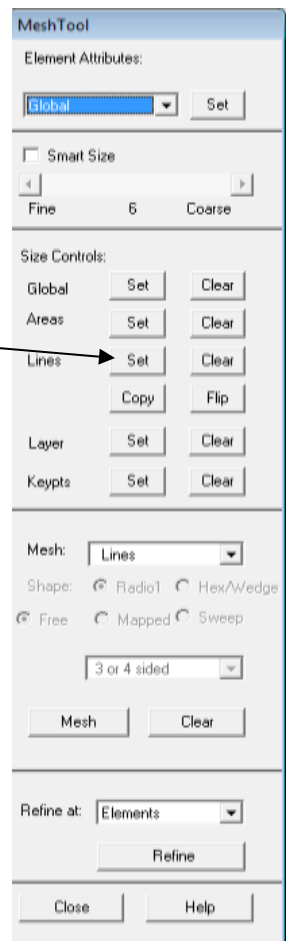
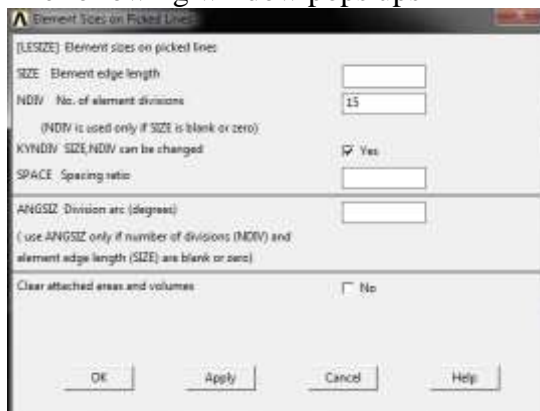
The following window pops up

Click on 'Set' under 'Lines'

Select both the lines

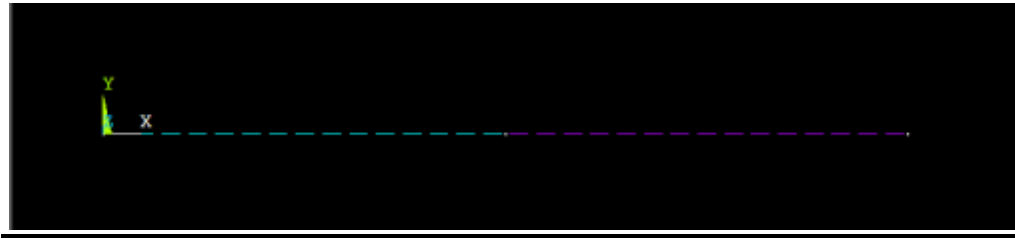
and click on OK

The following window pops up





Set number of element divisions to 15 and click on OK  
 The display on the graphical interface changes to



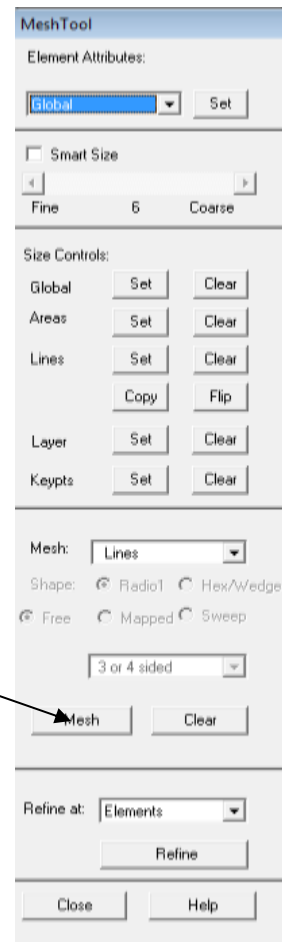
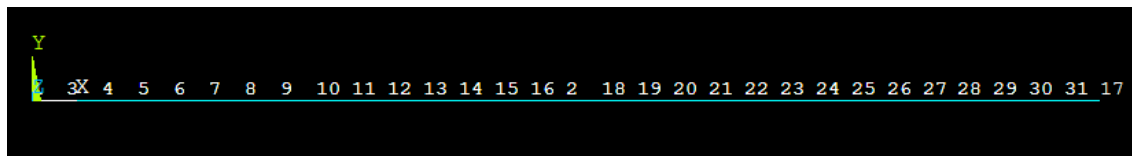
Now click on 'Mesh' in the 'Mesh Tool' window

Select both the lines from the graphical interface.  
 Click OK

Display the element and node numbers.

Utility menu → PlotCtrls → Numbering

Toggle on the 'Node Numbers' option. (If the drawing is disappeared, please go to, Plot (from utility menu on the top) → Multi-Plots) → OK



After finishing this step, go to the List to check the element, nodes material assignment.

Utility menu → List → Elements → Nodes+Attr+Realconstant

ELIST Command

File

LIST ALL SELECTED ELEMENTS. <LIST NODES>

ELEM	MAT	TYP	REL	ESY	SEC	NODES		
1	1	1	1	0	1	1	3	0
2	1	1	1	0	1	3	4	0
3	1	1	1	0	1	4	5	0
4	1	1	1	0	1	5	6	0
5	1	1	1	0	1	6	7	0
6	1	1	1	0	1	7	8	0
7	1	1	1	0	1	8	9	0
8	1	1	1	0	1	9	10	0
9	1	1	1	0	1	10	11	0
10	1	1	1	0	1	11	12	0
11	1	1	1	0	1	12	13	0
12	1	1	1	0	1	13	14	0
13	1	1	1	0	1	14	15	0
14	1	1	1	0	1	15	16	0
15	1	1	1	0	1	16	2	0
16	1	1	1	0	1	2	18	0
17	1	1	1	0	1	18	19	0
18	1	1	1	0	1	19	20	0
19	1	1	1	0	1	20	21	0
20	1	1	1	0	1	21	22	0
ELEM	MAT	TYP	REL	ESY	SEC	NODES		
21	1	1	1	0	1	22	23	0
22	1	1	1	0	1	23	24	0
23	1	1	1	0	1	24	25	0
24	1	1	1	0	1	25	26	0
25	1	1	1	0	1	26	27	0
26	1	1	1	0	1	27	28	0
27	1	1	1	0	1	28	29	0
28	1	1	1	0	1	29	30	0
29	1	1	1	0	1	30	31	0
30	1	1	1	0	1	31	17	0

## STEP 7: Apply Boundary Conditions and Loads

### ***Apply B.C's***

Solution → Define loads → Apply → Structural → Displacement → On Nodes.  
Select the two end nodes (click on nodes at the most left and right) → OK (or middle click).

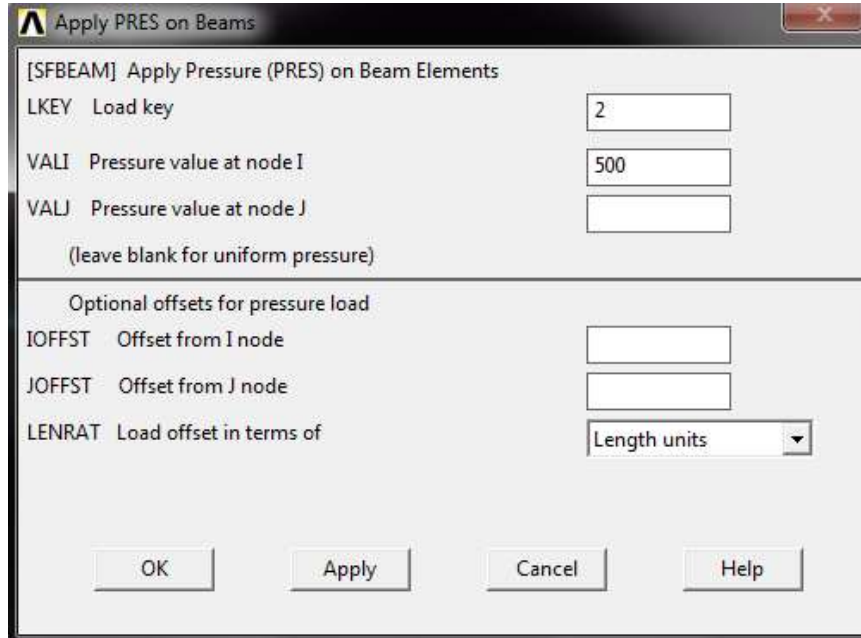
Select ALL DOF in the dialog box appeared → OK.

The Boundary conditions will be displayed on the graphics window.

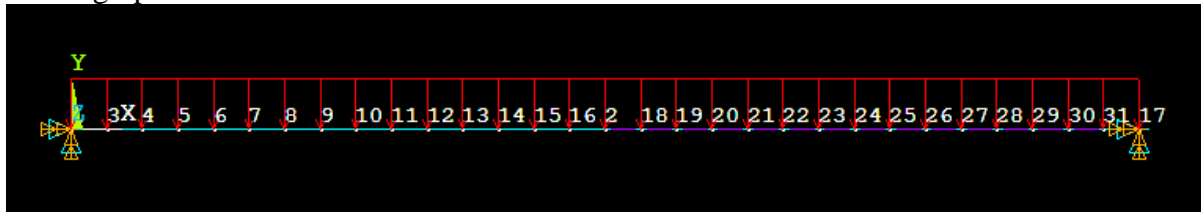
### ***Apply Pressure on Beam***

Solution → Define loads → Apply → Structural → Pressure → On Beams

Use the Box method to select all the elements and middle click. The following dialog box appears:



Enter the value of Load key=2, pressure = 500 and click OK. The pressure will be displayed on the graphics window.



### **STEP 8: Solve the problem**

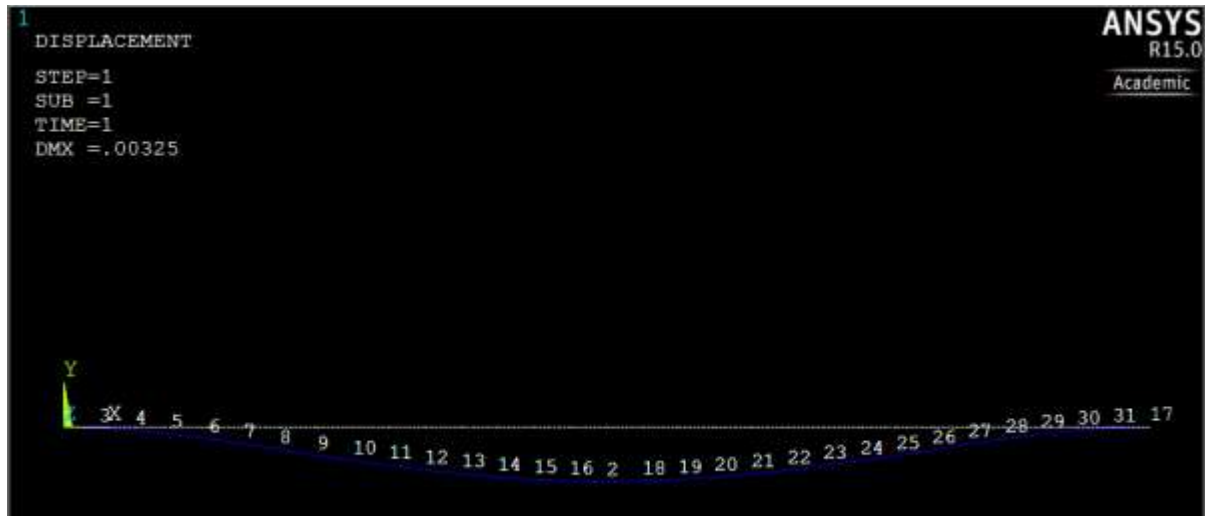
Solution → Solve → Current LS.

Click *OK* to solve.

### **STEP 9: View results.**

General Postproc → Plot Results → Deformed Shape

Select *Def + undeformed* and select *OK*.



### Nodal Solution

General Postproc → List Results → Nodal Solution → DOF solution → Displacement vector sum → OK

PRNSOL Command

File

PRINT U NODAL SOLUTION PER NODE

\*\*\*\*\* POST1 NODAL DEGREE OF FREEDOM LISTING \*\*\*\*\*

LOAD STEP= 1 SUBSTEP= 1  
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	-0.32497E-02	0.0000	0.32497E-02
3	0.0000	-0.53374E-04	0.0000	0.53374E-04
4	0.0000	-0.20029E-03	0.0000	0.20029E-03
5	0.0000	-0.41991E-03	0.0000	0.41991E-03
6	0.0000	-0.69295E-03	0.0000	0.69295E-03
7	0.0000	-0.10017E-02	0.0000	0.10017E-02
8	0.0000	-0.13298E-02	0.0000	0.13298E-02
9	0.0000	-0.16628E-02	0.0000	0.16628E-02
10	0.0000	-0.19875E-02	0.0000	0.19875E-02
11	0.0000	-0.22923E-02	0.0000	0.22923E-02
12	0.0000	-0.25671E-02	0.0000	0.25671E-02
13	0.0000	-0.28036E-02	0.0000	0.28036E-02
14	0.0000	-0.29947E-02	0.0000	0.29947E-02
15	0.0000	-0.31351E-02	0.0000	0.31351E-02
16	0.0000	-0.32209E-02	0.0000	0.32209E-02
17	0.0000	0.0000	0.0000	0.0000
18	0.0000	-0.32209E-02	0.0000	0.32209E-02
19	0.0000	-0.31351E-02	0.0000	0.31351E-02
20	0.0000	-0.29947E-02	0.0000	0.29947E-02
21	0.0000	-0.28036E-02	0.0000	0.28036E-02
22	0.0000	-0.25671E-02	0.0000	0.25671E-02
23	0.0000	-0.22923E-02	0.0000	0.22923E-02
24	0.0000	-0.19875E-02	0.0000	0.19875E-02
25	0.0000	-0.16628E-02	0.0000	0.16628E-02
26	0.0000	-0.13298E-02	0.0000	0.13298E-02
27	0.0000	-0.10017E-02	0.0000	0.10017E-02
28	0.0000	-0.69295E-03	0.0000	0.69295E-03
29	0.0000	-0.41991E-03	0.0000	0.41991E-03
30	0.0000	-0.20029E-03	0.0000	0.20029E-03
31	0.0000	-0.53374E-04	0.0000	0.53374E-04

MAXIMUM ABSOLUTE VALUES

NODE	0	2	0	2
VALUE	0.0000	-0.32497E-02	0.0000	0.32497E-02

The displacement at node 2 is -0.0032497 m in the Y direction.

### Reaction Solution

General Postproc → List Results → Reaction Solution → All items → OK

```
PRRSOL Command
File

PRINT REACTION SOLUTIONS PER NODE

***** POST1 TOTAL REACTION SOLUTION LISTING *****

LOAD STEP=      1  SUBSTEP=      1
TIME=      1.0000  LOAD CASE=      0

THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

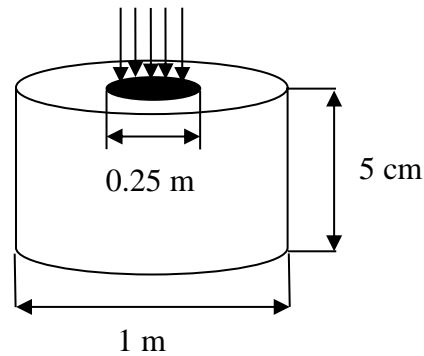
      NODE      FX      FY      FZ      MX      MY      MZ
      1      0.0000      2500.0      0.0000      0.20840E-13  0.25871E-12  4162.0
     17      0.0000      2500.0      0.0000      0.20840E-13 -0.25871E-12 -4162.0

TOTAL VALUES
VALUE      0.0000      5000.0      0.0000      0.41679E-13  0.11117E-24  0.19536E-08
```

Note: Due to variances of computation rounding, the final results could be a little different from the results above.

## 1.2 Analysis of a Circular Plate Using Axisymmetric Element

A circular aluminum plate with 1m diameter and 5cm thickness is loaded by a uniform pressure of 0.5Pa distributed over a 0.25m diameter circle at the center of the plate. The outer edges of the plate are completely fixed from motion. Find the maximum deflection and von Mises stress. Use Young's modulus = 70GPa, Poisson's ratio=0.33.

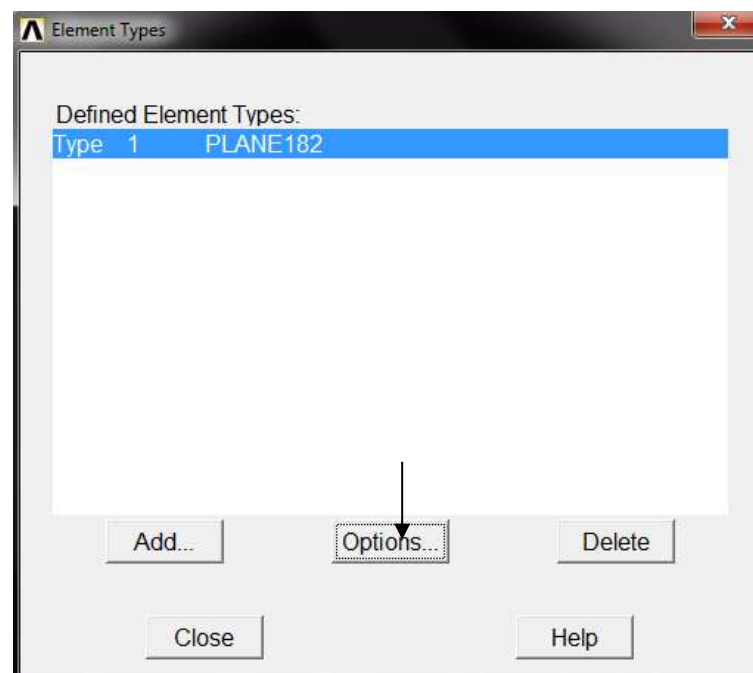


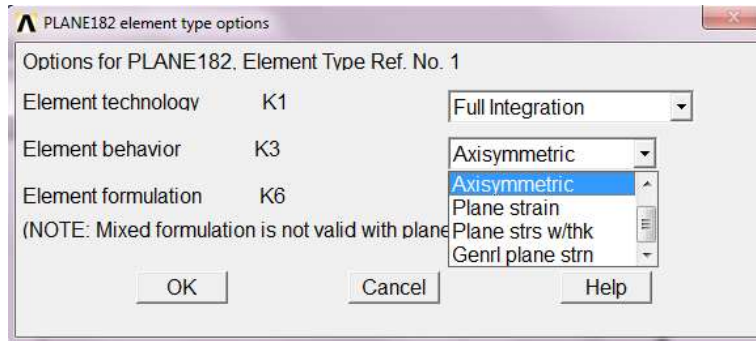
### **STEP 1:**

Preferences → Structural → OK

### **STEP 2: Define Element Type**

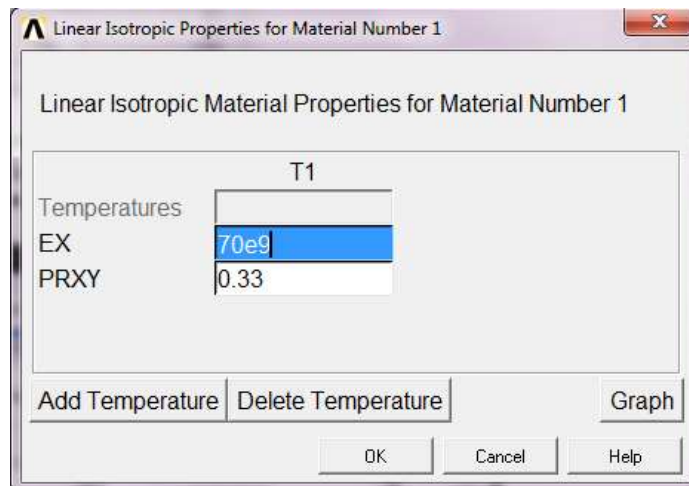
Preprocessor → Element Type → Add/Edit/Delete → Add... → Solid  
→ Quad 4 node 182 → OK → Close *Element Types* dialog box →  
Options → Element Behavior K3: Axisymmetric → OK





### **STEP 3: Define Material Properties**

Preprocessor → Material Props → Material Models → Structural →  
Linear → Elastic → Isotropic



Enter Young's Modulus (*EX*) and Poisson's ratio (*PRXY*) for the material and click OK.

### **STEP 4: Create the model**

*Create keypoints:*

Preprocessor → Modeling → Create → Keypoints → In Active CS

Enter Keypoint number and X, Y, Z Location in active CS

Keypoint Number	X	Y	Z	
1	0.0	0.0	0.0	→ Apply
2	0.5	0.0	0.0	→ Apply
3	0.5	0.05	0.0	→ Apply
4	0.125	0.05	0.0	→ Apply
5	0.0	0.05	0.0	→ OK

The Keypoint numbers appear on the Graphics Window.

### Create Lines:

Preprocessor → Modeling → Create → Lines → Lines → Straight Line  
Select Keypoint 1 and Keypoint 2  
Select Keypoint 2 and Keypoint 3  
Select Keypoint 3 and Keypoint 4  
Select Keypoint 4 and Keypoint 5  
Select Keypoint 5 and Keypoint 1 → Close the dialogue box

### Create Areas:

Preprocessor → Modeling → Create → Areas → Arbitrary → By Lines →  
Select all the lines created → OK

### STEP 5 Meshing

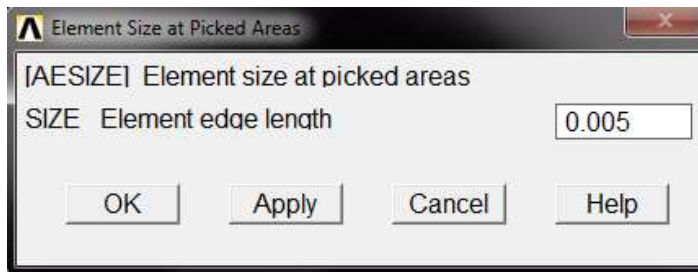
Preprocessor → Meshing → Mesh Tool

The following window pops up

Click on 'Set' on the right of 'Areas'

Select the area just created  
and click on OK

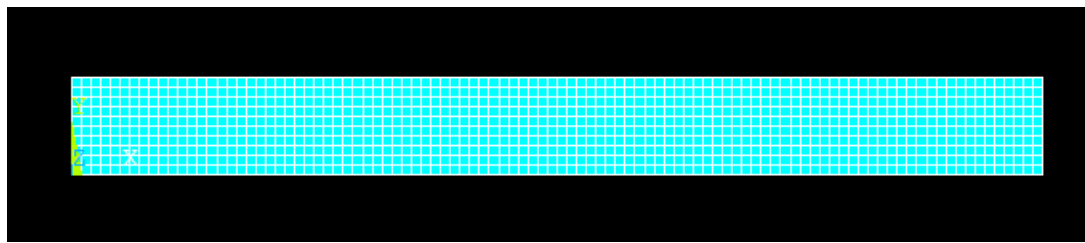
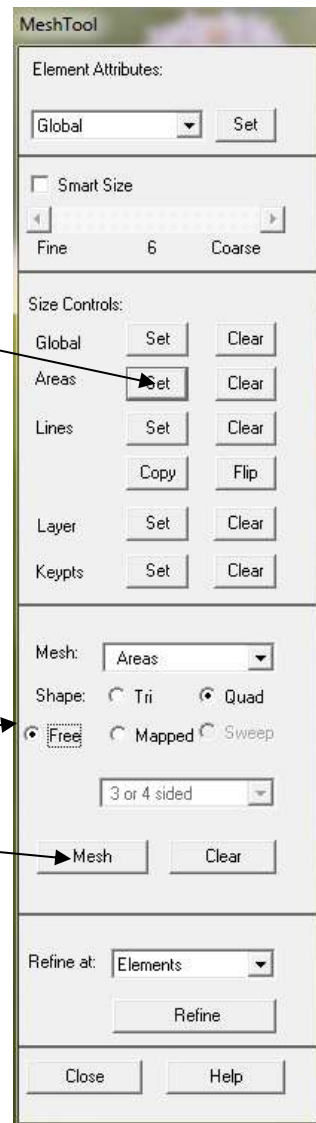
The following window pops up



Set element edge length as 0.005 and click on OK

Set Mesh domain as Areas, shape as Quad, Free

Click mesh → select the area just created → OK



After finishing this step, go to the List to check the element, nodes and material assignment.

Utility menu → List → Elements → Nodes+Attr+Realconstant

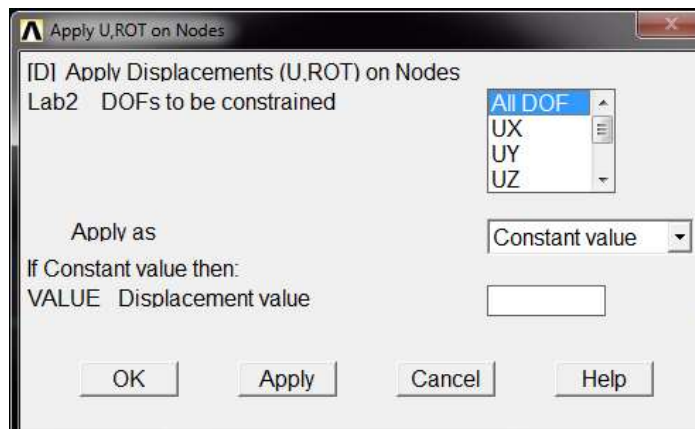
ELIST Command									
File									
LIST ALL SELECTED ELEMENTS. <LIST NODES>									
ELEM	MAT	TYP	REL	ESY	SEC	NODES			
1	1	1	1	0	1	1111	1110	1017	1018
2	1	1	1	0	1	1018	1019	1020	1111
3	1	1	1	0	1	1111	834	835	1110
4	1	1	1	0	1	1111	1020	833	834
5	1	1	1	0	1	835	836	1109	1110
6	1	1	1	0	1	836	837	1108	1109
7	1	1	1	0	1	1109	1016	1017	1110
8	1	1	1	0	1	1108	1015	1016	1109
9	1	1	1	0	1	837	838	1107	1108
10	1	1	1	0	1	1107	1014	1015	1108
11	1	1	1	0	1	838	839	1106	1107
12	1	1	1	0	1	1106	1013	1014	1107
13	1	1	1	0	1	839	840	1105	1106
14	1	1	1	0	1	1105	1012	1013	1106
15	1	1	1	0	1	840	841	1104	1105
16	1	1	1	0	1	1104	1011	1012	1105
17	1	1	1	0	1	841	842	1103	1104
18	1	1	1	0	1	1103	1010	1011	1104
19	1	1	1	0	1	842	843	1102	1103
20	1	1	1	0	1	1102	1009	1010	1103

## STEP 6: Apply Boundary Conditions and Loads

### Apply B.C's

Main Menu → Solution → Define loads → Apply → Structural →  
 Displacement → On Lines.  
 Select the right line → OK (or middle click).

The following dialog box appears:



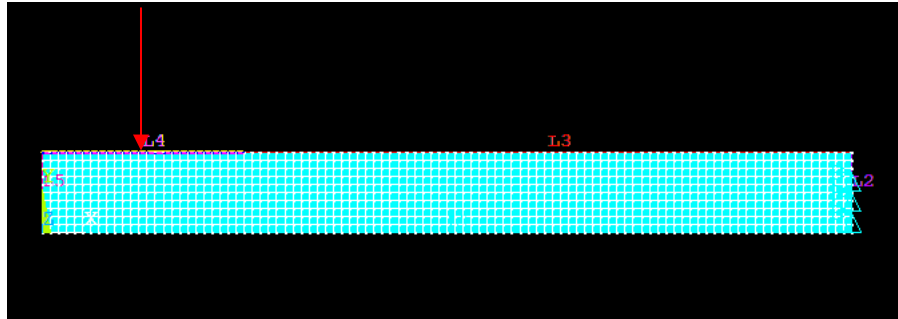
Select *ALL DOF* and click *OK*.

The Boundary conditions are displayed on the graphics window.

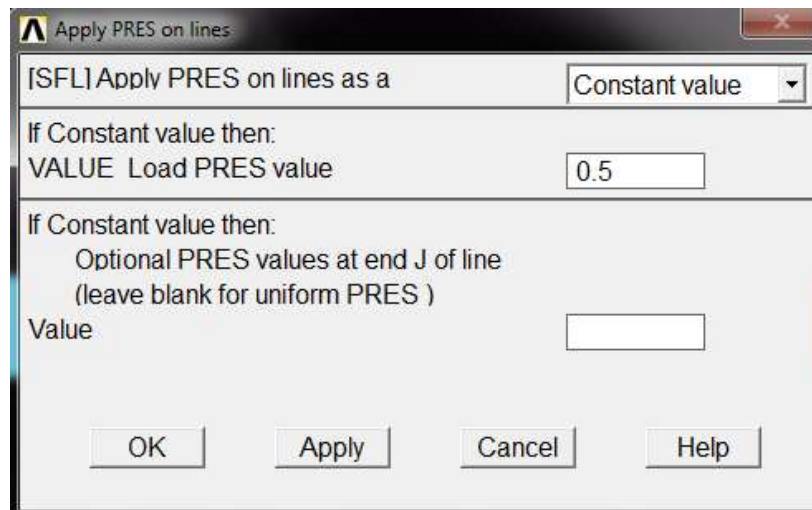
### Apply Pressure

Solution → Define loads → Apply → Structural → Pressure → On  
 Lines.  
 Select the left line on the top and click *OK*.

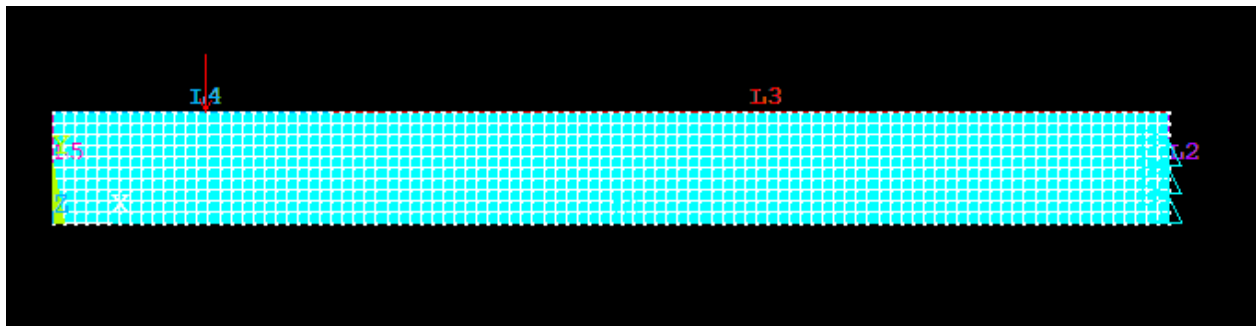




The following dialog box appears:



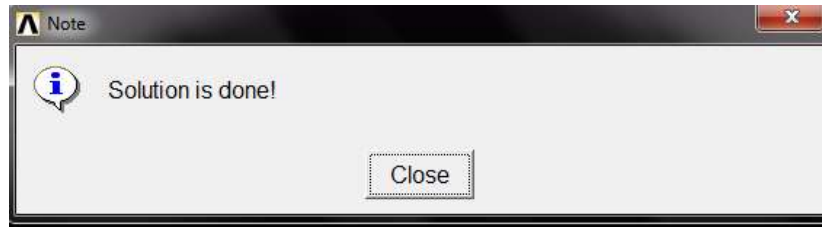
Set Load PRES value as 0.5 and click OK.  
The Graphic Display should look like the figure below.



### **STEP 7: Solve the problem**

Solution → Solve → Current LS.

A dialog box appears. Click *OK* to solve. Once the solution is done, the following dialog box appears:-

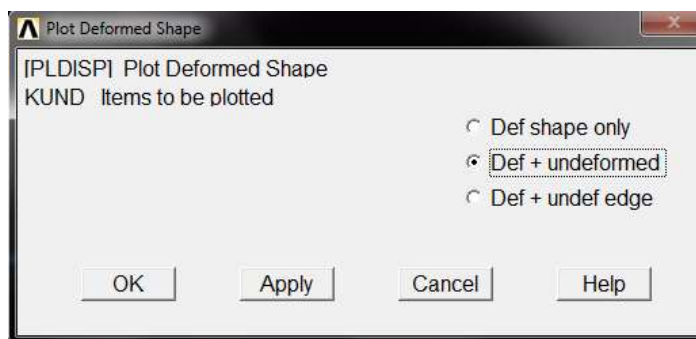


Close the dialog box.

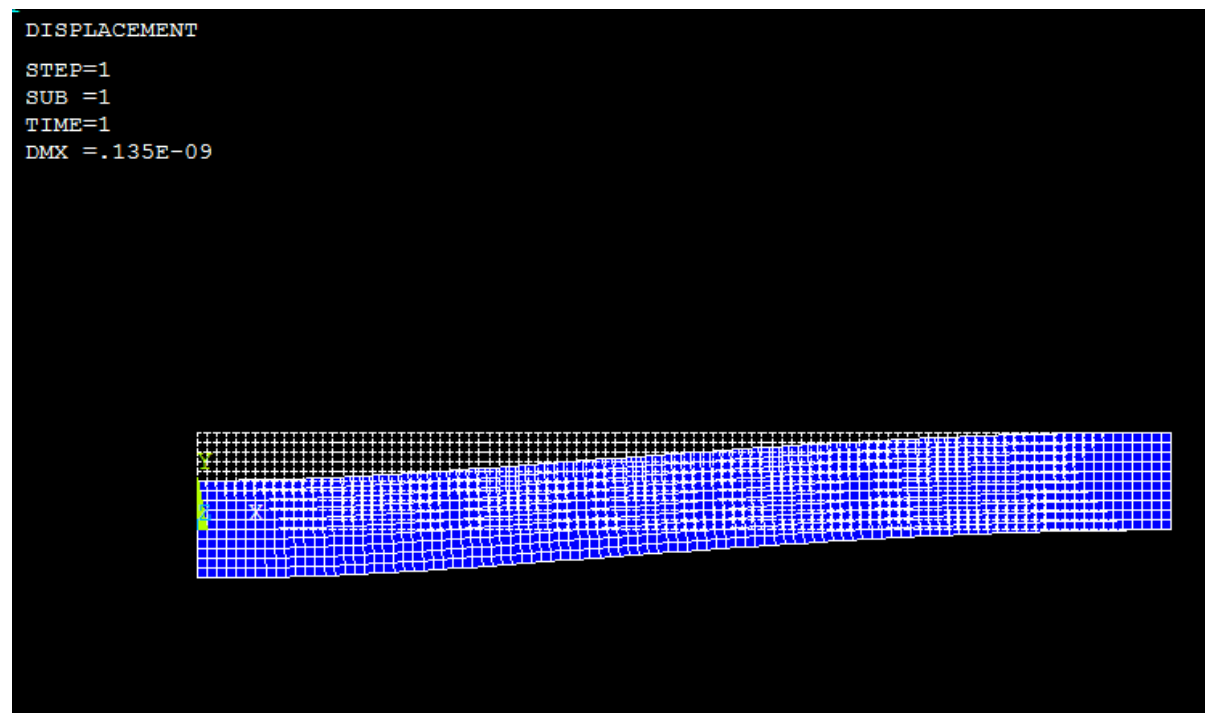
**STEP 8: View results.**

Main Menu → General Postproc → Plot Results → Deformed Shape

The following dialog box appears:



Select *Def + undeformed* and select *OK*.



## Nodal Solution

General Postproc → List Results → Nodal Solution → DOF Solution  
→ Y-Component of displacement

```
***** POST1 NODAL DEGREE OF FREEDOM LISTING *****
LOAD STEP=      1 SUBSTEP=      1
TIME=      1.00000 LOAD CASE=      0
THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM
      NODE      UY
      1111 -0.13419E-09
MAXIMUM ABSOLUTE VALUES
      NODE      ?
      VALUE -0.13533E-09
```

**The maximum displacement in Y direction happens at node 7, the value is 0.13533E-9 m.**

**Note: The node numbers in your model will depend upon the model building and meshing sequence you use and may be different from what is shown here.**

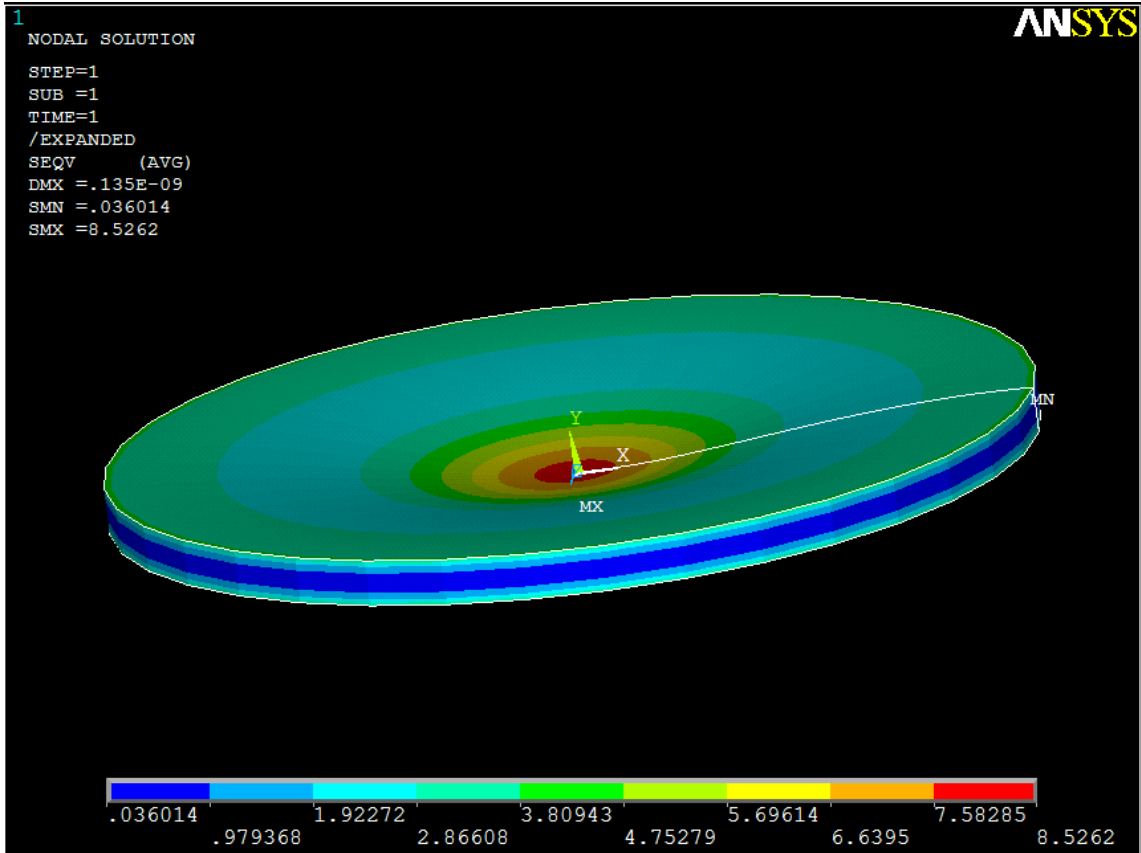
General Postproc → List Results → Nodal Solution → Stress → von Mises stress

```
      1109  0.13722  -0.46488E-01-0.43371  0.57093  0.50480
      1110  0.10430  -0.46465E-01-0.40074  0.50504  0.44906
***** POST1 NODAL STRESS LISTING *****
PowerGraphics Is Currently Enabled
LOAD STEP=      1 SUBSTEP=      1
TIME=      1.00000 LOAD CASE=      0
NODAL RESULTS ARE FOR MATERIAL      1
      NODE      S1      S2      S3      SINT      SEQU
      1111  0.72579E-01-0.46449E-01-0.36899  0.44157  0.39571
MINIMUM VALUES
      NODE      113      1      1      225      225
      VALUE -1.2504  -8.2071  -8.3310  0.41537E-01 0.36014E-01
MAXIMUM VALUES
      NODE      2      2      121      2      2
      VALUE  8.2382  8.1137  1.2449  8.5877  8.5262
```

**The maximum von Mises stress happens at node 2, the value is 8.5262 pa.**

To view the node number, go to Utility menu → PlotCtrls → Numbering → Toggle on the 'Node Numbers' option → OK (If the drawing disappeared, please go to, Plot (from utility menu on the top) Multi-Plots)

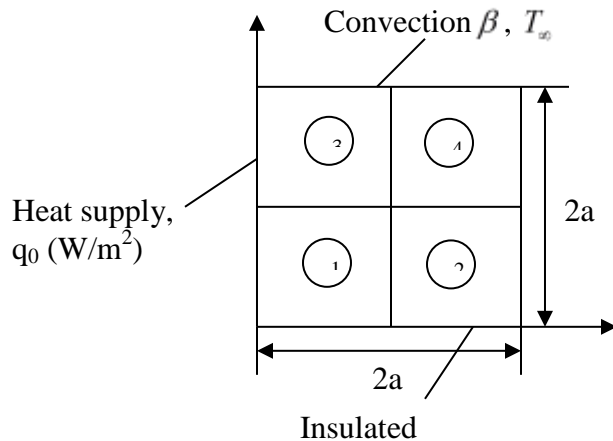
To view the full model, go to Utility menu → PlotCtrls → style → Symmetry Expansion → 2D Axi-symmetric → Full expansion → OK



# HEAT TRANSFER ANALYSIS

## Problem 8.30 (Text)

Consider steady-state heat conduction in a square region of side  $2a$ . Assume that the medium has conductivity of  $k$  (unit:  $W/(m \cdot ^\circ C)$ ) and uniform heat (energy) generation of  $g_0$  (unit:  $W/m^3$ ). Plot the contour of temperature distribution and list the nodal temperatures.



$$k = 30 \text{ W}/(m \cdot ^\circ C), \beta = 60 \text{ W}/(m^2 \cdot ^\circ C)$$

$$T_\infty = 0^\circ C, T_0 = 100^\circ C, a = 1 \text{ cm}$$

$$q_0 = 2 \times 10^5 \text{ W}/m^2, g_0 = 10^7 \text{ W}/m^3$$

### STEP 1:

Preferences → Thermal → OK

### STEP 2: Define Element Type

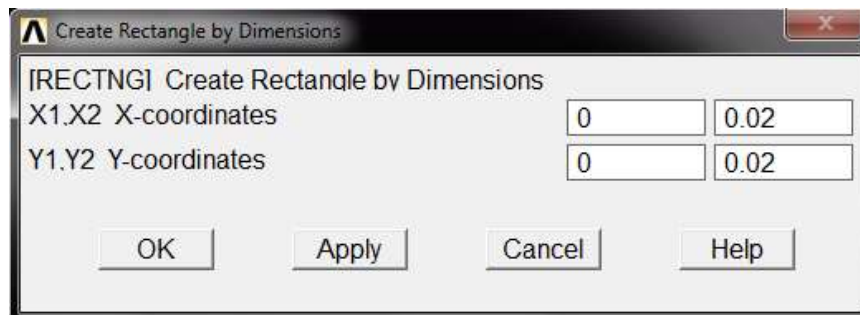
Preprocessor → Element Type → Add/Edit/Delete → Add... → Thermal Mass  
→ Solid → Quad 4 node 55 → OK → Close *Element Types* dialog box

### STEP 3: Define Material Properties

Preprocessor → Material Props → Material Models → Thermal → Conductivity  
→ Isotropic → KXX → 30 → OK → Close dialog box

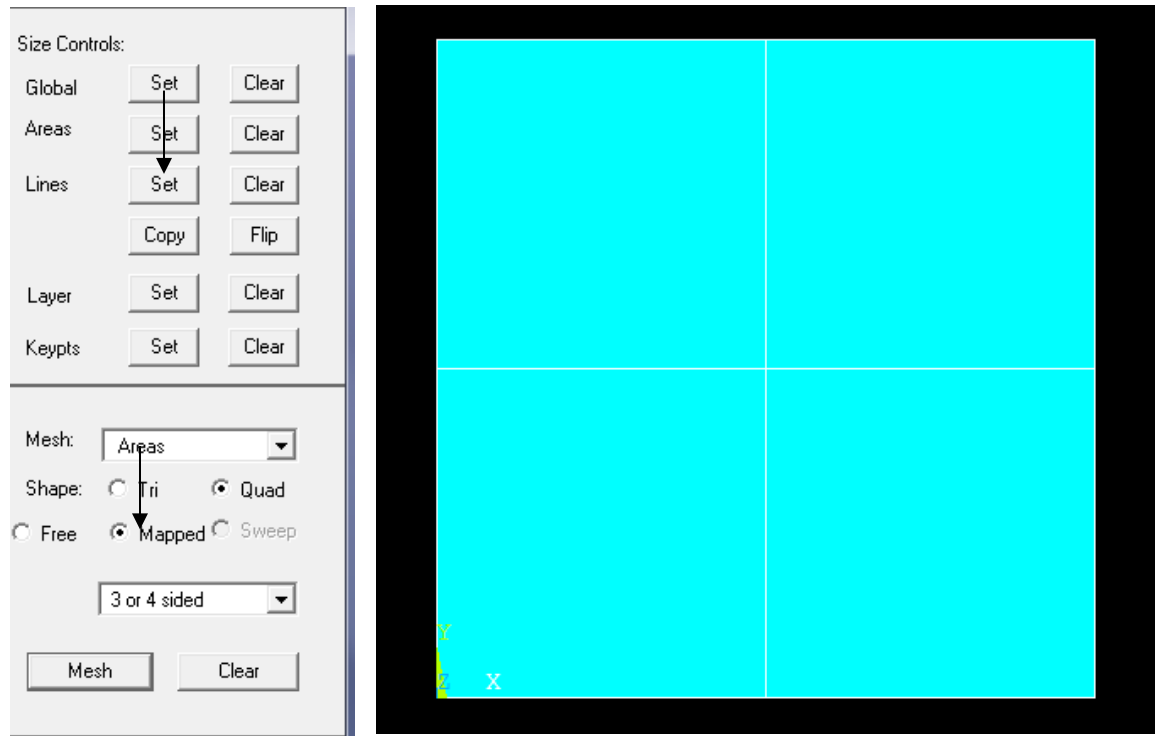
### STEP 4: Create the Model

Preprocessor → Modeling → Create → Areas → Rectangular → By  
Dimensions → Input the following values → OK



## **STEP 5: Mesh the Model**

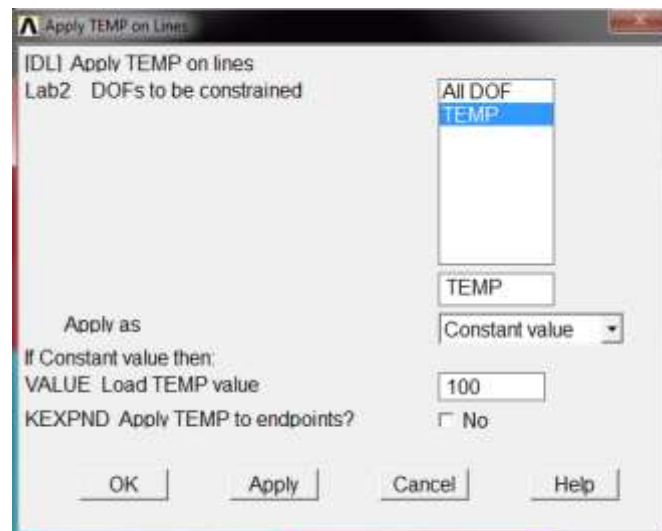
Preprocessor → Meshing → Mesh Tool → Lines → Set → Click the four edges of the square → OK → NDVI → input 2 → OK → Click “Mapped” → Mesh → Click the Square → OK → Close



## **STEP 6: Apply Boundary Conditions and Loads**

*Apply temperature on the right edge*

Preprocessor → Loads → Define Loads → Apply → Thermal → Temperature → On Lines → Click the right edge of the square → OK → Input the following values → OK



### ***Apply Heat Flux on the left edge***

Apply → Thermal → Heat Flux → On Lines → Click the left edge of the square →  
OK → Input the following values → OK

The dialog box is titled "Apply HFLUX on lines". It contains a dropdown menu labeled "[SFL] Apply HFLUX on lines as a" with "Constant value" selected. Below this, it says "If Constant value then:". There are two input fields: "VALI Heat flux" with the value "2e5" and "VALJ Heat flux" which is empty. A note says "Optional HFLUX values at end J of line (leave blank for uniform HFLUX)". At the bottom are "OK", "Cancel", and "Help" buttons.

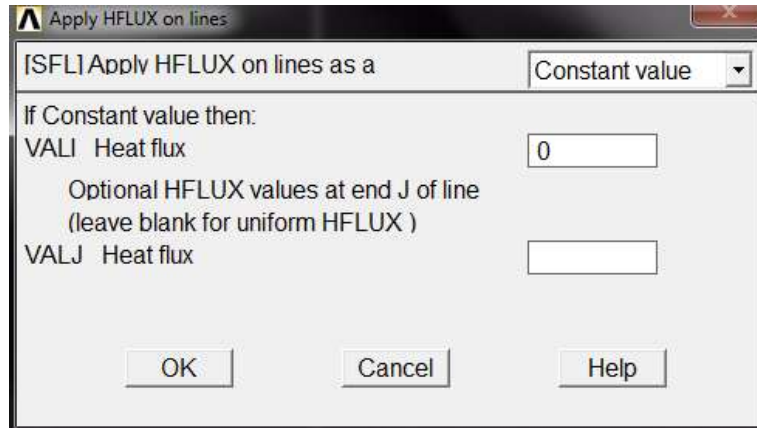
### ***Apply Convection on the top edge***

Apply → Thermal → Convection → On Lines → Click the top edge of the square →  
OK → Input the following values → OK

The dialog box is titled "Apply CONV on lines". It has two sections. The first section has a dropdown "[SFL] Apply Film Coef on lines" set to "Constant value", followed by "If Constant value then:" and "VALI Film coefficient" with the value "60". The second section has a dropdown "[SFL] Apply Bulk Temp on lines" set to "Constant value", followed by "If Constant value then:" and "VAL2I Bulk temperature" with the value "0". Below these are "Optional CONV values at end J of line (leave blank for uniform CONV)" with input fields for "VALJ Film coefficient" and "VAL2J Bulk temperature", both empty. At the bottom are "OK", "Cancel", and "Help" buttons.

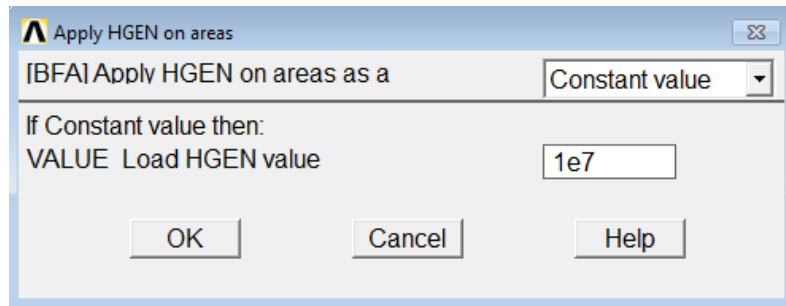
### ***Apply Insulated B.C. on the bottom edge***

Apply → Thermal → Heat Flux → On Lines → Click the bottom edge of the square →  
OK → Input the following values → OK



***Apply Volume heat generation over the whole modeling***

Apply → Thermal → Heat Generat → On Areas → Select the area created → OK (or middle click) → Input the following values → OK



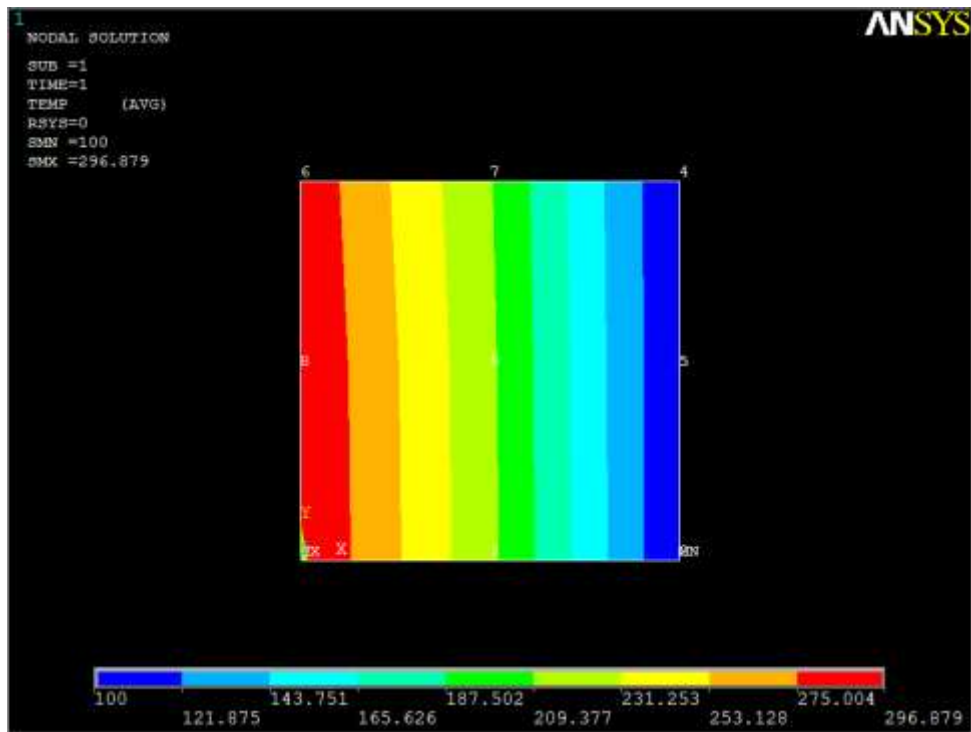
**STEP 7: Solve the problem**

Solution → Solve → Current LS.  
Click *OK* to solve.

**STEP 8: View results.**

General Postproc → Plot Results → Contour Plot → Nodal Solu → DOF Solution → Nodal Temperature → OK



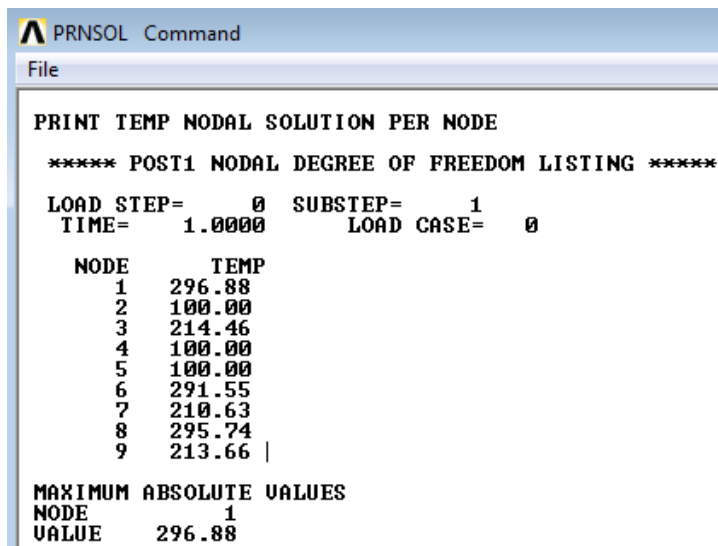


*View the temperature value at all nodes*

Main Menu → Plot → Mult-plots

Main Menu → Numbering → Node Numbers → Click “On” → OK

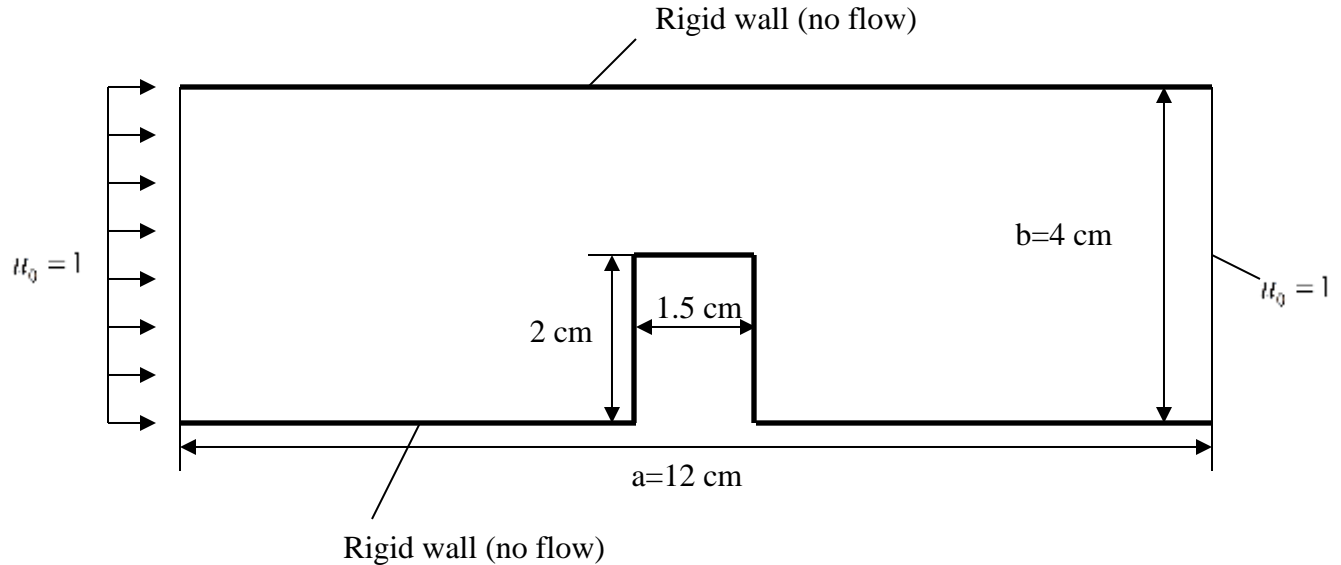
General Postproc → List Results → Nodal Solution → DOF Solution → Nodal Temperature → OK



# FLUID MECHANICS

## Problem 8.42 (Text)

For the fluid problem shown below, determine the steady velocity distribution of the domain

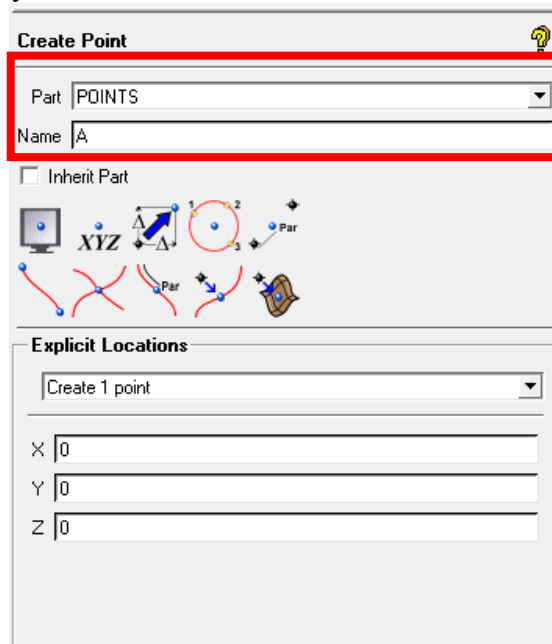


### STEP 1: Open ICEM CFD & Specify file name & its path

Start → Program → ANSYS 15.0 → Meshing → ICEM CFD 15.0 → File → New Project → Browse to a working directory → Save

### STEP 2: Create the vertices to create lines and the surface of the domain

Geometry (function tab) → Create Point → Explicit Coordinates → Enter the X, Y, Z coordinates → Apply



Before creating a point, do not forget to change part name to POINTS as show above and keep the name unchanged during these points' creation.

**Note:** If your version of ICEM doesn't display the same create point options as above, adjust the following settings.

1. Settings → Selection → Auto pick mode should be turned OFF.
2. Settings → Geometry Options → Name new geometry must be turned ON.
3. Settings → Geometry Options → Replace same name item must be turned ON.
4. Settings → Geometry Options → Inherit Part Name → Create new must be toggled ON.
5. Settings → Geometry Options → Create surface topology must be turned OFF.

Create the following set of points with the names given below:

NAME	A	B	C	D	E	F	G	H
X	0	0.0525	0.0525	0.0675	0.0675	0.12	0.12	0
Y	0	0	0.02	0.02	0	0	0.04	0.04
Z	0	0	0	0	0	0	0	0

In the model tree, Geometry → (right click) Points → Show Points Names, all the points' name created could be showed, and make sure no duplication of same points.

### **STEP 3: Create lines from the vertices created**

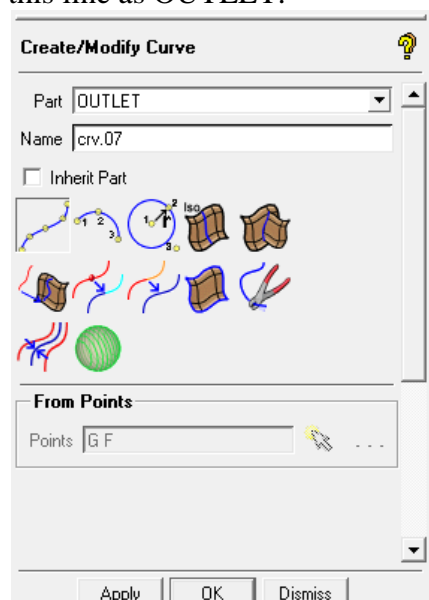
Geometry (function tab) → Create/modify curve → From points → Change part name to INLET → At the blank Points, fill in A H → APLY (You can also select points A & H → MIDDEL=DONE)

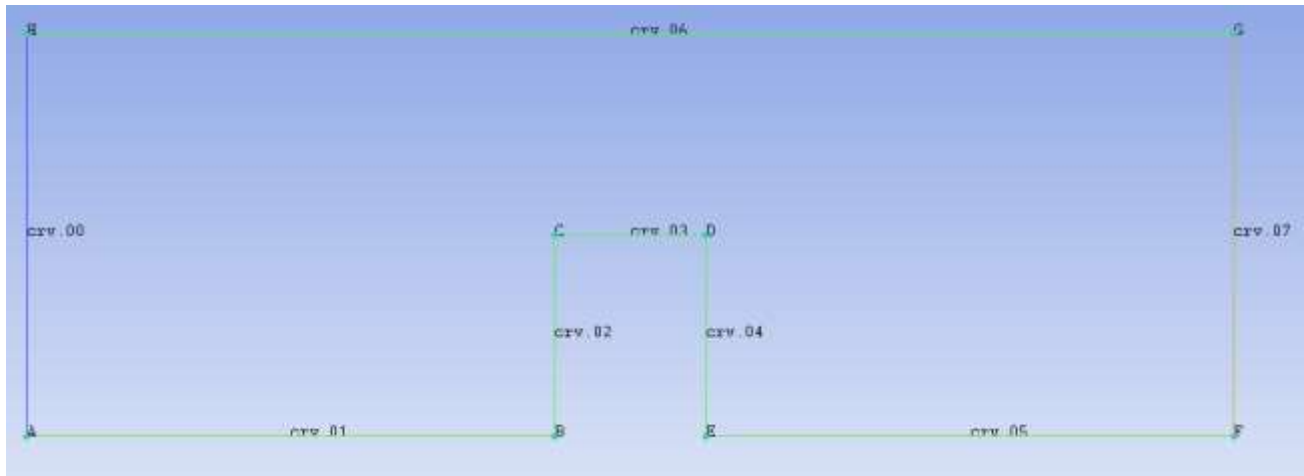
Similarly, change part name to WALL → select AB, middle click → select BC, middle click → select CD, middle click → select DE, middle click → select EF, middle click → select HG, middle click → Dismiss, all these lines are belonged to part WALL.

**Note:** Do not change part name when creating WALL lines.

In the model tree, right click on Geometry → Curves → Show Points Names, all the curves' name created could be showed, and make sure no duplication of same curve.

Similarly, join GF, and name this line as OUTLET.

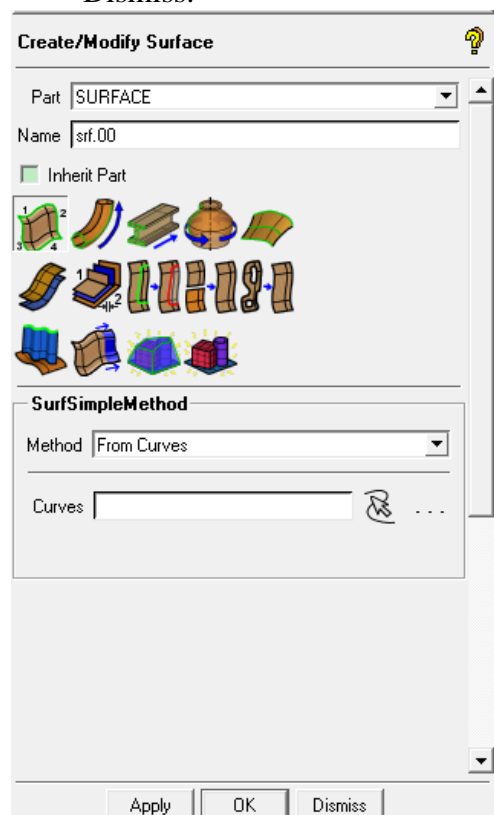




Note: After every step is done, click File → Save Project to make sure the changes made are saved in time.

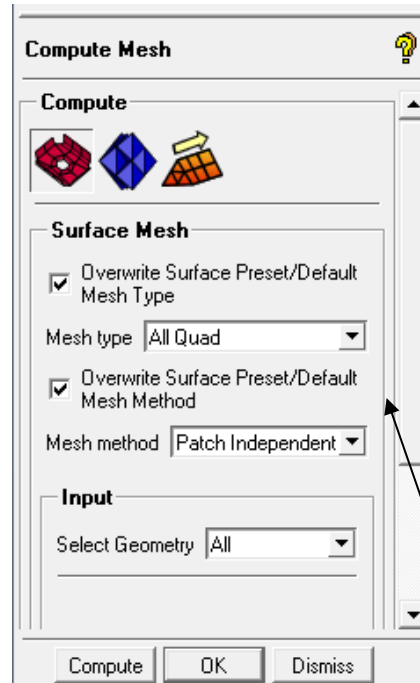
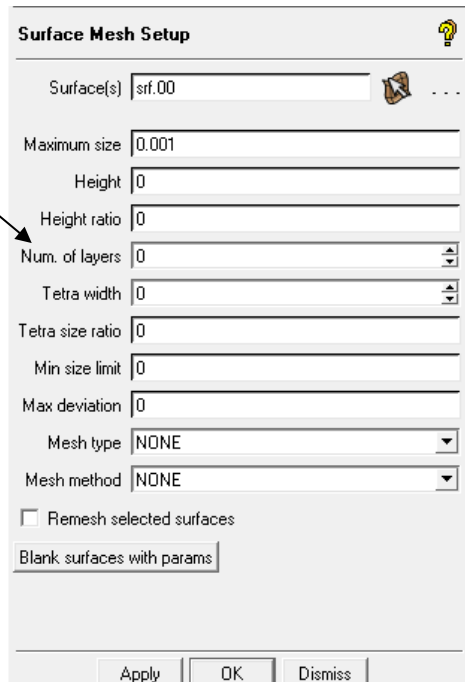
#### **STEP 4: Create the surface**

Geometry (function tab) → Create/modify surface → Change Part Name to SURFACE → Simple surface → Methods → From curves → Select all the lines created → Middle click → APPLY → Dismiss.

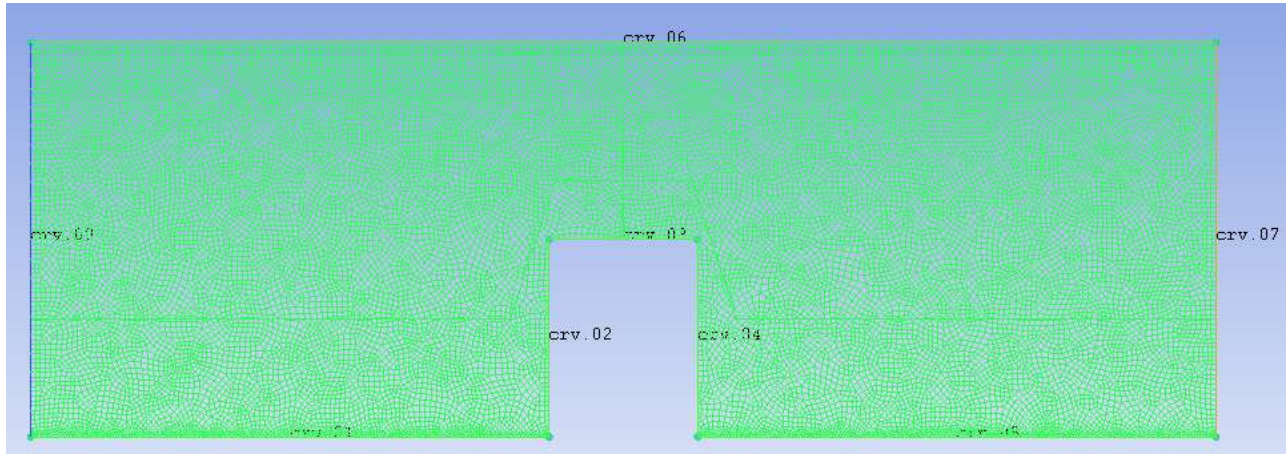


#### **STEP 5: Surface meshing**

Mesh (function tab) → Surface Mesh Setup → Surface → Select the SURFACE created → Middle click → Maximum size → 0.001 → OK.



Mesh (function tab) → Compute Mesh → Surface Mesh Only → Check on Overwrite Surface Preset/Default Mesh Type → Mesh Type → All Quad → Check on Overwrite Surface Preset/Default Mesh Method → select Patch Independent → Select Geometry → ALL → Compute.

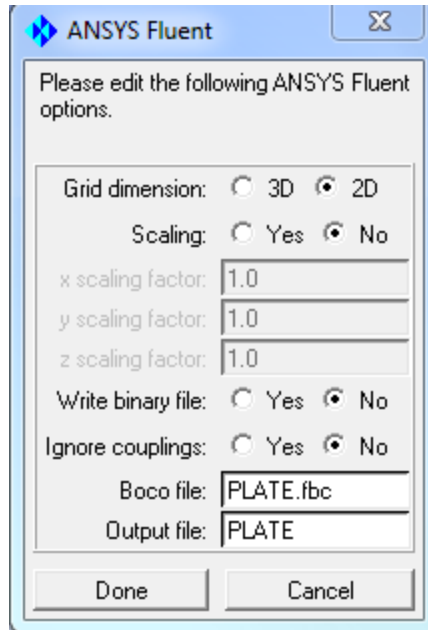


### **STEP 6: Select the solver**

Output (function tab) → Select solver → Output Solver=ANSYS Fluent → Common Structural Solver=ANSYS → OK

### **STEP 7: Saving the mesh file**

Output (function tab) → Write input → save project first=YES → Open the .uns file → Save the file in 2D → Change output file name to PLATE → DONE.



The extension of the mesh file would be .msh, which can be read by most of the CFD solvers including ANSYS CFX, Star CD, OpenFOAM and FLUENT.

### **STEP 8: Import mesh file into CFX**

Start → Program → ANSYS 15.0 → Fluid Dynamics → CFX 15.0 → CFX-Pre 15.0  
 File → New case → General → OK  
 File → import → mesh → Select the .msh file just created (Files of type choose FLUENT (\*.cas, \*.msh) ) → Open.

### **STEP 9: Set up steady-state case definition**

In the model tree, double click **Analysis Type** → select steady state → OK  
 In the model tree, double click **Default Domain**,  
 Basic setting (tab) → Change material to Air at 25 degree C → Ref. Pressure= 1 atm.  
 Fluid Models (tab) → turbulence → (none) laminar → Combustion → None → OK.

### **STEP 10: Define boundary conditions**

In the model tree, under **Simulation** → **Flow analysis** → right click Default domain → Insert → Boundary → Type Inlet → OK  
 Enter the following:  
 Basic settings (Tab) → Boundary Type=Inlet → Location=INLET →  
 Boundary details (Tab) → Flow Regime = Subsonic → Mass & Momentum → Option=Cart. Vel. Components → U=1 m/s, V=W=0 m/s.

In the model tree, under **Simulation** → **Flow analysis** → right click Default domain → Insert → Boundary → Type OUTLET → OK  
 Enter the following:  
 Basic settings (Tab) → Boundary Type=outlet → Location=OUTLET  
 Boundary details (Tab) → Flow Regime = Subsonic → Mass & Momentum → Option=Cart. Vel. Components → U=1 m/s, V=W=0 m/s.

In the model tree, under **Simulation** → **Flow analysis** → right click Default domain → Insert → Boundary → Type symmetry → OK

Enter the following:

Basic settings (Tab) → Boundary Type=symmetry → Location=Primitive 2D C, Primitive 2D D (use 'Shift' key) → OK

In the model tree, under **Simulation** → **Flow analysis** → right click Default domain → Insert → Boundary → Type wall → OK

Enter the following:

Basic settings (Tab) → Boundary Type=wall → Location= wall

Boundary details (Tab) → Mass & Momentum → Option=No Slip Wall → Apply OK.

### **STEP 11: Set up solver controls**

In the model tree, under **Simulation** → **Flow analysis** → Solver → Double Click Solver Control.

Advection Scheme → High resolution

Minimum iterations → 1

Maximum iterations → 300

Convergence criteria:

Residual type → RMS

Residual target → 1E-4 → OK

### **STEP 12: Run**

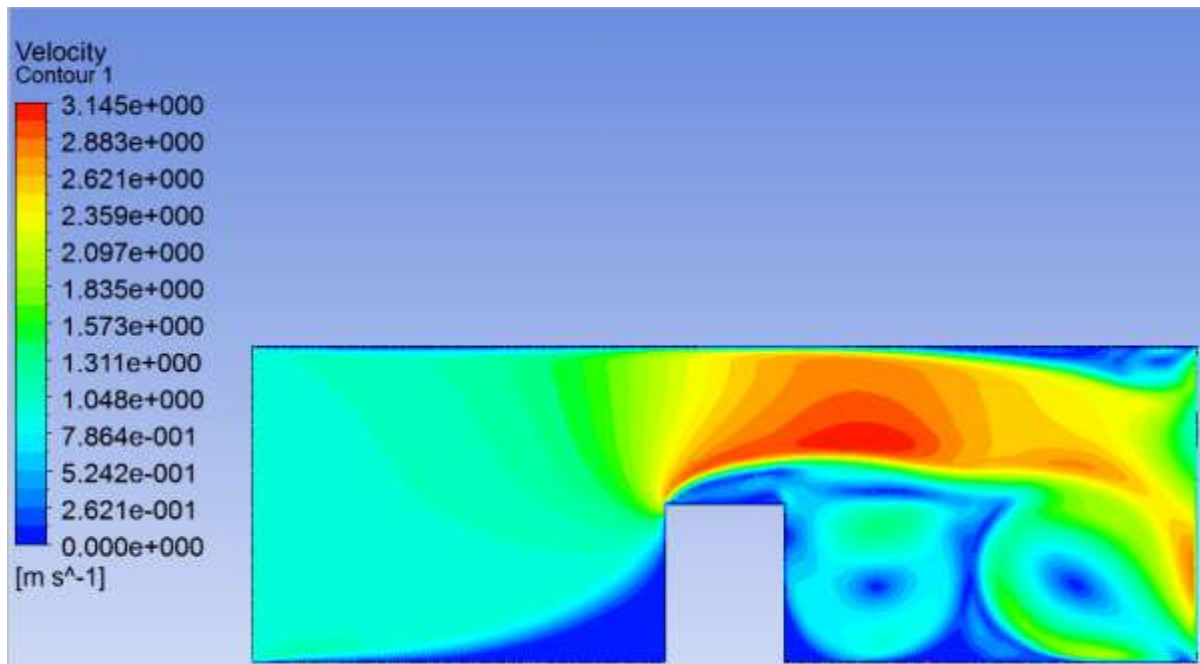
Right click on the drawing window → Start solver → Define Run → Save → Start Run.

### **STEP 13: Post processing of results.**

Menu (On CFX-Solver Manager) → Launch CFD-Post with a Results File → OK.

Menu → Insert → Contour → Name: Contour 1.

Geometry (Tab) → Location=symmetry → Variable=velocity → Number of contours → 25 → Apply.



**Velocity Contour**

**Note:** The results will vary slightly depending on the mesh size and element type