Getting started with ANSYS

ANSYS is a general purpose finite element program. It is used widely in the industry. It is marketed for both the Windows and the Unix operating systems. However in the College of Engineering it is available only on the HP-UX workstations. If you are using a Windows NT computer you must first use an X Windows application to establish a remote connection to a Unix workstation. When connected you can then access ANSYS as if you are working directly on a Unix workstation.

Launching ANSYS

At the command prompt type **ansys** and you will see the ANSYS launcher. It is a simple window containing buttons that activate the ANSYS program and other auxiliary programs.

When you click on **ANSYS** (using the left mouse button), you will see a dialog box containing several options. Leave all other options with their default values except for the following two.

> a. **Working directory**: this is the directory from which the ANSYS run will be executed. All database files will be saved in this directory.

> b. **Initial jobname**: this is the jobname, which will be used as the file name prefix for all files generated by the ANSYS run. The default jobname is **file**.

After entering the appropriate information press **Run** on the interactive Set Up dialog box. Ansys will create several windows and may ask questions related to licensing and then will be ready for performing a finite element analysis. Alternatively, you may launch ANSYS by clicking on **ANSYS NOW,** the program will run with the current (or the default) setting.

You can create models for analysis either graphically by clicking mouse or by entering text on a command line. The default is the GUI (Graphical User Interface). In the Command line mode you can type the input data line by line in a small window titled **ANSYS Input**. You can also prepare an input data file for either a part of the model or the entire model by using a text editor (such as *pico*) and then type **/input, inputfile** in the ANSYS command line window. Here the **inputfile** is the name of the input data file. If you enter part of the input data, you can continue typing in the rest of it. This is called interactive run.

Alternatively, you can prepare the complete input data file and then click **Batch** button from the Set Up dialog box. This *batch mode* is very useful for solving large problems. Click the button after the **input file name** option and choose your input file in the popup window. A **jobname.out** file will be generated by ANSYS to store all the output results. When working with the command line interface, a line starting with one of the following three forms is considered a comment and is ignored during execution.

> (1) /COM
> (2) C***
> (3) !

Basic Features of ANSYS

The ANSYS program is organized into several processors. A typical ANSYS analysis involves using three processors.

1. *Preprocessing* (PREP7 processor), where you provide data such as the geometry, materials and element types to the program.
2. *Solution* (SOLUTION processor), where you define analysis type and options, apply loads and initiate the finite element solution. Boundary conditions maybe specified in either PREP7 or SOLUTION.
3. *Postprocessing* (POST1 or POST26 processors), where you review the results of the analysis through graphics display and tabular listings. POST26 is for dynamic analysis where the time-history of designated variables are available.

Interacting with AYSYS

The Graphics User Interface (GUI) of ANSYS consists of six main windows. These windows can be resized, moved and closed. They are briefly described below.

1. ANSYS Utility Menu. This menu is usually near the top of the screen. It contains ten pull-down menus for performing common operations.

2. ANSYS Input. This small window is usually directly below the Utility Menu. This window can be used for typing commands or entering numerical data during an interaction run.

3. ANSYS Toolbar. This small window is usually at the right of the Input window. It contains buttons for commonly used operations such as saving data base to disk and quitting the program. To safeguard the analysis in case of a program crash, the **Save-DB** button should be used frequently to save the current work to the disk.

4. ANSYS Main Menu. This is perhaps the most frequently used Menu in an interactive run. It provides access to the primary ANSYS functions through a series of cascading menus. This menu is usually located on the left side of the screen directly below the Input window.

5. ANSYS Graphics. This window, covering most of the remaining screen, is used for graphic displaying.

6. ANSYS Output: Any text messages generated by ANSYS are placed in this window. It is usually hidden behind the Graphics window. However, it is strongly encouraged to periodically review the information in this window to see exactly how ANSYS is interpreting the information supplied through the under interface, and what is wrong.
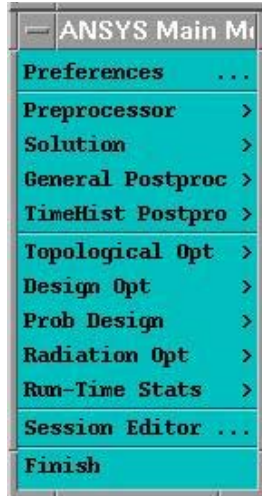
ANSYS main menu

ANSYS has a sophisticated GUI consisting menus, dialog boxes and graphic picks. However, each GUI action has a corresponding text command. This feature makes ANSYS convenient for situation that require several repetition of a series steps. You can prepare a text files containing these commands using any text editor and then type **/input**, *filename* in the ANSYS command line window.

During an interactive run, ANSYS automatically writes commands equivalent to user's graphical action to a file called the log file (*filename.log*). This is a text file and is saved in the working directory. This file is very useful if you want to repeat the same analysis with perhaps slightly modified values for some of the parameters. If you are interested in using this file you should rename it first because it will be overwritten an soon as you enters ANSYS the next time. It is also important to remember that the log file contains everything that you did in an interactive session, including mistakes and Exit command that you used to get out the of program. Therefore some editing is always necessary before you can use it as an input file.

Conventions Used in Describing Menu Selections

ANSYS uses a combination of cascading menus, input windows and dialog boxes. In the lab handouts required selections to perform a specific task are shown through a series of key words separated by ">" symbol. For example

ANSYS Utility Menu > PlotCtrls > Capture Image > file > Print > Print to: [**lp -dlaser**] > OK

This line means that we start with the ANSYS Utility Menu and choose PlotCtrls. This choice leads to a submenu from which we select Capture Image. This menu opens up a window with a file menu. From this menu we select Print option which leads to the Print to: [**lp -dlaser**] dialog box which is closed by clicking on the OK button.

Different mouse buttons also perform different tasks. Most ordinary mouse clicks are performed using the left mouse button. If another mouse button must be used it is shown by attaching a small figure showing three rectangles. The dark rectangle corresponds to the button that must be used for that required click. For

example a click with the middle mouse button is indicated by ▯▮▯.

Re-entering ANSYS

The database created by a previous run can be resumed at any time. Assuming a pervious database file **truss.db** exists, then the following sequence must be used in order to continue working with the model.
- Pick **Run Interactive Now** from the launcher menu.
- ANSYS Utility Menu > File
- Resume from ... > Files [**truss.db**] > OK

Printing Graphics Files

There are two ways to print graphisc:
- From the ANSYS Utility Menu choose PlotCtrls > Capture Image > file > Print > Print to: **[lp -dlaser]** > OK
- ANSYS Utility Menu > PlotCtrls > Hard Copy > OK

With the second option a postscript file will be created and named as *file.ps00.* After exiting ANSYS, you can print this postscript file on a laser print by typing:
        lp -dlaser *file.ps00*

How to get Help

Ansys on-line help system is very convenient. For X-window version 5.3 or later, here is how to access on-line help system. Double click on ANSYS Help System and it will bring the document navigation window. Navigate menu provides ways to navigate through the document such as table of contents, word search, go back and forth. Also, notice that some words or phrases are boxed, and if you double click on the boxed words or phrases, it will take you to the source of more information on the items. They are called hypertext links. For ANSYS 6.0, you can also access the online documentation by clicking on **?** from the launcher.

In addition, the Engineering library has a complete set of ANSYS user manuals, including *procedures, commands*, *element* and *theory*. They are almost the same as the on-line help documents.

Files Created by ANSYS

ANSYS creates a large number of files, some are useful to users while others are for use only by the program itself. Depending on their use, files may be written in text or binary form. Most generated files have the name *filename.ext,* where *filename* default to jobname, and *ext* is the extension that identifies the contents of the file.

Ext	Type	Description
Db	Binary	Database
Rst	Binary	Results
Err	Text	error and warning messages
Log	Text	command input history
Out	Text	output results

The output file (*filename.out*) is a text file that contains all the print out you request in an interactive session and the error messages. It is always good habit to review this file to see how the program is interpreting the user commands and how the analysis is proceeding. The database file (filename.db).is a binary file that contains all the data defining the model and the current results. This file is created when you exiting the program with **Save-DB,** or when you issue the **Save** command. The results file (*finename.rst*) contains all the data that allows you to resume the analysis from this point.

MENG412: Computer Aided Design
Spring 2006

<u>ANSYS Primer</u>

Ansys is a commercial finite element program designed to handle large-scale, complex loading, multi-physics problems {fluid flows, stress analysis of solid structures, heat transfer, electromagnetics}. In this course, we will only look at structural analysis problems.

This primer will tell you how to get started with Ansys. You can also get online help and look through some of the tutorial examples provided within Ansys.

Launching Ansys

Windows Start Menu: Programs→Class Software→Ansys →Run Interactive Now

You can set Ansys to offer you options only for structural analysis by setting:

Ansys Main Menu: Preferences (select Structural from pop-up Window)

When you launch Ansys, several windows and menu bars open. These include:
- Ansys Utility Menu bar located on the top of your screen
- Ansys Toolbar
- Ansys Input Window
- Ansys Main Menu
- Ansys Graphics Window
- Ansys Output Window (may be located behind the Graphics Window)

As you run Ansys, other windows and menus might open from time to time.

Saving your Ansys session

There are two save commands that are useful to know. It is important to save your work periodically using:

Ansys Toolbar: save_db

If you mess up at a later step, you can always resume using:

Ansys Toolbar: resume_db

from the previously saved step.

Note also that save_db does not mean that you can quit Ansys and return at a later time to resume. Someone else might have logged on and started a new run. In order to properly save your work you must use:

Ansys Utility Menu: File→Save as {input filename and store in your own directory or on your own storage media}.

Making Hardcopies of Ansys Results

You can get a hardcopy of specific Ansys results in several ways. There is a printer icon that lets you capture images to file or to a printer. Under the Ansys Utility Menu: PlotCtrls, there are other hardcopy options. It is probably wise to make hardcopy files (say as JPEG or TIFF) and to then import these into your word processor.

Exiting Ansys

It is extremely important that you do not leave Ansys hanging. There are only a limited number of licenses available (A license manager program doles out license tokens and retrieves them when done) and hanging licenses are a nuisance to everybody!

You should exit Ansys using:

Ansys Toolbar: Quit (select "Save …" or "No Save" from pop-up Window as desired)

If you "hang" Ansys inadvertently (by accidentally closing Windows, by crashing your computer etc), please let us know immediately.

Finite Element Simulations

Any finite element simulation involves three stages:

Preprocessing: In this stage, you do the following:

1. Create the structural geometry.
2. Choose the type of material to use
3. Choose the type of elements to use
4. "Mesh", ie discretize the continuous structure into a discrete structure composed of nodes and elements.

Solving: In this stage, you do the following:

5. Apply a particular set of boundary conditions and loads to your structure
6. Solve for the displacements/stresses/strains in the structure.

Postprocessing: In this stage, you do the following:

7. Post-process your analysis results to monitor the deformed shape of the structure, to get the maximum normal and shear stresses, to obtain plots of displacements, stresses etc along paths that you create, etc.

It is best to learn how to navigate through Ansys by means of examples.

Example: Tensile Loading of a Plate With a Hole

Consider a large, thin, square plate of sides s containing a small circular hole of radius **a** at its center. The plate is subjected to simple tensile stress of $\sigma_o = 1\times10^6$ N/m^2 on its vertical edges. The sides of the plate are s = 2m and the radius of the hole is a = 0.1m. The plate is 0.001m thick and is made of aluminium with Young's modulus E = 0.7×10^{11} N/m^2, and Poisson's ratio $\nu = 0.3$.
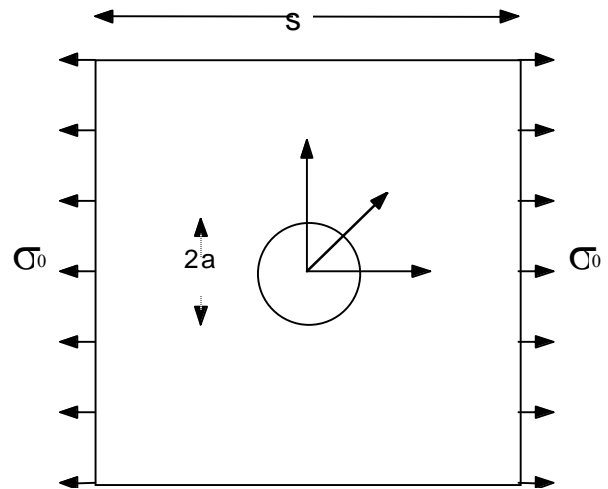
Fig 1.: Plate with a hole

Preprocessing:

Step 1. <u>Define Geometry</u>: From the symmetry of the problem, it is sufficient for us to model only one quadrant of the structure.

Ansys Main Menu: Preprocessor
    - Modeling- Create
      -Areas-Rectangle
        By 2 Corners
         {Enter:WP X = 0; WP Y = 0; Width = 1; Height = 1}
         Apply to create rectangle
         OK to close dialog box

Ansys Main Menu: Preprocessor
    - Modeling- Create
      -Areas-Circle
        Solid Circle
         {Enter:WP X = 0; WP Y = 0; Radius = 0.1}
         Apply to create circle
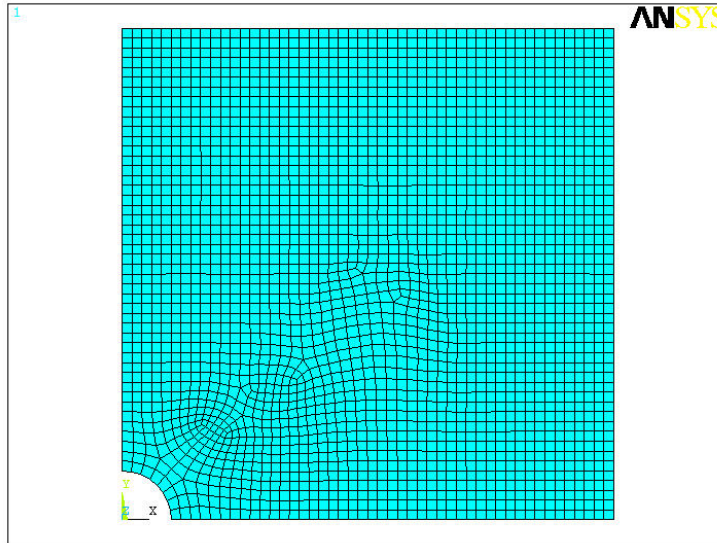   OK to close dialog box
Ansys Main Menu: Preprocessor
    - Modeling-Operate
      -Booleans-Divide
        -Area by Area-
         cursor pick rectangle Apply
         cursor pick circle Apply
         OK to close dialog box
Now is a good time to save your work: Ansys Toolbar: save_db

Step 2. <u>Define Element Type</u>:

Ansys Main Menu: Preprocessor
                              Element Type
                                    Add/Edit/Delete
                                          Add {select 'Solid' 'Quad 4node 42' OK}
                        {In options, pick 'plane stress w/thick'} OK

Ansys Main Menu: Preprocessor
                              Real constants
                                    Add/Edit/Delete: Add:
                                    {select element type chosen, type '0.001' for thickness}

Step 3. <u>Define Material Type</u>:

Ansys Main Menu: Preprocessor>Material Props>Material Models>Structural (double click)>Linear>elastic>isotropic

        Input Mat'l Properties {EX=Young's: '0.7E11'; PRXY=Poisson's Ratio '0.3'; OK}

Now is a good time to save your work: Ansys Toolbar: save_db

Step 4. <u>Create Mesh</u>:

Ansys Main Menu: Preprocessor: Meshing
                              Meshtool {a new Meshtool Window opens}
                                 Meshtool: {select Size Controls: Global ; Set}
                                                               {set Size '0.02'; OK}
                                 Meshtool: Mesh {a new Mesh Areas window opens}
                                   Mesh Areas: {cursor select plate area to mesh; Apply; OK}

Fig. 2: Ansys-generated mesh for the plate-hole problem

<u>Solving:</u>

Step 5. <u>Apply Displacement Constraints</u>: From symmetry, the bottom edge cannot displace vertically and the left edge cannot displace horizontally.

Ansys Main Menu: Solution
              -Define Loads-Apply
               -Structural-Displacement
                    On Line {cursor pick bottom edge; Apply}
                                  {select 'UY'; Displacement value '0'; Apply}
Ansys Main Menu: Solution
              -Define Loads-Apply
               -Structural-Displacement
                    On Line {cursor pick left edge; Apply}
                                  {select 'UX'; Displacement value '0'; Apply}

Step 6. <u>Apply Loads</u>: The applied tensile load on the right edge can be treated as negative pressure is tensile load!

Ansys Main Menu: Solution
              -Define Loads-Apply
               -Structural-Pressure
                    On Line {cursor pick right edge; Apply}
                                {Pressure value '-1e6'; Apply} OK

Now is a good time to save your work: Ansys Toolbar: save_db

Step 7. <u>Solve</u>:
Ansys Main Menu: Solution
              -Solve-Current LS (load step)

Close Solution Done Window

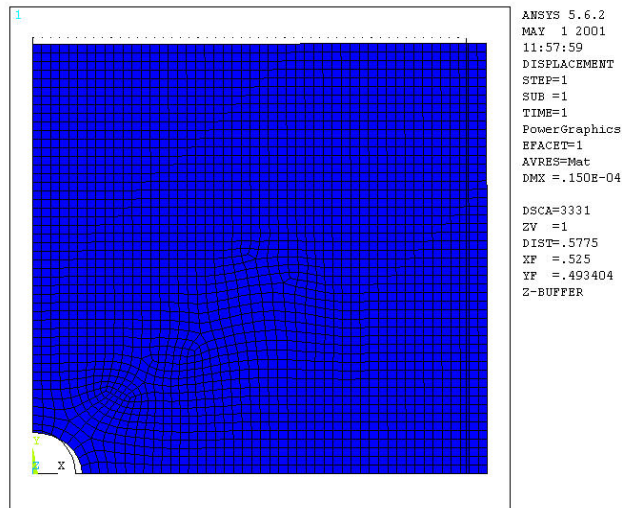## Post-processing:

Step 8. Plot Deformed Shape:
Ansys Main Menu: General Postprocessor
                        Plot Results
                                Deformed Shape {select Def + Undef edge}

Fig 3: Deformed shape overlaid on outline of undeformed shape
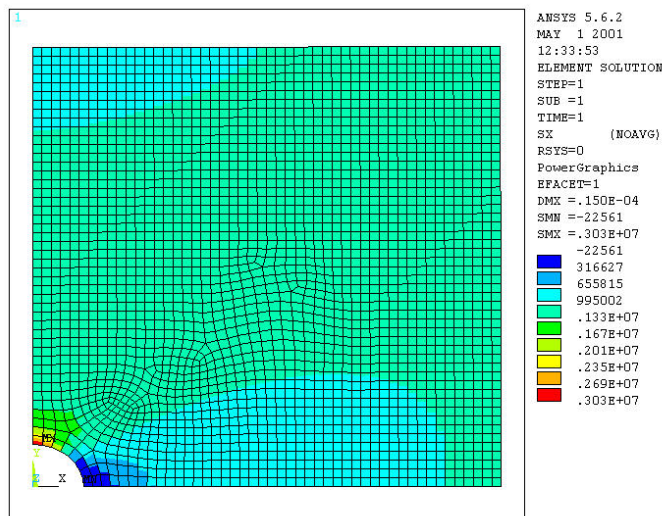
Step 9. Contour Plots of Stresses:

Let us look at σ_{xx} throughout the structure as a contour plot.

Ansys Main Menu: General Postprocessor
                        Plot Results
                         -Contour Plot-Element Solution {select 'Stress'; 'SX'; OK}

Fig. 4: Contour plot of σ_{xx}.

Step 10. <u>Plots on User Defined Paths:</u>
Let us plot σ_{xx} along the left and bottom edges.
Ansys Main Menu: General Postprocessor
      Path Operations
       Define Path
        By nodes {cursor select all nodes on left edge; Apply}
           {define path name: 'leftedge'; OK}

Ansys Main Menu: General Postprocessor
      Path Operations
       Map onto path {select 'Stress'; 'SX'}
       Plot path Item>On Graph>(select appropriate item to plot)
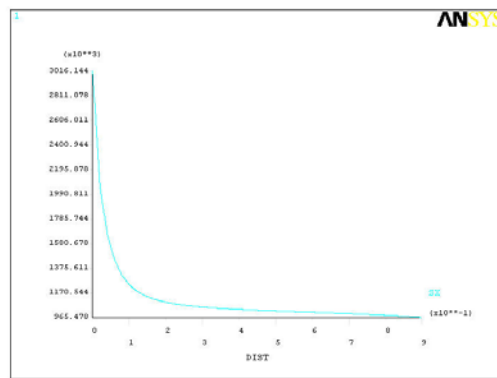
Fig 5: Plot of σ_{xx} along the left edge – note stress concentration close to the hole.

Ansys Main Menu: General Postprocessor
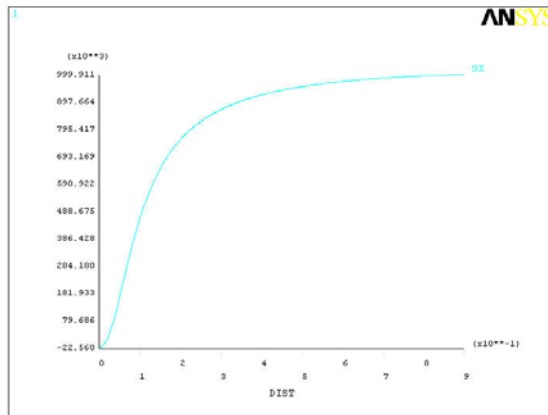   Path Operations
       Define Path
        By nodes {cursor select all nodes on bottom edge; Apply}
           {define path name: 'bottom'; OK}
Ansys Main Menu: General Postprocessor
      Path Operations
       Map onto path {select 'Stress'; 'SX'}
       Plot path Item>On Graph>(select appropriate item to plot)

Fig 6: Plot of σ_{xx} along the bottom edge – note stress varies from zero at the hole to the applied value far away from it.

Example: Pipe Under Pressure

Consider a very long hollow pipe with internal radius r_i = 1.0m and outer radius r_o=1.2m. The pipe is under internal pressure p_i = 1MPa.
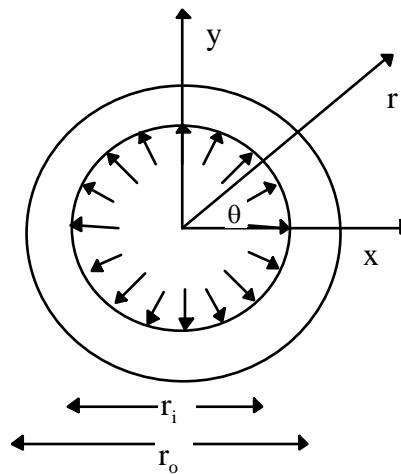

Figure 7: Pipe Under Internal Pressure

Step 1. <u>Define Geometry</u>: From the symmetry of the problem, it is sufficient for us to model only one quadrant of the structure.

Ansys Main Menu: Preprocessor
                - Modeling- Create
                    -Areas-Circle>Partial Annulus:
                {Enter:WP X = 0; WP Y = 0; Rad-1 = 1; Theta-1 = 0; Rad-2=1.2; Theta-2=90}
                        Apply to create annulus
                            OK to close dialog box

Now is a good time to save your work: Ansys Toolbar: save_db

Step 2. <u>Define Element Type</u>:

Ansys Main Menu: Preprocessor
                           Element Type
                                 Add/Edit/Delete
                                         Add {select 'Solid' 'Quad 8node 82' OK} Close

Step 3. <u>Define Material Type:</u>

Ansys Main Menu: Preprocessor>Material Props>Material Models>Structural (double click)>Linear>elastic>isotropic

        Input Mat'l Properties {EX=Young's: '0.7E11'; PRXY=Poisson's Ratio '0.3'; OK}

Now is a good time to save your work: Ansys Toolbar: save_db

Step 4. <u>Create Mesh:</u>

Ansys Main Menu: Preprocessor: Meshing
                           Meshtool {a new Meshtool Window opens}
                              Meshtool: {select Size Controls: Global ; Set}
                                                    {set Size '0.02'; OK}
                           Meshtool: Mesh {a new Mesh Areas window opens}
                              Mesh Areas: {cursor select plate area to mesh; Apply; OK}

Figure 8: Ansys Generated Mesh

<u>Solving:</u>

Step 5. <u>Apply Displacement Constraints</u>: From symmetry, the bottom edge cannot displace vertically and the left edge cannot displace horizontally.

Ansys Main Menu: Solution
                        -Define Loads-Apply
                          -Structural-Displacement
                                On Line {cursor pick bottom edge; Apply}
                                        {select 'UY'; Displacement value '0'; Apply}
Ansys Main Menu: Solution
                        -Define Loads-Apply
                          -Structural-Displacement
                                On Line {cursor pick left edge; Apply}
                                        {select 'UX'; Displacement value '0'; Apply}

Step 6. Apply Loads: Pressure load on the inner surface

Ansys Main Menu: Solution
                        -Define Loads-Apply
                          -Structural-Pressure
                                On Line {cursor pick right edge; Apply}
                                        {Pressure value '1e6'; Apply} OK

Now is a good time to save your work: Ansys Toolbar: save_db

Step 7. Solve:
Ansys Main Menu: Solution
                        -Solve-Current LS (load step)
                         Close Solution Done Window

Post-processing:

Step 8. Plot Deformed Shape:
Ansys Main Menu: General Postprocessor
                        Plot Results
                                Deformed Shape {select Def + Undef edge}
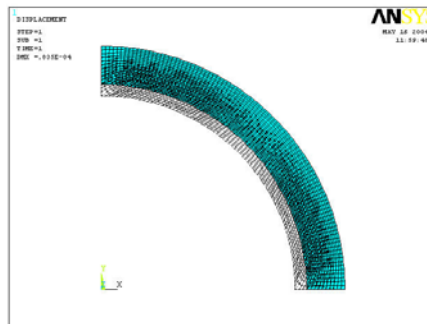
Fig 9: Deformed shape overlaid on outline of undeformed shape

Step 9. Contour Plots of Stresses:

Let us look at stresses throughout the structure as a contour plot.

Ansys Main Menu: General Postprocessor
                    Plot Results
                        -Contour Plot-Element Solution {select 'Stress'; von-Mises; OK}
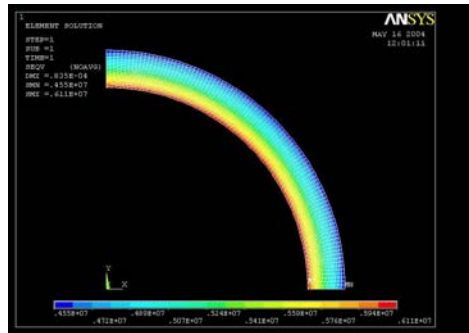
Fig 10: Contour plot of von-Mises stress

Example: A Balcony Truss

We will analyze a 2D "balcony" truss using Ansys. The problem set- up is shown in the figure below. The modulus of elasticity for each truss element is 1.9×10^6 lb/in², and each member has a cross-sectional area of 8 in².
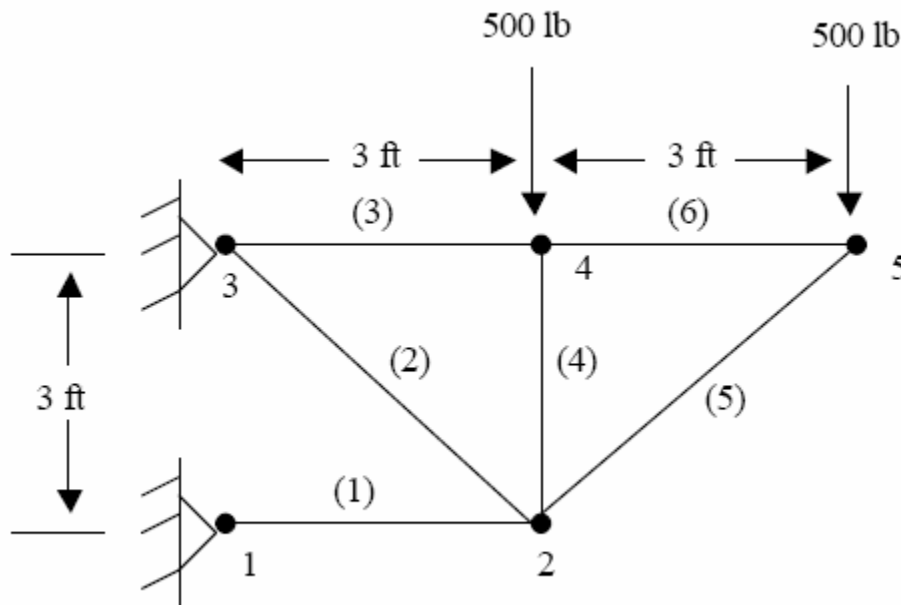
Fig. 11: A 2D balcony truss

Element	Connected to Nodes	Node	Coordinates (in inches)
1	1 and 2	1	(0,0)
2	2 and 3	2	(36,0)
3	3 and 4	3	(0,36)
4	2 and 4	4	(36,36)
5	2 and 5	5	(72,36)
6	4 and 5		

Step 1. Define Element Type:
Ansys Main Menu: Preprocessor
                              Element Type
                                   Add/Edit/Delete
                                          Add {select 'Link '2dSpar" OK} Close

Step 2. Define Material Type:

Ansys Main Menu: Preprocessor -> Real Constants -> Add/Edit/Delete….
Click on Add (notice that Link1 is listed), then in the new window click OK
A pop-up window will appear listing 3 boxes to fill in: Real Constant Set#, Cross Sectional Area, and Initial Strain. Leave Real Constant Set# at 1, take the Cross Sectional Area to be 8, and the
Initial Strain to be 0.
Click OK, finally close the Real Constants window by clicking the X in the upper right hand corner

Ansys Main Menu: Preprocessor>Material Props>Material Models>Structural (double click)>Linear>elastic>isotropic

        Input Mat'l Properties {EX=Young's: '1.9e6'; leave blank PRXY=0.3 ; OK}

Now is a good time to save your work: Ansys Toolbar: save_db

Step 3. Define Geometry / Create Mesh:

Create Nodes:
Ansys Main Menu: Preprocessor
                              - Modeling- Create
                                   -Nodes>On Working Plane:
                         {Enter:nodal coordinates in the pop-up window box as follows: 0,0 ENTER etc
                                          OK to close dialog box
Now is a good time to save your work: Ansys Toolbar: save_db

Create Elements:
Ansys Main Menu: Preprocessor
                              - Modeling- Create
                                   -Elements>User Numbered>Thru Nodes:
                         {Cursor pick pairs of nodes for each element following the list in the table above}
                                          OK to close dialog box

Now is a good time to save your work: Ansys Toolbar: save_db

Solving:

Step 4. Apply Displacement Constraints: From symmetry, the bottom edge cannot displace vertically and the left edge cannot displace horizontally.

Ansys Main Menu: Solution
                        -Define Loads-Apply
                          -Structural-Displacement
                                On Nodes {cursor pick node 1; Apply}
                                            {select 'All DOF'; Displacement value '0'; Apply}
                                On Nodes {cursor pick node 3; Apply}
                                            {select 'All DOF'; Displacement value '0'; Apply}

Step 5. Apply Loads:
Ansys Main Menu: Solution
                        -Define Loads-Apply
                          -Structural-Force
                                On Nodes {cursor pick nodes 4 and 5; Apply}
                                            {select 'FY'; Force value '-500'; Apply}
Now is a good time to save your work: Ansys Toolbar: save_db

Step 6. Solve:
Ansys Main Menu: Solution
                        -Solve-Current LS (load step)
                         Close Solution Done Window

Post-processing:

Step 7. Plot Deformed Shape:
Ansys Main Menu: General Postprocessor
                        Plot Results
                                Deformed Shape {select Def + Undef edge}
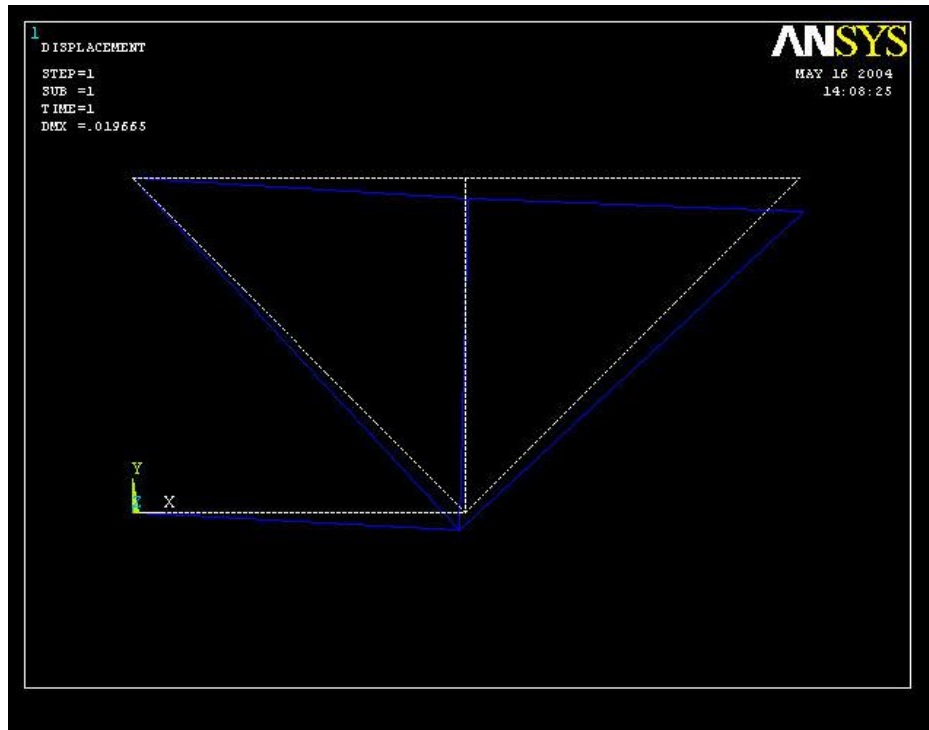
Fig 12: Deformed shape overlaid on outline of undeformed shape

Step 8. <u>Listing of Stresses</u>:

Let us look at stresses in various elements of the structure as a text file.

- Click General Postproc -> Element Table -> Define Table. A window entitled, "Element Table
Data" should appear. Click Add.
- Now a window entitled, "Define Additional Element Table Items" should appear. For "User label for
item", type Axforce. In the window directly below, scroll down until "By sequence num" is found, and
click on it. Now, in the window to the right, click on "SMISC,". Finally, in the window below "SMISC,"
type a 1 next to the "SMISC", and click Apply. Ignore the "Eff NU for EQU strain" line.
- By clicking Apply, the same "Define Additional…" window reappears. For "User label.." type Axstress.
Find "By sequence num" again, choose "LS" instead of "SMISC", and type a 1 next to the "LS" below,
and click OK.
- Close the "Element Table Data" window.
- Click General Postproc -> Element Table -> List Element Table. Click on both AXFORCE and
AXSTRESS, then click OK. A table showing the force and stress in each element should appear.
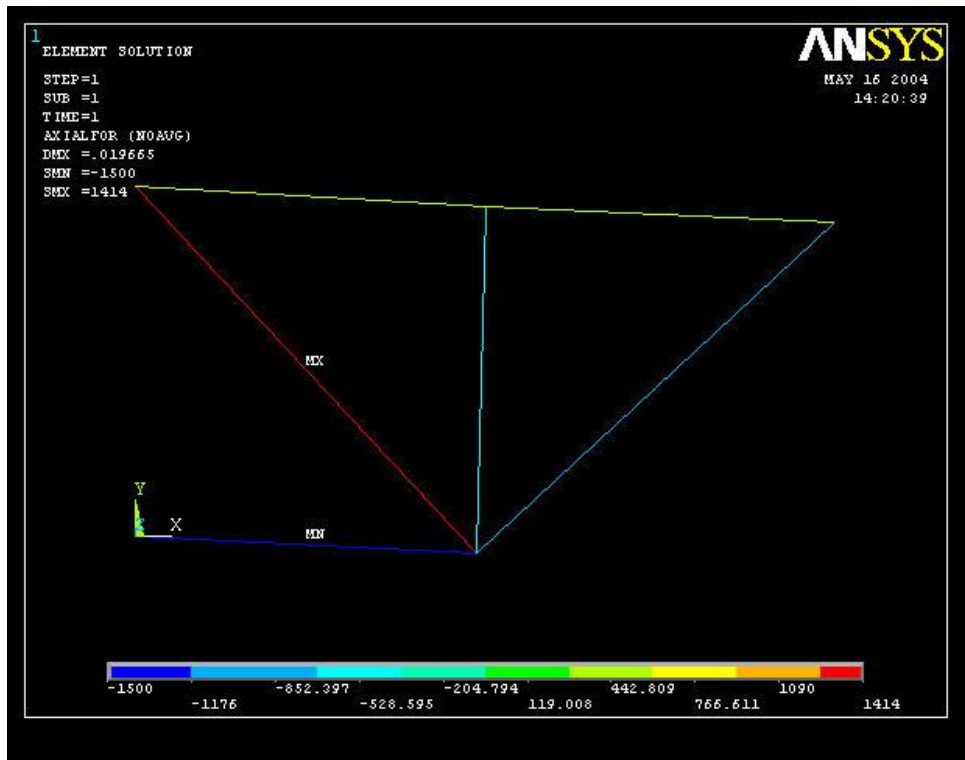
Figure 14: Forces/Stresses in the truss elements

Ansys Lab — Heat Flow

Lab exercise

Determine the temperature of over a solid with a circular groove as shown in the figure. A hot liquid flows along the groove and keeps the temperature of the circular surface at $T_1 = 150^0C$. The top and side surfaces are well insulated. The bottom surface is in contact with a coolant that maintains at $T_2 = 0^0C$. The solid is made of two materials, with thermal conductivities K1=10 W/m k, K2=20 W/m l, R1=5m, R2=7m.

Liquid
T1

R1

R2   Area 1 (K1)

10 m

Area 2 (K2)

20 m

T2
Coolant
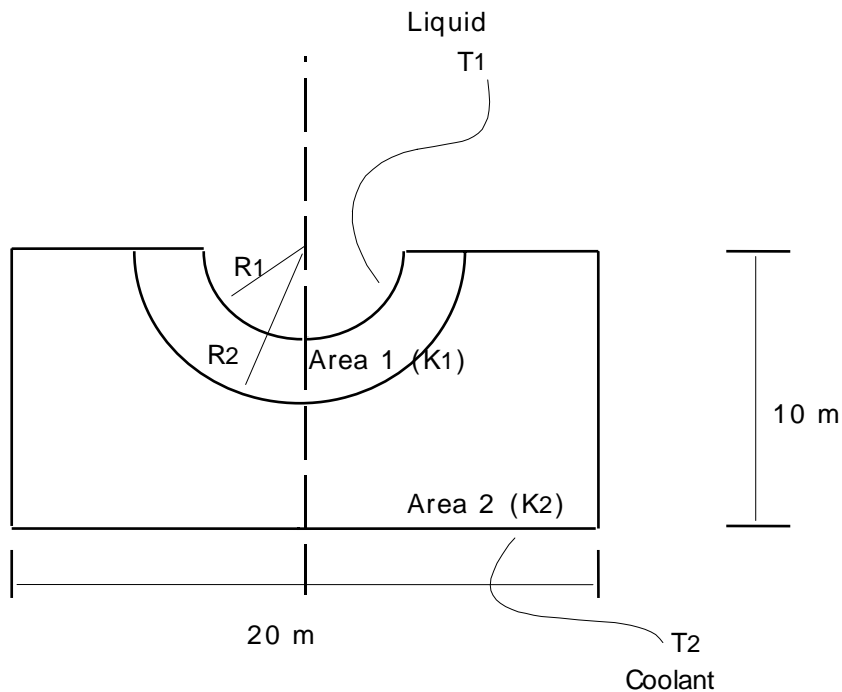
Initial Setup

1. Enter the ANSYS program by using the launcher

• Click **ANSYS now**... from the launcher menu.
• Type heatLab in the Initial jobname entry field of the dialog box.
• Pick Run to apply the information.

2. Specify a title for the problem.

• ANSYS Utility Menu > File
• Change Title > Enter new title [**Heat Transfer**] > OK

3. Set up the graphics area.

Important note: The following settings for the Workplane, PlotCtrls, etc. are appropriate only for this lab exercise. For each problem you must use geometry information from the model to decide on the appropriate settings. The minimum and maximum dimensions are easy to determine from the model geometry. The Spacing determines the number of grid lines that you'll see on the screen. The Snap Incr controls the smallest value that you can select graphically. The selected position is rounded using the Snap Incr.

It is usually not necessary to manually enter the numbers in the View Settings in PlotCtrls. It is far more convenient to simply play with the graphics controls to see when the model looks right on the screen. We have given the numerical values in this lab to make sure that all of you see exactly the same graphics.

• ANSYS Utility Menu
• Workplane >WP Settings > Grid and Triad
   Snap Incr[**1.0**]
   Spacing[**1.0**]
   Minimum[**0.0**]
   Maximum[**10.0**]
   > OK
•Workplane > Display Working Plane
•PlotCtrls > View Settings > Magnification > DVAL
   User Specified [**10.0**] > OK
•PlotCtrls > View Settings > Focus Point > XF
   User Specified XF, YF, ZF [**5**] [**5**] [**5**] > OK

Preprocessor

From the ANSYS Main Menu choose Preprocessor and use the following menus to construct a finite element model

1. Define the element type (PLANE55, the 2-D, 4 node, thermal solid, and material properties)

• ANSYS Main Menu
• Preprocessor > Element Type > Add/Edit/Delete > Add
   Thermal **Solid** > [**Quad 4node 55**]
   > OK > Close
• Preprocessor > Material Props > Material Models > Thermal > Conductivity > **Isotropic**
   KXX [**10**] > OK
• Preprocessor > Material Props > Material Models > Material > Newmodel > Define Material ID [**2**] > OK
> Thermal > Conductivity > **Isotropic**
   KXX [**20**] > OK

2. Create the solid model

  (1). Creating one 10 x 10 square.

• ANSYS Main Menu
• Preprocessor > -Modeling- **Create**
 Areas- **Rectangle** > By 2 Corners

   ■🔲🔲[WP = (0, 10)]

   ■🔲🔲[WP = (10, 0)]
> OK

  (2). Create a quarter annulus.

• ANSYS Main Menu
• Preprocessor > -Modeling- **Create**

    -Areas- **Circle** > Partial Annulus

        ■□□ [WP = (0, 10)]

        ■□□ [Rad-1 = 7, Theta-1 =270]

        ■□□ [Rad-2 = 0, Theta-2 =360]
        > OK

    (3). Use a boolean operation to subtract annulus from square (area 2).

• ANSYS Main Menu
• Preprocessor > -Modeling- **Operate**

    -Booleans- **Subtract** > Areas

        ■□□ [Square area]
        □■□

        ■□□ [quarter annulus]
        > OK

    (4). Create second annulus (area 1).

• ANSYS Main Menu
• Preprocessor > -Modeling- **Create**

    -Areas- **Circle** > Partial Annulus

        ■□□ [WP = (0, 10)]

        ■□□ [Rad-1 = 7, Theta-1 =270]

        ■□□ [Rad-2 = 5, Theta-2 =360]
        > OK
• ANSYS Utility Menu
• PlotCtrls > Numbering > Plot Numbering Controls > Area > **On** > OK
Note: The PlotCtrls menu has a large number of options that you can select to show the model in different
ways. Try different options so you can get familiar with the available choices.
• ANSYS Main Menu
• Preprocessor > -Modeling- **Operate**

    -Booleans- **Glue** > Areas > Pick All > OK
It is a good idea to periodically save the database. Ansys is fairly robust but it does crash once in a while.
After a crash you can resume from the point where you last saved the database.
• ANSYS Toolbar > SAVE_DB

3. Mesh the model.

• ANSYS Main Menu
• Preprocessor > Meshing >Mesh Attributes**>**

    Picked Areas **>** ■□□ [Area 1] > OK > MAT [**1**] > OK
    Picked Areas **>** ■□□ [Area 2] > OK > MAT [**2**] > OK

• Preprocessor > -Meshing- **Size Cntrls>Manualsize**

      -Lines- Picked Lines

      [inner circle; r=5]
      > OK > NDIV - No. of element divisions [**10**] > OK

      -Lines- Picked Lines

      [top left horizontal line]
      > OK > NDIV - No. of element divisions [**2**] > OK

      -Lines- Picked Lines

      [top right horizontal line]
      > OK > NDIV - No. of element divisions [**4**] > OK

• Preprocessor > Meshing > **Mesh** > Areas > Free: [Pick all] > OK
• ANSYS Toolbar > SAVE_DB

Solution Process

From the ANSYS Main Menu choose Solution and use the following menus to define the boundary conditions and the initial solutions.

1. **Apply boundary conditions:**

    (1). Specify temperature along surfaces that contact with liquid and coolant

• ANSYS Main Menu
      • Solution > -Define Loads- **Apply**
      -Thermal - Temperature > On Nodes

      [all nodes along liquid surface.]
      > OK > VALUE [150.0] > OK

      [all nodes along coolant surface.]
      > OK > VALUE [**0.0**] > OK

    (2). Apply insulation constraints along top and side surfaces and symmetric line**.**

• ANSYS Main Menu
      • Solution > -Define Loads- **Apply**
      -Thermal - Heat Flux > On Lines

      [The top two horizontal lines]
      > OK > VAL I [0.0] > Apply
      -Thermal - Heat Flux > On Lines

      [The right vertical lines]
      > OK > VAL I [0.0] > Apply
      -Thermal - Heat Flux > On Lines

      [The two left vertical lines]

      OK > VAL I [0.0] > OK

2. **Initiate the solution**

• ANSYS Main Menu
•Solution > -Solve- **Current LS**

Postprocess

From the ANSYS Main Menu choose General Postprocessor and use the following menus to review the results

1. Plot temperature contour, and thermal gradient vector.

• ANSYS main Menu
• General Postproc > Plot Results
      -Contour Plot- Nodal Solu...
      DOF solution [**TEMP**]

2. Plot thermal flux vector

• General Postproc > Plot Results
      -Vector Plot- Predefined > Flux & gradient [**TF**] > OK

3. Plot the temperature gradient

• General Postproc > Plot Results
      -Vector Plot- Predefined > Flux & gradient [**TG**] > OK

Lab Report

Submit a concise report for the lab exercise. In general, the report should include a description of the problem, the results you obtained, discussions about the results, and any particular issues you would like to comment. For this lab, for example, you should identify what is differential equation that we are solving, what are the boundary conditions, what are the material constants and so on. For the results part, it is suggested that you include at least the following plots

(1). Finite element model
(2). Temperature contour
(3). Heat flux vector plot
(4). Temperature gradient