APENDIX 2
BRIEF INTRODUCTION TO ANSYS

This is a brief introduction to using ANSYS, including a quick explanation of the stages of analysis, how to start ANSYS, and the use of the windows in ANSYS. ANSYS is a general-purpose finite element-modeling package for numerically solving a wide variety of mechanical 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.

SOLUTION STAGES

In general, a finite element solution may be broken into the following three stages. This is a general guideline that can be used for setting up any finite element analysis.

1. PREPROCESSING: DEFINING THE PROBLEM

The major steps in preprocessing are given below:

• Define keypoints/lines/areas/volumes
• Define element type and material/geometric properties
• Mesh lines/areas/volumes as required

The amount of detail required will depend on the dimensionality of the analysis (i.e. 1D, 2D, axisymmetric and 3D).

2. SOLUTION: ASSIGNING LOADS, CONSTRAINTS AND SOLVING

Here we specify the loads (point or pressure), constraints (translational and rotational) and finally solve the resulting set of equations.

3. POSTPROCESSING: FURTHER PROCESSING AND VIEWING OF THE RESULTS

In this stage one may wish to see:

• Lists of nodal displacements
• Element forces and moments
• Deflection plots
• Stress contour diagrams
STARTING UP ANSYS

ANSYS can create rather large files when running and saving; be sure that your local drive has space for it.

GETTING THE PROGRAM STARTED

There are two ways that you can start up ANSYS:

1. PC Windows application
2. Unix Windows application

PC WINDOWS START UP

Starting up ANSYS in PC Windows is simple:

1. Start Menu
2. Programs
3. ANSYS 5.7 or ANSYS 7.1
4. Run Interactive (ANSYS 5.7) or Configure ANSYS Classic (ANSYS 7.1)

Note: You can directly run the ANSYS Classic in ANSYS 7.1 to start the program. However, it is recommended that you use the step 4.

The following figure shows the window for ANSYS 5.7 and also for ANSYS 7.1 if you do the step 4.

UNIX WINDOWS START UP

Starting the Unix version of ANSYS involves a few more steps. You need special instructions.
ANSYS ENVIRONMENT

The ANSYS Environment contains 2 windows: a Main Window and an Output Window. Note that there is somewhat difference among the versions of ANSYS.

1. MAIN WINDOW

The first window is from ANSYS5.7 and the other two are from ANSYS7.0 and 7.1.

Within the Main Window are 5 divisions:
UTILITY MENU

The Utility Menu contains functions that are available throughout the ANSYS session, such as file controls, selections, graphic controls and parameters.

INPUT WINDOW

The Input Line shows program prompt messages and allows you to type in commands directly.

TOOLBAR

The Toolbar contains push buttons that execute commonly used ANSYS commands. More push buttons can be added if desired.

MAIN MENU

Functions used to build and analyze the FE model are located in the ANSYS Main Menu and its submenus. The Main Menu contains the primary ANSYS functions, organized by preprocessor, solution, general postprocessor and design optimizer and other special functions. It is from this menu that the vast majority of modeling commands are issued. This is where you will note the greatest change between previous versions of ANSYS and version 7.0 and 7.1. However, while the versions appear different, the menu structure has not changed.

The Main Menu lets the user switch to any processor to obtain access to the desired commands.

![ANSYS Processor Structure Diagram]

ANSYS processor structure
1. PREFERENCES

The "Preferences" dialog box allows you to choose the desired engineering discipline for context filtering of menu choices. By default, menu choices for all disciplines are shown, with non-applicable choices "dimmed" based on a set of element types in your model. If you prefer not to see the dimmed choices at all, you should turn on filtering. For example, turning on structural filtering completely suppresses all thermal, electromagnetic, and fluid menu topics.

2. PREPROCESSOR

Lets the user build the model geometry and assign element types, geometric and material properties.

3. SOLUTION

Lets the user select the type of analysis, apply the boundary conditions to the FE model and invoke the solution process of the FE equations.

4. GENERAL POSTPROCESSOR

Lets the user view results obtained in the FE analysis for the entire model for static or steady-state analyses. It will also provide the instantaneous results at selected points in time for transient and dynamic analyses.

5. TIME HISTORY POSTPROCESSOR

Lets the user view results obtained in a transient or dynamic FE analysis for selected points of the model as a function of time.

6. FINISH

Exits the current processor and returns to the Begin level. Other functions such as Design Opt, Radiation Matrix and Run-Time Stats are additional modules that are only required for certain, special types of analyses. Upon entering a processor a submenu will automatically open which lets the user choose from existing commands within that processor. Existing, but currently not available commands due to the stage of the analysis are greyed out and cannot be selected. In general, commands are only available within their processor and cannot be accessed from another processor. However, there are some exceptions. Also, utility commands are always immediately accessible and are independent of the current processor.
GRAPHICS WINDOW
The Graphic Window is where graphics are shown and graphical picking can be made. It is here where you will graphically view the model in its various stages of construction and the ensuing results from the analysis.

2. OUTPUT WINDOW
The Output Window shows text output from the program, such as listing of data etc. It is usually positioned behind the main window and can be put to the front if necessary.

ANSYS INTERFACE
There are two methods to use ANSYS. The first is by means of the Graphical User Interface or GUI. This method follows the conventions of popular Windows and X-Windows based programs.

The second is by means of command files. The command file approach has a steeper learning curve for many, but 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.

For the beginner it is recommended to use the GUI. The tutorials in this website are designed to teach the GUI. However, many of you may find the command file simple and more efficient to use once you have invested a small amount of time into learning the code.

SAVING YOUR JOB
It is good practice to save your model at various points during its creation. Very often you will get to a point in the modeling where things have gone well and you like to save it at the point. In that way, if you make some mistakes later on, you will at least be able to come back to this point. To save your model, select Utility Menu Bar -> File -> Save As Jobname.db. Your model will be saved in a file called jobname.db, where jobname is the name that you specified in the Launcher when you first started ANSYS.

It is a good idea to save your job at different times throughout the building and analysis of the model to backup your work in case of a system crash or other unforeseen problems.
Beware that there is no `nm(n` command in ANSYS. It is a best way to save the model before you do any modification.

**RECALLING OR RESUMING A PREVIOUSLY SAVED JOB**

Frequently you want to start up ANSYS and recall and continue a previous job. There are two methods to do this:

Using the *Launcher...*, in the ANSYS *Launcher*, select *Interactive* and specify the previously defined *jobname*. Then when you get ANSYS started, select *Utility Menu -> File -> Resume Jobname.db*.

This will restore as much of your database (geometry, loads, solution, etc) that you previously saved. Or, start ANSYS and select *Utility Menu -> File -> Resume from...* and select your job from the list that appears.

**ANSYS FILES**

A large number of files are created when you run ANSYS. If you started ANSYS without specifying a jobname, the name of all the files created will be `FILE.*` where the * represents various extensions described below. If you specified a jobname, say *Frame*, then the created files will all have the file prefix, *Frame* again with various extensions:

- **frame.db**
  
  Database file (binary). This file stores the geometry, boundary conditions and any solutions.

- **frame.dbb**
  
  Backup of the database file (binary).

- **frame.err**
  
  Error file (text). Listing of all error and warning messages.

- **frame.out**
  
  Output of all ANSYS operations (text). This is what normally scrolls in the output window during an ANSYS session.

- **frame.log**
  
  Log file or listing of ANSYS commands (text). Listing of all equivalent ANSYS command line commands used during the current session.

- **etc...** Depending on the operations carried out, other files may have been written. These files may contain results, etc.
PRINTING TEXT RESULTS TO A FILE

ANSYS produces lists and tables of many types of results that are normally displayed on the screen. However, it is often desired to save the results to a file to be later analyzed or included in a report.

1. Stresses: instead of using Plot Results to plot the stresses, choose List Results.
   Select Elem Table Data, and choose what you want to list from the menu. You can pick multiple items. When the list appears on the screen in its own window, Select File/Save As... and give a file name to store the results.

2. Any other solutions can be done in the same way. For example select Nodal Solution from the List Results menu, to get displacements.

3. Preprocessing and Solution data can be listed and saved from the List menu in the Utility Menu bar. Save the resulting list in the same way described above.

PLOTTING OF FIGURES

There are two major routes to get hardcopies from ANSYS. The first is a quick a raster-based screen dump, while the second is a scalable vector plot.

1. QUICK IMAGE SAVE

   When you want to quickly save an image of the entire screen or the current 'Graphics window', select: Utility menu bar/PlotCtrls/Hard Copy...

   In the window that appears, you will normally want to select 'Graphics window', 'Monochrome', 'Reverse Video', 'Landscape' and 'Save to:'. Then enter the file name of your choice. Press 'OK'. This raster image file may now be printed on a PostScript printer or included in a document.

2. BETTER QUALITY PLOTS

   The second method of saving a plot is much more flexible, but takes a lot more work to set up. No introduction here.

UNITS

As with most engineering software tools, ANSYS does associate units with any of its input supplied by the user. You may solve your problems in a variety of units. ANSYS requires that all geometry dimensions, properties, and input loads be consistent with respect to a units system. This means that the usage of units in ANSYS must be kept
consistent throughout the analysis. By default, ANSYS supports the MKS system of units (meter, kilogram, second). For example, if the user inputs all geometric data in metres and all forces and loading in Newton, the resulting stress will be given in N/m² = Pascal. However, this further requires that all additional information has been given in these units, i.e., the modulus of elasticity must also be defined in N/m².

This usage of unit consistency must be strictly observed by the user and remains valid without exception. Commonly used unit combinations in structural analysis are m-N-Pa, in-lbf-psi, and mm-N-MPa. However, in principle any combination of units is valid, as long as they are consistent. Thus for example, describing the geometry of a model in millimetres, but all applied forces in pounds, will result in the odd but valid stress measure lbf/mm². Some pitfalls remain. For example, describing all dimensions in millimetres and all forces in Newton in a static analysis will result in the convenient stress measure MPa. However, when computing the natural frequencies of the same FE model in a dynamic analysis, the results will be off by a factor of 31.6228 = \sqrt{1000}. This is because 1 N = 1 kg\cdot m/s², i.e., the occurrence of metre in the unit Newton makes it strictly speaking inconsistent in combination with millimetres.

Vibration and transient analysis require that the mass of the structure be entered in units consistent with the other units in the model. Some North American industries normally work in inches-pounds-seconds. This requires that mass be represented as pounds/in/sec². Pounds here means "pounds force", the force with which 1.0 g of gravity pulls on the mass. This means dividing the weight in "pounds force", or the density in pounds/in³, by the number 386.1 (more accurate than 32.2*12=386.4), which is the acceleration due to gravity expressed in inches per second squared (in/sec²). In consequence, when mass and mass density have been defined this way (the density of steel, which depends on the alloy, if given as 0.2836 lb/in³ would be entered into ANSYS as 0.0007345) it is necessary to enter 1.0 g of gravity as 386.1 in/sec² to let ANSYS apply the correct force due to gravity on the structure. Loads will be entered in pounds. Pressures and stresses will be referred to as pounds per square inch. ANSYS refers to these units as "BIN" (see the /UNITS command for "British system using inches", noting that the /UNITS command is for annotation of the database, and has no effect on the analysis or data).
ANSYS does not care what units are used, nor does it issue warnings. The analyst must be consistent in the set of units in one model, to avoid errors. Getting the mass and mass density into the correct units is particularly important if any form of vibration, transient, or transient heat transfer work will be done. Tip: Check the values for typical materials in the ANSYS material library as a guide, even if you do not use these exact materials. A comparison will indicate if your values are in the right range. The ANSYS materials library includes material values in various systems of units. Many design codes will, for example, give densities in lb/in^3, where pounds is actually the weight expressed as "pounds force".

SOLID MODEL AND FINITE ELEMENT MODEL

ANSYS distinguishes two different models that are built by the user throughout a FE analysis: the solid model and the FE model. The solid model is a geometric description of the structure to be analysed and is best compared to a model built using CAD software. In ANSYS the solid model consists of keypoints, lines, areas and volumes. A solid model can be built "bottom up", which means that initially created keypoints define lines, which define areas, which define volumes, or "top down", where a higher order entity (e.g. a volume) is defined using one of ANSYS create functions and all required lower order entities (areas, lines, keypoints) are automatically defined with it. In general the solid model is used to create the FE model, consisting of nodes and elements only. This process is called "meshing". During meshing, ANSYS automatically places nodes and elements into the solid model, such that it fills out the solid model in an optimal way. This is done to ease the very elaborate tasks of defining every single node and element individually by the user. However, it needs to be kept in mind that the subsequent FE analysis will be based on the FE model only and is completely independent of the original solid model. Therefore, the analyst should plan ahead carefully how much detail is to be included into a solid model in order to not perform unnecessary work. Fillet radii of 1 mm will be meaningless when meshing the solid model with an average element edge length of 5 mm.

It now should be obvious that the creation of a solid model is not required when conducting a FE analysis. The FE model can just as well be created manually, which sometimes proves easier for specific modelling purposes.