# Tutorial #1:The Essential ANSYS.

The purpose of this tutorial is to help you become familiar with the environment that you will be working in this quarter. Please take the time to understand this tutorial well before going further – it will save you time in the long run!

## Part 1. Introduction to ANSYS

### I. Getting started from a LINUX window

#### A. What machines can I run ANSYS on

Version 11 of ANSYS is loaded on the pods at the Pod Cluster in Terman room 104 (http://www-tecc.stanford.edu/).

### II. Using ANSYS

#### A. Where do I start?

You can start ANSYS from any directory, however it is best to start ANSYS in a directory that is specific to each project or session; that way you won't load up your home directory with the output files that ANSYS creates with each use.

 Make a ME309 directory

```
pod7:/> mkdir ME309
pod7:/> cd ME309
```

 Start ANSYS:

```
pod7:/ME309 >launcher
```

 Run ANSYS:

- *In the Simulation Evironment dropdown choose ANSYS (second on the list)*

- *Change the" working directory" to ME309 directory by browsing and selecting it.*

- *Change the "jobname" from file to example, and click "run".*

a) Click *Run* to bring up the ANSYS graphical user interface (GUI).

2. The box titled ANSYS Main Menu provides a good outline for how ANSYS works.

   a) **Preprocessor:** Used to build geometry, define material properties, mesh, apply loads, and define boundary conditions for the model. It is the most important, and time intensive part of the analysis.

   b) **Solution**: Allows you to define, solve and refine the solution according to analysis type.

   c) **General postprocessor**: Allows you to get the results of the solution for the entire model at given load steps.

   d) **Time history postprocessor**: Allows you to get results of the solution for variables of interest over all time steps (transient or harmonic analysis).

## B. How can I use ANSYS?

There are four ways to use ANSYS, they are summarized below according to level of interaction required between the program and the user.

1. **GUI:** The Graphical User Interface consists of everything except the box titled ANSYS input window. Most of the work in this class will involve this "point and click" approach.

2. **Command prompt:** The command prompt allows you to execute user-entered commands that are formatted to be readable by ANSYS. Everything that you can do with the mouse can be done through the command line. Most people use a combination of command line and the GUI. Type the following in the command line:

   **Help, nsel**

This brings up the help file for the nsel command. Next type nsel in the command line, notice that it shows the nsel command format above the line.

3. **.log file:** Alternatively, commands can be written into a text file which can then be read by ANSYS. The text file can be created in any editor, or by saving all of your ANSYS commands while you work – it does this automatically. You can load this file in to ANSYS and it will execute each command in the text file. It is extremely helpful to keep a "clean" log file. It will make the debugging process much easier. We'll try it later.

4. **Batch Mode:** The aforementioned .log file, with a list of commands, serves as an input file for running ANSYS in Batch mode. When working on your final projects, you may have an analysis solution that takes a long time, or you may want to repeat an analysis many times while changing one or two design variables – this is where batch mode comes in handy.

## C. The FEA Process using ANSYS:

1. Nodes and Elements

   a) FEA is based on nodes. Elements are used to define the relationship between the nodes: linear, nonlinear, rigid, etc. Model development usually begins with elements. Choosing the element type makes you think about what you are physically modeling, before you create the geometry. The type of element you chose determines:

      - *the degrees of freedom for each node (ux, uy, uz, rotx, roty, rotz, temp), as well as the discipline (structural, thermal, etc.)*
      - *the characteristic shape of the element (line, quadrilateral, brick, etc.)*
      - *and whether the element lies in 2D or 3D space*

2. Real Constants

   a) The next step is to define a set of real constants for the elements; certain elements need parameters defined (i.e. cross-sectional properties). Once you've decided on the element type, check the online element manual to see what real constants are required for each element type that you use.

b) Type *help, plane42* in the command line. You will see the details about this 2-D element including the real constant options.

3. Material Properties

a) Depending on the application, material properties can be linear, non-linear, and/or anisotropic.

b) Linear material properties:

- *Linear material properties can be constant or temperature-dependent, and isotropic or anisotropic. Isotropic material properties only need properties in the X-direction. ANSYS will make the Y and Z-direction properties equal to the X-direction properties defined.*

- *There are other material properties that have default values (i.e. Poisson's ratio, shear modulus, etc.) so be sure to check the material property table to make sure the values are correct.*

- *You can also choose the material properties from the ANSYS material library for isotropic linear materials.*

c) Temperature dependent material properties:

- *Temperature dependent material properties may be input in the form of a polynomial equation (i.e. linear, quadratic, cubic, quartic). ANSYS then evaluates the function at discrete points using linear interpolation between points and constant extrapolation beyond extreme points.*

- *You can also define a table of temperatures and corresponding material property values. These values can also be plotted to check their distribution, and edited using ANSYS.*

d) Non-linear material properties:

- *Non-linear material data is input using tables.*

- *These tables can also be plotted and edited in ANSYS.*

- *Linear and non-linear material properties can be saved to a text file or read from a text file.*

4. Active sets of entities

   a) Just as you can have more than one type of element in your model, you can have more than one set of real constants and material properties. ANSYS assigns a numeric label (starting with 1) to each Element Type, Real Constant Set, and Material Property Set. This way ANSYS knows what properties to assign to the elements you are about to create, these elements and properties are said to be "active".

   b) Upon creating elements, applying loads, or other operations, you can declare a set of nodes to be active. That is, nodes that are in the set defined are the only nodes that will receive your next commands (e.g. delete). This operation becomes useful when there are many nodes in a model, or if the model is 3D with many nodes stacked on top of each other in space.

   c) You can declare any set of similar entities this active or inactive – nodes, elements, keypoints.

5. Building Geometry

   There are three fundamental ways to build FEA models:

   a) Solid Modeling: You can use ANSYS for "solid modeling" (like CAD) to describe the geometric shape of the model and then use the ANSYS automesher to create nodes and elements. You are still allowed control over element size and shape to some extent. All pre-processors have different auto-meshing algorithms and some create better FEA meshes than others. However, no code works for every geometry, and hence it is easy to come up with a structure that cannot be meshed automatically (you'll see).

   b) Direct Generation: You can use "direct generation" by manually defining the location of *each* node and then "connecting" them to create elements. ANSYS can also copy patterns of existing nodes and elements, and perform other operations to make this less time consuming. If you chose direct generation remember:

      - *Nodes should be placed at locations of interest*

      - *Elements must be "connected", and they have aspect ratio and warpage limitations that you must keep in mind when modeling*

   c) CAD Import: You can import geometry from a CAD package and then model it using the ANSYS automesher. This method is the least desirable from a FEA standpoint.

Most models are created using a combination of automatic features and manual ones.

6. Loads (boundary conditions)

   a) "Boundary conditions" represent how your structure interacts with the outside world: Constraints, forces, pressure, gravity, etc. These things are ultimately applied to the nodes. You can specify displacements (i.e. ux, uy, uz, rotx, roty, rotz), or you can specify loads (i.e. Force/Moment, Pressure, Temperature) at each node. Displacements and loads can also be applied to ANSYS CAD entities (i.e. lines, areas, etc.) This way ANSYS will map the loads from the line, say, to the nodes on that line, even if you change your mesh.

7. Solution

   a) This is where ANSYS takes the database information and creates a matrix, the size of which can be quite large, say 500,000 X 500,000 or small, say 10X10. Then ANSYS uses efficient numerical solution routines to solve the equations. You will learn about this in class.

8. Results

   a) Using the postprocessor, you can retrieve the desired information: Displacements, stresses, natural frequencies, etc. and display it in many ways.

## D. What output do I get from ANSYS?

1. The initial jobname specified in the Interactive window is the prefix for all of the files that ANSYS creates. Recall that, if you don't specify a filename ANSYS uses the default filename "file". Each time the analysis is completed, these files will be **overwritten** if you do not specify a different jobname. The files created are saved into the directory where you started ANSYS unless otherwise specified.

2. Here is a summary of the most commonly used files that ANSYS creates for an analysis called *homework1*:

   a) **homework1.db:** This is the main database file that has all of the information about your model including geometry, material information, load conditions, boundary conditions, etc. The .db file is created when you save your model. (This is the most important file)

b) **homework1.log:** This file contains a running list of commands that you entered into ANSYS during that session. The .log file can help you in many ways:

- *If you chose to use the GUI or the ANSYS input window and somehow lose your model, the .log file could help you reconstruct the model without having to start over from scratch.*

- *Since the .log file is a running list of commands, the file can be edited to remove any command errors entered into the GUI or ANSYS input window. This corrected file can then be used as a text file or batch file for running your program.*

- *If you should find yourself running out of disc space, saving this file will allow you to retain all of the information necessary to build and run the file at a later time. It's smaller than the .db file, especially if it's compressed.*

c) **homework1.err**: This file keeps a record of all error messages that have occurred during your current and past sessions.

d) **homework1.mat**: This file is created during solution phase and stores the element matrix used during solution.

e) **homework1.esave**: This file is created during solution phase and stores the element data during solution.

f) **homework1.tri**: This is created during solution and stores the triangularized element matrix used during solution (this can be large).

g) **homework1.rst:** This is created during solution and stores the results of your solution ($2^{nd}$ most important file).

3. The only files that are critical to save are the .db, .rst (or .rth) and .log files. The rest of them can be safely deleted at the end of a session to save space.

**E. That's a lot of files, what if I do not have enough memory in my account?**

1. This question may not arise when going through these simple tutorials, but when solving homework sets or the final project, it will be **very important** that you be aware of the memory needed to create, solve, and save your models.

2. Although each of your accounts has a decent amount of disk space available, a moderately sized model can consume that space easily.

3. Delete all unnecessary files and analysis

## a) Some Tips

**F. Help!**

1. **The ANSYS Help System** is a fast and easy way to find an answer. Found on the **ANSYS Utility Menu**, Help provides you with thorough descriptions of ANSYS procedures, commands, elements, and theory. Surf through the help menus a little bit to see what's there. The TA's would like it if you would look there first before asking questions (at least later in the quarter), especially when you get advanced beyond the TA's ANSYS knowledge! If you think you will be referring to certain pages often, you can save yourself time by printing them and keeping them with these tutorials. Watch out for your print quota though!

2. There is also plenty of good help resources online at places like: http://ansys.net/ansys/?mycat=links. Just try a search for "ANSYS help resources" or something similar.

**G. Save and Resume**

1. Use SAVE_DB (on the ANSYS Toolbar) liberally so that you can save your work as you go along.

2. If you should ever run into trouble, you can always choose resume (near the graphics window), and that will return ANSYS to the last saved version of the database. This feature might come in handy if some geometry, or a mesh you created didn't come out right, or if your

computer crashes (it might!). It is better to resume the file than run the risk of having nodes or elements in the problem that you didn't completely erase using another method.

## H. Graphics

1. Don't trust everything you see or sometimes don't see in the graphics screen. Sometimes, certain entities may be turned off. Choosing which entities to display is useful for creating the model, as we'll see later.

2. To quickly check to see a list of all nodes or elements that are active go to the Utility Menu under **List>>** and then chose what you want listed. A pop-up file will appear with the information you requested. You can print this window, but I don't recommend it, especially for large files.