

EE 505 MEMS Sensors and Actuators Batch ANSYS Tutorial

Objective

This tutorial was created to introduce you to ANSYS by simulating the bending of a cantilever beam due to an applied force.

Background

ANSYS is a general purpose Finite Element Analysis (FEA) software package. There are a wide variety of problems in statics and dynamics that it can solve or approximate; mechanical, thermal, acoustic, electromagnetic, and electrical, to name a few, including coupled and non-linear problems.

ANSYS 10.0 has two methods for entering commands and building models. The first method is a GUI from which most of the commands can be accessed and is arguable easier to learn. However, as with other CAD software, an investment in time in learning the second method, command line and batch file functionality, is well worth it. We will only go through the command line method, since this is more powerful. Also, with batch files models can be easily modified without having to rebuild the whole model.

Overview of FEA

FEA software solves electrical, mechanical, and other problems by discretizing the analysis region. In order to analyze a region the model must be given a set of nodes and a mesh. In addition, there must be an element assigned to each set of nodes. The nodes define the boundaries of each individual element, and the entire collection of elements defines the analysis region (the geometry of the model). ANSYS provides a wide variety of elements for different analysis types (1D, 2D, 3D, thermal, mechanical, electrical, etc). The elements are defined by a set of nodes, and by an element type. The role of the element is to provide the equations that relate the degrees of freedom to the nodes.

As an example: A mesh of a 3D cube that has 1000 nodes (10 nodes per side) and 3 DOF at each node (x, y, z) is converted by ANSYS into a 3000 x 3000 matrix before solving. Doubling the number of nodes per side results in a system of 24000 equations (a 24000 x 24000 matrix). For this reason it is desirable to use as few nodes as necessary to get the necessary results. FEA solvers are all routines that are specialized for solving large systems of equations (matrices).

In ANSYS and other FEA software, you are going to be responsible for keeping track of units. There is no functionality for entering material properties in any available unit and having the software maintain consistency. It is useful to write down all your properties with units, and ensure that they are consistent. Forgetting this will give you results that are either lucky or many orders of magnitude off.

FEA is a powerful tool, but it is not a replacement for all engineering analysis. The inviolate rule of FEA analysis is to never trust the results of an analysis unless you can analytically confirm a simpler case. The ability to do this will confirm that your units are correct, that you are using an adequate mesh density, and that you are using a suitable element type for the analysis. Blindly following the results of an FEA analysis will surely lead to trouble.

Procedural Overview

An analysis is generally broken into 3 main parts:

- 1) Pre-processing
 - a. Generation of nodes
 - b. Generation of elements
 - c. Assigning boundary conditions
 - d. Assigning material properties
- 2) Solving
 - a. Application of loads or fields
 - b. Applying solution parameters (i.e. linear / non-linear)
 - c. Selecting analysis values to be saved
 - d. Solution of equations
- 3) Post-processing
 - a. Interpretation of saved results
 - b. Numerical processing
 - c. Visualization

Important Note:

ANSYS has many commands, most of which have many options. **YOU MUST LEARN TO USE THE HELP DOCUMENTATION!!!**

Procedure

We will analyze a 3D cantilever beam as shown below. Determine the nodal deflections in the z direction, stress in the x direction, and stress in the y direction.

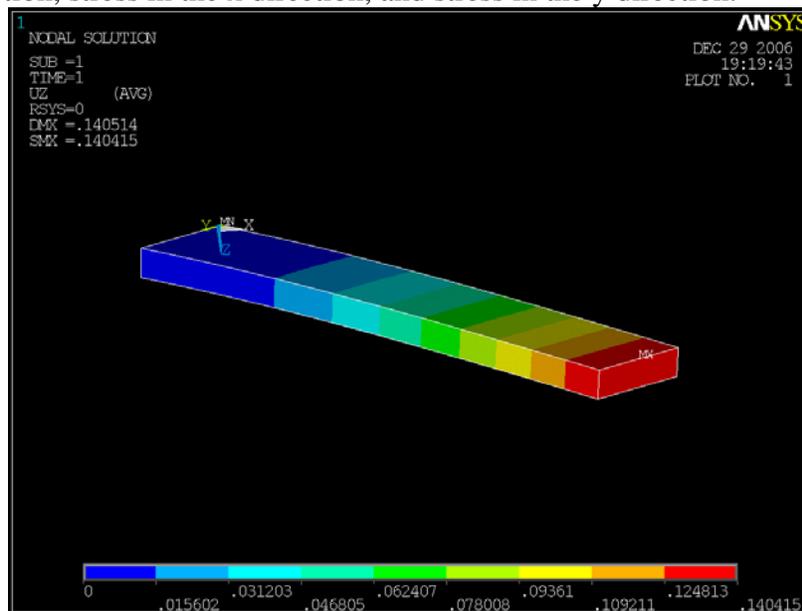


Figure 1. Cantilever Beam

Pre-processing

- Step 1. Enter the pre-processor to setup the model.

Command: /prep7
Enters the model creation preprocessor.

Step 2. Dimensional Analysis.

ANSYS, like other FEA software packages, does not keep track of units, therefore it is imperative that you define your units and stick with them. A MathCAD file can be useful to check your dimensional analysis. For this tutorial we will be using the following system of units.

```
!      DIMENSIONS
! [F] = uNewton
! [L] = um
! [T] = seconds
```

Force will be in micro-newtons, distances will be in microns, and time will be in seconds. Make sure to use these dimensions when specifying material properties, boundary conditions, and applied forces.

Step 3. Enter material properties.

For this tutorial, our objective is to model a cantilever beam that has been etched out of single crystal silicon, so the material properties will be defined for silicon. A good source for the values of material properties is www.matweb.com. You will need to enter Young's Modulus (EX) and Poisson's Ratio (PRXY). Although you don't have to enter the thermal conductivity (KXX), ANSYS will give you a warning if you do not (Note: ANSYS will still successfully solve the problem even if the thermal conductivity is not specified). Since there is only one material in this model, we will call it material 1 (mat = 1).

Command: mp, lab, mat, c0
Defines a linear material property as a constant.

Example: mp, ex, 1, 112400

Step 4. Specify KEYPOINTS to input the device geometry.

For this introductory tutorial, we will model the cantilever beam as a simple rectangular box. Therefore we will need 8 KEYPOINTS to specify the beam. It is also time to pick our coordinate system. It is often convenient to imagine the cantilever beam being micro-fabricated in a silicon wafer where the top surface of the wafer corresponds to the top surface of the cantilever beam. We will make this top surface the XY plane and define the positive z direction to be going down into the surface of the wafer.

Command: K, NPT, X, Y, Z
Defines a KEYPOINT.

Example: k, 1, 0, 0, 0

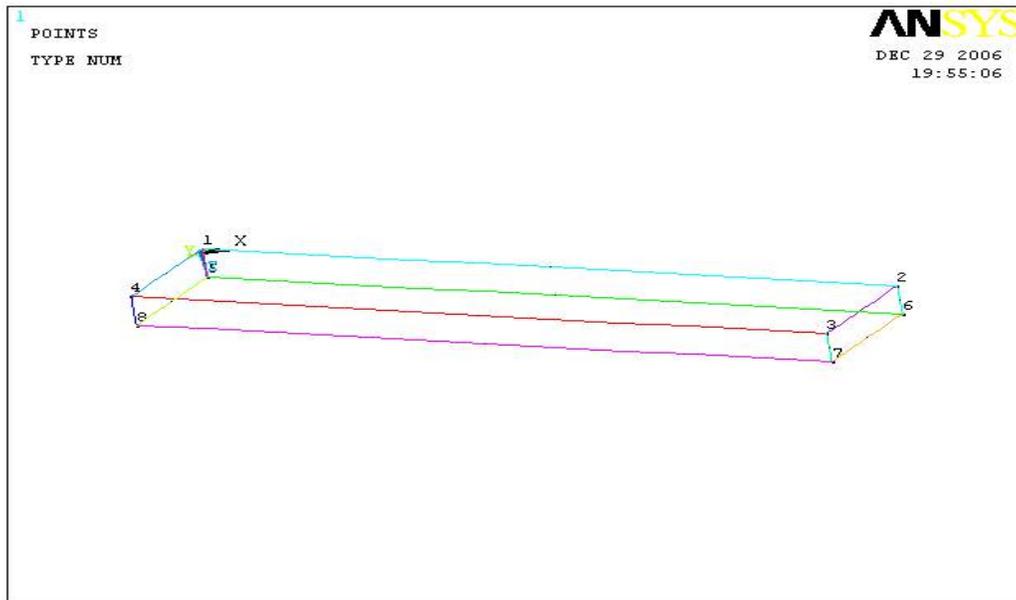


Figure 2. KEYPOINTS for cantilever beam model in ANSYS. Note: the lines were added to help visualize the cantilever beam.

Step 5. Define the volume of the cantilever beam.

Defines a volume (and its corresponding lines and areas) through eight (or fewer) existing KEYPOINTS. KEYPOINTS must be input in a continuous order. The order of the KEYPOINTS should be around the bottom and then the top. Missing lines are generated "straight" in the active coordinate system and assigned the lowest available numbers. Missing areas are generated and assigned the lowest available numbers.

Command: V, P1, P2, P3, P4, P5, P6, P7, P8
Defines a volume through KEYPOINTS.

Example: v, 1, 2, 3, 4, 5, 6, 7, 8

Step 6. Define your element type.

There are many 2D and 3D element types available in ANSYS depending on the analysis you want to do. For this tutorial we will use element type solid226. This element type has the structural capabilities needed for this analysis, as well as piezoresistive capabilities that we will make use of in the future. For a description of this element type, search for "solid226" in the ANSYS help documentation. Don't worry about the keyoption for now, just specify it as 11 like the example below. If you are curious and must know, read the ANSYS help documentation for solid226.

Command: ET, mat, element type, keyoption
Defines the element type for the material

Example: et, 1, solid226, 11

Step 7. Mesh the volume.

The mesh density determines how accurate your results will be and also how much time it will take your poor computer to crunch through the calculations. There's no free lunch, so this will always be a trade off to consider. Simply using the automated mesh commands would look like this:

```
mat, 1  
vmesh, all
```

However, this does not give a satisfactory mesh density as it is too coarse. Adding two more commands allows us to specify the number of elements per length, width, and thickness of the cantilever beam.

```
smrsize, off  
desize, 4, 4, 20  
mat, 1  
vmesh, all
```

This produces a good starting point for comparison with the analytical results. The "smrsize" command tells ANSYS to allow us to manually specify the mesh density. The "desize" command specifies the divisions or number of elements per dimension. The "mat" command tells ANSYS to associate the material properties of material 1 with the automatically generated volume elements. Finally, "vmesh" does exactly what it sounds like it does, it meshes the volume.

Solving

Step 8. Define boundary conditions (BCs) and apply a force at the end of the cantilever beam.

The vmesh command used in step 7 automatically generates nodes throughout the entire volume of the model. For a cantilever beam, we need to fix the nodes at one end of the beam and apply the force to one or more nodes on the other end of the beam in the z direction.

The easiest way to fix the displacement of a select group of nodes is to use the node select commands. Search for "NSEL" in the ANSYS help documentation.

Command: NSEL, type, item, comp, vmin, vmax
Selects a subset of nodes.

Example: The following commands select all the nodes in the $x=0$ plane.

```
nselect, all
nplot
nselect, s, loc, x, 0, 0
nplot
d, all, ux, 0
d, all, uy, 0
d, all, uz, 0
```

The first “nselect” command selects all the nodes in the model in order to start with the full set and pare it down from there. The “nplot” command is necessary after each “nselect” command or else you will loose your new set. The second “nselect” command selects all nodes with an x coordinate of 0 from the previous set (all the nodes in the model). Now we plot these nodes that have $x=0$. Then we fix their displacement in the X, Y, and Z directions using the “d” command.

To apply the force on the cantilever beam we can use the “nselect” commands to isolate the node at the center of the width of the cantilever beam and opposite of the fix end. Then apply the force to this single node. You will need the “f” command to apply the force once you have selected this node using the nselect commands.

Command: F, NODE, LAB, VALUE
Specifies force loads at nodes.

Example: f, all, fz, 10

Step 9. Solve

Enter the solution mode of ANSYS.

Command: /solu
Enters the solution processor.

And finally, solve.

Command: solve
Starts a solution

Post-processing

Step 10. Plot results.

Enter the post-processor.

Command: /post1
Enters the database results post-processor.

Plot the displacement in the z direction.

Command: PLNSOL, ITEM
Displays results as continuous contours.

Example: plnsol, uz

Step 11. Compare ANSYS results with analytical results.