LASER INTERFEROMETER GRAVITATIONAL WAVE OBSERVATORY

LIGO Laboratory / LIGO Scientific Collaboration

Ansys 7.0 tutorial in non-linear blade deflection analysis

Eoin J. Elliffe

Distribution of this document:
LIGO Science Collaboration

This is an internal working note
of the LIGO Project.

California Institute of Technology
LIGO Project – MS 18-34
1200 E. California Blvd.
Pasadena, CA 91125
Phone (626) 395-2129
Fax (626) 304-9834
E-mail: info@ligo.caltech.edu

Massachusetts Institute of Technology
LIGO Project – NW17-161
175 Albany St
Cambridge, MA 02139
Phone (617) 253-4824
Fax (617) 253-7014
E-mail: info@ligo.mit.edu

LIGO Hanford Observatory
P.O. Box 1970
Mail Stop S9-02
Richland WA 99352
Phone 509-372-8106
Fax 509-372-8137

LIGO Livingston Observatory
P.O. Box 940
Livingston, LA 70754
Phone 225-686-3100
Fax 225-686-7189

http://www.ligo.caltech.edu/

1　Introduction

1.1　Blade Deflection Analysis

Is it not a valid assumption that a blade undergoes a geometric nonlinearity when it responds to a load, as there is both a large displacement and/or rotation? It does not however possess a non-linear stress-strain relationship, which would imply non-linearity's in the material.

Ansys uses the "Newton-Raphson" method to solve non-linear problems. The load is sub-divided into a series of load increments. The load increments can be applied over several load steps. This method evaluates the out of balance load vector, which is the difference between the restoring forces (the loads corresponding to the element stresses) and the applied loads.

The program then performs a linear solution, using the out-of-balance loads, and checks for convergence. If convergence criteria are not satisfied the out-of-balance load vector is re-evaluated, the stiffness matrix is updated.

Non-linear analysis requires 3 steps

- Load steps are defined explicitly over a time span. Loads are assumed to vary linearly within load steps.

- Within each load step, you can direct the program to perform several solutions to apply the load gradually.

- At each sub step, the program will perform a number of equilibrium iterations to obtain a converged solution.

The Ansys program gives you a number of choices when you designate convergence criteria, you can base convergence checking on forces, moments, displacements or rotations or any combination of these items.

Additionally each item can have a different convergence tolerance value.

2　Conservative –Vs- Non Conservative

When loads are taken away the system recovers back to its initial state this is assumed to be the case for the cantilever blades under inspection, therefore it is a conservative system and is path independent. That is loads can usually be applied in any order in any number of increments without affecting the end results.

3　Load and Displacement directions

1. Loads applied maintain constant direction.

2. Forces will change direction "Following" the elements as they under go large rotations.

Case 1. Applies to the blades. Accelerations and concentrated forces maintain their original orientation, regardless of element orientation.

Nodal co-ordinate system orientations are not updated in a large deflection analysis. Calculated displacements are therefore output in the original directions.

The out-of-plane stiffness of a structure can be significantly affected by the state of in plane stress in that structure.

The coupling between in-plane stress and transverse stiffness is known as stress stiffening and can be switched on or off depending on validity of non-uniform stress distribution. It is most pronounced in thin, highly stressed structures, such as cables or membranes and maybe considered in membranes >1mm thick. The theory of stress stiffening is explained in detail in the Ansys help files. This option should be considered for blade modeling depending on the geometry and output of initial runs of the non-linear algorithm. This point will be expanded on later in this document.

4. Modeling

Open up Ansys and set filename

Got to the workplane menu and Change active CS to global cylindrical.

ASIDE: If you refer to the lower mode cleaner blade design for the LASTI project

See below.
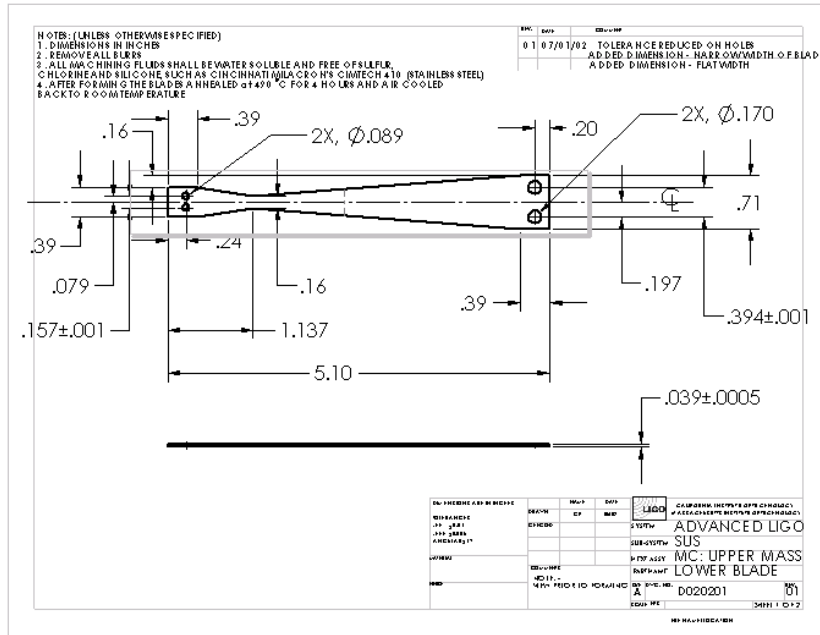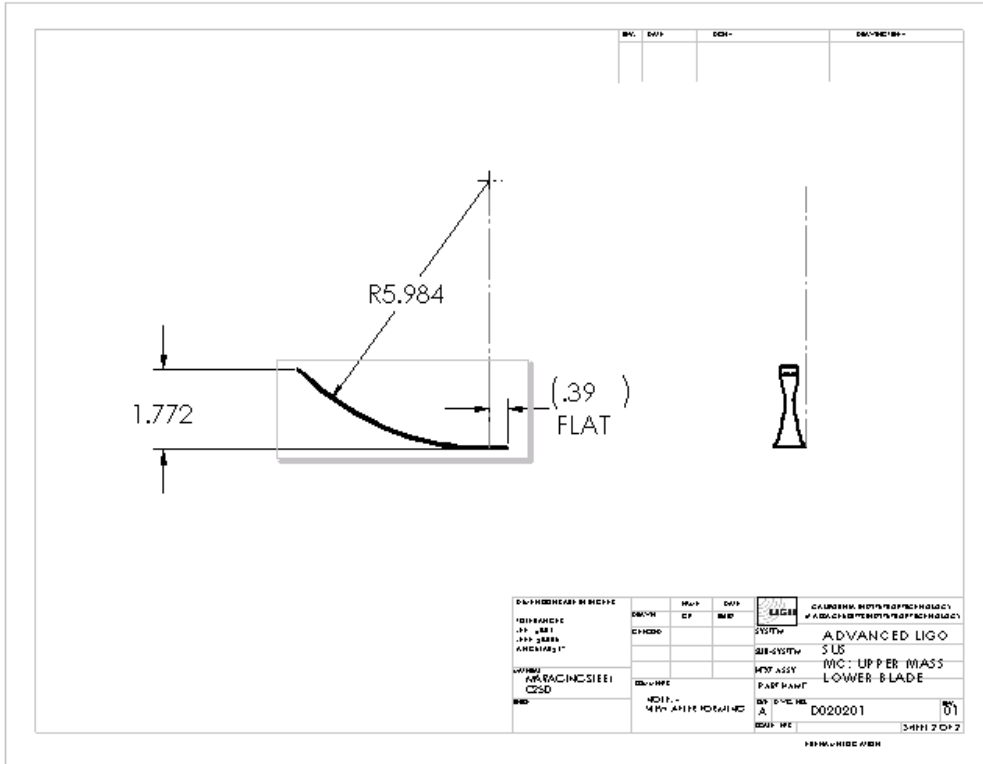
The radius of curvature R is: 5.984 inches

By geometrical analysis it can be shown that the deflection d. is related to the radius of curvature by the equation $d = R (1 - \cos(L/R))$ and in this case is 1.772 inches.

Where L is the total length of the curved part of the blade.

When modelling the blade in Ansys we can ignore the clamped area of the blade.

This means for the blade above we have 8 defining points that determine it's shape.
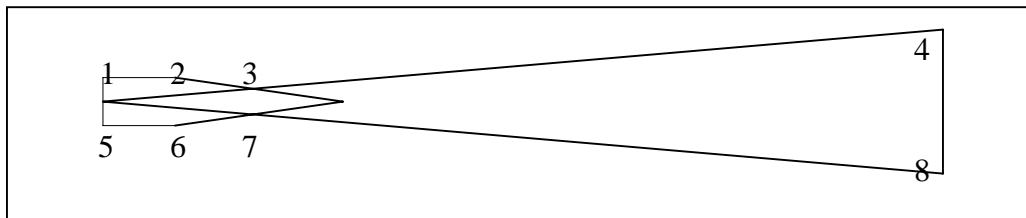
But we wish to draw a curved blade, to do this we can set an imaginary origin at point 4 or 8 from the diagram above.

We then change the pts to fit cylindrical coordinates using the radius of curvature provided.

Go to Modelling → Create → Keypts → in active coordinate system.

(I will be working in mm's.), Here is where we will enter in the defining pts of the blades shape.
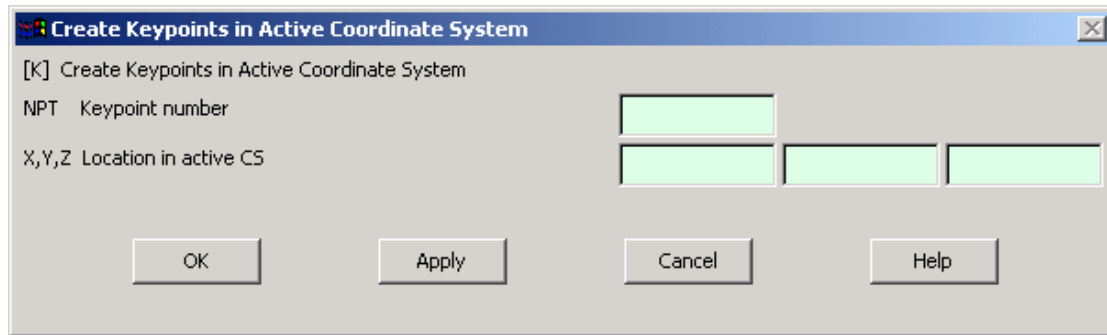
You should end up with the coordinates below simply using

R*theta = segment length. They should be something like: *(please note that the factor of 25.4 is just changing inches to millimetres)*

| R (x)	Θ (y)	Z (z)
5.984*25.4	45.0974	0.16*25.4
5.984*25.4	45.0974	0.55*25.4
5.984*25.4	41.36326	0.16*25.4
5.984*25.4	41.36326	0.55*25.4
5.984*25.4	34.21086	0.275*25.4
5.984*25.4	34.21086	0.435*25.4
5.984*25.4	0	0
5.984*25.4	0	0.71*25.4

After each coordinate entered press Apply rather than OK so that the form remains to enter in the next coordinate but be careful with the last co-ordinate! Just press OK
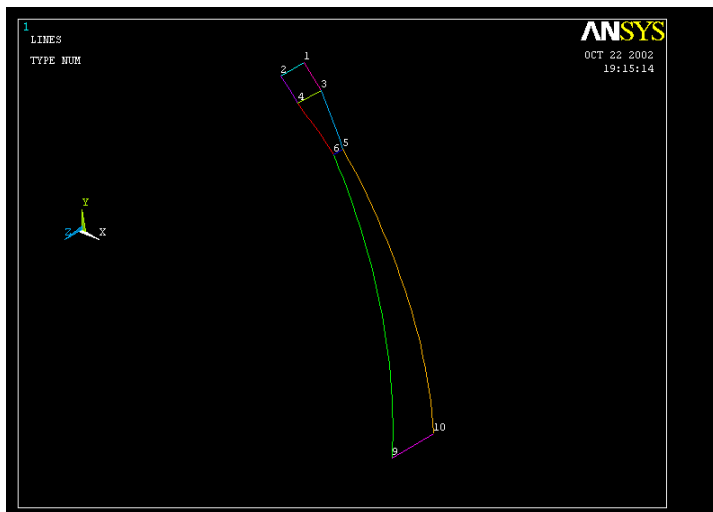
Change view to isometric by selecting Plotctrls → pan zoom rotate
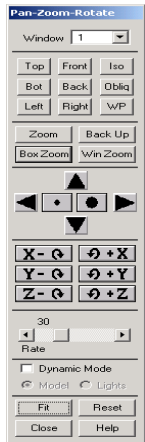
Then select **iso** from the above form and then select **fit** in case all keypoints created are not displayed.
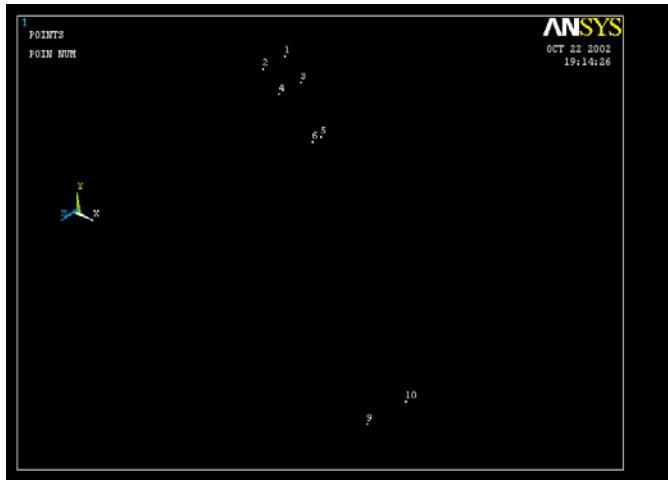
Now navigate to Create→ Lines → In Active Coord and draw the lines as shown in diagram below.

Now that you have done that you notice that you have three areas each needs to be defined so go to Modelling →create → areas →arbitary → by lines

And now select four lines that enclose an area and repeat the procedure for the three areas.You can then turn on area numbering by going to Plotctrls → numbering
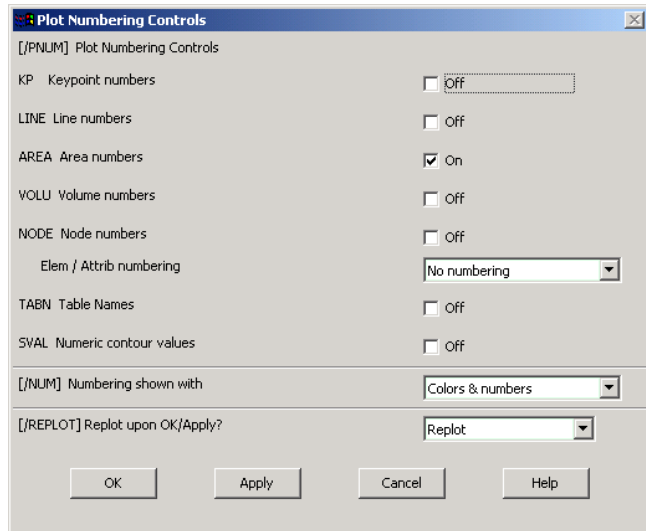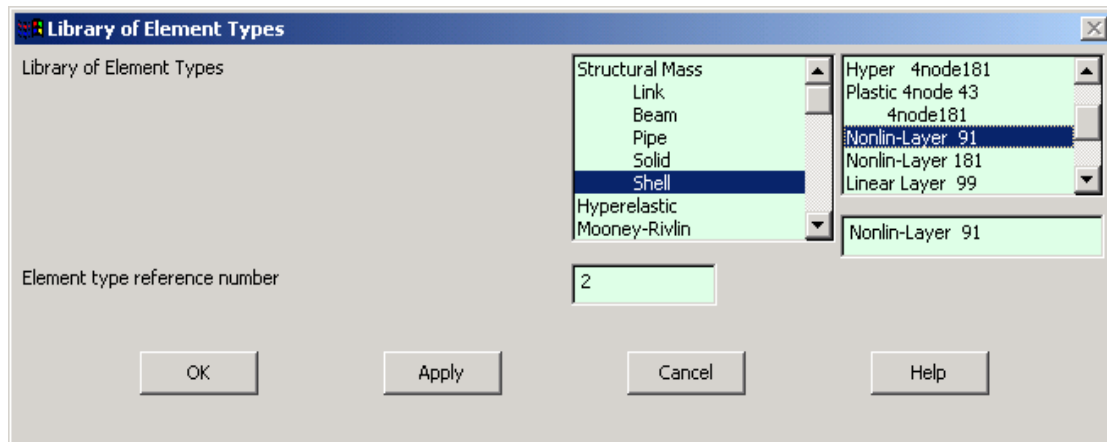

Select area numbers on and then OK !!

You should now have something like this

It is now time to shell mesh the area created.

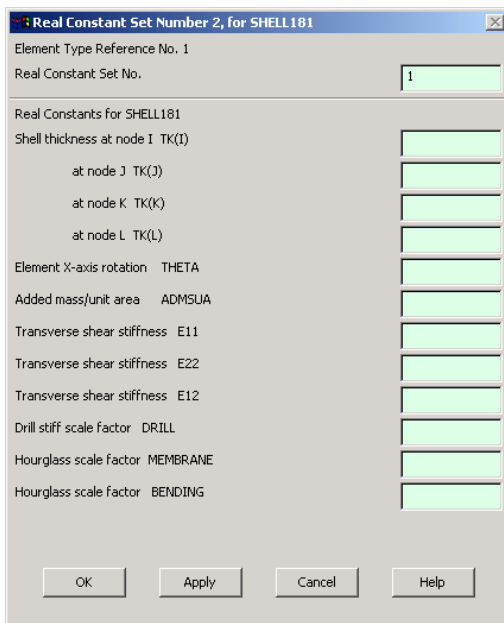Navigate to Preprocessor→Element type → Add/Edit/Delete → Add

Then choose Shell and Nonlin-layer 91 this a very accurate element suited to our analysis detailed above. Click OK, until you get back out to the main screen.

Now go to Preprocessor→Real constants→Add

You should get the pop-up to the left now click OK!

You now get the menu screen to the left. Here you can now give your element a virtual thickness value. Since the thickness doesn't vary you can just set the Shell thickness at node I TK(I).

Set this box to 1.5. That is a thickness of 1.5mm. (this entry should go in the second box down from the top…..)
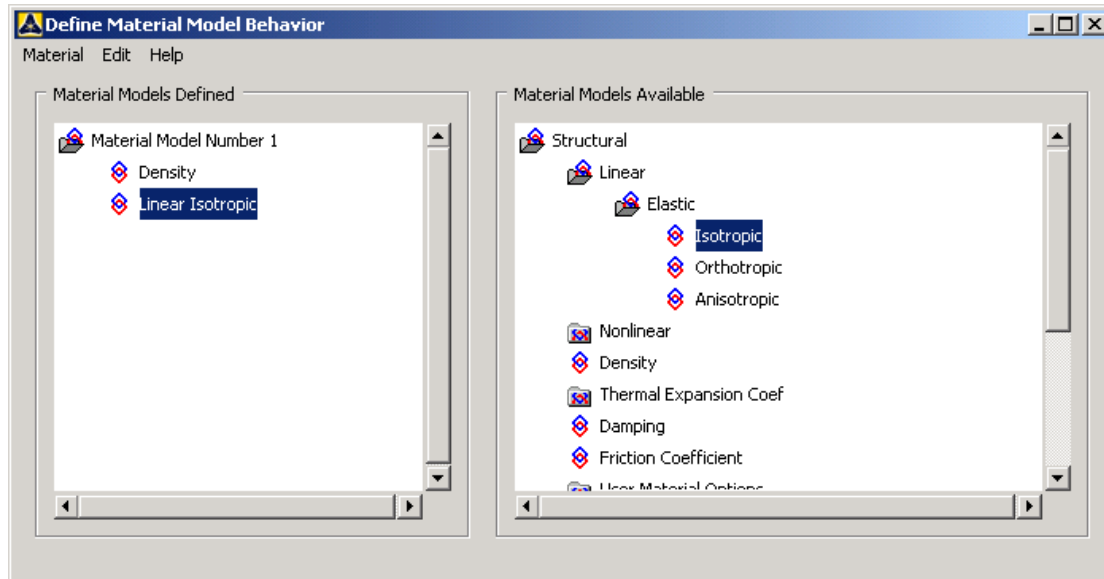
We now can define what our shell elements are made from

Preprocessor→Material props→Material models→Click on Isotropic you can now enter your Young's modulus and Possion's ratio.

Here you must remember that you are working in units of mm's so that the young's modulus will be N/mm^2. For maraging steel this will be 176e3 N/mm^2 rather than 176GPa..

Poission's ratio is unitless so just enter 0.3

Under Nonlinear click on density this should be quoted in Kg's/mm^3. For maraging steel this is 8000 kg/m^3 X 10^{-9} to convert to Kg/mm^3 or 8e-6!

We are now ready to mesh

Go to meshing→ mesh tool
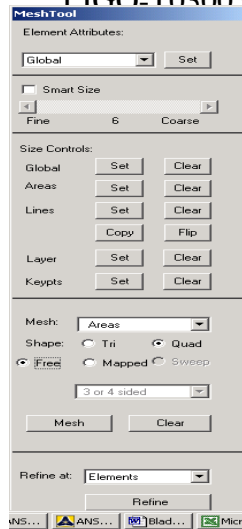
The pop-up menu above should appear.

You can now define the size of your mesh.

There is a variety of ways to do this but we shall only mention one here, that is to select the Global set button and enter the element size of 5 this means that the average element size is 5mm's. With the Shape of the mesh set to Quad and Free we can now click on the MESH button.

Here we are asked to choose what we wish to mesh if we select the Pick all button we should now have something like the next picture below!!
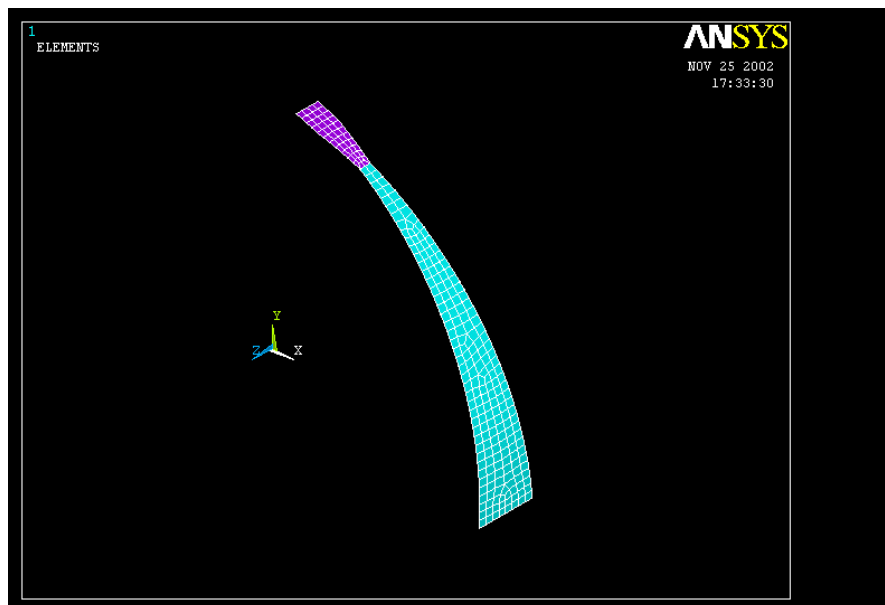
Next step is to apply the constraints and forces on the maraging steel curved cantilever blade. First of all we need to change the view so that it is easy to select the correct nodes.

Click on the Plotctrls→Pan zoom rotate button to access the viewing controls, this then allows us to view the base nodes full on…figure it out!!

Navigate to Preprocessor→Loads→Apply→Structural→Displacement → On nodes
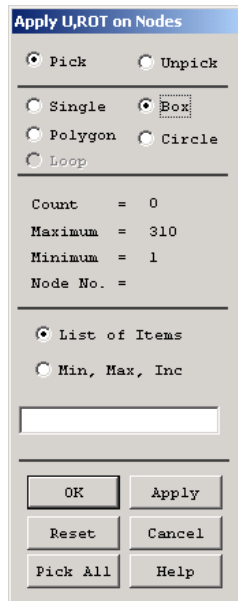
Select the "Box" radio button this allows you to draw a box around the nodes you wish to constrain.

Now draw a box around the base nodes, if you are unsure as which are the base nodes just issue the nplot command in the command line and you will clearly see the bottom row of nodes that you wish to constrain.

Having done this you should now get a pop-up that explains itself.
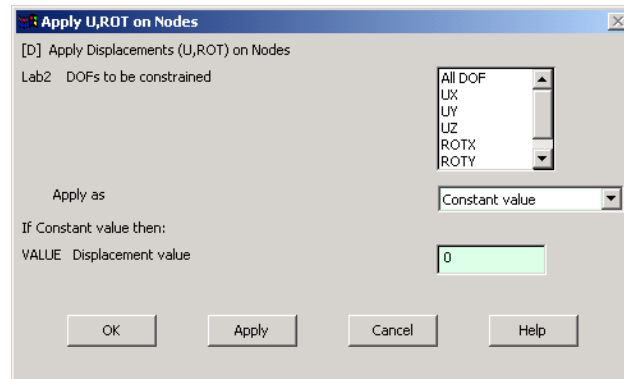
Make sure **All DOF** (all degrees of freedom) is selected and that the displacement value is set to zero. Press OK.

We can now view our model in such away as to make it viable to select the centre node on the tip of the blade so that we may apply a Force to it.

On the View controls I press **iso** then box zoom on the tip of the blade. You can now issue the **nplot** (node plot) command to view the middle node you want to apply a force to.

Now go to Preprocessor→Loads→apply→structural→ Force/moment →on nodes
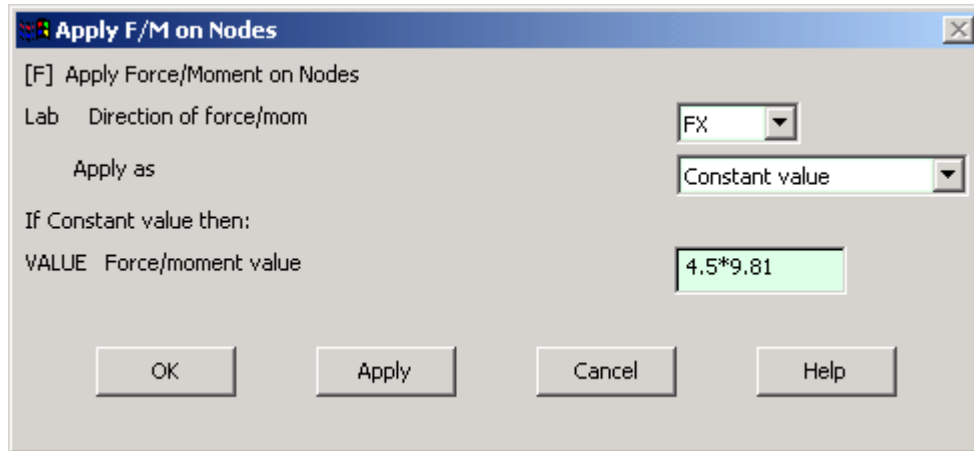
Enter a value similar to the one above, we will apply 1.5Kg's in the positive X-direction, instead of 4.5Kg's as shown, multiply 1.5*9.81 (gravity to convert to newtons.)

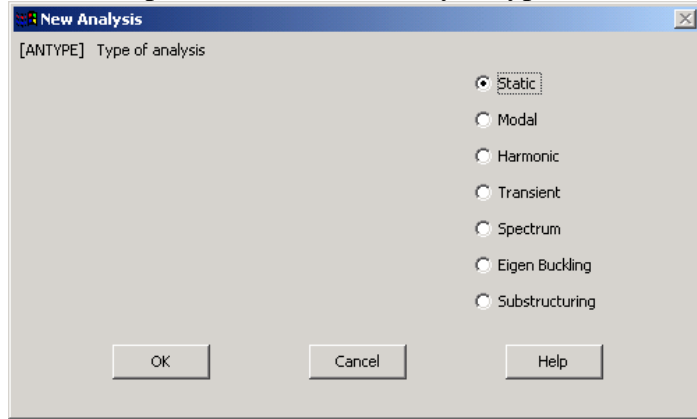To view the model with all forces applied go to Plotctrls→symbols

Now choose to show all BC + reaction and click OK

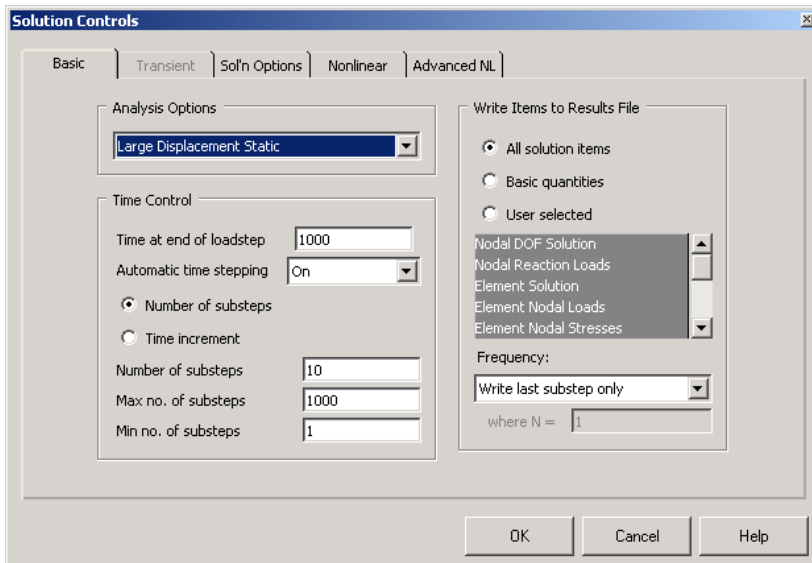You should now see the picture below (or similar).
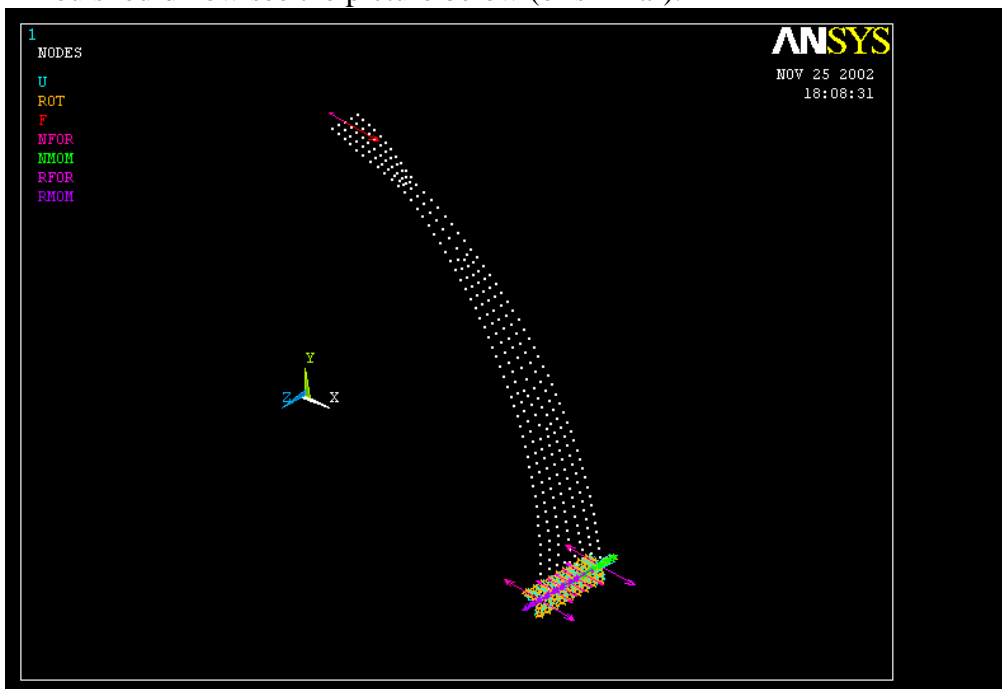
We now must define a non-linear analysis.

The menu path is Solution→Analysis type→New Analysis

Click Ok

Now select Solution→ Analysis type→Sol'n controls

Set the values as shown above. Please look in the Help menu to understand the controls on this form. If you are familiar with programming, you're right, they are just defining a convergence loop and defining the content of your output file**.

Click Ok. A really nice idea is to set the *"time at end of loadstep"* to be equal to the load you apply to the tip of the blade (see end of document for more details!!)

You are now ready to run the model
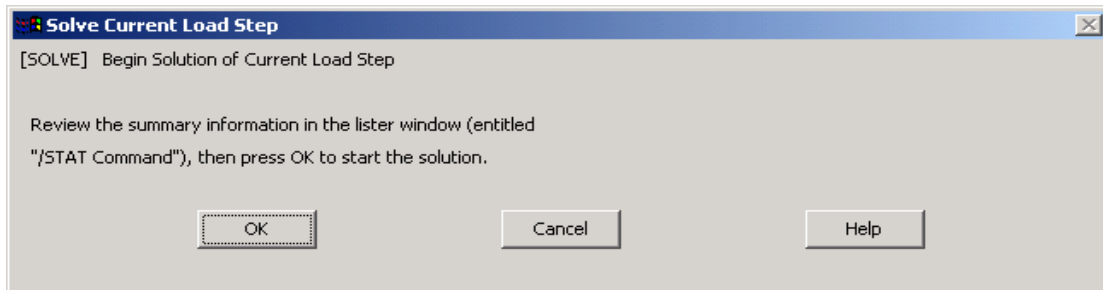
Solution➔Solve➔Current LS ………………OK!

Once the model has finished running you can click on the General Post-processing.

This will give you all sorts of beautiful results one of particular interest in the case of the blades is the deflection in the X-direction.

We can generate an output file, which will give us the displacement of every node,

To do this we navigate to General postproc➔ List Results ➔Nodal solution
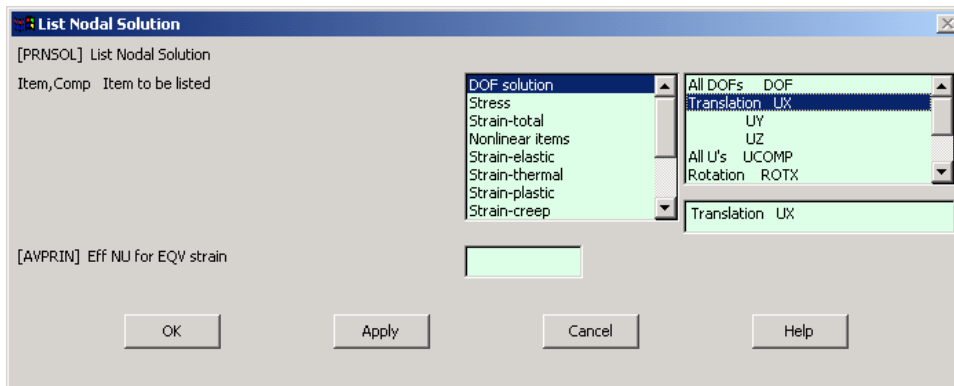
Select as shown above DOF solution ➔ Translation UX

This will provide you with a listing of node numbers along side the displacement of that node in mm's. To see how the nodes are numbered go to Plotctrls➔Numbering➔and turn on node numbers.

3.1 Tips

If you wish to find out the maximum value of the displacement, scroll down to the end of the file. Strictly speaking the top nodes should all have the same value, this may not be the case but the variation in these values is negligibly small. However there is a way around this you can concatenate the nodes at the top of the blade. Or easier still you can divide the force you apply by the number of nodes on the tip of the blade and apply this force to the number of nodes. You can also create nodes and apply the force at distances away from the cantilever blade, which will not be explained here but is very easy to accomplish.

Another handy tip as mentioned earlier is to set the *"time at end of loadstep"* to be equal to the load you apply to the tip of the blade. After the model is run you can navigate to **TimeHist Postpro** in this window to can plot a graph of "Time" versus Displacement for any node. Which since time is set to be equal to load this graph is equivalent to a "Load" versus Displacement graph.

3.2 Errors/warnings

I have run numerous blades using the technique above with no errors. However it may be that warnings such as "small pivot equations are unsolved". Please refer to the Ansys help files, where this problem is fully documented.

Any Questions or comments are very welcome.

Contact: e.elliffe@physics.gla.ac.uk