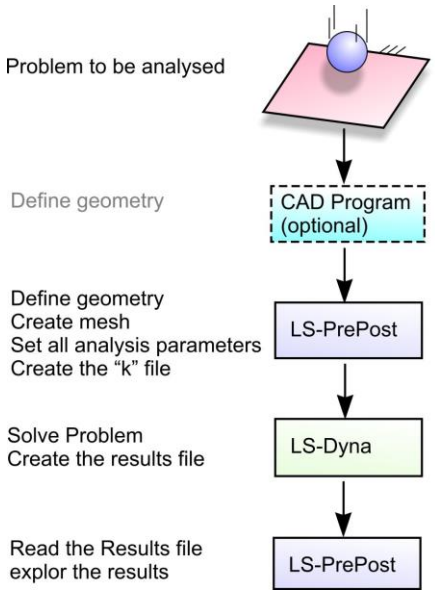
Engineering 9093 – Ice Class Ship Structures

Tutorial #1
Introduction to LSDyna: Simple Cantilever

By C. Daley

Overview

LSDyna™ is a special type of finite element program produced by Livermore Software Technology Corporation (LSTC). The program is particularly well suited to problems involving impact and complex non-linear behaviour (eg buckling, plasticity, fracture). For this introduction, we will examine a simple cantilever bar. We will be demonstrating with the release 4.0 (Beta) of LS-PrePost. LS- PrePost is a free program that is used to prepare simulations and examine results. You can install this program on any computer you want (ie at home or on your laptop) and prepare models. The calculation program is called LSDyna. It is not free, but you can use it at the university. The sketch below shows the sequence of steps to perform a simulation:



Unlike the quasi-static simulations that are typically done in FE programs such as ANSYS, LSDYNA (normally) performs dynamic simulations, where time is actual time (in seconds or milliseconds). So even though we will start with a problem that could be solved quasi-statically, LSDyna will solve it dynamically. By dynamically, we mean that LSDyna will take the inertia of the elements and their accelerations into account. So all elements must have a specified density and mass, something not required for a quasi-static analysis in ANSYS. (Note - ANSYS-Explicit is like LSDyna)

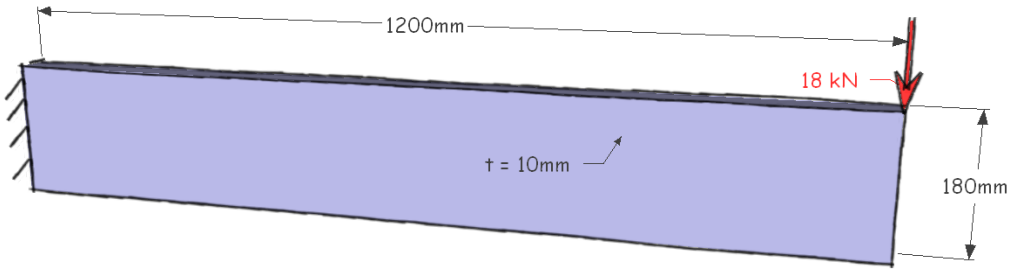
Learning Objectives

The lab is intended to show the basic user interface of LS-PrePost and LSDyna. The lab will cover the necessary steps to permit the simplest of structural analyses.

LSDYNA Model #1 – simple cantilever

Step 1: describe and sketch the problem:

In this first example we will model a simple steel cantilever, to see how the simple structure responds to load. The problem is sketched below.



The problem description is as follows:

Geometry:	1.200 x .180 x .010 m
Load:	18000 N applied at the end of the cantilever in the string direction
Supports:	the base is fixed in all degrees of freedom, all other boundaries are free.
Material:	Steel, with E = 200e9 Pa (2e11 N/m2), σy = 3e8 Pa, Et= 1e9 Pa
Units:	N, m, Pa, s NOTE: The user must use consistent units

Step 2: estimate expected results (analytically):

The bar has the following properties:

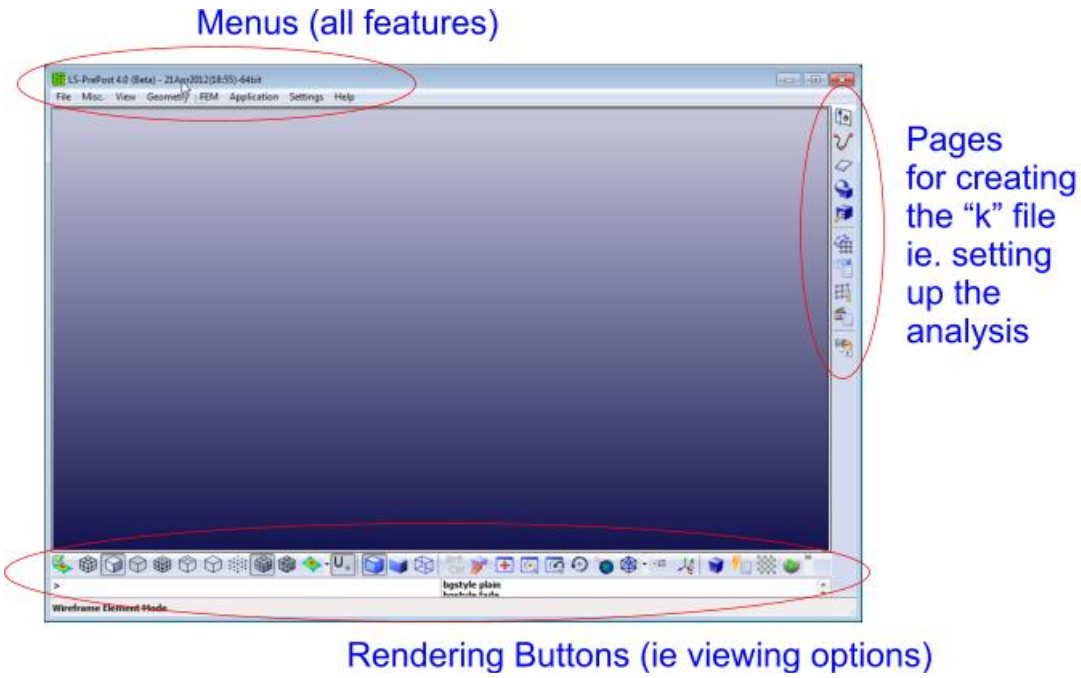
Moment of inertia :	I = 1/12 t h3 = 10 x 180^3 /12 = 4.860e06 mm4
Section Modulus:	Z = I/(h/2) = 54000 mm3
Base Bending Moment:	M = 18000 x 1200 N-mm
Maximum stress (at base):	sig = M/Z = 400 N/mm2 or MPa
Maximum deflection:	d=FL3/(3 EI) = 10 mm

It is likely that the LSDYNA results will be close to these, and close to the ANSYS analysis of this problem that we did in the course 5003. The % error will depend on the assumptions, but differences of say +- 10% would not be unusual.

Step 3: open LS-PrePost

1) First, Open LS-PrePost which should be on the desktop, with Icon  or 

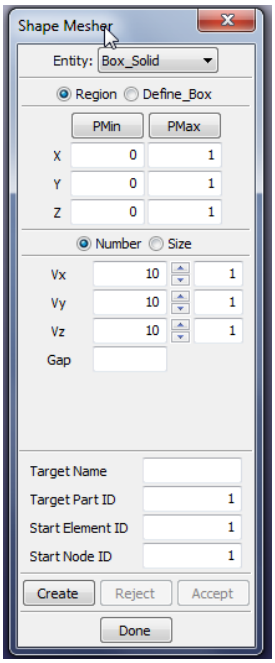
2) The window will be as shown below. There are pull down menus at the top of the screen, a variety of rendering buttons for display options at the bottom, and a set of modeling and editing icons on the right side. There is an older version of the interface that you can see by pressing F11. You might want that interface if you want to follow an old tutorial example (many are available on line). We will use the new GUI icon interface.



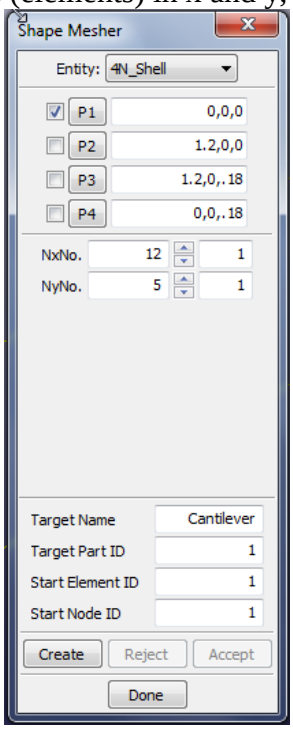
Step 4: Directly create the FE model (nodes and element mesh)

1) Click the **element and mesh** button:  . which opens up a new set of icons for meshing.

2) Click the **shape mesher** button:  . which opens a dialog window to directly create a mesh. (Note: in this case there will be no CAD model - we go straight to a finite element mesh)

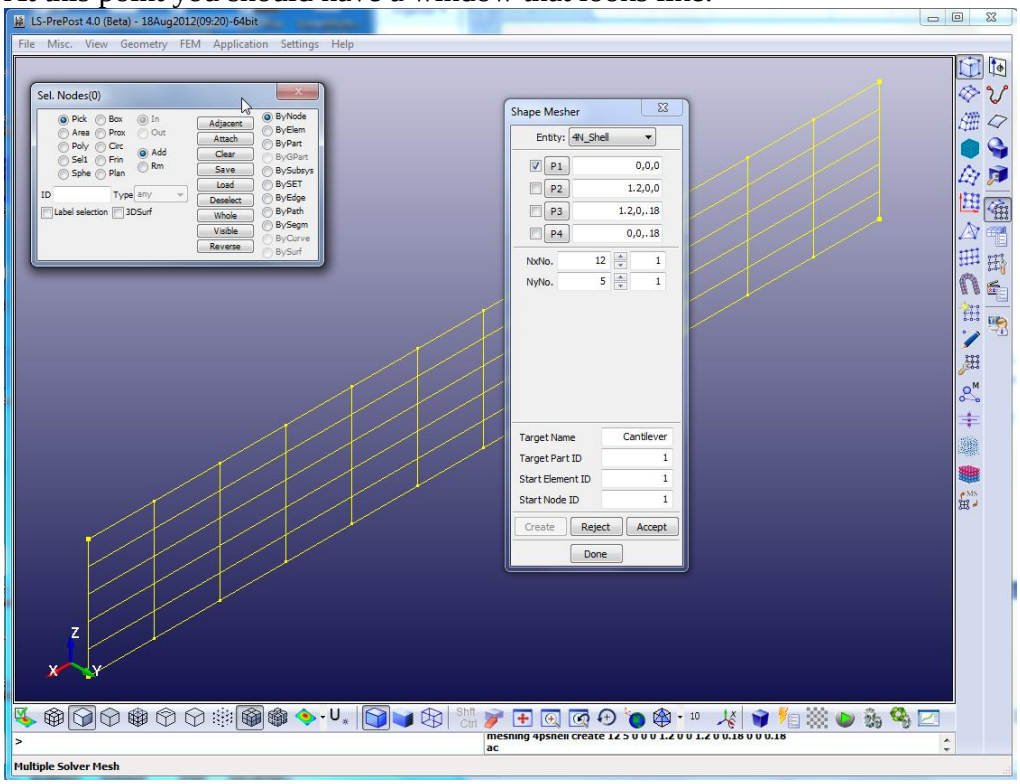
The initial dialog look like: 

In the **Entity** box at the top, select: **4N_Shell** and fill in the 4 corner coordinates, the number of divisions (elements) in x and y, and the **TargetName**,

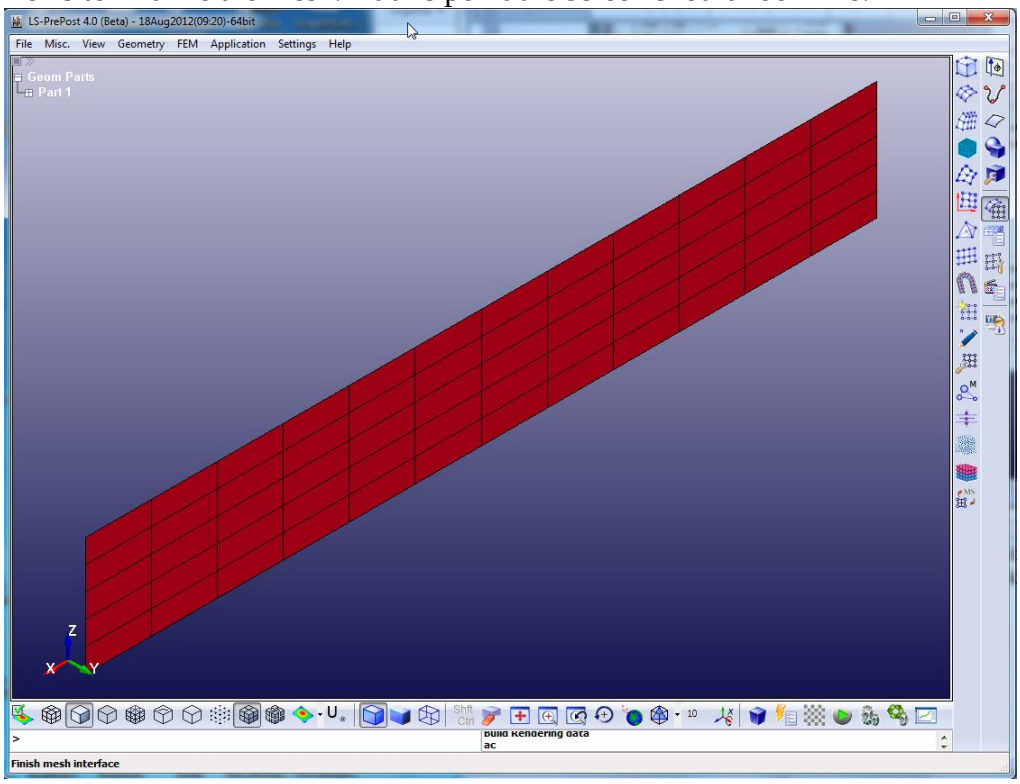
so it looks like this: 

When you click on **Create**, a trial mesh is created. To see it you will need to change the view to isometric by clicking on the **IsoMetric** icon at the bottom of the LSDyna window: 

At this point you should have a window that looks like:



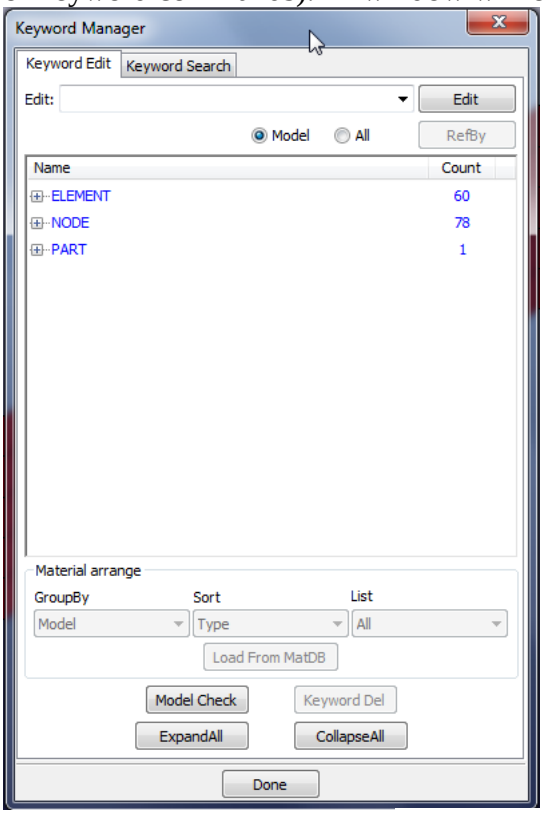
You can edit the values, and even reject the trial mesh. If it's ok you click **Accept** and **Done** to finalize the mesh. At this point the screen should look like:

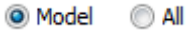


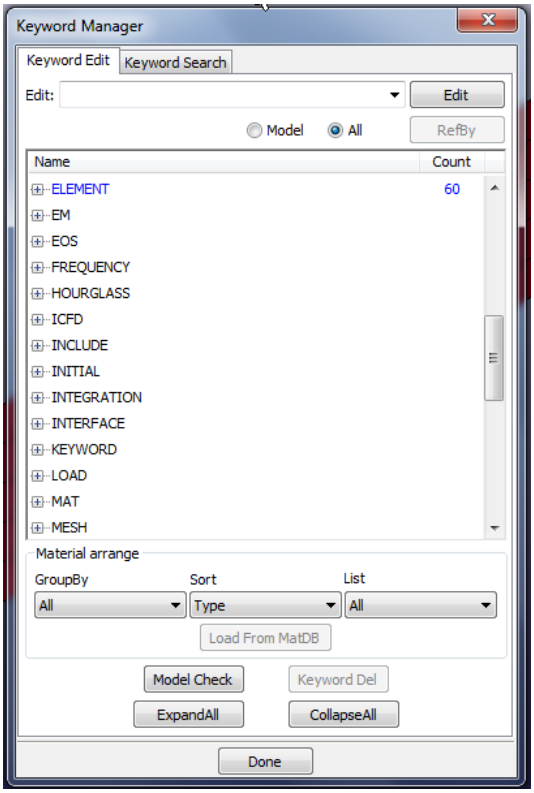
Step 5: Define the material properties

1) Click the **model and part** button:  which will bring up a new set of icons.

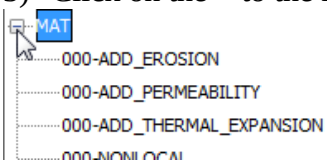
2) Click the **keyword manager** button:  which opens a dialog window to directly create keywords (Note: LS-Dyna files are called .k files because they are essentially a list of keyword commands). A window will open that looks like:

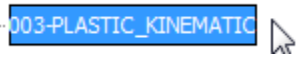


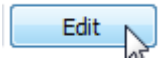
At this point the model option  is selected, and there are 3 categories of keyword already there (element, node and part). Note that there are 60 elements, along with the 78 nodes and 1 part. The part was created automatically when we created the mesh (a mesh has nodes and elements). Select the option and the window will display all the possible keywords:



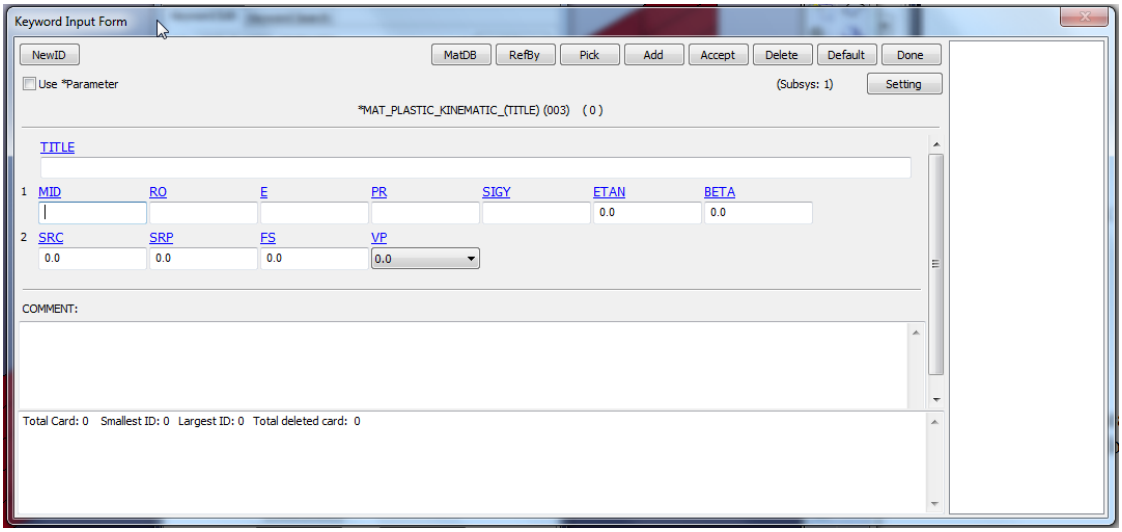
3) Click on the + to the left of **MAT** to open the material options:



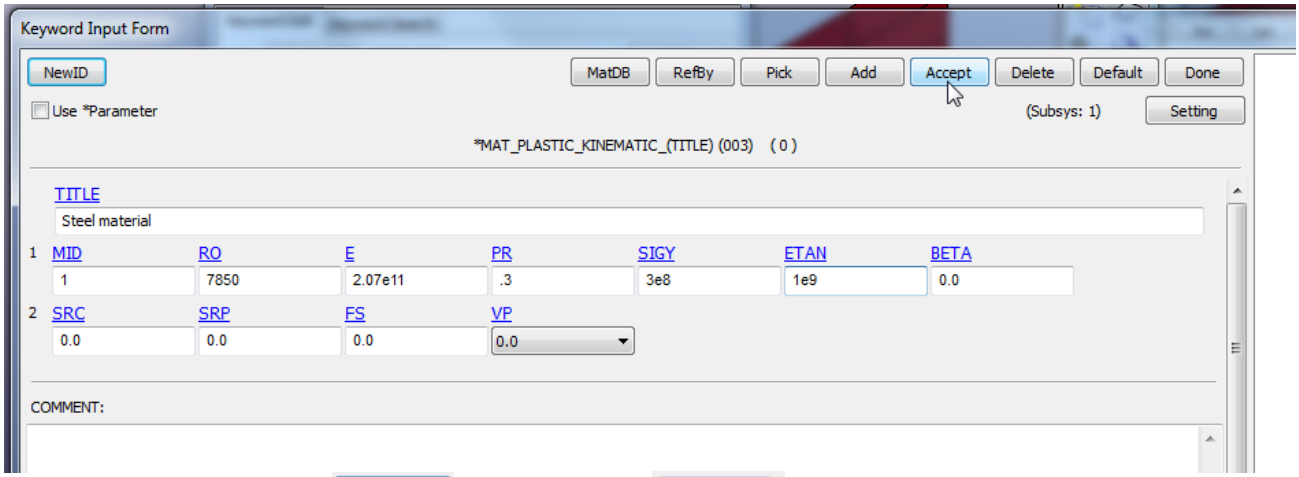
4) Select 

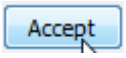
and then click on 

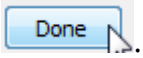
near the top of the window. A **Keyword Input Form** will open to allow input of the properties for this material model:

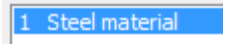


5) click on **NewID** and fill in the form. Enter values for TITLE, RO, E, PR, SIGY, ETAN. Other options are left at default values;



then press 

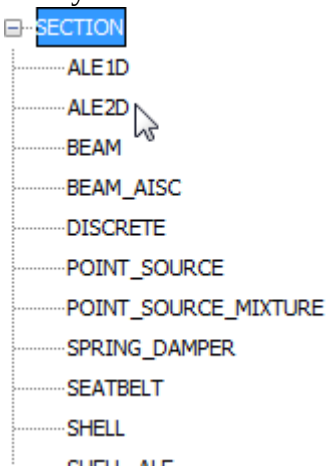
and then press 

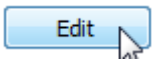
. After pressing Accept, you will see 

in the right hand space of the form. This material model describes elasto-plastic behavior in steel. This is similar to the bilinear-kinematic material property that can be specified in ANSYS.

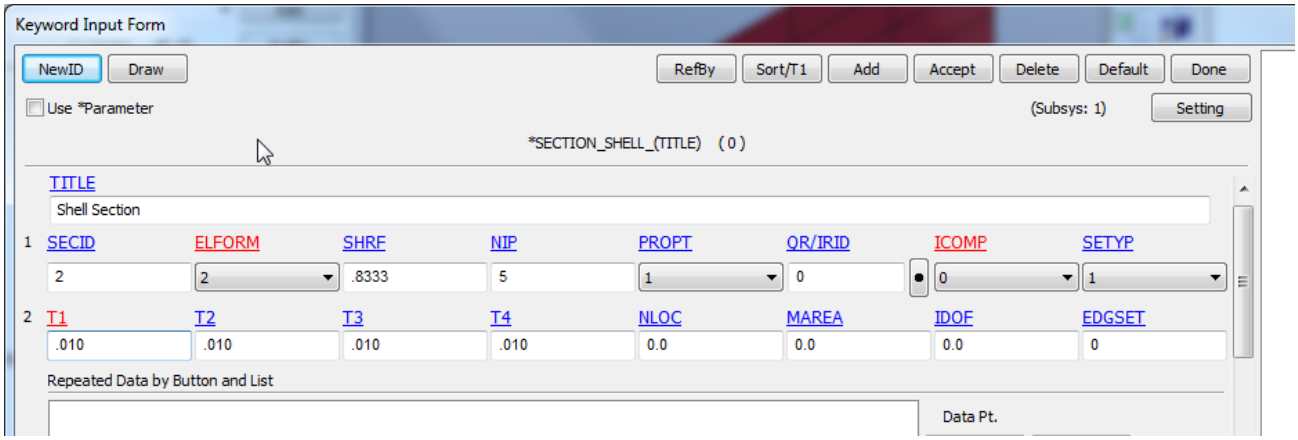
Step 6: Define the section properties

1) In LSDyna the section properties contain information about the element parameters and the type of physics employed. In the **Keyword Manager** window, scroll down until you see the **Section** Keyword. Click on the + and then select **Shell**



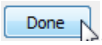
2) Click on 

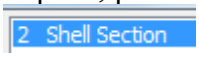
near the top of the window. A **Keyword Input Form** will open to allow input of the properties for this section model. click on **NewID** and fill in the form as follows:



The form shows that the plate elements will have a thickness of 0.010 m (10mm), that there are 5 integration points through the thickness (NIP=5) and the shear factor SHRF is 0.833. When you enter the first thickness T1, all other values (T2 etc) are made identical. These are the thicknesses of all for corners of this shell section.

When complete, press 

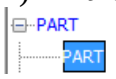
and then press 

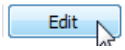
. After pressing Accept, you will see 

in the right hand space of the form.

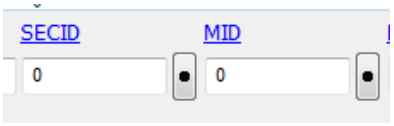
Step 7: Apply the material and section properties to the model (to the 'Part')

1) In the **Keyword Manager** window, select the **Part** Keyword, press + and select **Part**:



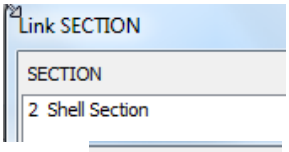
Then press 

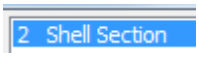
. The part is defined, but the section and material information is not linked to the part (there are zeros under SECID and MID);

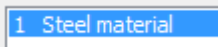


.

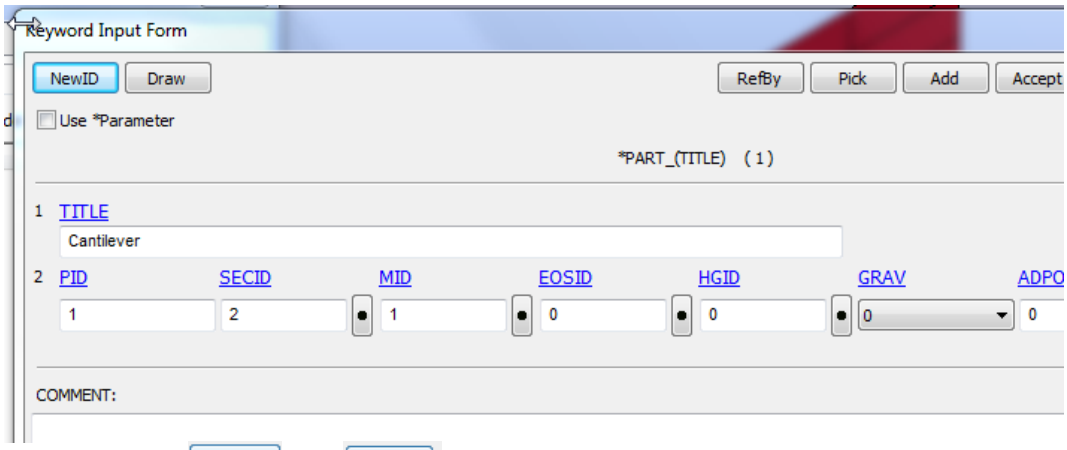
Press the black dot to the right of SECID, and you see the following window open;



Select 

and press **Done**. When you do the SECID becomes 2. Click on the black dot to the right of MID and select 

and press **Done**. With this the window will look like:



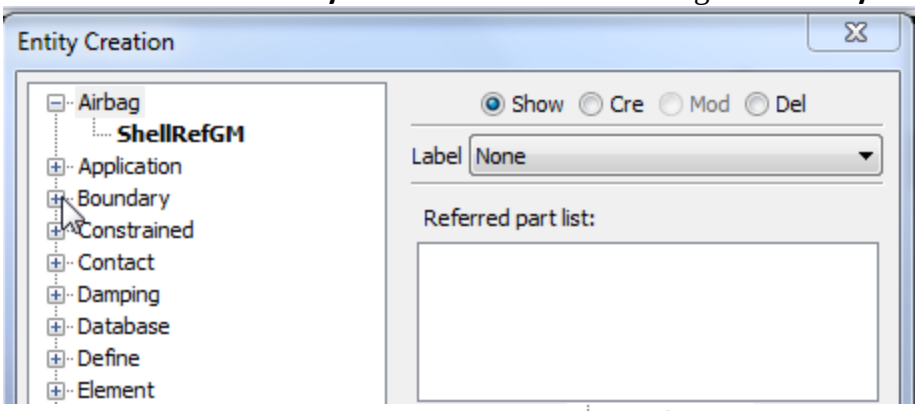
then press Accept and Done. We now have a finite element model with material and section properties.

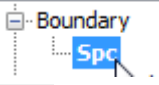
Step 8: Apply boundary conditions (fix the end of the bar')

Rather than using the **Keyword Manager** window, we will use a special tool to create the boundary conditions (an app) :

The **Model and Part** button already selected: 

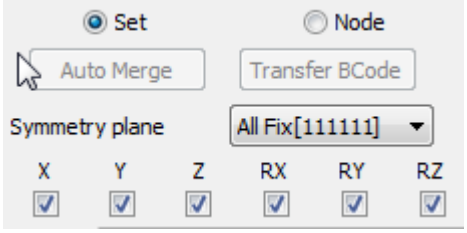
Click on the **Create Entity** button.  This will bring us an **Entity Creation** form.



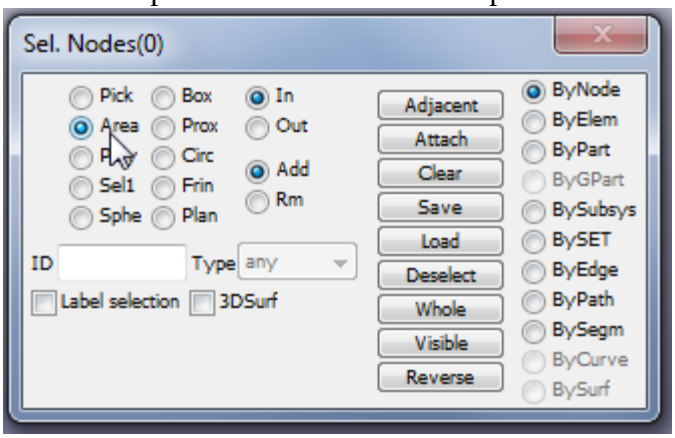
Select the + **Boundary** and then select **Spc** : 

Now change to Create mode by selecting 

Make sure the **Set** option selected and all 6 degrees of freedom selected:

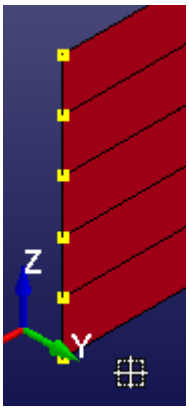


Now you need to select the nodes at the left end of the model. A **Set Nodes** dialog box should be open: Make sure the **Area** option is selected:

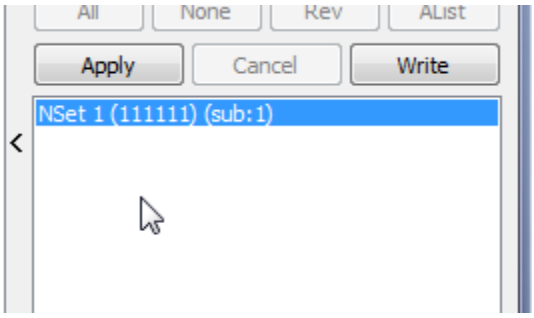


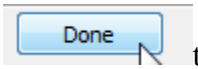
Now the cursor has changed to a selection box. Select all the nodes at the end of the model and hit 

:



You will see that a set of nodes has been created:



Now hit 

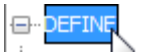
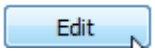
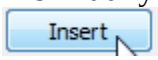
to close this activity.

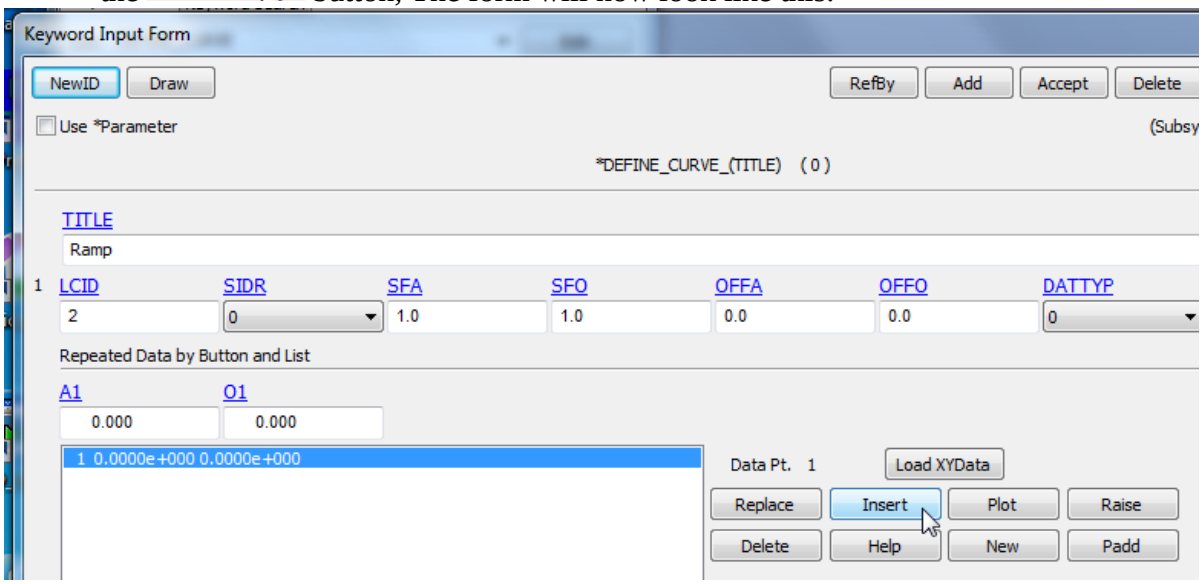
Step 9: Apply a force to the free end of the bar.

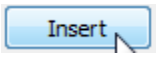
This activity requires a few sub-steps. We need to define a curve to tell LSDyna how to apply the force in time (remember - everything in LSDyna happens in real time). And we need to know the node number of the top right corner of the model.

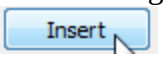
1) Define a standard ramp curve:

Click the **model and part** button:  and click the **keyword manager** button: 

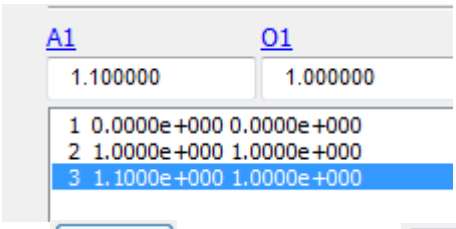
Now Scroll to **DEFINE** , open it (hit +)  and select  and then the  key at the top. This will bring up a **Keyword Input Form** to create the curve. The form is initially blank. Hit **NewID**, give it the Title *Ramp* and with A1 and O1 both = 0.0 hit the  button, The form will now look like this.

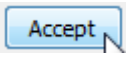
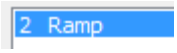


Now edit the A1 and O1 boxes to 1 and 1, and hit  again.

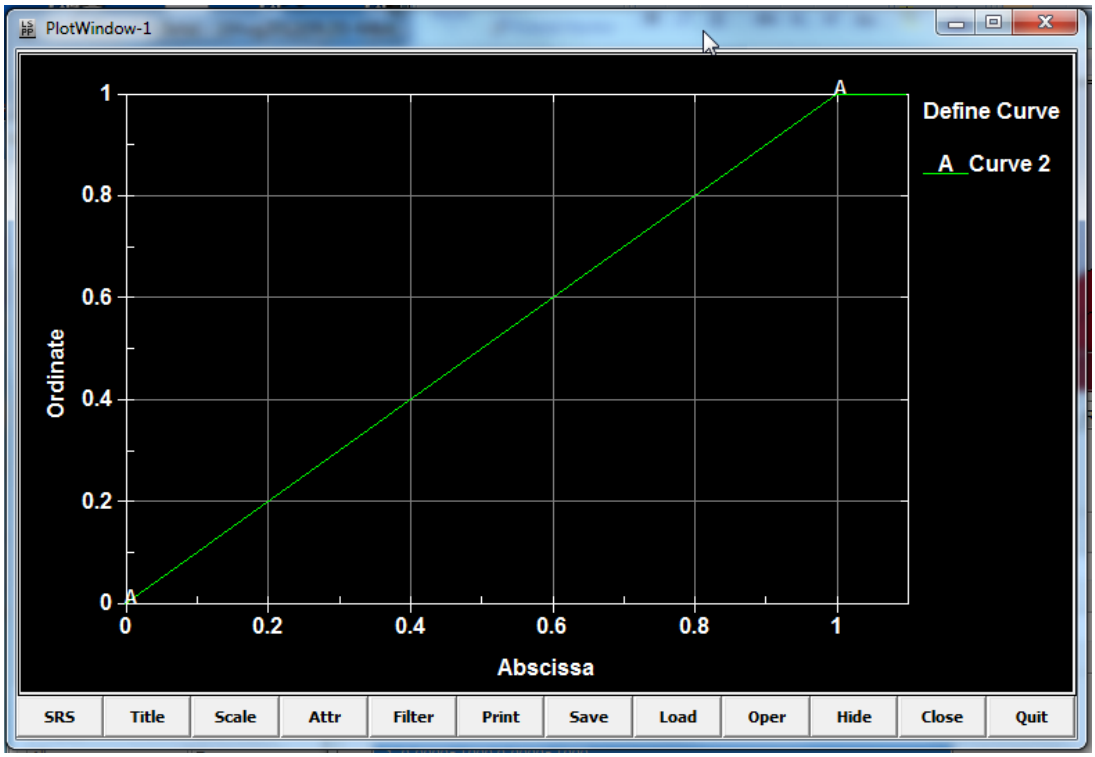
Now edit the A1 and O1 boxes to 1.1 and 1, and hit  again.

Now the table will look like:



Hit  and you will see  in the right hand side of the form.

To check the curve, hit the Plot button to see the curve plotted. You should see:



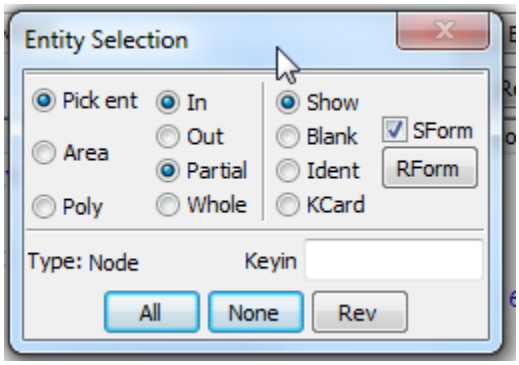
Now **Close** the plot window
If you did not put the small plateau on the curve, it would drop to zero at the end and would give strange results. We will simulate 1 sec of time, so this curve has an X range of 0 to 1 (plus 10% we won't use). The vertical range is 1, but we will scale that to 18000N. If you planned to simulate 2 seconds, this curve would need an x range of 0 to 2.

Now hit Done to close the **Keyword Input Form**.

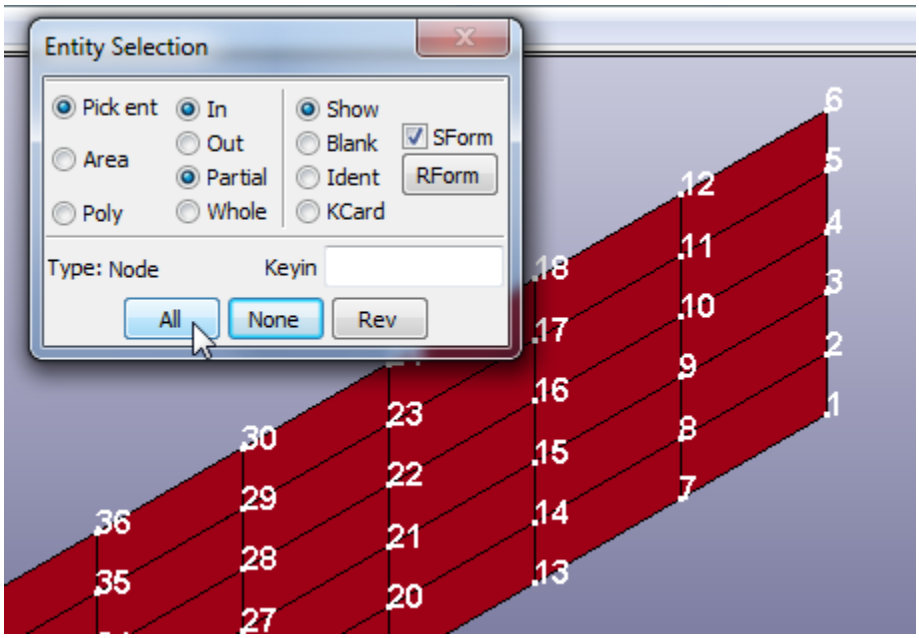
2) Find the Node number at the point for the load.

Go back to the **Keyword Manager** (select Model to simplify the task) and select **NODE** (and **NODE** again)

Now hit Edit and you will see the full list of nodes. You could edit the nodes, but we just want to see them. Select the Draw command on the **Keyword Input Form**. The form then shrinks and an **Entity Selection** window appears:



Click the All button to see all the nodes labeled:

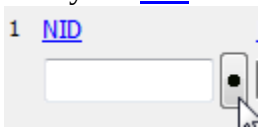


We will apply our force to the node with number 6.
Hit None to remove the labels. Restore the **Keyword Input Form** window and hit Done.

3) Define the force.

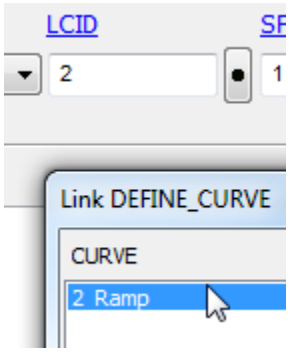
Return to the **Keyword Manager** window, select All and then LOAD

Now select NODE_POINT, and Edit. Now you see the input form. Hit the black dot by the NID box:



and select node 6. and hit Done. In the DOF box select the **3** to specify the z direction.

Hit the black dot for the LCID box to specify the load curve. Select the ramp curve we defined earlier and hit Done.



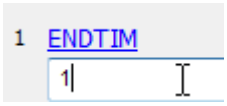
Now scale the force by 18000. Type *-18000* into the SF box. This will mean that the load will ramp up from 0 to 18000N over the course of 1 second. And the load will act downwards (-z).
(Note - if you want to see help for one of the boxes - just put the cursor in the box and look at the bottom of the window)
hit Accept and Done.

Step 10: Specify the duration of the simulation.

Return to the Keyword Manager and select CONTROL

Scroll down to find TERMINATION and then click Edit

Put 1 in the ENDTIM box :

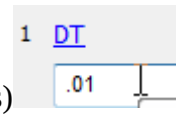


hit Accept and Done

Step 11: Specify output frequency.

LSDyna will solve the problem with a very short timestep. There would likely be far too much output if you were to look at all the data for every solution timestep. Instead you can specify the frequency of the output. Return to the **Keyword Manager** and select the **DATABASE** and **BINARY_D3PLOT** Keywords. and then click Edit

Specify an output timestep DT for the d3plot data of 0.01 seconds (10ms)

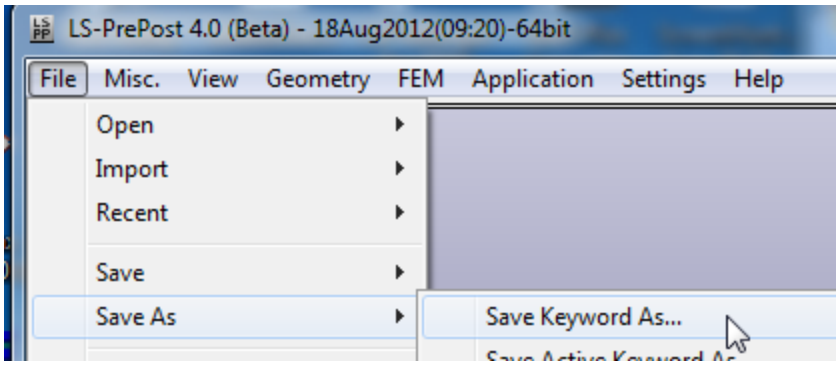


hit Accept and Done

In the **Keyword Manager** window hit Done to close it

Step 12: Save the .k file

Under the **File** menu select the **Save Keyword As** command and save the file as *Cant1.k* in its own folder (to keep the output files all in one place)
Note - you will have to type *Cant1.k* , as LSPrePost will not automatically add the .k extension.

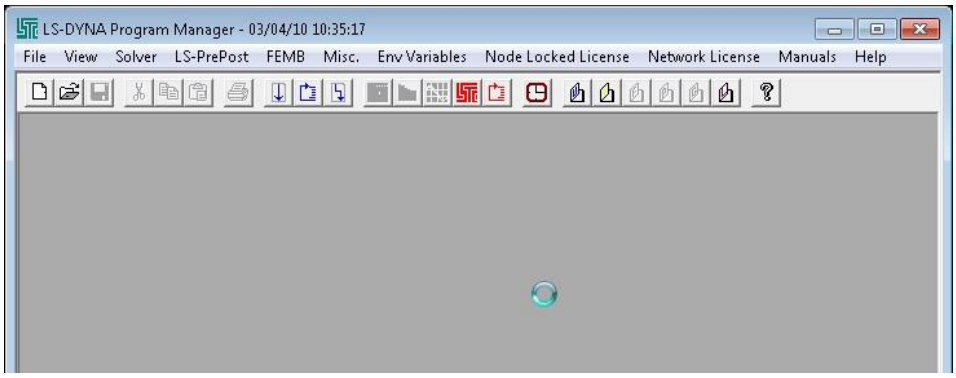


The .k file is a simple text file with all the problem information listed as either Keywords (lines starting with *) or associated data (no *) . There are also comments (lines beginning with $#). The .k file that we have just created is in the appendix below. You should examine the .k file to see how the problem input data is represented.

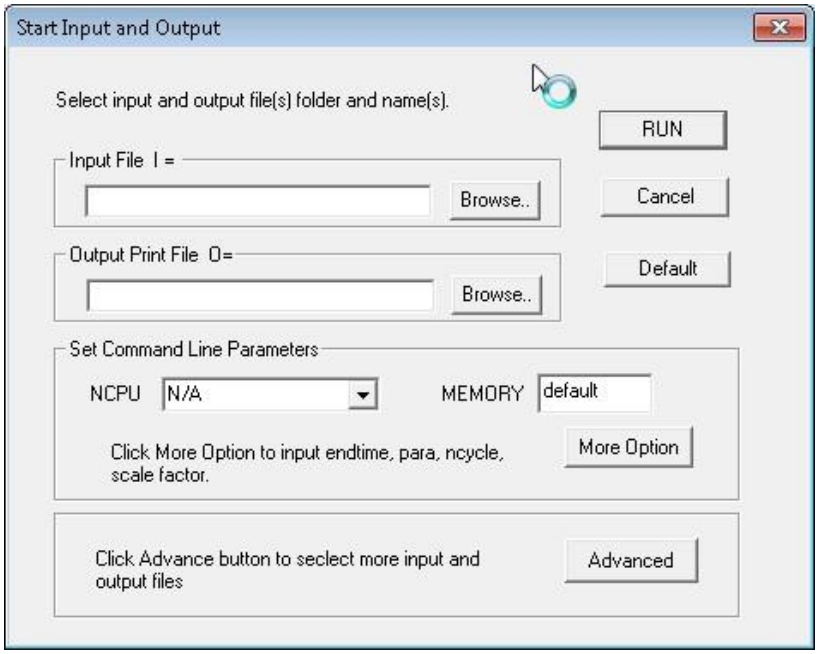
Step 13: Run the LS-Dyna Analysis

1) Click on the **LS-Dyna Manager** Program 

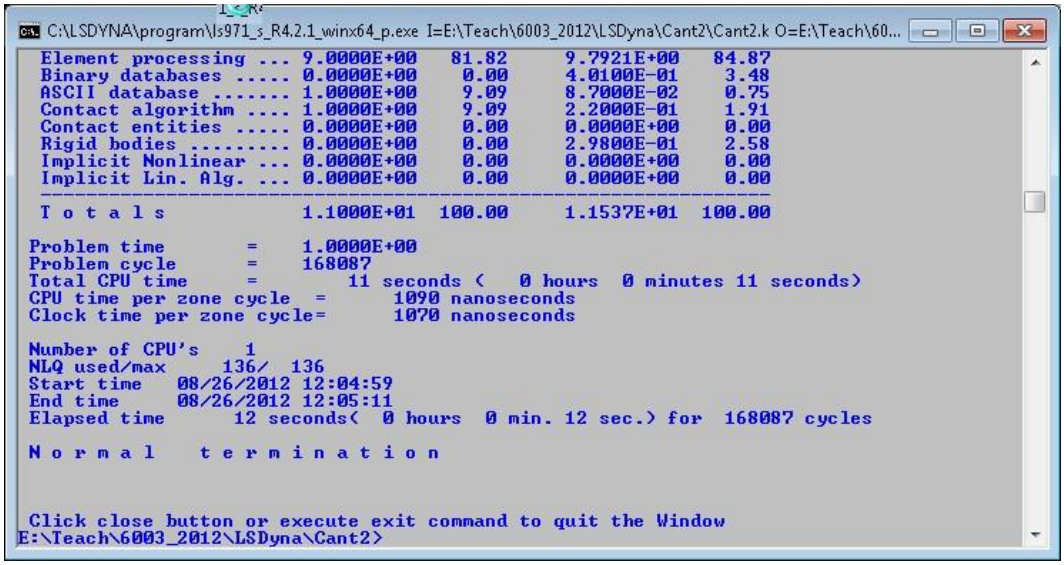
, which opens the window.:



In the top menu, select **Solver** and select **Start LS-Dyna** Analysis. Next you see the input screen. Use the **Browse** button to select the *Cant1.k* file. By default the same folder as the .k file is in will be the folder where the output is sent.



When the .k file is selected, press the **RUN** button. A command window will pop up showing the computation steps. When finished is should say **`Normal termination`** :

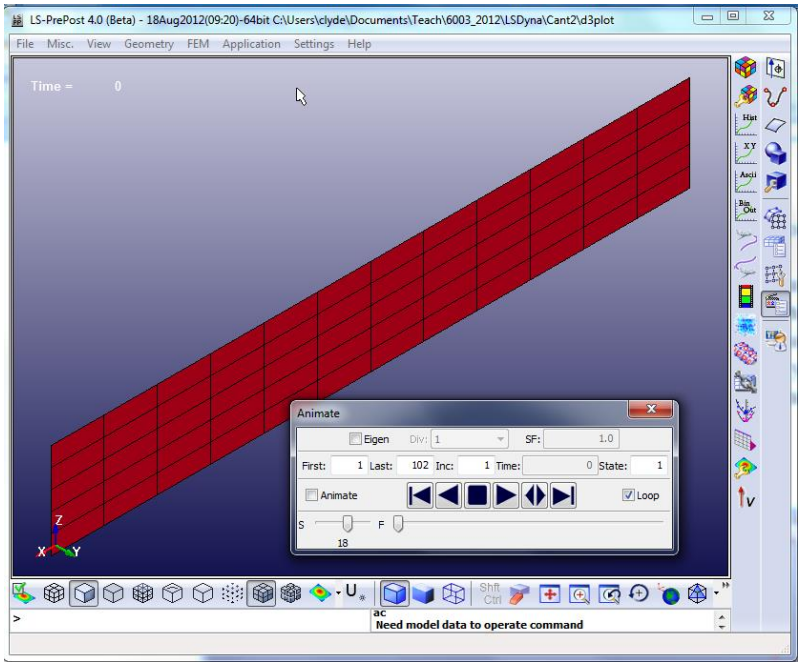


Step 14: Examine results of the LS-Dyna Analysis

1) Re-Open LS-PrePost.

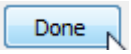
2) From the **File** Menu, Select **Open** and **LS-Dyna Binary Plot**.

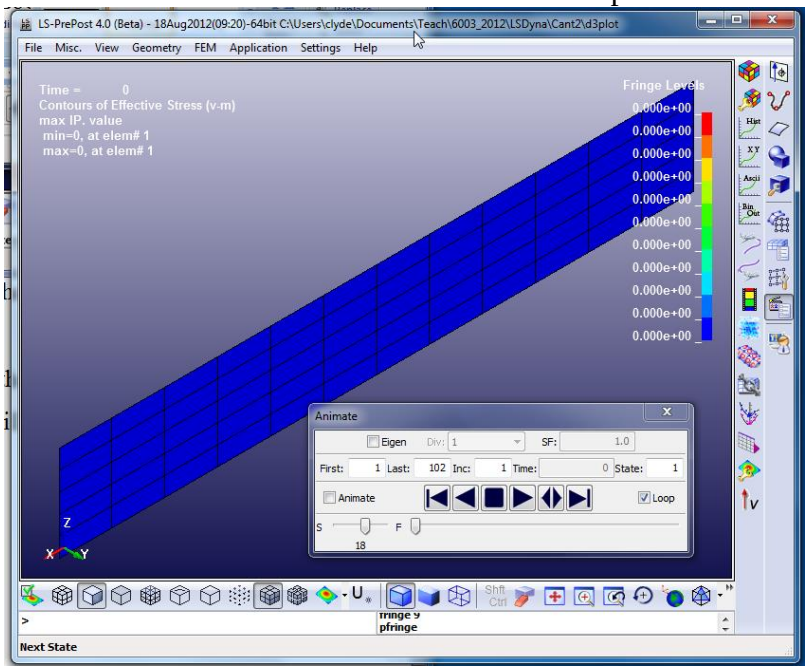
3) In the same folder as the .k file should be a file called **d3plot** - open it. Now you should see the screen as shown below.



You can hit the Play button to annimate the bar over the 1 second of analysis.

To see more data, hit the **Post** button  , then hit the **Fringe Component** 

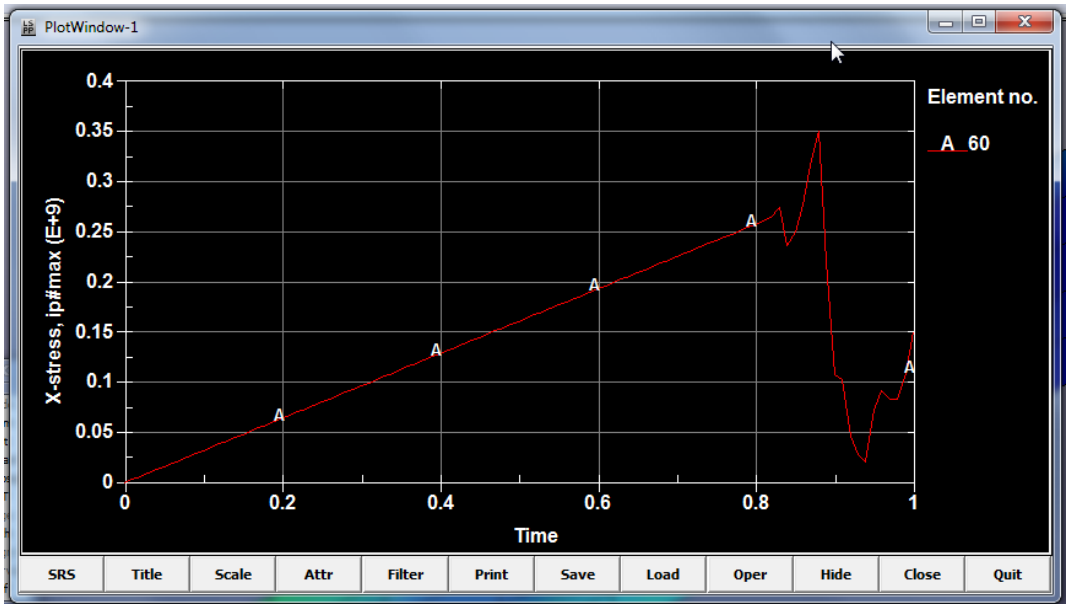
select **Von Mises Stress** and  and the plot will become:



Now you will see the animated stresses. You can select **Static** under the **Fringe Range** button  to keep a constant range of stress during the animation.

4) There are many more results than can be plotted. For example click on the **History** button 

, select the **Element** option, select one element on the model, select one output (say **X-stress**) and hit **Plot**. You should see something like:



Summary of Steps

Step 1: describe and sketch the problem:
Step 2: estimate expected results (analytically):
Step 3: open LS-PrePost
Step 4: Directly create the FE model (nodes and element mesh)

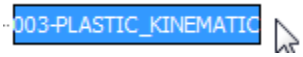
element and mesh button:  .

shape mesher button:  .

Entity box: **4N_Shell**

Step 5: Define the material properties

MAT to open the material options:



Step 6: Define the section properties

Section

Shell

Step 7: Apply the material and section properties to the model (to the 'Part')
Step 8: Apply boundary conditions (fix the end of the bar')

Boundary

Spc

Step 9: Apply a force to the free end of the bar.

Define a standard ramp curve:
Find the Node number at the point for the load.

LOAD

NODE_POINT

Step 10: Specify the duration of the simulation.

CONTROL

TERMINATION

Step 11: Specify output frequency.

DATABASE

BINARY_D3PLOT

Step 12: Save the .k file

Save Keyword As

Step 13: Run the LS-Dyna Analysis

should say **`Normal termination`** :

Step 14: Examine results of the LS-Dyna Analysis

Open LS-PrePost.

Open LS-Dyna Binary Plot.

d3plot

Appendix A - The .k file

```
$# LS-DYNA Keyword file created by LS-PrePost 4.0 (Beta) - 18Aug2012(09:20)
$# Created on Aug-26-2012 (09:05:32)
*KEYWORD
*BOUNDARY_SPC_SET
$#     nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         1         1         1
*SET_NODE_LIST_TITLE
NODESET(SPC) 1
$#      sid       da1       da2       da3       da4    solver
         1     0.000     0.000     0.000     0.000MECH
$#     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        73        74        75        76        77        78         0         0
*NODE
$#   nid               x               y               z      tc      rc
       1           0.000           0.000           0.000       0       0
       2           0.000           0.000           0.036000    0       0
       3           0.000           0.000           0.072000    0       0
       4           0.000           0.000           0.108000    0       0
       5           0.000           0.000           0.144000    0       0
       6           0.000           0.000           0.180000    0       0
       7           0.100000        0.000           0.000       0       0
       8           0.100000        0.000           0.036000    0       0
       9           0.100000        0.000           0.072000    0       0
      10           0.100000        0.000           0.108000    0       0
      11           0.100000        0.000           0.144000    0       0
      12           0.100000        0.000           0.180000    0       0
      13           0.200000        0.000           0.000       0       0
      14           0.200000        0.000           0.036000    0       0
      15           0.200000        0.000           0.072000    0       0
      16           0.200000        0.000           0.108000    0       0
      17           0.200000        0.000           0.144000    0       0
      18           0.200000        0.000           0.180000    0       0
      19           0.300000        0.000           0.000       0       0
      20           0.300000        0.000           0.036000    0       0
      21           0.300000        0.000           0.072000    0       0
      22           0.300000        0.000           0.108000    0       0
      23           0.300000        0.000           0.144000    0       0
      24           0.300000        0.000           0.180000    0       0
      25           0.400000        0.000           0.000       0       0
      26           0.400000        0.000           0.036000    0       0
      27           0.400000        0.000           0.072000    0       0
      28           0.400000        0.000           0.108000    0       0
      29           0.400000        0.000           0.144000    0       0
      30           0.400000        0.000           0.180000    0       0
      31           0.500000        0.000           0.000       0       0
      32           0.500000        0.000           0.036000    0       0
      33           0.500000        0.000           0.072000    0       0
      34           0.500000        0.000           0.108000    0       0
      35           0.500000        0.000           0.144000    0       0
      36           0.500000        0.000           0.180000    0       0
      37           0.600000        0.000           0.000       0       0
      38           0.600000        0.000           0.036000    0       0
      39           0.600000        0.000           0.072000    0       0
      40           0.600000        0.000           0.108000    0       0
      41           0.600000        0.000           0.144000    0       0
      42           0.600000        0.000           0.180000    0       0
      43           0.700000        0.000           0.000       0       0
      44           0.700000        0.000           0.036000    0       0
      45           0.700000        0.000           0.072000    0       0
      46           0.700000        0.000           0.108000    0       0
      47           0.700000        0.000           0.144000    0       0
      48           0.700000        0.000           0.180000    0       0
      49           0.800000        0.000           0.000       0       0
      50           0.800000        0.000           0.036000    0       0
      51           0.800000        0.000           0.072000    0       0
      52           0.800000        0.000           0.108000    0       0
      53           0.800000        0.000           0.144000    0       0
      54           0.800000        0.000           0.180000    0       0
      55           0.900000        0.000           0.000       0       0
      56           0.900000        0.000           0.036000    0       0
      57           0.900000        0.000           0.072000    0       0
      58           0.900000        0.000           0.108000    0       0
      59           0.900000        0.000           0.144000    0       0
      60           0.900000        0.000           0.180000    0       0
      61           1.000000        0.000           0.000       0       0
      62           1.000000        0.000           0.036000    0       0
      63           1.000000        0.000           0.072000    0       0
      64           1.000000        0.000           0.108000    0       0
      65           1.000000        0.000           0.144000    0       0
      66           1.000000        0.000           0.180000    0       0
      67           1.100000        0.000           0.000       0       0
      68           1.100000        0.000           0.036000    0       0
      69           1.100000        0.000           0.072000    0       0
      70           1.100000        0.000           0.108000    0       0
      71           1.100000        0.000           0.144000    0       0
      72           1.100000        0.000           0.180000    0       0
      73           1.200000        0.000           0.000       0       0
      74           1.200000        0.000           0.036000    0       0
      75           1.200000        0.000           0.072000    0       0
      76           1.200000        0.000           0.108000    0       0
      77           1.200000        0.000           0.144000    0       0
      78           1.200000        0.000           0.180000    0       0
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
  0.010000         0         0         0         0
$#   ioopt
         0
*SECTION_SHELL_TITLE
Shell Section
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         2         2  0.833300         5         1         0         0         1
$#      t1        t2        t3        t4      nloc     marea      idof    edgset
  0.010000  0.010000  0.010000  0.010000     0.000     0.000     0.000         0
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
  1.000000         0     0.000     0.000     0.000
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       7       8       2       1       0       0       0       0
       2       1       8       9       3       2       0       0       0       0
       3       1       9      10       4       3       0       0       0       0
       4       1      10      11       5       4       0       0       0       0
       5       1      11      12       6       5       0       0       0       0
       6       1      13      14       8       7       0       0       0       0
       7       1      14      15       9       8       0       0       0       0
       8       1      15      16      10       9       0       0       0       0
       9       1      16      17      11      10       0       0       0       0
      10       1      17      18      12      11       0       0       0       0
      11       1      19      20      14      13       0       0       0       0
      12       1      20      21      15      14       0       0       0       0
      13       1      21      22      16      15       0       0       0       0
      14       1      22      23      17      16       0       0       0       0
      15       1      23      24      18      17       0       0       0       0
      16       1      25      26      20      19       0       0       0       0
      17       1      26      27      21      20       0       0       0       0
      18       1      27      28      22      21       0       0       0       0
      19       1      28      29      23      22       0       0       0       0
      20       1      29      30      24      23       0       0       0       0
      21       1      31      32      26      25       0       0       0       0
      22       1      32      33      27      26       0       0       0       0
      23       1      33      34      28      27       0       0       0       0
      24       1      34      35      29      28       0       0       0       0
      25       1      35      36      30      29       0       0       0       0
      26       1      37      38      32      31       0       0       0       0
      27       1      38      39      33      32       0       0       0       0
      28       1      39      40      34      33       0       0       0       0
      29       1      40      41      35      34       0       0       0       0
      30       1      41      42      36      35       0       0       0       0
      31       1      43      44      38      37       0       0       0       0
      32       1      44      45      39      38       0       0       0       0
      33       1      45      46      40      39       0       0       0       0
      34       1      46      47      41      40       0       0       0       0
      35       1      47      48      42      41       0       0       0       0
      36       1      49      50      44      43       0       0       0       0
      37       1      50      51      45      44       0       0       0       0
      38       1      51      52      46      45       0       0       0       0
      39       1      52      53      47      46       0       0       0       0
      40       1      53      54      48      47       0       0       0       0
      41       1      55      56      50      49       0       0       0       0
      42       1      56      57      51      50       0       0       0       0
      43       1      57      58      52      51       0       0       0       0
      44       1      58      59      53      52       0       0       0       0
      45       1      59      60      54      53       0       0       0       0
      46       1      61      62      56      55       0       0       0       0
      47       1      62      63      57      56       0       0       0       0
      48       1      63      64      58      57       0       0       0       0
      49       1      64      65      59      58       0       0       0       0
      50       1      65      66      60      59       0       0       0       0
      51       1      67      68      62      61       0       0       0       0
      52       1      68      69      63      62       0       0       0       0
      53       1      69      70      64      63       0       0       0       0
      54       1      70      71      65      64       0       0       0       0
      55       1      71      72      66      65       0       0       0       0
      56       1      73      74      68      67       0       0       0       0
      57       1      74      75      69      68       0       0       0       0
      58       1      75      76      70      69       0       0       0       0
      59       1      76      77      71      70       0       0       0       0
      60       1      77      78      72      71       0       0       0       0
*DEFINE_CURVE_TITLE
Ramp
$#     lcid      sidr       sfa       sfo      offa      offo    dattyp
         2         0  1.000000  1.000000     0.000     0.000         0
$#               a1                ol
            0.000             0.000
        1.000000          1.000000
        1.100000          1.000000
*MAT_PLASTIC_KINEMATIC_TITLE
Steel material
$#     mid        ro         e        pr      sigy      etan      beta
         1 7850.00002.0700E+11  0.300000 3.0000E+8 1.0000E+9     0.000
$#     src       srp        fs        vp
     0.000     0.000     0.000     0.000
*PART
$# title
Cantilever
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         2         1         0         0         0         0         0
*LOAD_NODE_POINT
$#     nid       dof      lcid        sf       cid        m1        m2        m3
         6         3        2-18000.000         0         0         0         0
*END
```

Self Study Exercises: Student:_____

For each of these exercises, be prepared to show the instructor your results.

Exercise #1 – Change the material model . Open the .k file. Find and delete the material model. Now create a new MAT model using 001-Elastic (ie no yielding) re-save the model as Cant2 in a new folder (make sure output goes to new folder). Run the model and fill in table below for time = 1 sec.

Deflection at end	Original model	MAT 001-Elastic
deflection at end [mm]:		
Eqv. Stress at base [MPa]		
Eqv.Stress at end [MPa]		
Comments: ?		

Ex#1 Initials of Instructor_____

Exercise #2 – Redo the analysis starting with a CAD step. Start by drawing a plane and then use the **Auto mesher** to make a mesh of triangles. Repeat all the rest of the steps.

Deflection at end	With Triangular mesh
deflection at end [mm]:	
Eqv. Stress at base [MPa]	
Eqv.Stress at end [MPa]	
Comment:	

Ex#2 Initials of Instructor_____