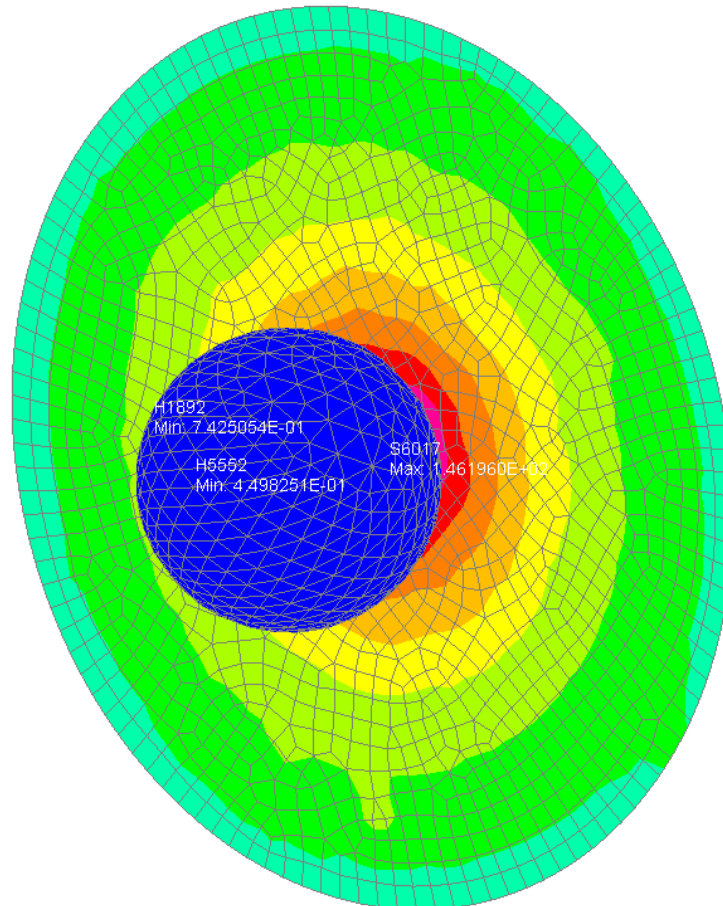# First Dyna/Primer Tutorial

## Laurence Marks

January 2022

In this first tutorial we use Primer and LS-Dyna to model a steel ball impacting an aluminium plate. The mesh is available as a Nastran BDF file (actually created in Abaqus). By completing this exercise you should learn about the program structure of Primer and Dyna, how to define a basic explicit solution and how to look at the results.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com   P a g e | **1**

Problem Description

In this problem a steel ball (25mm diameter) hits an aluminium disc at 10m/s. The high speed, short duration of the event, with the ball free to move, makes it an ideal application of an explicit solver.
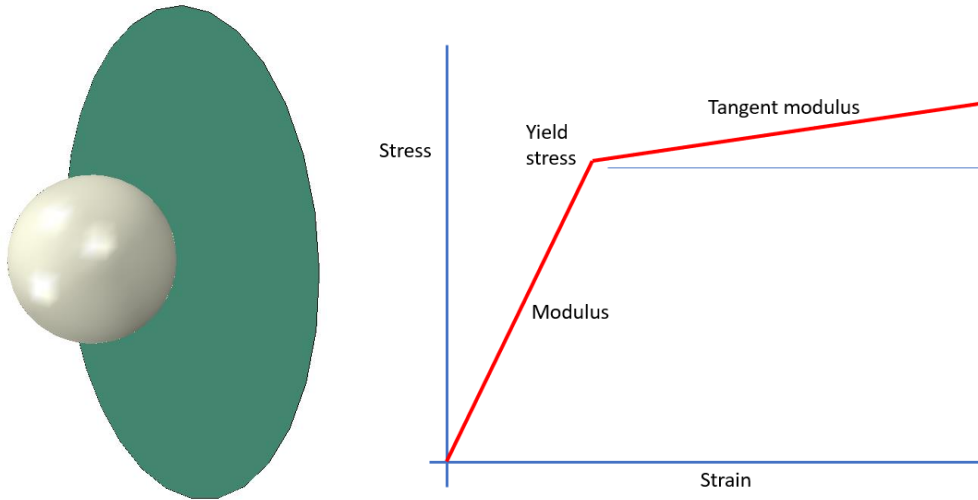

The model is solved twice, initially with elastic material properties, then with plastic properties. In the second case we can observe plastic deformation and hence a permanent shape change. We keep the steel properties linear but use a bi-linear material definition for the aluminium plate.

The ball is meshed using lower order tetrahedral elements, the aluminium plate with lower order, thin shell elements.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com   P a g e | **2**
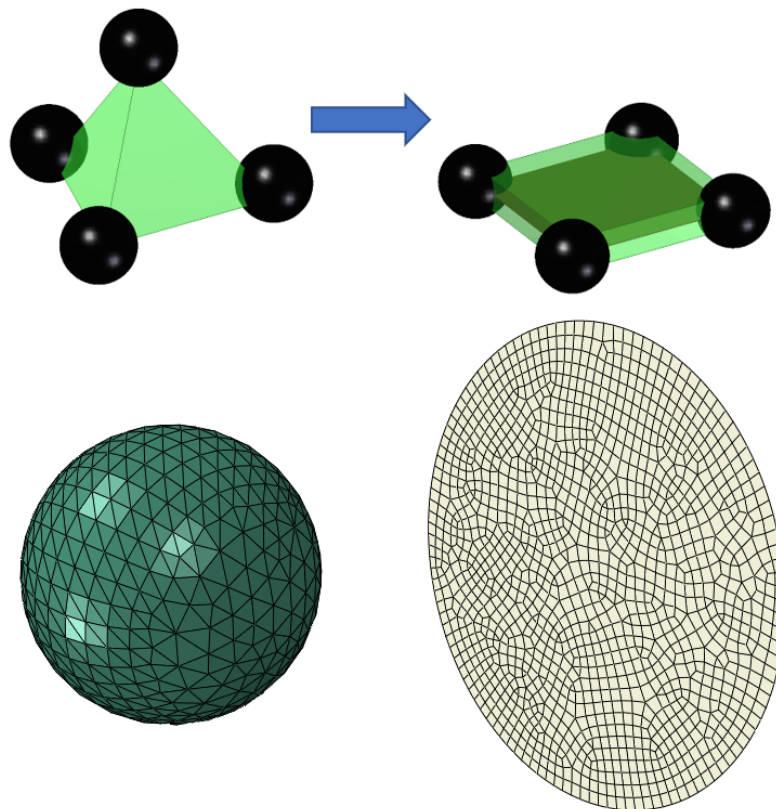
Step 1: Read meshed model from Abaqus via Nastran BDF file
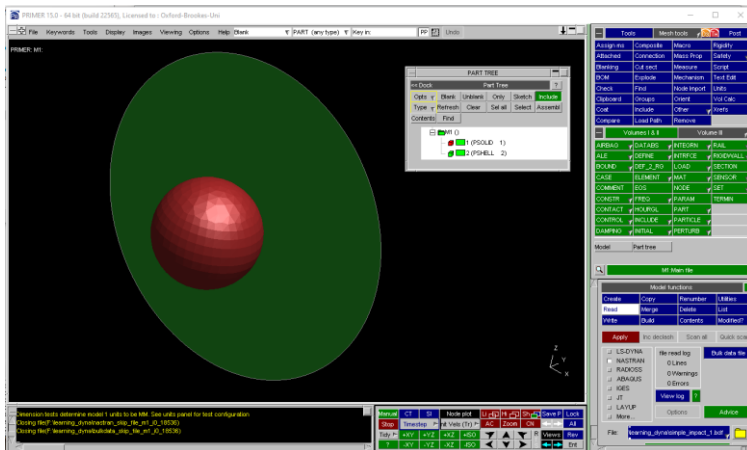
Question 1: Where do you switch on the element boundaries?

Translate abaqus inp file as shown below

```
F:\learning_dyna> abaqus tonastran job=simple_impact_1
looking for an .inp file ...
Old job files exist. Overwrite? (y/n): y
Abaqus JOB simple_impact_1
BEGIN toNastran FILE TRANSLATION
running traditional toNastran ...
Run C:\SIMULIA\EstProducts\2020\win_b64\code\bin\aba2nas.exe
------------------------------------------------
Abaqus/toNastran Release 6.20-1
  reading Abaqus input: simple_impact_1.inp
creating Nastran input: simple_impact_1.bdf
writing diagnostics to: simple_impact_1.log
WARNING: some Abaqus keywords were ignored. See .log file for details
NOTE: wrote 2613 grids.
NOTE: wrote 7099 elements.
NOTE: wrote 2 materials.
END toNastran FILE TRANSLATION
Abaqus JOB simple_impact_1 COMPLETED

F:\learning_dyna>
```
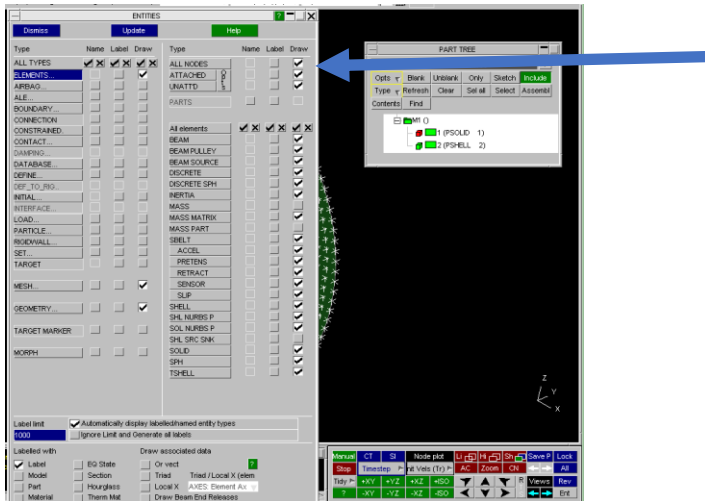
Step 2: Fix periphery of target

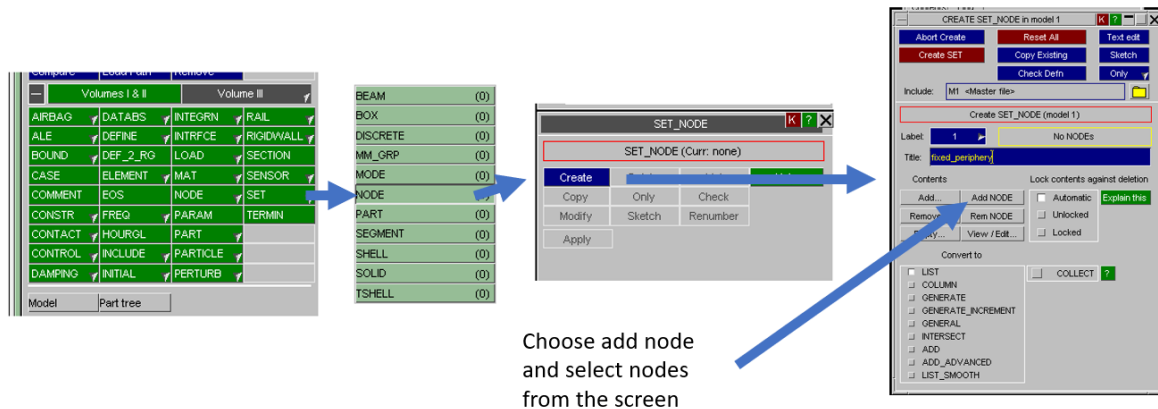Create node set for definition of constraint (SPC) (much easier than doing this as a single operation)
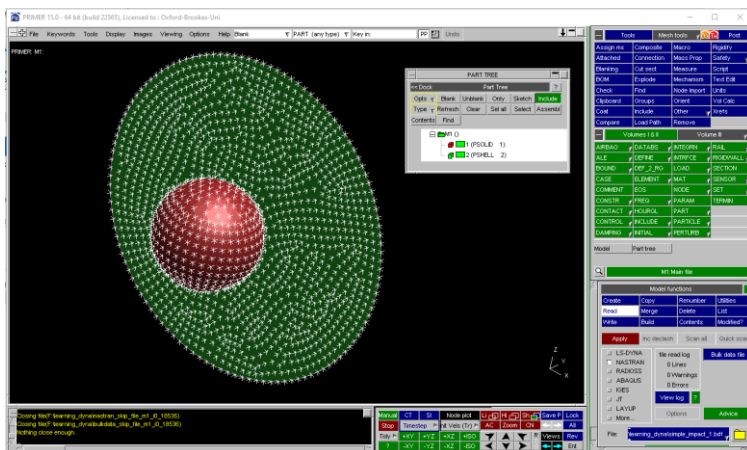
Switch on display of nodes – "ent" on display menu region, followed by select drawing all nodes, as shown below.
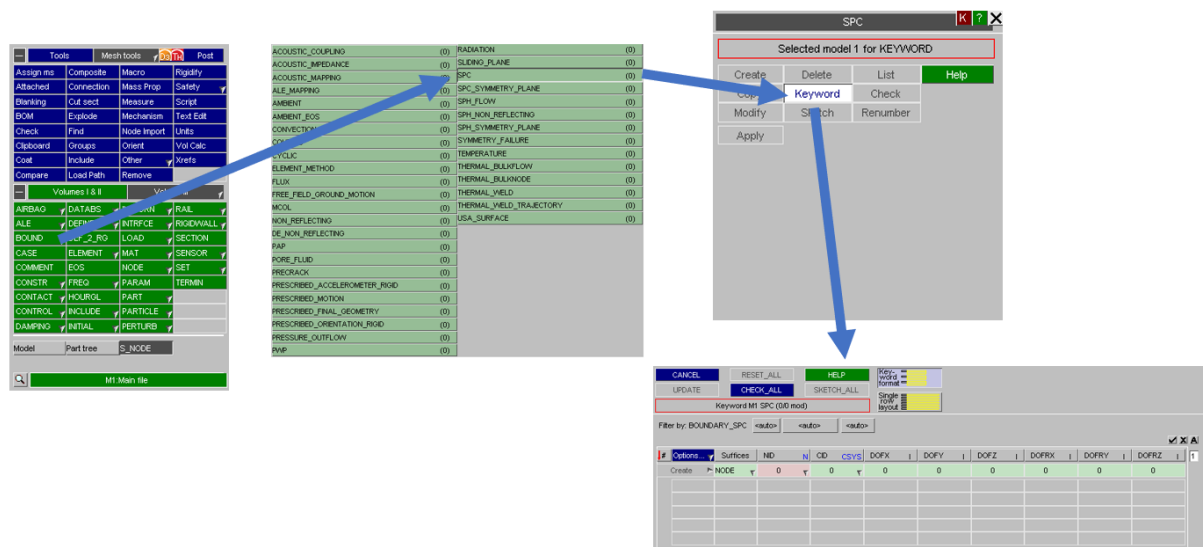
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com   P a g e | **3**

Now create a set, call it something like "fixed_periphery_nodes"

Choose add node and select nodes from the screen

Select "add items" and create set. Now we need to create the boundary condition, called an SPC in Primer.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com   P a g e | **4**

The keyword editor dialog is used to define the actual nature of the SPC. We are going to fix all degrees of freedom.

1. Change the zeros to 1's in the DOF's that you want to fix – that's all of them.
2. Select nodes (NSID select the set we created) then hit create and update.
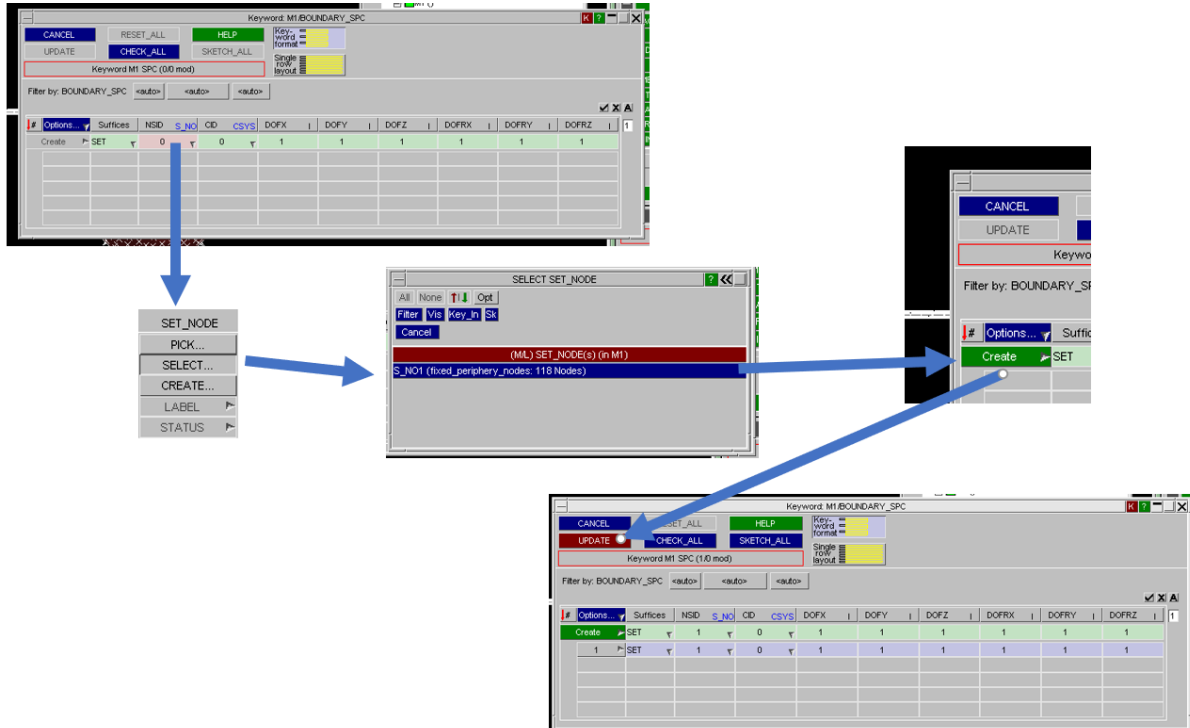
You should now have fixed the periphery of the plate. That was simple wasn't it? Its one command in Abaqus CAE.
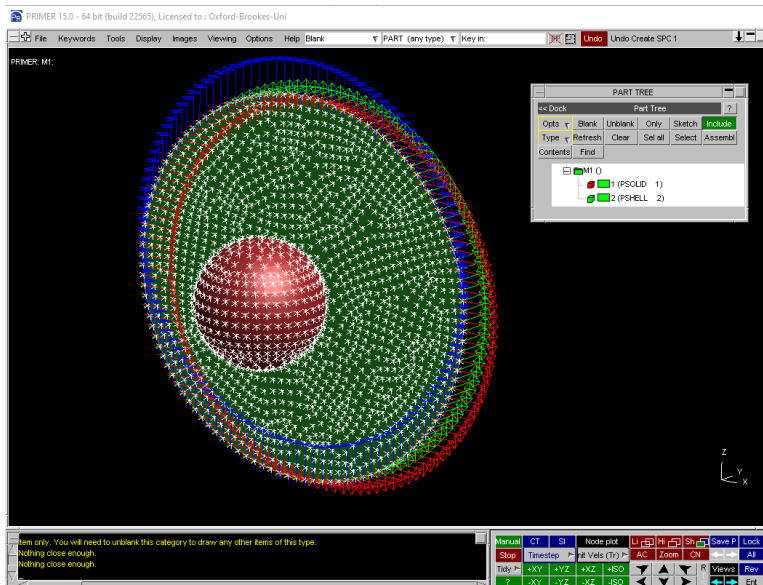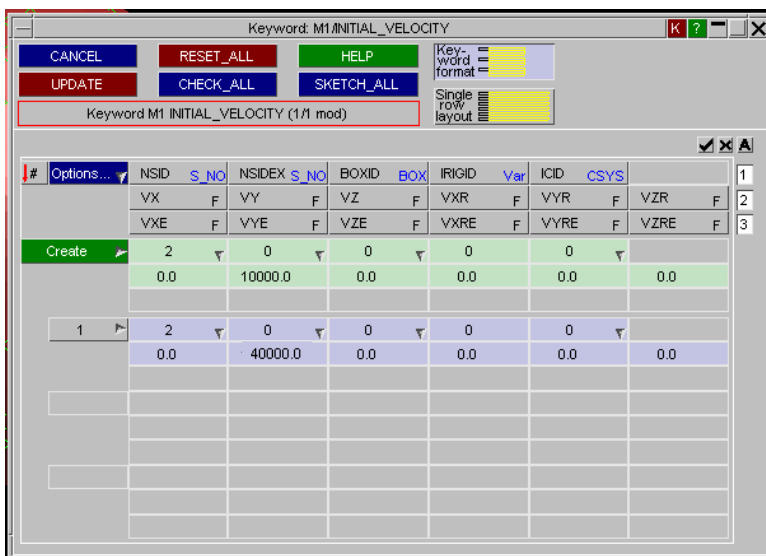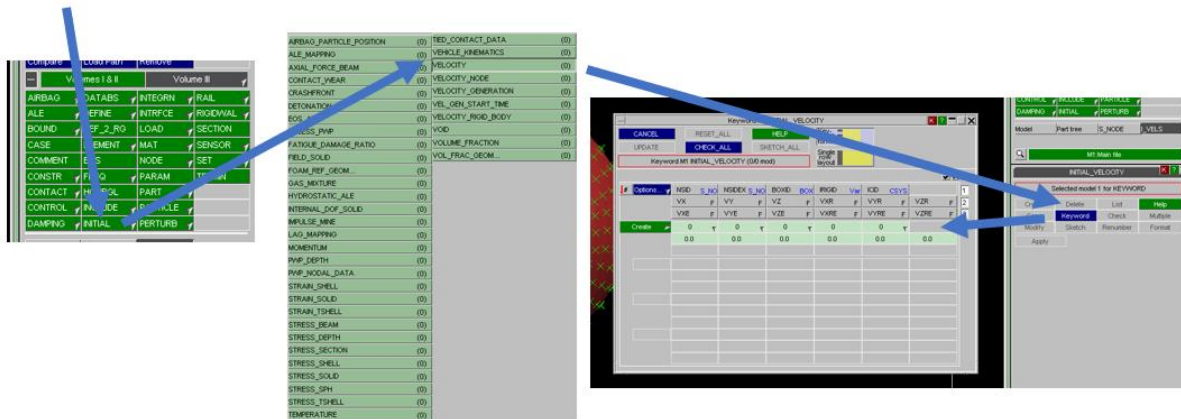
Step 3: Apply initial velocity to ball.

Create a node set for the initial velocities. Call it "initial_velocity" and select all the nodes in the ball. (Click add node then drag a window round the ball.)
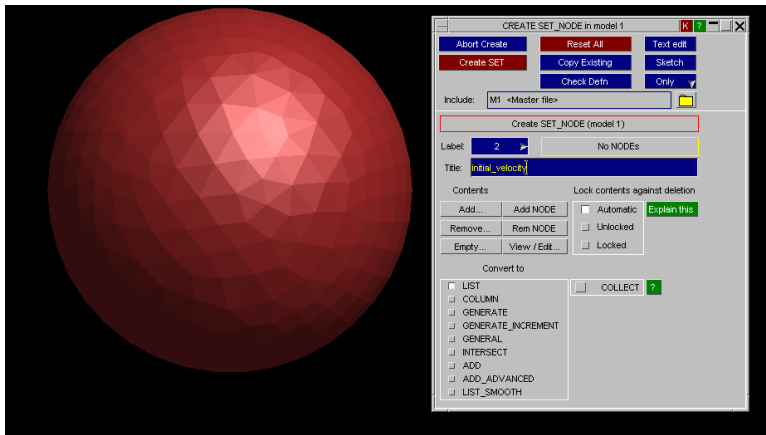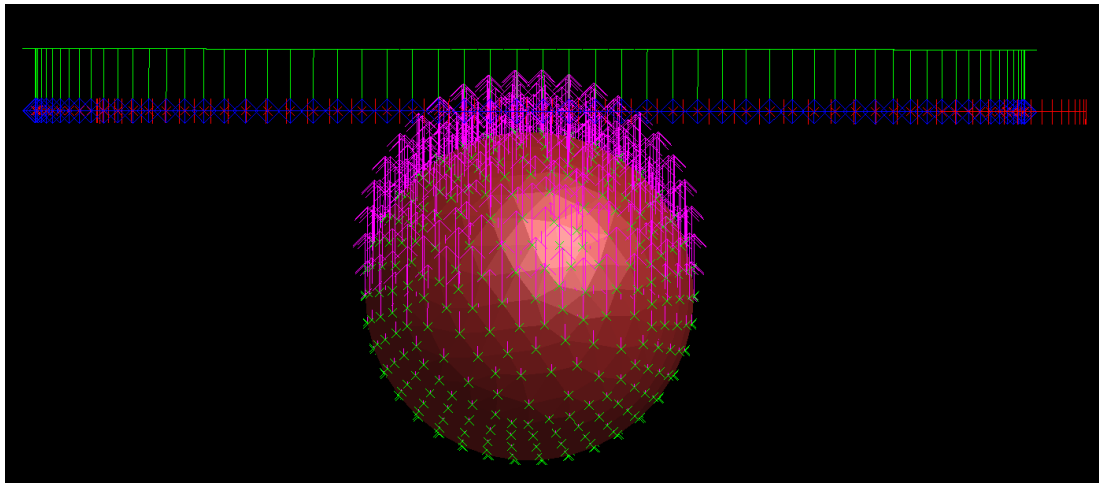

We now define an initial velocity for the set "initial_velocity

Set the initial velocity to be 40e3 mm/s2 in the y direction – if this has worked the screen updates with some arrows in the nodes on the ball.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Step 4: Define the contact

We have to define the nature of the interaction of the ball and plate. Rather than selecting "keyword" this time we select create
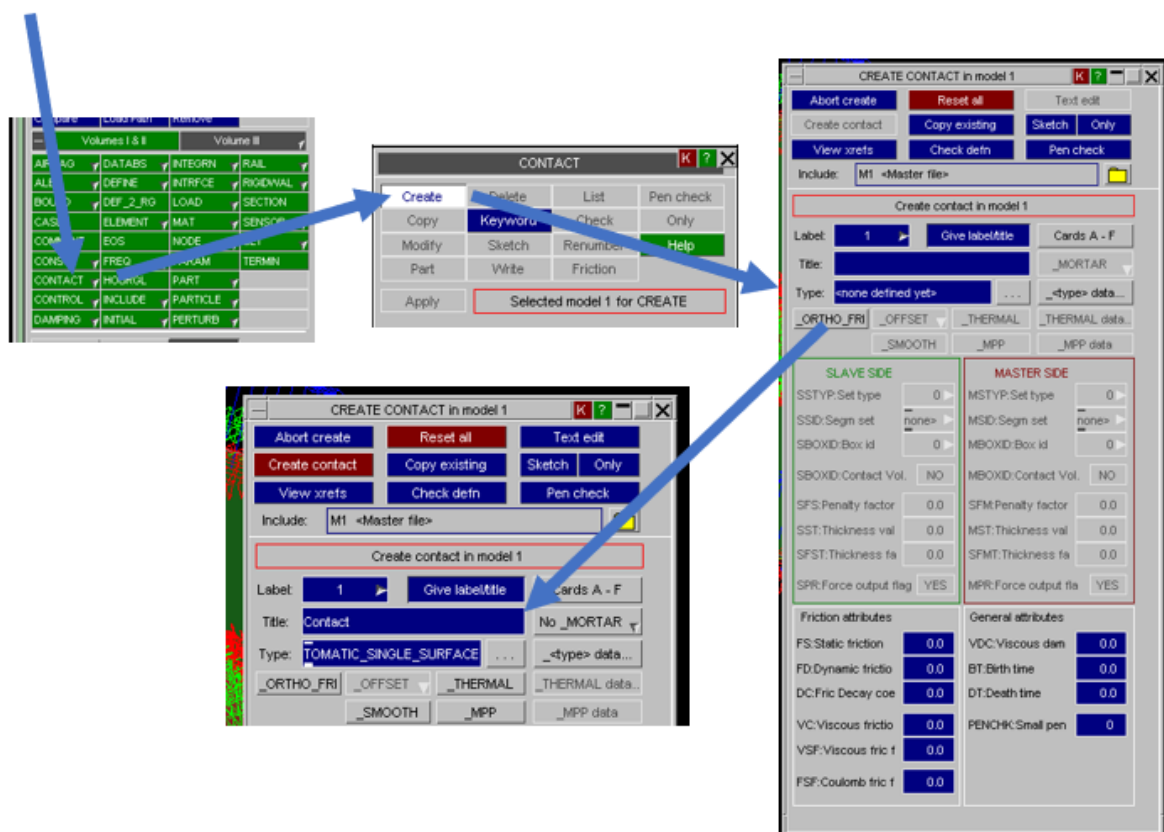
Give the interaction the original name "contact" and select "automatic single surface". There are many contact options – this isn't the place to look at the differences. Hit create contact and the contact is defined.
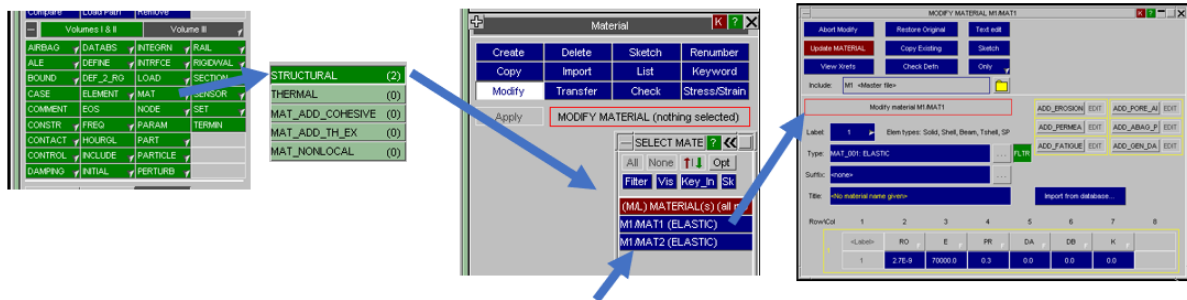
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com   P a g e | **7**
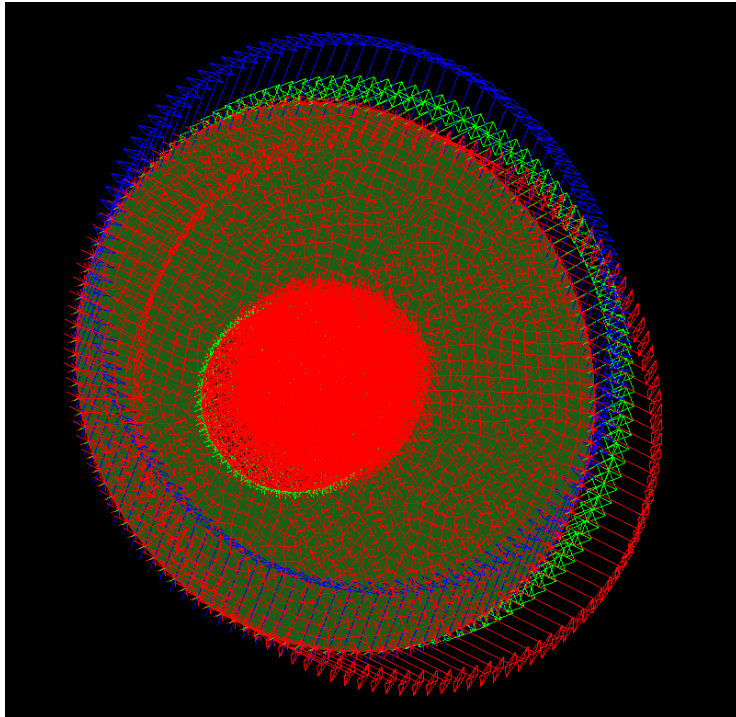
The screen is now an awful mess.

Step 5: Review the material definitions

We have defined material properties in the Abaqus model. These are linear, and in fact the first runs we'll do will use a linear material definition.


If you place the cursor over the M1MAT1 and M1MAT2 buttons parts of the model with each material will highlight

These properties are fine – we'll change them to a simple non-linear model after the first run. At this stage "abort modify".

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com   P a g e | **8**

## Step 6: Define some controls for the Dyna Solver

Obviously there are lots of parameters that the solver uses. In this initial example we are only going to modify a couple of the defaults.
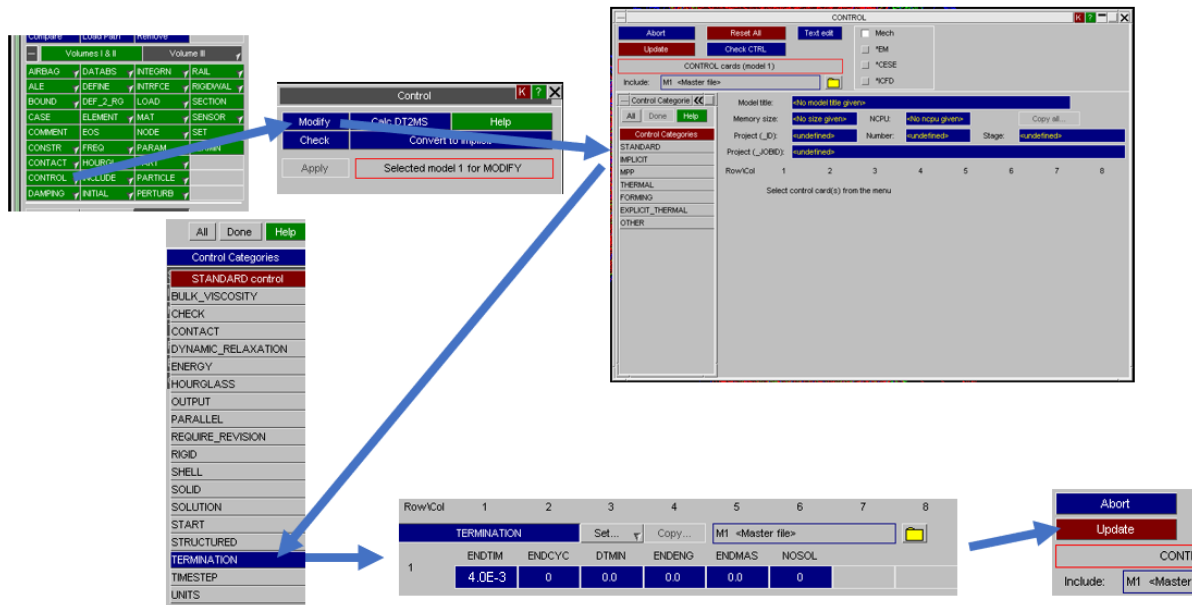
So we select control, we go to modify, then select standard. We set the termination time of the solution to 4e-3s.

Step 7: Write the solver file

The role of Primer is to write a file for the Dyna solver. You'll probably have to close loads of dialog boxes in the bottom left hand side of the screen to get back to this one.
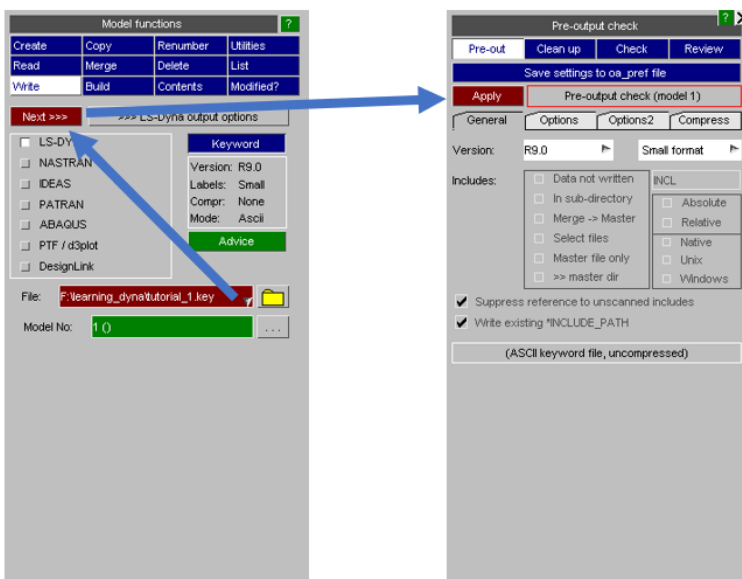
Give the file a name like "tutorial_1.key" Don't close primer.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Step 8: Run the model in LS Dyna

Chose SMP and only chose single precision if you don't care that much about the answer

Select the key file you wrote in section 7

If the problem is small 1 or 2 processors are actually quicker than more

Submit starts the run

When its running Dyna reports progress to a text window. When it has finished "hit any key to continue"

Laurence Marks
l.marks@brookes.ac.uk
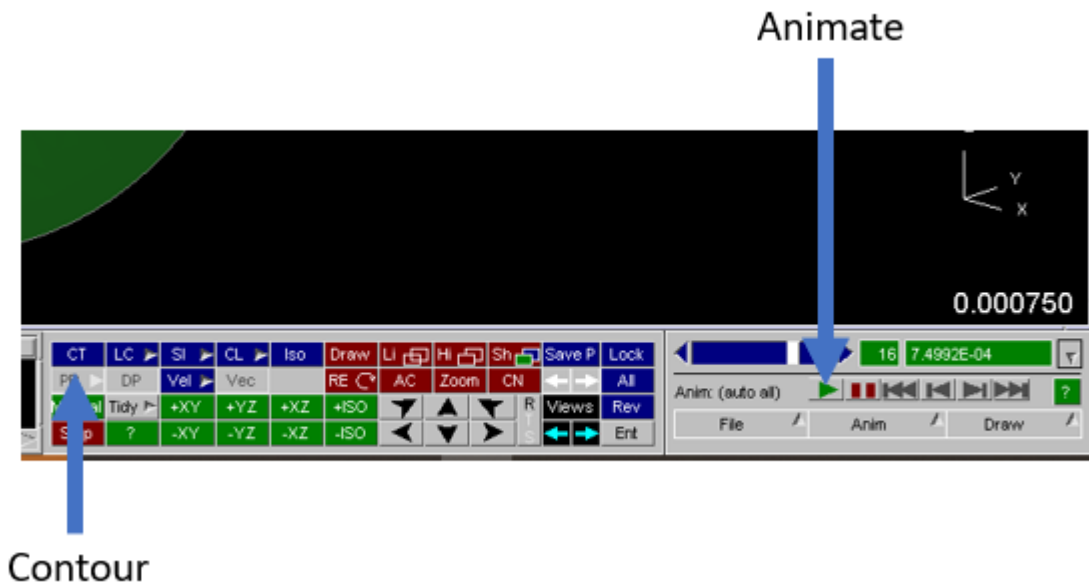laurencemarks64@googlemail.com   P a g e | **10**

Step 9: Post Processing (displaying the results)

If we go back to the LS-Dyna environment we can fire up the D3Plot post processor. If you chose the results file with the same name as the key file with a PTF extension rather than key.
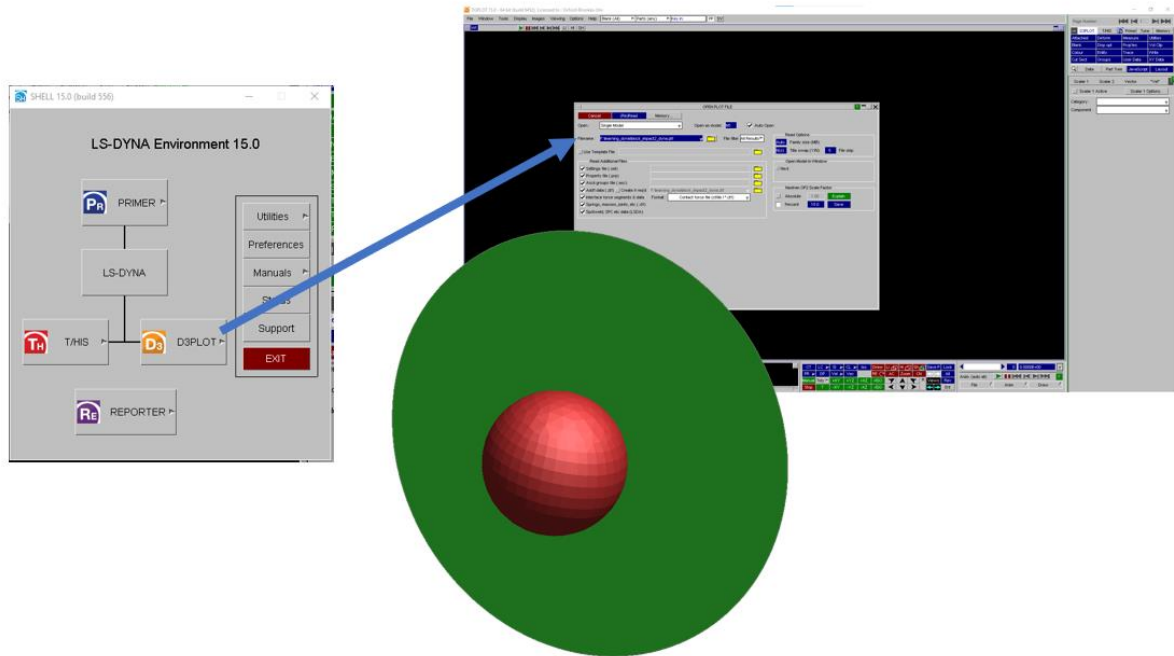

We'll now see the familiar plate and ball. Hit the play button and the ball will bounce off the plate.


To display the stresses push the contour button. What you'll get is essentially nothing as the default plot type is plastic strain and because the properties in this first run are linear you haven't got any.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com   P a g e | **11**
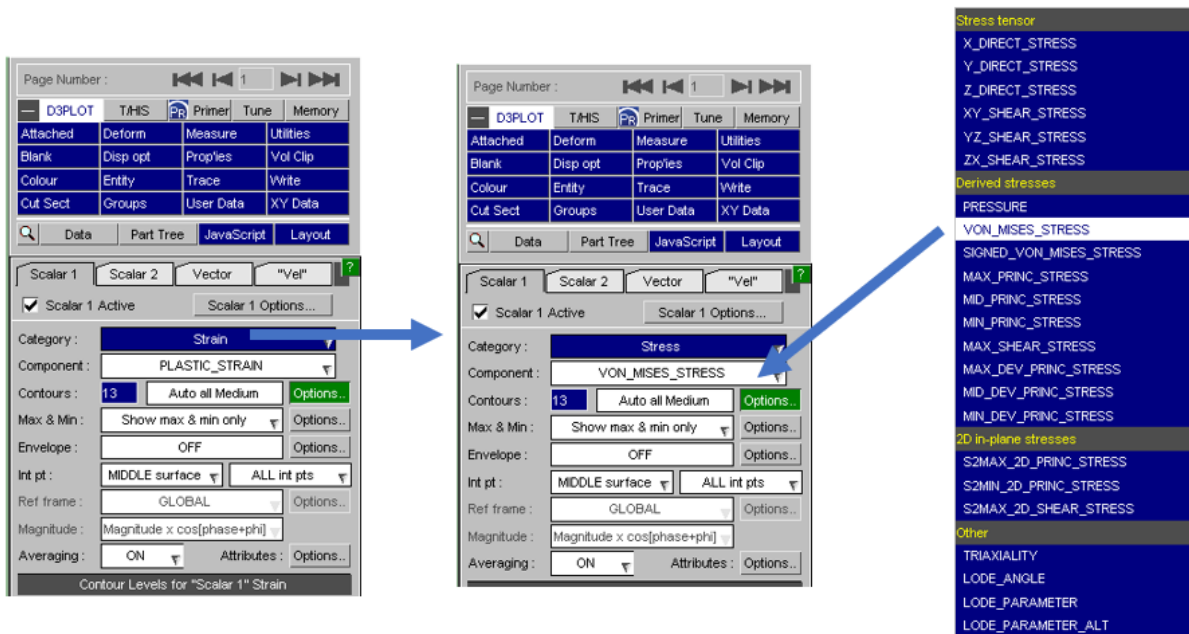
So select "stress" as the category and select "von_mises_stress". There is no such thing as Signed_von_mises_stress.
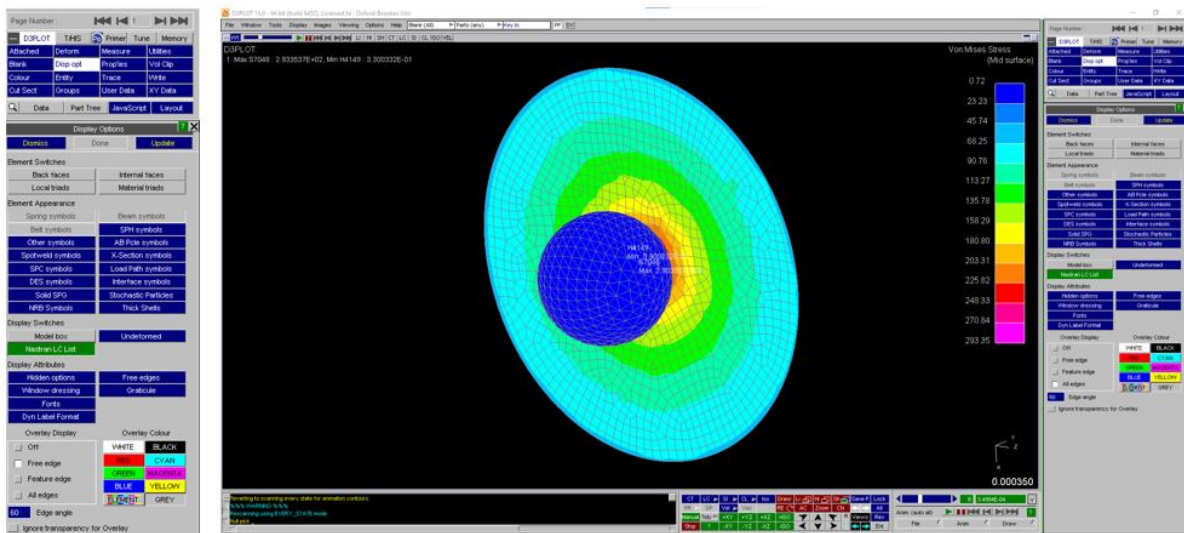

We can plot element boundaries by selecting Disp_opt at the top right hand side of the screen. Choose "all edges" at the bottom of the dialog box. We then get a series of stress plots through the history of the impact.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com   P a g e | **12**

Step 10: Updating the material to allow for yielding

If we go back to Primer we can update the aluminium properties to allow yielding to occur. Select MAT from the menu and select MAT1.
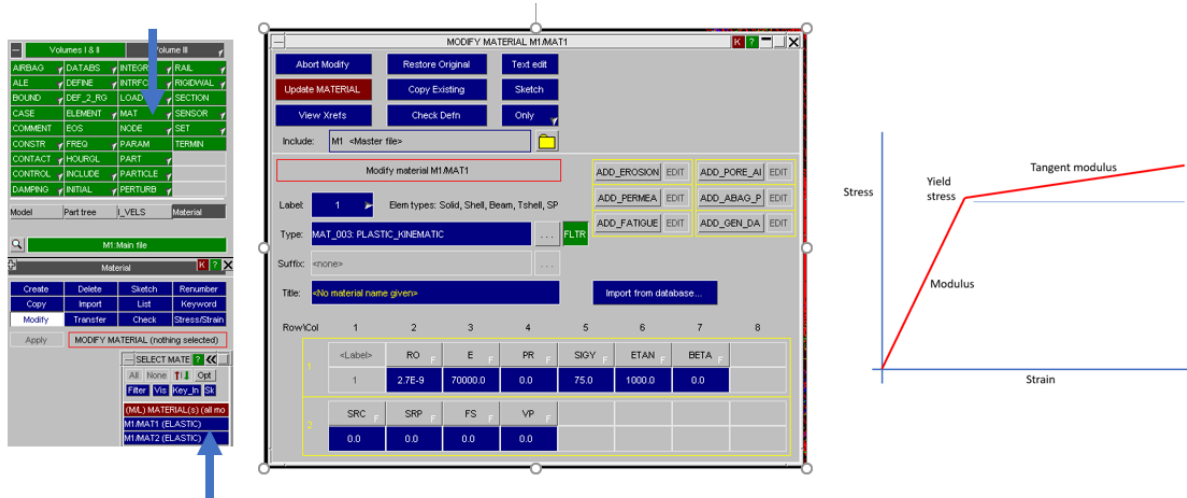
Change the material type to MAT_03_PLASTIC_KINEMATIC and define a yield of 75MPa (SIGY) and a tangent modulus of 1000MPa (ETAN). Update material.

STEP 11: Write a new file and re-run the model

Use a new model file – something like tutorial_1_plastic and re-run it – repeat step 8. You can repeat stage 9 to view the results. The ball should have put a big dent in the ally disc.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com