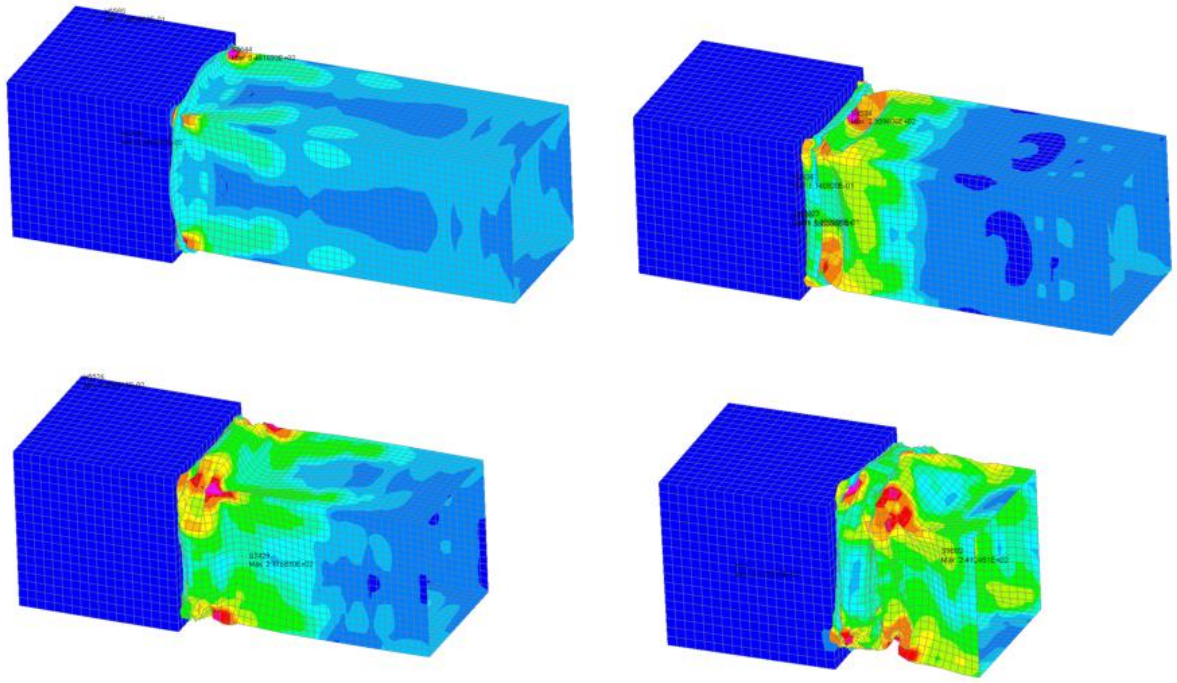
2nd Dyna/Primer Tutorial

Tube Crushing Impact

Laurence Marks

January 2022



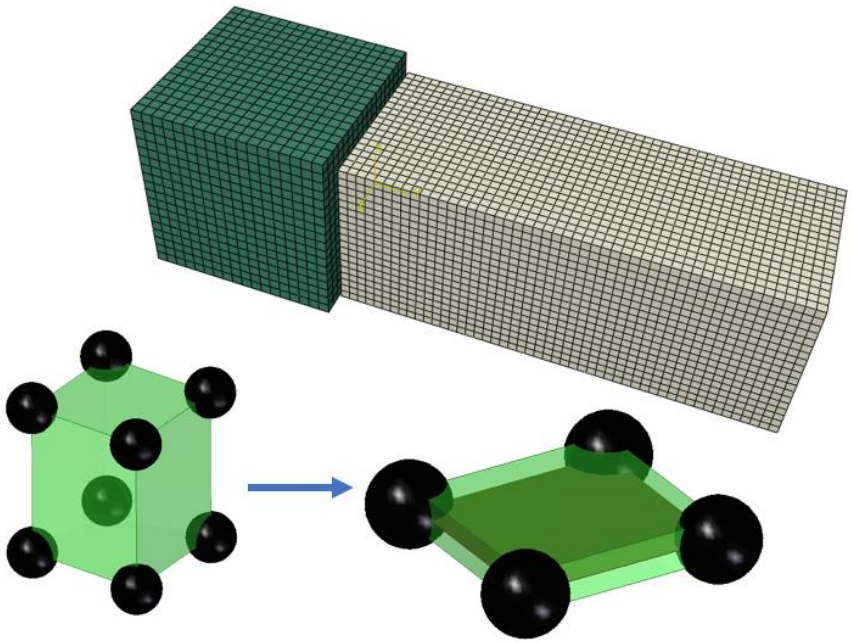
In this second tutorial we use the same steps a in tutorial 1 to create a tube crush scenario. You need to have completed tutorial 1 for tutorial 2 to make any sense.

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

Problem Description

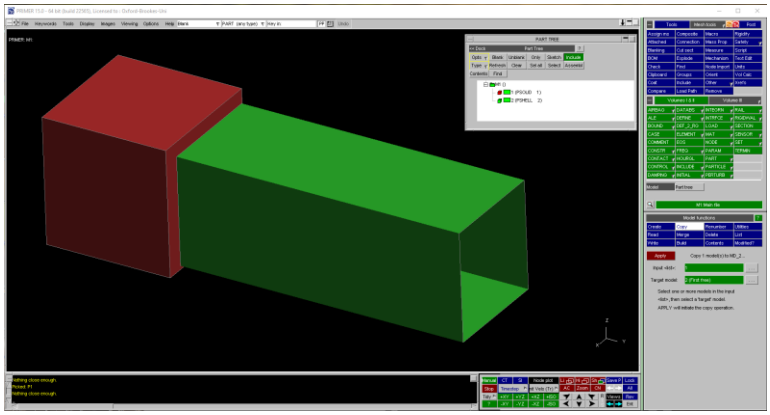
In this problem a steel block hits an aluminium disc at 20m/s. The impact progressively crushes the tube dissipating energy as the tube is plastically deformed.

The block is meshed using lower order hexahedral elements, the aluminium tube with lower order, thin shell elements.



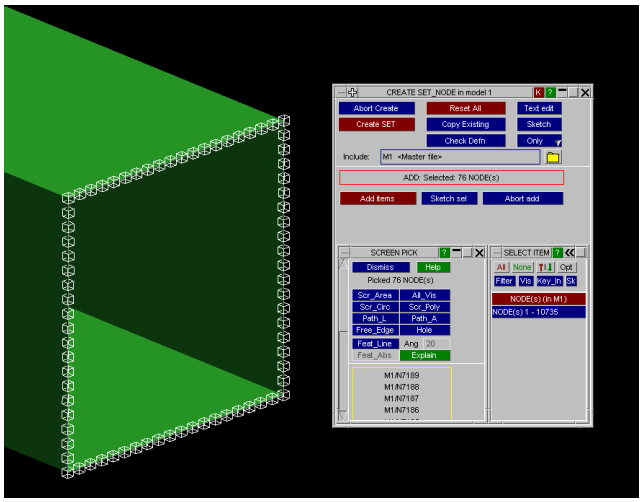
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Step 1: Read meshed model from Abaqus via Nastran BDF file

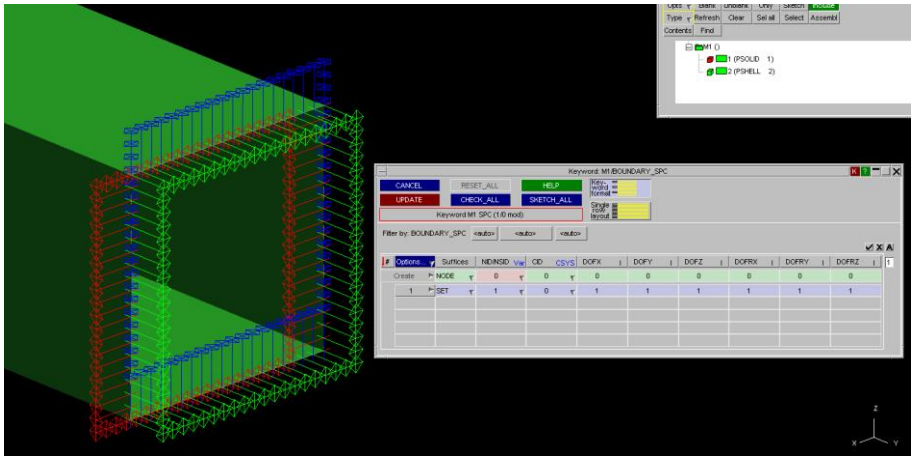


Step 2: Fix periphery of target

We are going to fix the end of the tube away from the impacting block. As in tutorial 1 we are going to create a set and then apply SPC's to that set.



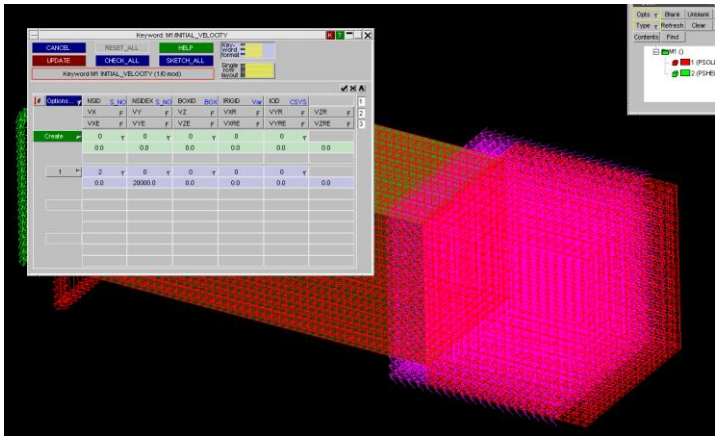
Create a set of nodes called fixed_end_of_box – as shown above and restrain all degrees of freedom as shown below.



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

Step 3: Apply initial velocity to ball.

Now 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.)

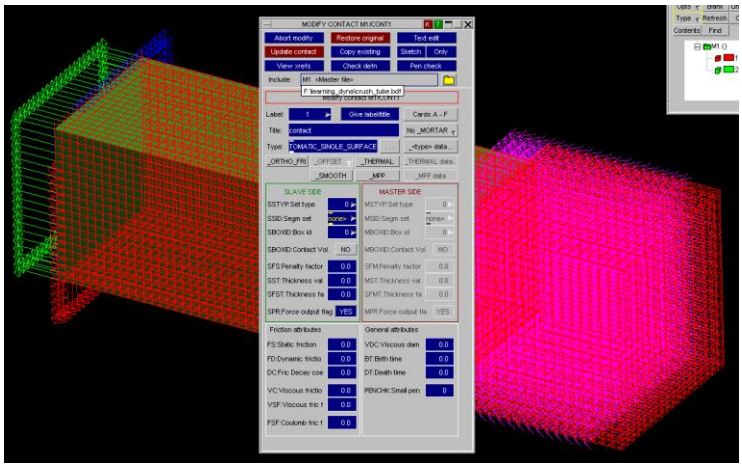


We now define an initial velocity for the set “initial_velocity” of 20e3mm/s.

Step 4: Define the contact

We have to define the nature of the interaction of the block and tube. This contact considers self contact which is critical to the deformation modelled.

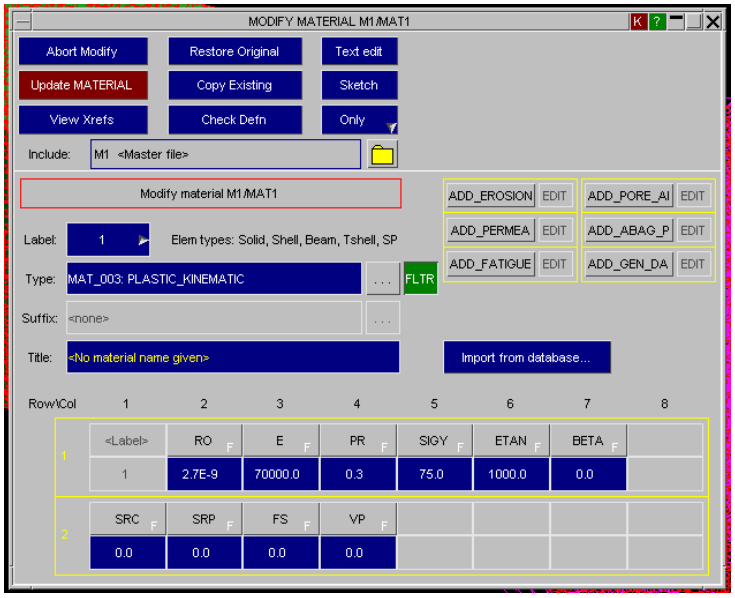
As in tutorial 1 we give the interaction the original name “contact” and select “automatic single surface”. There are many contact options – this isn’t the place to look at the differences. Hit create contact and the contact is defined.



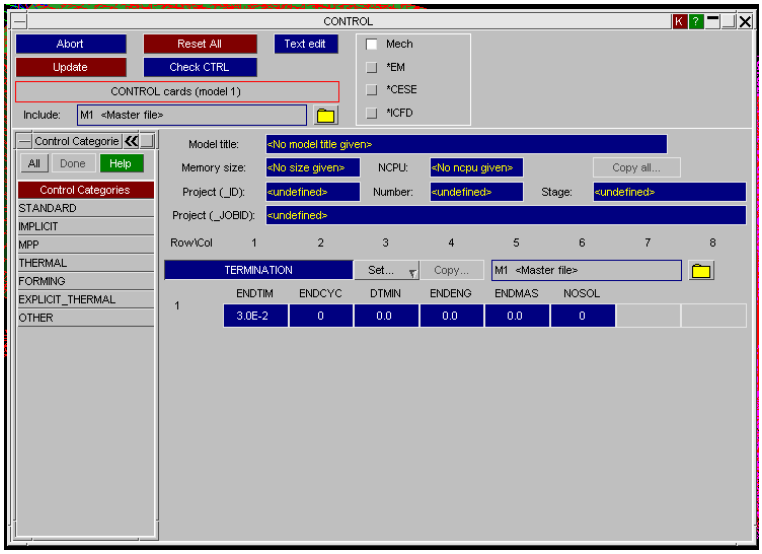
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Step 5: Review the material definitions

This model doesn't really make much sense with linear properties so we update the material properties of the aluminium section to consider yield and post yield behaviour. This is the same material model as in tutorial 1.



Step 6: Define some controls for the Dyna Solver



So we select control, we go to modify, then select standard. We set the termination time of the solution to 3e-3s.

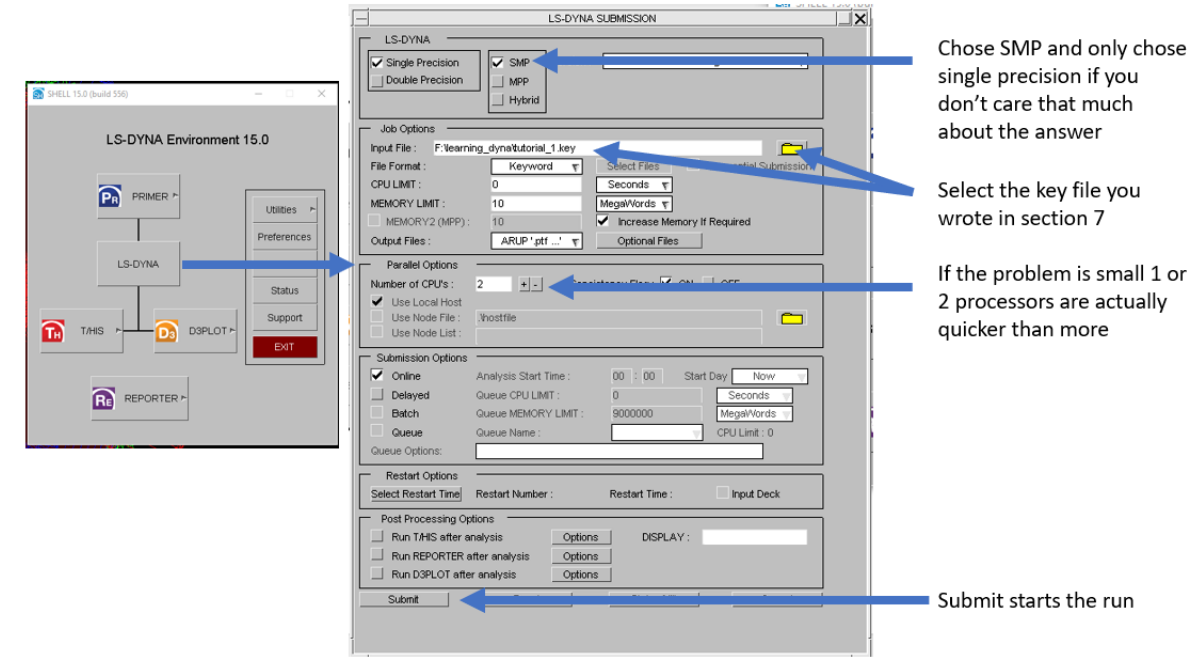
Step 7: Write the solver file

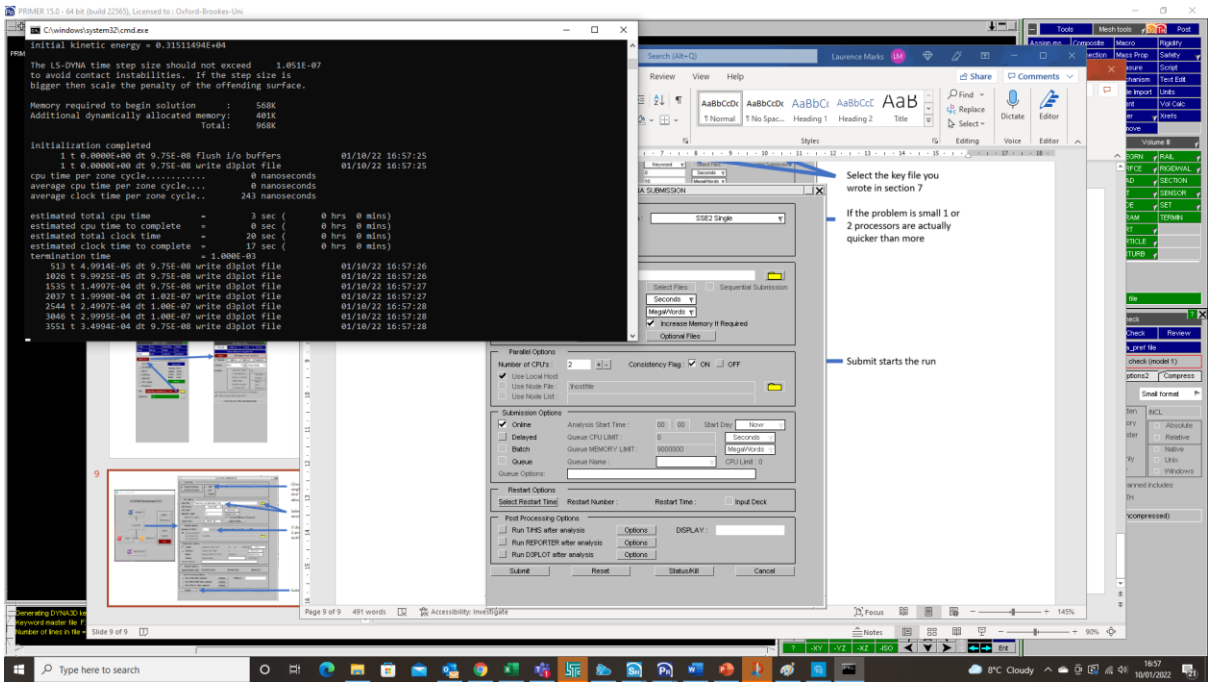
Write the solver file as you did in tutorial 1.

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

Step 8: Run the model in LS Dyna

The same as tutorial 1 again.





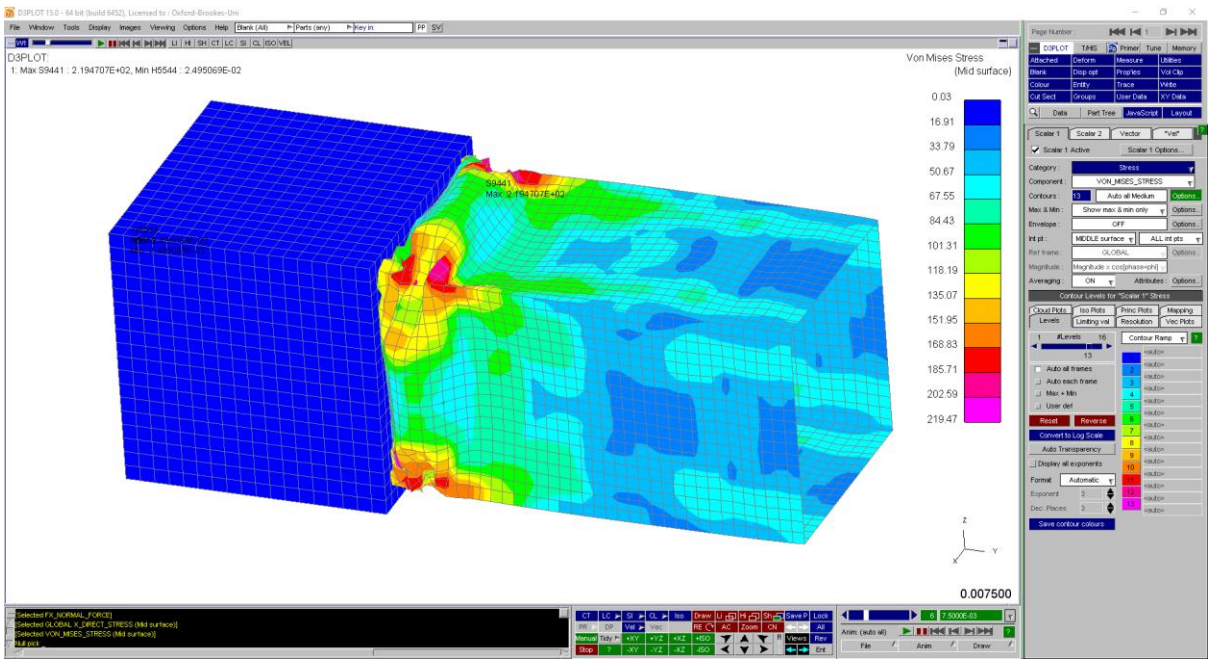
When its running Dyna reports progress to a text window. When it has finished "hit any key to continue"

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

Step 9: Post Processing (displaying the results)

When we have read the results file into the post processor we can change the screen settings before we proceed.

We can set the background colour to white, change the text to black and plot all element edges.



We change the results set we plot to Von_mises_stress and move the results slider bar. We can also animate the results.

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