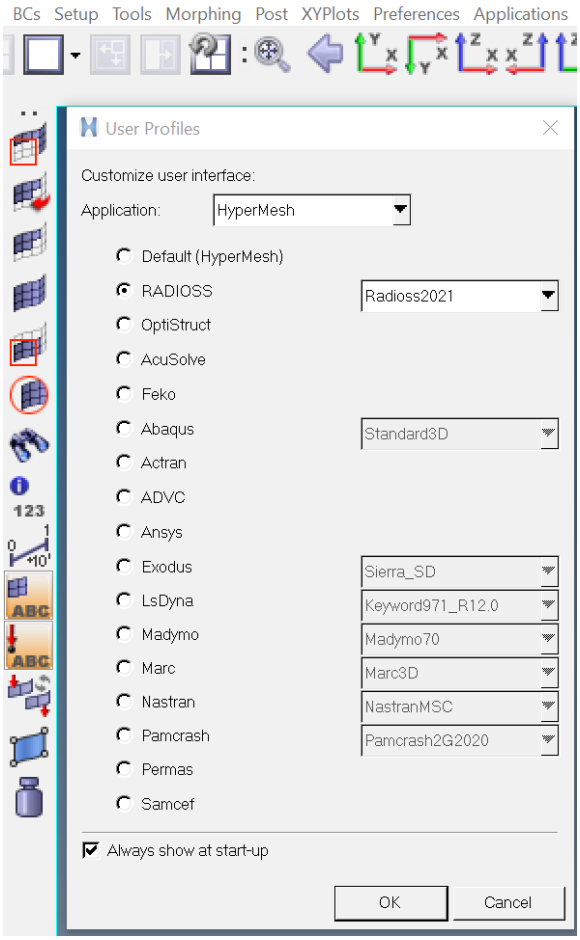
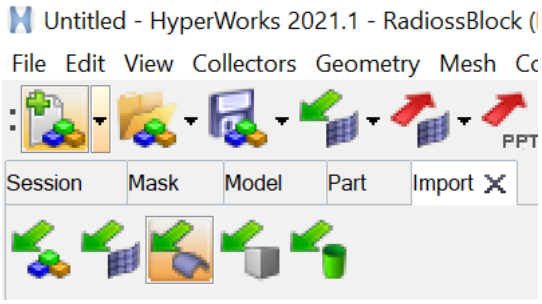
SD-12 Exercise – Drop Test Analysis HyperMesh (Radioss)

Step 1: OPENING THE MODEL

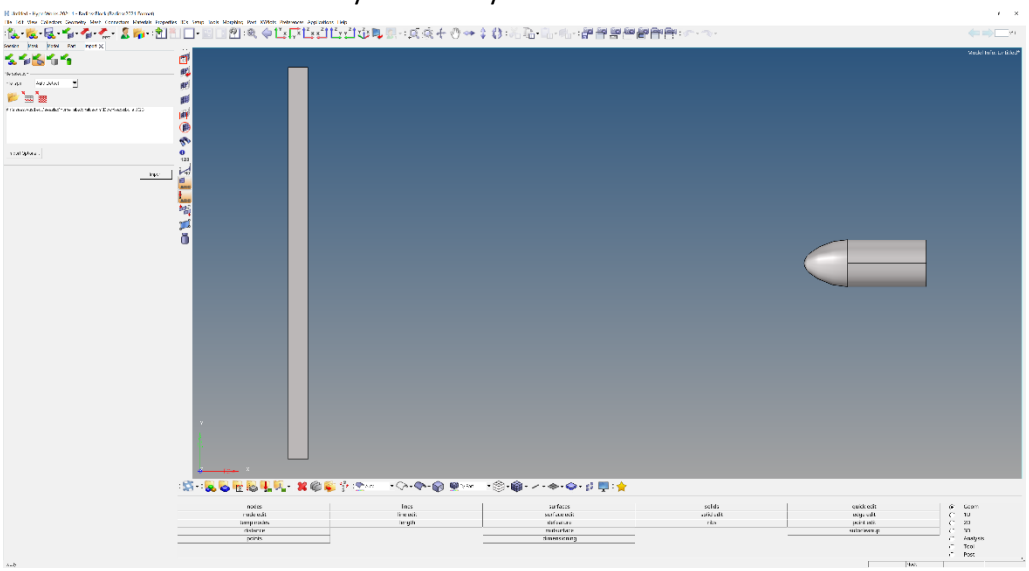
1. When opening HyperMesh you will see a user profile. Make sure that it is on Radioss and keep all other settings the same.



2. Next got to file -> import -> geometry and click the vanilla folder under file type. Browse and select **bullet. Igs** from the downloaded location.

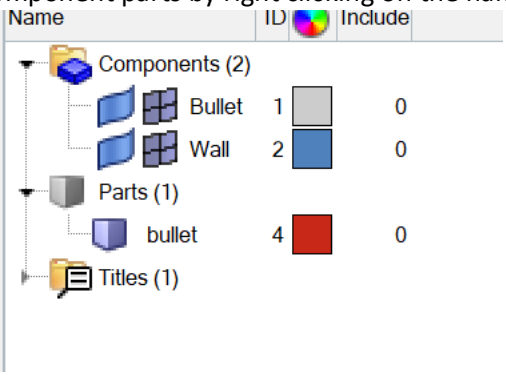


The picture below should be what you see on your screen now.



Remark: In this window you can change many features, some main features are:

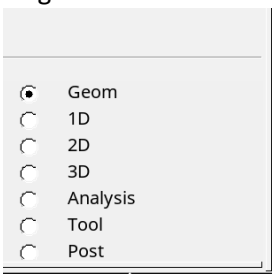
- Change the name of the component parts by right clicking on the name and then renaming.

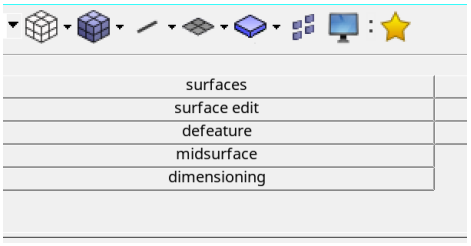


- You can also switch between wire and shaded view of the part from the  icon
- Rotate the part by holding the control button and the left button on your mouse or zoom in and out with the roller on the mouse.
- Other features are shown below:

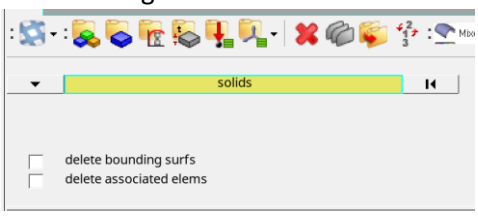
Step 2: SETTING UP A MESH

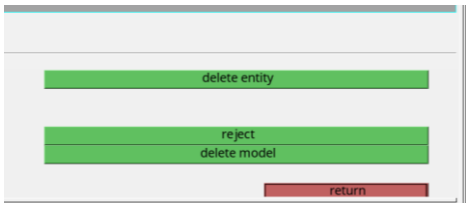
1. First a midsurface needs to be created. In the panel at the bottom make a **2d midplane** on the wall by going to **mid-surface**. Once in the midsurface panel first click the **yellow block** and then drag over the entire wall to select it. Click enter.



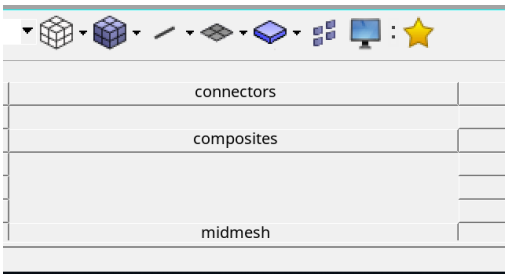


2. Next Click the **f2 key** to be able to delete the wall. Click the **yellow box** and then hold shift and make drag the curser over the wall to make a box. **Click delete entity**.

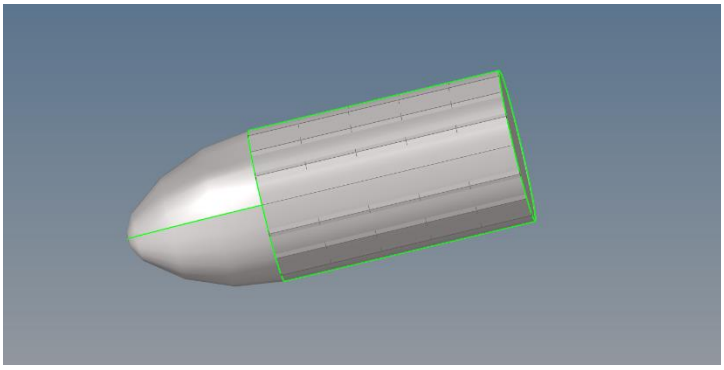


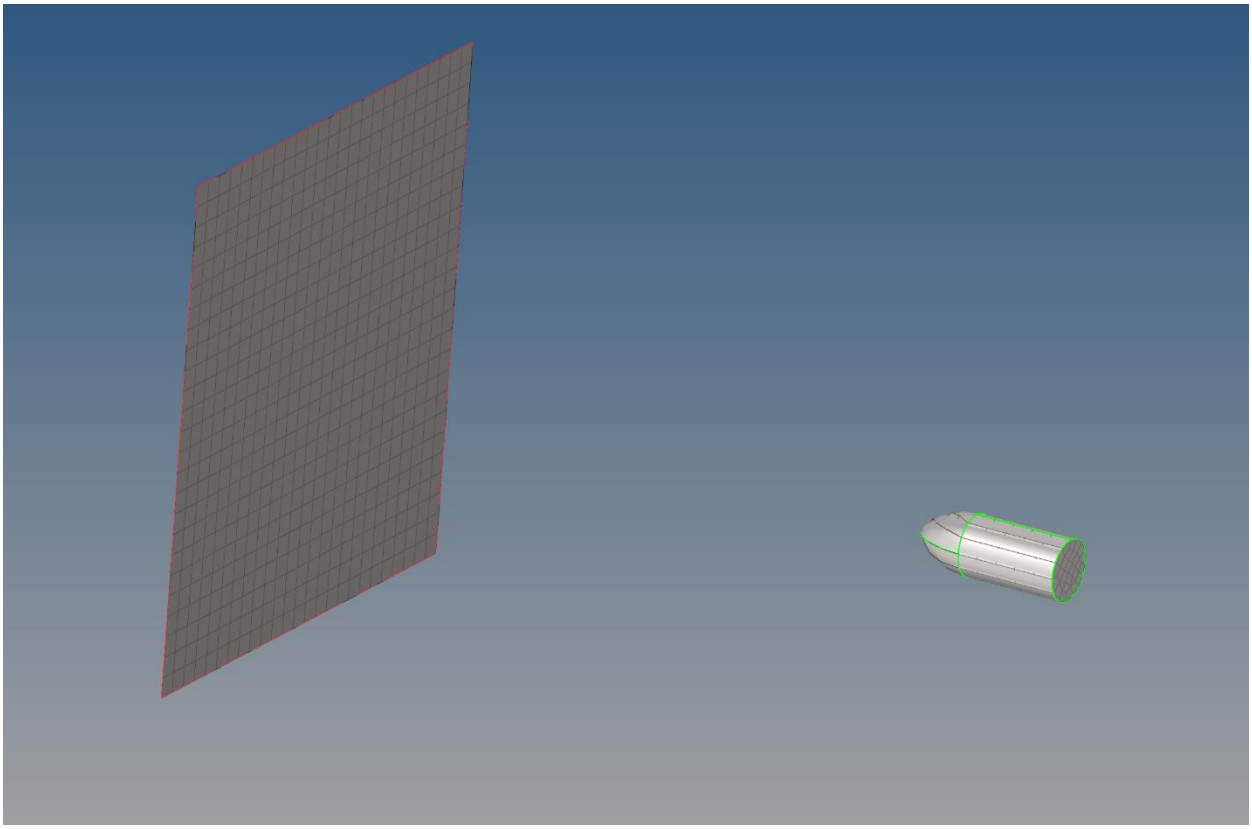


3. In the panel at the bottom go to 2D -> midmesh -> set an **element size** to **2** and **mesh type** as **mixed**. Then click the cylindrical part of the bullet. This will create a mesh on the selected surface. To make a more even mesh, click at where the numbers are to add more nodes until the lines are parallel.

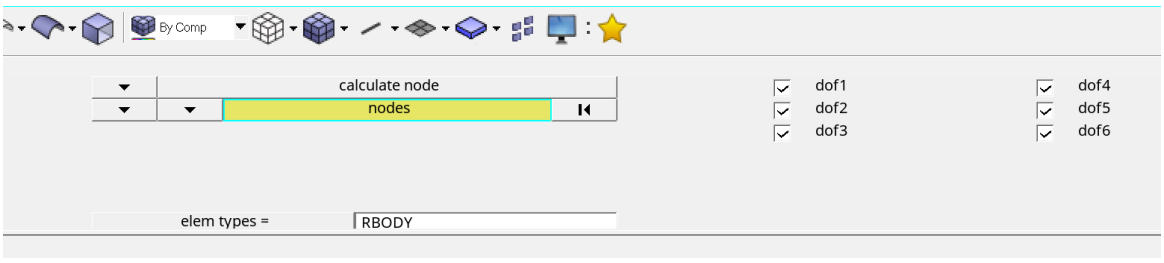


4. Repeat for all other surfaces on the bullet and the 2d wall. Your model should now look like below.

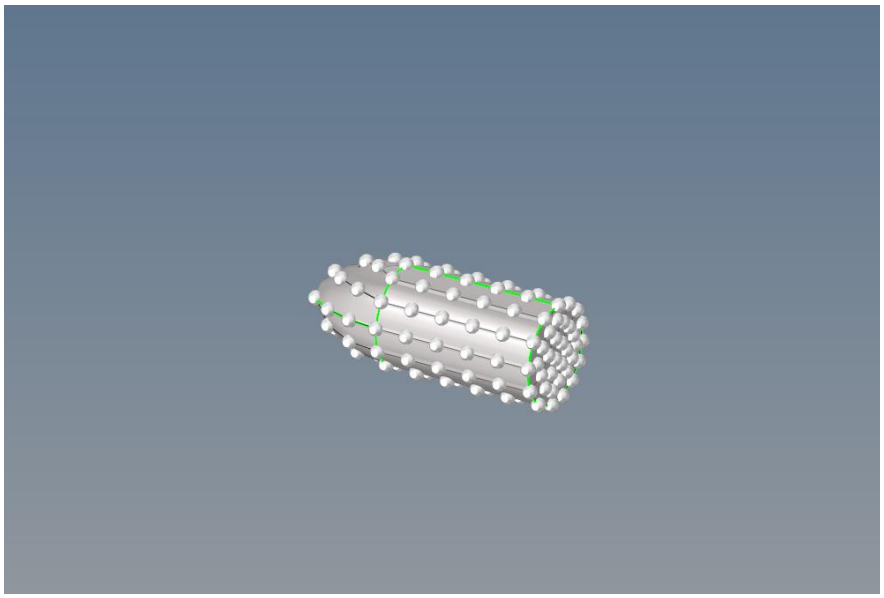




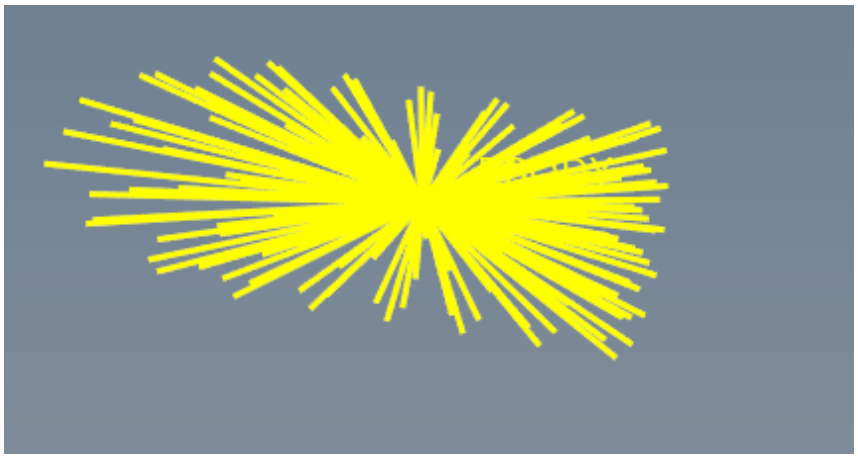
5. Next, in the model window on the right, right click -> create -> component and rename it rigids. Then go to the **bottom panel** and select **1D**. There click **rigids** and make sure to have the setting as seen below:



This will create nodes that are fixed compared to one another and will look like the picture below:

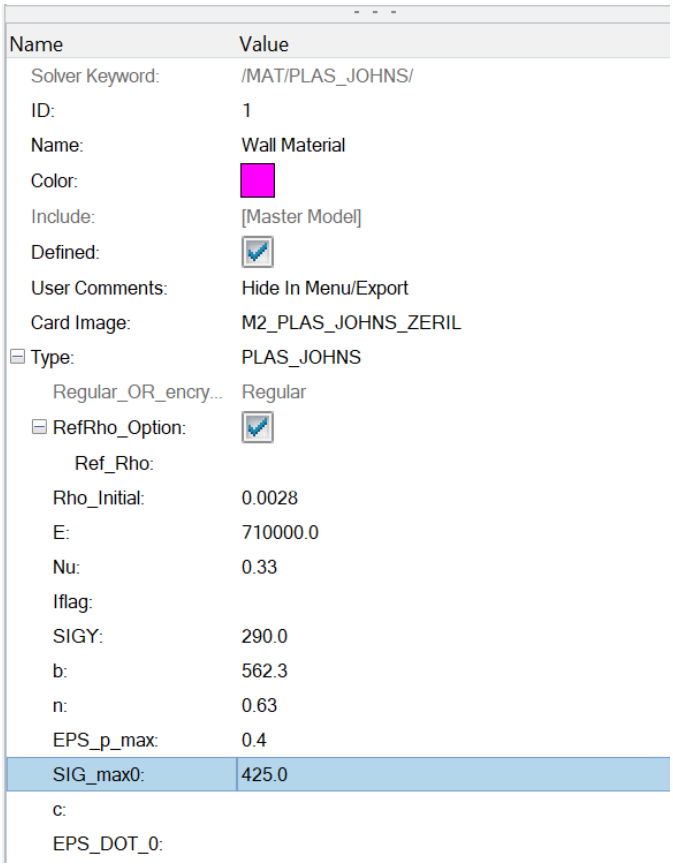


Remark: To check if it worked properly you can look to see what it did by suppressing the bullet component. In the model window. Notice how all the yellow lines converge at the center. That center node is what is called the master node in Radioss and controls the other nodes (Basically of the node moves the others will as well).



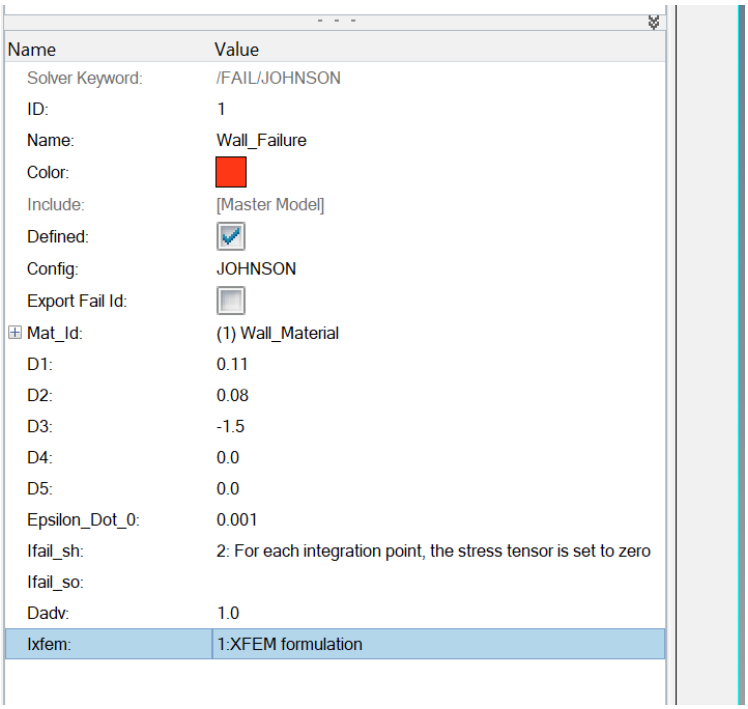
Step 3: MAKING MATERIAL AND PROPERTIE ID'S

1. Right click in the model folder -> create-> material, copy the values in the figure below:

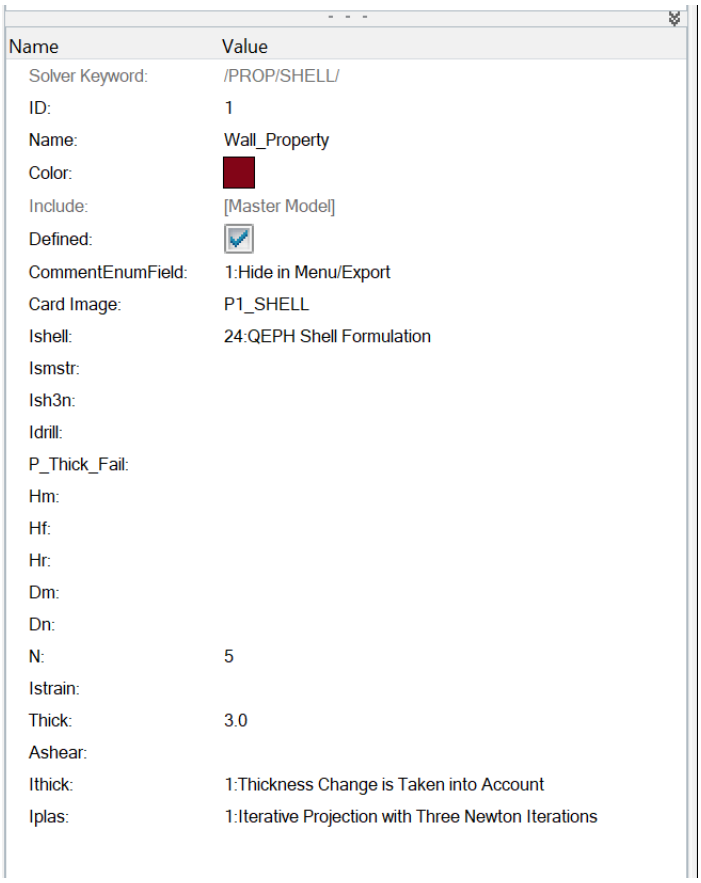


2. Duplicate the wall by right clicking the wall material just made in the model window and clicking duplicate. Rename it Bullet_material and make **eps_p_max** and **sig_max0** have values of zero to make the bullet is a rigid body.

3. Create a new **failure Criteria** for wall. Do this by, right clicking in the model -> create -> failure and rename the failure folder just created to wall failure. Replace values as seen below:

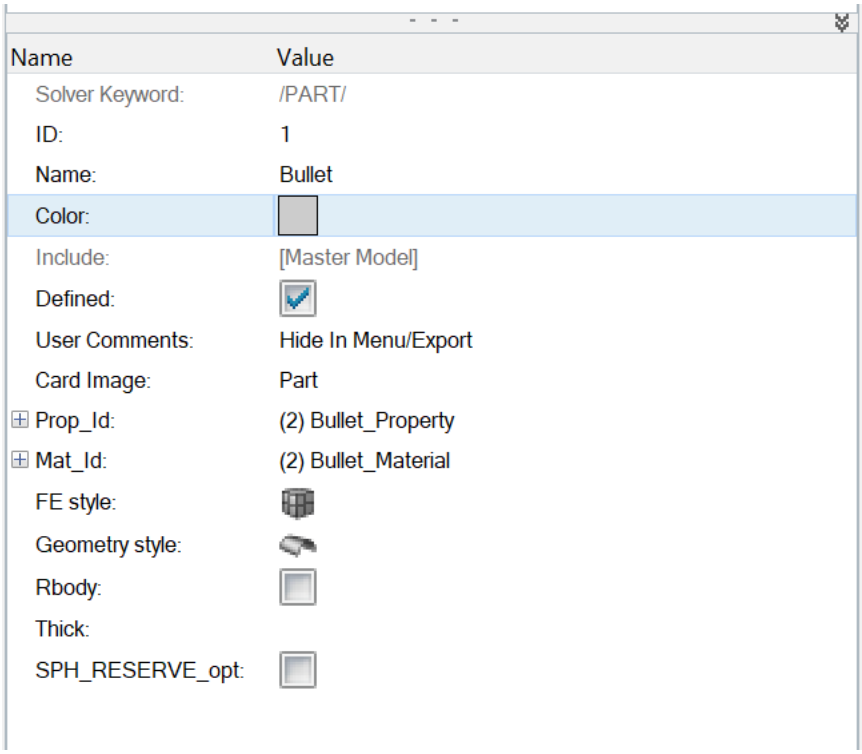


4. After creating the materials, **property id's** are also needed. To do this right click -> create -> property and fill in the values below:



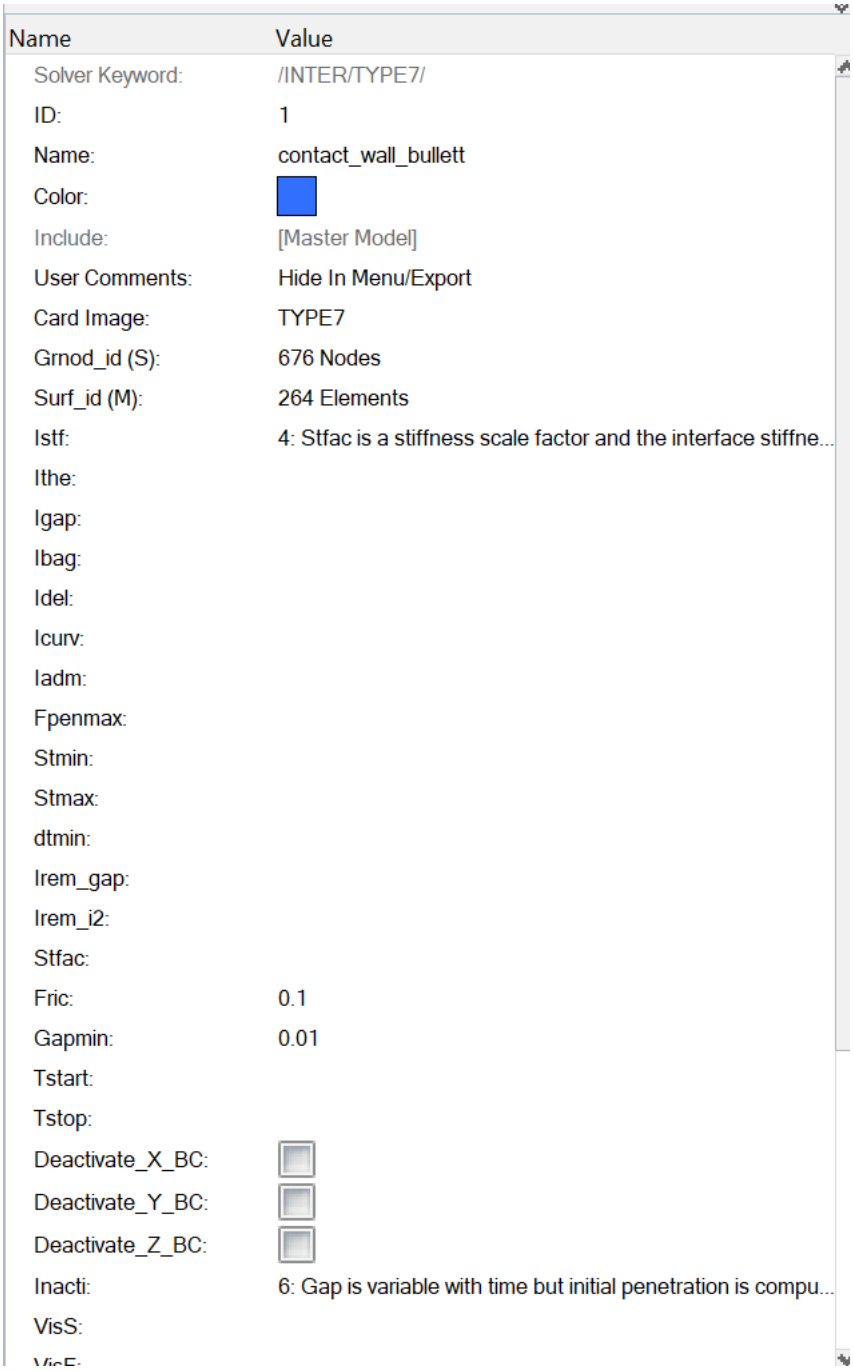
5. Repeat step 12 for the bullet.

6. **Assign** the **property id** and **material** to the bullet and the wall by selecting one of the components, clicking on the **prop_id** and **Mat_id** and selecting the appropriate one for each.



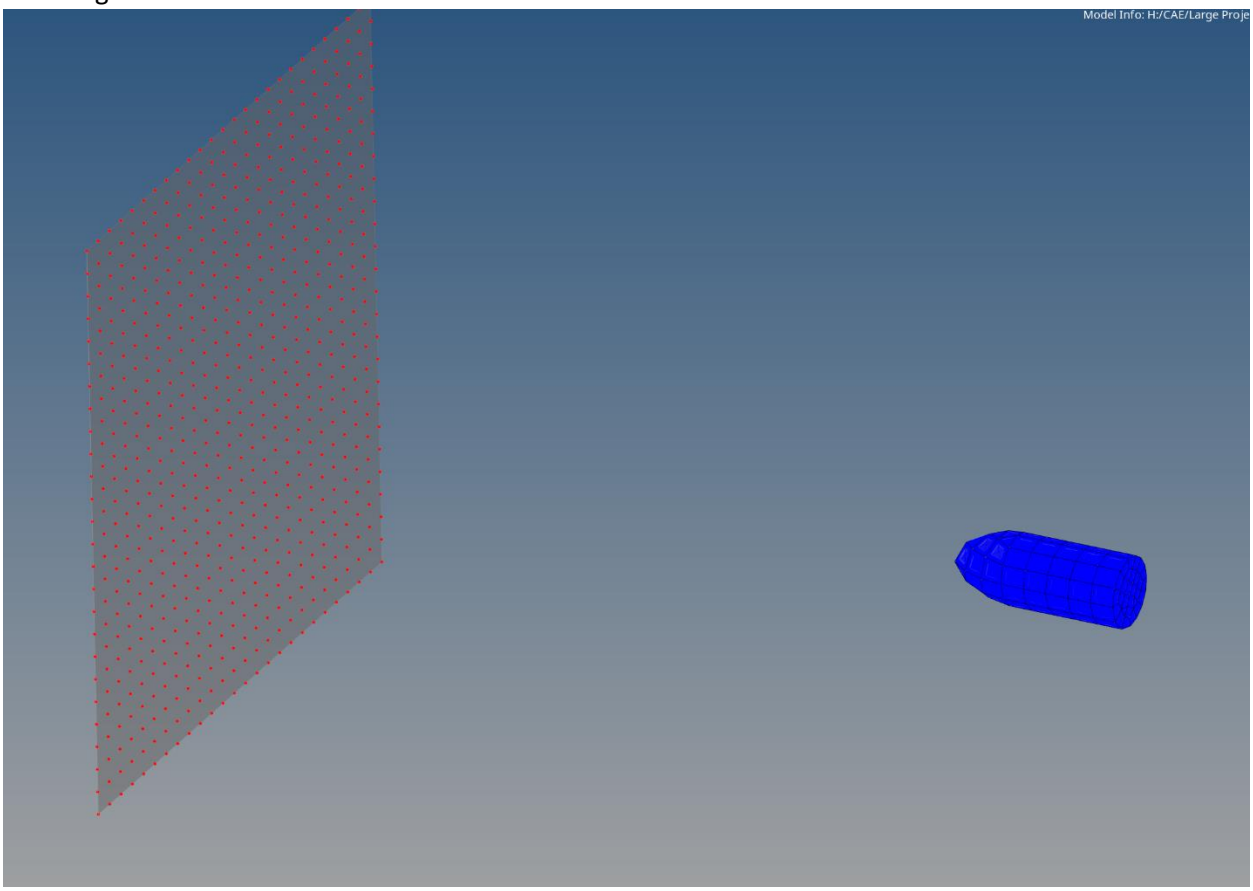
Step 4: CREATING CONTACTS

1. Start by creating a type 7 contact between the wall and the bullet. Do this by right clicking the model window and going to contact. Once created, this will create a group folder, in the group folder rename the subgroup to contact. Change the settings as seen in the figure. When selecting the Grnod_id, choose the wall by clicking shift and dragging a box around the wall. Then for the Surf_id, do the same for the bullet.



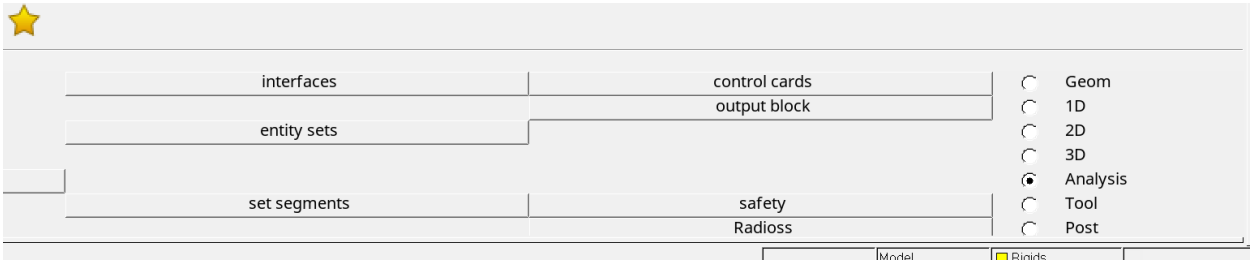
Note: Further down you also need to set Iform to 2.

Remark: You can check to see if it is right by right clicking on the contact you just made and selecting review. The wall should be in red and the bullet in blue.

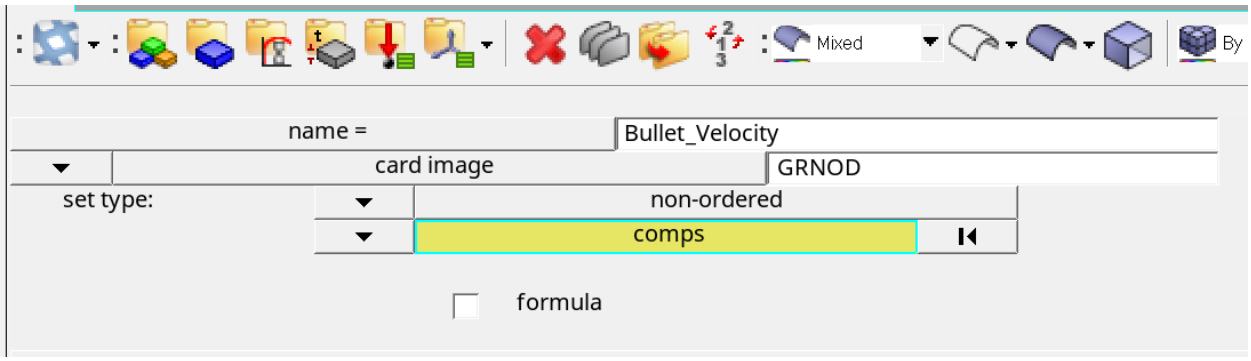


Step 5: ASSIGNING BOUNDARY CONDITIONS AND OUTPUT BLOCK

1. Start by clicking analysis in the panel at the bottom, then click entity sets.



2. In entity set name the first set **Bullet Velocity**, set card image to **GRNOD**, set type to **non-order** and have the **yellow box as comps**. Once this is done click the yellow box and select the bullet component to use. Then click **create**.



3. Now make another entity set labeled wall nodes with the only change is that the **yellow box** should say **nodes** now. Then click on nodes in the **yellow box** go to **by geo** and select the right and left edge of the wall. Click **create**.

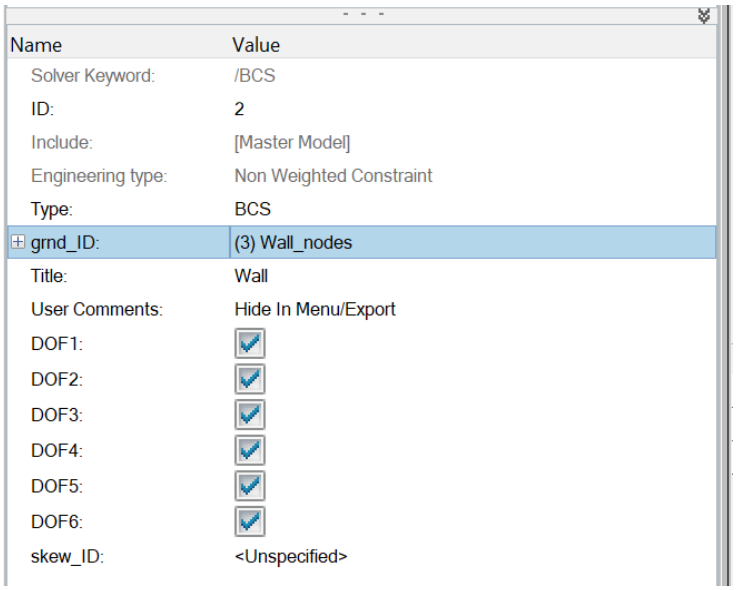
Note: You should now see three sets in your model window.

4. The next step is to open solver, if it is not already a tab next to model, go to view-> browser->hypermesh->solver.

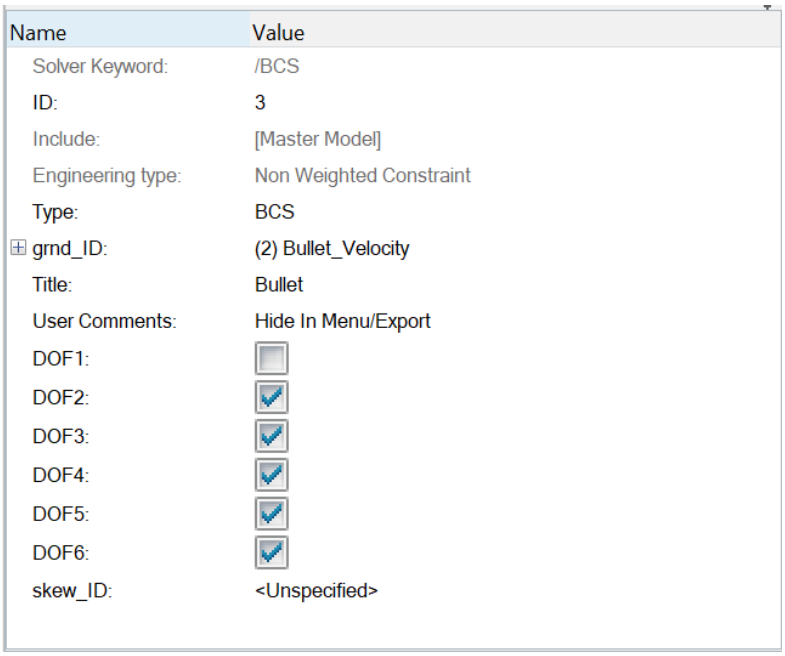
5. The in the solver window click create->Boundary Conditions->**INIVEL**. Rename the INIVEL to Velocity, select the Bullet_Velocity set for **grnd_ID** and change the velocity component to (-555555,0,0). This will give the bullet an initial velocity in mm/s.

Name	Value
Solver Keyword:	/INIVEL/TRA
ID:	1
Include:	[Master Model]
Engineering type:	Directional Velocity
Type:	INIVEL
⊞ grnd_ID:	(2) Bullet_Velocity
Title:	Velocity
User Comments:	Hide In Menu/Export
type:	TRA: Translational Material Velocity
Velocity Components ...	-555555, 0, 0
skew_ID:	<Unspecified>

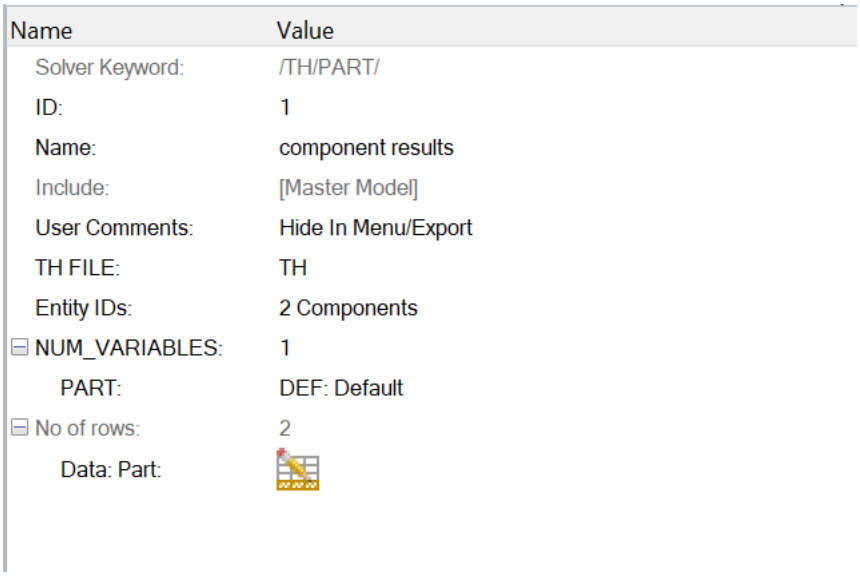
6. Now for the wall, right click -> click create->Boundary conditions->BCS. Rename as wall and then limit all the degrees of freedom by checking **DOF1-DOF6**. Specify the **grnd_ID** as wall_nodes.



7. Create another BCS for the bullet, check every DOF besides **DOF1**, which represents the x axis.



8. Now it's time to create an output block to get results. Do this by right clicking in the model window and selecting output block. Rename the output block as component results. For entities IDs select both components (the bullet and wall). Set **num_variables** to 1 and **Part** as DEF: default.



9. Make another output block for contacts and select contact_wall_bullet_ for the **entities IDs**. Keep everything else the same as the components results.

10. Save your file in a location of your choice.

11. Under analysis in the bottom panel click **Control cards**. Locate and enter the values for each card below:

- ENG_ANIM_DT -> click on it -> set Tfreq value to 5e-5
- ENG_ANIM_ELEM -> check EPSP, ENERGY,VONM and HOURG
- ENG_MON -> check MON_ON_OFF
- ENG_RUN -> set Tstop to 0.005
- ENG_TFILE -> set Time_frequency to 5e-5

12. Save the model again.

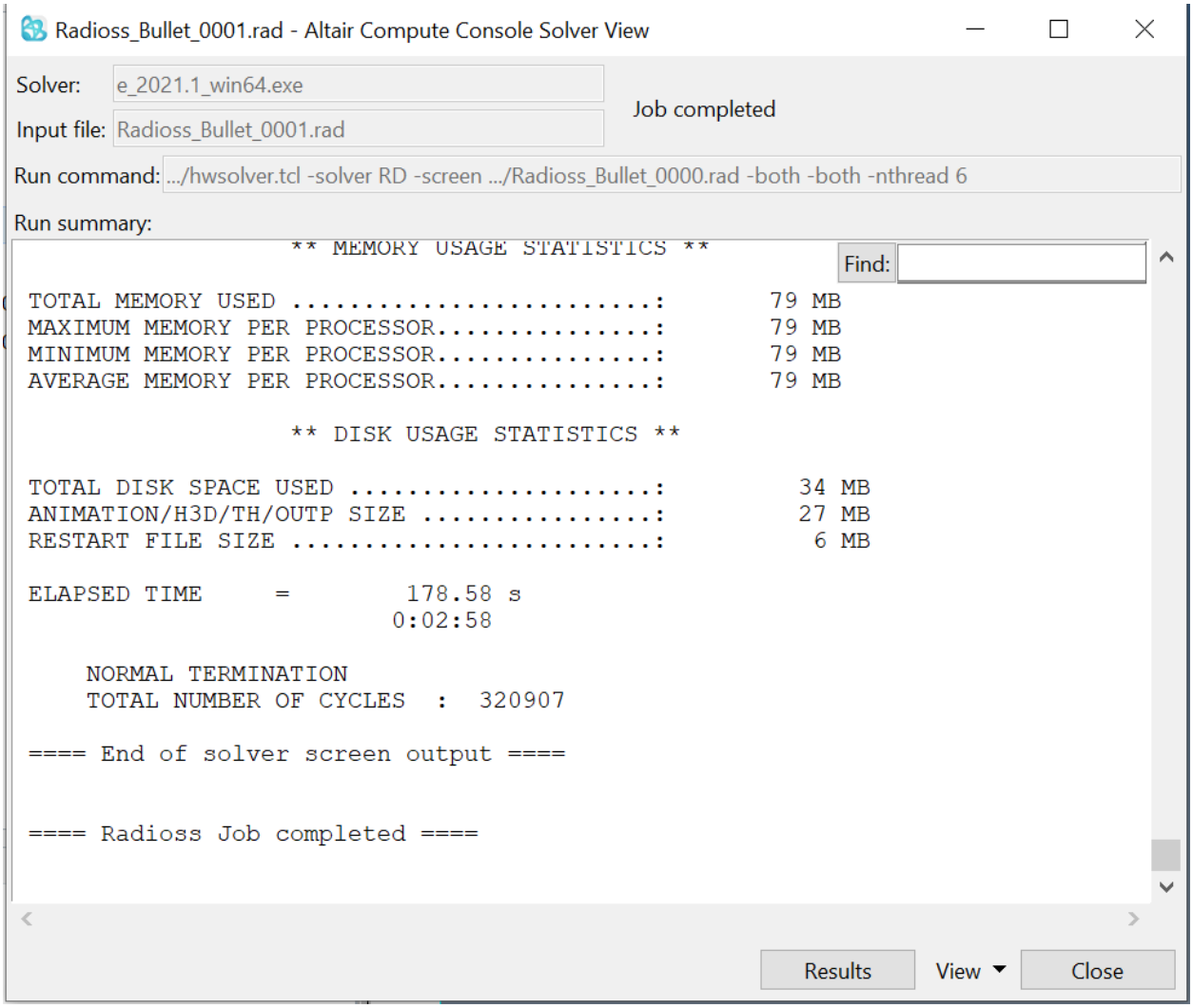
Step 6: RUN THE ANALYSIS

1. Once this is done return to the analysis panel and select **Radioss**.

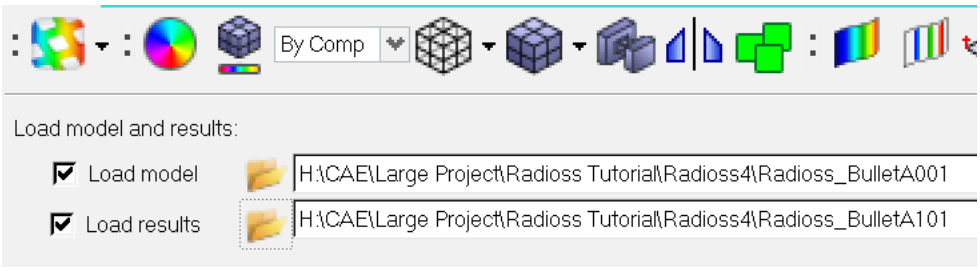
2. Save the input file that it has specified ending in **XXXX _0000.rad** in the same place as the model.

3. Under options type **–both –nthread 6**. (Note there is a space between both and -n as well as thread and 6)

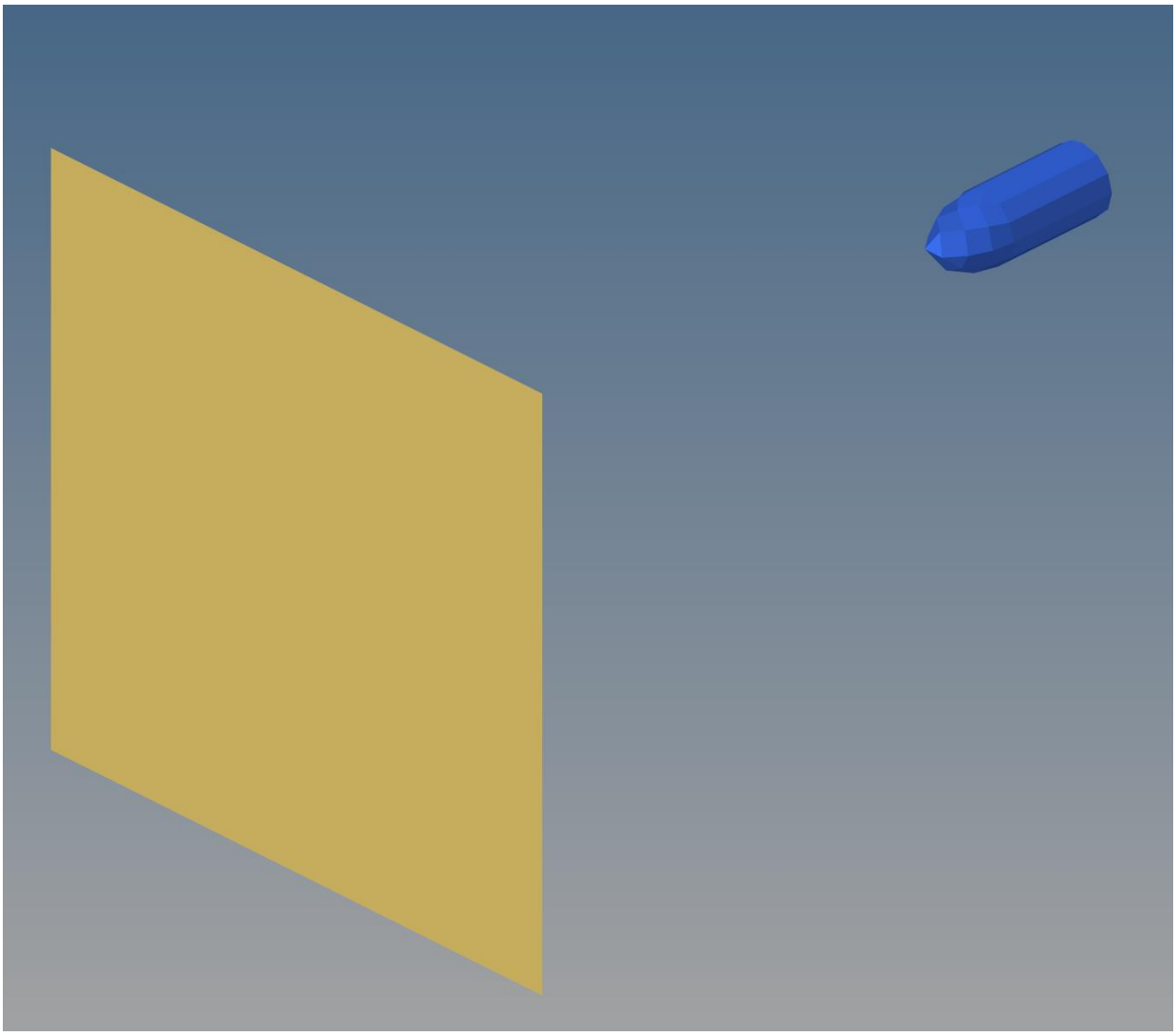
4. Click **Radioss** to run the simulation. A console should pop up with the solver running. Wait until it is done. When it is done you should see this in the console.



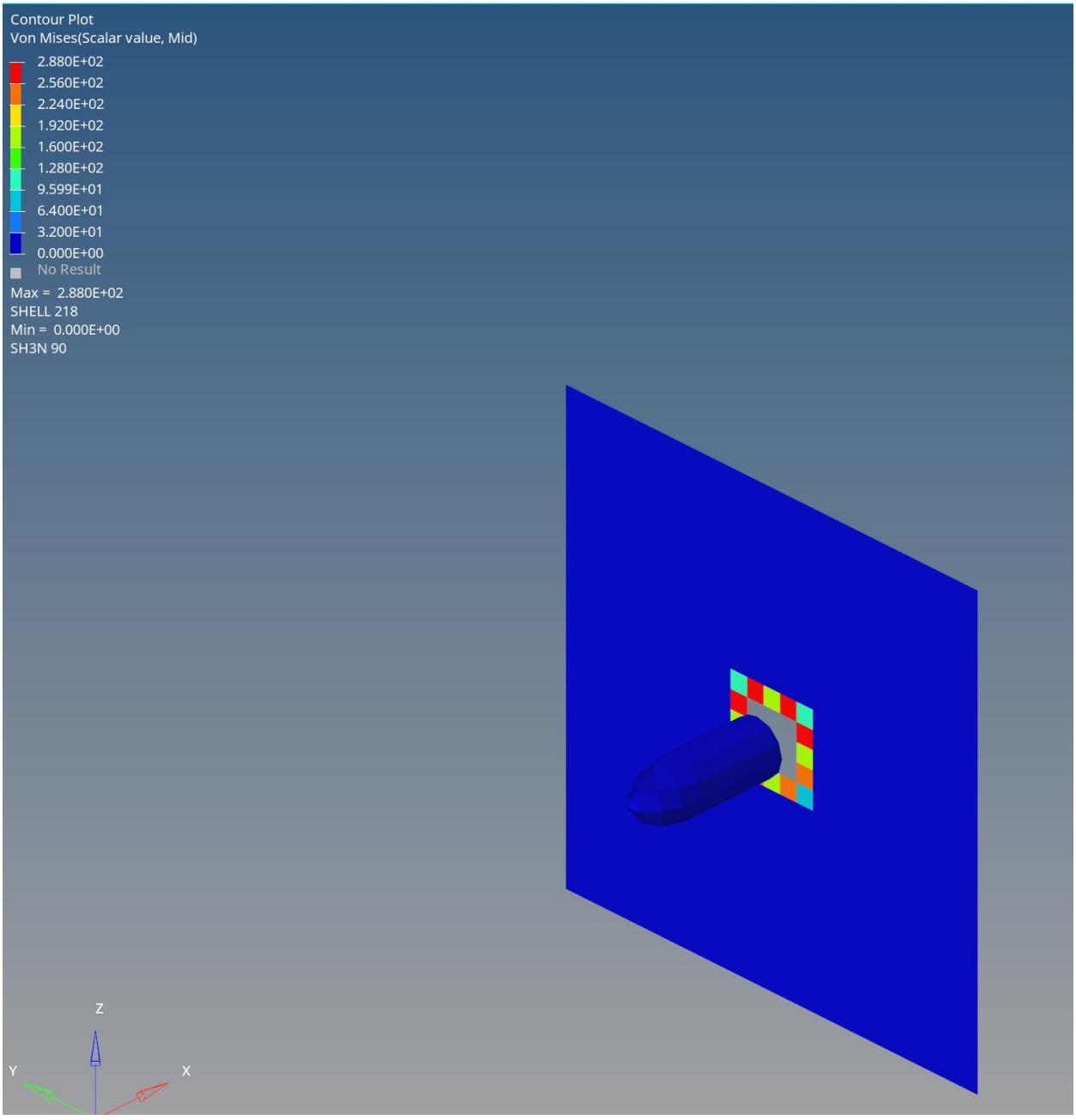
5. Open HyperView. In the bottom panel, you should see load model and load results. In the load model, open the file that ends in **A001**. For the load results select the **last run file**. A figure of what this will look like is below:



6. Once this is done, click apply in the bottom left corner of the software. Once completed you should see a the wall and bullet in hyperview:



7. You can the click the contour icon to see the displacement or the Von Mises Stress of the bullet and wall. Click the play button icon will allow you to see the bullet go through the wall. A snapshot of the video can be seen below:


Contour Plot
Von Mises(Scalar value, Mid)
2.880E+02
2.560E+02
2.240E+02
1.920E+02
1.600E+02
1.280E+02
9.599E+01
6.400E+01
3.200E+01
0.000E+00
No Result
Max = 2.880E+02
SHELL 218
Min = 0.000E+00
SH3N 90
Z
Y
X