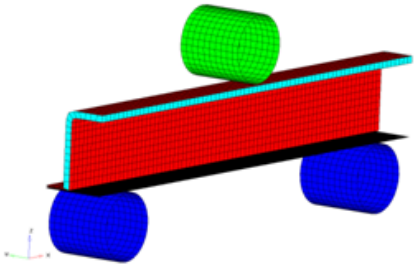
Three Point Bending with HyperMesh - RD-3595

To set up 3-point bending model with symmetric boundary conditions in Y direction.



Model Description

- UNITS: Length (mm), Time (s), Mass (ton), Force (N) and Stress (MPa)
- Simulation time: in Engine file [0 – 6.601e-002 s]
- Only one half of the model is modeled because it is symmetric.
- The supports are totally fixed. An imposed velocity of 1000 mm/s is applied on the Impactor in the (–Z) direction
- Model size = 370mm x 46.5mm x 159mm
- Honeycomb Material /MAT/LAW28: HONEYCOMB
  - $\rho = 3.0e^{-10}$ ton/mm^3 [Rho_I] Initial density
  - $E_{ij} = 200$ MPa [E11], [E22] and [E33] Young's modulus
  - $G_{ij} = 150$ MPa [G11], [G22] and [G33] Shear modulus
- Elasto-Plastic Material /MAT/LAW36: Inner, Outer and Flat
  - $\rho = 7.85^{-9}$ ton/mm^3 [Rho_I] Initial density
  - E = 210000 MPa [E] Young's modulus
  - $\upsilon = 0.29$ [nu] Poisson's ratio
  - Strain Curve:

	0	1	2	3	4	5	6	7	8	9
STRAIN	0	0.012002	0.014003	0.018003	0.022002	0.026003	0.030006	0.032	0.033005	0.033523
STRESS	325	335.968	343783	349.245	358.649	372.309	383.925	388.109	389.292	389.506

- Elastic Material /MAT/PLAS_JOHNS: Impactor
  - $\rho = 8e^{-9}$ ton/mm^3 [Rho_I] Initial density
  - E = 208000 MPa [E] Young's modulus
  - $\upsilon = 0.29$ [nu] Poisson's ratio

Exercise

Step 1: Load the RADIOSS User Profile

1. Launch HyperMesh 10.0.
2. From the ***Preferences*** menu, click ***User Profiles...***.
3. From the pop up window, select ***RADIOSS***, then choose ***Block 90*** from the pull-down menu.
4. Click ***OK***.

Step 2: Retrieve the RADIOSS file

1. From pull-down menu, click ***File***.
2. Click ***import...***

3. Click ***File***, navigate to the correct directory, and select `BENDING_0000.rad`.
4. Click ***Apply***.
5. Click ***Close*** to close the window.

Step 3: Create and Assign material and property for FOAM

1. On the ***Collectors*** menu, click ***Edit*** and select ***Components*** subpanel or from the toolbar click on ***component*** icon  and go to the ***Update*** page.
2. Click on ***comps*** and select ***Foam***.
3. Make sure ***card image =*** is set to ***Part***.
4. Click on ***create mat*** tab to create material.
5. For ***mat name =***, enter `Foam`.
6. Set ***type =*** to ***OTHER***.
7. For ***card image*** = select ***M28_HONEYCOMB***.
8. Click ***create/edit***.
9. Input the following values:

   Rho_I: `3.000e-10`

   E11: `200.000`

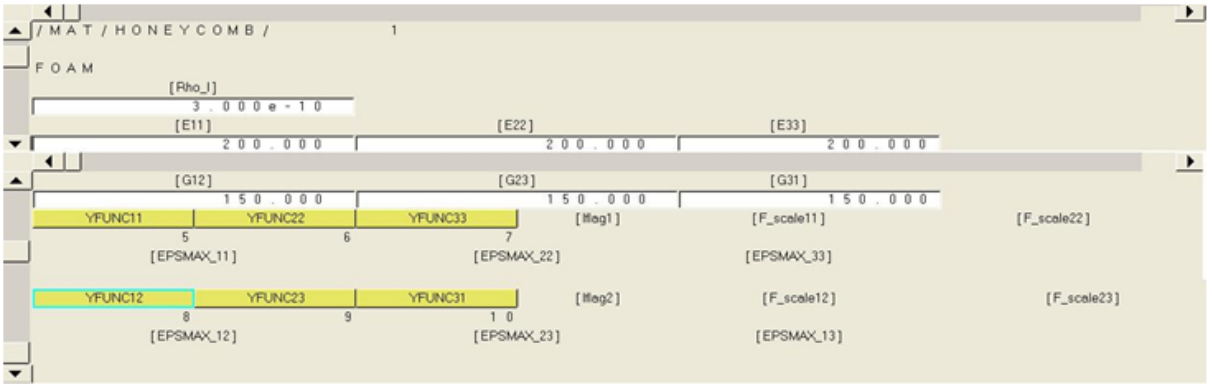
   E22: `200.000`

   E33: `200.000`

   G12: `150.000`

   G23: `150.000`

   G31: `150.000`



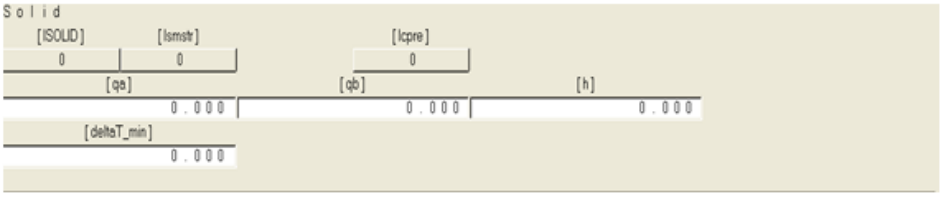
10. Click ***return*** to exit the panel and return to component panel.
11. Click on ***create prop*** tab to create property.
12. For ***prop name =***, enter `Foam`.
13. Set ***type =*** to ***VOLUME***.
14. Select ***card image*** and set to ***P14_Solid***.
15. Click ***create/edit***.
16. Enter the following values:

    qa: `0.000`

    qb: `0.000`

    h: `0.000`

    deltaT_min: `0.000`



17. Click ***return*** twice exit the panel.
18. Click ***update*** to update the component with created property and material.

Step 4: Create and Assign material and property for the component Inner

1. On the ***Collectors*** menu, click ***Edit*** and select ***Components*** subpanel or from the toolbar, click on ***component*** icon  and go to the ***Update*** page.
2. Click on ***comps*** and select Inner.
3. Make sure ***card image =*** is set to ***Part***.
4. Click on ***create mat*** tab to create material.
5. For ***mat name=***, enter `Inner`.
6. For ***type =***, select ***ELASTO-PLASTIC***.
7. For ***card image =*** select ***M36_PLAS_TAB*** .
8. Click ***create/edit***.
9. Input the following values:

   Rho_I: `7.85e-09`

   E: `210000.000`

   nu: `0.290`

   EPS_max: `0.000`

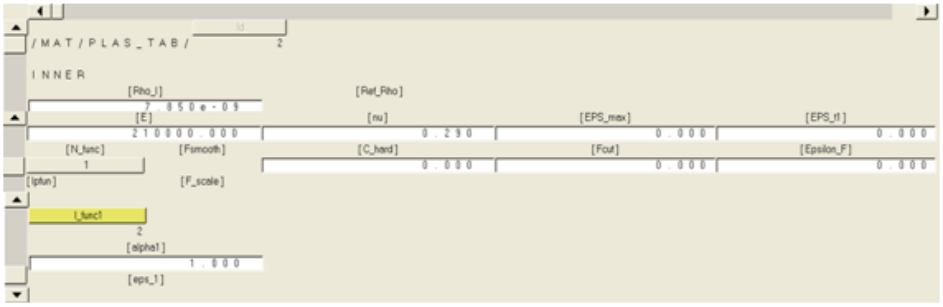
   EPS_t1: `0.000`

   C_hard: `0.000`

   Fcut: `0.000`

   Epsilon_F: `0.000`

   alpha1: `1.000`



10. Click ***return*** to exit the panel and return to component panel.
11. Click on ***create prop*** tab to create property.
12. For ***prop name =***, enter `Inner`.
13. Set ***type =*** to ***SURFACE***.
14. For ***card image =***, select ***P1_SHELL***.
15. For ***thickness=***, enter `9.119e-01`.
16. Click ***create/edit***.
17. Enter the following values:

   Hm: `0.00`

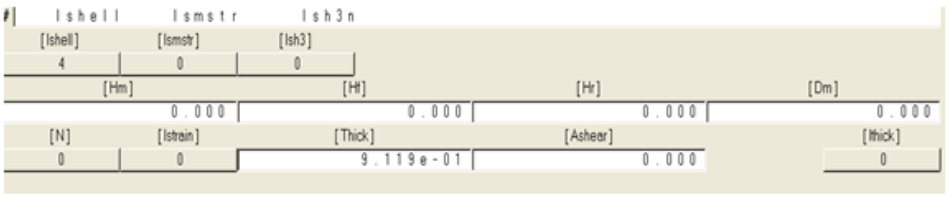
   Hf: `0.00`

   Hr: `0.00`

   Dm: `0.00`

   Thick: `9.119e-01`

   Ashear: `0.000`



18. Click ***return*** to exit the panel.
19. Click ***update*** to update the component with created property and material.

Step 5: Create and Assign material and property for the components Outer

1. On the ***Collectors*** menu, click ***Edit*** and select ***Components*** subpanel or from the toolbar click on ***component*** icon  and go to the ***Update*** page.
2. Click on ***comps*** and select ***Outer***.
3. Make sure ***card image =*** is set to ***Part***.
4. Click on ***create mat*** tab to create material.
5. For ***mat name=***, enter `Outer`.
6. For ***type =***, select ***ELASTO-PLASTIC***.
7. Change the ***card image =*** to ***same as*** and pick ***Inner***.
8. Click ***create*** to create material ***Outer*** with same values as material Inner.
9. Click ***return*** to return to the component panel.
10. Click on ***create prop*** tab to create property.
11. For ***prop name =***, enter `Outer`.
12. Set ***type =*** to ***SURFACE***.
13. Change the ***card image =*** to ***same as*** and pick ***Inner***.
14. Click ***create*** to create property ***Outer*** with same values as property Inner.
15. Click ***return*** to return to the component panel.
16. Click ***update*** to update the component with created property and material.

Step 6: Create and Assign material and property for the components Flat

Follow the procedure described in Step 5 with Outer replaced by ***Flat***.

Step 7: Create and assign material and property for Impactor

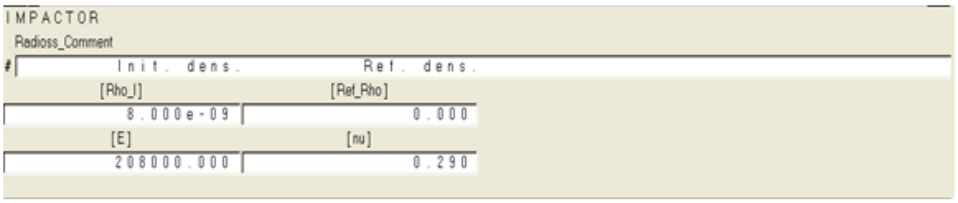
1. On the ***Collectors*** menu, click ***Edit*** and select ***Components*** subpanel or from the toolbar click on ***component*** icon  and go to the ***Update*** page.
2. Click on ***comps*** and select ***Impactor***.
3. Make sure ***card image =*** is set to ***Part***.
4. Click on ***create mat*** tab to create material.
5. For ***mat name=***, enter `Impactor`.
6. For ***type =***, select ***ELASTIC***.
7. For ***card image =***, select ***M1_ELAST***.
8. Click ***create/edit***.
9. Input the following values:

   Rho_I: `8.000e-09`

   Ref_Rho: `0.000`

   E: `208000.000`

   nu: `0.290`

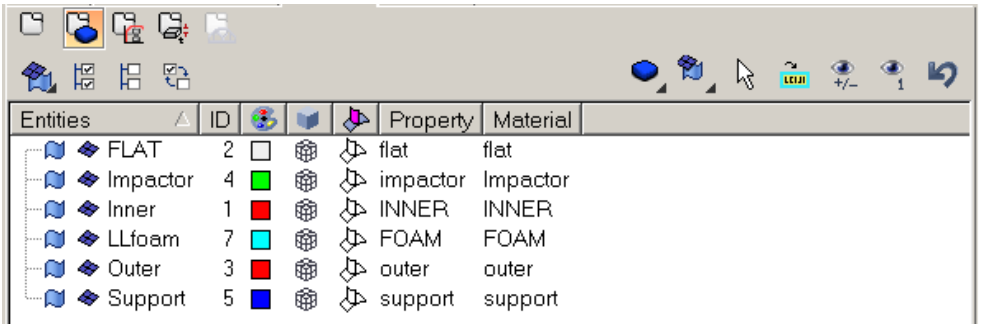


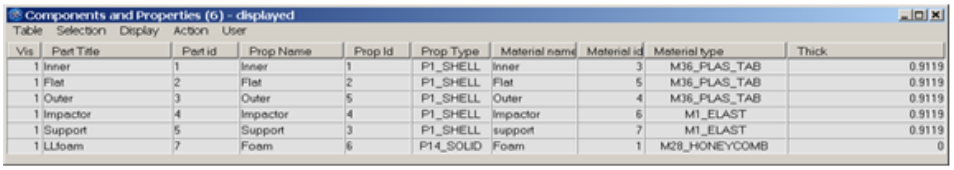
10. Click ***return*** to return to the component menu.
11. Click on ***create prop*** tab to create property.
12. For ***prop name =***, enter `Impactor`.
13. Set ***type =*** to ***SURFACE***.
14. Change the ***card image =*** to ***same as*** and pick ***Inner***.
15. Click ***create*** to create material ***Outer*** with same values are material Inner.
16. Click ***return*** to return to the component panel.
17. Click ***update*** to update the component with created property and material.

Step 8: Create and assign material and property for Support

Follow the same procedures as in Step 5. Create a copy of Impactor property and material with name ***support*** and assign it to component support.

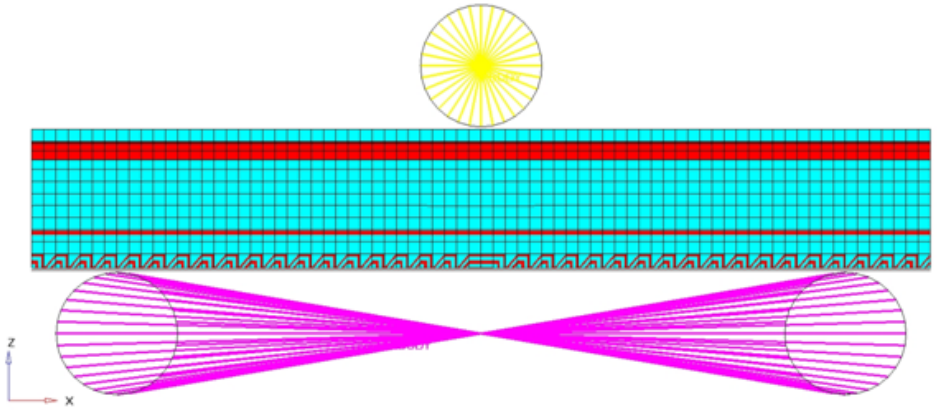
After completing Step 8, open the ***component*** view of the ***model*** browser or component table to check the correct assignment.





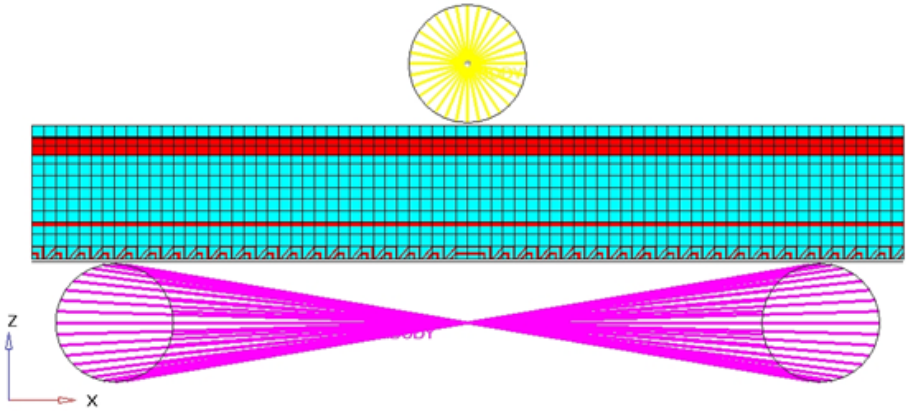
Step 9: Create a rigid body to make Impactor and Support Rigid

1. On the toolbar, click ***collector panel*** ().
2. Go to ***create*** subpanel.
3. For ***name=***, enter `Impact rigid`.
4. Select any color for easy visualization.
5. Switch to ***no card image***.
6. Switch to ***no property***.
7. Click ***create***.
8. Click ***return*** to exit the panel.
9. Go to ***1D page***, go to the ***rigids*** panel.
10. Ensure that you are in the ***create*** sub-panel.
11. For ***dependent*** switch to ***comps***.
12. For ***primary node*** switch ***calculate node***.
13. Click ***comps***.
14. Select ***Impactor***, then click ***select***.
15. Click ***create***.
16. Click ***return*** to exit the panel.
17. Similarly, create rigid body for ***Support*** component in a collector with the name "`Support rigid`" using sub-Steps 7.1 to 7.13.

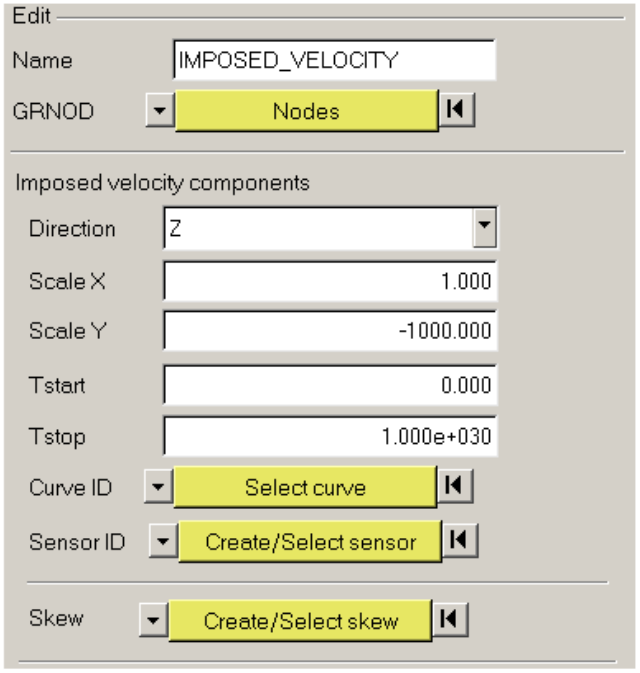


Step 10: Define imposed velocity and boundary condition for the impactor

1. From the ***Utility*** page, start the ***BC's Manager***.
2. For ***Name***, enter `IMPOSED_VELOCITY`, set ***Select type*** to ***Imposed velocity*** and set the ***GRNOD*** to ***Nodes***.
3. Click nodes and select the master node of the rigid body as shown in the following image.



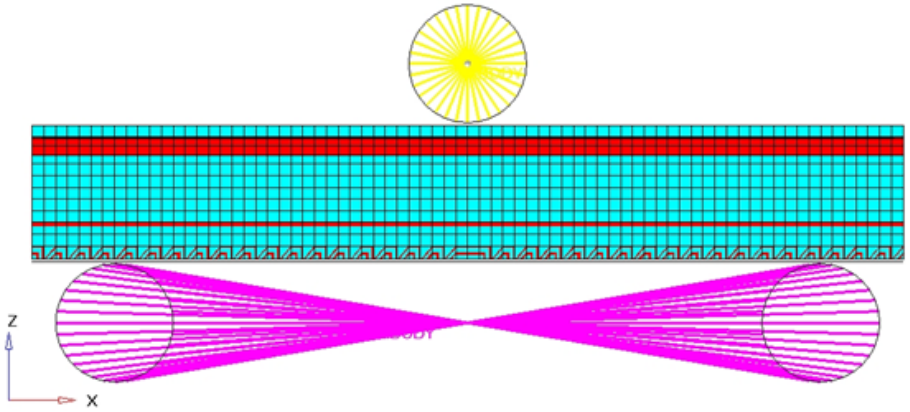
4. Set the ***Direction*** as ***Z***.
5. Set ***Scale Y*** to `-1000.0` as the direction of velocity is opposite to the global Z axis.
6. Set the ***Curve ID*** to ***Select curve*** .
7. Select the ***predefined curve*** to ***Func1***.
8. Click ***create*** to create the imposed velocity boundary condition.

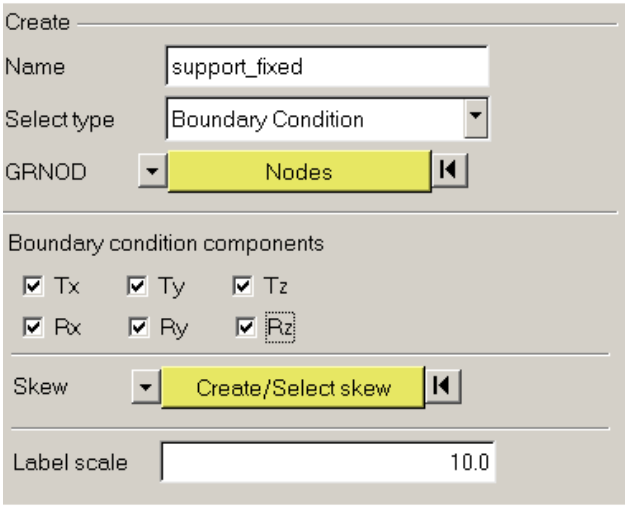


9. For ***Name***, enter `Impactor_constraints`, set ***Select type*** to ***Boundary condition*** and set the ***GRNOD*** to ***Nodes***.
10. Click ***nodes*** and select the ***master node*** of the rigid body.
11. Check all the degrees of freedom to constrain, except Tz.
12. Click ***create*** to create the boundary condition.

Step 11: Define fixed boundary condition for the support

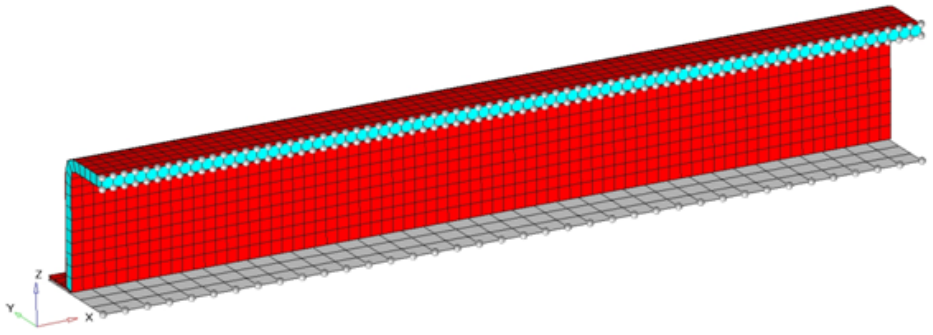
1. From the ***Utility*** page, start the ***BC's Manager***.
2. For ***Name***, enter `support_fixed`, set ***Select type*** to ***Boundary condition*** and set the ***GRNOD*** to ***Nodes***.
3. Select the master node of the rigid body created on Supporter as shown in the following image.
4. Check all the degrees of freedom to constraint.
5. Click ***create*** to create the boundary condition.

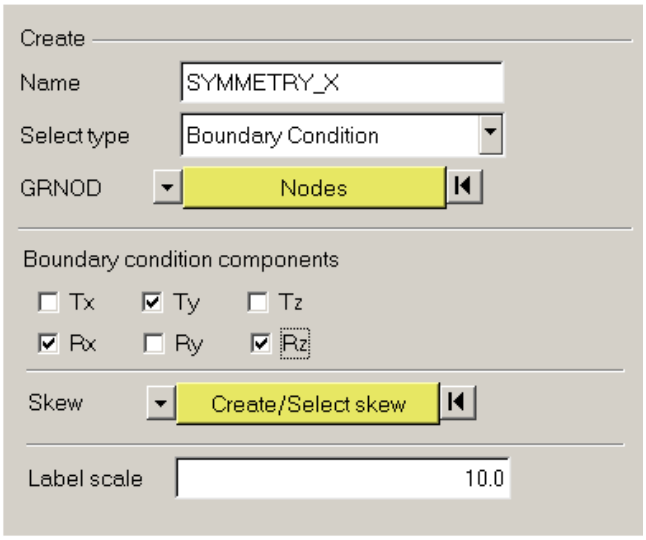




Step 12: Define symmetry boundary condition for the foam, inner, outer and flat

1. From the ***Utility*** page, start the ***BC's Manager***.
2. For ***Name***, enter `SYMMETRY_XZ`, set ***Select type*** to ***Boundary condition*** and set the ***GRNOD*** to ***Nodes***.
3. Select the nodes of the ***foam, inner, outer*** and ***flat*** as shown in the following image.
4. Check the degrees of translational degrees of freedom Y and rotational degrees of freedom X and Z to constraint.
5. Click ***create*** to create the boundary condition.

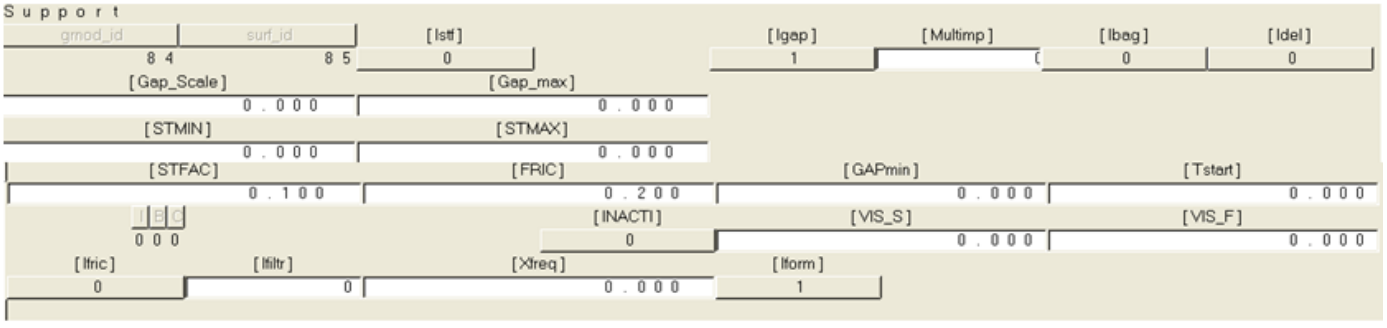




6. Click ***close*** to exit the BC Manager.

Step 13: Define contacts between the beam and the support

1. Enter the ***interfaces*** panel or from ***Tools*** ▶ ***Create cards*** ▶ ***Inter*** ▶ ***Type 7***.
2. For ***name =***, enter `Support`.
3. Set ***type =*** to ***TYPE7***.
4. Click ***create***.
5. Go to the ***add*** sub-panel.
6. Switch ***master selector*** to ***comps***.
7. Click the yellow ***comps*** button.
8. From the ***list of comps***, select ***Support***.
9. Click ***select*** ▶ ***update***.
10. Set the ***slave selector*** to ***comps***.
11. Click the yellow ***comps*** button.
12. Select the component ***FLAT***.
13. Click ***select*** ▶ ***update***.
14. Go to the ***card image*** sub-panel.
15. Click ***edit***.
16. Enter the values as in the following image:

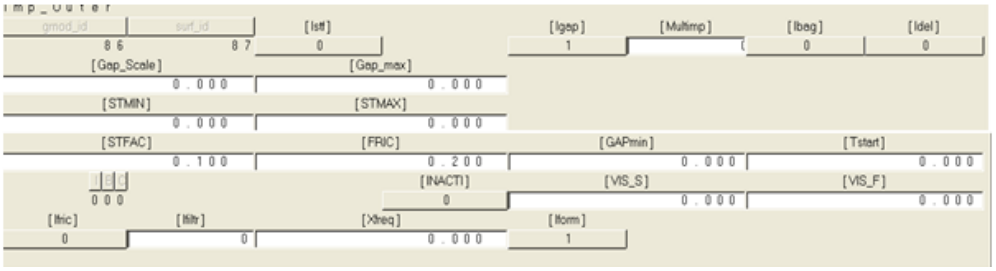


17. Click ***return*** twice to exit the panel.

Step 14: Define contacts between the impactor and the outer

1. From ***Analysis*** page ▶ ***interfaces*** panel ▶ ***create*** sub-panel.
2. For ***name =***, enter `Imp_Outer`.
3. Set ***type =*** to ***TYPE7***.
4. Click ***create***.
5. Go to the ***add*** sub-panel.
6. For ***master***, select ***comps***.
7. Click the yellow ***comps*** button.
8. From the ***list of comps***, select ***Impactor***.

9. Click ***select*** ▶ ***update***.
10. For ***slave***, select ***sets***.
11. Click the yellow ***sets*** button.
12. From the ***list of comps***, select ***Outer*** created previously.
13. Click ***select*** ▶ ***update***.
14. Go to the ***card image*** sub-panel and click ***edit***.

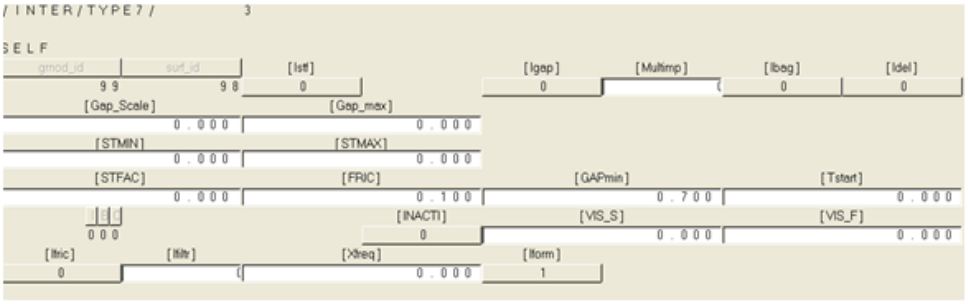


15. Click ***return*** twice to exit the panel.

Step 15: Define the self contact between the beam components

1. From the ***Analysis*** page, enter the ***interfaces*** panel, ***create*** sub-panel.
2. For ***name =***, enter `Self`.
3. Set ***type =*** to *TYPE7*.
4. Click ***create***.
5. Go to the ***add*** sub-panel.
6. Set the ***master selector*** to ***comps***.
7. Click the yellow ***comps*** button.
8. From the ***list of comps***, select ***Inner, Outer***, and ***Flat***.
9. Click ***select*** ▶ ***update***.
10. Set the ***slave selector*** to ***comps***.
11. Click the yellow ***comps*** button.
12. From the ***list of comps***, select ***Inner, Outer***, and ***Flat***.
13. Click ***select*** ▶ ***update***.
14. Go to ***card image*** subpanel.
15. Click ***edit***.

    Enter values as shown in the following image:



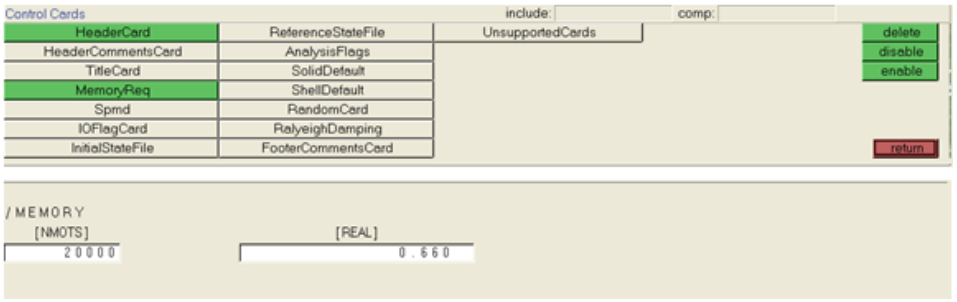
16. Click ***return*** twice to exit the panel.

Step 16: Create Interface time history

1. Go to ***Analysis*** page, then ***output block*** panel.
2. For ***name=***, enter `IMPACTOR`.
3. Switch the ***entity selector*** to ***groups***.
4. Click ***groups*** and select the interface ***Imp_Outer*** from the list.
5. Click ***select*** ▶ ***create*** ▶ ***edit***.
6. For ***VAR*** field, enter `DEF`
7. Click ***return*** twice to exit the panel.

Step 17: Allocate Required Memory

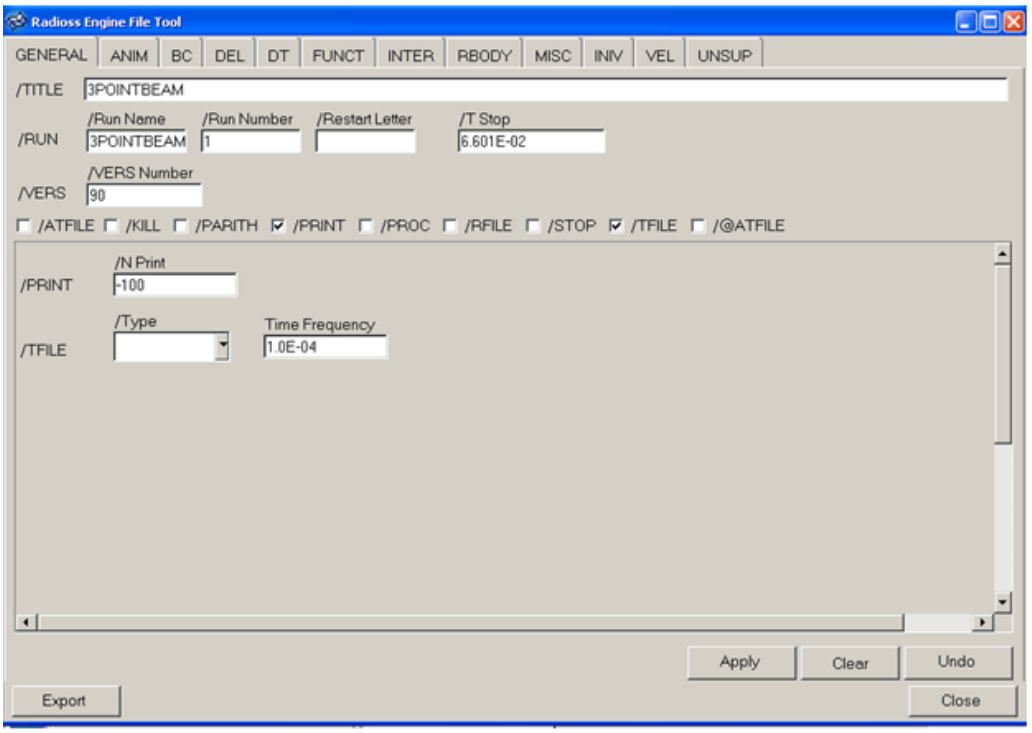
1. From the ***main*** menu, go to the ***Analysis*** page ▶ ***Control Cards*** sub-panel.
2. Click ***MemoryReq***.
3. Click ***NMOTS*** and enter `20000` as depicted in the following image.

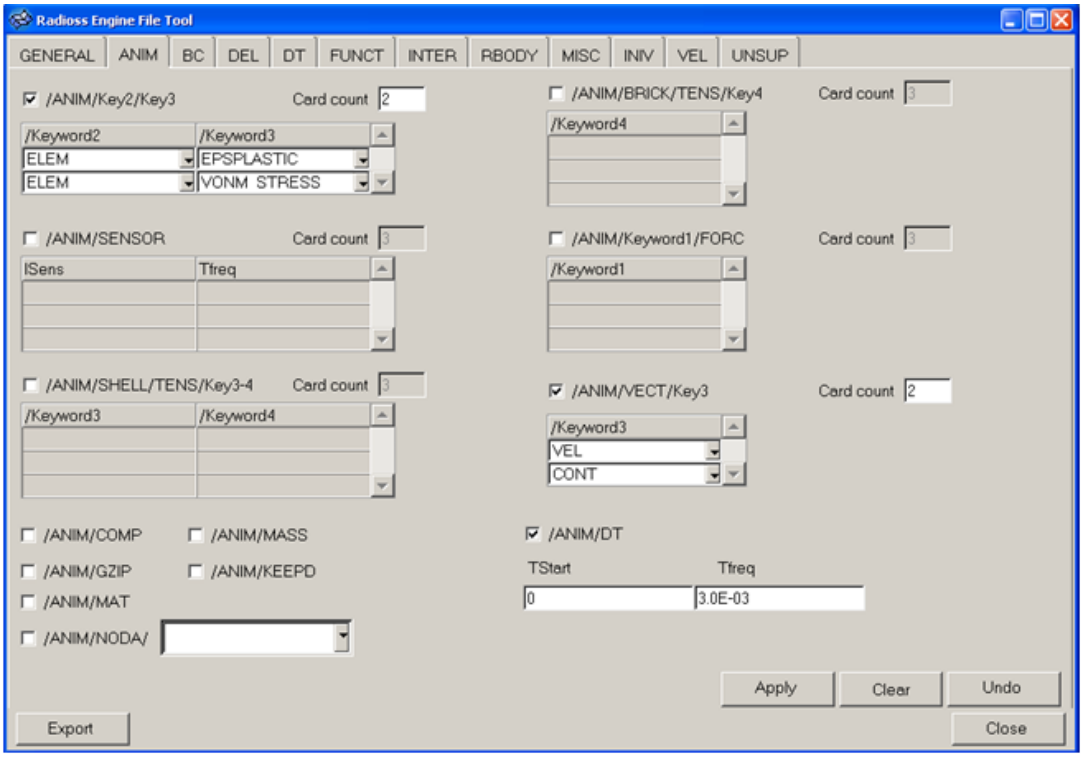


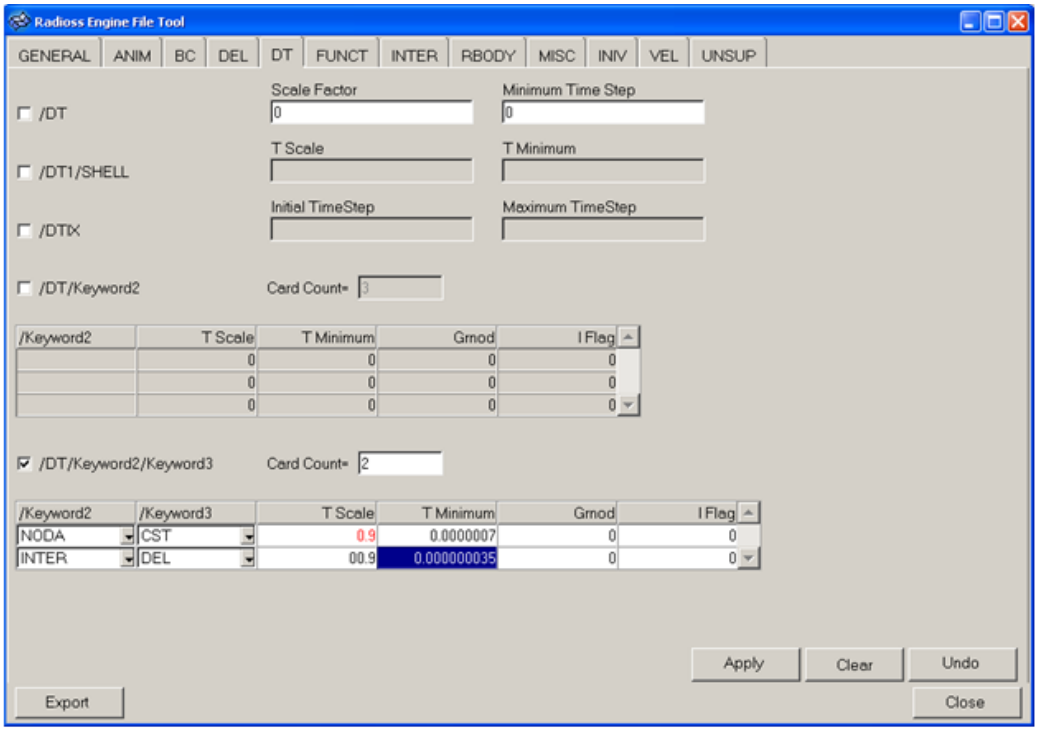
Step 18: Create output requests on control cards

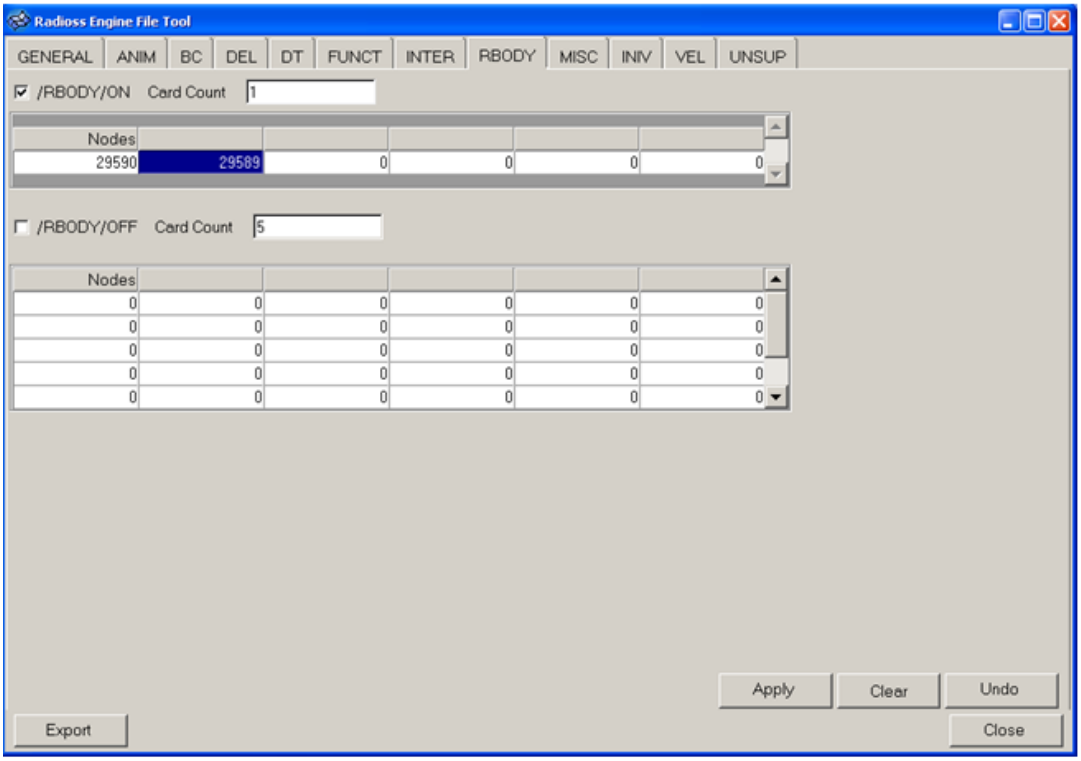
1. In the ***Utility*** Browser ▶ ***RADIOSS Tools*** menu page, click ***Engine File***.

   Enter the values as shown in each of the windows:




Radioss Engine File Tool
GENERAL
ANIM
BC
DEL
DT
FUNCT
INTER
RBODY
MISC
INIV
VEL
UNSUP
/ANIM/Key2/Key3
Card count
2
/Keyword2
/Keyword3
ELEM
EPSPLASTIC
ELEM
VONM STRESS
/ANIM/BRICK/TENS/Key4
Card count
3
/Keyword4
/ANIM/SENSOR
Card count
3
ISens
Ttreq
/ANIM/Keyword1/FORC
Card count
3
/Keyword1
/ANIM/SHELL/TENS/Key3-4
Card count
3
/Keyword3
/Keyword4
/ANIM/VECT/Key3
Card count
2
/Keyword3
VEL
CONT
/ANIM/COMP
/ANIM/MASS
/ANIM/GZIP
/ANIM/KEEPD
/ANIM/MAT
/ANIM/NODA/
/ANIM/DT
TStart
Ttreq
0
3.0E-03
Apply
Clear
Undo
Export
Close


Radioss Engine File Tool
GENERAL
ANIM
BC
DEL
DT
FUNCT
INTER
RBODY
MISC
INIV
VEL
UNSUP
/DT
Scale Factor
0
Minimum Time Step
0
/DT1/SHELL
T Scale
T Minimum
/DTIX
Initial TimeStep
Maximum TimeStep
/DT/Keyword2
Card Count=
3
/Keyword2
T Scale
T Minimum
Gmod
I Flag
/DT/Keyword2/Keyword3
Card Count=
2
/Keyword2
/Keyword3
T Scale
T Minimum
Gmod
I Flag
NODA
CST
0.9
0.0000007
0
0
INTER
DEL
00.9
0.000000035
0
0
Apply
Clear
Undo
Export
Close

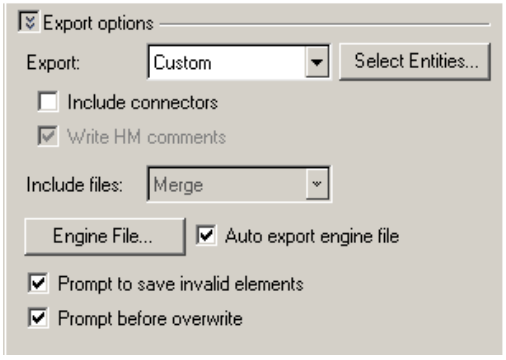


Select the master nodes of the two RBODY In the model for RBODY/ON nodes.

Note: The above values are the master node IDs of the respective RBODY of Impactor and Support. These may vary for different models.

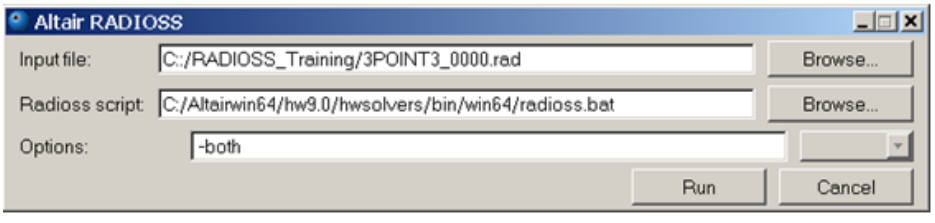
Step 19: Export the model

1. From the pull-down menu, select ***File***, then ***Export...***.
2. In the ***Export*** panel, from the ***Export type:*** pull-down menu select ***FE Model***.
3. Under ***File selection***, click the folder icon to select the name of the file (***BENDING***) to export and the destination directory of your choosing.
4. In the ***Export*** panel, click the arrows icon next to ***Export options*** to expand the panel.
5. For ***Export:*** select ***Custom***.
6. Deactivate the ***Include connectors*** option.
7. Activate the ***Auto export engine file*** option.
8. Activate the ***Prompt to save invalid elements*** option.
9. Activate the ***Prompt before overwrite*** option.



10. Click ***Apply*** ▶ ***Close***.

Step 20: Open RADIOSS Manager from windows Start menu



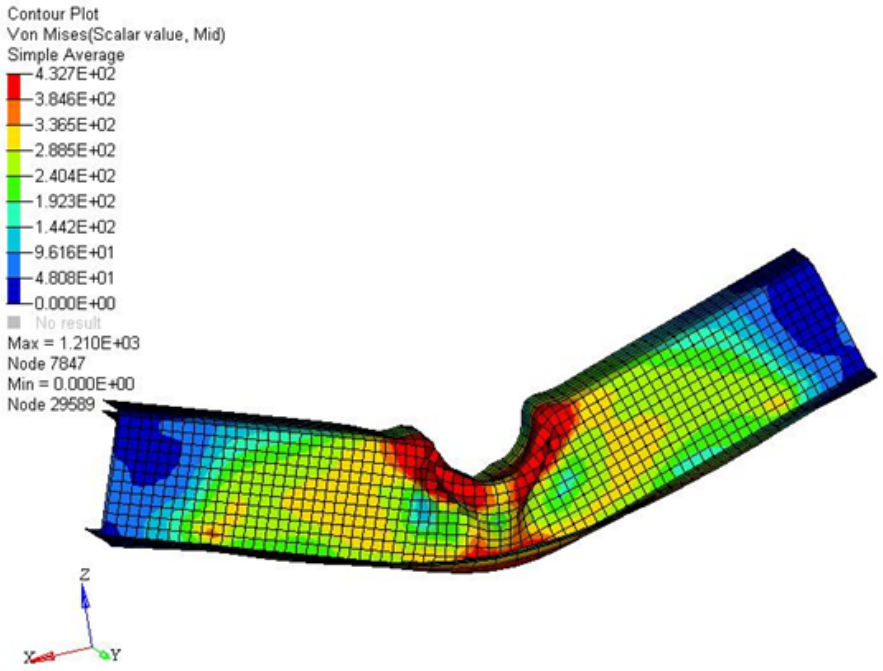
Step 21: Run the model `3POINT3_0000.rad` using RADIOSS Manager on the class_exercise folder with the

option: `-both`

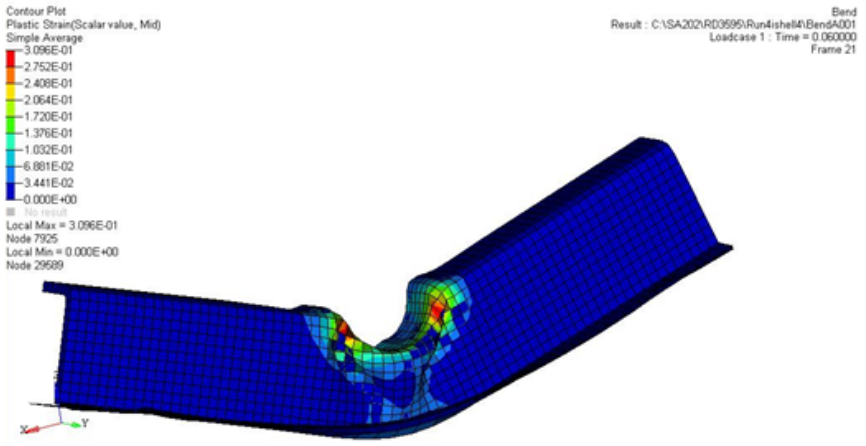
Step 22: Review the listing files for this run and verify on the results

1. See if there are any warnings or errors in `.out` files.
2. Using HyperView, plot the displacement, strain contour and vectors.

EXERCISE EXPECTED RESULTS



von Mises Stress Contour (MPa)



Plastic Strain Contour

Go To

RADIOSS, MotionSolve, and OptiStruct Tutorials