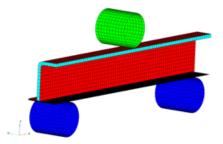
Three Point Bending with HyperMesh - RD-3595

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
 - $P = 3.0e^{-10} \text{ ton/mm}^3$ [Rho_I] Initial density
 - E_{ii} = 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
 - $P = 7.85^{-9} \text{ ton/mm}^3$ [Rho_I] Initial density
 - E = 210000 MPa [E] Young's modulus
 - v = 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

 P = 8e⁻⁹ ton/mm³ 	[Rho_I] Initial density
--	-------------------------

- E = 208000 MPa [E] Young's modulus
- u = 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

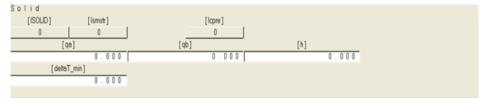
 - **G23**: 150.000
 - **G31**: 150.000

	▲ / MAT/HONEYCOMB/	1		<u>}</u>
	FOAM			
	[Rho_1]			
	3.000e-10 [E11]	[E22]	[E33]	
•	200.000	200.000	200.000	
	[G12]	[G23]	[G31]	<u>,</u>
	150.000	150.000	150.000	
	YFUNC11 YFUNC22	YFUNC33 [Hag1]	[F_scale11]	[F_scale22]
	5 6 [EPSMAX_11]	7 [EPSMAX_22]	[EPSMAX_33]	
	YFUNC12 YFUNC23	YFUNC31 [Hag2]	[F_scale12]	[F_scale23]
	[EPSMAX_12]	[EPSMAX_23]	[EPSMAX_13]	
-				

- 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

/ MAT / PLAS_T	A B /	2		•
INNER [Pho_] 7	350 e - 0 9	[Ret_Rho]		
 [E] 		[nu]	[EPS_mex]	[EPS_f1]
2 1 0	0000.000	0.290	0.000	0.000
[N_func]	[Fsmooth]	[C_hard]	[Fout]	[Epsilon_F]
[lplun]	[F_scale]	0.000	0.000	0.000
				
2				
[aipha	1.000			
[eps_]				

- 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:

	owing values.			
Hm : 0.00				
Hf : 0.00				
Hr : 0.00				
Dm : 0.00				
Thick : 9.11	9e-01			
Ashear: 0.0	000			
∦ Ishell	lsmstr	lsh3n		
[Ishell]	[Ismstr]	[lsh3]		
4	0	0		
[H	m]	[Ht]	[Hr]	[Dm]
	0.000	0.000	0.000	0.000
[N]	[Istrain]	[Thick]	[Ashear]	[thick]
0	0	9.119e-01	0.000	0

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_1:8.000e-09

Ref_Rho: 0.000

E: 208000.000

nu: 0.290

1	MPACTOR	
ļ	Radioss_Comment	
#	Init. dens.	Ref.dens.
	[Rho_I]	[Ref_Rho]
Γ	8.000e-09	0.000
	[E]	[nu]
Γ	208000.000	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

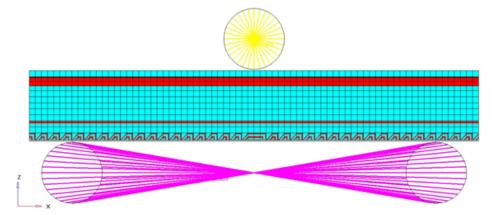
Follow the same procedures as in Step 5. Create a copy of Impactor property and material with name support and assign it to component support.

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

🔁 🗄 🗄	₹. ₹.) 🛍 🔓	° ₩ +/-	1
Entities) 🛞		🕭 Р	roperty	Material				
💭 🐟 FLA	λT.	2 🗆	1	J⊅ fla	t	flat				
📜 🐟 Imp	actor	4 🗖	ŵ	ÅÞ im	pactor	Impactor				
				Y.	•	•				
🗆 💭 🗢 Inn	er	I 📕	F	\D⊳ IN	NER	INNER				
🗆 🕽 🐟 LLfi	bam	7 🗖	B	JA FC)AM	FOAM				
😭 🐟 Out	er	3 🗖	ŵ	Åου	ter	outer				
				Ŷ						
🇆 🕽 🗇 Sup	pon	5 🗖	ŵ	Ø⊃ su	pport	support				
			w .							_ [] ×
able Selection Disp		User		Prop Id	Prop Type	Material name	Material id	Material type	Thick	
able Selection Disp	lay Action			Ptop Id	Prop Type P1_SHELL	Material name	Material id	Material type M36_PLAS_TAB	Thick	0.9119
ible Selection Disp is PartTitle	lay Action	User Prop N		Prop Id			Material id 3 5		Thick	
able Selection Disp is PartTitle 1 Inner 1 Flat 1 Outer	lay Action	User Prop N Inner Flat Outer	Vame	Prop Id 1 2 5	P1_SHELL P1_SHELL P1_SHELL	Inner Flat Outer	Material id 3 5 4	M36_PLAS_TAB M36_PLAS_TAB M36_PLAS_TAB	Thick	0.9119
able Selection Disp <u>fis PartTitle</u> 1 Inner 1 Flat 1 Outer 1 Impactor	lay Action	User Prop N Inner Flat Outer Impacto	Varme Ir	Prop Id 1 2 5 4	P1_SHELL P1_SHELL P1_SHELL P1_SHELL	Inner Flat Outer Impactor	Material id 3 5 4 6	M36_PLAS_TAB M36_PLAS_TAB M36_PLAS_TAB M36_PLAS_TAB M1_ELAST	Thick	0.9119 0.9119 0.9119
/is PartTitle 1 Inner 1 Flat 1 Outer	lay Action	User Prop N Inner Flat Outer	Varme Ir	Prop Id 1 2 5 4 3	P1_SHELL P1_SHELL P1_SHELL	Inner Flat Outer Impactor support	Material id 3 5 4 6 7	M36_PLAS_TAB M36_PLAS_TAB M36_PLAS_TAB	Thick	0.9119

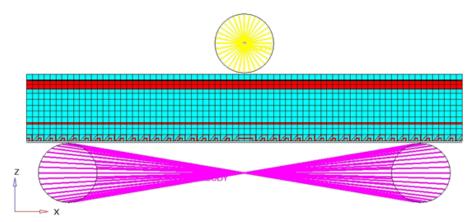
Step 9: Create a rigid body to make Impactor and Support Rigid

- 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.

Edit	
Luit	
Name	IMPOSED_VELOCITY
GRNOD	▼ Nodes I
Imposed vel	ocity components
Direction	Z
Scale X	1.000
Scale Y	-1000.000
Tstart	0.000
Tstop	1.000e+030
Curve ID	Select curve
Sensor ID	Create/Select sensor
Skew	Create/Select skew

- 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

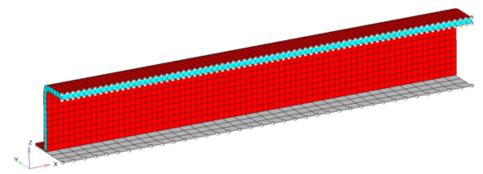
- 1. From the *Utility* page, start the *BC's Manager*.
- 2. For Name, enter support_fixed, set Select type to Boundary condition and set the GRNOD to Nodes.
- 3. Select the master node of the rigid body created on Supporter as shown in the following image.
- 4. Check all the degrees of freedom to constraint.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

Z	
Ĭ	
Z A	

Create	
Name	support_fixed
Select type	Boundary Condition
GRNOD -	Nodes K
Boundary condi	tion components
Tx T	Гу 🔽 Тz
🗹 Rx 🔽 🗹	Ry 🔽 Rz
Skew 💽	Create/Select skew
Label scale	10.0

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.



Create	
Name	SYMMETRY_X
Select type	Boundary Condition
GRNOD -	Nodes 📕
Boundary cond	
Skew 💌	Create/Select skew
Label scale	10.0

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:

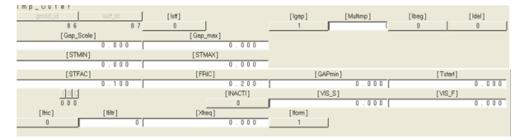
Support			
gmod_id surf_id	[lstf]	[lgap] [Multimp]	[lbag] [ldel]
84 85	0	1 (0 0
[Gap_Scale]	[Gap_max]		
0.000	0.000		
[STMIN]	[STMAX]		
0.000	0.000		
[STFAC]	[FRIC]	[GAPmin]	[Tstart]
0.100	0.200	0.000	0.000
IBO	[INACTI]	[VIS_S]	[VIS_F]
0 0 0	0	0.000	0.000
[lfric] [lfiltr]	[Xireq]	[lform]	
0 0	0.000	1	

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.

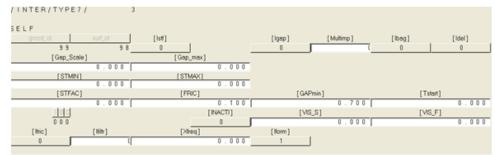


15. Click return twice to exit the panel.

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.

Control Cards		include:		comp:	
HeaderCard	ReferenceStateFile	UnsupportedCards			delete
HeaderCommentsCard	AnalysisFlags				disable
TitleCard	SolidDefault				enable
MemoryReq	ShellDefault				
Spmd	RandomCard				
IOFlagCard	RalyeighDamping				
InitialStateFile	FooterCommentsCard				return
/ MEMORY [NMOTS] 20000	[REAL] 0.6	6 0			

Step 18: Create output requests on control cards

 In the *Utility* Browser ► *RADIOSS Tools* menu page, click *Engine File*. Enter the values as shown in each of the windows:

😒 Radioss E	ngine File 1	fool												
GENERAL	ANIM	BC	DEL	DT	FUNCT	INTER	RBODY	MISC		VEL	UNSUP			
/TITLE 3	POINTBE	AM												
-	'Run Nam 3POINTBE		'Run Nu 1	mber	/Restart	Letter	/T Stop 6.601E-02	2	-					
-	VERS Nu 30	mber												
C /ATFILE	E /KILL	. 🗆 /F	PARITH	V /	PRINT E	/PROC	/RFILE	🗆 /STO	P 🗹 /	TFILE	□ /@ATF	LE		
/PRINT	/N Prin -100	vt.	_											-
/TFILE	/Type		•	Time 1.0E-	Frequency 04	-								
•														•
											Apply		Clear	Undo
Export														Close

🕸 Radioss Engine File Tool	
GENERAL ANIM BC DEL DT FUNCT INTER F	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	/ANIM/Keyword1/FORC Card count
ISens Ttreq	/Keyword1
ANIM/SHELL/TENS/Key3-4 Card count 3	ANIM/VECT/Key3 Card count 2
/Keyword3 /Keyword4	/Keyword3
F /ANIM/COMP F /ANIM/MASS	₽ /ANIM/DT
□ /ANIM/GZIP □ /ANIM/KEEPD	TStart Tfreq
I /ANIM/MAT	0 3.0E-03
T /ANIM/NODA/	
	Apply Clear Undo
Export	Close

😵 Radioss Engine	File Tool											
GENERAL	IM BC	DEL	DT	FUNCT	INTER	RBOD	MISC		VEL	UNSUP		
Г /ОТ			Scale 0	Factor		_	Minimum Ti 0	me Step		-		
F /DT1/SHELL			T Sce	le			T Minimum]		
Г /ОТТХ			Initial	TimeStep			Meximum T	ïmeStep]		
□ /DT/Keyword	2		Card (Count= 3								
/Keyword2	т	Scale	Т	Minimum		Grnod		Flag 🔺	1			
		0		0		0		0				
		0		0		0		0				
✓ /DT/Keyword	2/Keyword3		Card 0	Count= 2								
/Keyword2	/Keyword3			T Scale		imum	(Grnod		l Flag 🔺		
	CST			0.9		00007		0		0		
INTER .	DEL	-		00.9	0.0000	00035		0		0 🔻		
										Apply	Clear	Undo
Export												Close

🕸 Radioss Engine F	ile Tool						
GENERAL ANI	M BC DEL	DT FUNCT	INTER RBODY	Y MISC INIV	VEL UNSUP		
RBODY/ON	Card Count 1						
Nodes					^		
29590	29589	0	0	0	0		
					V		
F /RBODY/OFF	Card Count 5						
Nodes					-		
0	0	-	0	0	0		
0	0	-	0	0	0		
0	0	0	0	0	0		
0	0	0	0	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.

Export options										
, Enport option										
Export:	Custom	•	Select Entities							
🔽 Write HM	comments									
Include files: Merge *										
Engine File 🔽 Auto export engine file										
Prompt to save invalid elements										
Prompt befo	ore overwrite									

10. Click Apply ► Close.

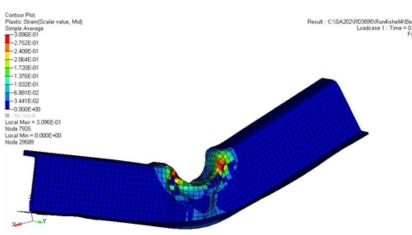
Step 20: Open RADIOSS Manager from windows Start menu

Altair RADIC	DSS		>	ĸ
Input file:	C::/RADIOSS_Training/3POINT3_0000.rad		Browse	
Radioss script:	C:/Altairwin64/hw9.0/hwsolvers/bin/win64/radioss.bat		Browse	
Options:	-both		Y	
		Run	Cancel	

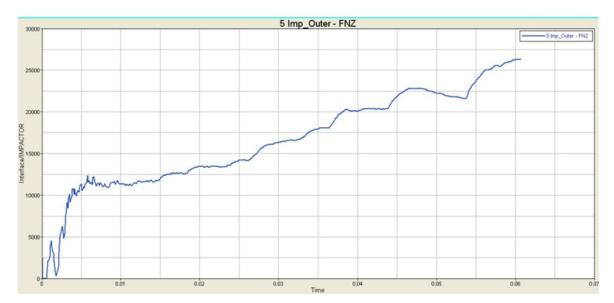
Step 22: Review the listing files for this run and verify on the results

- 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 Contour Plot Von Mises(Scalar value, Mid) Simple Average 4.327E+02 -3.846E+02 -3.365E+02 -2.885E+02 2.404E+02 1.923E+02 1.442E+02 -9.616E+01 4.808E+01 0.000E+00 -Max = 1.210E+03 Node 7847 Min = 0.000E+00 Node 29589 ; Y x von Mises Stress Contour (MPa) Result : C:\SA202\RD3595\Run4ishell4\Bend4001 Loadcase 1 : Time = 0.06000 Frame 21







Go To

RADIOSS, MotionSolve, and OptiStruct Tutorials