

# LS-Dyna/Pre-post Tutorial 1

Ball and Plate Impact

# Introduction

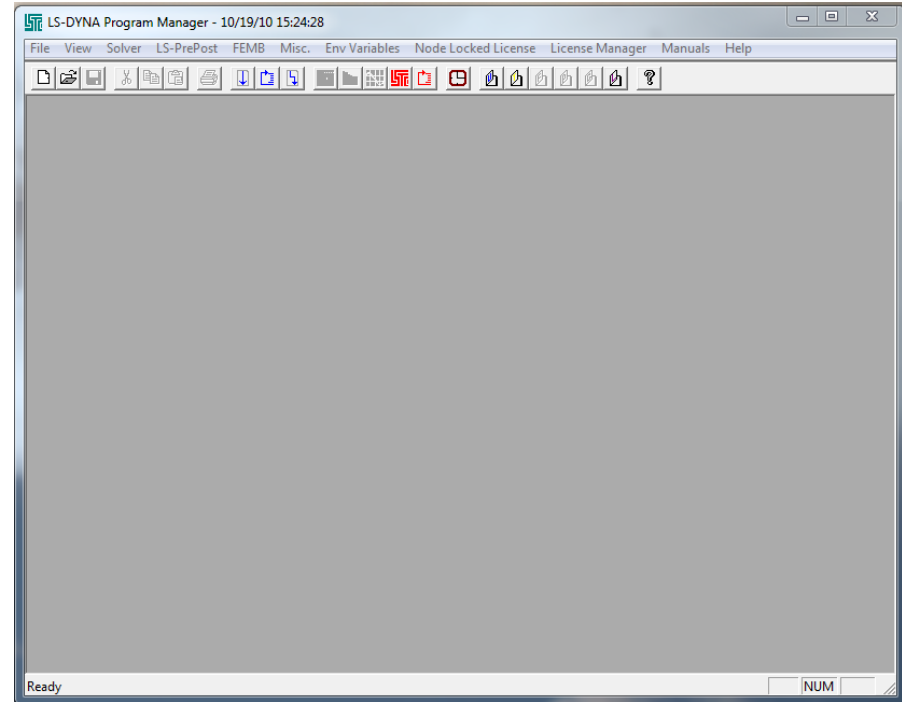
- This tutorial is an introduction LS-DYNA modeling using LS-Prepost
- In this tutorial, you will learn to:
  - Mesh a 2D plate and a 3D sphere
  - Assign section and materials properties to parts
  - Create node sets
  - Apply boundary constraints to node sets
  - Apply initial velocity conditions
  - Define part-to-part contacts
  - Set the simulation time, and define the output interval
  - Use the animation controls
  - Plot rigid body and nodal time-histories
  - Plot fringe components
- It is important that you process what you are doing – this is not a cookbook exercise – you will need to understand what is possible in each step, not just what you need to get done.

# Start the Program Manager and LS-PrePost

Much of your project management can be done from the LS\_DYNA Program Manager

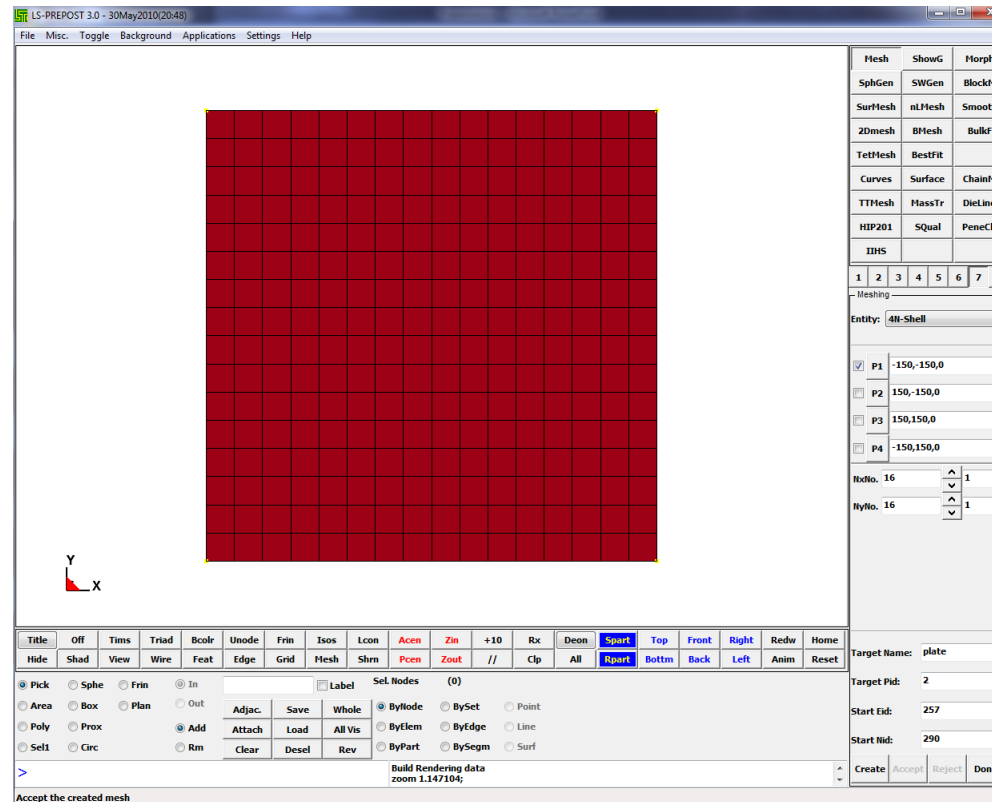
- From the Start Menu, select the LS-DYNA Manager
- Click on the LS-PrePost button on the menu bar and select “Start LS-PrePost”.
- Depending on the version you are running, you may need to press the F11 key to toggle to the old style menu that is used in this tutorial.
- The LS-PrePost application will appear

In this tutorial you will model an rigid ball impacting an elastic-plastic plate.



# Step 1: Create Plate Mesh

- Go to **page 6** and select **Mesh**
- In the Entity menu, select **4N-Shell**
- Enter the coords of the plate corners:
  - P1: -150,-150,0
  - P2: 150,-150,0
  - P3: 150,150,0
  - P4: -150,150,0
- Enter the # of elements in each dimension
  - NxNo: 16
  - NyNo: 16
- Enter Target Name
  - plate
- Click *Create*
- Click *Accept*



Mesh	ShowG	Morph
SphGen	SWGen	BlockM
SurfMesh	nLMesh	Smooth
2Dmesh	BMesh	BulkF
TetMesh	BestFit	
Curves	Surface	ChainM
TTMesh	MassTr	DieLine
HIP201	SQual	PeneCk
IIHS		

1 2 3 4 5 6 7 D

Meshing

Entity: 4N-Shell

☒ P1 -150,-150,0

☐ P2 150,-150,0

☐ P3 150,150,0

☐ P4 -150,150,0

NxNo. 16

NyNo. 16

Target Name: plate

Target Pid: 2

Start Eid: 257

Start Nid: 290

Create Accept Reject Done

# Step 2: Create Ball Mesh

Mesh	ShowG	Morph
SphGen	SWGen	BlockM
SurfMesh	nLMesh	Smooth
2Dmesh	BMesh	BulkF
TetMesh	BestFit	
Curves	Surface	ChainM
TTMesh	MassTr	DieLine
HIP201	SQual	PeneCk
IIHS		

1	2	3	4	5	6	7	D
---	---	---	---	---	---	---	---

Meshing

Entity: **Sphere\_Solid**

Radius: 50

density: 6

Center[Position]

X	Y	Z
0	0	51

Direct1

dx	1	dx	0
dy	0	dy	1
dz	0	dz	0

Direct2

Target Name: **ball**

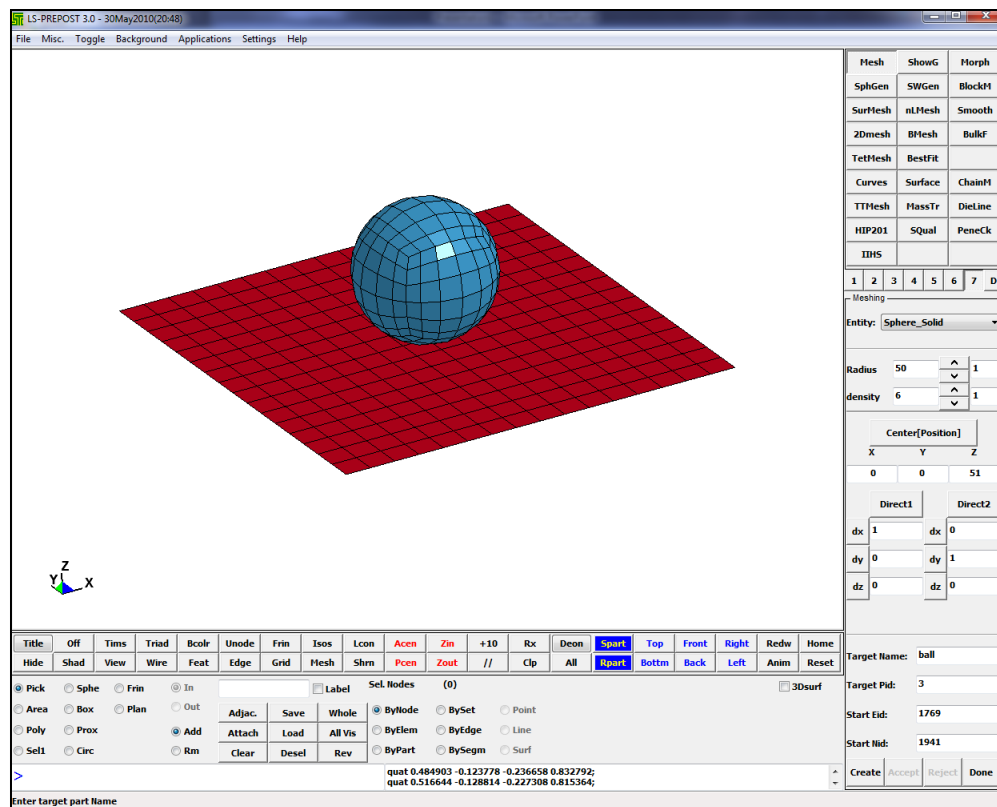
Target Pid: 3

Start Eid: 1769

Start Nid: 1941

Create Accept Reject Done

- In the Entity menu, select **Sphere\_Solid**
- Enter the ball radius and element density:
  - Radius: 50
  - Density: 6
- Enter the center location
  - X: 0
  - Y: 0
  - Z: 51
- Enter Target Name
  - ball
- Click *Create*
- Click *Accept*
- Click *Done*



# Step 3: Define Materials

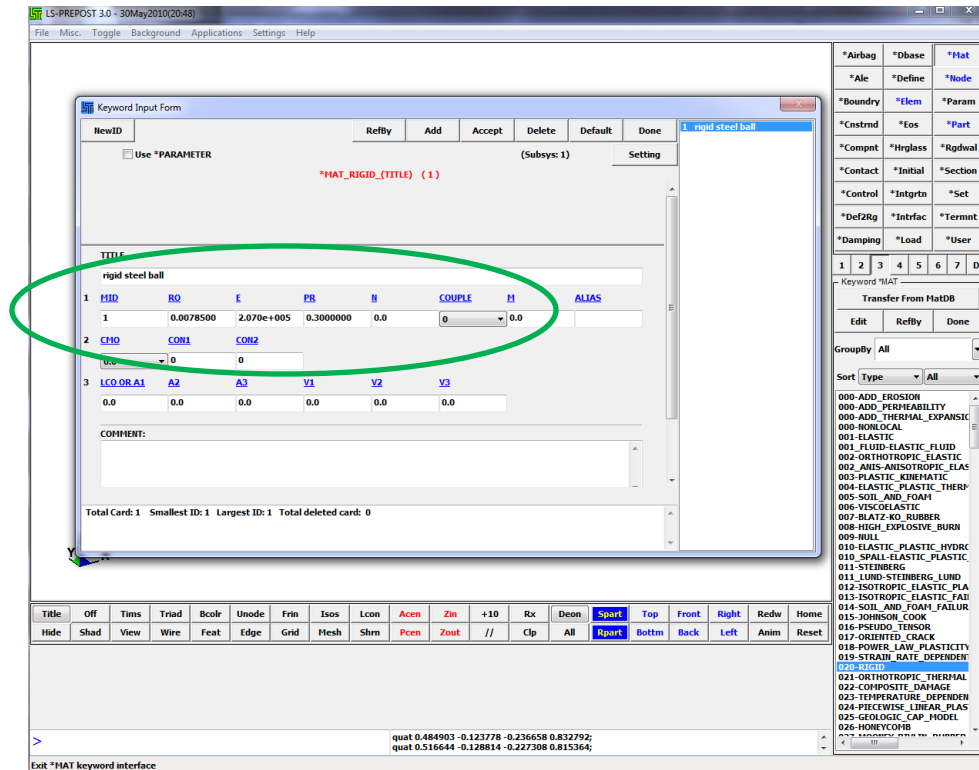
*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User

1	2	3	4	5	6	7	D
Keyword *MAT							
Transfer From MatDB							
Edit	RefBy	Done					
GroupBy All							
Sort Type All							
000-ADD_EROSION 000-ADD_PERMEABILITY 000-ADD_THERMAL_EXPANSIC 000-NONLOCAL 001-ELASTIC 001-FLUID-ELASTIC_FLUID 002-ORTHOTROPIC_ELASTIC 002-ANIS-ANISOTROPIC_ELAS 003-PLASTIC_KINEMATIC 004-ELASTIC_PLASTIC_THERP 005-SOIL_AND_FOAM 006-VISCOELASTIC 007-BLATZ-KO_RUBBER 008-HIGH_EXPLOSIVE_BURN 009-NULL 010-ELASTIC_PLASTIC_HYDRC 010-SPALL-ELASTIC_PLASTIC 011-STEINBERG 011-LUND-STEINBERG_LUND 012-ISOTROPIC_ELASTIC_PLA 013-ISOTROPIC_ELASTIC_FAI 014-SOIL_AND_FOAM_FAILURE 015-JOHNSON_COOK 016-PSEUDO_TENSOR 017-ORIENTED_CRACK 018-POWER_LAW_PLASTICITY 019-STRAIN_RATE_DEPENDEN 020-RIGID 021-ORTHOTROPIC_THERMAL 022-COMPOSITE_DAMAGE 023-TEMPERATURE_DEPENDEN 024-PIECEWISE_LINEAR_PLAS 025-GEOLOGIC_CAP_MODEL 026-HONEYCOMB 027-MOONEY-RIVLIN_RUBBER							

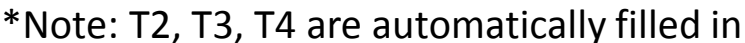
- Go to **page 3** and select **\*Mat**
- Select GroupBy: **All**, sort by **Type**
- Select 020-Rigid and click *Edit*
- Click *NewID* in the pop-up, and enter:
  - TITLE: rigid steel ball
  - RO: 7.85E-3
  - E: 207E+3
  - PR: 0.3

- Click *Accept*
- Select 024-PIECEWISE LINEAR PLASTICITY and click *Edit*
- Click *NewID*, and enter
  - TITLE: deformable plate
  - RO: 7.85E-3
  - E: 207E+3
  - PR: 0.3
  - SIGY: 200
  - ETAN:2E+3

- Click *Accept*
- Click *Done*



- In **page 3**, select **\*Section**
- Select SOLID from the list, and click *Edit*
- Click *NewID* in the pop-up, and enter:
  - **TITLE: solid ball**
  - **ELFORM = 1**
- Click *Accept*
- Click *Done*
- Select SHELL from the list, and click *Edit*
- Click on *NewID* and enter:
  - **TITLE: plate section**
  - **ELFORM = 2**
  - **NIP = 5**
  - **T1 = 1**
- Click *Accept*
- Click *Done*



1	2	3	4	5	6	7	D
Keyword SECTION							
Edit		RefBy		Done			
All				Model			
ALE2D BEAM BEAM_AISC DISCRETE POINT_SOURCE POINT_SOURCE_MIXTURE SPRING_DAMPER SEATBELT SHELL SHELL_ALE SHELL_EFG SHELL_THERMAL [*]SOLID (1) SOLID_ALE SOLID_EFG SPH TSHELL							

# Step 5: Define Parts

- In **page 3**, select **\*Part**
- Click on **[\*]PART (2)** and select *Edit*
- Select Part 1 on the list in the pop-up, and enter:
  - TITLE:** plate
- Click on the SECID button and select:
  - 2 plate section**
- Click on the MID button and select:
  - 2 deformable plate**
- Click *Accept*
- Select Part 2 and enter:
  - TITLE:** ball
- Click on SECID button and select:
  - 1 solid ball**
- Click on MID button and select:
  - 1 rigid steel ball**
- Click *Accept*
- Click *Done*

LS-PREPOST 3.0 - 30May2010(20:48)

File Misc Toggle Background Applications Settings Help

Keyword Input form

NewID Draw RefBy Pick Add Accept Delete Default Done

☐ Use \*PARAMETER (Subsys: 1) Setting

\*PART\*(TITLE) (2)

PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	THID
1	2	2	0	0	0	0	0

COMPEN...

Total Card: 2 Smallest ID: 1 Largest ID: 2 Total deleted card: 0

Title Off Time Triad Bcolr Unode Frin Isos Lcon Acon Zn +10 Rx Deon **Surf** Top Front Right Redw Home  
Hide Shad View Wire Feat Edge Grid Mesh Shm Pcon Zout // Clp All **Rpart** Bottom Back Left Anim Reset

Setting Subsystem Id

quat 0.484903 -0.123778 -0.236658 0.832792;  
quat 0.516644 -0.128814 -0.227308 0.815364;

*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User

1 2 3 4 5 6 7 D

Keyword \*PART

Edit RefBy Done

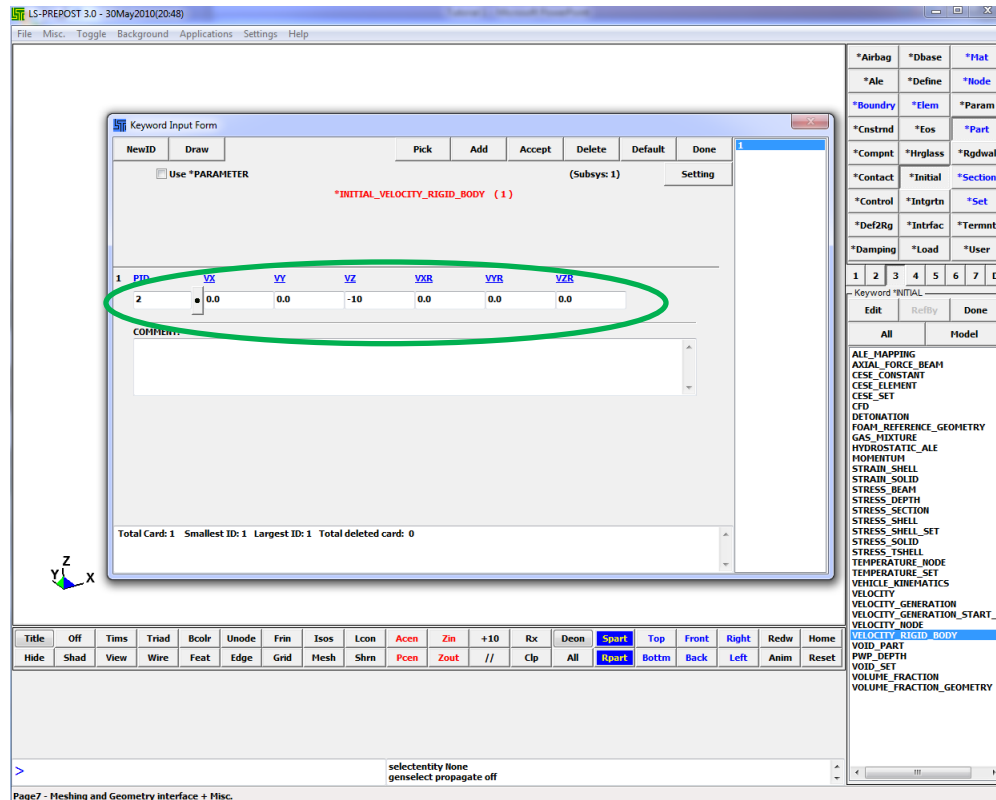
All Model

[\*]PART (2)



# Step 6: Define Initial Velocity

- In **page 3**, select **\*Initial**
- Select **VELOCITY\_RIGID\_BODY** from the list and click *Edit*
- In the pop-up, click on the PID button and select:
  - 2 ball
- Select VZ and enter:
  - VZ: -10
- Click *Accept*
- Click *Done*



*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	Section
*Control	Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User

1	2	3	4	5	6	7	D
---	---	---	---	---	---	---	---

Keyword *INITIAL	Edit	RefBy	Done
All	Model		

ALE\_MAPPING  
 AXIAL\_FORCE\_BEAM  
 CESE\_CONSTANT  
 CESE\_ELEMENT  
 CESE\_SET  
 CFD  
 DETONATION  
 FOAM\_REFERENCE\_GEOMETRY  
 GAS\_MIXTURE  
 HYDROSTATIC\_ALE  
 MOMENTUM  
 STRAIN\_SHELL  
 STRAIN\_SOLID  
 STRESS\_BEAM  
 STRESS\_DEPTH  
 STRESS\_SECTION  
 STRESS\_SHELL  
 STRESS\_SHELL\_SET  
 STRESS\_SOLID  
 STRESS\_TSHLL  
 TEMPERATURE\_NODE  
 TEMPERATURE\_SET  
 VEHICLE\_KINEMATICS  
 VELOCITY  
 VELOCITY\_GENERATION  
 VELOCITY\_GENERATION\_START\_1  
 VELOCITY\_NODE  
 VELOCITY\_RIGID\_BODY  
 VOID\_PART  
 PWP\_DEPTH  
 VOID\_SET  
 VOLUME\_FRACTION  
 VOLUME\_FRACTION\_GEOMETRY

## Step 7: Define Contact

- In **page 3**, select **\*Contact**
- Select **AUTOMATIC\_SURFACE\_TO\_SURFACE** from the list and click *Edit*
- Click *NewID* in the pop-up, and enter:
  - **TITLE: ball to plate contact**
- Set SSTYP and MSTYP to 3
- Click on the SSID button and select:
  - **1 plate**
- Click MSID and select:
  - **2 ball**
- Enter the following:
  - **FS: 0.1**
  - **FD: 0.1**
- Check box **A** and set **SOFT** equal to 2
- Click *Accept*
- Click *Done*

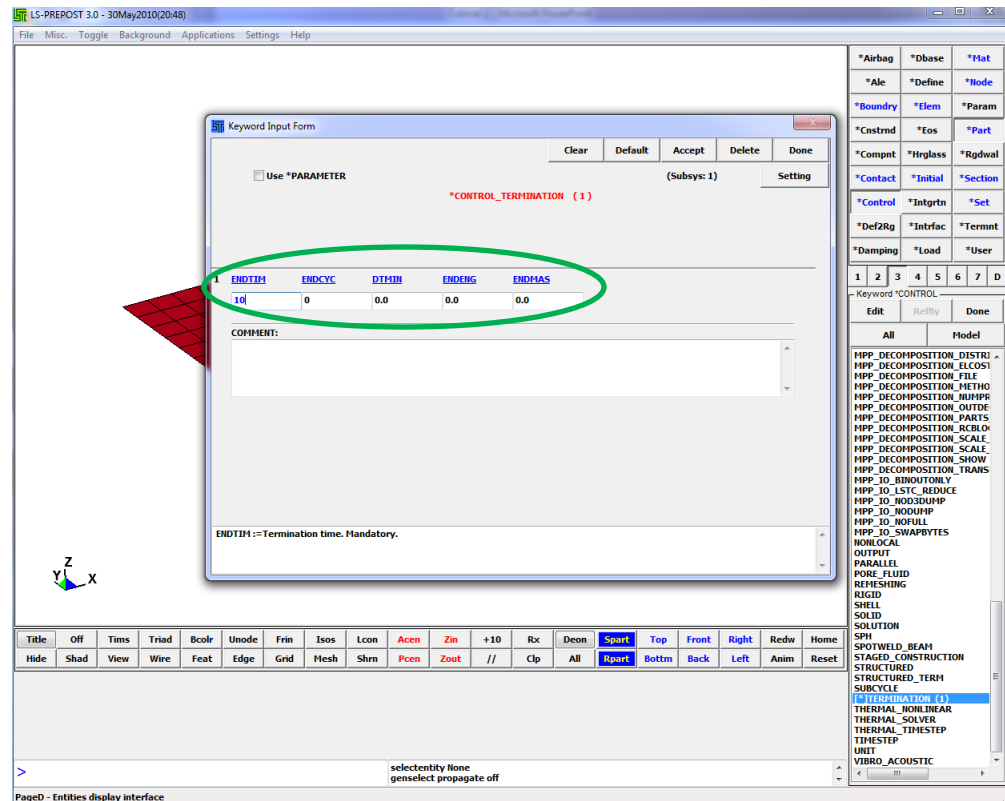
[illegible]

*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User

Keyword CONTACT		
Edit	RefBy	Done
All	Model	
AIRBAG_SINGLE_SURFACE		
AUTOMATIC_GENERAL		
AUTOMATIC_GENERAL_MPP		
AUTOMATIC_GENERAL_INTERI		
AUTOMATIC_NODES_TO_SURF		
AUTOMATIC_NODES_TO_SURF		
AUTOMATIC_ONE_WAY_SURF		
AUTOMATIC_ONE_WAY_SURF		
AUTOMATIC_ONE_WAY_SURF		
AUTOMATIC_ONE_WAY_SURF		
AUTOMATIC_SINGLE_SURFACE		
AUTOMATIC_SINGLE_SURFACE		
AUTOMATIC_SURFACE_TO_SU		
AUTOMATIC_SURFACE_TO_SU		
AUTOMATIC_SURFACE_TO_SU		
AUTOMATIC_SURFACE_TO_SU		
CONSTRAINT_NODES_TO_SURI		
CONSTRAINT_SURFACE_TO_SU		
DRAWBEAD		
DRAWBEAD_INITIALIZE		
ERODING_NODES_TO_SURFACE		
ERODING_SINGLE_SURFACE		
ERODING_SURFACE_TO_SURF		
FORCE_TRANSDUCER		
FORCE_TRANSDUCER_CONSTR		
FORCE_TRANSDUCER_PENALTY		
FORMING_NODES_TO_SURFACE		
FORMING_NODES_TO_SURFACE		
FORMING_ONE_WAY_SURFACE		
FORMING_ONE_WAY_SURFACE		
FORMING_SURFACE_TO_SURF		
FORMING_SURFACE_TO_SURF		
GUIDED_CABLE		
GUIDED_CABLE_SET		
NODES_TO_SURFACE		
NODES_TO_SURFACE_SMOOTH		
NODES_TO_SURFACE_INTERFEI		
ONE_WAY_SURFACE_TO_SURF		
ONE_WAY_SURFACE_TO_SURF		
ONE_WAY_SURFACE_TO_SURF		

# Step 8: Set End Time

- In **page 3**, select **\*Control**
- Select **TERMINATION** from the list and click *Edit*
- Enter:
  - **ENDTIME: 10**
- Click *Accept*
- Click *Done*



*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User

1	2	3	4	5	6	7	D
---	---	---	---	---	---	---	---

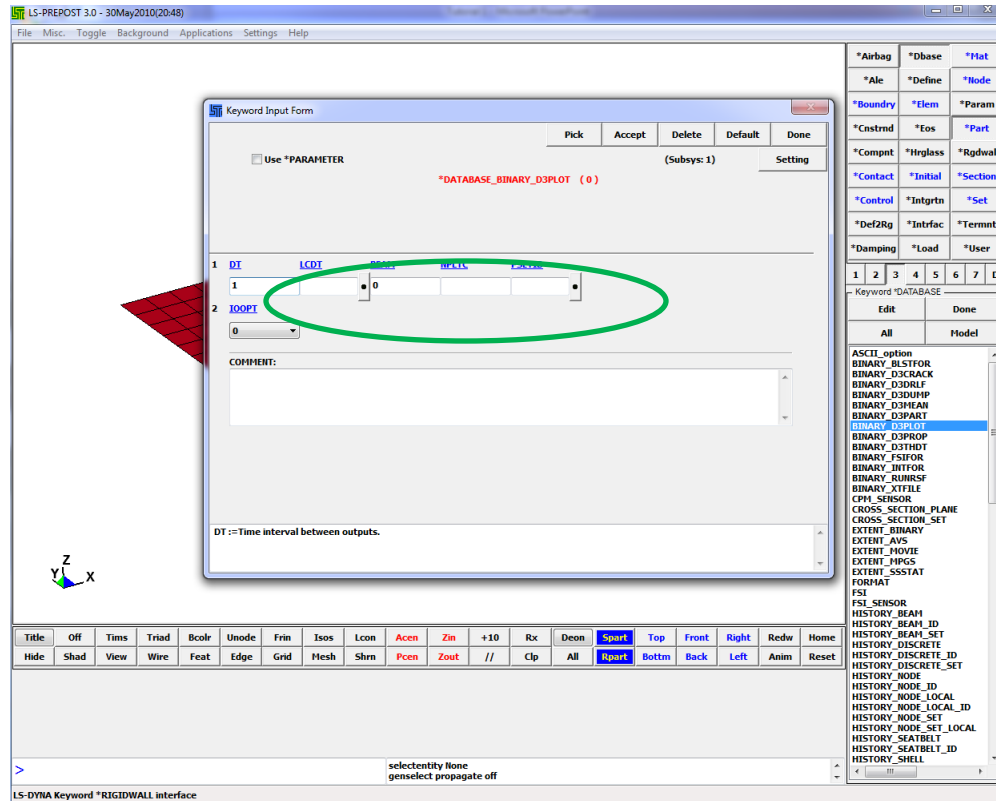
Keyword = CONTROL		
Edit	RefBy	Done
All	Model	
MPP_DECOMPOSITION_DISTRI		
MPP_DECOMPOSITION_ELCOST		
MPP_DECOMPOSITION_FILE		
MPP_DECOMPOSITION_METHO		
MPP_DECOMPOSITION_NUMPR		
MPP_DECOMPOSITION_OUTDE		
MPP_DECOMPOSITION_PARTS		
MPP_DECOMPOSITION_RCBLO		
MPP_DECOMPOSITION_SCALE		
MPP_DECOMPOSITION_SCALE		
MPP_DECOMPOSITION_SHOW		
MPP_DECOMPOSITION_TRANS		
MPP_IO_BINOUTONLY		
MPP_IO_LSTC_REDUCE		
MPP_IO_NOD3DUMP		
MPP_IO_NODUMP		
MPP_IO_NOFULL		
MPP_IO_SWAPBYTES		
NONLOCAL		
OUTPUT		
PARALLEL		
PORE_FLUID		
REMESHING		
RIGID		
SHELL		
SOLID		
SOLUTION		
SPH		
SPOTWELD_BEAM		
STAGED_CONSTRUCTION		
STRUCTURED		
STRUCTURED_TERM		
SUBCYCLE		
*TERMINATION (1)		
THERMAL_NONLINEAR		
THERMAL_SOLVER		
THERMAL_TIMESTEP		
TIMESTEP		
UNIT		
VIBRO_ACOUSTIC		

# Step 9: Set Output Frequency

*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User
1	2	3

Keyword	DATABASE
Edit	Done
All	Model
ASCII_option	
BINARY_BLSTFOR	
BINARY_D3CRACK	
BINARY_D3DRLF	
BINARY_D3DUMP	
BINARY_D3MEAN	
BINARY_D3PART	
BINARY_D3PLOT	
BINARY_D3PROP	
BINARY_D3THDT	
BINARY_FSIFOR	
BINARY_INTFOR	
BINARY_RUNRSF	
BINARY_XTFILE	
CPM_SENSOR	
CROSS_SECTION_PLANE	
CROSS_SECTION_SET	
EXTENT_BINARY	
EXTENT_AVS	
EXTENT_MOVIE	
EXTENT_MPGS	
EXTENT_SSSTAT	
FORMAT	
FSI	
FSI_SENSOR	
HISTORY_BEAM	
HISTORY_BEAM_ID	
HISTORY_BEAM_SET	
HISTORY_DISCRETE	
HISTORY_DISCRETE_ID	
HISTORY_DISCRETE_SET	
HISTORY_NODE	
HISTORY_NODE_ID	
HISTORY_NODE_LOCAL	
HISTORY_NODE_SET	
HISTORY_NODE_LOCAL_ID	
HISTORY_NODE_SET_LOCAL	
HISTORY_SEATBELT	
HISTORY_SEATBELT_ID	
HISTORY_SHELL	

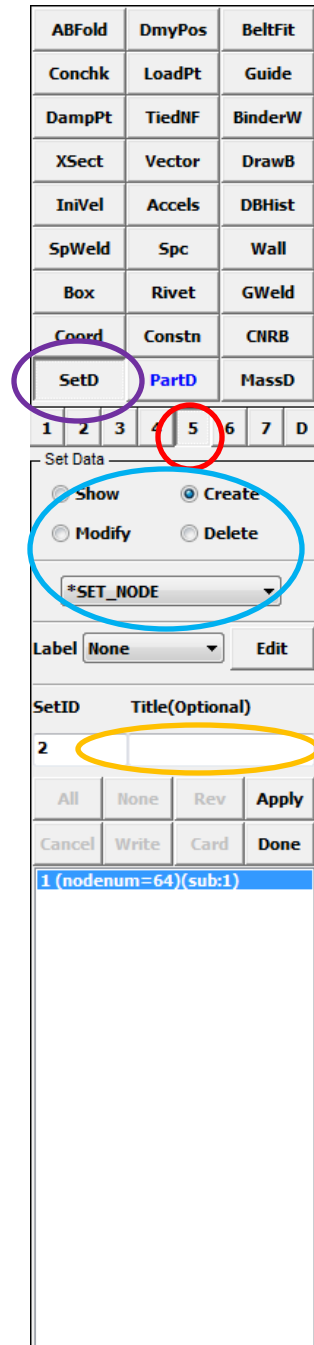
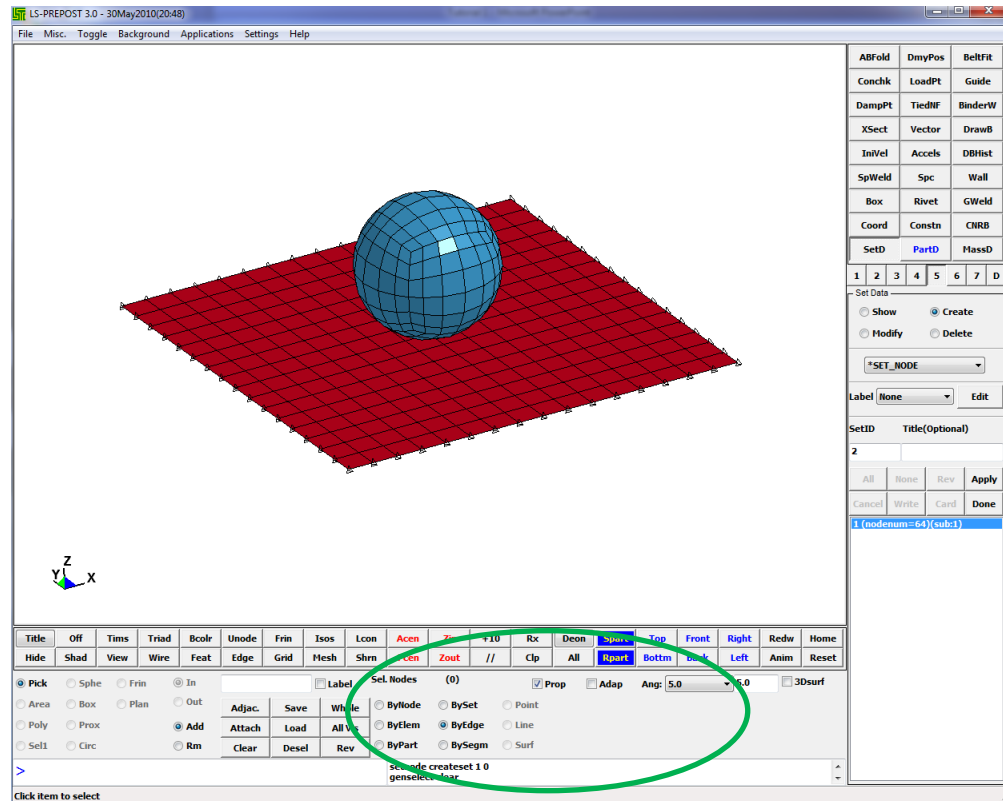
- In **page 3**, select **\*Dbase**
- Select **BINARY\_D3PLOT** from the list and click *Edit*
- Enter:
  - DT: 1
- Click *Accept*
- Click *Done*



# Step 10: Define Edge Nodes Set

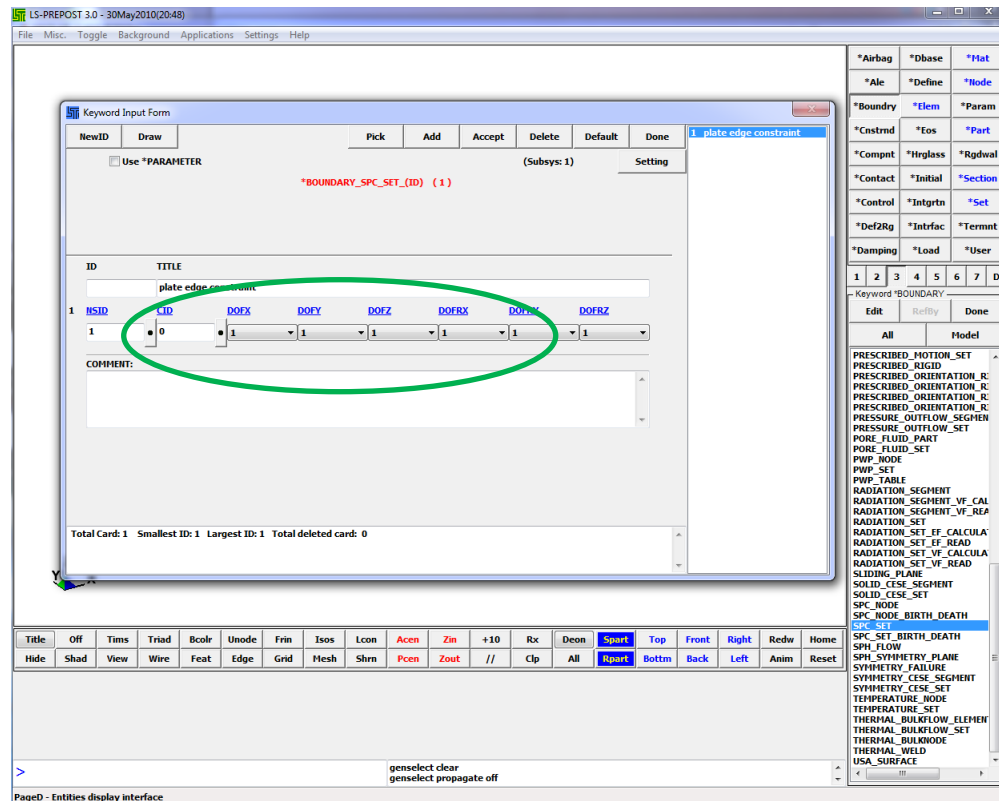
- Go to **page 5** and select **SetD**
- Select **Create** and **\*SET\_NODE**
- Enter **edge nodes** in the Title Box
- On the bottom menu, select **ByEdge**
- Click on **Prop**
- Click along each of the four plate edges so that all the edge nodes are selected
- Click *Apply*
- Click *Done*

\*Note: nodes can be removed by clicking with the right mouse button



# Step 11: Define Boundary Constraint

- Go to **page 3** and select **\*Boundary**
- Select **SPC\_SET** from the list and click *Edit*
- Enter **plate edge constraint** in the Title Box
- Click on the NSID button and select:
  - 1 edge nodes
- Change DOFX, DOFY, DOFZ, DOFRX, DOFRY, and DOFRZ to 1
- Click *Accept*
- Click *Done*



*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User

1	2	3	4	5	6	7	D
---	---	---	---	---	---	---	---

Keyword: BOUNDARY

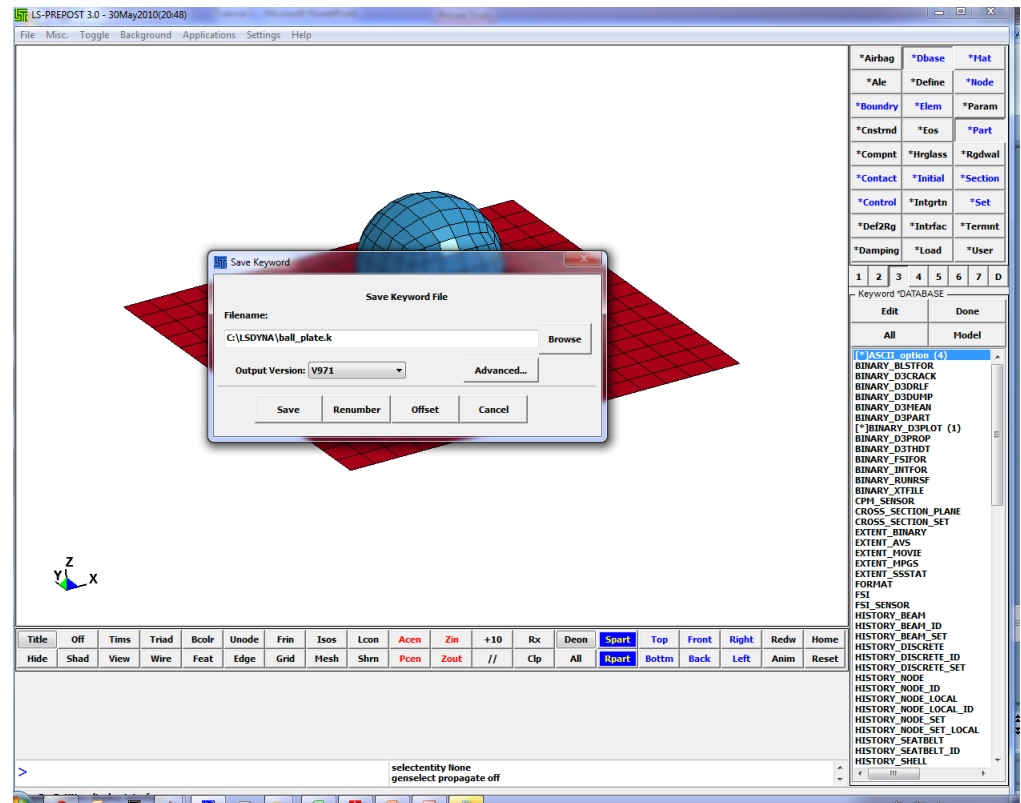
Edit	RefBy	Done
------	-------	------

All	Model
-----	-------

PRESCRIBED\_MOTION\_SET  
 PRESCRIBED\_RIGID  
 PRESCRIBED\_ORIENTATION\_R  
 PRESCRIBED\_ORIENTATION\_R  
 PRESCRIBED\_ORIENTATION\_R  
 PRESSURE\_OUTFLOW\_SEGME  
 PRESSURE\_OUTFLOW\_SET  
 PORE\_FLUID\_PART  
 PORE\_FLUID\_SET  
 PWP\_NODE  
 PWP\_SET  
 PWP\_TABLE  
 RADIATION\_SEGMENT  
 RADIATION\_SEGMENT\_VF\_CAL  
 RADIATION\_SEGMENT\_VF\_REA  
 RADIATION\_SET  
 RADIATION\_SET\_EF\_CALCULA  
 RADIATION\_SET\_EF\_READ  
 RADIATION\_SET\_VF\_CALCULA  
 RADIATION\_SET\_VF\_READ  
 SLIDING\_PLANE  
 SOLID\_CESE\_SEGMENT  
 SOLID\_CESE\_SET  
 SPC\_NODE  
 SPC\_NODE\_BIRTH\_DEATH  
 SPC\_SET  
 SPC\_SET\_BIRTH\_DEATH  
 SPH\_FLOW  
 SPH\_SYMMETRY\_PLANE  
 SYMMETRY\_FAILURE  
 SYMMETRY\_CESE\_SEGMENT  
 SYMMETRY\_CESE\_SET  
 TEMPERATURE\_NODE  
 TEMPERATURE\_SET  
 THERMAL\_BULKFLOW\_ELEME  
 THERMAL\_BULKFLOW\_SET  
 THERMAL\_BULKNODE  
 THERMAL\_WELD  
 USA\_SURFACE

# Step 12: Save File

- File → Save Keyword
- Enter filename as **ball\_plate.k**
- Click *Save*
- You can exit Prepost now

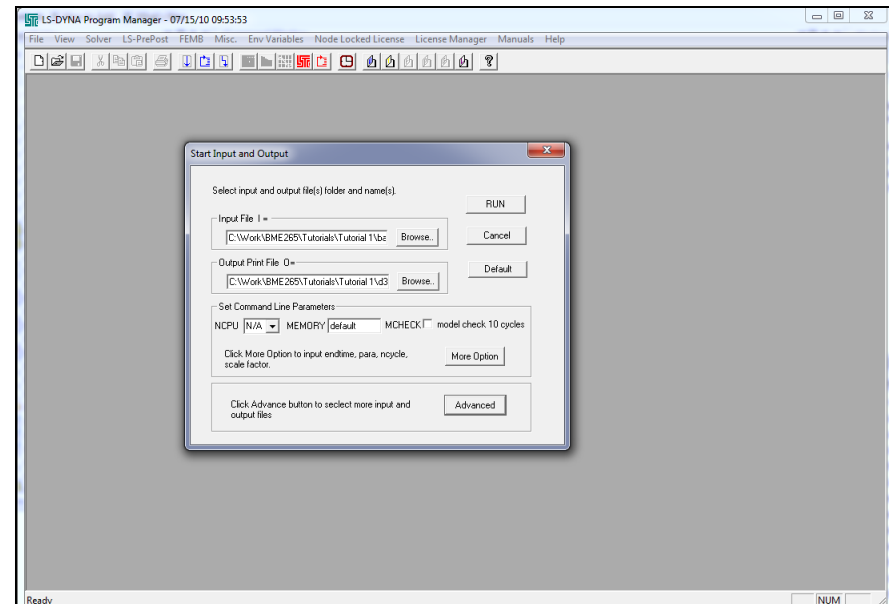


# Step 13: Run Simulation

- In the LS-DYNA Program Manager, select Solver → Start LS-DYNA Analysis
- Click on the first Browse button and locate **ball\_plate.k**
- Click *RUN*
- A command window will appear showing you the simulation process
- When the simulation is finished, you should see:

## Normal termination

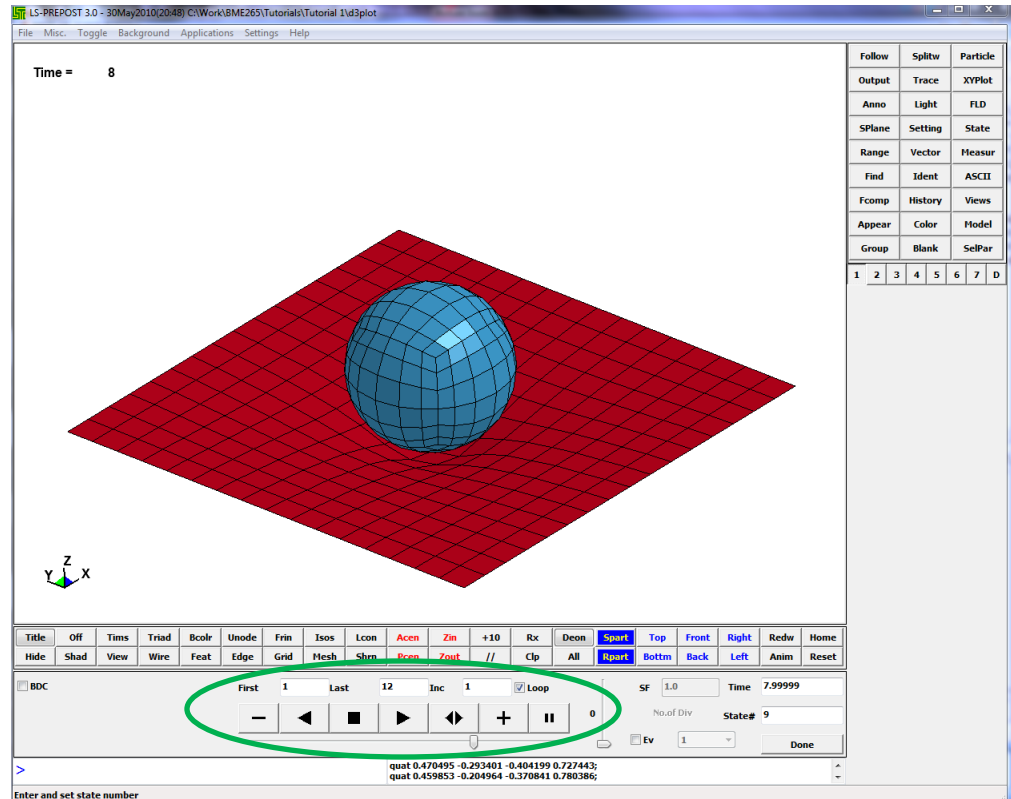
- If you see  
**Error termination**  
then there was an error in your model  
and the simulation did not finish





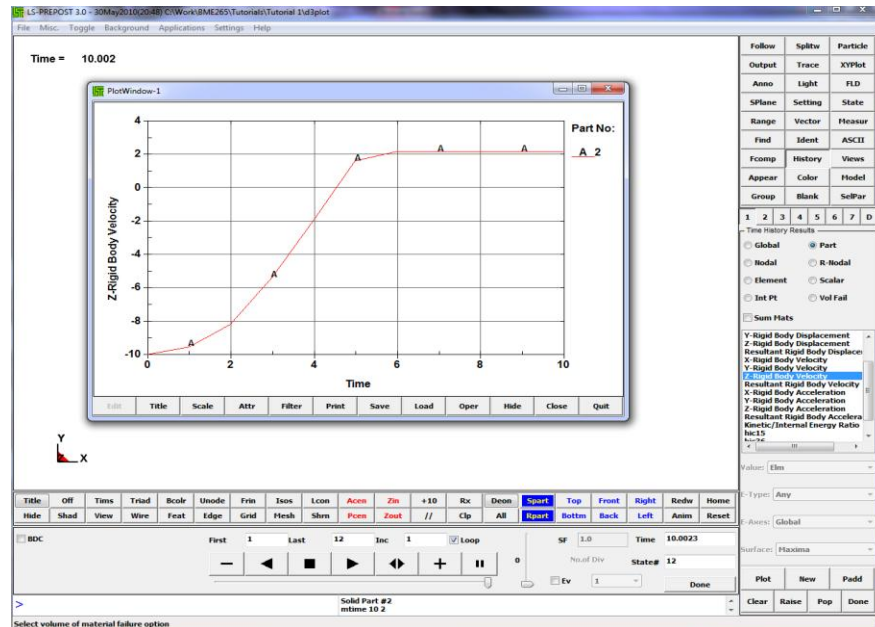
# Step 14: Post-Processing (Animation)

- If the simulation ran correctly, start up LS-Prepost again
- Go to File → Open → LS-DYNA Binary Plot
- Find the file **d3plot** that is in the same directory as the model **k** file you just ran
- You can run animation of the simulation using the **animation control panel** in the bottom menu



# Step 15a: Post-Processing (Rigid Body)

- Go to **page 1**, select Setting
- Select Hic/CSI const. and set the Time units to msec and Gravity Constant to 0.00981 and click *Apply*
- In **page 1**, select **SelPar**
- Select H2, which should only leave the ball model showing
- In **page 1**, select **History**, and choose **Part**
- Select **Z-Rigid Body Velocity** from the list
- Click on the model of the ball
- Click on *Plot*
- Click *Quit* to close plot



Follow	Splitw	Particle
Output	Trace	XYPlot
Anno	Light	FLD
SPlane	Setting	State
Range	Vector	Measur
Find	Ident	ASCII
Fcomp	History	Views
Appear	Color	Model
Group	Blank	SelPar

1	2	3	4	5	6	7	D
---	---	---	---	---	---	---	---

Time History Results	
<input type="radio"/> Global	<input checked="" type="radio"/> Part
<input type="radio"/> Nodal	<input type="radio"/> R-Nodal
<input type="radio"/> Element	<input type="radio"/> Scalar
<input type="radio"/> Int Pt	<input type="radio"/> Vol Fail
<input type="checkbox"/> Sum Mats	
Y-Rigid Body Displacement Z-Rigid Body Displacement Resultant Rigid Body Displacement X-Rigid Body Velocity Y-Rigid Body Velocity <b>Z-Rigid Body Velocity</b> Resultant Rigid Body Velocity X-Rigid Body Acceleration Y-Rigid Body Acceleration Z-Rigid Body Acceleration Resultant Rigid Body Acceleration Kinetic/Internal Energy Ratio hic15 hic26	
Value:	Elm
E-Type:	Any
E-Axes:	Global
Surface:	Maxima

Plot	New	Padd
Clear	Raise	Pop
Done		

# Step 15b: Post-Processing (Rigid Body)

Follow	Splitw	Particle
Output	Trace	XYPlot
Anno	Light	FLD
SPlane	Setting	State
Range	Vector	Measur
Find	Ident	ASCII
Fcomp	History	Views
Appear	Color	Model
Group	Blank	SelfPar

1	2	3	4	5	6	7	D
---	---	---	---	---	---	---	---

Time History Results

☐ Global
 ☒ Part
 ☐ Nodal
 ☐ R-Nodal
 ☐ Element
 ☐ Scalar
 ☐ Int Pt
 ☐ Vol Fail

☐ Sum Mats

Y-Rigid Body Displacement  
 Z-Rigid Body Displacement  
 Resultant Rigid Body Displacement  
 X-Rigid Body Velocity  
 Y-Rigid Body Velocity  
 Z-Rigid Body Velocity  
 Resultant Rigid Body Velocity  
 X-Rigid Body Acceleration  
 Y-Rigid Body Acceleration  
 Z-Rigid Body Acceleration  
 Resultant Rigid Body Acceleration  
 Kinetic/Internal Energy Ratio  
 hic15

Value: Elm

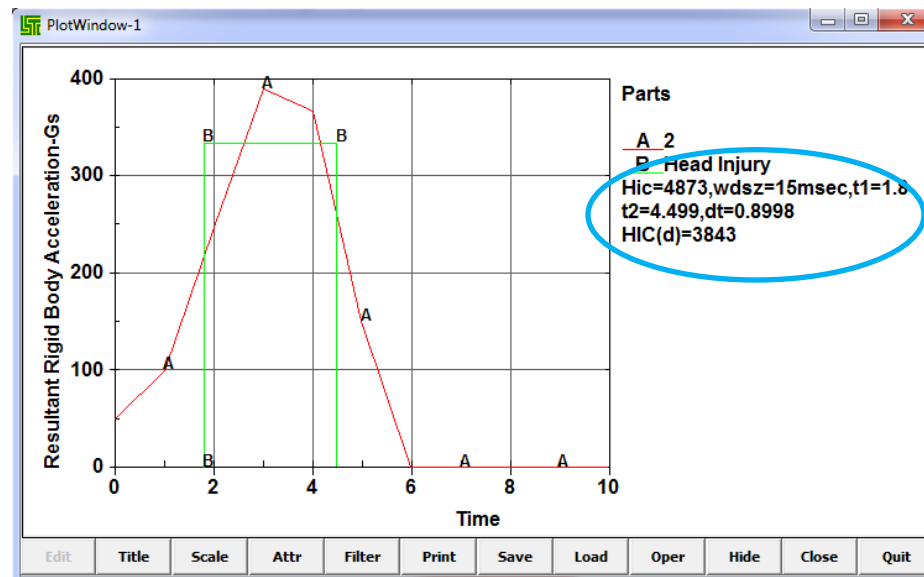
E-Type: Any

E-Axes: Global

Surface: Maxima

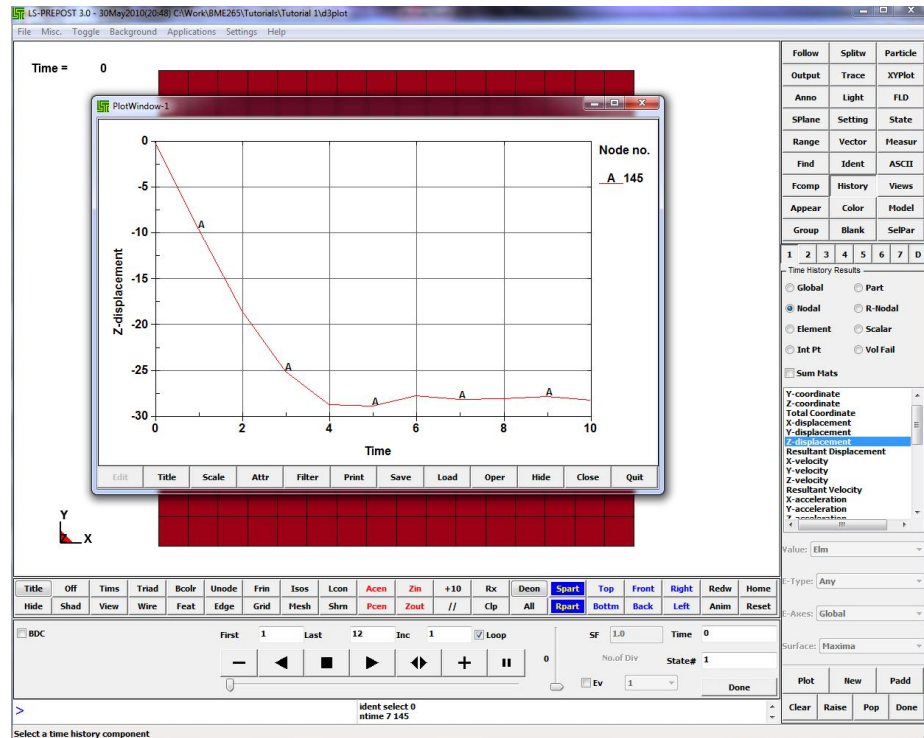
Plot New Padd  
 Clear Raise Pop Done

- With part 2 still selected, select **hic15** from the list
- hic15 calculates the HIC value with a maximum time window of 15 ms
- Click on *Plot*
- The two curves shown are the **resultant acceleration**, and the **time window** used for maximum HIC
- Information on the **HIC value** and the **time window** is beside the plot
- NOTE: This HIC is sensitive to the time step of the acceleration plot. Decrease the output time step in Step 9 to DT = 0.1 for a more accurate calculation.
- Click *Quit* to close plot



# Step 16: Post-Processing (Nodal)

- In **page 1**, select **SelPar**
- Select S1, which should only leave the plate model showing
- In **page 1**, select **History**, and choose **Nodal**
- Select **Z-displacement** from the list
- Click on the center node of the plate
  - It should indicate that it is selected
- Click on **Plot**
- Click **Quit** to close plot



Follow	Splitw	Particle
Output	Trace	XYPlot
Anno	Light	FLD
SPlane	Setting	State
Range	Vector	Measur
Find	Ident	ASCII
Fcomp	History	Views
Appear	Color	Model
Group	Blank	SelPar

1	2	3	4	5	6	7	D
---	---	---	---	---	---	---	---

Time History Results

☐ Global
 ☐ Part
 ☒ Nodal
 ☐ R-Nodal

☐ Element
 ☐ Scalar
 ☐ Int Pt
 ☐ Vol Fail

☐ Sum Mats

Y-coordinate  
 Z-coordinate  
 Total Coordinate  
 X-displacement  
 Y-displacement  
**Z-displacement**  
 Resultant Displacement  
 X-velocity  
 Y-velocity  
 Z-velocity  
 Resultant Velocity  
 X-acceleration  
 Y-acceleration  
 Z-acceleration

Value: Elm

E-Type: Any

E-Axes: Global

Surface: Maxima

Plot	New	Padd
Clear	Raise	Pop
Done		

- In **page 1**, select **Fcomp**
- Select **Stress**, and choose **plastic strain** from the list
- In the **animation control panel**, set the current state to #12
  - This should be for time = 10

