LS-Dyna/Pre-post Tutorial 1

Ball and Plate Impact

Introduction

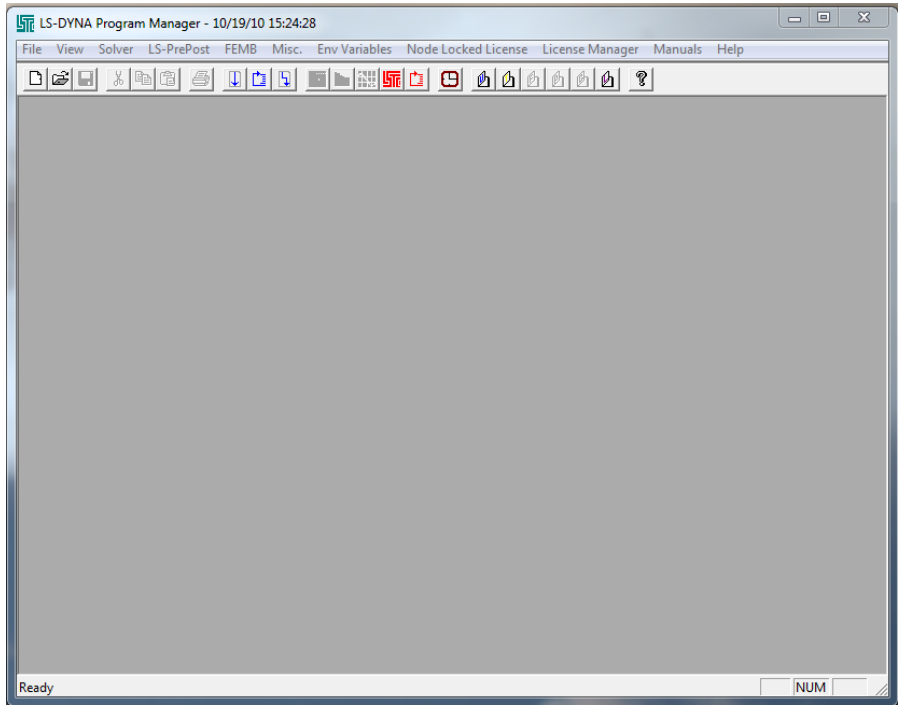
- This tutorial is an introduction LS-DYNA modeling using LS-Prepost
- In this tutorial, you will learn to:
  - Mesh a 2D plate and a 3D sphere
  - Assign section and materials properties to parts
  - Create node sets
  - Apply boundary constraints to node sets
  - Apply initial velocity conditions
  - 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

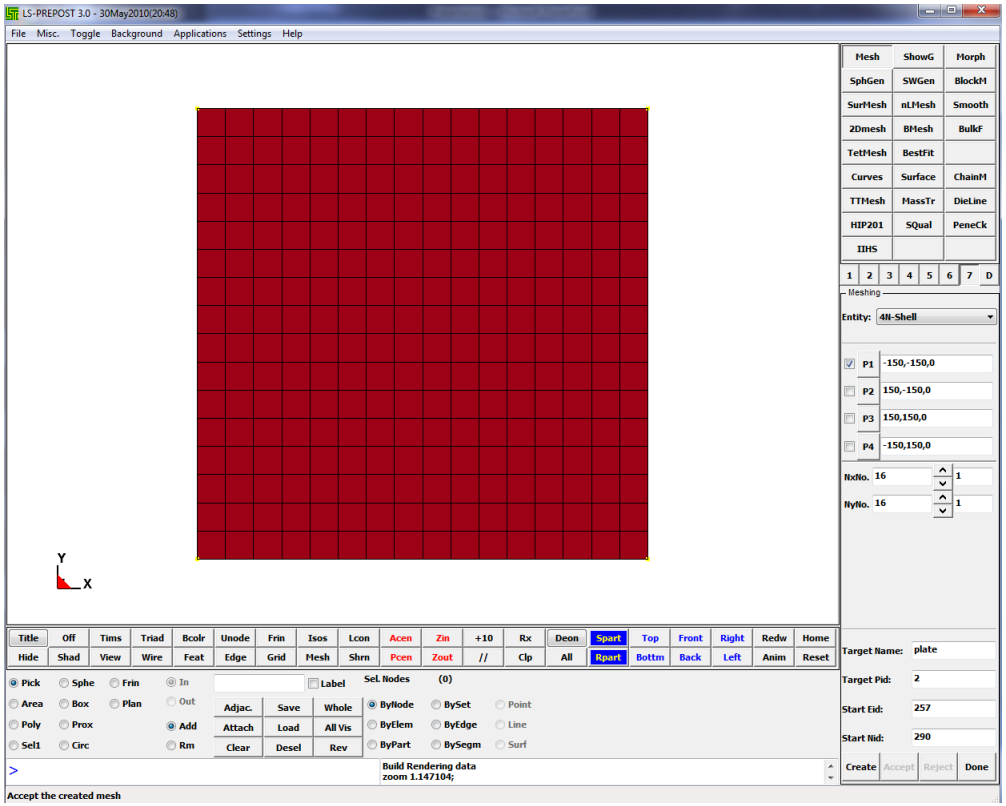
- From the Start Menu, select the LS-DYNA Manager
- Click on the LS-PrePost button on themenu bar and select “Start LS-PrePost”.
- Depending on the version you are running, you may need to press the F11 key to toggle to the old style menu that is used in this tutorial.
- The LS-PrePost application will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

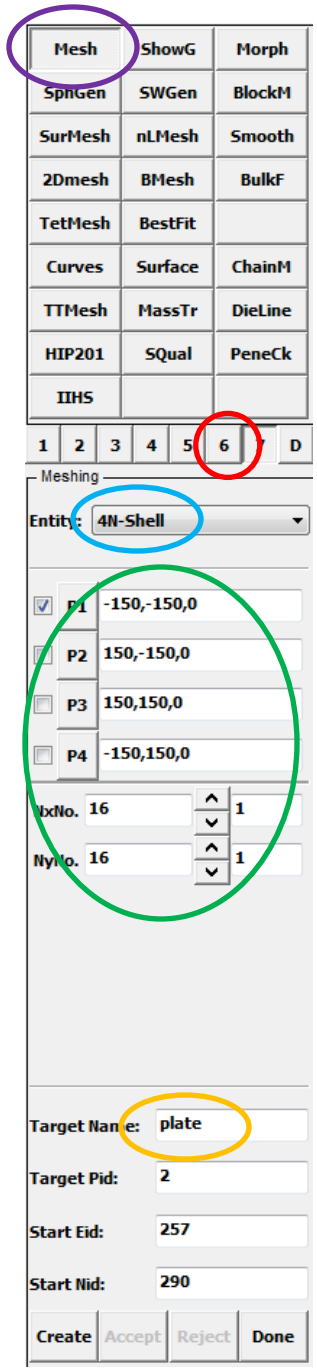
In this tutorial you will model an rigid ball impacting an elastic-plastic plate.



Step 1: Create Plate Mesh

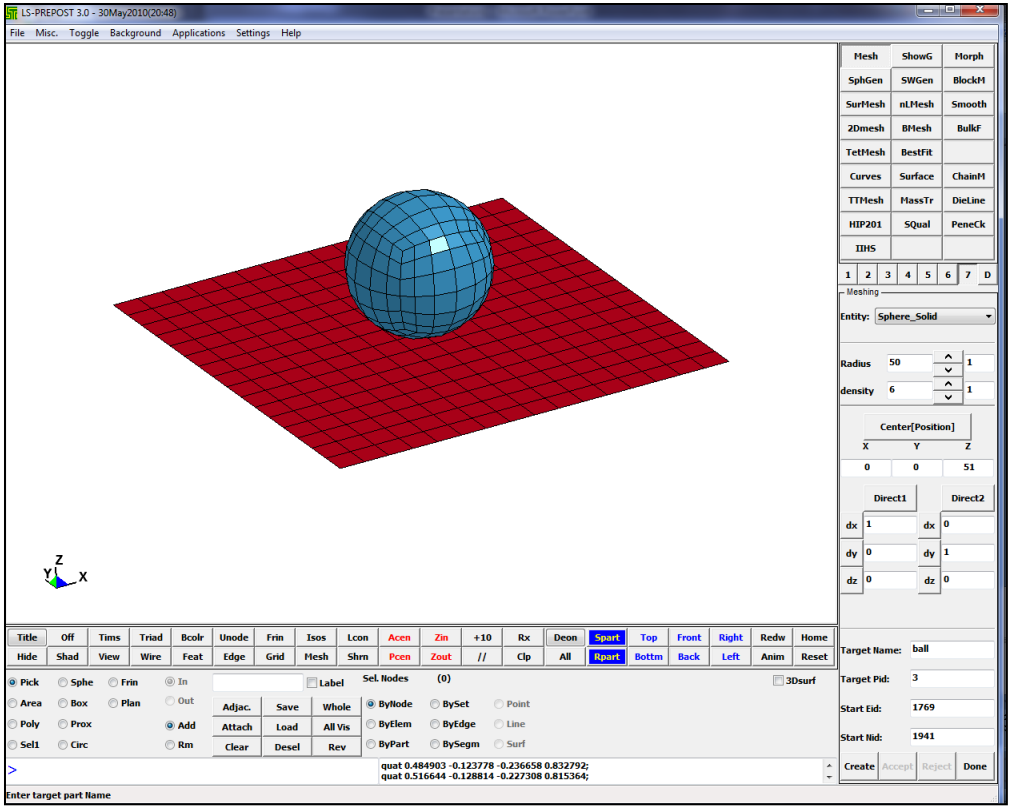
- Go to page 6 and select Mesh
- In the Entity menu, select 4N-Shell
- Enter the coords of the plate corners:
  - P1: -150,-150,0
  - P2: 150,-150,0
  - P3: 150,150,0
  - P4: -150,150,0
- Enter the # of elements in each dimension
  - NxNo: 16
  - NyNo: 16
- Enter Target Name
  - plate
- Click *Create*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

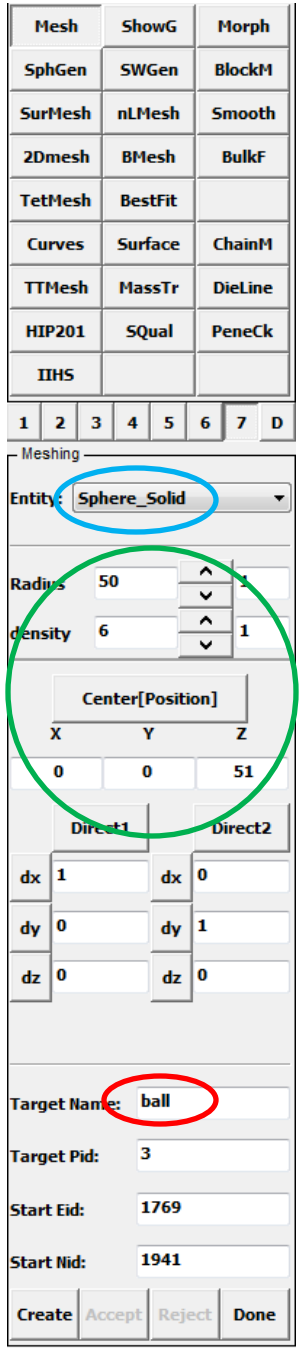




Step 2: Create Ball Mesh

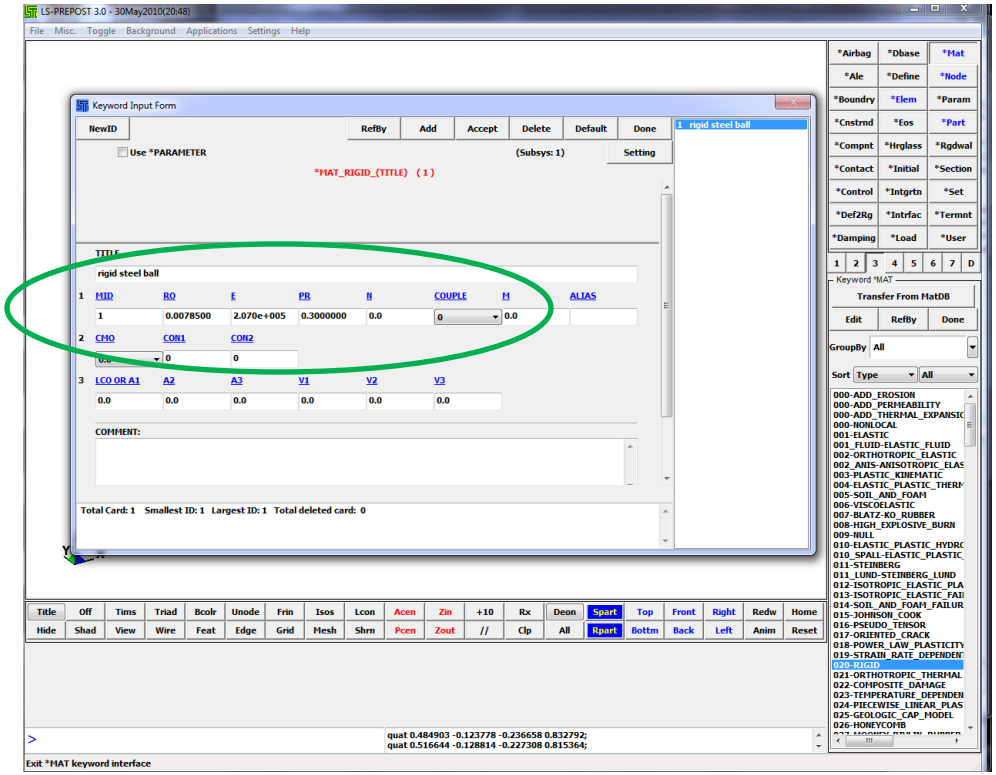
- In the Entity menu, select Sphere_Solid
- Enter the ball radius and element density:
  - Radius: 50
  - Density: 6
- Enter the center location
  - X: 0
  - Y: 0
  - Z: 51
- Enter Target Name
  - ball
- Click *Create*
- Click *Accept*
- Click *Done*

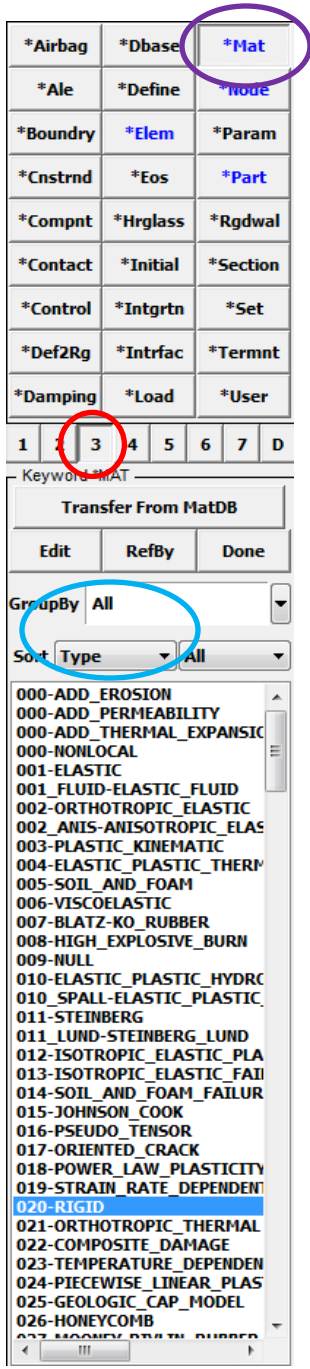




Step 3: Define Materials

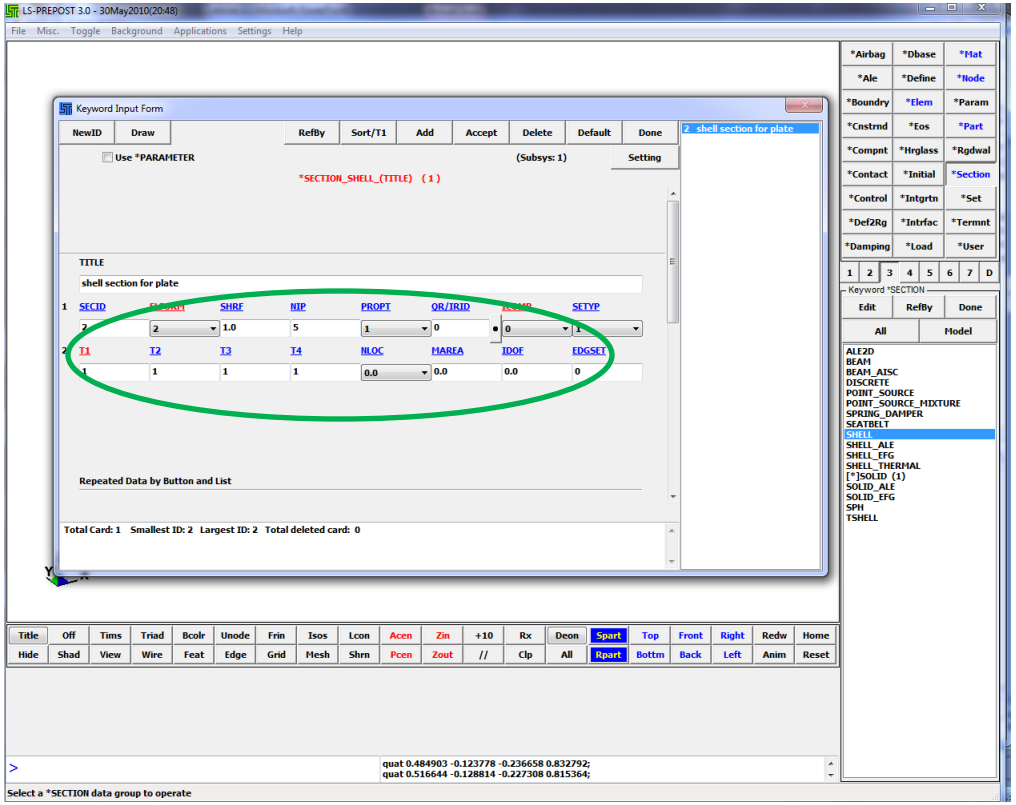
- Go to page 3 and select *Mat
- Select GroupBy: All, sort by Type
- Select 020-Rigid and click *Edit*
- Click *NewID* in the pop-up, and enter:
  - TITLE: rigid steel ball
  - RO: 7.85E-3
  - E: 207E+3
  - PR: 0.3
- Click *Accept*
- Select 024-PIECEWISE LINEAR_PLASTICITY and click *Edit*
- Click *NewID*, and enter
  - TITLE: deformable plate
  - RO: 7.85E-3
  - E: 207E+3
  - PR: 0.3
  - SIGY: 200
  - ETAN:2E+3
- Click *Accept*
- Click *Done*

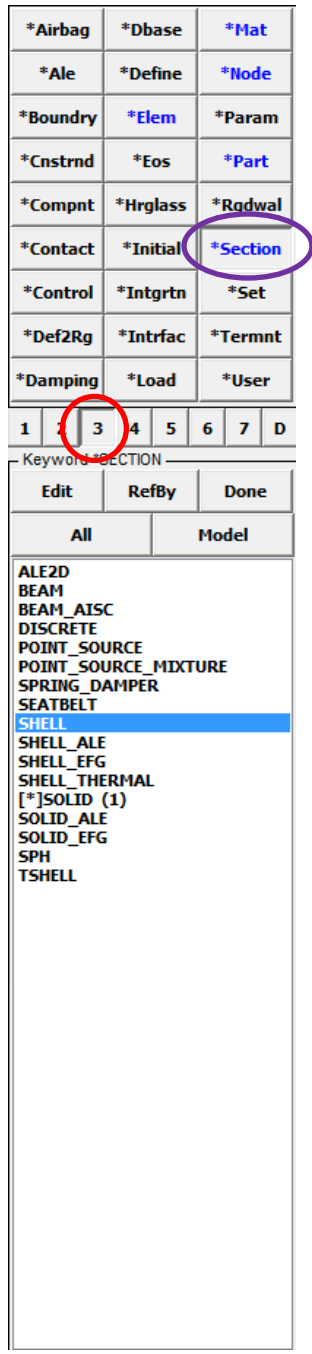




Step 4: Define Section Properties

- 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*

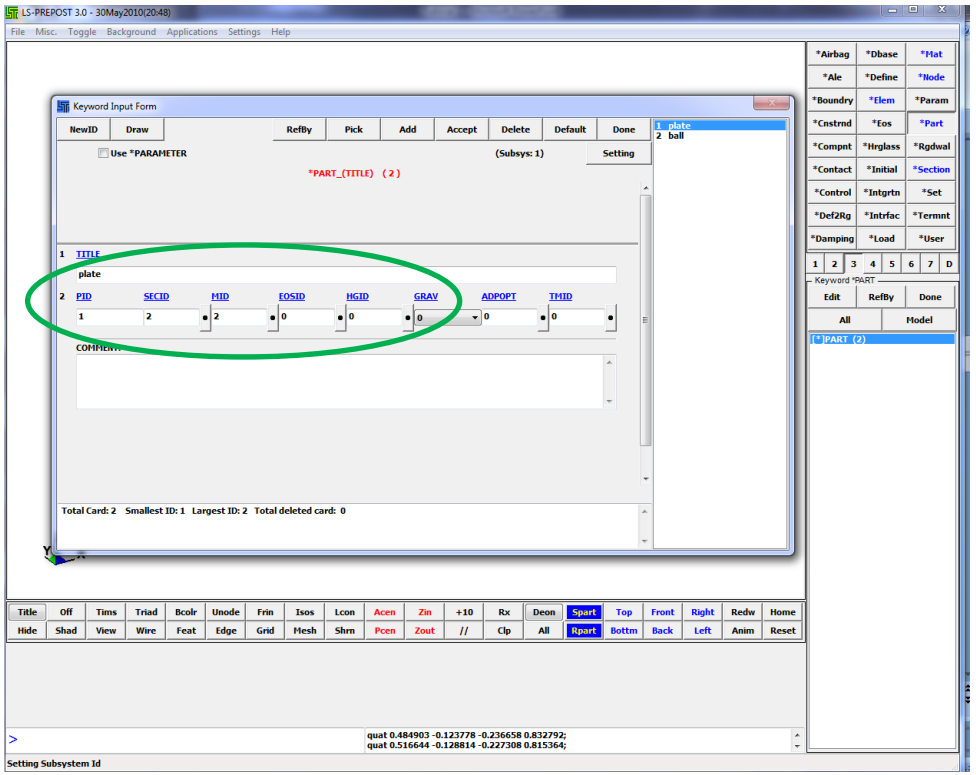




*Note: T2, T3, T4 are automatically filled in

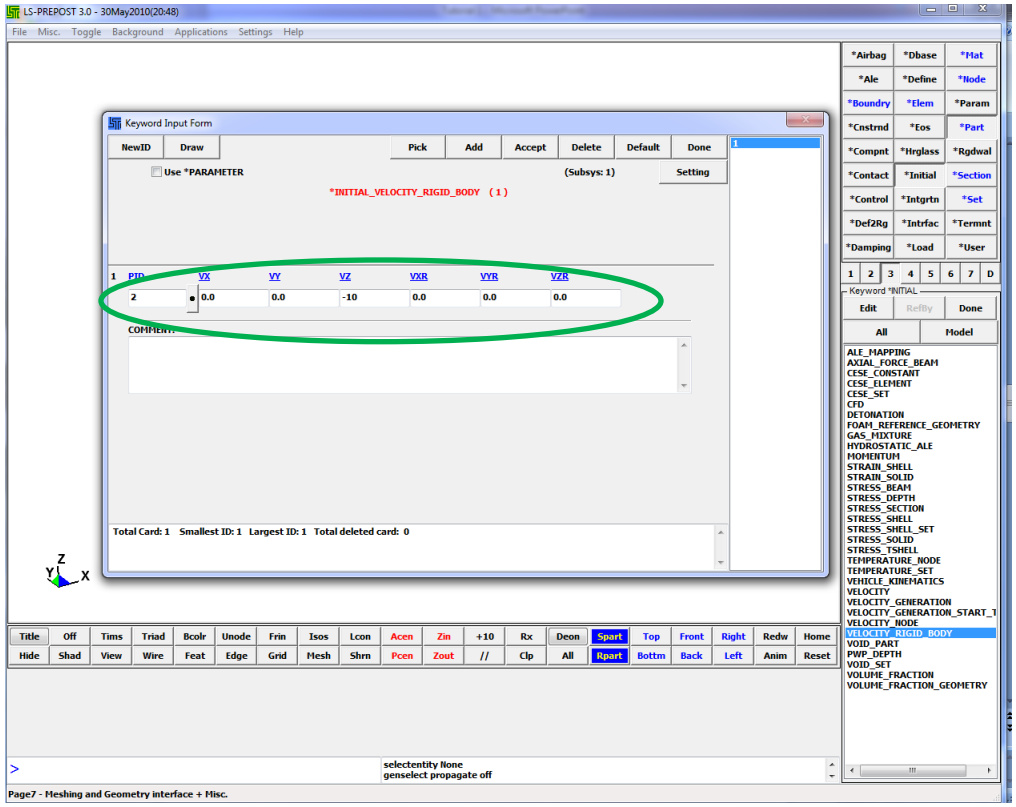
Step 5: Define Parts

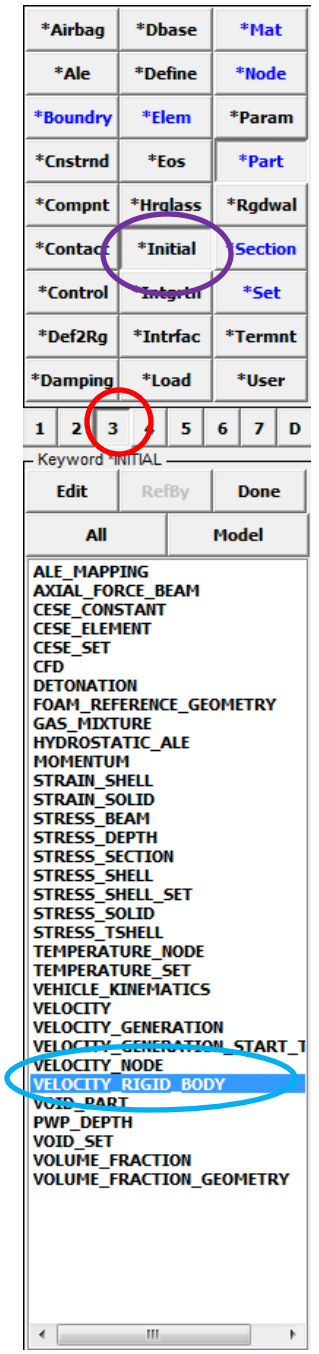
- In page 3, select *Part
- Click on [*]PART (2) and select *Edit*
- Select Part 1 on the list in the pop-up, and enter:
  - TITLE: plate
- Click on the SECID button and select:
  - 2 plate section
- Click on the MID button and select:
  - 2 deformable plate
- Click *Accept*
- Select Part 2 and enter: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



Step 6: Define Initial Velocity

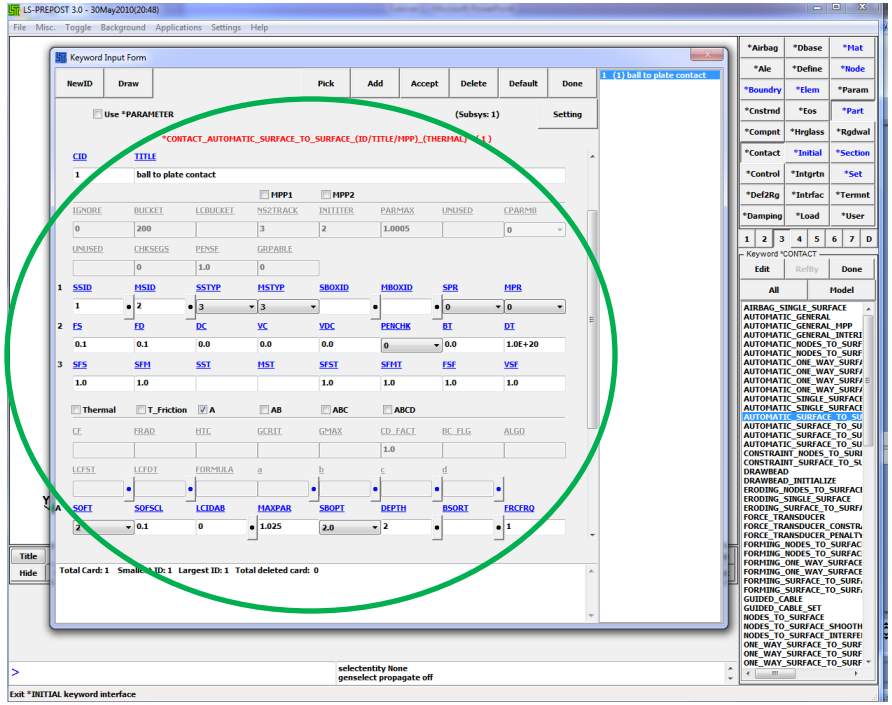
- In page 3, select *Initial
- Select VELOCITY_RIGID_BODY from the list and click *Edit*
- In the pop-up, click on the PID button and select:
  - 2 ball
- Select VZ and enter:
  - VZ: -10
- Click *Accept*
- Click *Done*

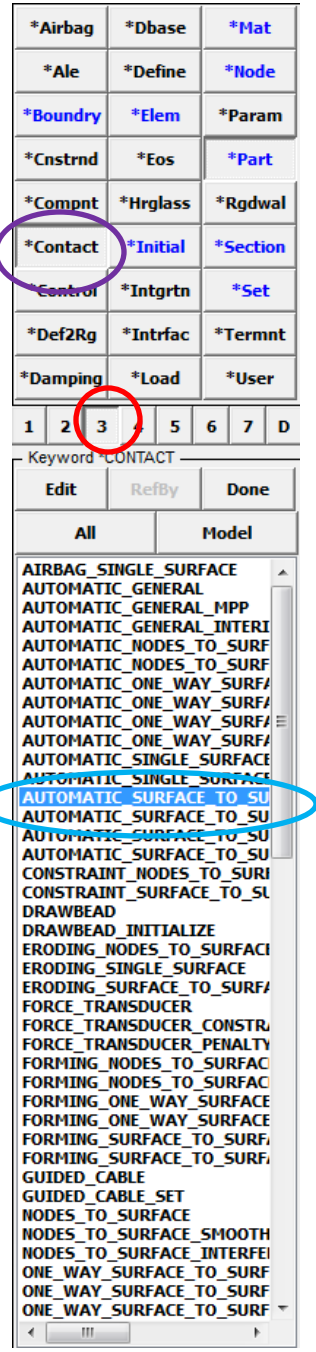




Step 7: Define Contact

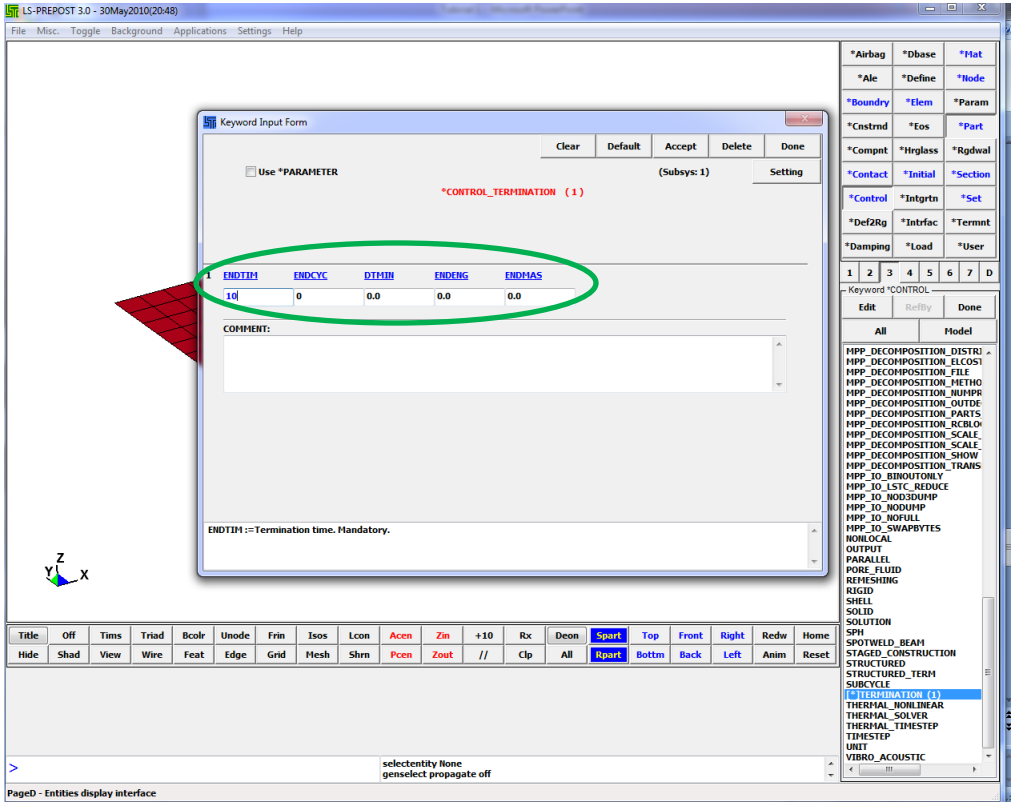
- In page 3, select *Contact
- Select AUTOMATIC_SURFACE_TO_SURFACE from the list and click *Edit*
- Click *NewID* in the pop-up, and enter:
  - TITLE: ball to plate contact
- Set SSTYP and MSTYP to 3
- Click on the SSID button and select:
  - 1 plate
- Click MSID and select:
  - 2 ball
- Enter the following: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





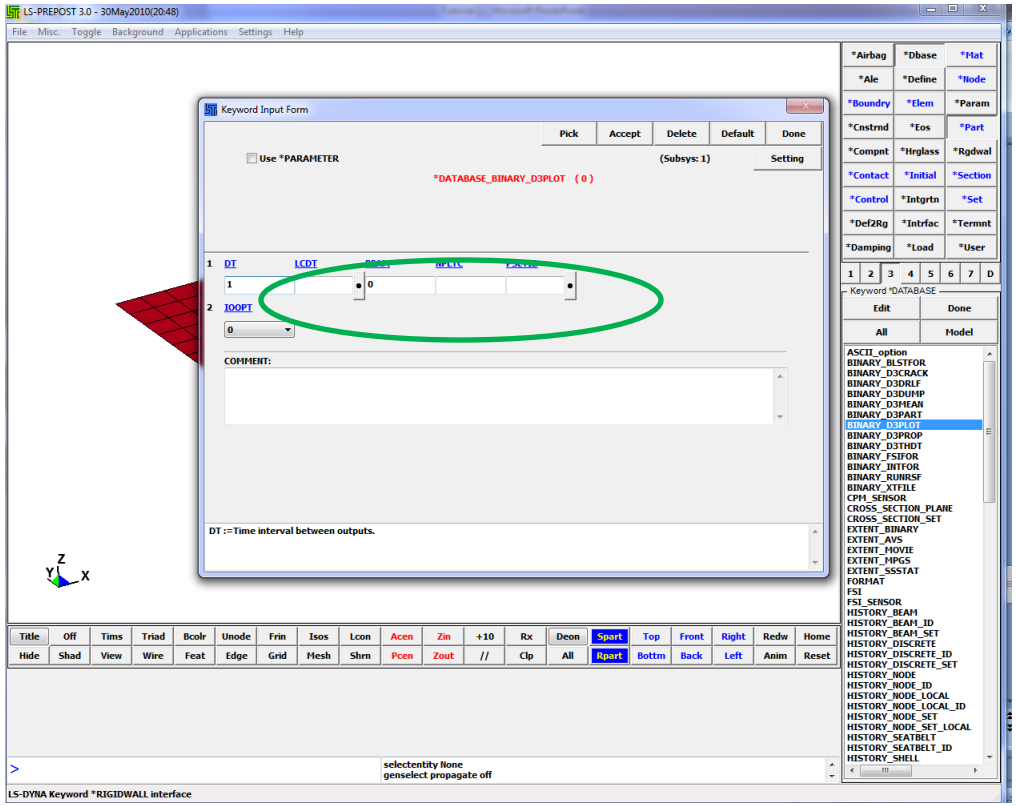
Step 8: Set End Time

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



Step 9: Set Output Frequency

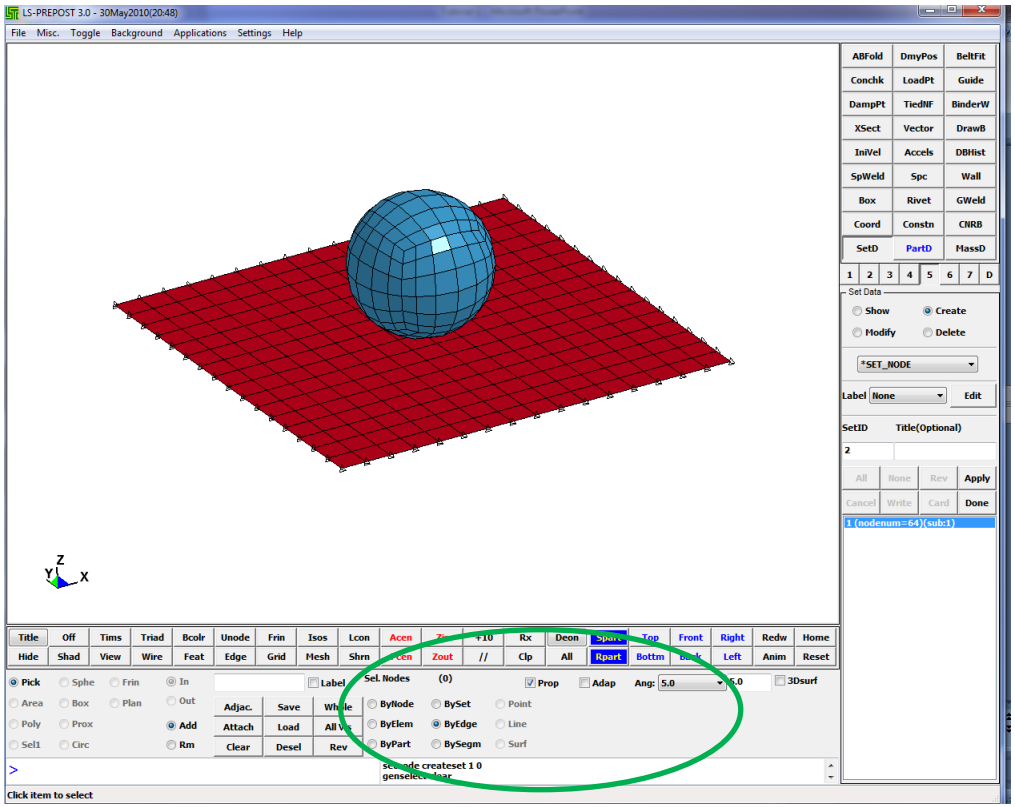
- In page 3, select *Dbase
- Select BINARY_D3PLOT from the list and click *Edit*
- Enter:
  - DT: 1
- Click *Accept*
- Click *Done*



Step 10: Define Edge Nodes Set

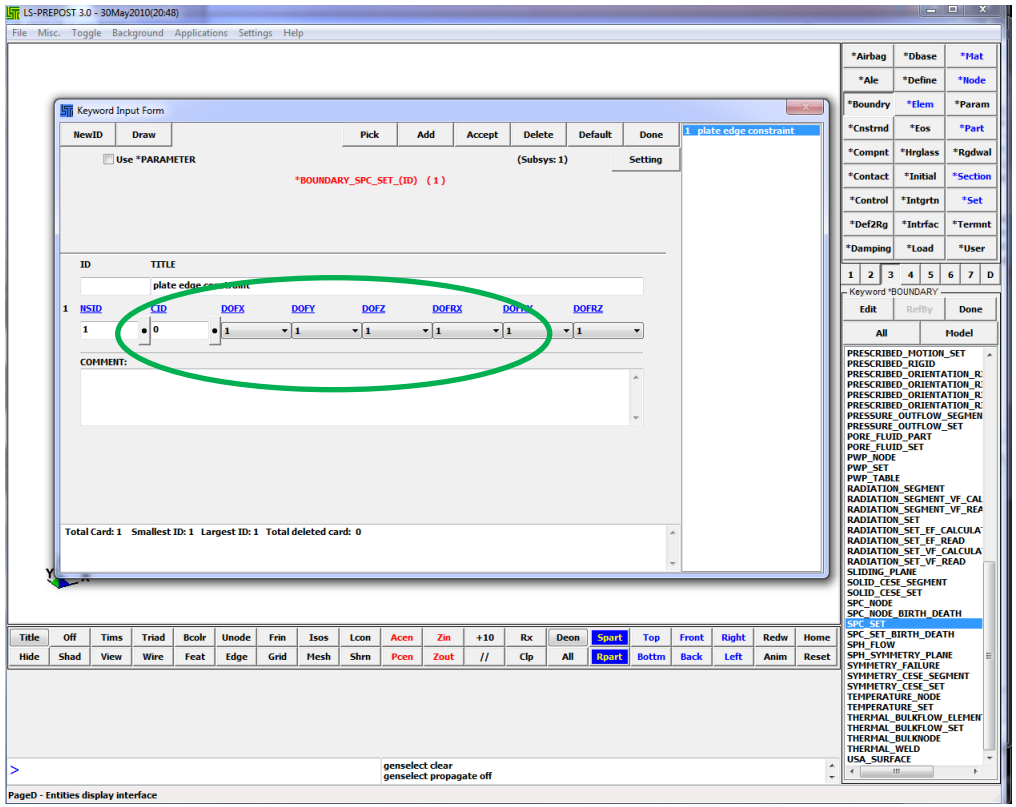
- Go to page 5 and select SetD
- Select Create and *SET_NODE
- Enter edge nodes in the Title Box
- On the bottom menu, select ByEdge
- Click on Prop
- Click along each of the four plate edges so that all the edge nodes are selected
- Click *Apply*
- Click *Done*

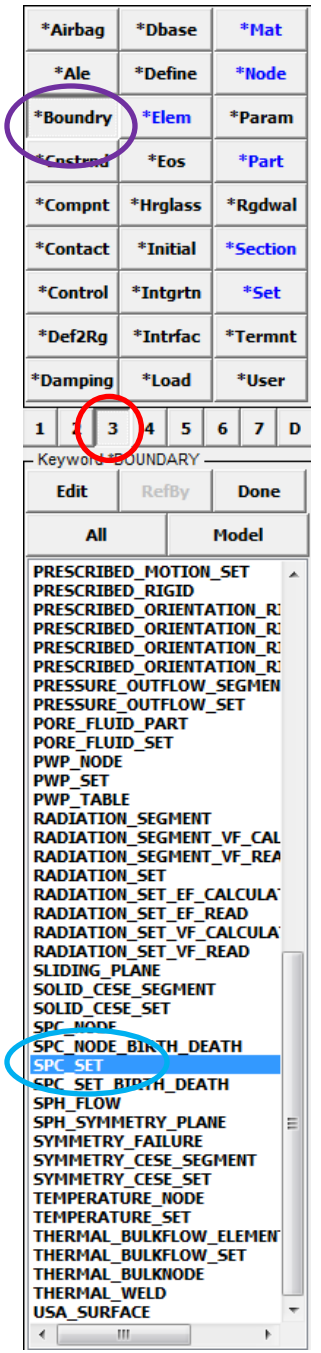
*Note: nodes can be removed by clicking with the right mouse button



Step 11: Define Boundary Constraint

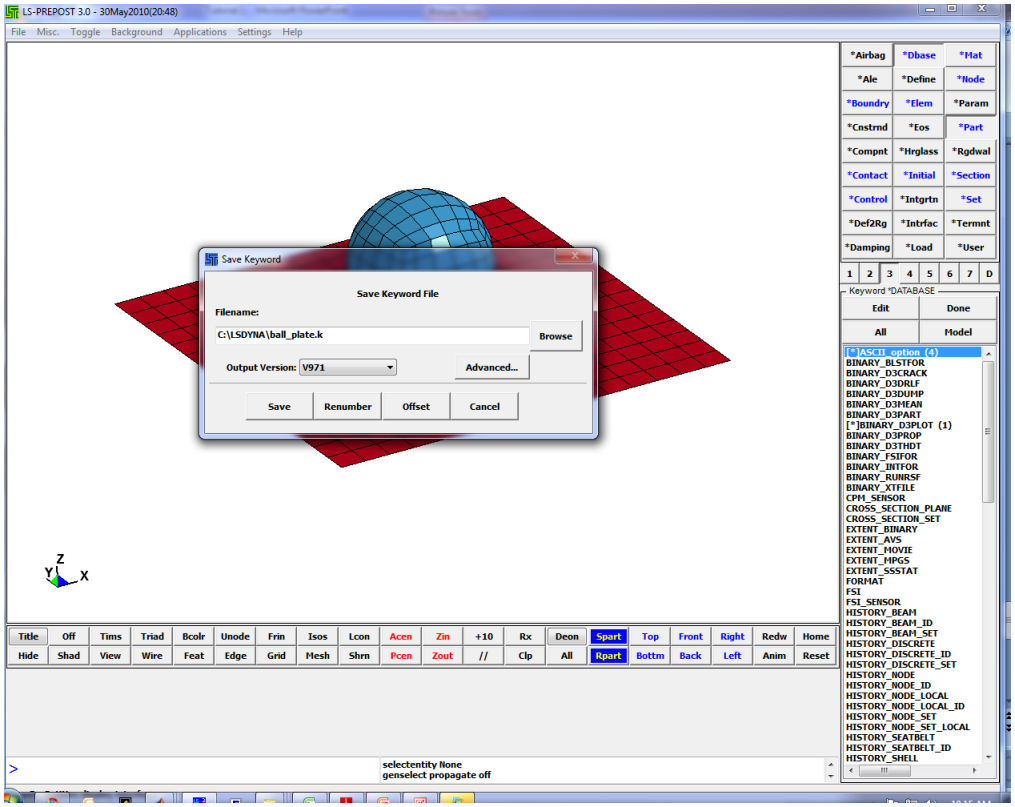
- Go to page 3 and select *Boundary
- Select SPC_SET from the list and click *Edit*
- Enter plate edge constraint in the Title Box
- Click on the NSID button and select:
  - 1 edge nodes
- Change DOFX, DOFY, DOFX, DOFRX, DOFRY, and DOFRZ to 1
- Click *Accept*
- Click *Done*





Step 12: Save File

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



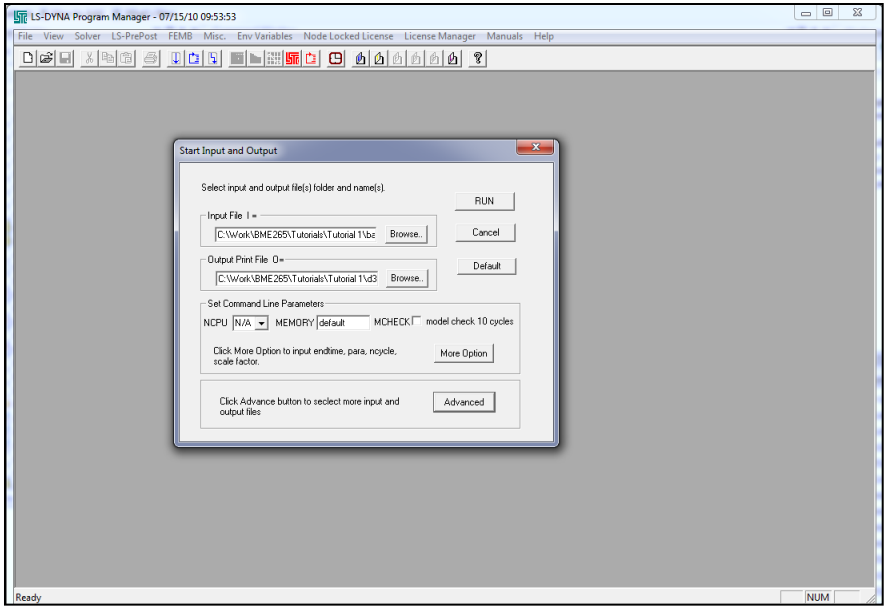
Step 13: Run Simulation

- 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

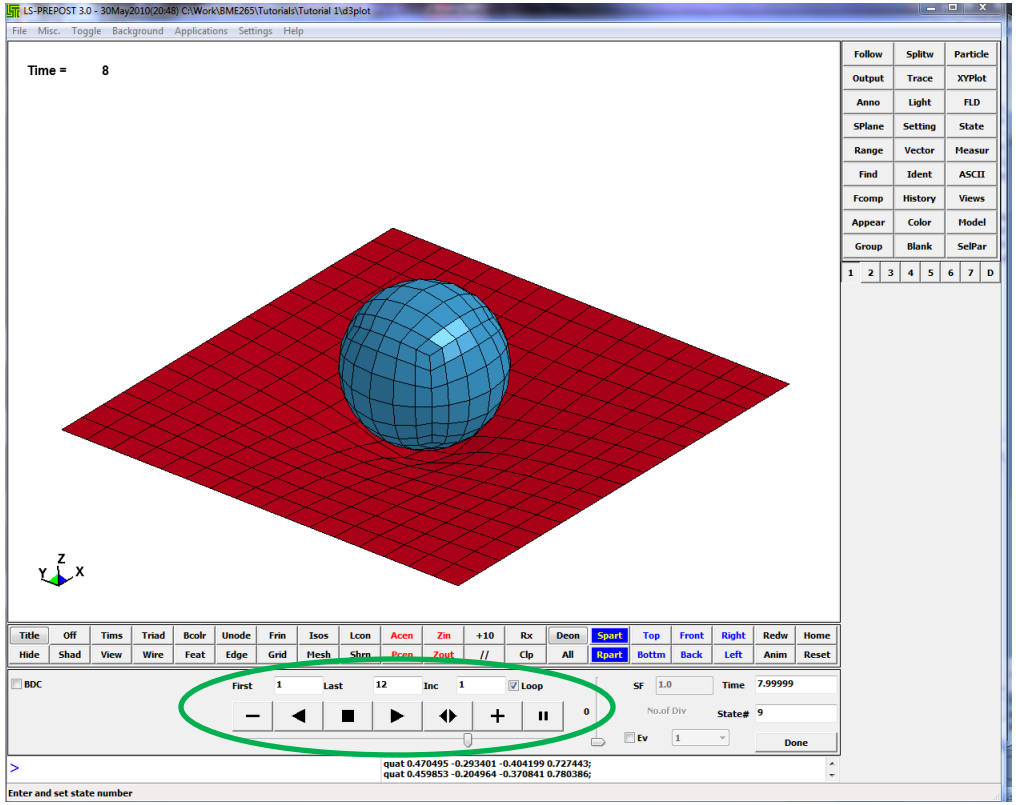
  Error termination

  then there was an error in your model and the simulation did not finish



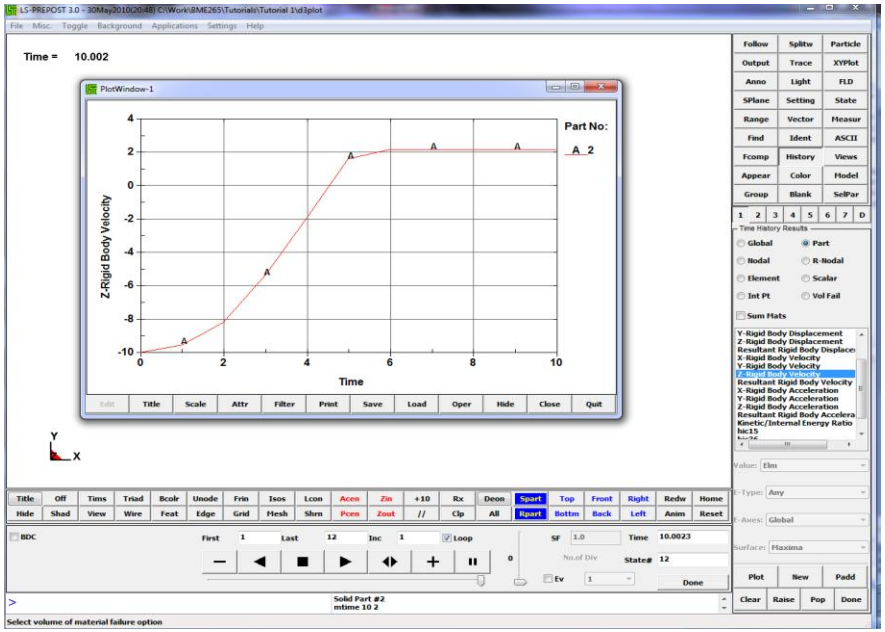
Step 14: Post-Processing (Animation)

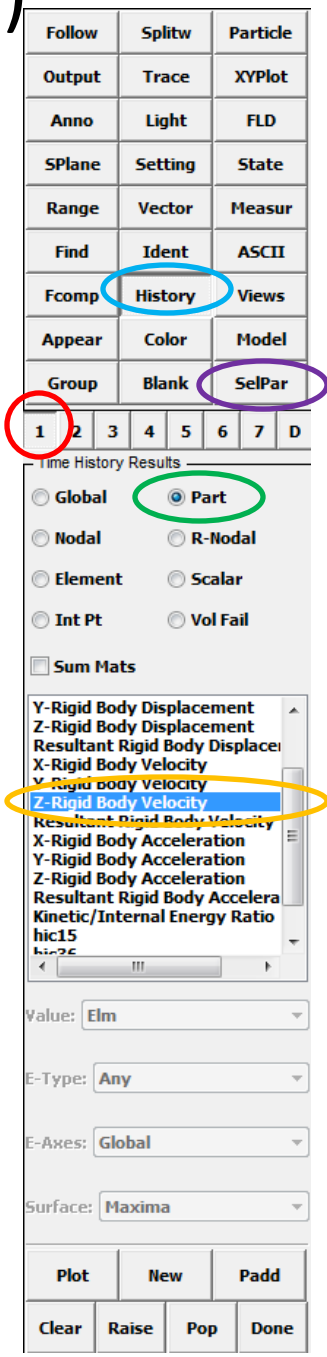
- If the simulation ran correctly, start up LS-Prepost again
- Go to File → Open → LS-DYNA Binary Plot
- Find the file d3plot that is in the same directory as the model k file you just ran
- You can run animation of the simulation using the animation control panel in the bottom menu



Step 15a: Post-Processing (Rigid Body)

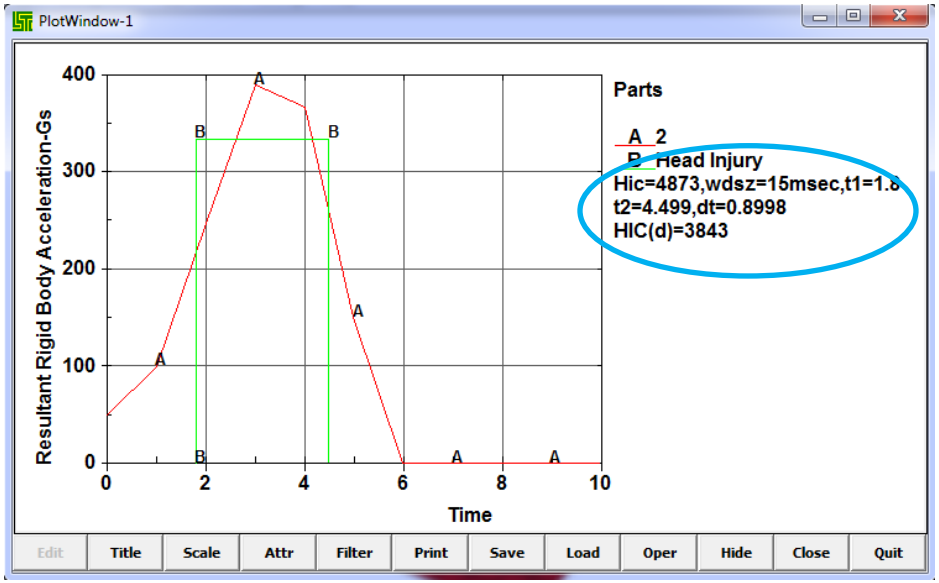
- Go to page 1, select Setting
- Select Hic/CSI const. and set the Time units to msec and Gravity Constant to 0.00981 and click *Aply*
- In page 1, select SelPar
- Select H2, which should only leave the ball model showing
- In page 1, select History, and choose Part
- Select Z-Rigid Body Velocity from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

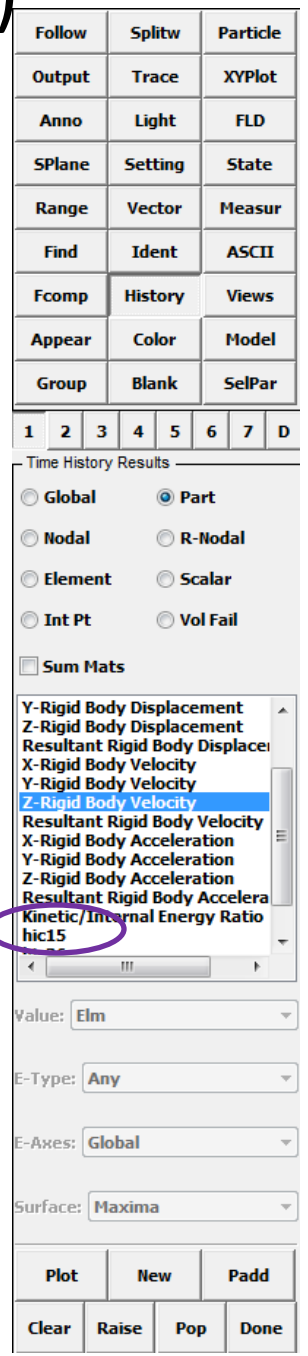




Step 15b: Post-Processing (Rigid Body)

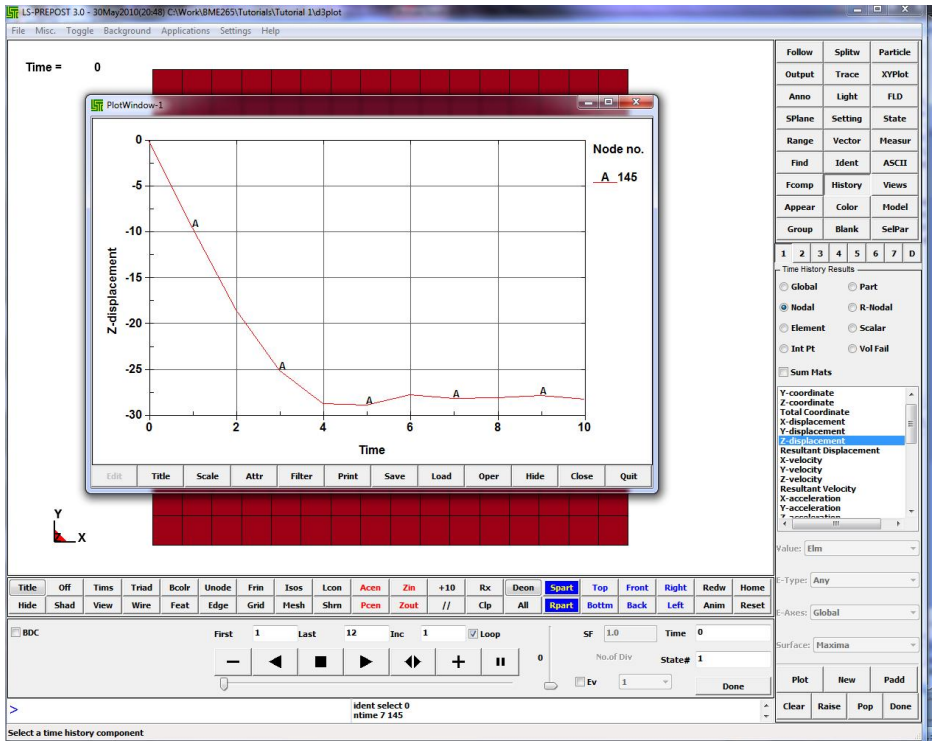
- With part 2 still selected, select hic15 from the list
- hic15 calculates the HIC value with a maximum time window of 15 ms
- Click on *Plot*
- The two curves shown are the resultant acceleration, and the time window used for maximum HIC
- Information on the HIC value and the time window is beside 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

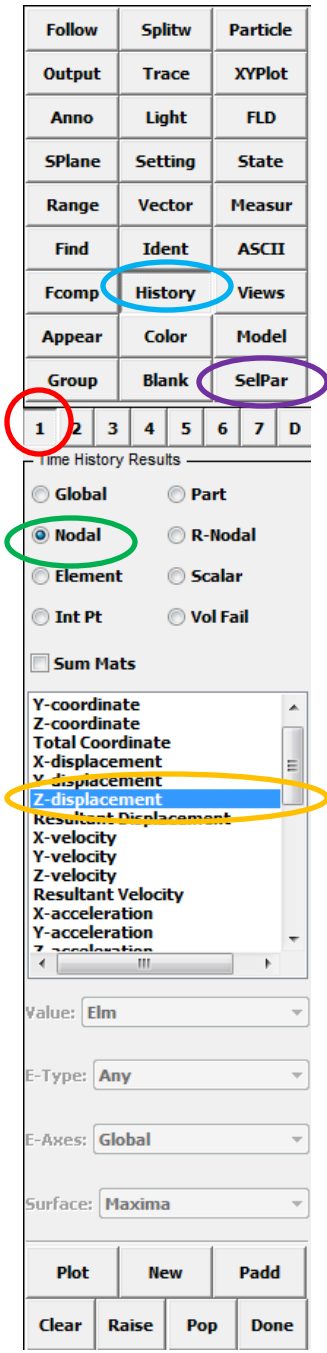




Step 16: Post-Processing (Nodal)

- 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





Step 17: Post-Processing (Fringe Plot)

- 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