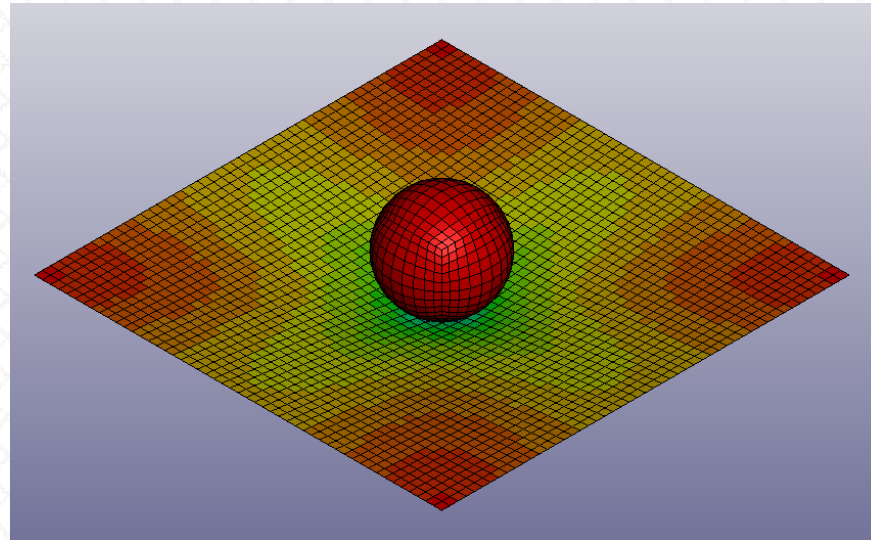


# Introduction to Ansys LS-DYNA Tutorial #1 Ball Impacting a Plate



Bronislav Piak, Simulation Engineer  
PADT, Inc.



We Make Innovation Work  
[www.padtinc.com](http://www.padtinc.com)

# Prerequisites

- Ansys LS-Dyna installed as part of Ansys 2020R2 package (or later release)
- LS-PrePost installed as part of Ansys 2020R2 package (or later release)
- LS-Run installed as part of Ansys 2020R2 package (or later release)



# Tutorial #1

We will model a ball impacting a plate:

## Plate

Dimensions: 200mm x 200mm x 0.1mm

Material: Steel

density  $\rho = 7.83 \times 10^{-6}$  kg/mm<sup>3</sup>

Elastic Modulus  $E = 207$  GPa

Poisson's ratio  $\nu = 0.3$

## Ball

Dimensions : radius = 25mm

Material: Steel

density  $\rho = 7.83 \times 10^{-6}$  kg/mm<sup>3</sup>

Elastic Modulus  $E = 207$  GPa

Poisson's ratio  $\nu = 0.3$

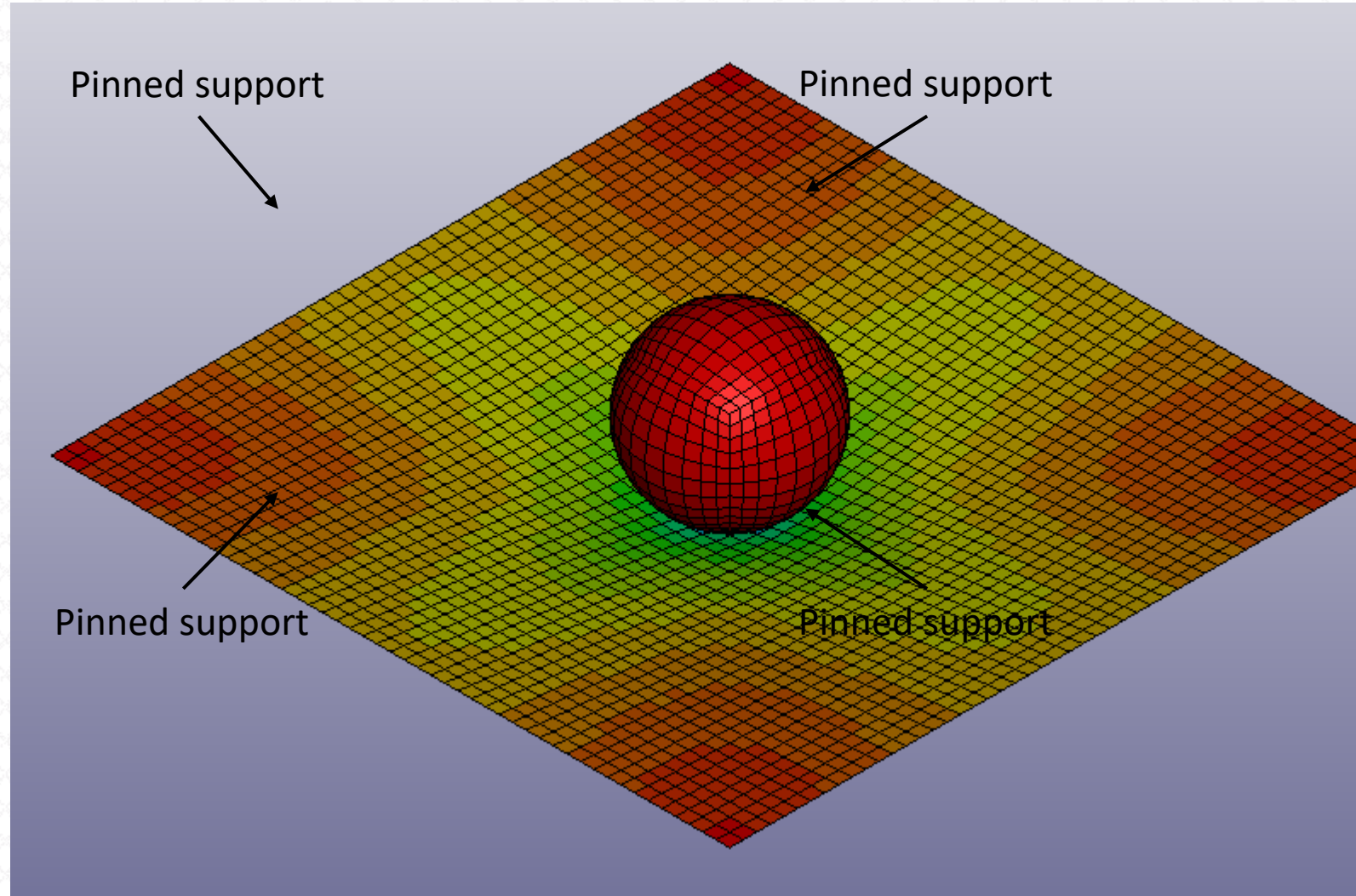
Ball will be modeled as a Rigid Body

## Boundary Conditions

Plate is pinned supported ( $U_x=0$ ,  $U_y=0$ ,  $U_z=0$ ,  
 $R_x=free$ ,  $R_y=free$ ,  $R_z=free$ ) along all 4 outer edges

## Initial Velocity

Ball Initial Velocity = 10mm/ms



# Units

LS-DYNA requires that the Units set used is consistent.

Definition of consistent units:

1 force unit = 1 mass unit \* 1 acceleration unit

1 acceleration unit = 1 length unit / (1 time unit)^2

1 density unit = 1 mass unit / (1 length unit)^3

We will use the [kg-mm-ms-kN] units set in this tutorial.

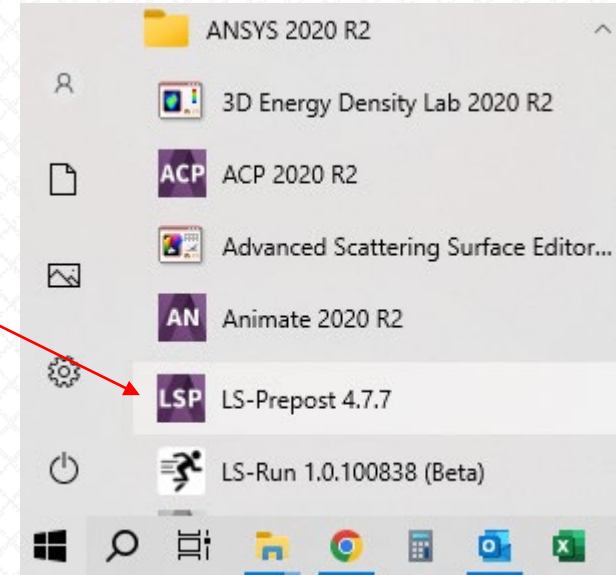
## Consistent set of units for Steel

|                        |        |      |           |                      |             | Consistent set of units for Steel |          |          |          |
|------------------------|--------|------|-----------|----------------------|-------------|-----------------------------------|----------|----------|----------|
| MASS                   | LENGTH | TIME | FORCE     | STRESS               | ENERGY      | DENSITY                           | YOUNG's  | 35MPH    | GRAVITY  |
|                        |        |      |           |                      |             | 56.33KMPH                         |          |          |          |
| kg                     | m      | s    | N         | Pa                   | J           | 7.83E+03                          | 2.07E+11 | 15.65    | 9.806    |
| kg                     | cm     | s    | 1.0e-02 N |                      |             | 7.83E-03                          | 2.07E+09 | 1.56E+03 | 9.81E+02 |
| kg                     | cm     | ms   | 1.0e+04 N |                      |             | 7.83E-03                          | 2.07E+03 | 1.56     | 9.81E-04 |
| kg                     | cm     | us   | 1.0e+10 N |                      |             | 7.83E-03                          | 2.07E-03 | 1.56E-03 | 9.81E-10 |
| kg                     | mm     | ms   | kN        | GPa                  | kN-mm       | 7.83E-06                          | 2.07E+02 | 15.65    | 9.81E-03 |
| g                      | cm     | s    | dyne      | dyne/cm <sup>2</sup> | erg         | 7.83E+00                          | 2.07E+12 | 1.56E+03 | 9.81E+02 |
| g                      | cm     | us   | 1.0e+07 N | Mbar                 | 1.0e+07 Ncm | 7.83E+00                          | 2.07E+00 | 1.56E-03 | 9.81E-10 |
| g                      | mm     | s    | 1.0e-06 N | Pa                   |             | 7.83E-03                          | 2.07E+11 | 1.56E+04 | 9.81E+03 |
| g                      | mm     | ms   | N         | MPa                  | N-mm        | 7.83E-03                          | 2.07E+05 | 15.65    | 9.81E-03 |
| ton                    | mm     | s    | N         | MPa                  | N-mm        | 7.83E-09                          | 2.07E+05 | 1.56E+04 | 9.81E+03 |
| lbf-s <sup>2</sup> /in | in     | s    | lbf       | psi                  | lbf-in      | 7.33E-04                          | 3.00E+07 | 6.16E+02 | 386      |
| slug                   | ft     | s    | lbf       | psf                  | lbf-ft      | 1.52E+01                          | 4.32E+09 | 51.33    | 32.17    |
| kgf-s <sup>2</sup> /mm | mm     | s    | kgf       | kgf/mm <sup>2</sup>  | kgf-mm      | 7.98E-10                          | 2.11E+04 | 1.56E+04 | 9.81E+03 |
| kg                     | mm     | s    | mN        | 1.0e+03 Pa           |             | 7.83E-06                          | 2.07E+08 |          | 9.81E+03 |
| g                      | cm     | ms   | 1.0e+1 N  | 1.0e+05 Pa           |             | 7.83E+00                          | 2.07E+06 |          | 9.81E-04 |



# Launch LS-Prepost

1. Start > ANSYS 2020 R2 > LS-Prepost



# Steps

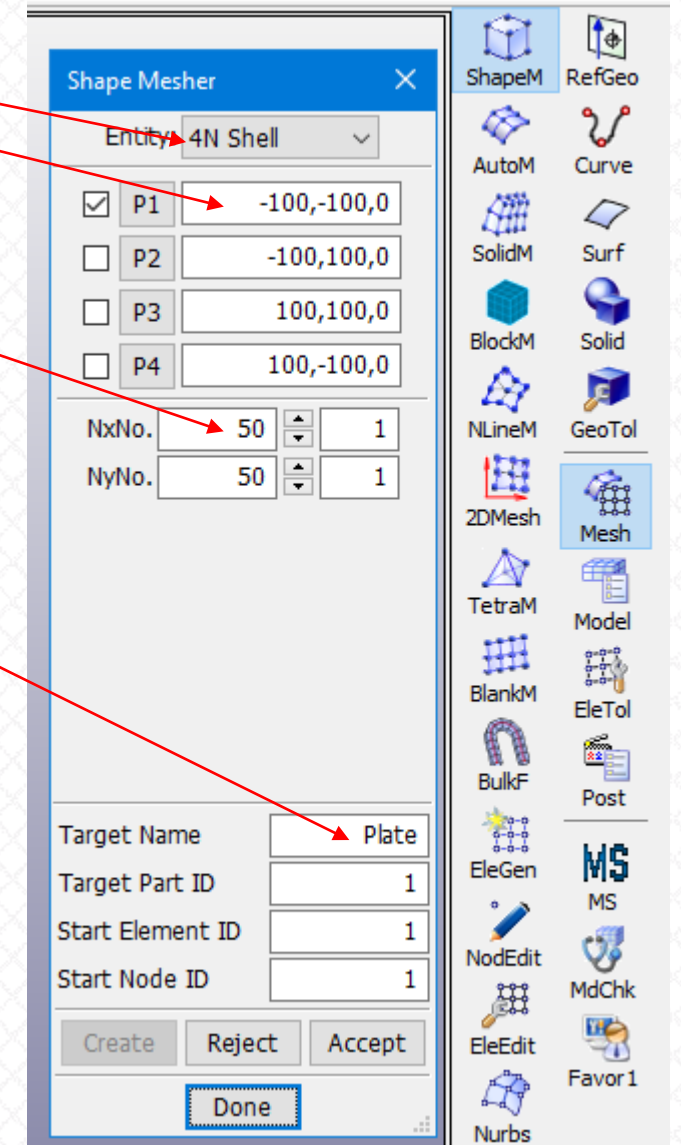
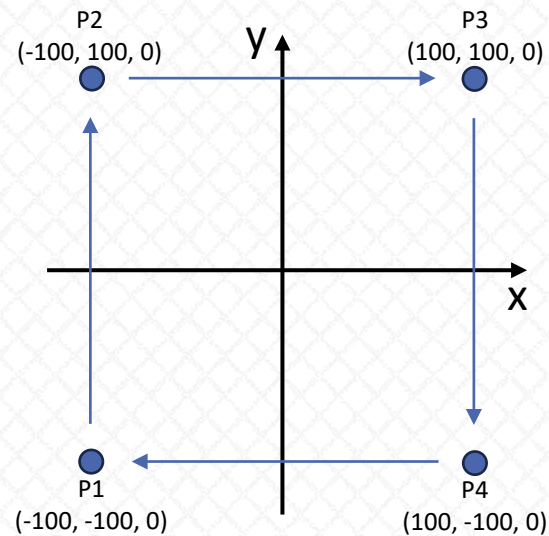
| Step # | Description                                     |   |
|--------|---|---|
| 1      | Create Geometry and Mesh                        | We will create geometry/FE mesh of a Plate and a Ball |
| 2      | Boundary Conditions                             |   |
| 3      | Material Properties                             |   |
| 4      | Section/Element Properties                      |   |
| 5      | Assign Material and Section Properties to Parts |   |
| 6      | Contact   |   |
| 7      | Initial Velocity                                |   |
| 8      | Analysis Time and Output Controls               |   |
| 9      | Submit Analysis in LS-Run                       |   |
| 10     | Postprocess results in LS-Prepost               |   |



# Create Plate

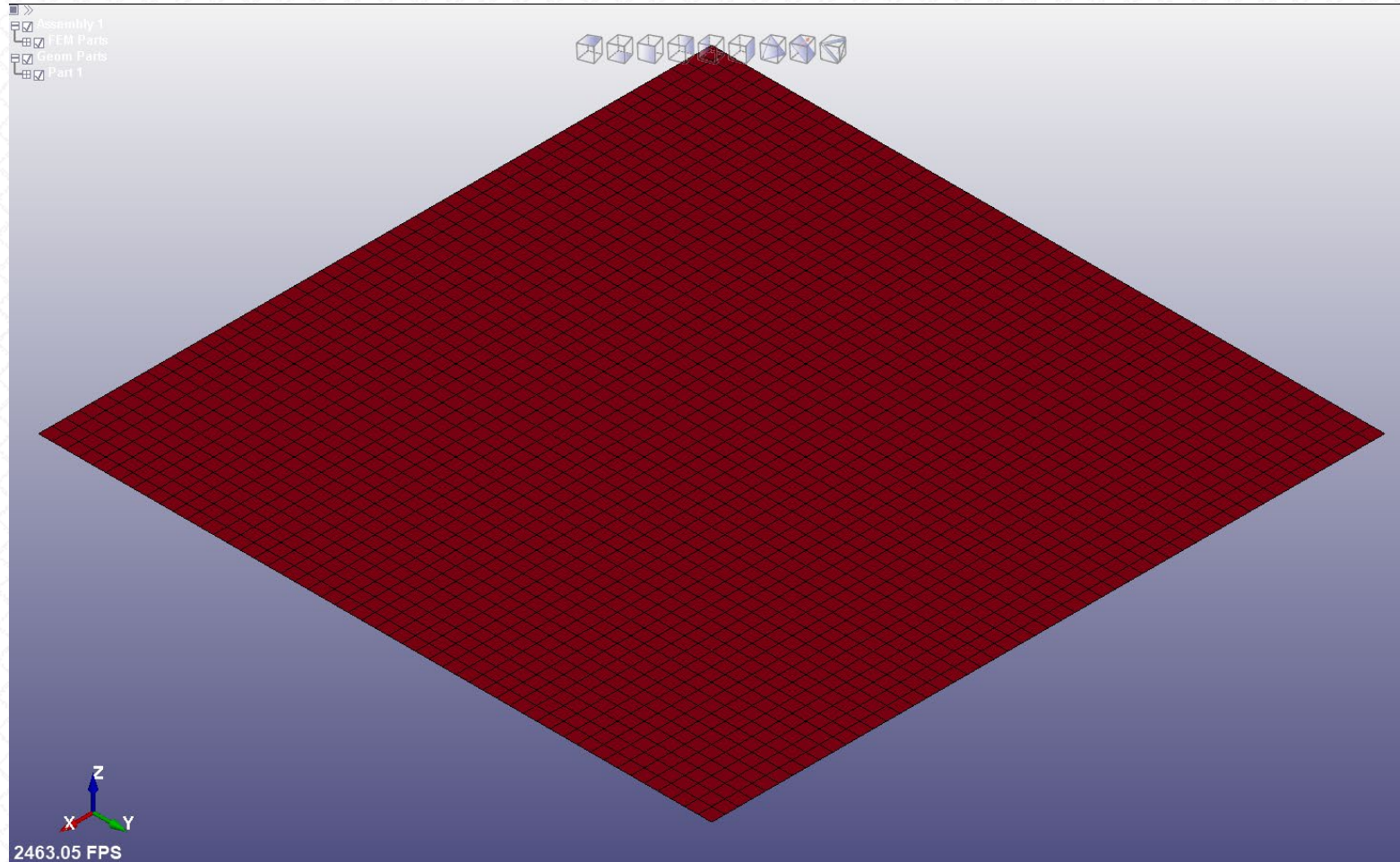
1. Mesh > ShapeM > Entity: 4N Shell
2. Enter x, y, z values for P1, P2, P3, P4 as shown
3. Enter NxNo and NyNo as shown
4. Target Name: Plate
5. Create
6. Accept
7. Done

Note:  
P1, P2, P3, P4 coordinates  
when creating 4N Shell



# Create Plate

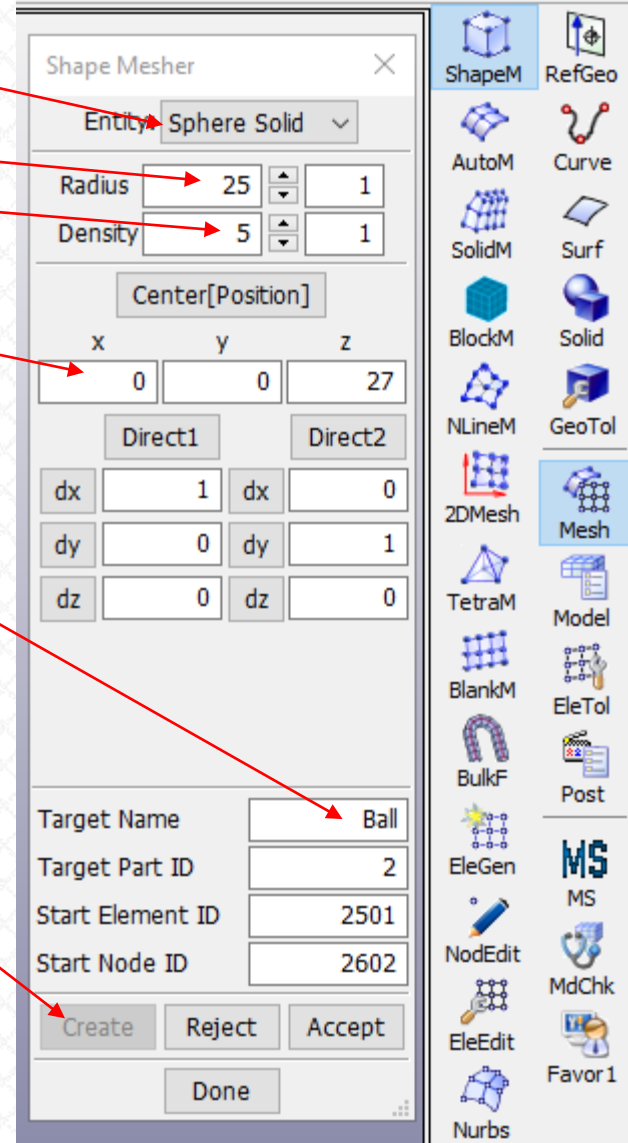
1. Shell Plate will be created





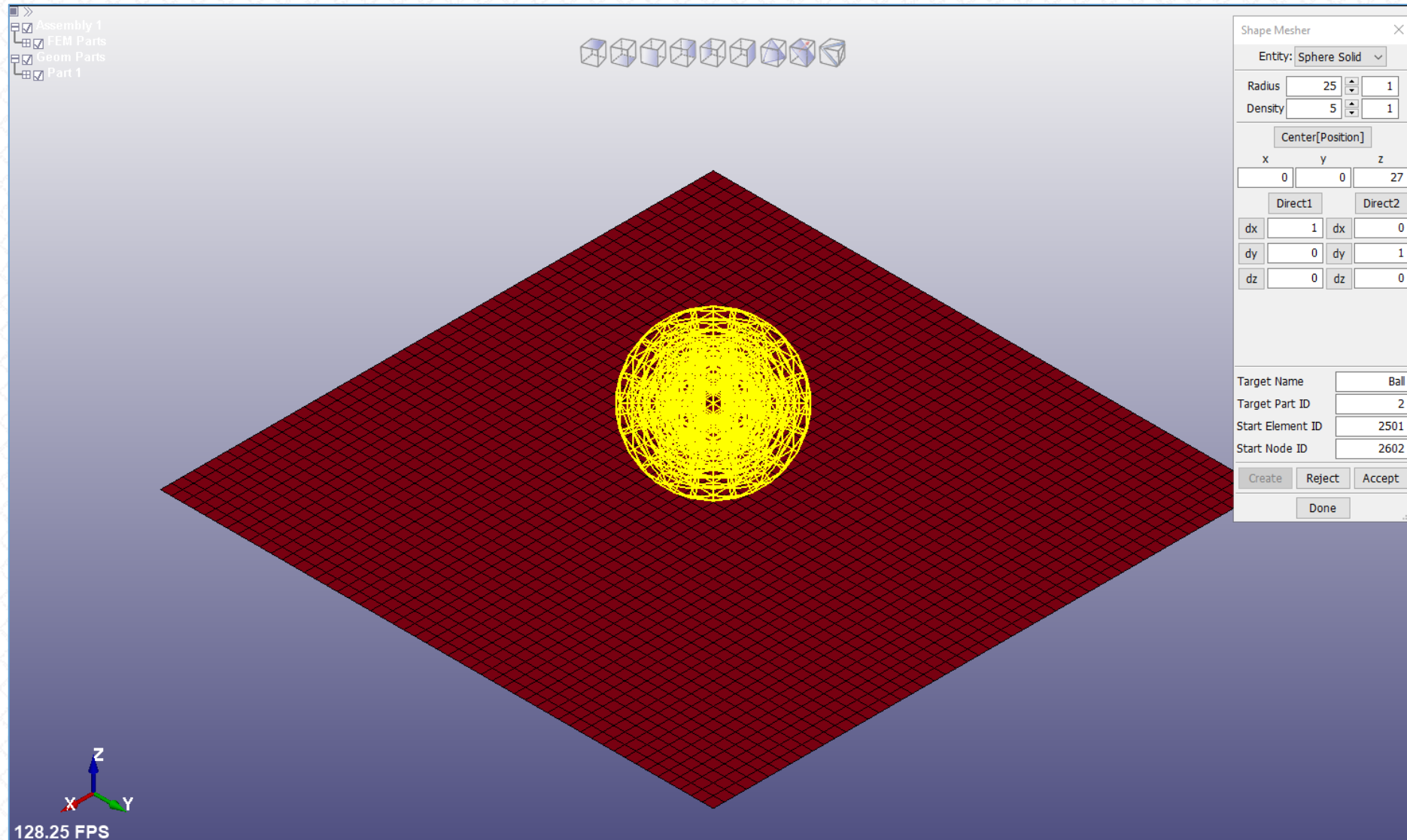
# Create Ball

1. Mesh > ShapeM > Entity: Sphere Solid
2. Radius: 25
3. Density: 5
4.  $x=0$ ,  $y=0$ ,  $z=27$
5. Target Name: Ball
6. Create



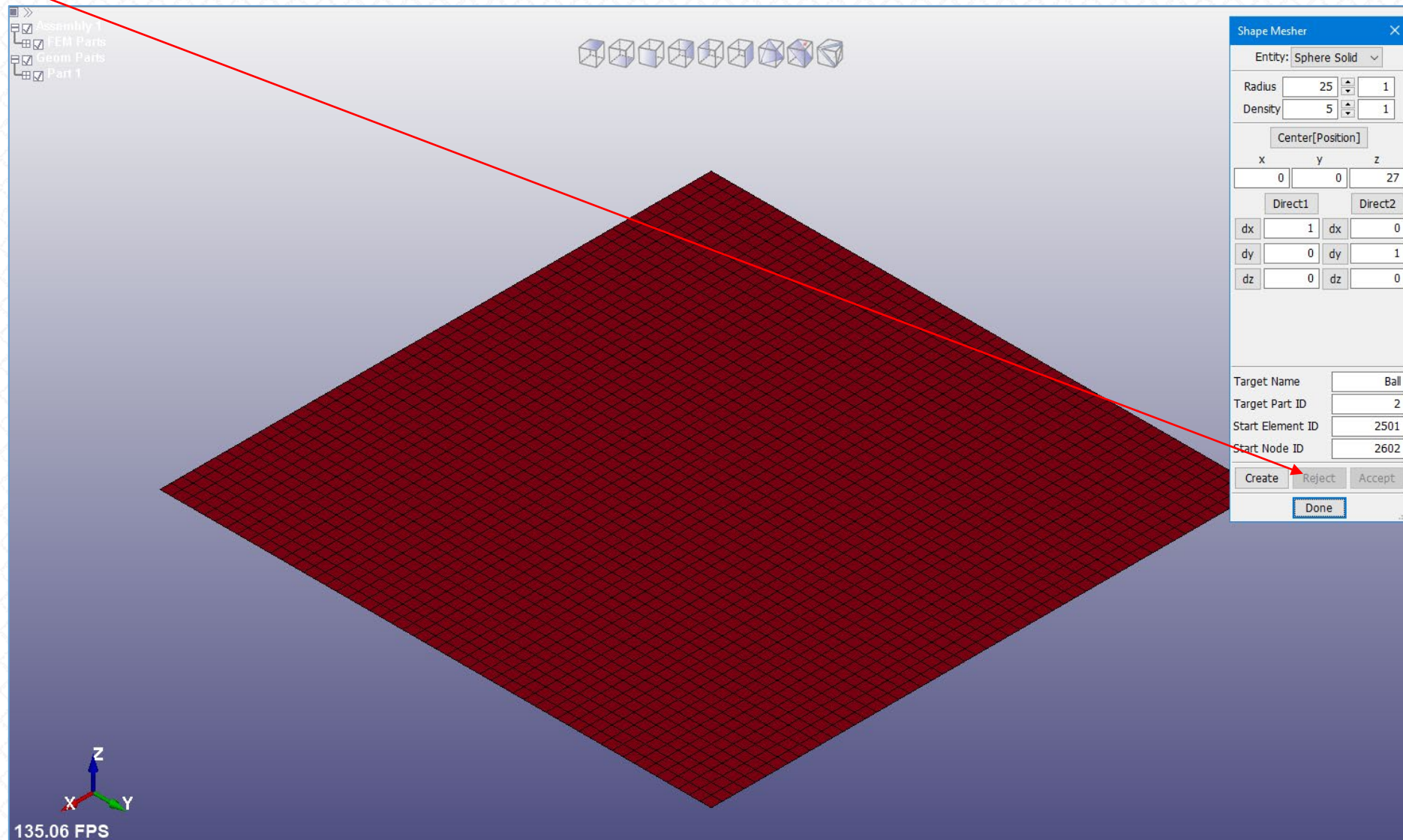
# Create Ball

1. Ball mesh will be shown:



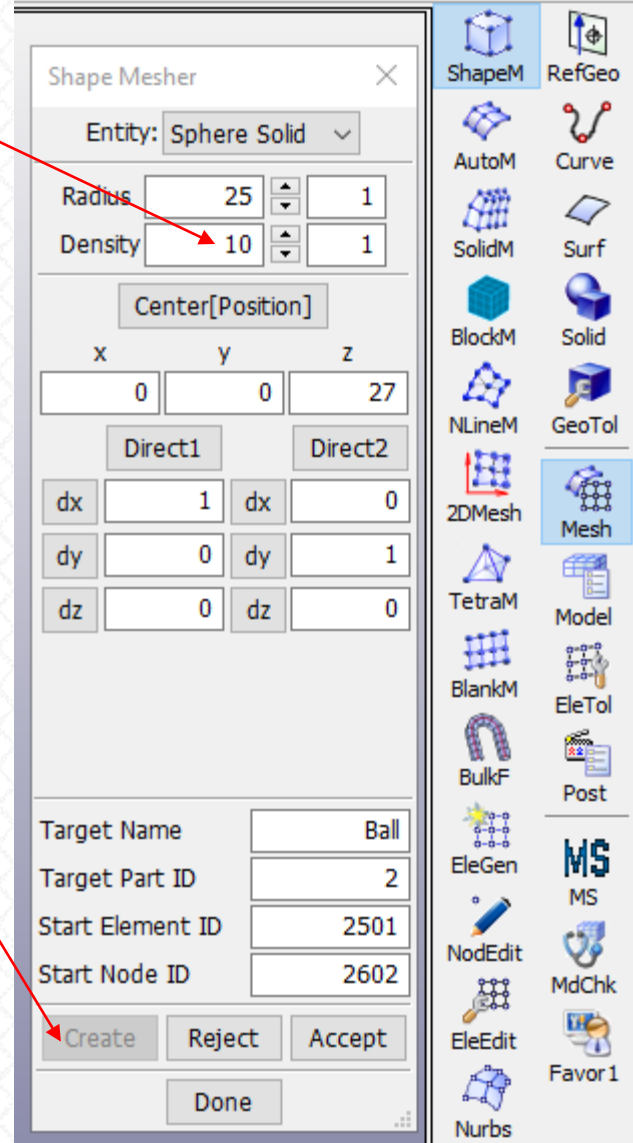
# Create Ball

## 1. Reject (we'd like to change Mesh Density)



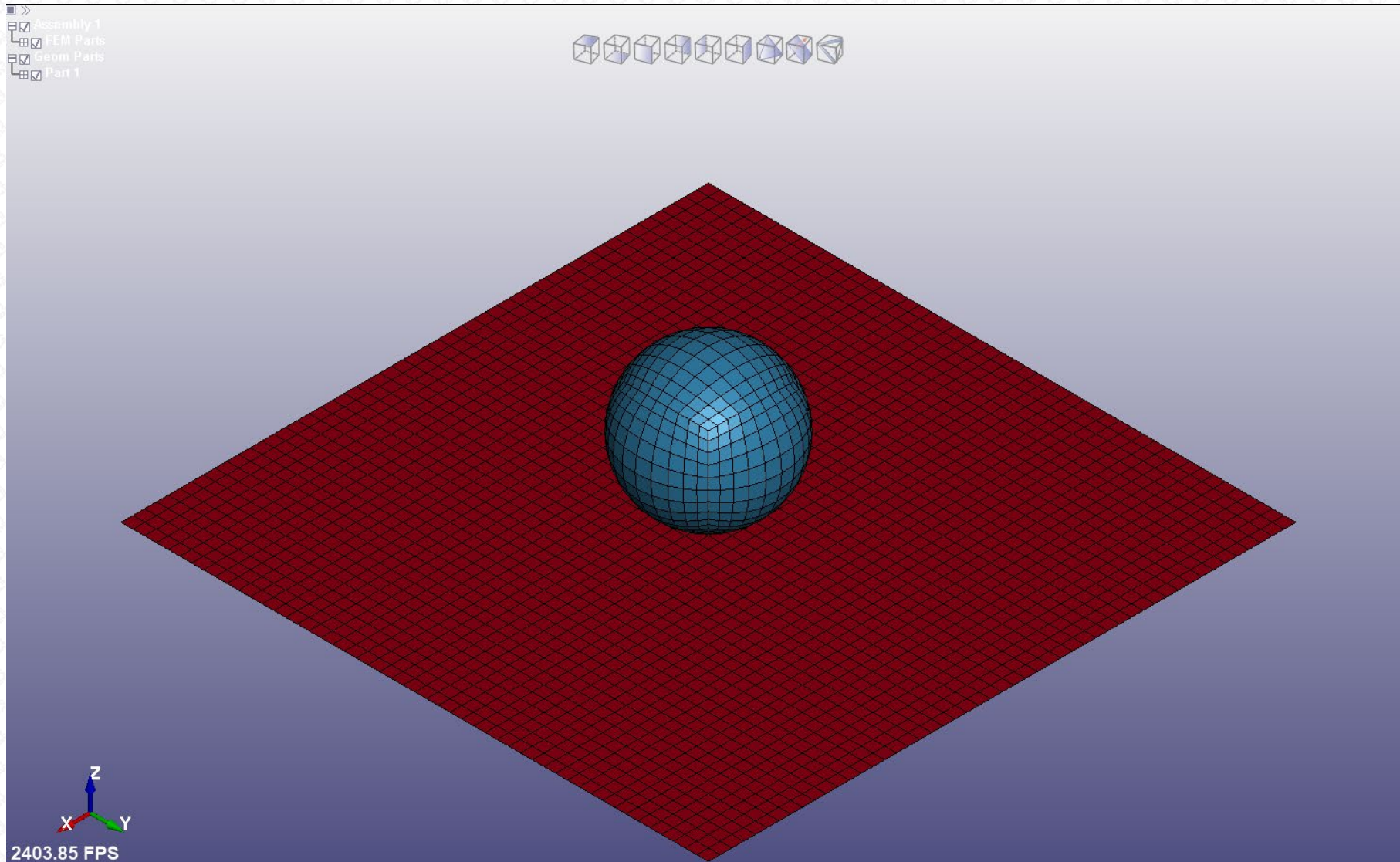
# Create Ball

1. Change Density to 10
2. Create
3. Accept
4. Done



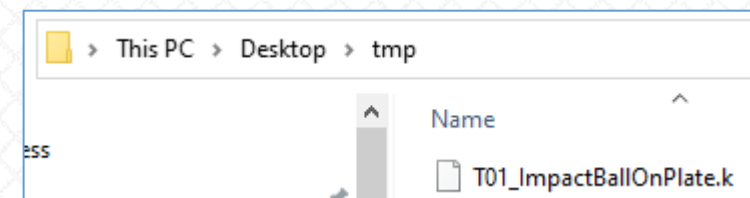
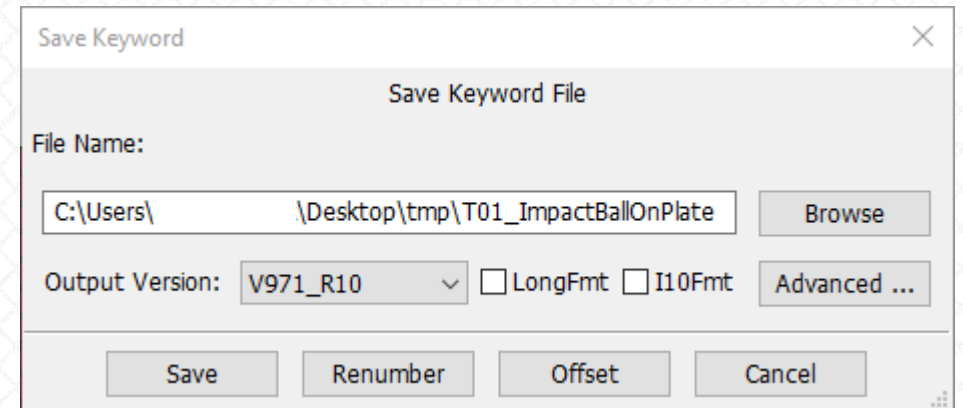
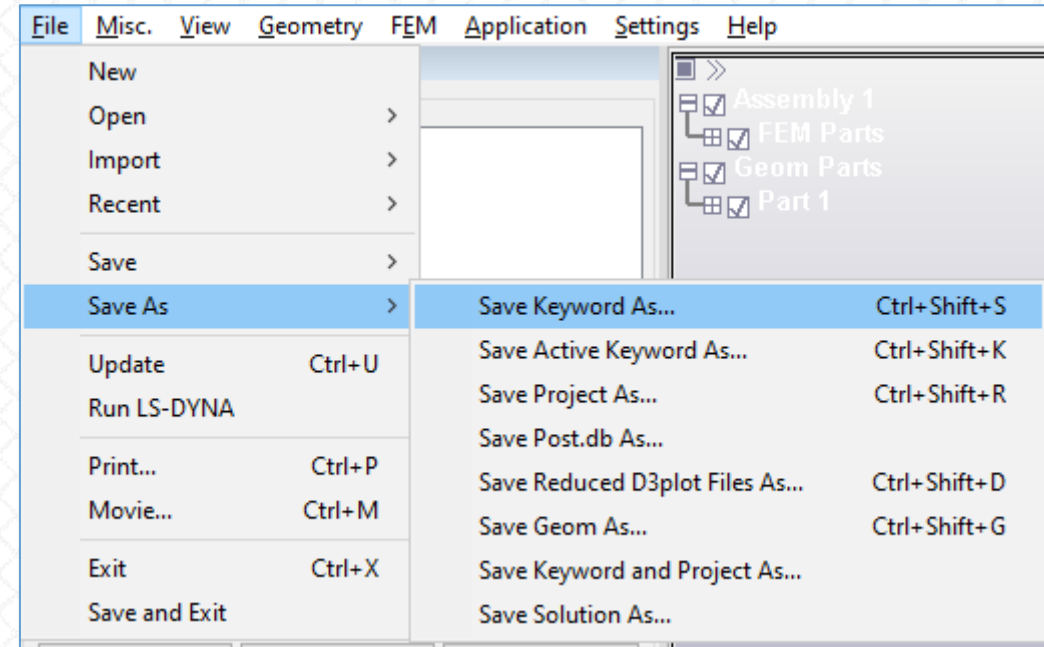
# Create Ball

1. Ball mesh will be created:



# Save

1. Remember to Save regularly, because LS-Prepost may close or freeze unexpectedly.
2. File > Save As > Save Keyword As...
3. “Save Keyword” popup window will appear
4. Browse to desired folder
5. Specify name. Give it .k extension
6. Hit “Save”

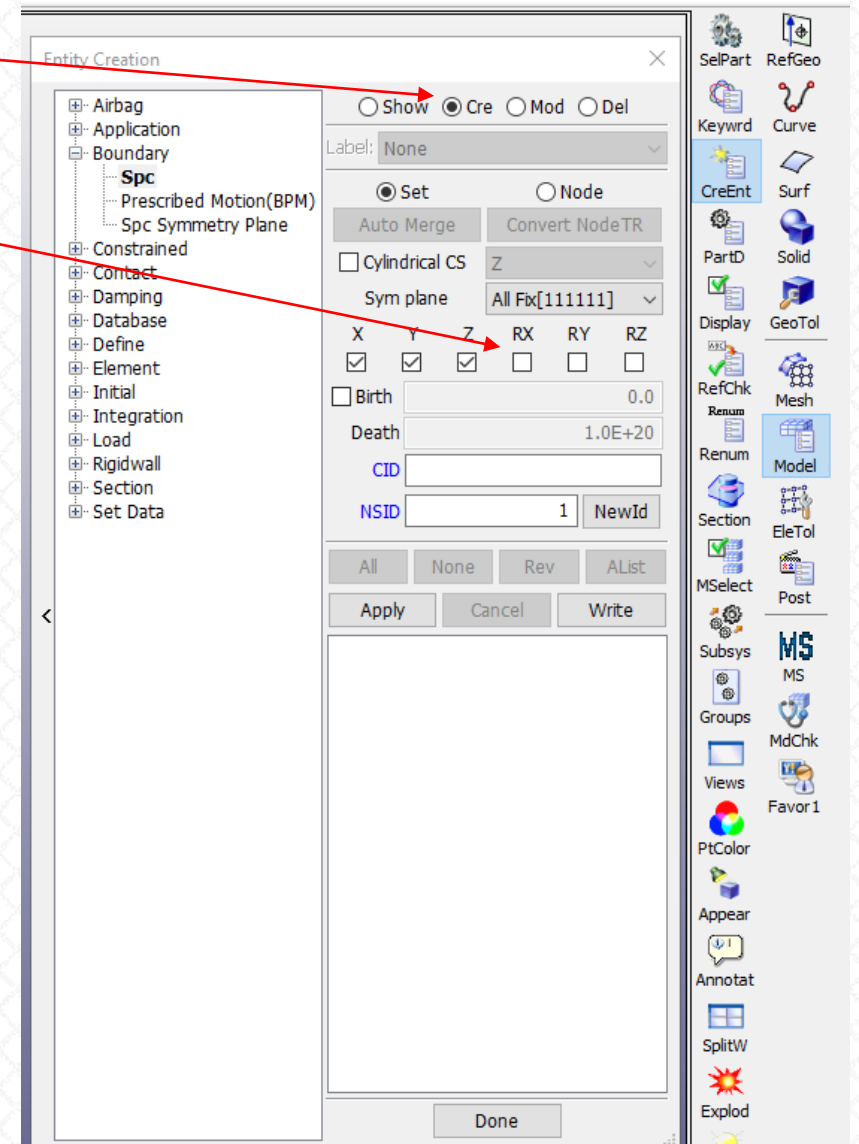


# Steps

| Step # | Description                                     |  |
|--------|---|--|
| 1      | Create Geometry and Mesh                        |  |
| 2      | Boundary Conditions                             | We will create a pinned support Boundary Condition for outer edges of the Plate. |
| 3      | Material Properties                             |  |
| 4      | Section/Element Properties                      |  |
| 5      | Assign Material and Section Properties to Parts |  |
| 6      | Contact   |  |
| 7      | Initial Velocity                                |  |
| 8      | Analysis Time and Output Controls               |  |
| 9      | Submit Analysis in LS-Run                       |  |
| 10     | Postprocess results in LS-Prepost               |  |

# Boundary Conditions

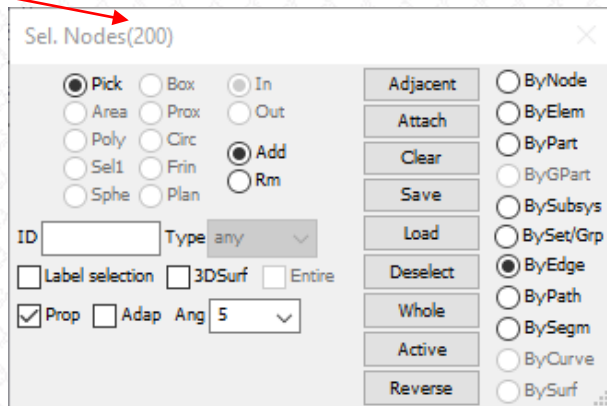
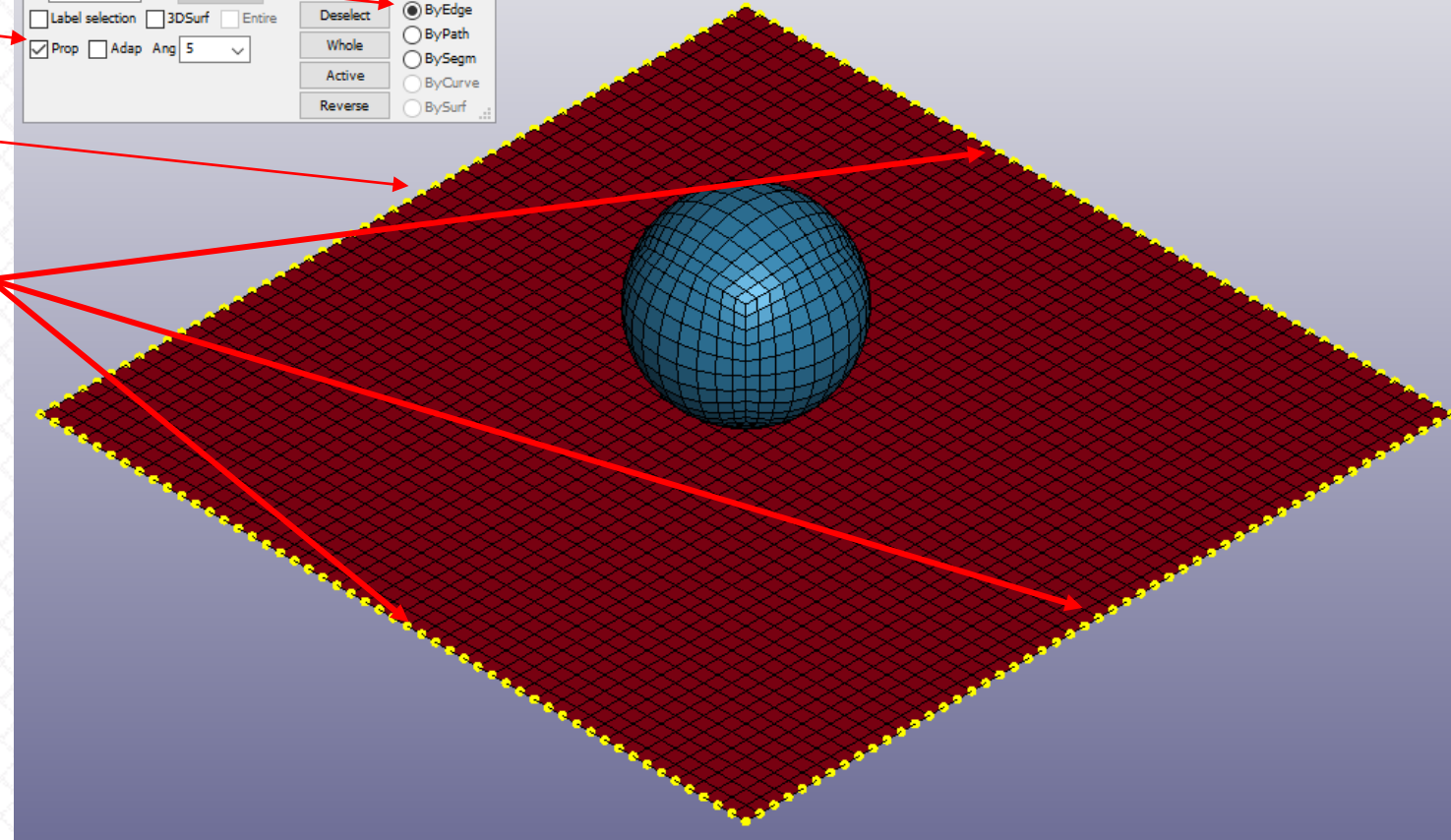
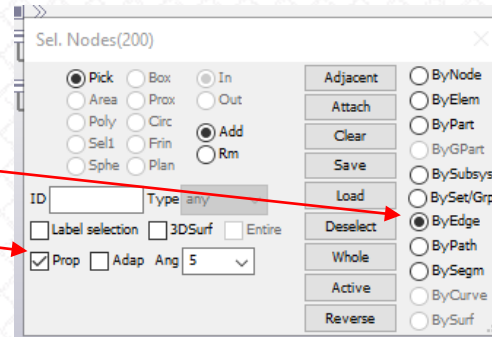
1. Model > CreEnt > Boundary > Spc
2. Switch radio button from “Show” to “Create”
3. Uncheck boxes: RX, RY, RZ





# Boundary Conditions

1. In Sel. Nodes dialog window select:
  - a) ByEdge
  - b) Prop
2. Hover mouse pointer over one of the outer edges
3. LMB (Left Mouse Button) click
4. This should select and highlight all nodes along this edge
5. Repeat Steps 2 – 4 for the rest of the outer edges
6. All nodes of the outer edges should be highlighted as shown.
7. "Sel. Nodes(200)" should show (200) meaning there are 200 nodes selected.

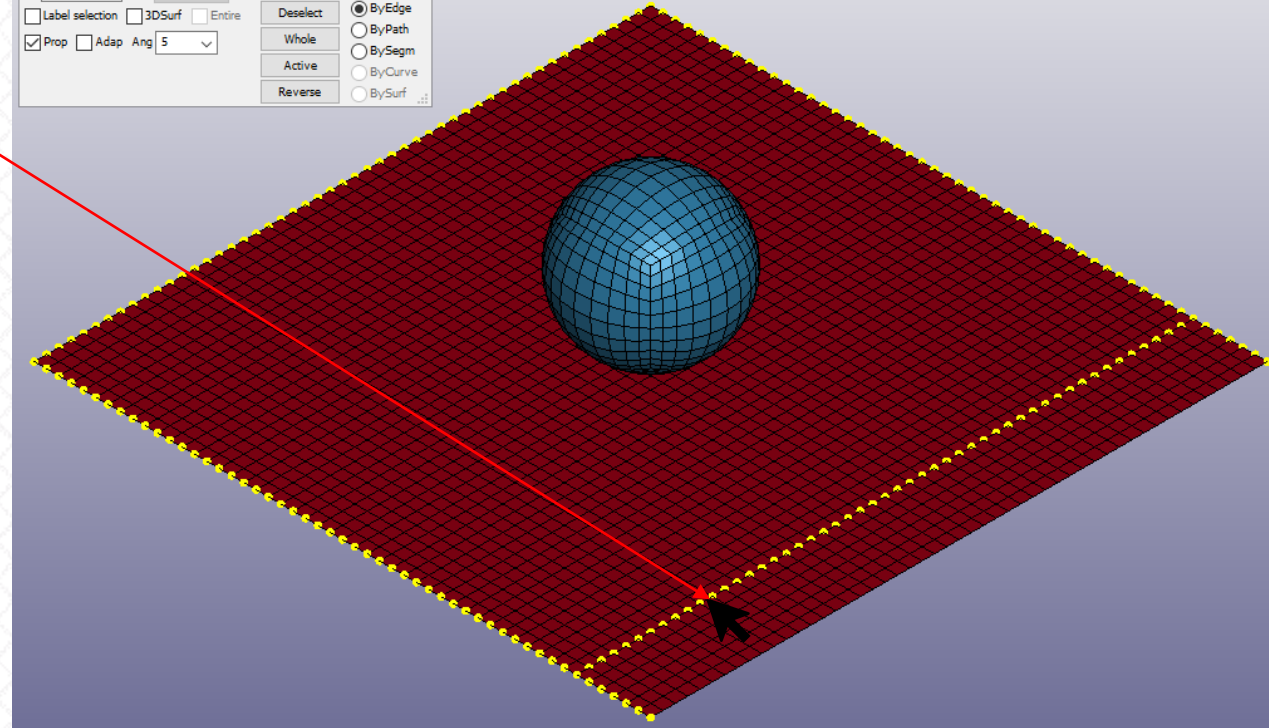
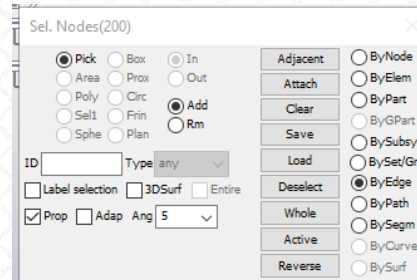
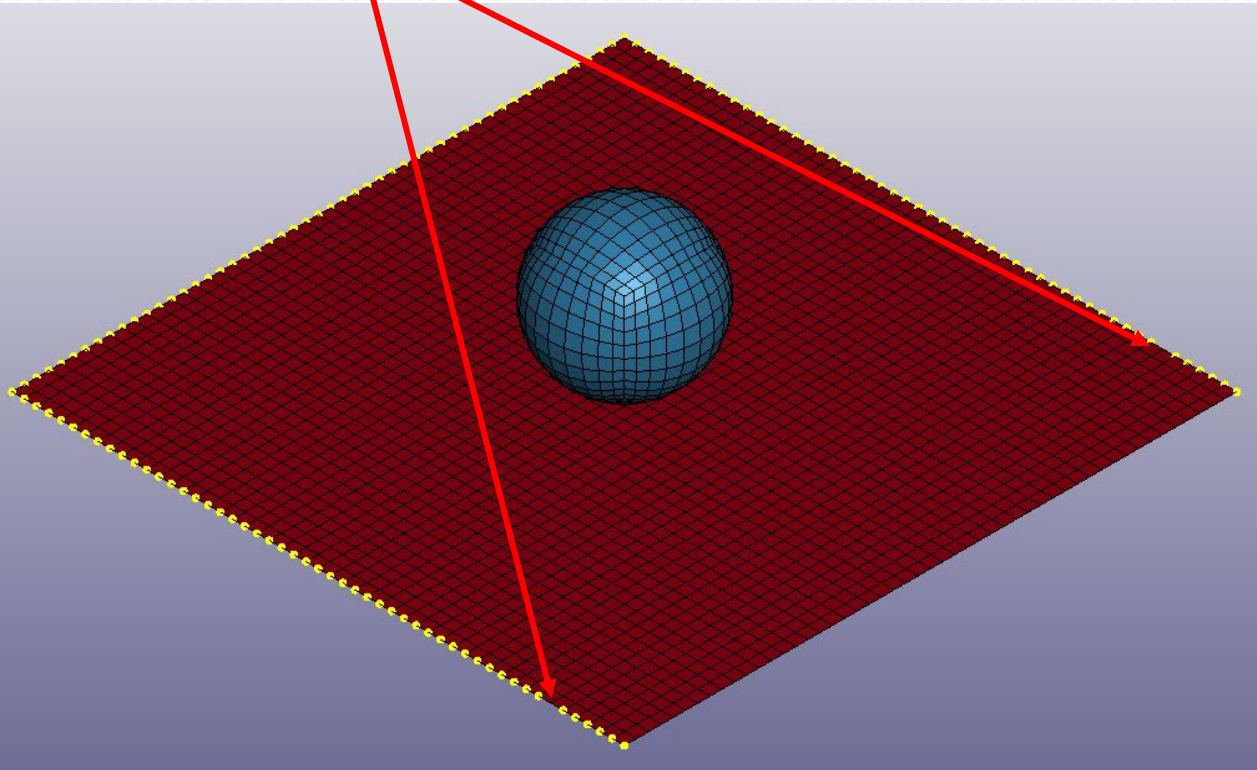


Note: see tip on next slide if you selected wrong nodes accidentally



# Boundary Conditions

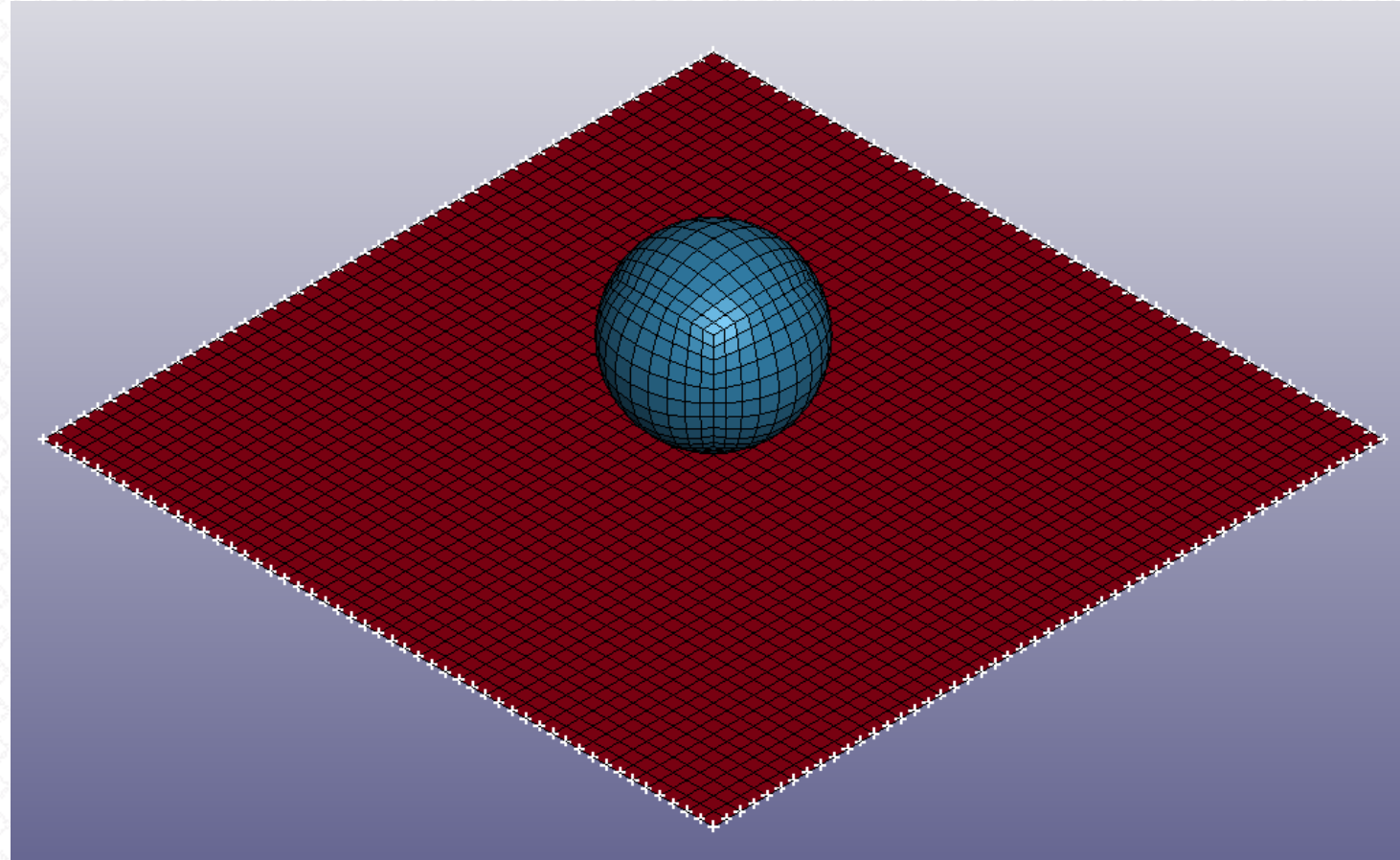
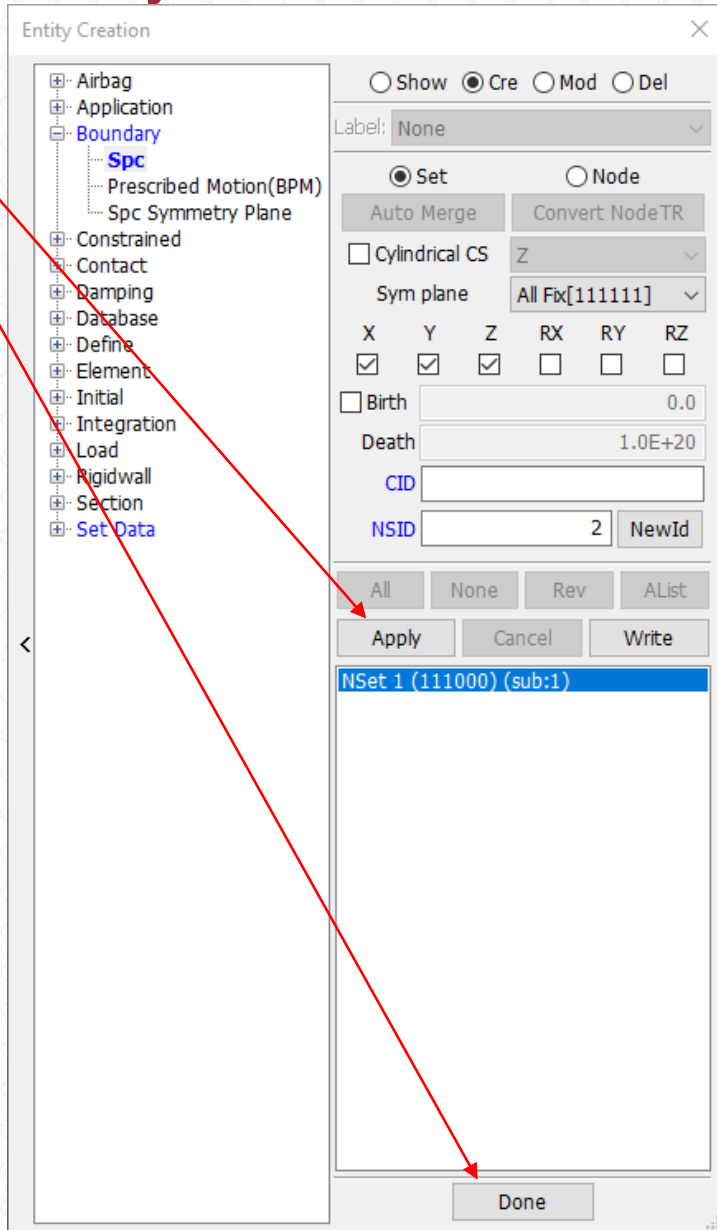
1. If you selected wrong nodes accidentally:
  - a) Hover mouse pointer over that region
  - b) RMB click
  - c) This should unselect all wrong nodes
  - d) Two nodes that we need were also unselected



- e) Select them back with LMB

# Boundary Conditions

1. Apply
2. Done



Each outer edge is fixed in X, Y, Z while is free to rotate in RX, RY, RZ.

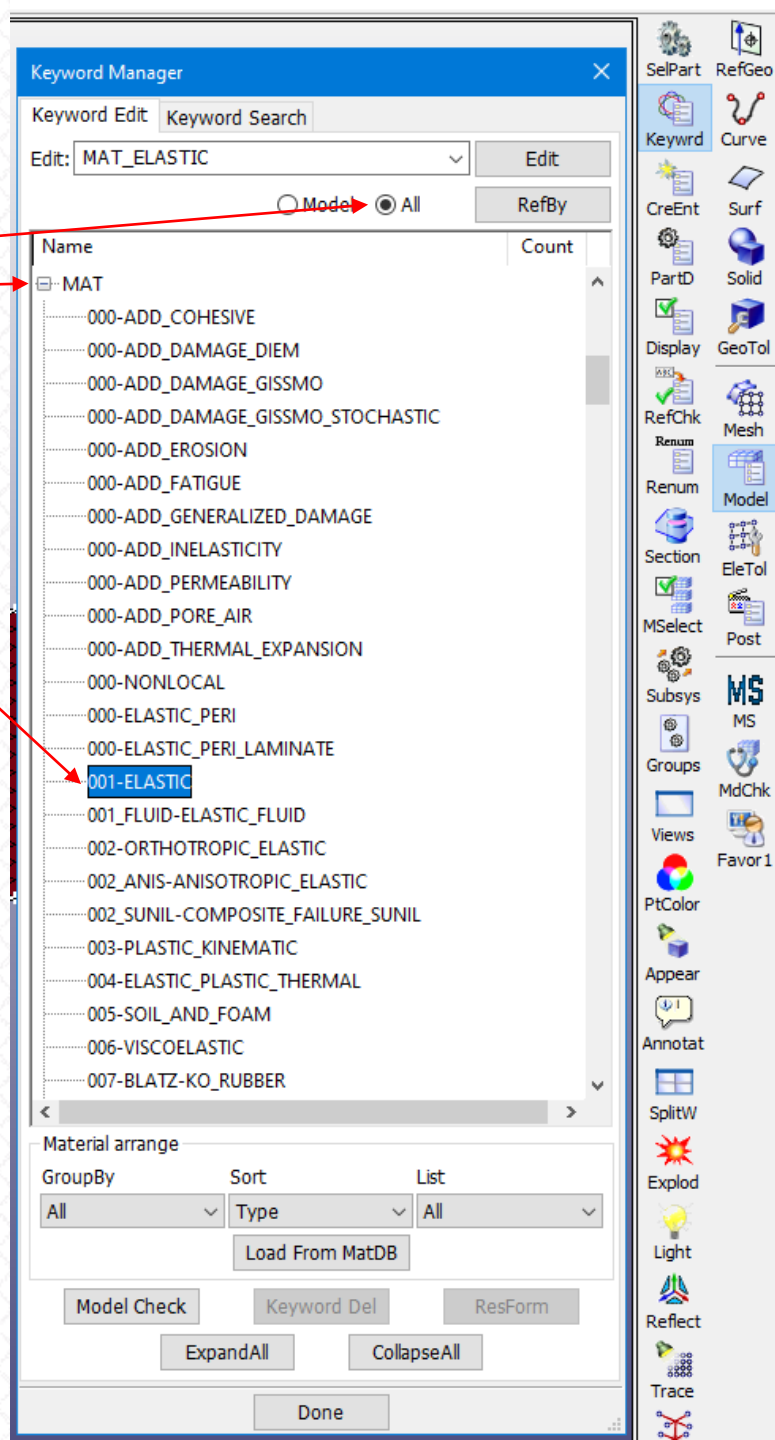


# Steps

| Step # | Description                                     |  |
|--------|---|--|
| 1      | Create Geometry and Mesh                        |  |
| 2      | Boundary Conditions                             |  |
| 3      | Material Properties                             | We will create Elastic material property for the Plate, and Rigid material property for the Ball |
| 4      | Section/Element Properties                      |  |
| 5      | Assign Material and Section Properties to Parts |  |
| 6      | Contact   |  |
| 7      | Initial Velocity                                |  |
| 8      | Analysis Time and Output Controls               |  |
| 9      | Submit Analysis in LS-Run                       |  |
| 10     | Postprocess results in LS-Prepost               |  |

# Material Properties

1. Model > Keywrđ
2. Switch radio button from “Model” to “All”
3. Expand [MAT]
4. Double click on [001-ELASTIC]



# Material Properties

1. Click "NewID"
2. TITLE: Steel
3. Type values for RO, E, PR as shown
  - a) Density RO =  $7.83 \times 10^{-6}$  kg/mm<sup>3</sup>
  - b) Elastic Modulus E = 207 GPa
  - c) Poisson's ratio PR = 0.3
4. Accept
5. Done

Keyword Input Form

NewID MatDB RefBy Pick Add Accept Delete Default Done 1 Steel

Use \*Parameter  Comment (Subsys: 1 T01\_PADT\_LS-DYNA\_BallOnPlate.k) Setting

\*MAT\_ELASTIC\_(TITLE) (001) (1)

TITLE  
Steel

| 1 | <u>MID</u> | <u>RO</u> | <u>E</u>  | <u>PR</u> | <u>DA</u> | <u>DB</u> | <u>NOT USED</u> |
|---|------------|-----------|-----------|-----------|-----------|-----------|-----------------|
|   | 1          | 7.830e-06 | 207.00000 | 0.3000000 | 0.0       | 0.0       | 0.0             |

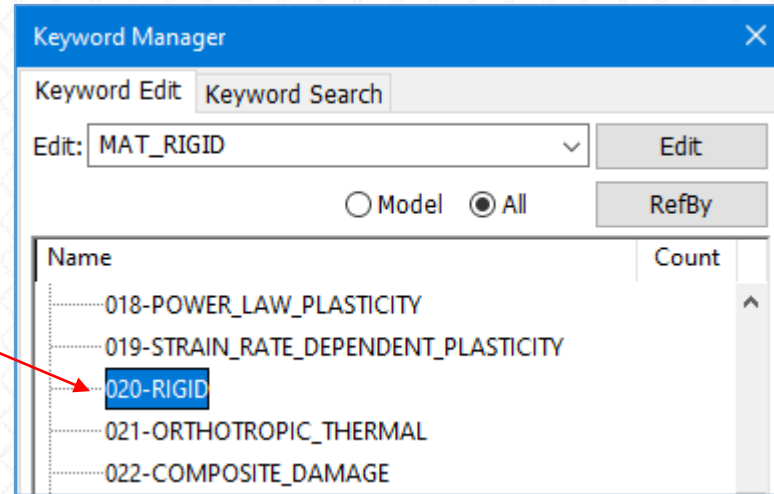
COMMENT:

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



# Material Properties

1. Under [MAT] scroll down to [020-RIGID]
2. Double click on [020-RIGID]



# Material Properties

1. Click "NewID"
2. TITLE: Rigid
3. Type values for RO, E, PR as shown
  - a) Density RO =  $7.83 \times 10^{-6}$  kg/mm<sup>3</sup>
  - b) Elastic Modulus E = 207 GPa
  - c) Poisson's ratio PR = 0.3
4. Accept
5. Done

Keyword Input Form

NewID MatDB RefBy Pick Add Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 T01\_PADT\_LS-DYNA\_BallOnPlate.k) Setting

\*MAT\_RIGID\_(TITLE) (020) (1)

TITLE  
Rigid

| 1 | <u>MID</u> | <u>RO</u> | <u>E</u>  | <u>PR</u> | <u>N</u> | <u>COUPLE</u> | <u>M</u> | <u>ALIAS</u> |
|---|------------|-----------|-----------|-----------|----------|---------------|----------|--------------|
|   | 2          | 7.830e-06 | 207.00000 | 0.3000000 | 0.0      | 0             | 0.0      |              |

| 2 | <u>CMO</u> | <u>CON1</u> | <u>CON2</u> |
|---|------------|-------------|-------------|
|   | 0.0        | 0           | 0           |

| 3 | <u>LCO_OR_A1</u> | <u>A2</u> | <u>A3</u> | <u>V1</u> | <u>V2</u> | <u>V3</u> |
|---|------------------|-----------|-----------|-----------|-----------|-----------|
|   | 0.0              | 0.0       | 0.0       | 0.0       | 0.0       | 0.0       |

COMMENT:

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

Note:  
Rigid material property will be assigned to the Ball.  
Components for which deformation is negligible and stress is unimportant may be modeled as rigid bodies.  
The elastic constants defined here are used for contact stiffness calculations. Thus, the constants should be reasonable.



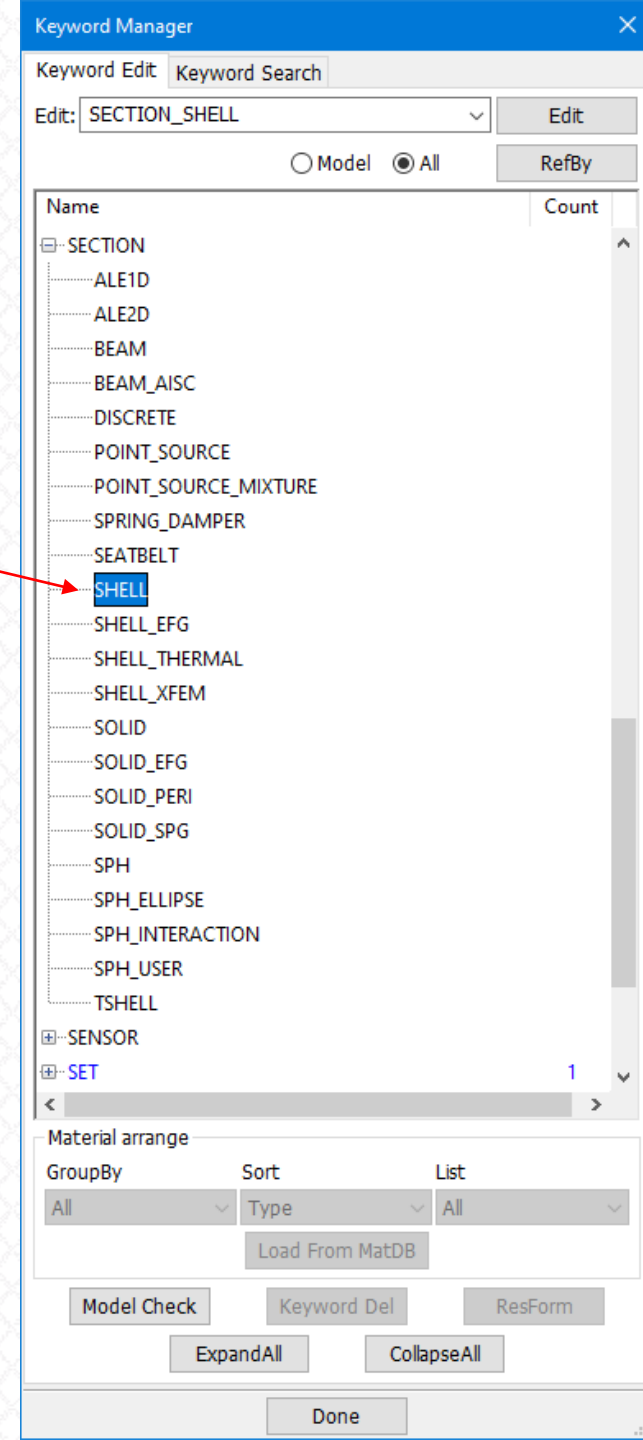


# Steps

| Step # | Description                                     |  |
|--------|---|--|
| 1      | Create Geometry and Mesh                        |  |
| 2      | Boundary Conditions                             |  |
| 3      | Material Properties                             |  |
| 4      | Section/Element Properties                      | We will create SHELL section property for the Plate, and SOLID section property for the Ball |
| 5      | Assign Material and Section Properties to Parts |  |
| 6      | Contact   |  |
| 7      | Initial Velocity                                |  |
| 8      | Analysis Time and Output Controls               |  |
| 9      | Submit Analysis in LS-Run                       |  |
| 10     | Postprocess results in LS-Prepost               |  |

# Section Properties / Element Formulation

1. Expand [SECTION]
2. Double click on [SHELL]



# Section Properties / Element Formulation

1. Click "NewID"
2. TITLE: Section Shell
3. Leave ELFORM = 2 (default, Belytschko-Tsay element formulation)
4. Type 0.1 in [T1] field
5. Press Enter
6. T2, T3, T4 fields should auto update with 0.1 value
7. Accept
8. Done

The screenshot shows the 'Keyword Input Form' for a 'Section Shell' element. The form is titled '\*SECTION\_SHELL\_(TITLE) (1)'. It includes a 'TITLE' field with the value 'Section Shell'. Below this is a table of properties with columns: SECID, ELFORM, SHRF, NIP, PROPT, OR/IRID, ICOMP, and SETYP. The first row (SECID 1) shows ELFORM = 2, SHRF = 1.0000000, NIP = 2, PROPT = 1, OR/IRID = 0, ICOMP = 0, and SETYP = 1. The second row (SECID 2) shows T1 = 0.1000000, T2 = 0.1000000, T3 = 0.1000000, T4 = 0.1000000, NLOC = 0.0, MAREA = 0.0, IDOF = 0.0, and EDGSET = 0. Below the table is a 'Repeated Data by Button and List' section with buttons for 'Replace', 'Insert', 'Delete', and 'Help'. At the bottom, it shows 'Total Card: 1', 'Smallest ID: 1', 'Largest ID: 1', and 'Total deleted card: 0'. Red arrows from the list on the left point to the 'NewID' button, the 'TITLE' field, the 'ELFORM' field, and the 'T1' field.

| SECID | ELFORM | SHRF      | NIP | PROPT | OR/IRID | ICOMP | SETYP |
|-------|--------|-----------|-----|-------|---------|-------|-------|
| 1     | 2      | 1.0000000 | 2   | 1     | 0       | 0     | 1     |
| 2     |        |           |     |       |         |       |       |

| T1        | T2        | T3        | T4        | NLOC | MAREA | IDOF | EDGSET |
|-----------|-----------|-----------|-----------|------|-------|------|--------|
| 0.1000000 | 0.1000000 | 0.1000000 | 0.1000000 | 0.0  | 0.0   | 0.0  | 0      |



# Section Properties / Element Formulation

1. Expand [SECTION]
2. Double click on [SOLID]

The screenshot shows the 'Keyword Manager' application window. At the top, there are tabs for 'Keyword Edit' and 'Keyword Search'. Below the tabs, there is an 'Edit:' dropdown menu set to 'SECTION\_SOLID', an 'Edit' button, and radio buttons for 'Model' and 'All' (selected). A 'RefBy' button is also present. The main area is a tree view with a 'Name' column and a 'Count' column. The tree is expanded to show the 'SECTION' folder, which contains several sub-items. The 'SOLID' item is highlighted with a blue selection box. A red arrow points from the second step of the instructions to this 'SOLID' item. Below the tree view, there is a 'Material arrange' section with 'GroupBy' (All), 'Sort' (Type), and 'List' (All) dropdown menus, and a 'Load From MatDB' button. At the bottom, there are buttons for 'Model Check', 'Keyword Del', 'ResForm', 'ExpandAll', 'CollapseAll', and 'Done'.

| Name                 | Count |
|----------------------|-------|
| SECTION              | 1     |
| ALE1D                |       |
| ALE2D                |       |
| BEAM                 |       |
| BEAM_AISC            |       |
| DISCRETE             |       |
| POINT_SOURCE         |       |
| POINT_SOURCE_MIXTURE |       |
| SPRING_DAMPER        |       |
| SEATBELT             |       |
| SHELL                | 1     |
| SHELL_EFG            |       |
| SHELL_THERMAL        |       |
| SHELL_XFEM           |       |
| <b>SOLID</b>         |       |
| SOLID_EFG            |       |
| SOLID_PERI           |       |
| SOLID_SPG            |       |
| SPH                  |       |
| SPH_ELLIPSE          |       |
| SPH_INTERACTION      |       |
| SPH_USER             |       |
| TSHELL               |       |
| SENSOR               |       |
| SET                  | 1     |



# Section Properties / Element Formulation

1. Click "NewID" →
2. TITLE: Section Solid →
3. Leave ELFORM = 1 (default, constant stress solid element formulation)
4. Accept
5. Done

Keyword Input Form

NewID Draw RefBy Add Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 T01\_PADT\_LS-DYNA\_BallOnPlate.k) Setting

\*SECTION\_SOLID\_(TITLE) (1)

TITLE

Section Solid

1 SECID ELFORM AET

1 1 0

Repeated Data by Button and List

Data Pt.

Replace Insert

Delete Help

Repeated Data by Button and List

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

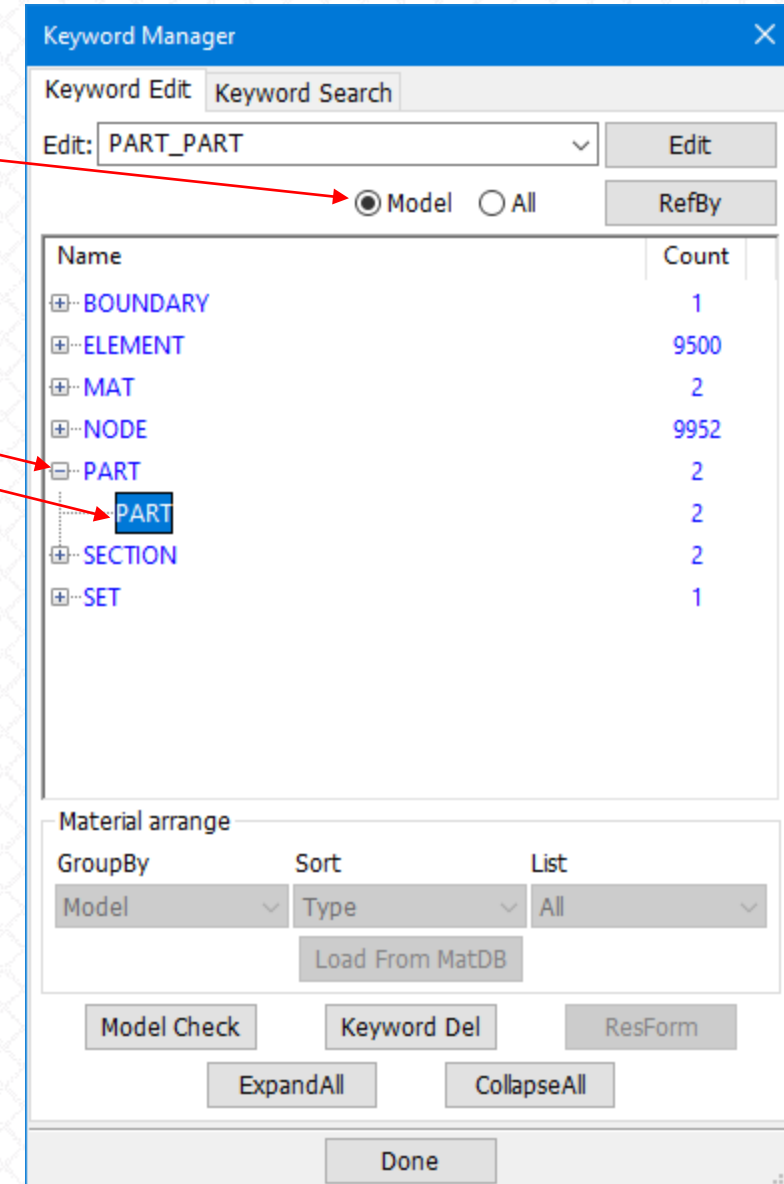


# Steps

| Step # | Description                                     |  |
|--------|---|--|
| 1      | Create Geometry and Mesh                        |  |
| 2      | Boundary Conditions                             |  |
| 3      | Material Properties                             |  |
| 4      | Section/Element Properties                      |  |
| 5      | Assign Material and Section Properties to Parts | We will assign created Material and Section properties to the Plate and Ball |
| 6      | Contact   |  |
| 7      | Initial Velocity                                |  |
| 8      | Analysis Time and Output Controls               |  |
| 9      | Submit Analysis in LS-Run                       |  |
| 10     | Postprocess results in LS-Prepost               |  |

# Assign Material and Section Properties to Parts

1. Switch radio button from "All" to "Model"
2. Expand [PART]
3. Double click on [PART]



# Assign Material and Section Properties to Parts

1. [1 Plate] part should be pre-selected
2. TITLE: Plate
3. Press [•] next to [SECID]
4. "Link SECTION" popup window will appear
5. Double click on [1 Section Shell]
6. Press [•] next to [MID]
7. "Link MAT" popup window will appear
8. Double click on [1 Steel]
9. Accept

Keyword Input Form

NewID Draw RefBy Pick Add Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 T01\_PADT\_LS-DYNA\_BallOnPlate.k) Setting

\*PART\_(TITLE) (2)

|   | TITLE | PID | SECID | MID | EOSID | HGID | GRAV | ADOPT | TMID |
|---|-------|-----|-------|-----|-------|------|------|-------|------|
| 1 | Plate | 1   | 1     | 1   | 0     | 0    | 0    | 0     | 0    |
| 2 |       |     |       |     |       |      |      |       |      |

COMMENT:

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

1 Plate  
2 Ball

Link SECTION

SECTION

1 Section Shell  
2 Section Solid

Read NewKwd NewEntry  
Cancel Done

Link MAT

MAT

1 Steel  
2 Rigid

Read NewKwd NewEntry  
Cancel Done





# Assign Material and Section Properties to Parts

1. Switch to [2 Ball] part

2. TITLE: Ball

3. Press [•] next to [SECID]

4. "Link SECTION" popup window will appear

5. Double click on [2 Section Solid]

6. Press [•] next to [MID]

7. "Link MAT" popup window will appear

8. Double click on [2 Rigid]

9. Accept

10. Done

Keyword Input Form

NewID Draw RefBy Pick Add Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 T01\_PADT\_LS-DYNA\_BallOnPlate.k) Setting

\*PART\_(TITLE) (2)

|   | TITLE | PID | SECID | MID | EOSID | HGID | GRAV | ADPOPT | TMID |
|---|-------|-----|-------|-----|-------|------|------|--------|------|
| 1 | Ball  |     |       |     |       |      |      |        |      |
| 2 |       | 2   | 2     | 2   | 0     | 0    | 0    | 0      | 0    |

COMMENT:

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

1 Plate  
2 Ball

Link SECTION

SECTION

1 Section Shell  
2 Section Solid

Read NewKwd NewEntry  
Cancel Done

Link MAT


MAT

1 Steel  
2 Rigid

Read NewKwd NewEntry  
Cancel Done



# Steps

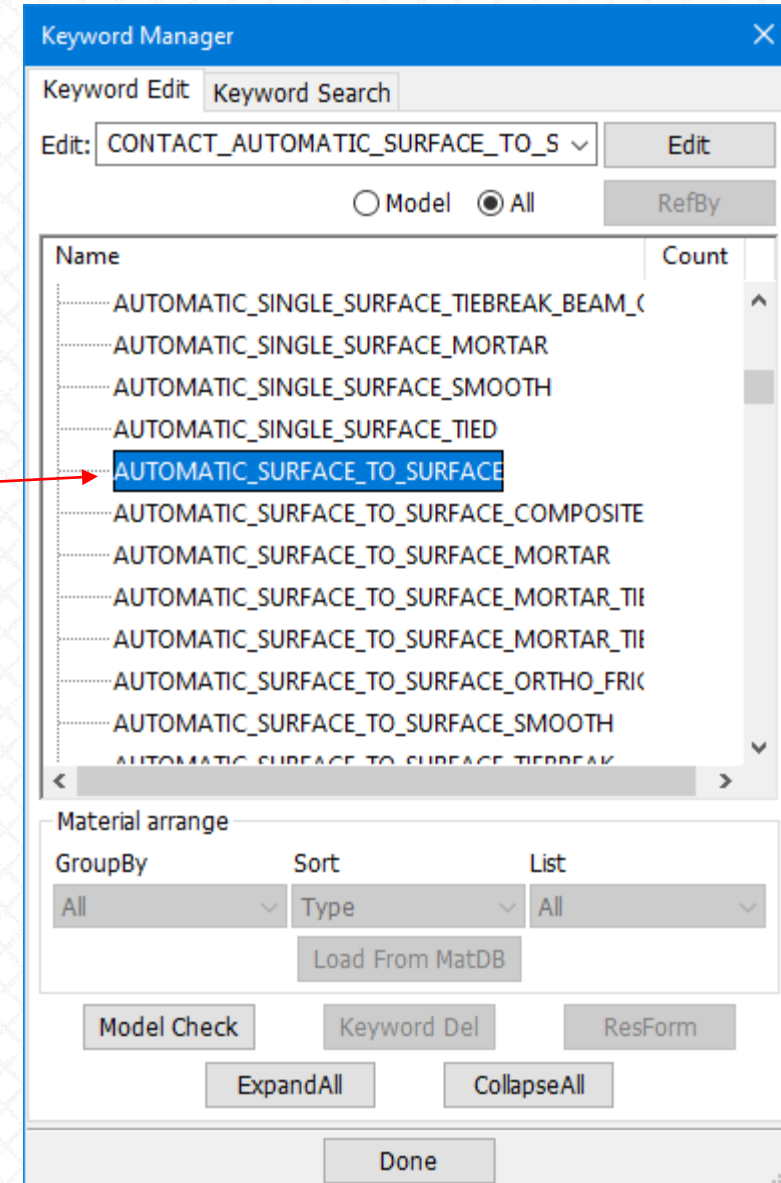
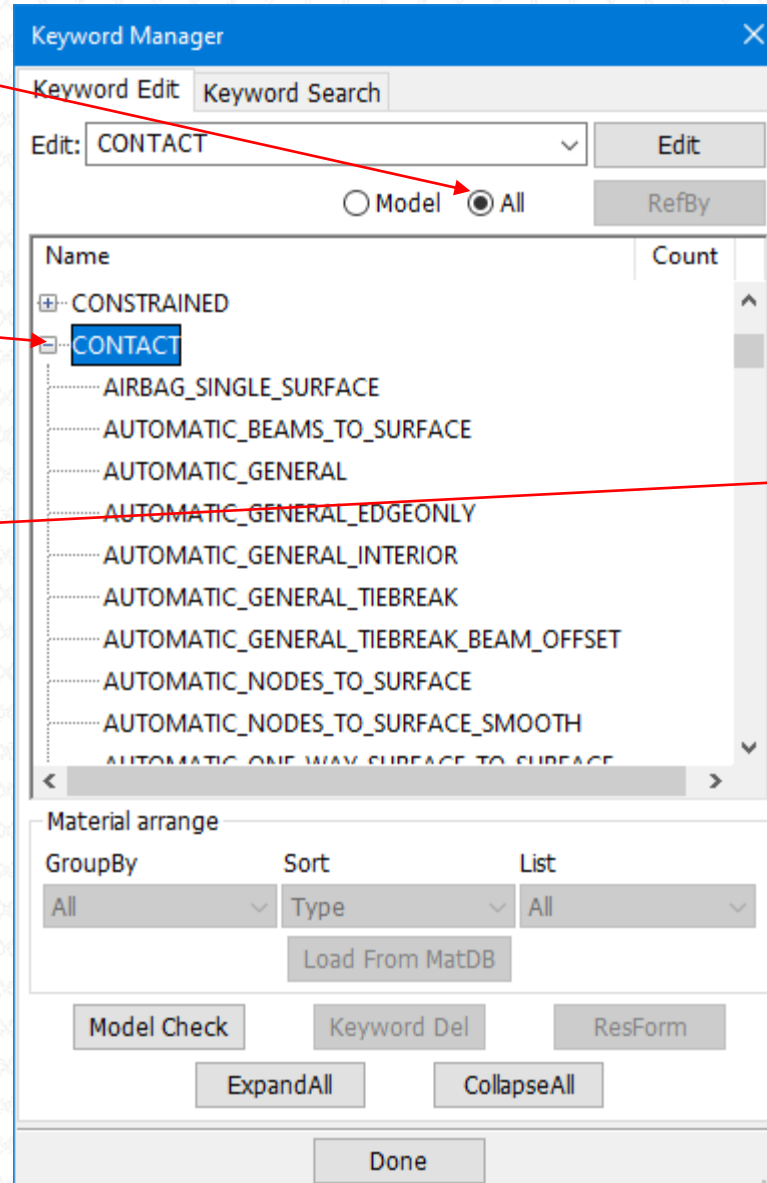
| Step #   | Description                                     |   |
|--|---|---|
| 1  | Create Geometry and Mesh                        |   |
| 2  | Boundary Conditions                             |   |
| 3  | Material Properties                             |   |
| 4  | Section/Element Properties                      |   |
| 5  | Assign Material and Section Properties to Parts |   |
|  6 | Contact   | We will define contact between the Ball and Plate |
| 7  | Initial Velocity                                |   |
| 8  | Analysis Time and Output Controls               |   |
| 9  | Submit Analysis in LS-Run                       |   |
| 10   | Postprocess results in LS-Prepost               |   |

# Contact

1. Switch radio button from “Model” to “All”

2. Expand [CONTACT]

3. Scroll down and double click on [AUTOMATIC\_SURFACE\_TO\_SURFACE]



# Contact

1. Click "NewID"
2. TITLE: Contact Ball to Plate
3. SSTYP: 3
4. MSTYP: 3
5. Press [•] next to [SSID]
6. "Link PART" popup window will appear
7. Double click on [2 Ball]
8. Press [•] next to [MSID]
9. "Link PART" popup window will appear
10. Double click on [1 Plate]
11. Accept
12. Done

Keyword Input Form

NewID Draw Pick Add Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 T01\_PADT\_LS\_DYNA\_BallOnPlate.k) Setting

\*CONTACT\_AUTOMATIC\_SURFACE\_TO\_SURFACE\_(ID/TITLE/MPP)\_(THERMAL) (1)

| CID | TITLE                 | MPP1                     | MPP2                     |
|-----|-----------------------|--------------------------|--------------------------|
| 1   | Contact Ball to Plate | <input type="checkbox"/> | <input type="checkbox"/> |

| 2 | IGNORE | BCKET | LCBCKT | MS2TRK | INITITR | PARMAX | UNUSED | CPARM8 |
|---|--------|-------|--------|--------|---------|--------|--------|--------|
|   | 0      | 200   |        | 3      | 2       | 1.0005 |        | 0      |

| 3 | UNUSED | CHKSEGS | FENSEF | CRPABLE |
|---|--------|---------|--------|---------|
|   |        | 0       | 1.0    | 0       |

| 4 | SSID                     | MSID                     | SSTYP | MSTYP | SBOXID | MBOXID | SPR | MPR |
|---|--------------------------|--------------------------|-------|-------|--------|--------|-----|-----|
|   | <input type="checkbox"/> | <input type="checkbox"/> | 3     | 3     | 0      | 0      | 0   | 0   |

| 5 | FS  | FD  | DC  | VC  | VDC | PENCHK | BT  | DT        |
|---|-----|-----|-----|-----|-----|--------|-----|-----------|
|   | 0.0 | 0.0 | 0.0 | 0.0 | 0.0 | 0      | 0.0 | 1.000e+20 |

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

Link PART

PART

- 1 Plate
- 2 Ball

Read NewKwd NewEntry

Cancel Done



# Steps

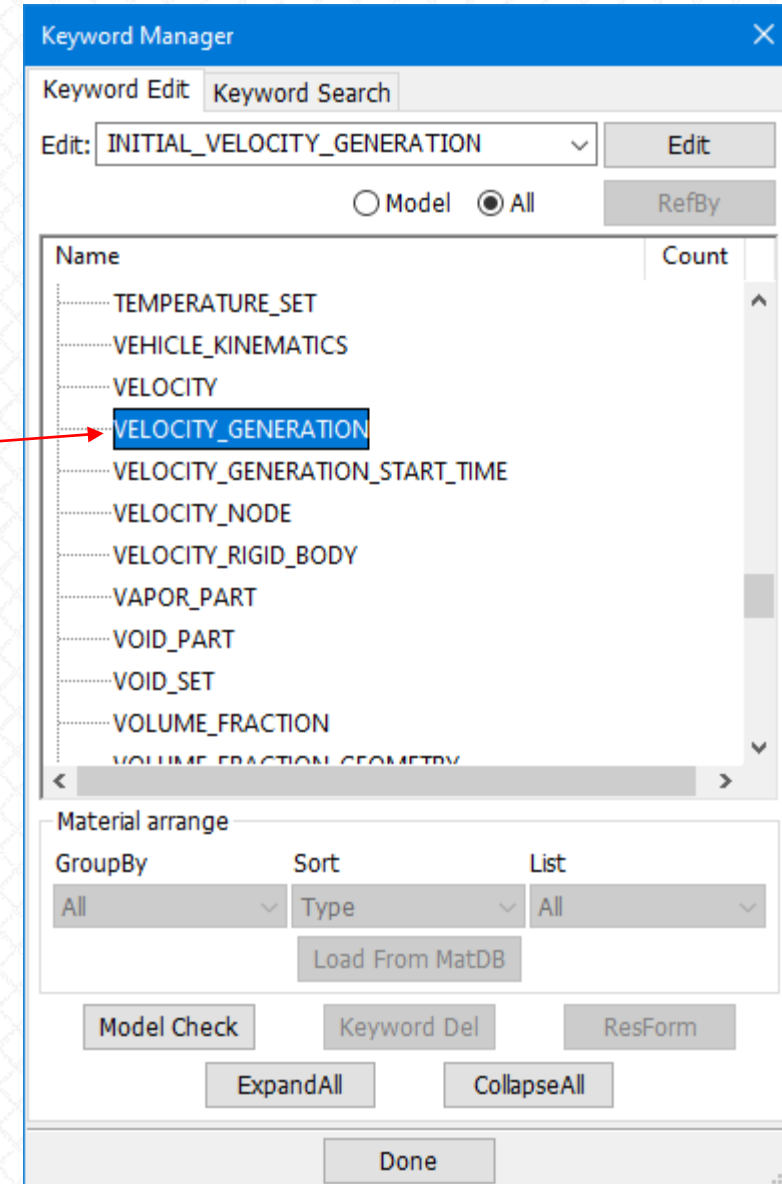
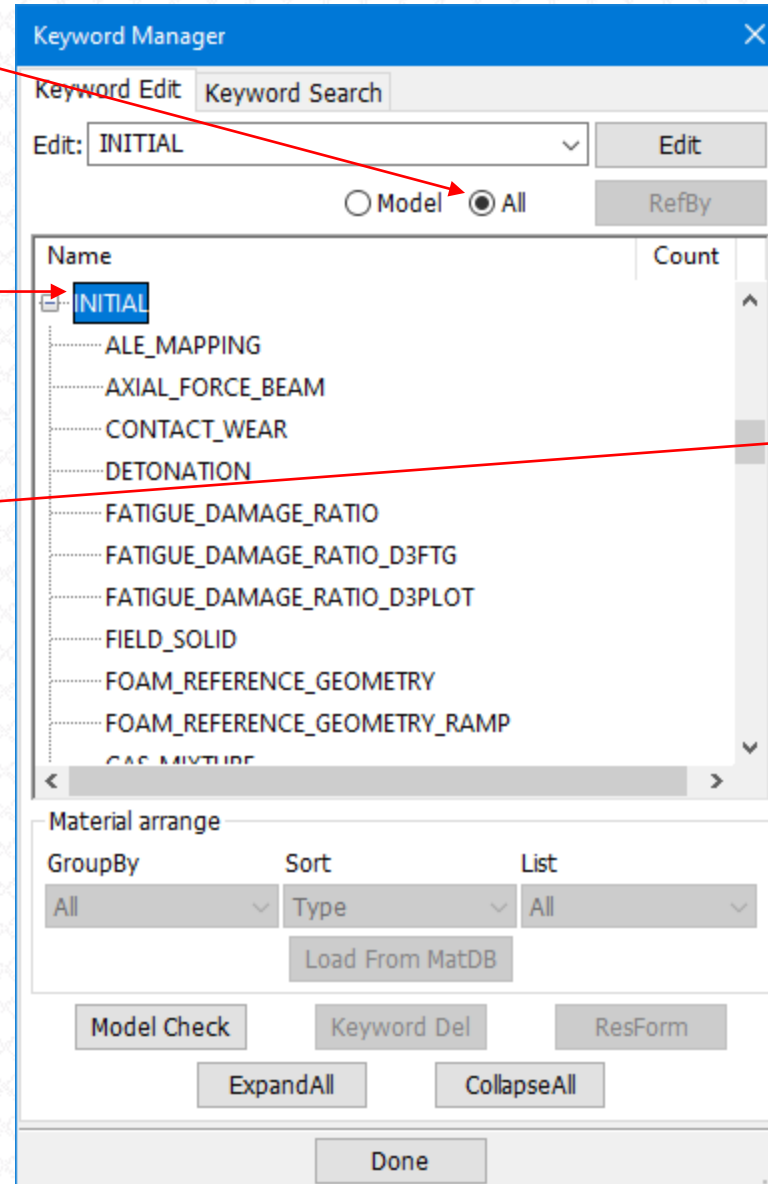
| Step # | Description                                     |   |
|--------|---|---|
| 1      | Create Geometry and Mesh                        |   |
| 2      | Boundary Conditions                             |   |
| 3      | Material Properties                             |   |
| 4      | Section/Element Properties                      |   |
| 5      | Assign Material and Section Properties to Parts |   |
| 6      | Contact   |   |
| 7      | Initial Velocity                                | We will assign initial velocity to the Ball |
| 8      | Analysis Time and Output Controls               |   |
| 9      | Submit Analysis in LS-Run                       |   |
| 10     | Postprocess results in LS-Prepost               |   |

# Initial Velocity

1. Select radio button "All" if it's not selected

2. Expand [INITIAL]

3. Scroll down and double click on [VELOCITY\_GENERATION]



# Initial Velocity

1. Click "NewID"
2. STYP: 2
3. Press [•] next to [NSID/PID]
4. "Link PART" popup window will appear
5. Double click on [2 Ball]
6. VZ = -10
7. Accept
8. Done

Keyword Input Form

NewID Draw Pick Add Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 T01\_PADT\_LS-DYNA\_BallOnPlate.k) Setting

\*INITIAL\_VELOCITY\_GENERATION (1)

|   | NSID/PID | STYP | OMEGA | VX  | VY  | VZ          | IVATN | ICID |
|---|----------|------|-------|-----|-----|-------------|-------|------|
| 1 | 2        | 2    | 0.0   | 0.0 | 0.0 | -10.0000000 | 0     | 0    |
| 2 |          |      |       |     |     |             |       |      |

COMMENT:

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

Link PART

PART

- 1 Plate
- 2 Ball

Read NewKwd NewEntity

Cancel Done



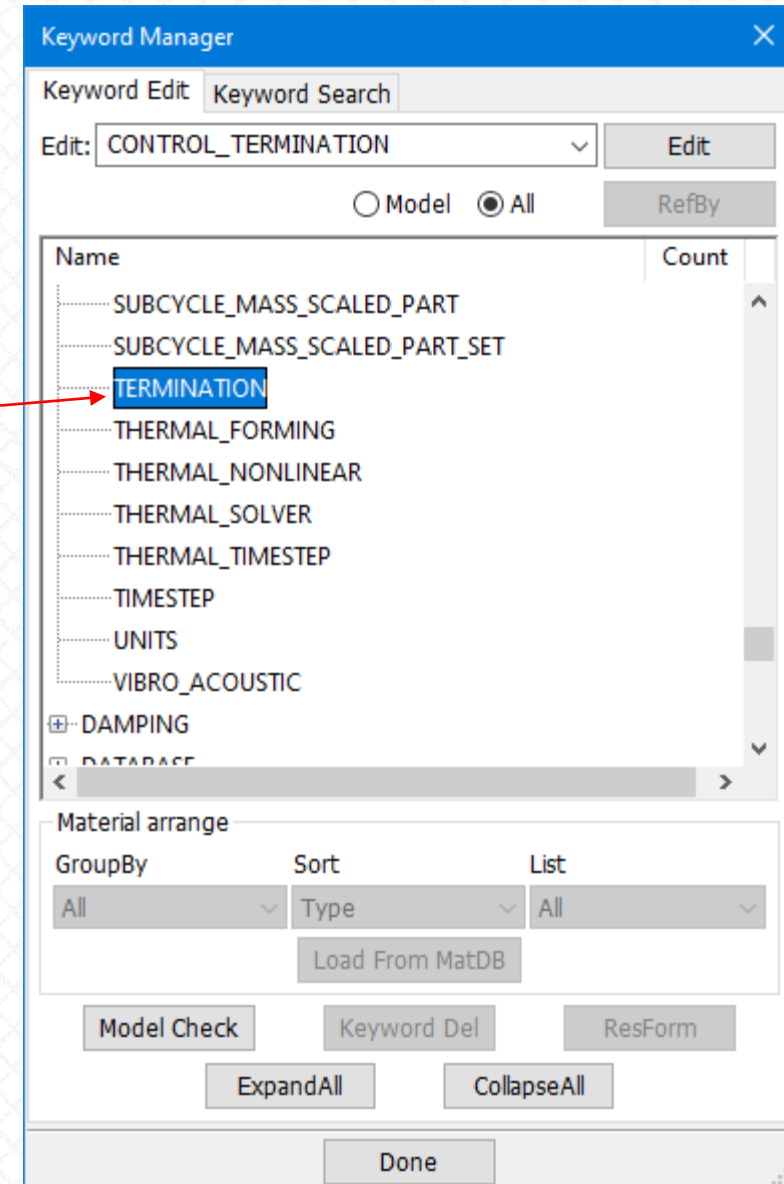
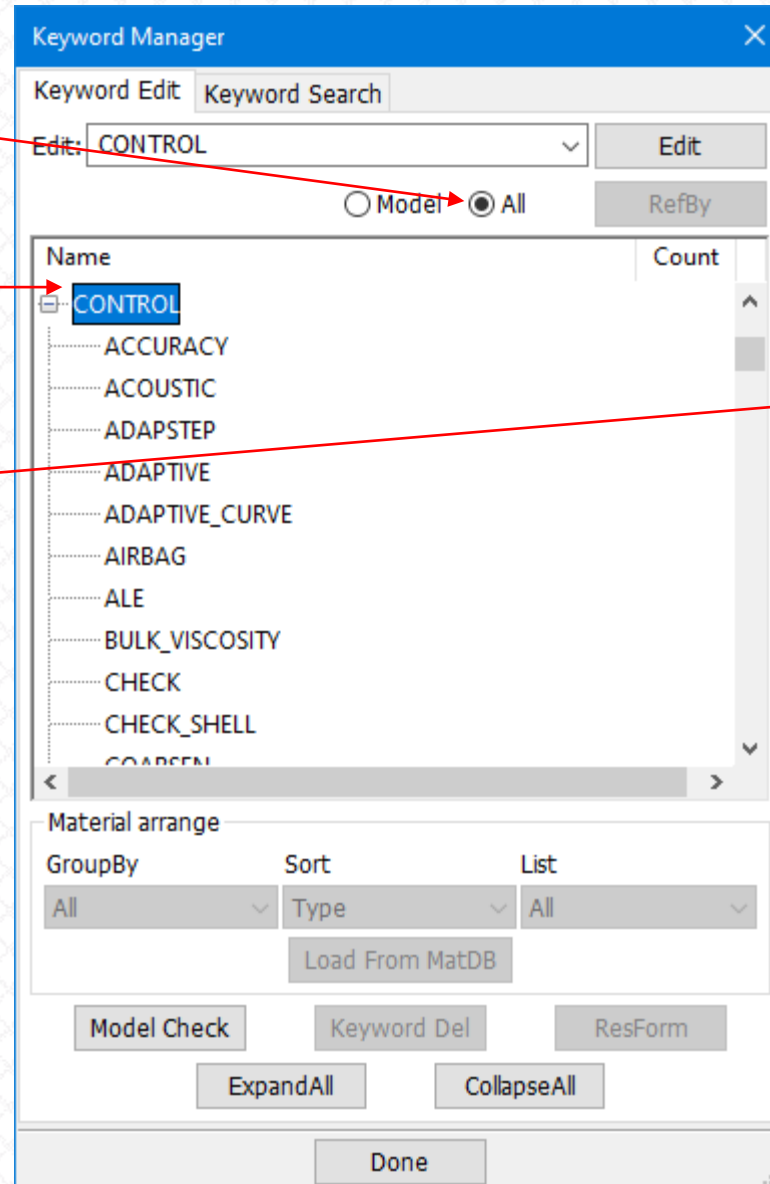
# Steps

| Step # | Description                                     |  |
|--------|---|--|
| 1      | Create Geometry and Mesh                        |  |
| 2      | Boundary Conditions                             |  |
| 3      | Material Properties                             |  |
| 4      | Section/Element Properties                      |  |
| 5      | Assign Material and Section Properties to Parts |  |
| 6      | Contact   |  |
| 7      | Initial Velocity                                |  |
| 8      | Analysis Time and Output Controls               | We will specify total duration of simulation and output controls |
| 9      | Submit Analysis in LS-Run                       |  |
| 10     | Postprocess results in LS-Prepost               |  |



# Analysis Time

1. Select radio button "All" if it's not selected
2. Expand [CONTROL]
3. Scroll down and double click on [TERMINATION]



# Analysis Time

1. ENDTIM: 10

Note:

ENDTIM = 10 means that we are modeling 10 ms of simulation where Ball impacts a Plate, and bounces back (we'll see that 10 ms is sufficient for that)

2. Accept

3. Done

Keyword Input Form

Use \*Parameter  Comment

(Subsys: 1 T01\_PADT\_LS-DYNA\_BallOnPlate.k)

\*CONTROL\_TERMINATION (1)

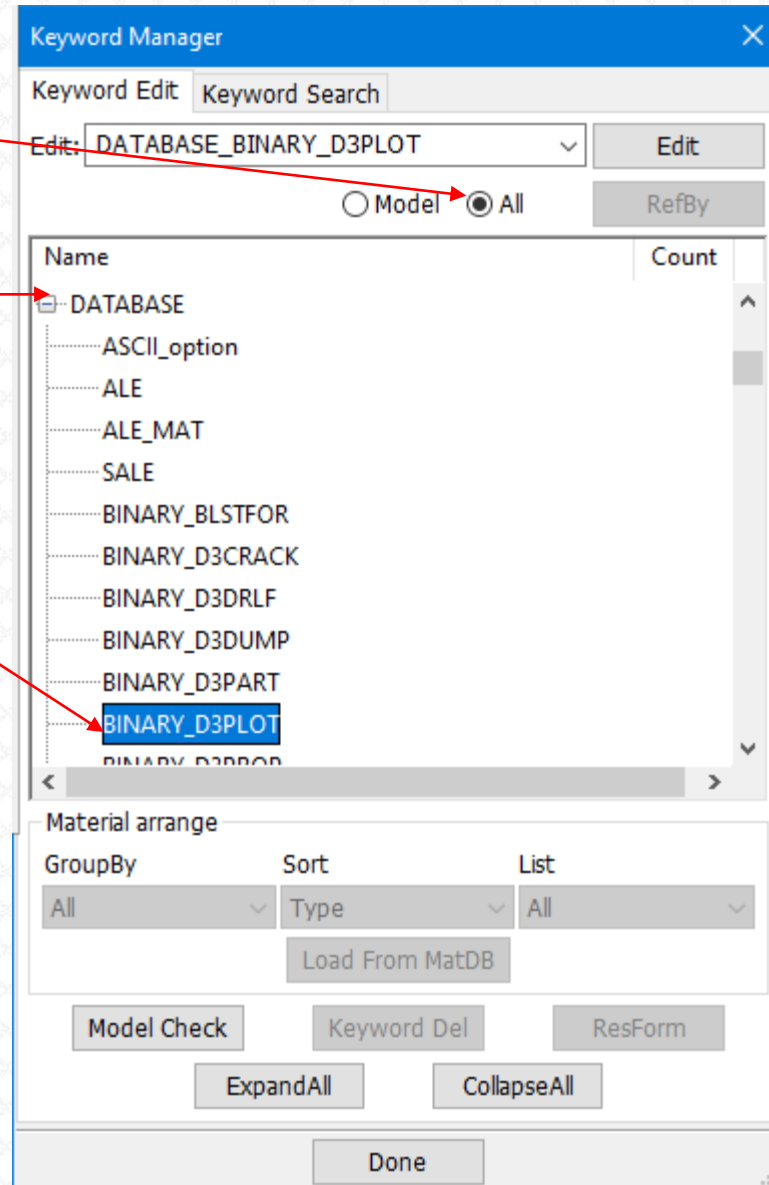
|   | <u>ENDTIM</u> | <u>ENDCYC</u> | <u>DTMIN</u> | <u>ENDENG</u> | <u>ENDMAS</u> | <u>NOSOL</u> |
|---|---------------|---------------|--------------|---------------|---------------|--------------|
| 1 | 10.0000000    | 0             | 0.0          | 0.0           | 1.000e+08     | 0            |

COMMENT:



# Output Controls

1. Select radio button "All" if it's not selected
2. Expand [DATABASE]
3. Double click on [BINARY\_D3PLOT]



# Output Controls

1. DT: 0.1

Note:

DT defines the time interval between output states.

In our case, there will be at least 100 states (because  $10 \text{ ms} / 0.1 \text{ ms} = 100$ ) with results written to disc.

2. Accept

3. Done

Keyword Input Form

Pick Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 T01\_PADT\_LS-DYNA\_BallOnPlate.k) Setting

\*DATABASE\_BINARY\_D3PLOT (1)

|   |           |      |        |        |        |      |
|---|-----------|------|--------|--------|--------|------|
| 1 | DT        | LCDT | BEAM   | NPLTC  | PSETID |      |
|   | 0.1000000 | 0    | 0      | 0      | 0      |      |
| 2 | IOOPT     | RATE | CUTOFF | WINDOW | TYPE   | PSET |
|   | 0         | 0.0  | 0.0    | 0.0    | 0      | 0    |

COMMENT:

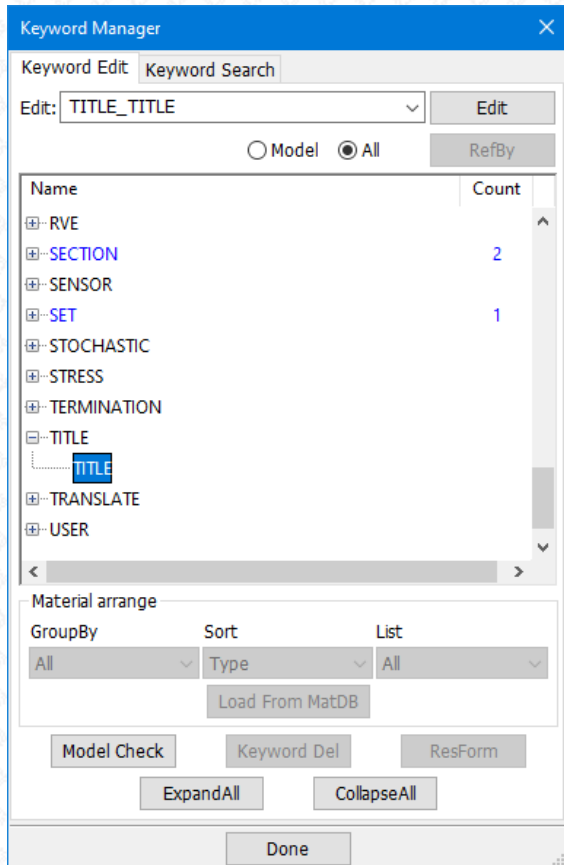
Total Card: 1 Smallest ID: 1 Largest ID: 1 Total deleted card: 0



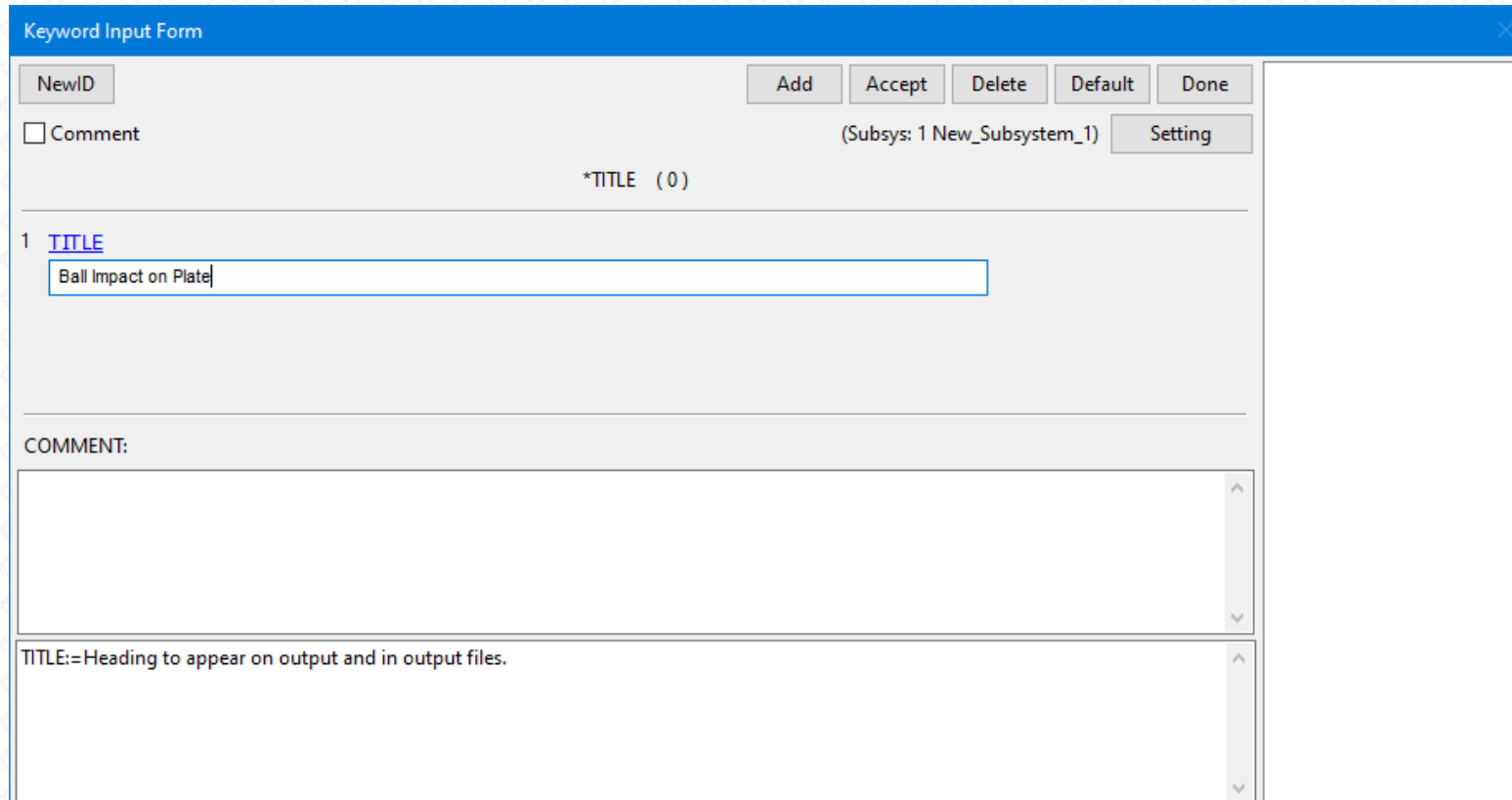
# Title

This step is optional, but a good practice.

1. Select radio button "All" if it's not selected
2. Expand [TITLE]
3. Double click on [TITLE]



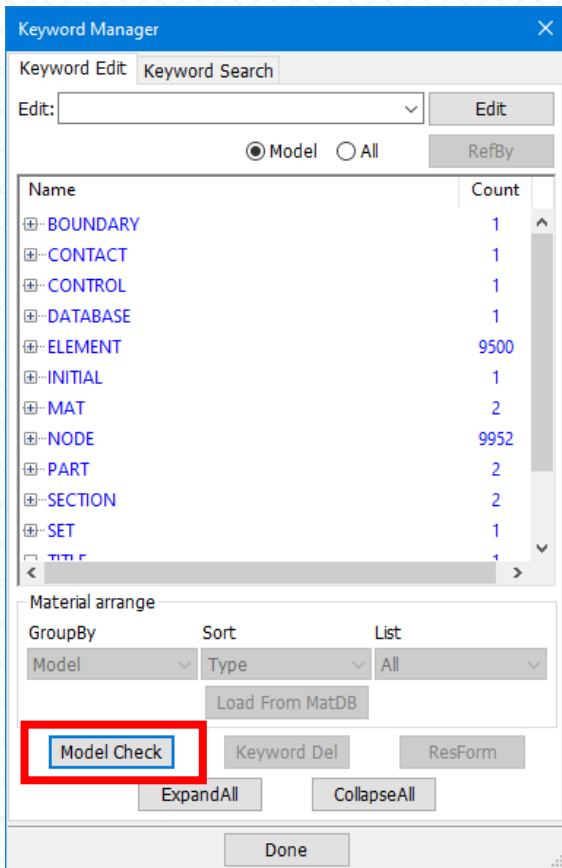
4. TITLE: Ball Impact on Plate
5. Accept
6. Done



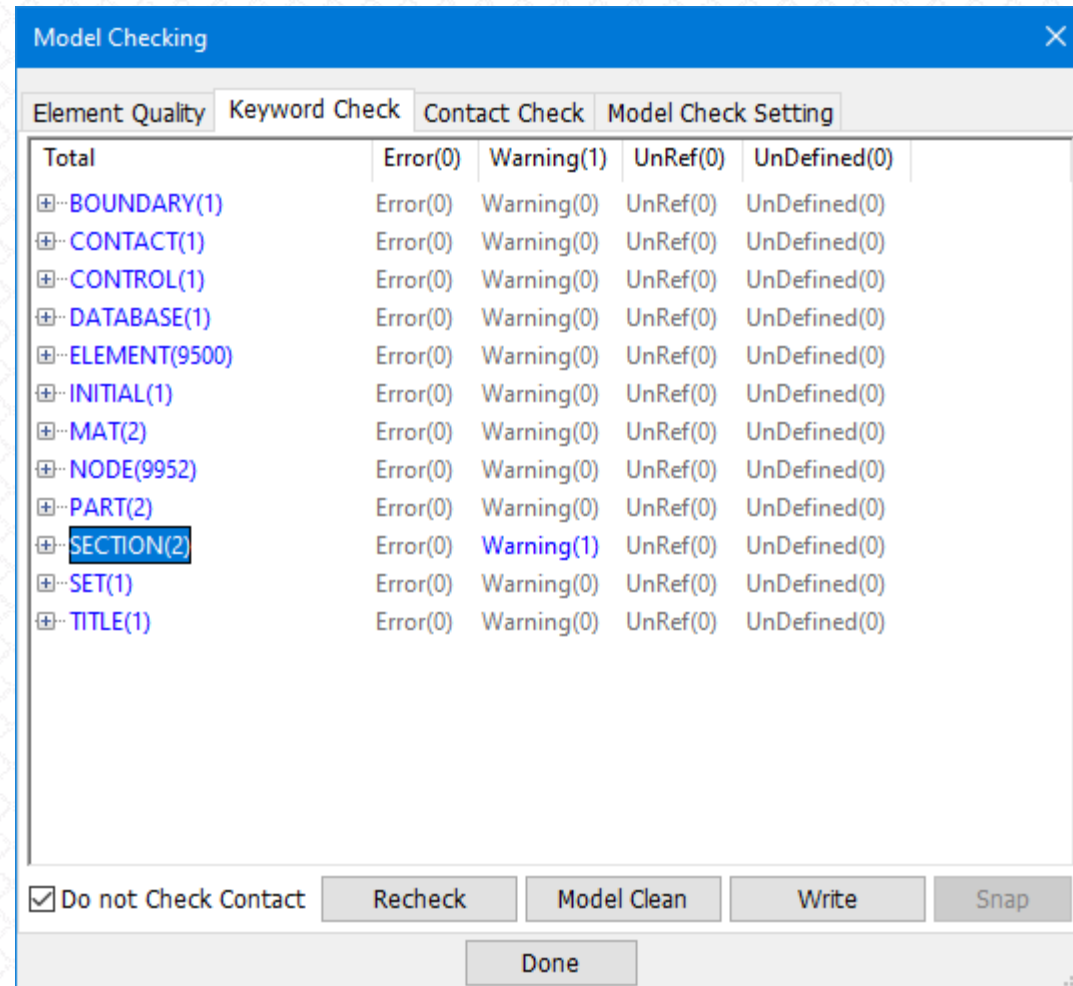
# Model Check

This step is optional, but a good practice making sure there are no errors before submitting a job in LS-Run.

1. Click “Model Check” button

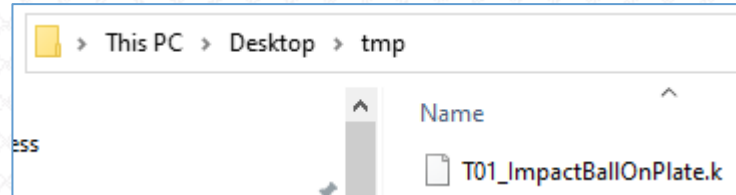


2. “Model Checking” popup window will appear
3. Make sure there are no Errors
4. Read Warnings, UnRef, UnDefined messages
5. Done



# Save

1. [Ctrl + S] to save .k file



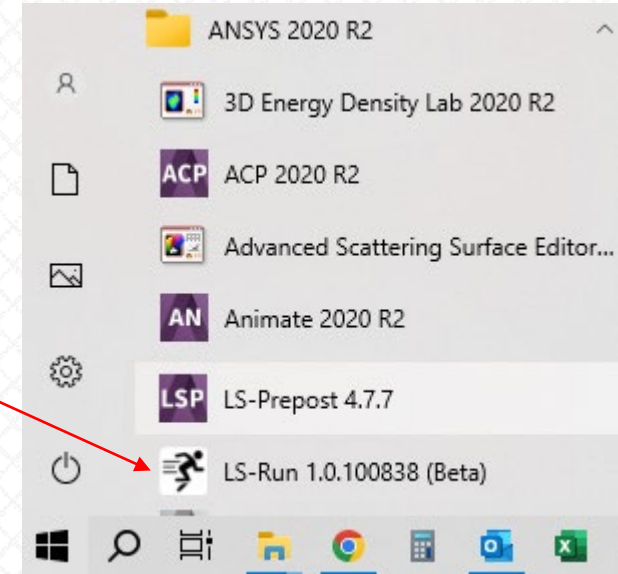
# Steps

| Step # | Description                                     |  |
|--------|---|--|
| 1      | Create Geometry and Mesh                        |  |
| 2      | Boundary Conditions                             |  |
| 3      | Material Properties                             |  |
| 4      | Section/Element Properties                      |  |
| 5      | Assign Material and Section Properties to Parts |  |
| 6      | Contact   |  |
| 7      | Initial Velocity                                |  |
| 8      | Analysis Time and Output Controls               |  |
| 9      | Submit Analysis in LS-Run                       | We will submit a job (created .k file) in LS-Run |
| 10     | Postprocess results in LS-Prepost               |  |



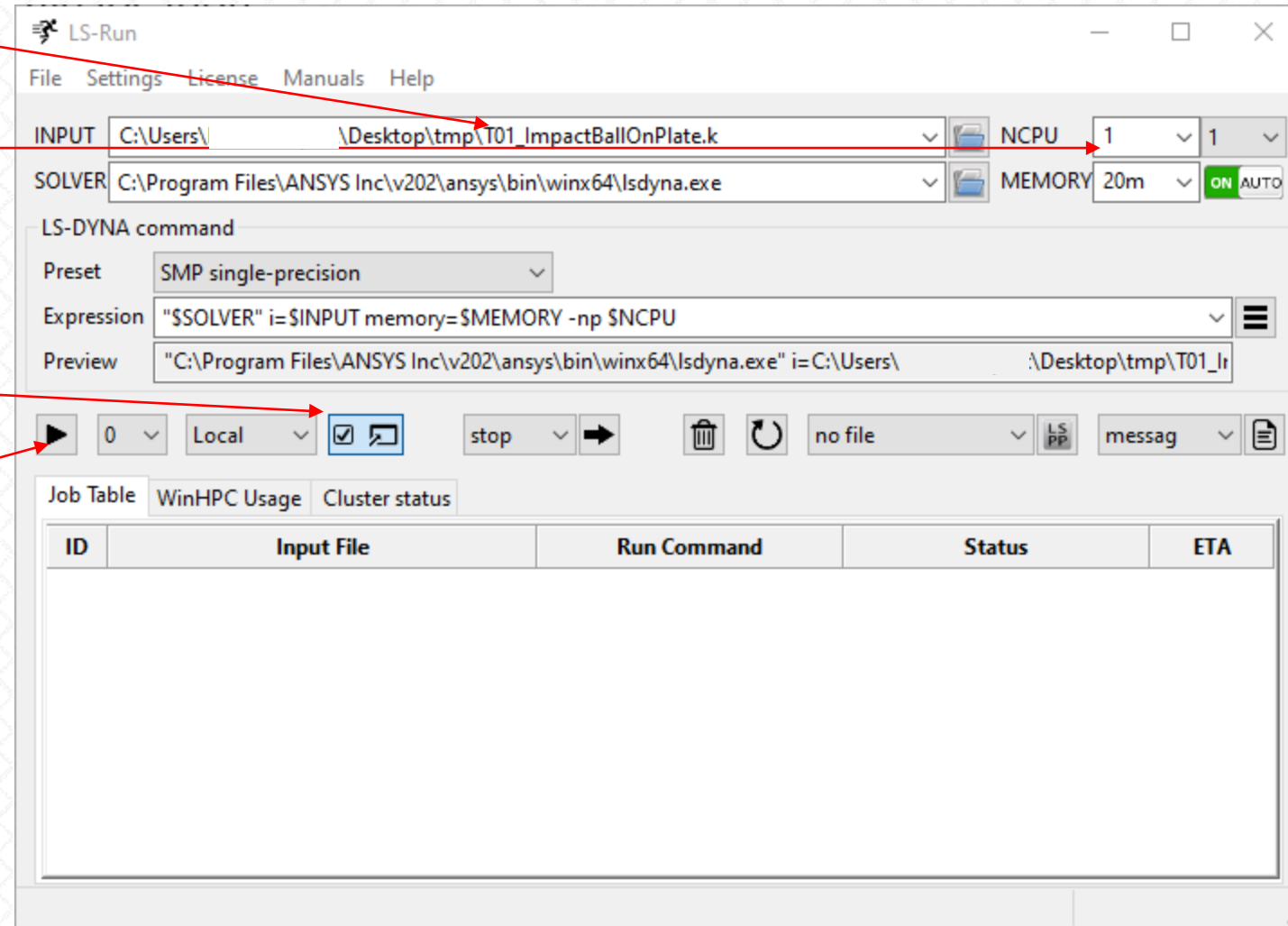
# Submit Analysis in LS-Run

1. Start > ANSYS 2020 R2 > LS-Run



# Submit Analysis in LS-Run

1. INPUT: browse to .k file location
2. NCPU: 1 (default)
  - a) Change to desired/available number of CPUs, if you'd like to speed up computation time
3. Check box to start LS-DYNA in a command prompt window.
  - a) Command prompt will display useful information during computations
4. Hit "Run"



# Submit Analysis in LS-Run

## 1. Command prompt window will display miscellaneous information

```
C:\Windows\SYSTEM32\cmd.exe
initial kinetic energy = 0.25289240E+02

The LS-DYNA time step size should not exceed 1.792E-04
to avoid contact instabilities. If the step size is
bigger then scale the penalty of the offending surface.

Memory required to begin solution      :    727K
Additional dynamically allocated memory:   1018K
Total:                                  1745K

initialization completed
  1 t 0.0000E+00 dt 6.68E-04 flush i/o buffers      09/06/23 20:05:36
  1 t 0.0000E+00 dt 6.68E-04 write d3plot file     09/06/23 20:05:36
cpu time per zone cycle.....          0 nanoseconds
average cpu time per zone cycle....    0 nanoseconds
average clock time per zone cycle..    143 nanoseconds

estimated total cpu time                =    0 sec (    0 hrs  0 mins)
estimated cpu time to complete          =    0 sec (    0 hrs  0 mins)
estimated total clock time              =   20 sec (    0 hrs  0 mins)
estimated clock time to complete        =   20 sec (    0 hrs  0 mins)
termination time                        = 1.000E+01
 150 t 9.9518E-02 dt 6.68E-04 write d3plot file   09/06/23 20:05:36
 300 t 1.9970E-01 dt 6.68E-04 write d3plot file   09/06/23 20:05:36
 450 t 2.9980E-01 dt 6.67E-04 write d3plot file   09/06/23 20:05:37
 600 t 3.9979E-01 dt 6.67E-04 write d3plot file   09/06/23 20:05:37
 750 t 4.9982E-01 dt 6.67E-04 write d3plot file   09/06/23 20:05:37
 900 t 5.9988E-01 dt 6.67E-04 write d3plot file   09/06/23 20:05:37
1050 t 6.9994E-01 dt 6.67E-04 write d3plot file   09/06/23 20:05:37
```

## 2. LS-Run window will display % completion (estimate)

The screenshot shows the LS-Run application window. At the top, there are menu options: File, Settings, License, Manuals, Help. Below the menu, there are input fields for INPUT (C:\Users\...\Desktop\tmp\T01\_ImpactBallOnPlate.k), SOLVER (C:\Program Files\ANSYS Inc\v202\ansys\bin\winx64\lsdyna.exe), NCPU (1), and MEMORY (20m). The LS-DYNA command section shows a Preset of 'SMP single-precision' and an Expression: "\$SOLVER" i=\$INPUT memory=\$MEMORY -np \$NCPU. The Preview shows the full command line. Below this is a control bar with a play button, a dropdown for '0', 'Local', a checkbox, a 'stop' button, a refresh button, a dropdown for 'd3plot', and a 'messag' button. At the bottom, there is a 'Job Table' with tabs for 'WinHPC Usage' and 'Cluster status'. The Job Table has columns for ID, Input File, Run Command, Status, and ETA. The first row shows ID 1, Input File C:\Users\...\Desktop\tmp\T01\_Ir, Run Command "C:\Program Files\ANSYS Inc\v202\ansys\bin\winx64\lsdyna.exe" i=C:\Users\bronislav.piak\Desktop\tmp\T01\_Ir, Status Running... 56%, and ETA 4s. A red arrow points from the text above to the 'Status' column of the Job Table.

| ID | Input File                      | Run Command  | Status         | ETA |
|----|---------------------------------|--|----------------|-----|
| 1  | C:\Users\...\Desktop\tmp\T01_Ir | "C:\Program Files\ANSYS Inc\v202\ansys\bin\winx64\lsdyna.exe" i=C:\Users\bronislav.piak\Desktop\tmp\T01_Ir | Running... 56% | 4s  |

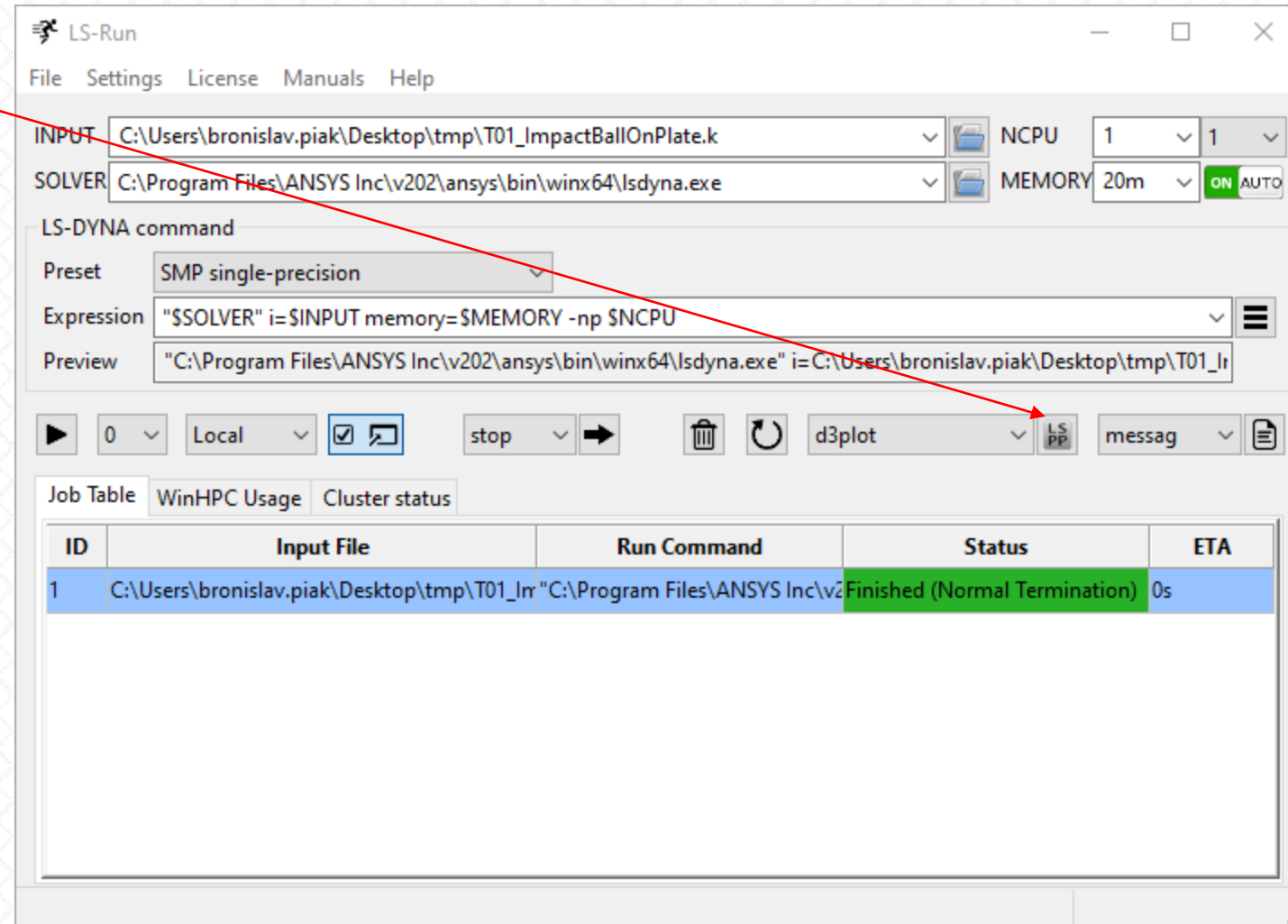


# Steps

| Step # | Description                                     |  |
|--------|---|--|
| 1      | Create Geometry and Mesh                        |  |
| 2      | Boundary Conditions                             |  |
| 3      | Material Properties                             |  |
| 4      | Section/Element Properties                      |  |
| 5      | Assign Material and Section Properties to Parts |  |
| 6      | Contact   |  |
| 7      | Initial Velocity                                |  |
| 8      | Analysis Time and Output Controls               |  |
| 9      | Submit Analysis in LS-Run                       |  |
| 10     | Postprocess results in LS-Prepost               | We will plot Von Misses stress and animate results |

# Postprocess Results in LS-Prepost

1. After LS-DYNA finished computations
2. Click “LS-PP” button to launch LS-Prepost and load the results file (d3plot) automatically



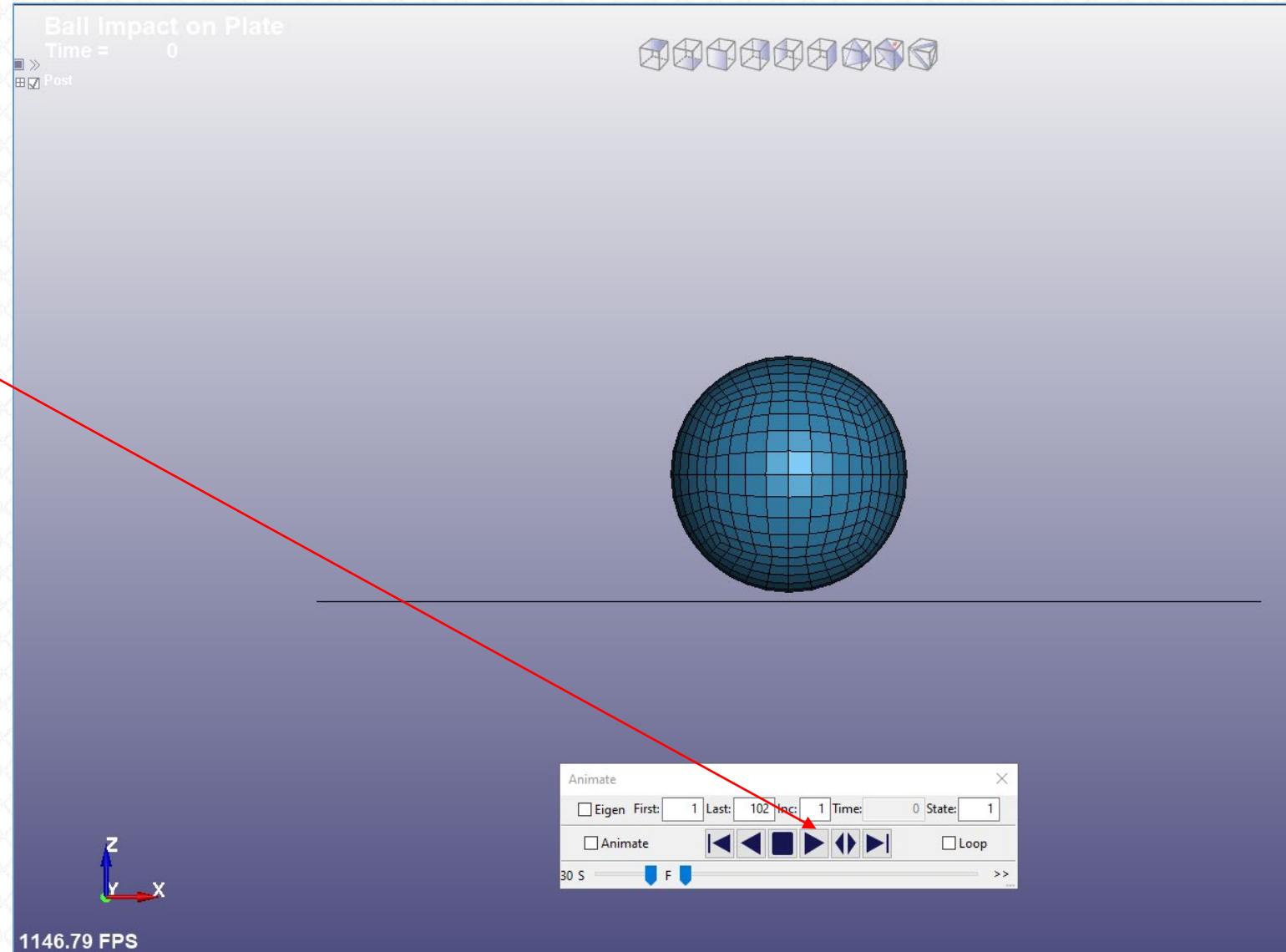
The screenshot shows the LS-Run application window. The interface includes a menu bar (File, Settings, License, Manuals, Help), input fields for INPUT and SOLVER, and a section for LS-DYNA command configuration. The command section includes a Preset dropdown (SMP single-precision), an Expression field, and a Preview field. A toolbar at the bottom contains various control buttons, including a play button, a stop button, and a button labeled "LS PP" which is highlighted with a red arrow. Below the toolbar is a Job Table with columns for ID, Input File, Run Command, Status, and ETA.

| ID | Input File                                 | Run Command                    | Status                        | ETA |
|----|--|--------------------------------|-------------------------------|-----|
| 1  | C:\Users\bronislav.piak\Desktop\tmp\T01_In | "C:\Program Files\ANSYS Inc\v2 | Finished (Normal Termination) | 0s  |

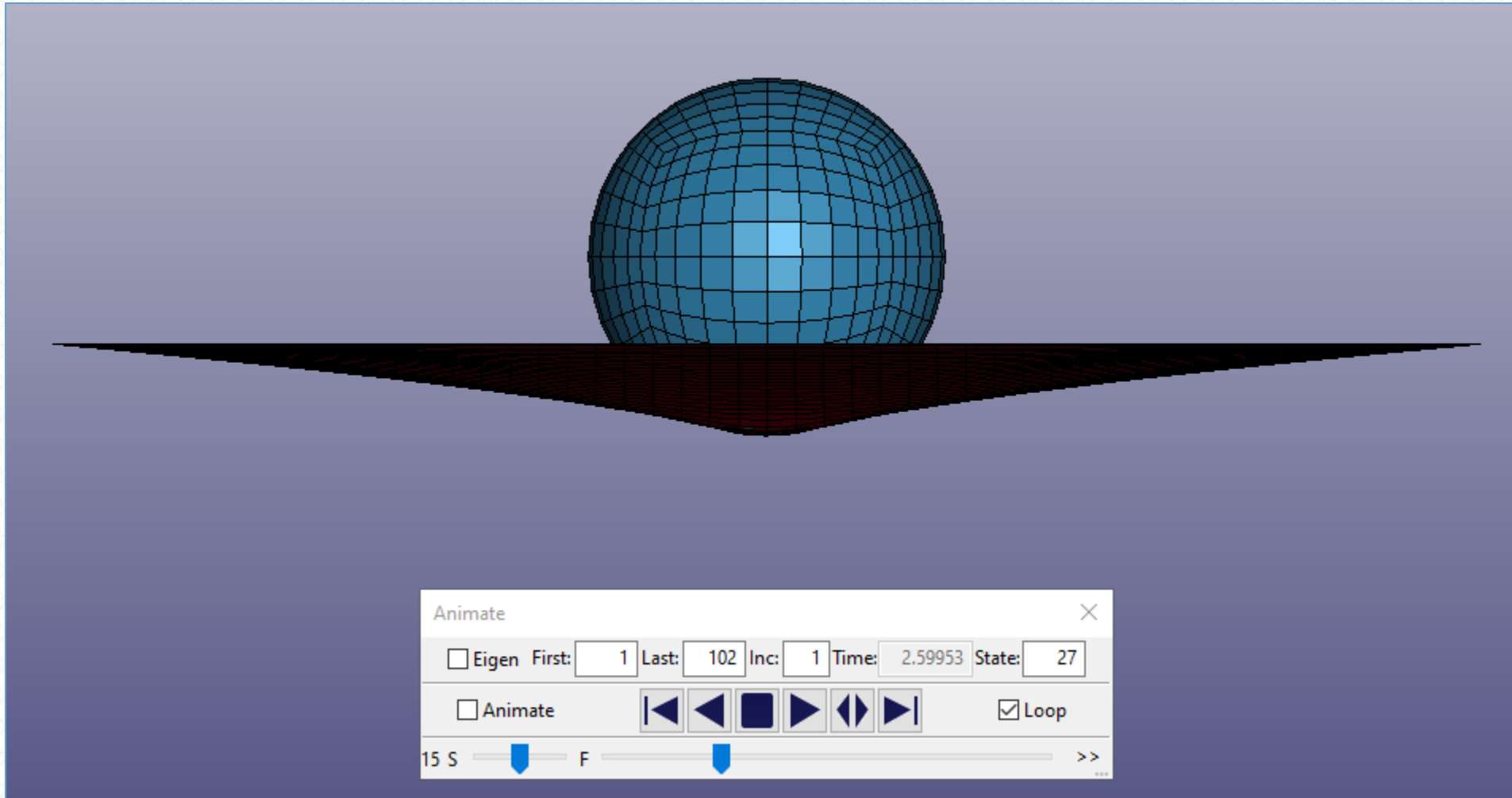


# Postprocess Results in LS-Prepost

1. Click on "Play" button to animate the results

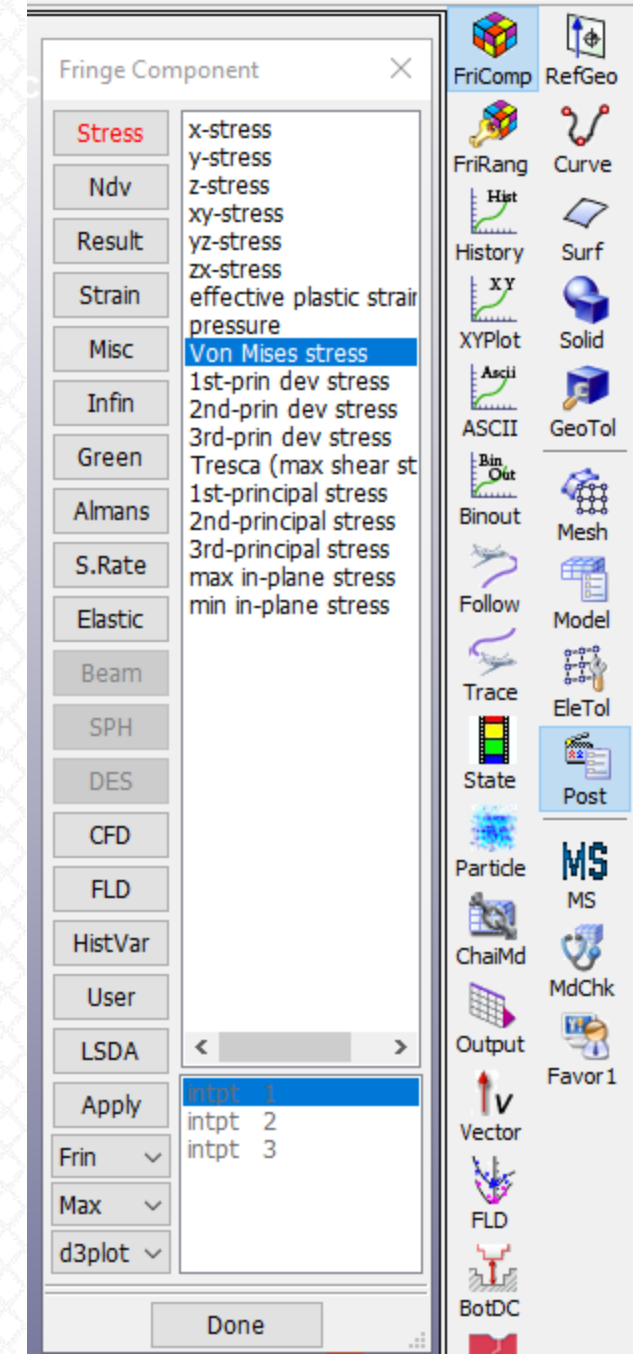


# Postprocess Results in LS-Prepost



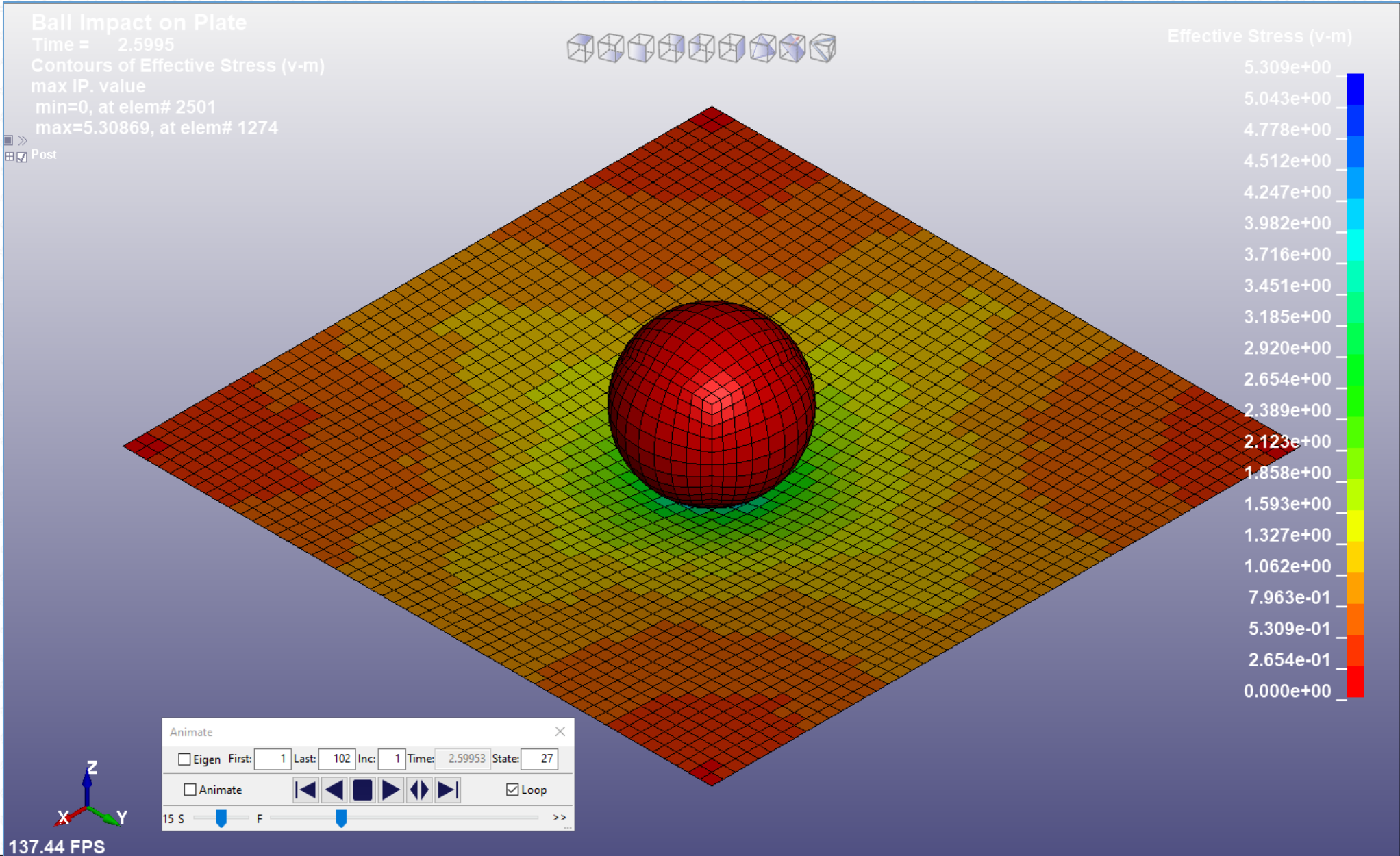
# Postprocess Results in LS-Prepost

1. Post > FriComp
2. In “Fringe Component” popup window:
  - a) Select “Stress” > Von Mises stress





# Postprocess Results in LS-Prepost



# Next Time...

Next time we will learn what is written to a .k file (LS-DYNA input/keyword deck file). How this file is structured, basic most important keywords. Therefore, stay tuned!

```
1  $# LS-DYNA Keyword file created by LS-PrePost(R) V4.7.7 - 17Feb2020
2  $# Created on Sep-11-2023 (08:23:52)
3  *KEYWORD
4  ☐ *TITLE
5  $#
6  Ball Impact on Plate
7  ☐ *CONTROL_TERMINATION
8  $#  endtim  endcyc  dtmin  endeng  endmas  nosol
9  ..... 10.0  0  0.0  0.01.000000E8  0
10 ☐ *DATABASE_BINARY_D3PLOT
11 $#  dt  lcdt  beam  npltc  psetid
12 ..... 0.1  0  0  0  0
13 $#  iopt  rate  cutoff  window  type  pset
14 ..... 0  0.0  0.0  0.0  0  0
15 ☐ *BOUNDARY_SPC_SET
16 $#  nsid  cid  dofx  dofy  dofz  dofrx  dofry  dofrz
17 ..... 1  0  1  1  1  0  0  0
18 ☐ *SET_NODE_LIST_TITLE
19 NODESET (SPC) 1
20 $#  sid  da1  da2  da3  da4  solver
21 ..... 1  0.0  0.0  0.0  0.0 OMECH
22 $#  nid1  nid2  nid3  nid4  nid5  nid6  nid7  nid8
23 ..... 2586  2587  2588  2589  2590  2591  2592  2593
24 ..... 2594  2595  2596  2597  2598  2599  2600  2601
```

