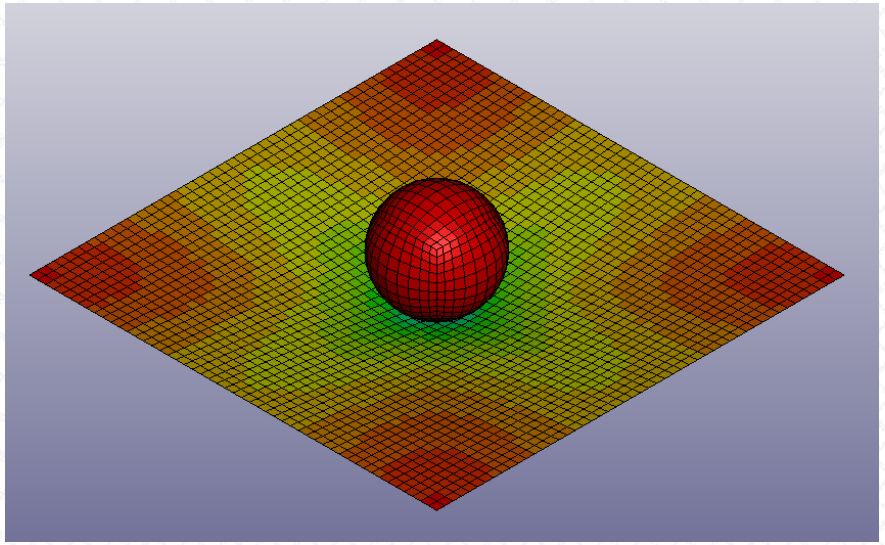
Introduction to Ansys LS-DYNA
Tutorial #1
Ball Impacting a Plate



Bronislav Piak, Simulation Engineer
PADT, Inc.

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

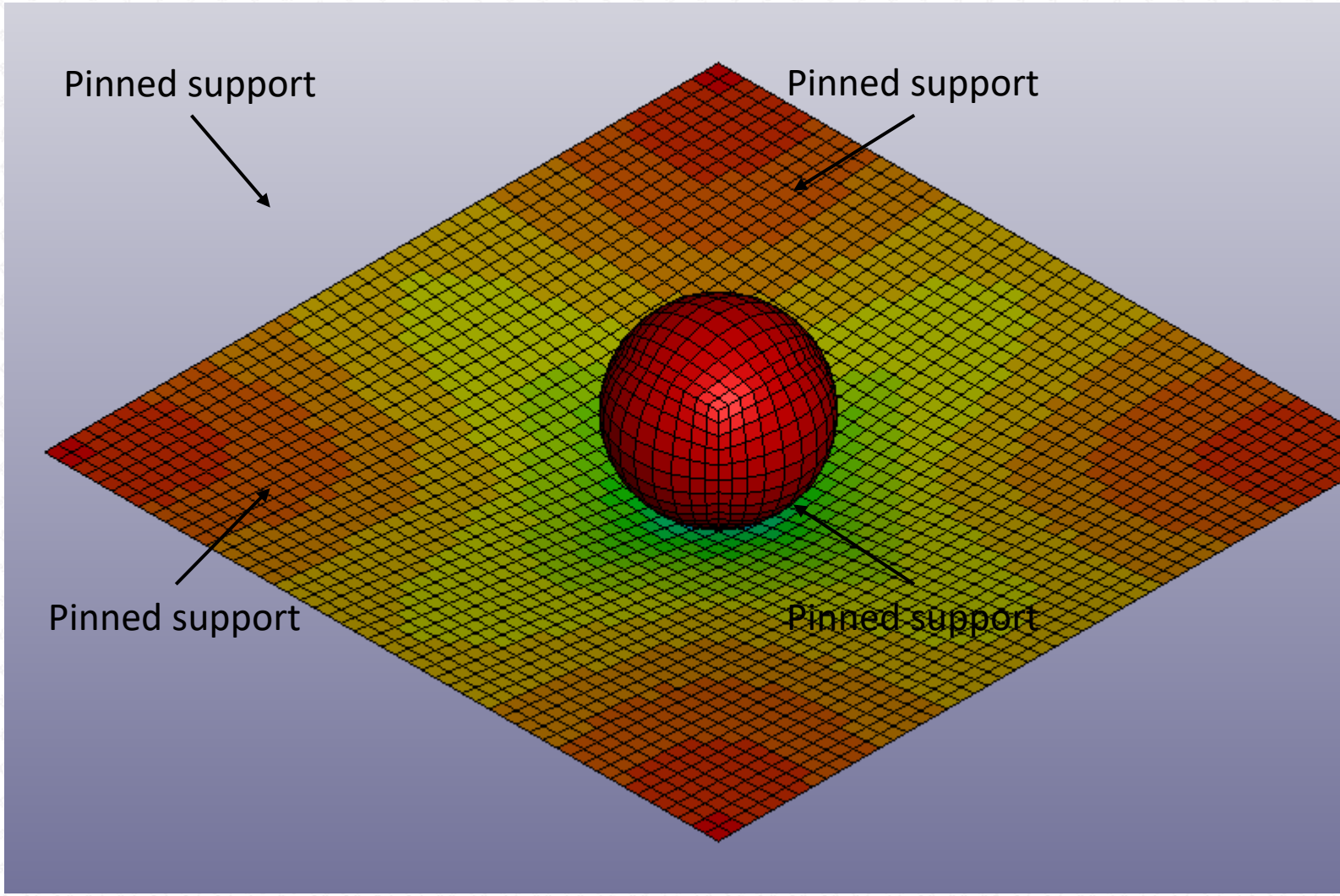
We will model a ball impacting a plate:

Plate
Dimensions: 200mm x 200mm x 0.1mm
Material: Steel
density RO = 7.83e-6 kg/mm3
Elastic Modulus E = 207 GPa
Poisson's ratio PR = 0.3

Ball
Dimensions : radius = 25mm
Material: Steel
density RO = 7.83e-6 kg/mm3
Elastic Modulus E = 207 GPa
Poisson's ratio PR = 0.3
Ball will be modeled as a Rigid Body

Boundary Conditions
Plate is pinned supported (UX=0, UY=0, UZ=0, RX=free, RY=free, RZ=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

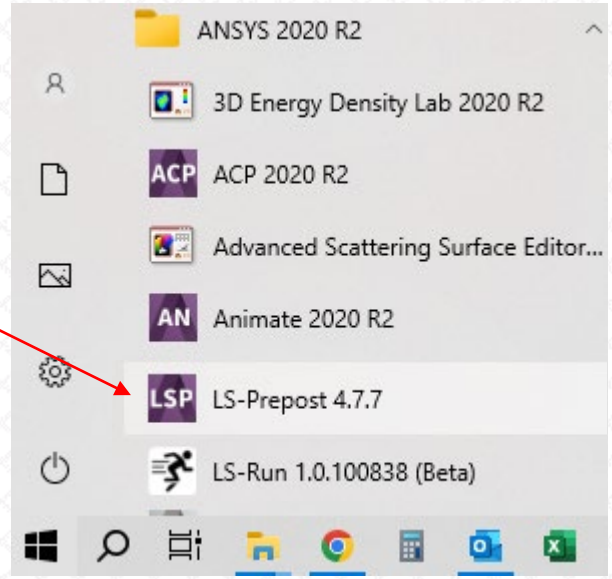
We will use the [kg-mm-ms-kN] units set in this tutorial.

Consistent set of units for Steel

							Consistent set of units for Steel			
MASS	**LENGTH**	**TIME**	**FORCE**	**STRESS**	**ENERGY**		**DENSITY**	**YOUNG's**	**35MPH 56.33KMPH**	**GRAVITY**
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^2	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^2/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^2/mm	mm	s	kgf	kgf/mm^2	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



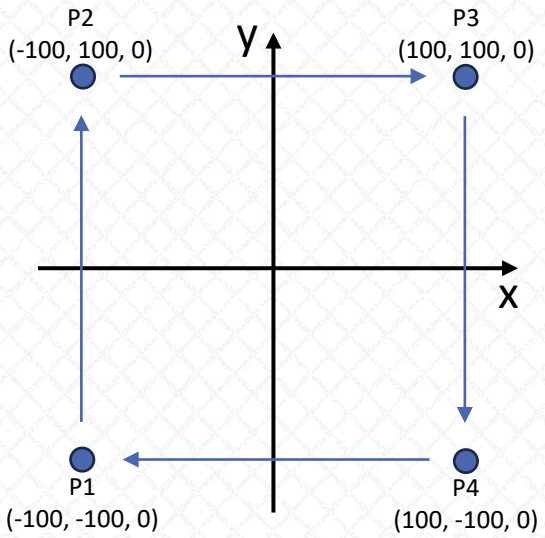
Steps

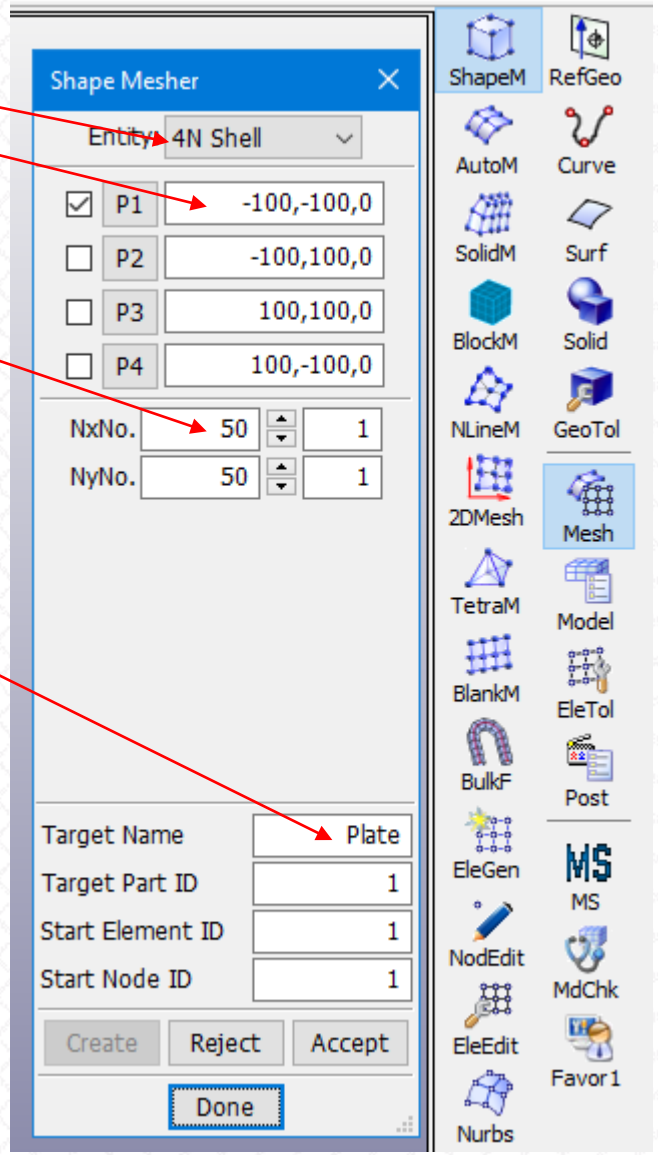
Step #	Desciption	
1	Create Geometry and Mesh	We will create geometry/FE mesh of a Plate and a Ball
2	Boundary Conditions	
3	Material Properties	
4	Section/Element Properties	
5	Assign Material and Section Properties to Parts	
6	Contact	
7	Initial Velocity	
8	Analysis Time and Output Controls	
9	Submit 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

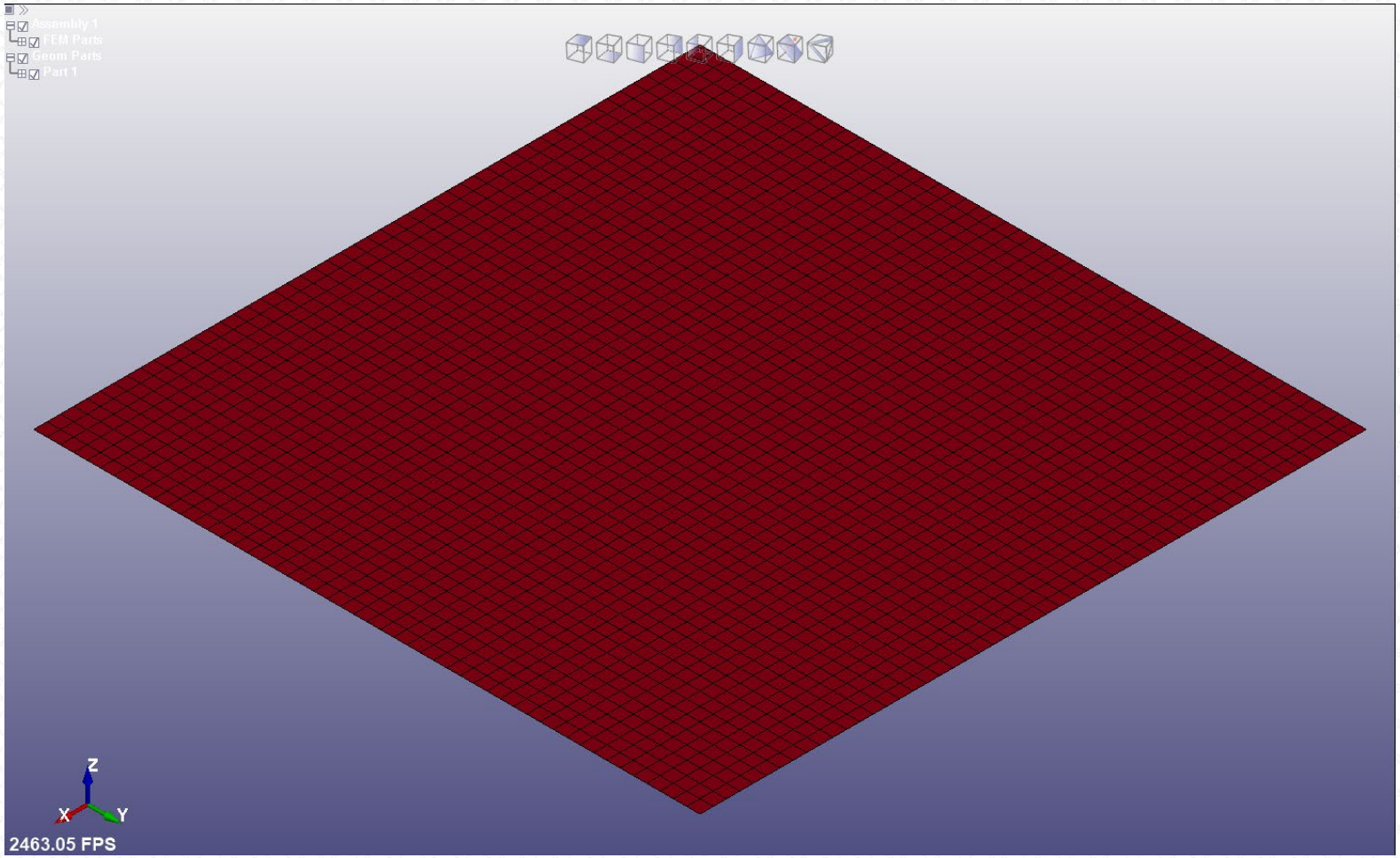
Note:
P1, P2, P3, P4 coordinates when creating 4N Shell





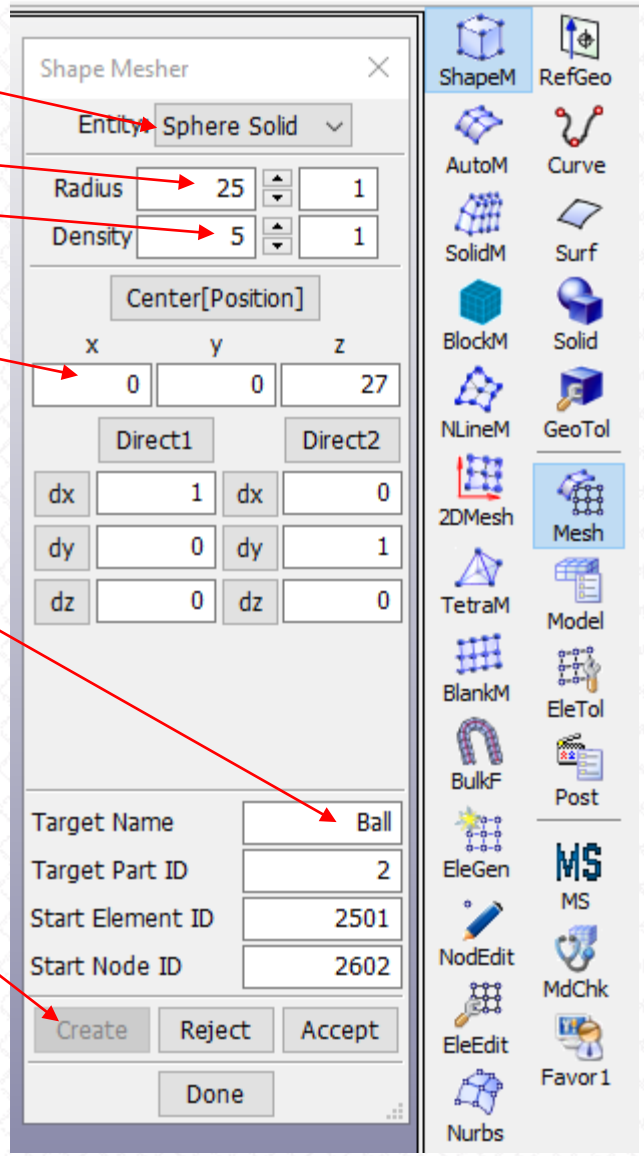
Create Plate

1. Shell Plate will be created



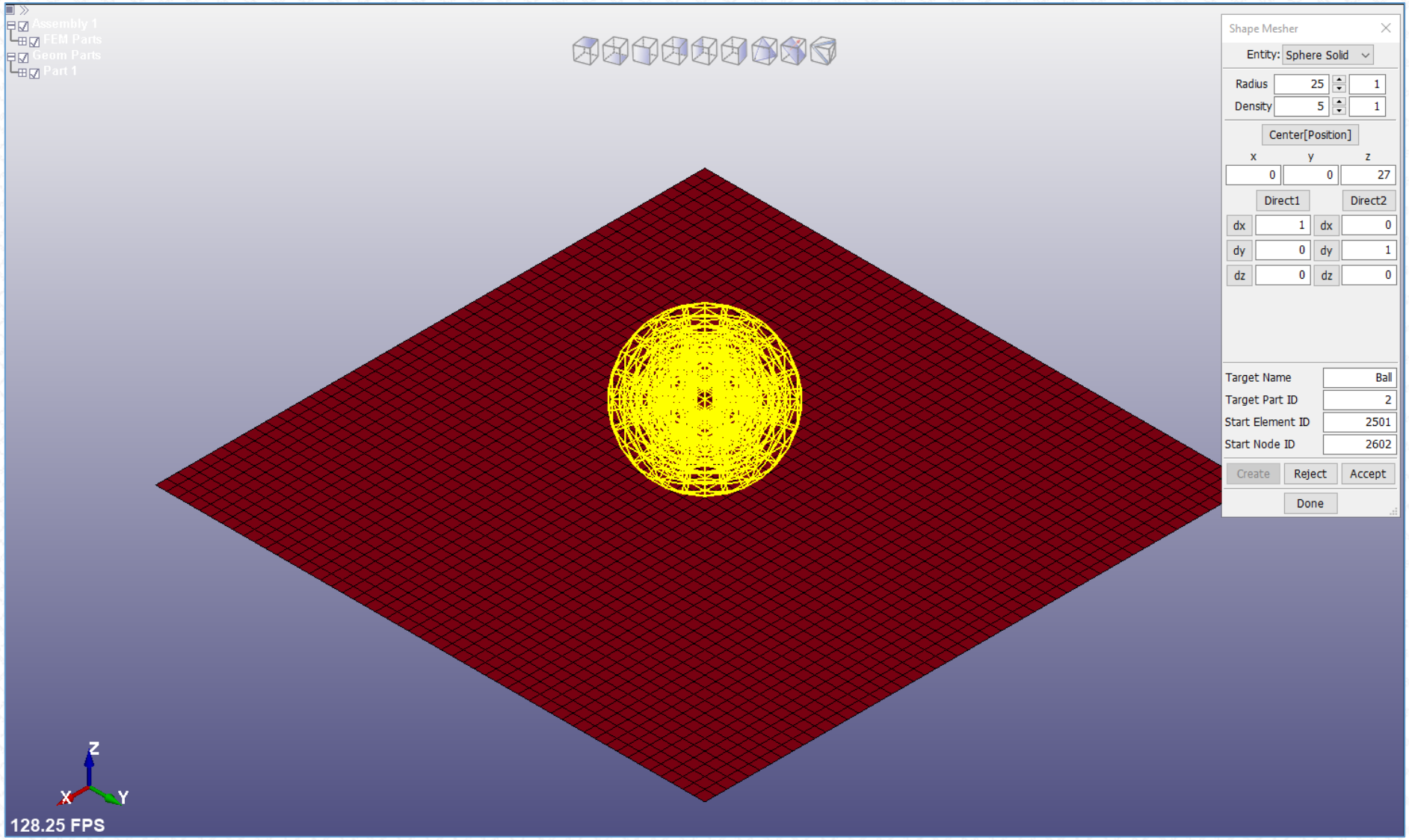
Create Ball

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



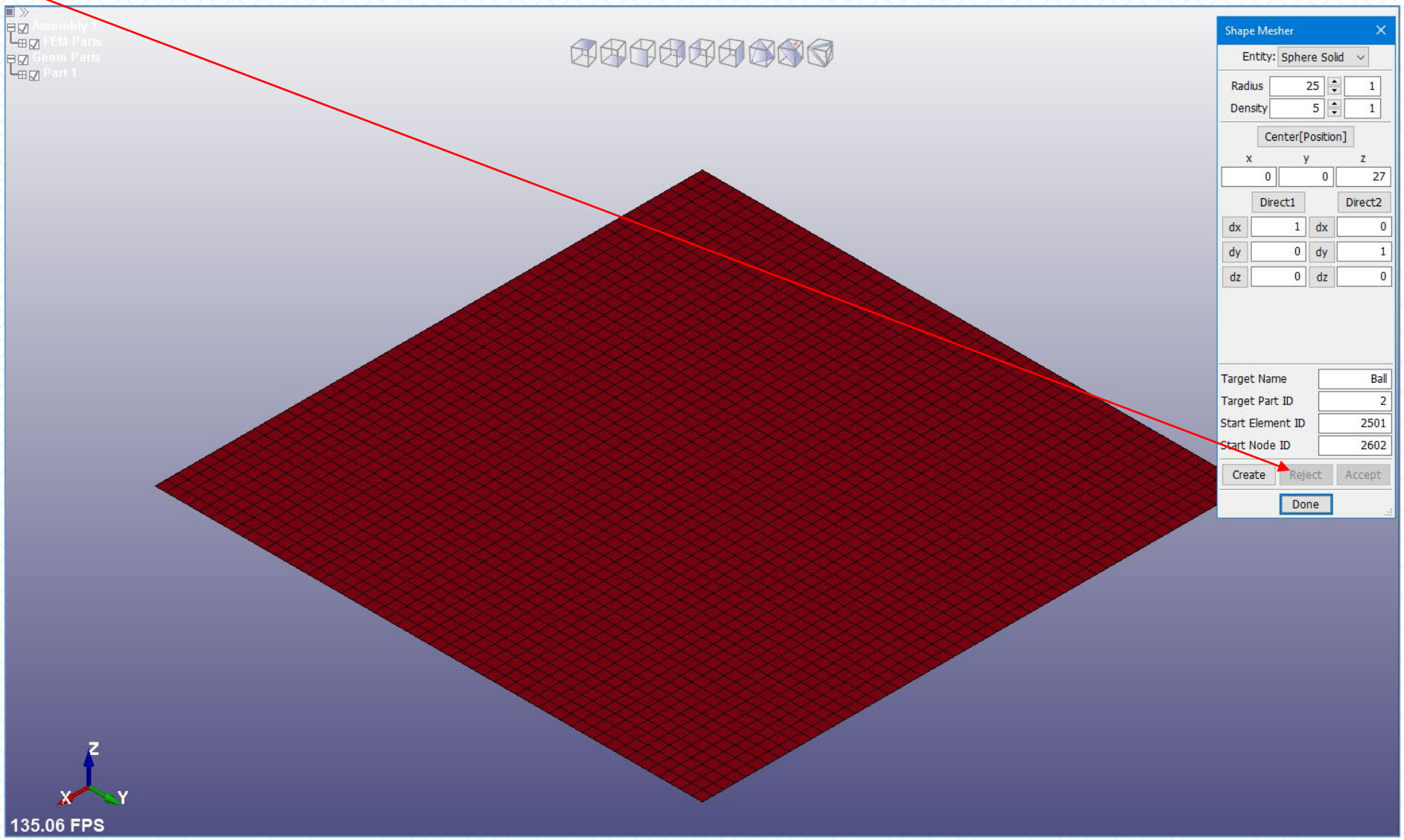
Create Ball

1. Ball mesh will be shown:



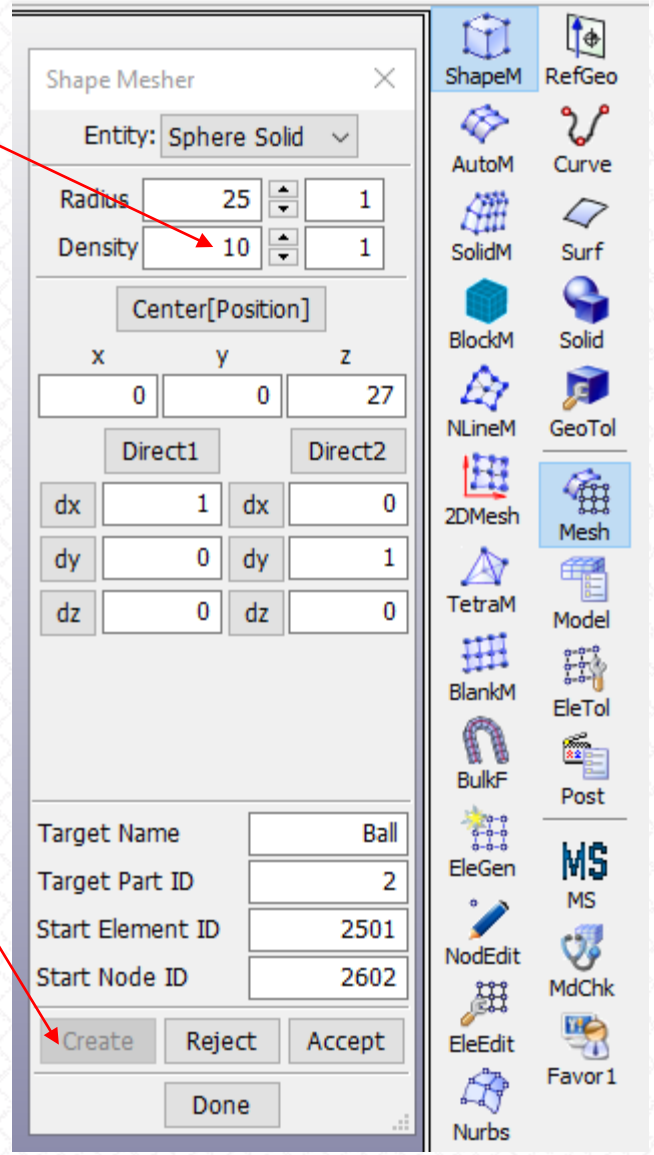
Create Ball

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



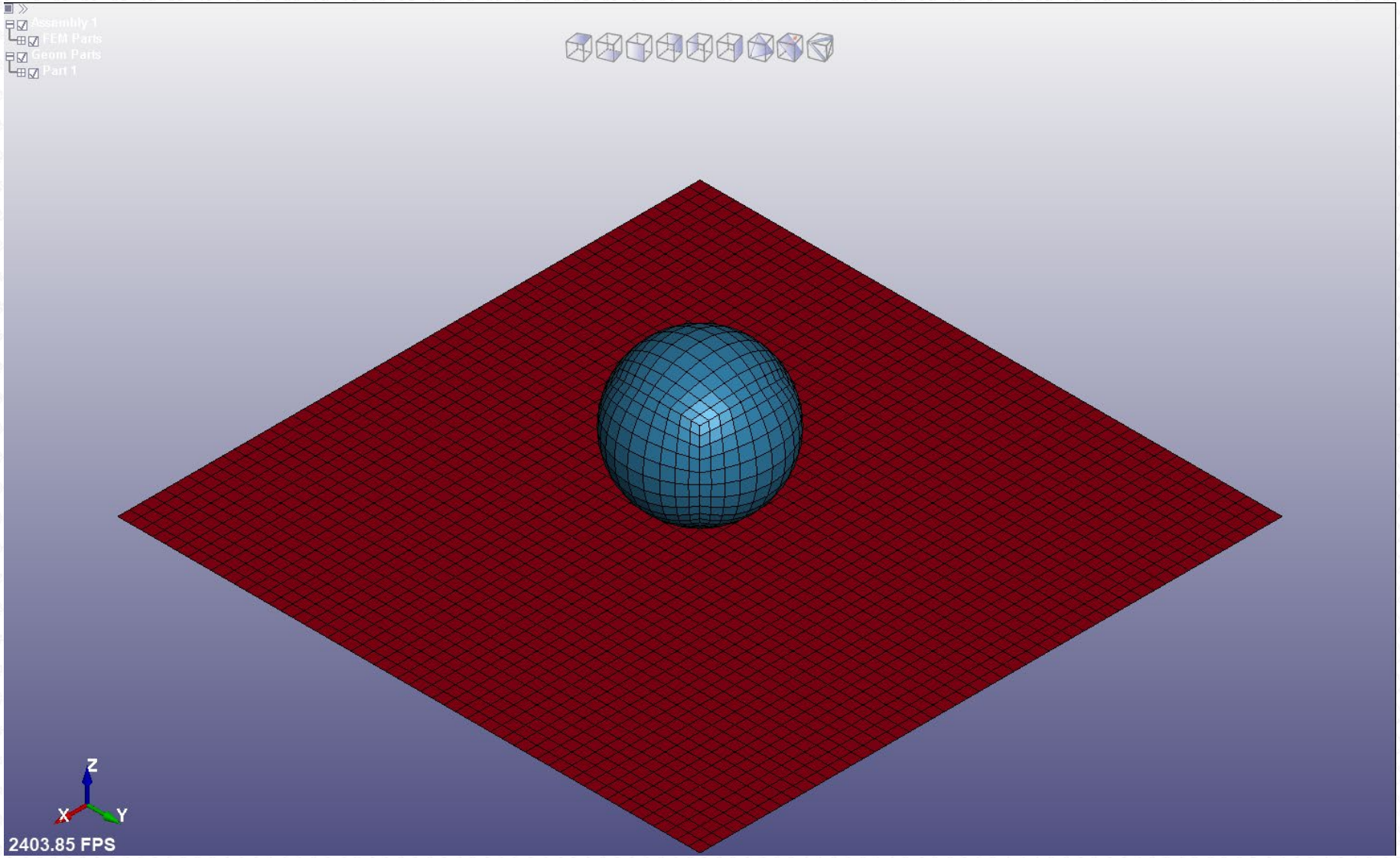
Create Ball

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



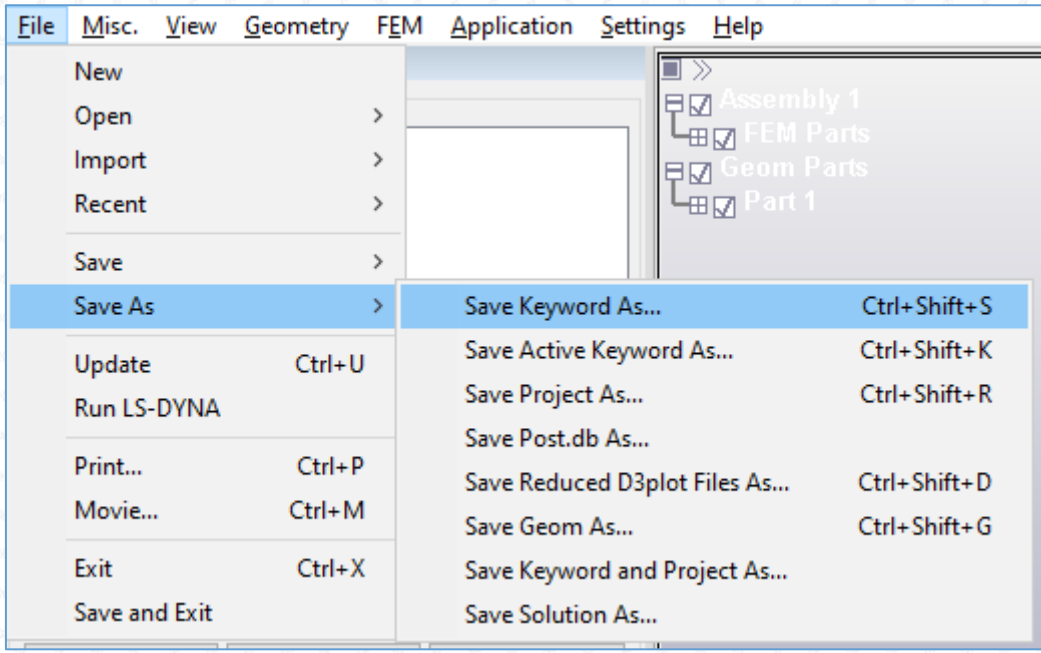
Create Ball

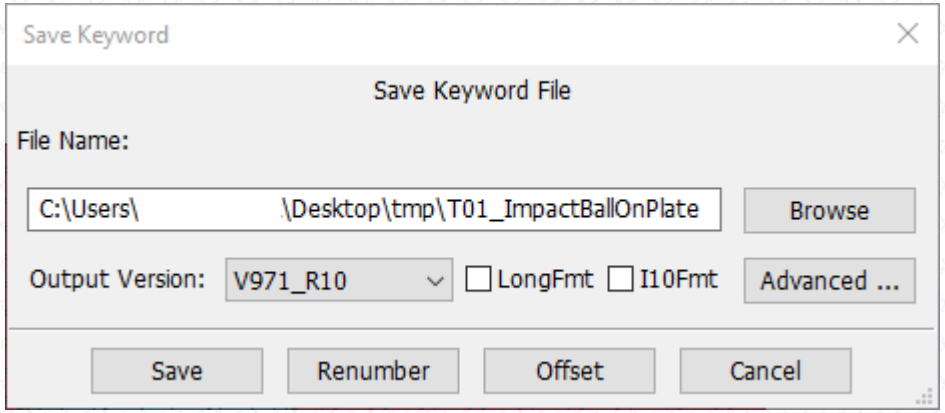
1. Ball mesh will be created:

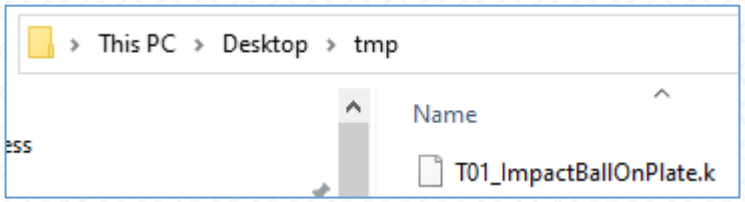


Save

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”





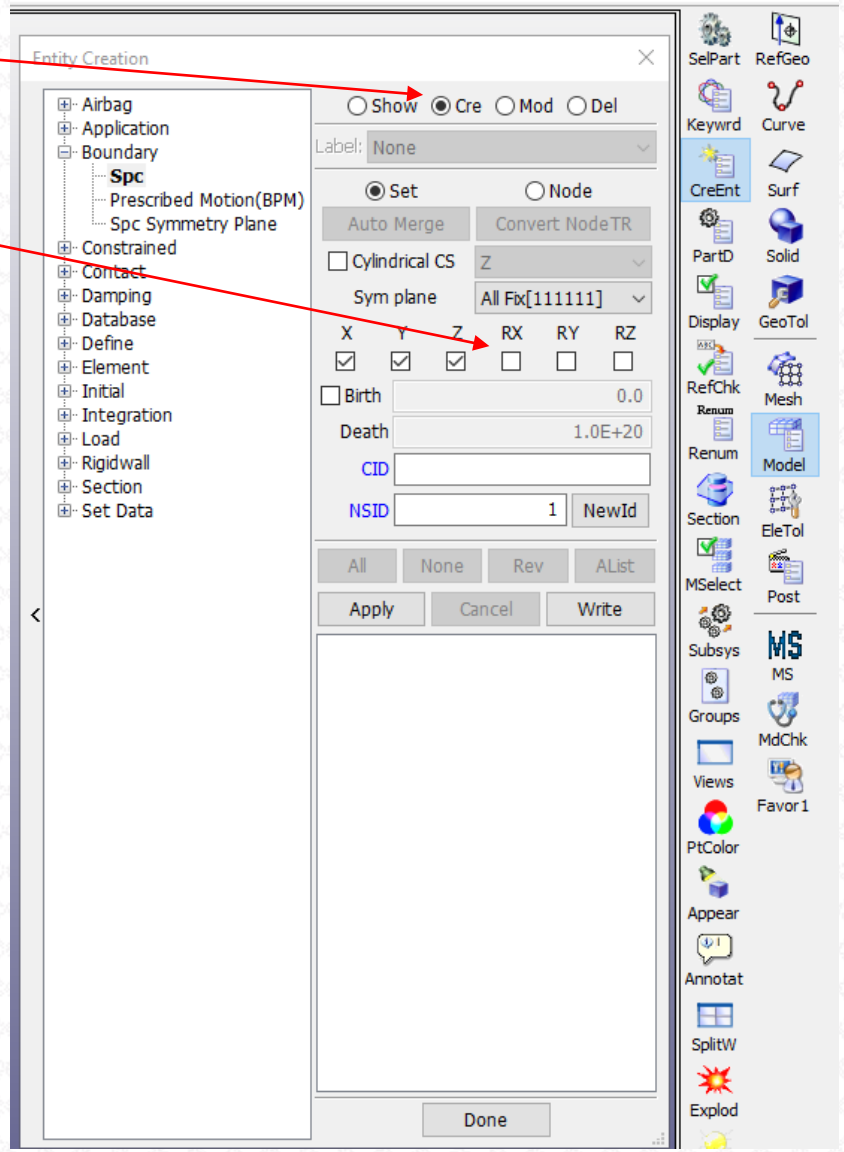


Steps

Step #	Desciption	
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	

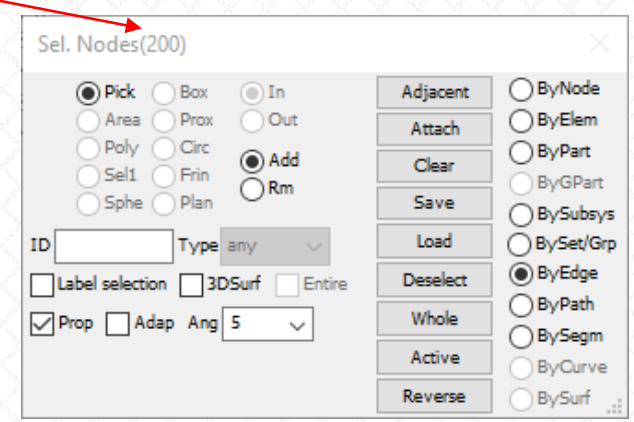
Boundary Conditions

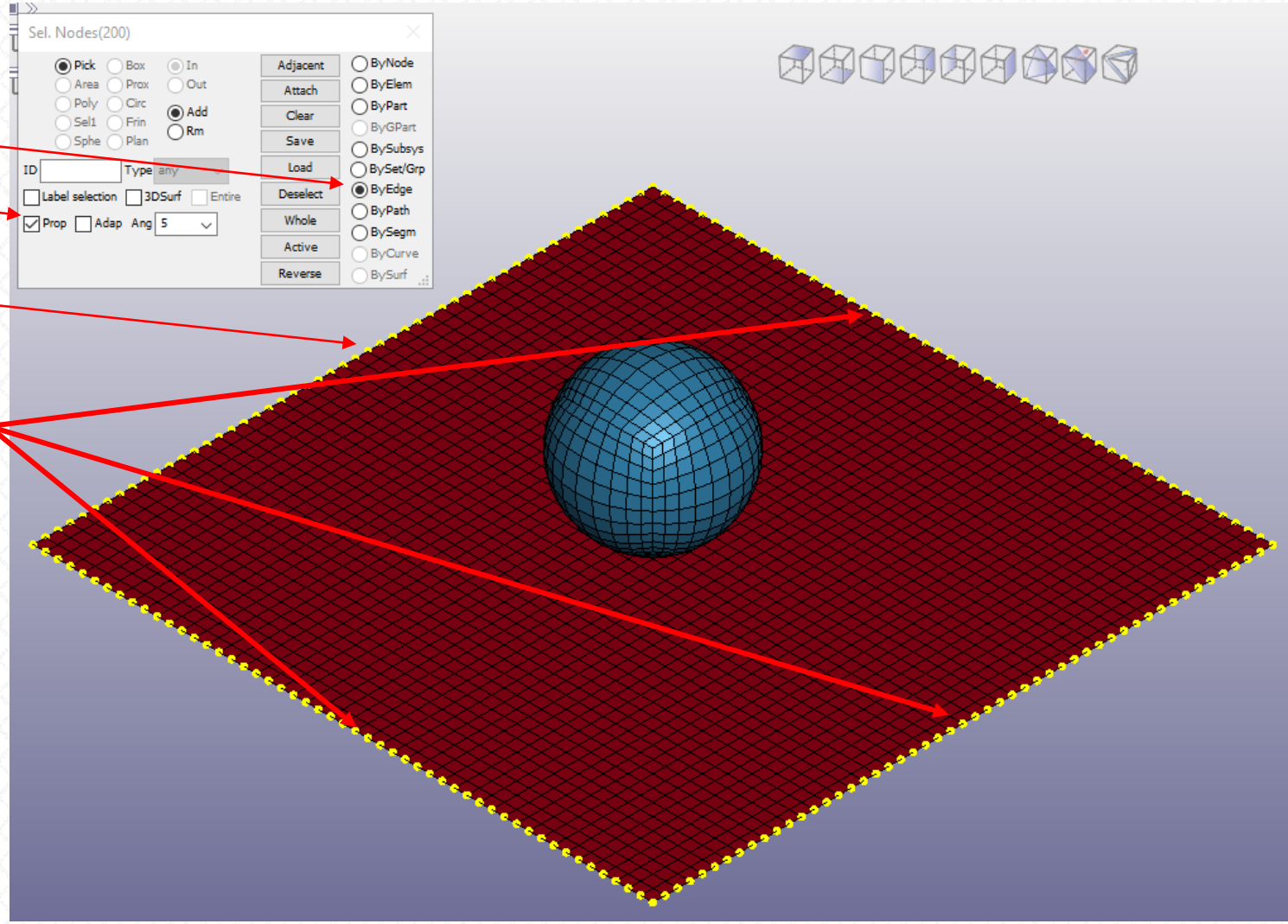
1. Model > CreEnt > Boundary > Spc
2. Switch radio button from “Show” to “Create”
3. Uncheck boxes: RX, RY, RZ



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.

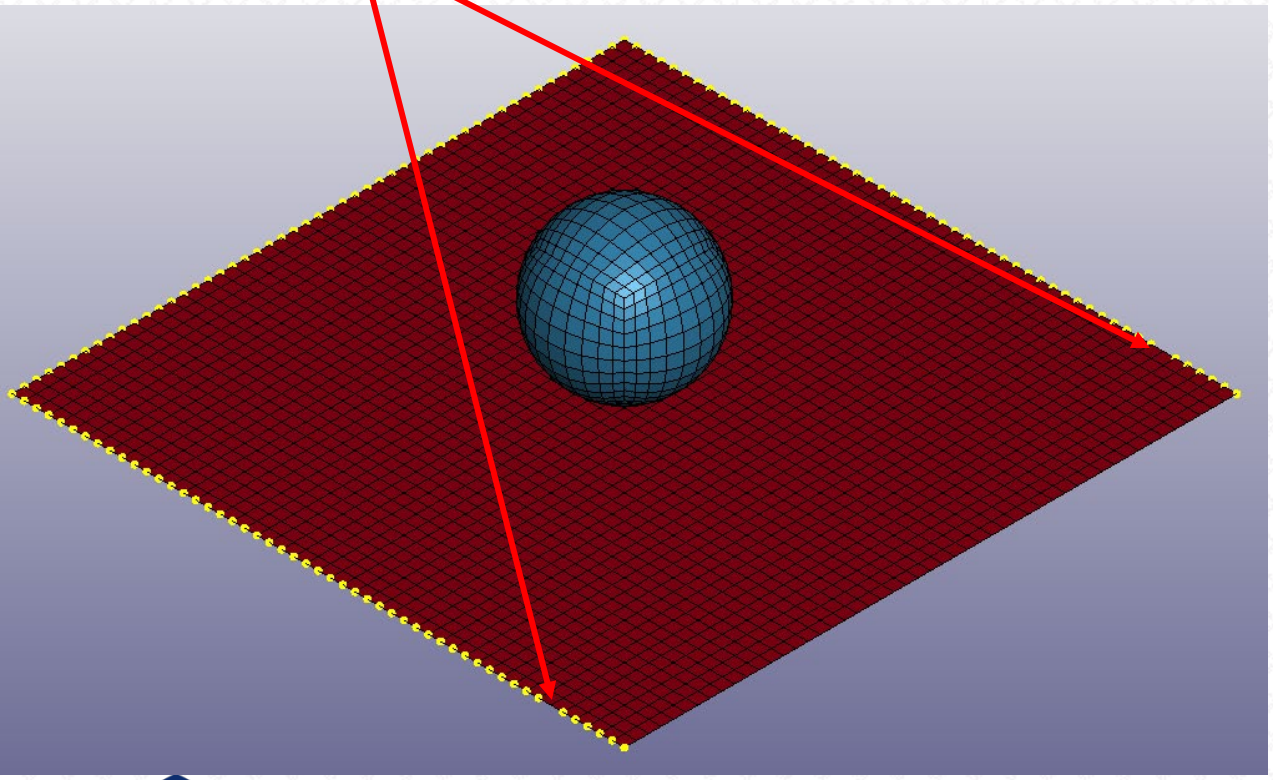


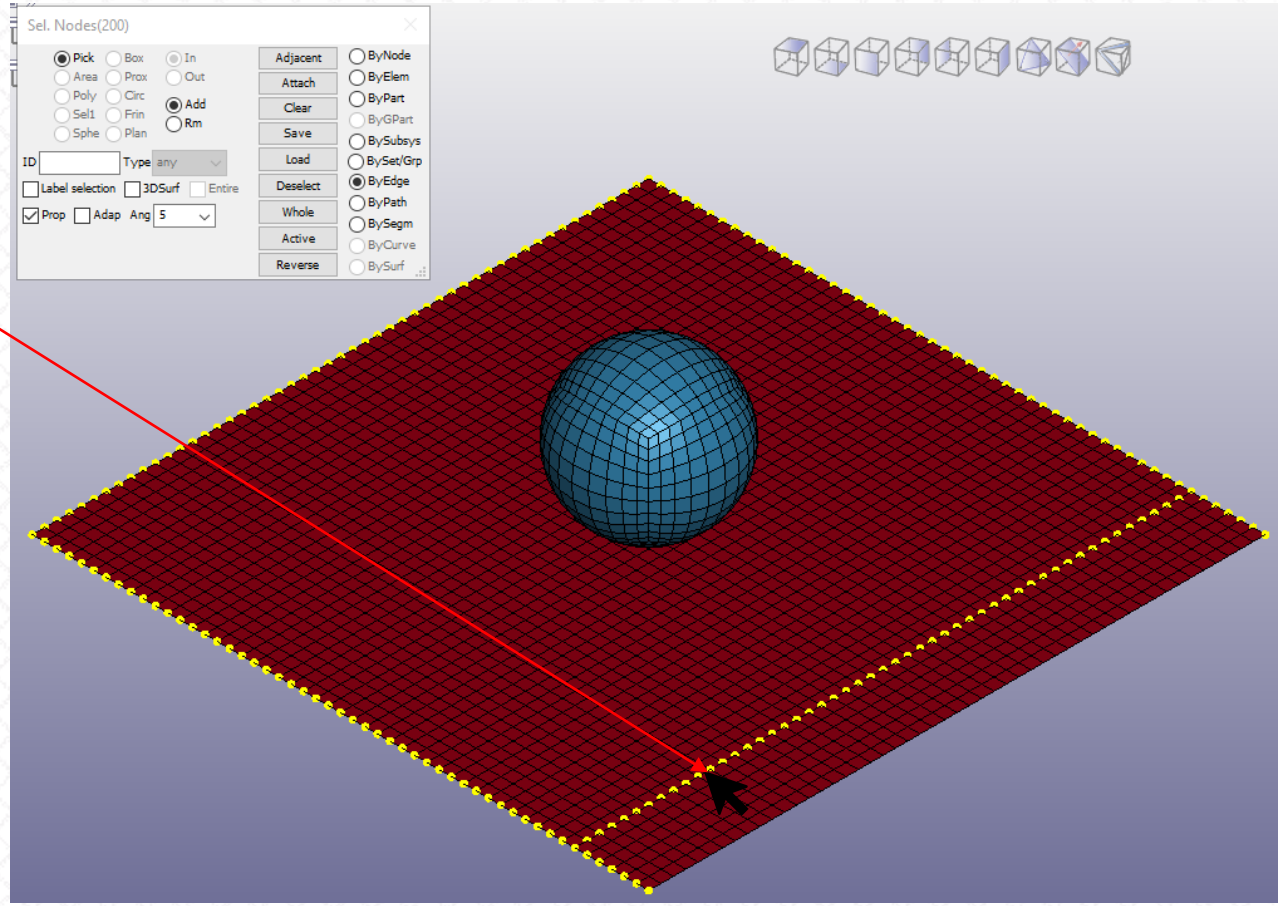


Note: see tip on next slide if you selected wrong nodes accidentally

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

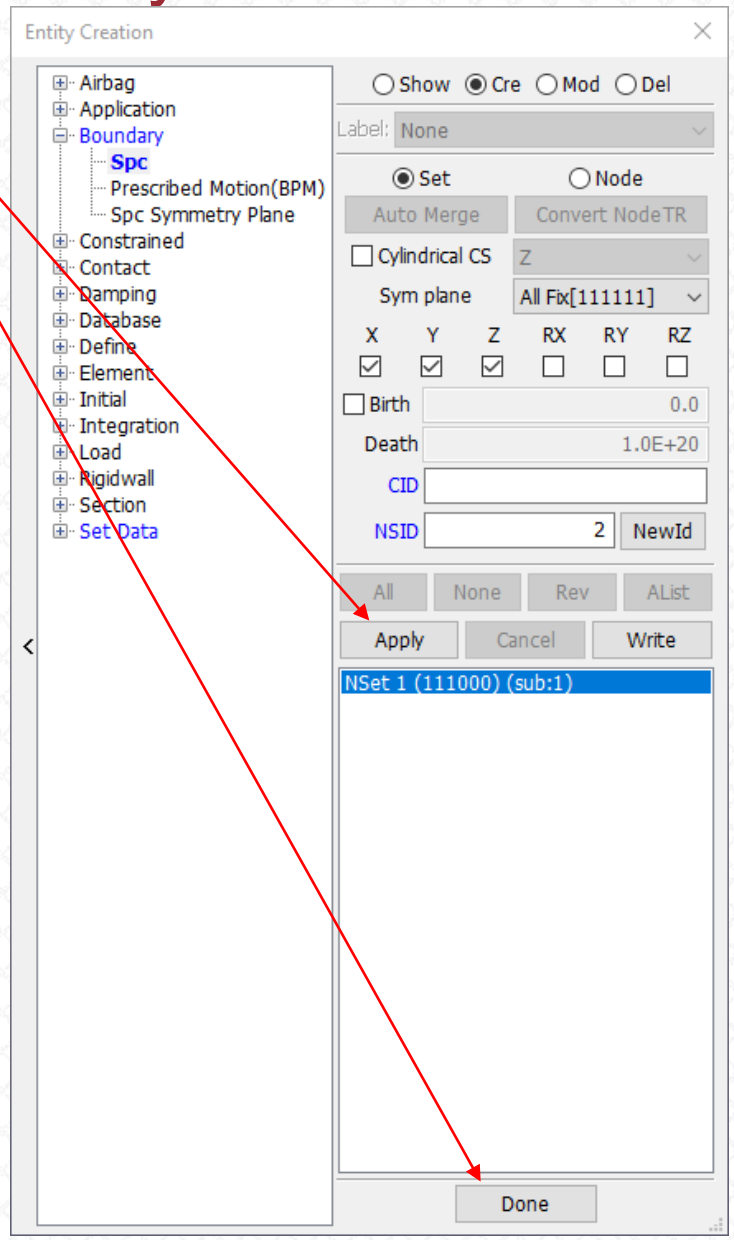


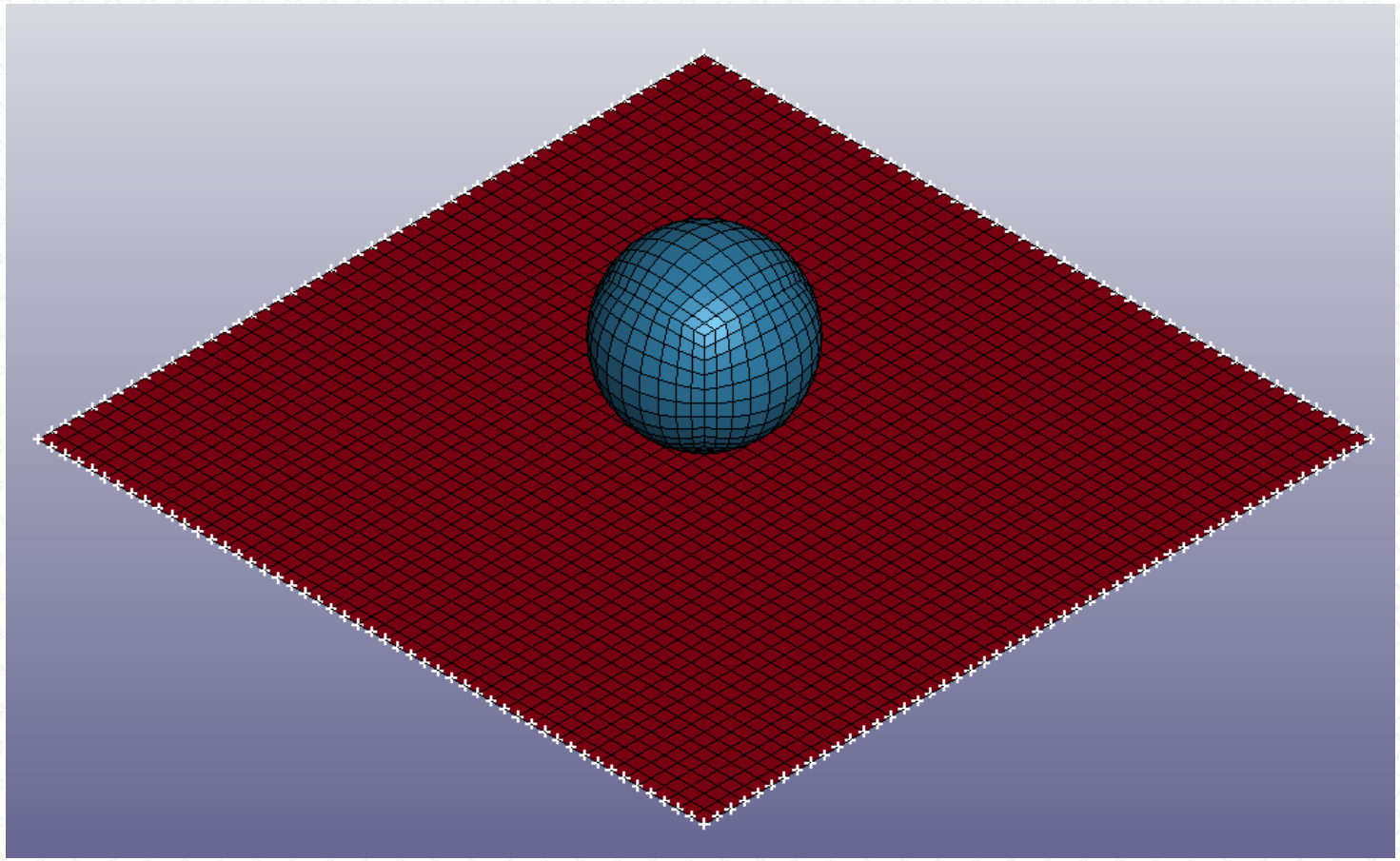


e) Select them back with LMB

Boundary Conditions

1. Apply
2. Done





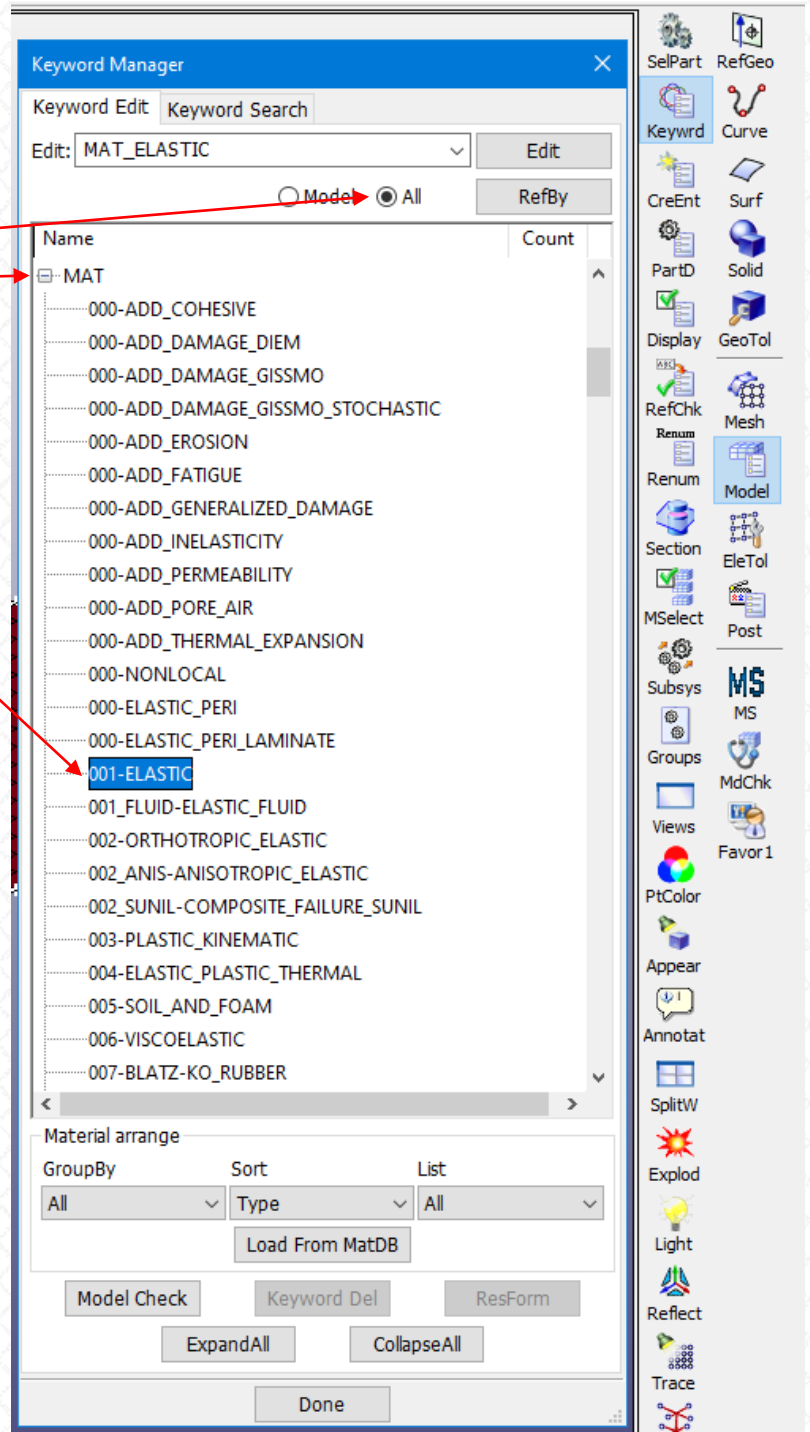
Each outer edge is fixed in X, Y, Z while is free to rotate in RX, RY, RZ.

Steps

Step #	Desciption	
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	

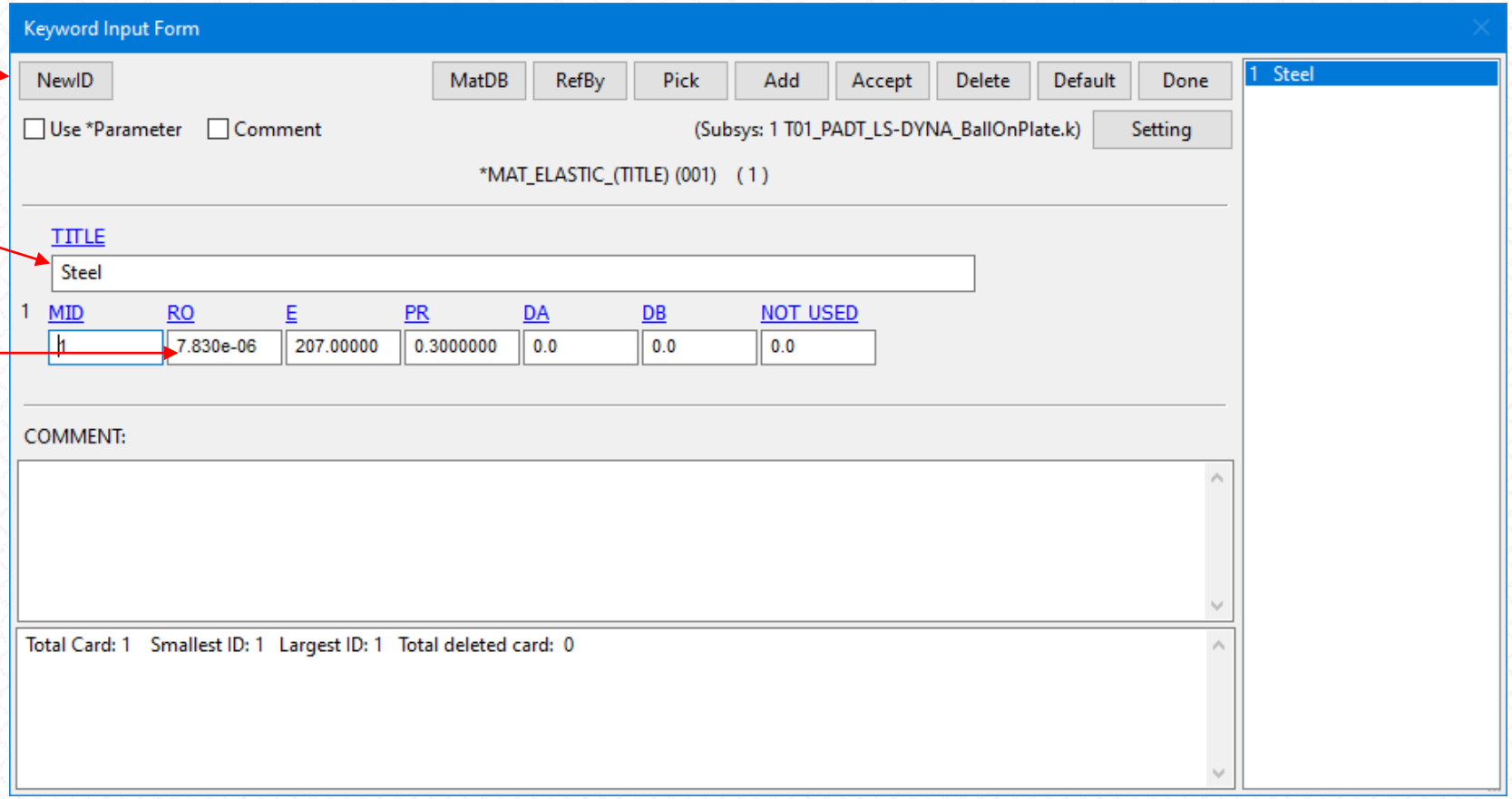
Material Properties

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



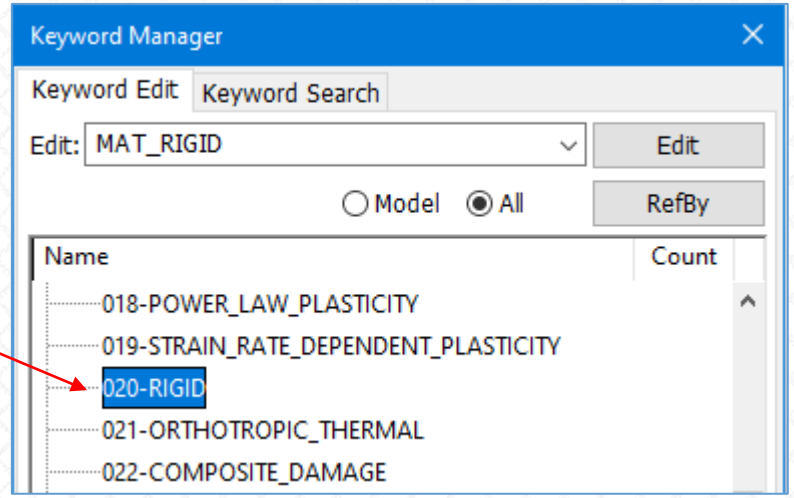
Material Properties

1. Click “NewID”
2. TITLE: Steel
3. Type values for RO, E, PR as shown
   a) Density RO = 7.83e-6 kg/mm3
   b) Elastic Modulus E = 207 GPa
   c) Poisson’s ratio PR = 0.3
4. Accept
5. Done



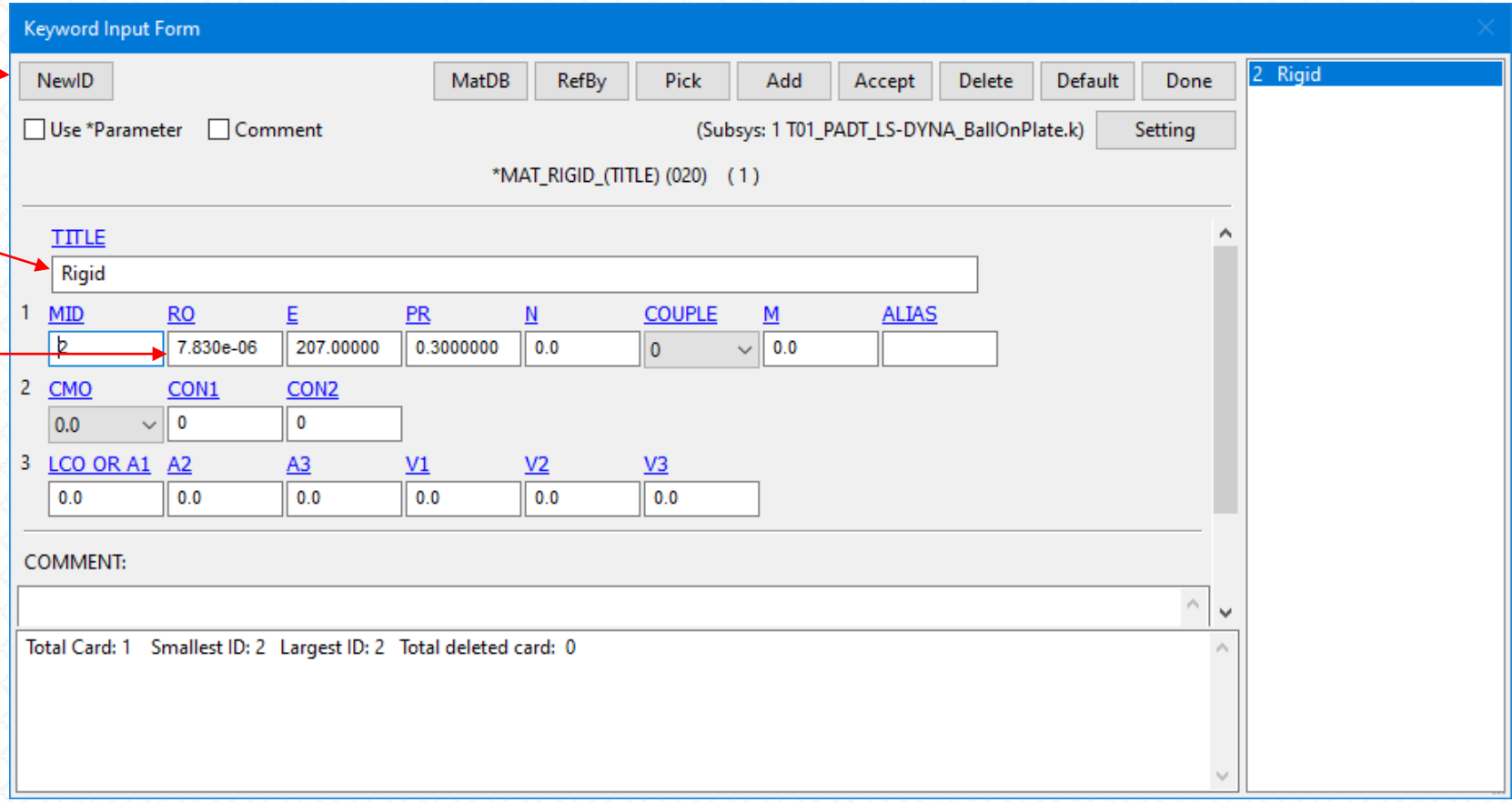
Material Properties

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 = 7.83e-6 kg/mm3
   b) Elastic Modulus E = 207 GPa
   c) Poisson’s ratio PR = 0.3
4. Accept
5. Done



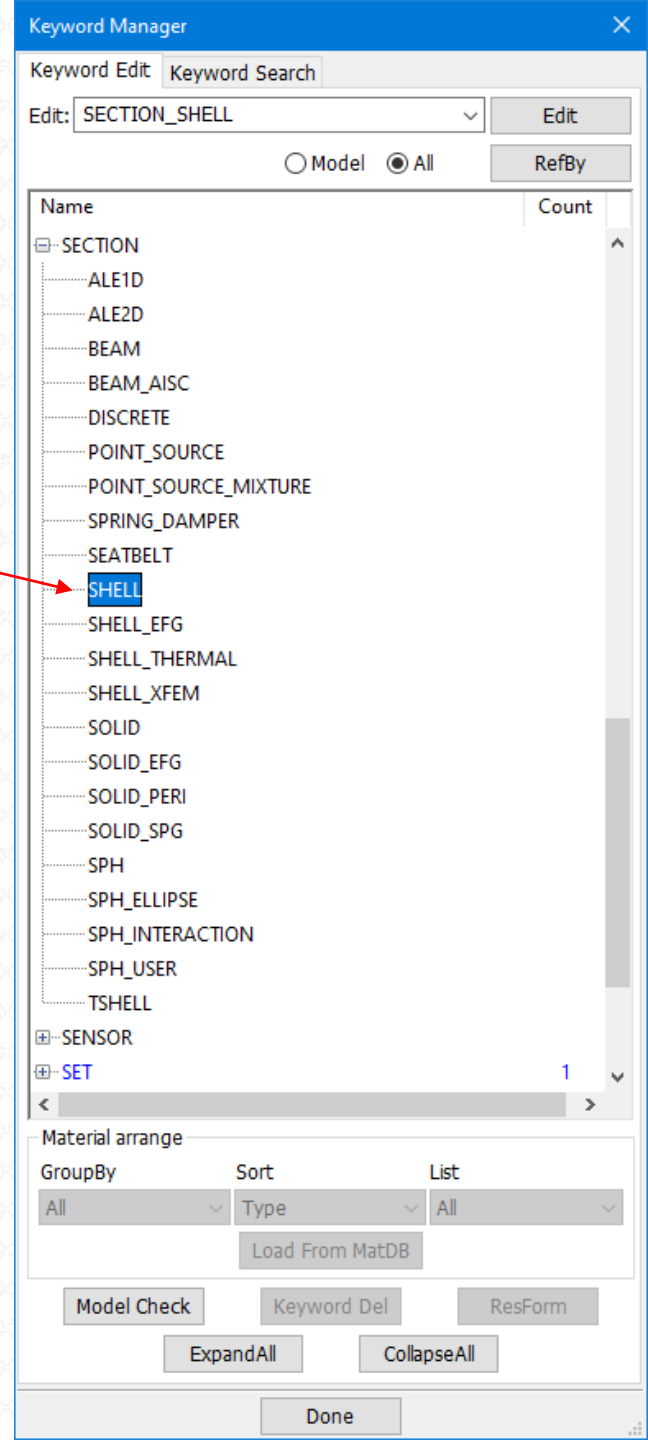
Note:
Rigid material property will be assigned to the Ball.
Components for which deformation is negligible and stress is unimportant may be modeled as rigid bodies.
The elastic constants defined here are used for contact stiffness calculations. Thus, the constants should be reasonable.

Steps

Step #	Desciption	
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	

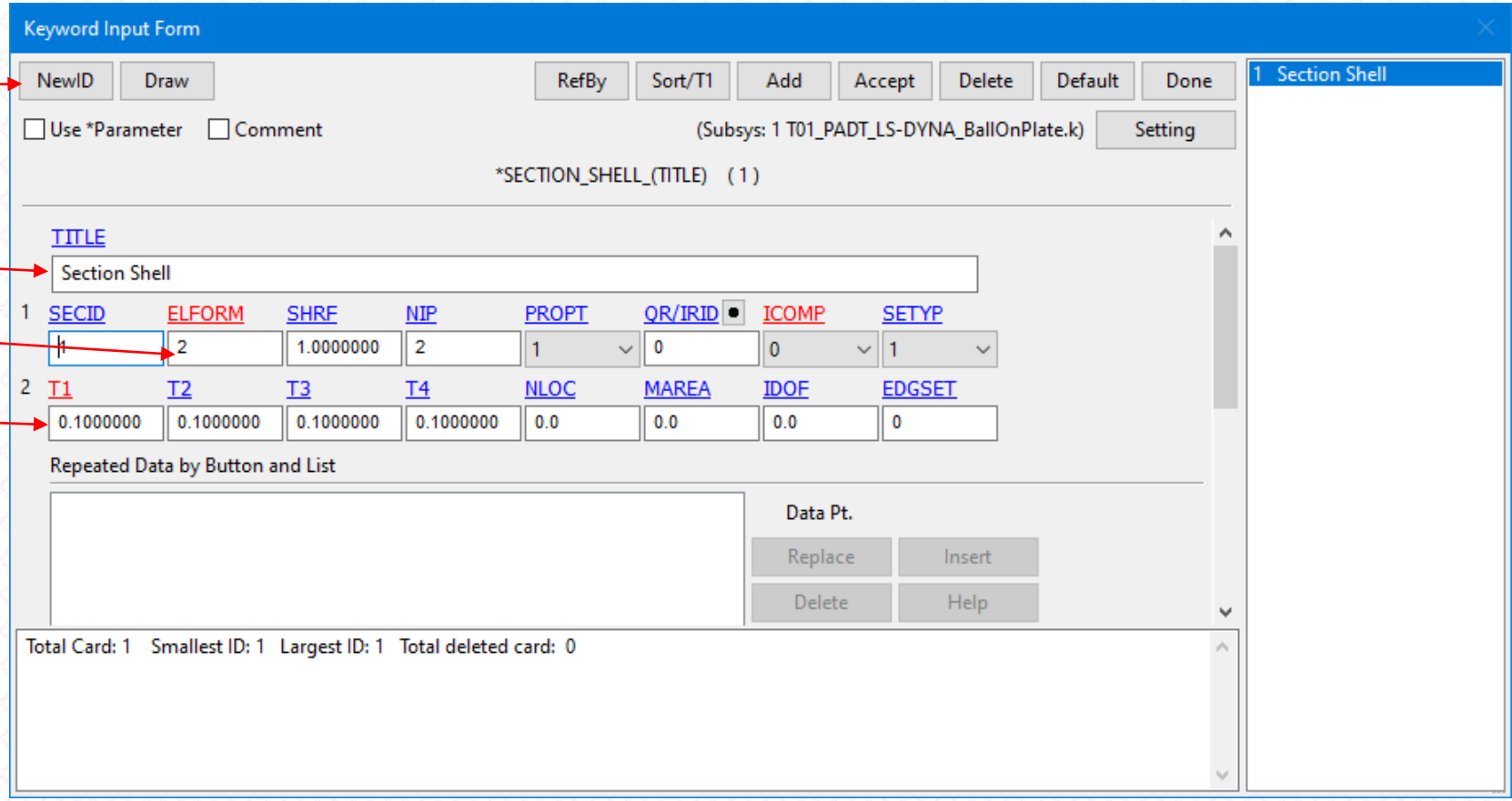
Section Properties / Element Formulation

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



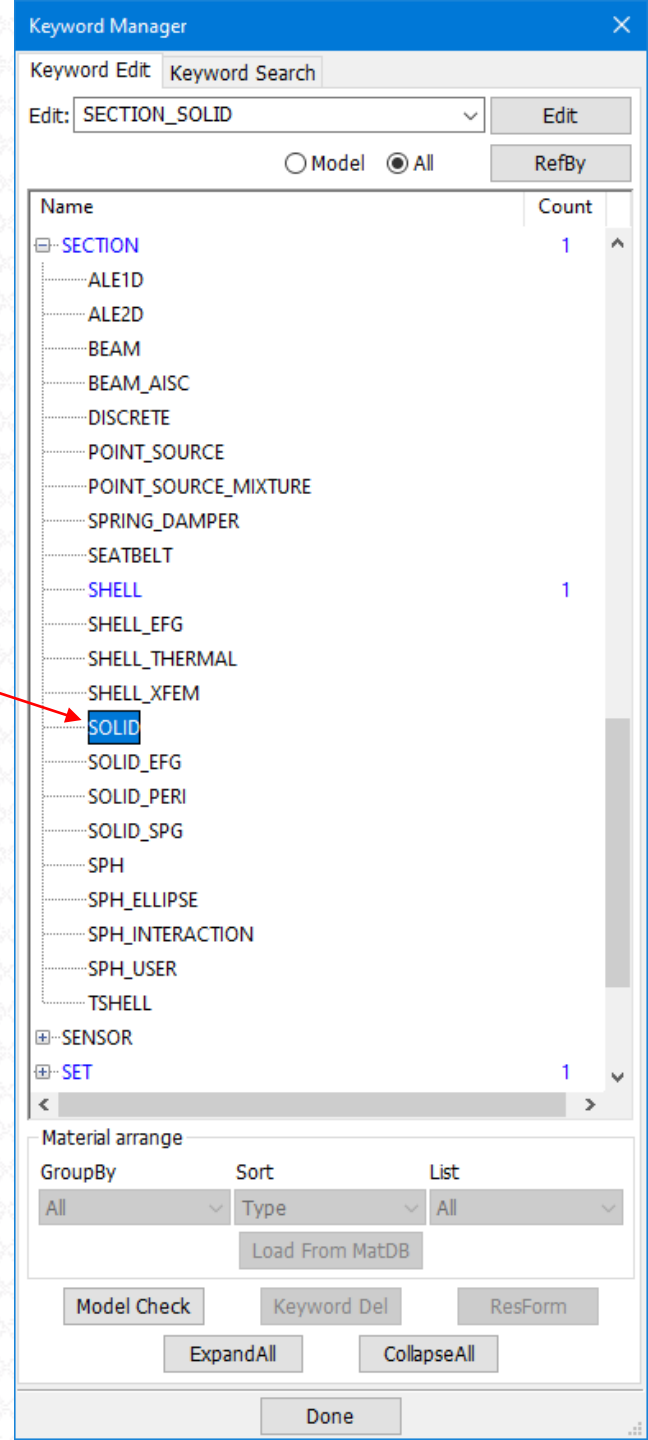
Section Properties / Element Formulation

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



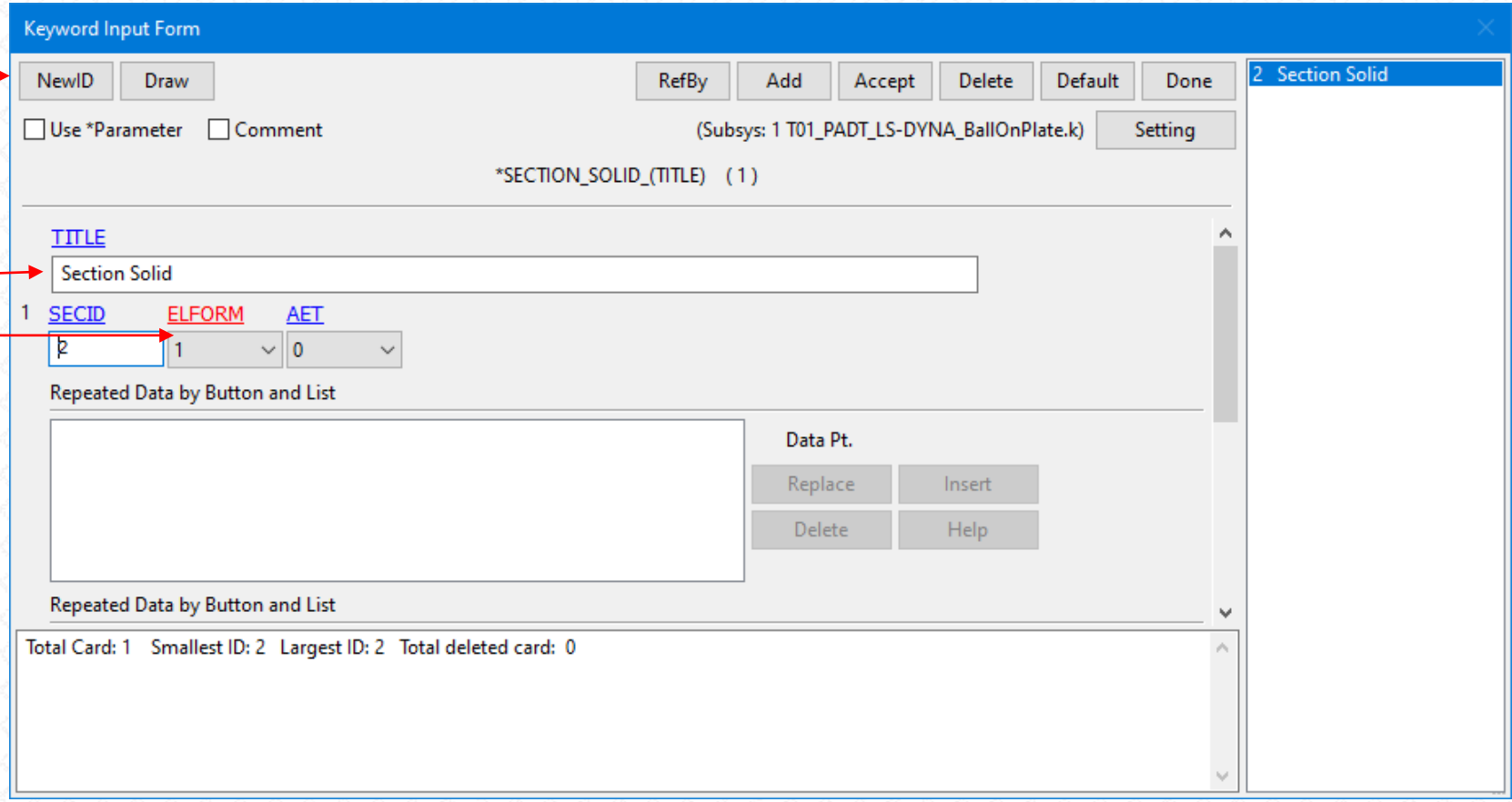
Section Properties / Element Formulation

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



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

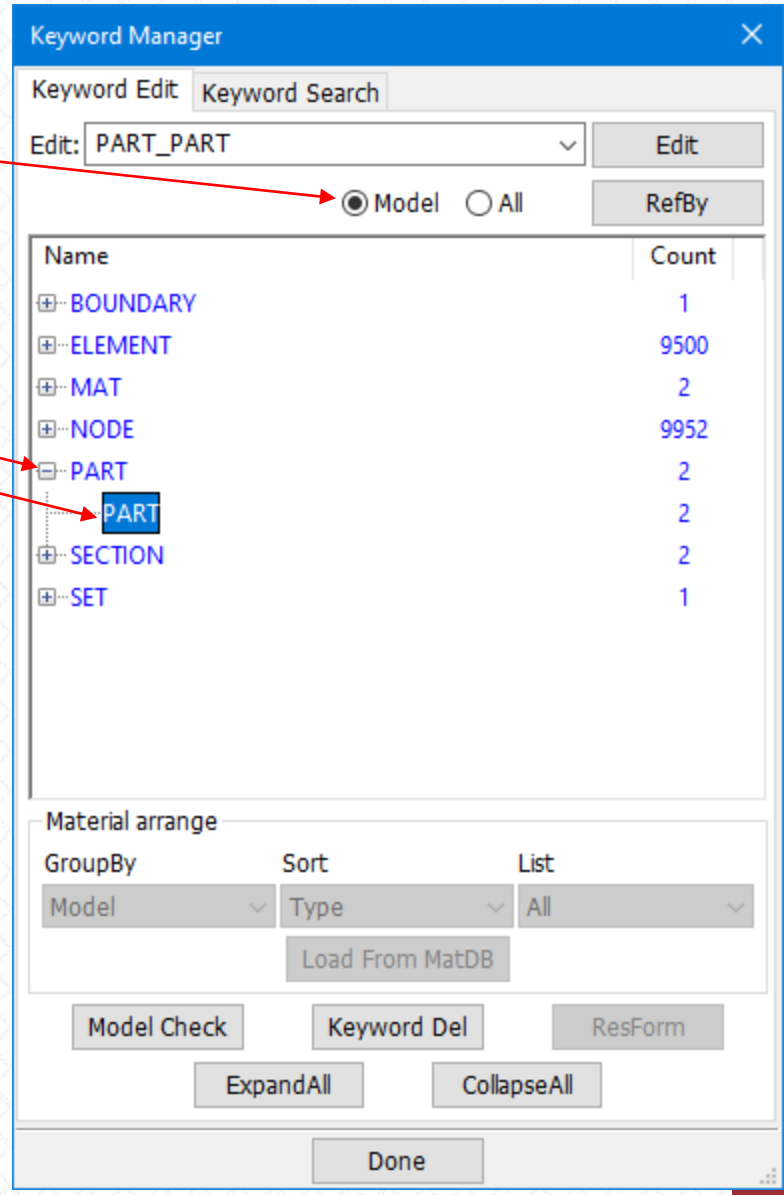


Steps

Step #	Desciption	
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	

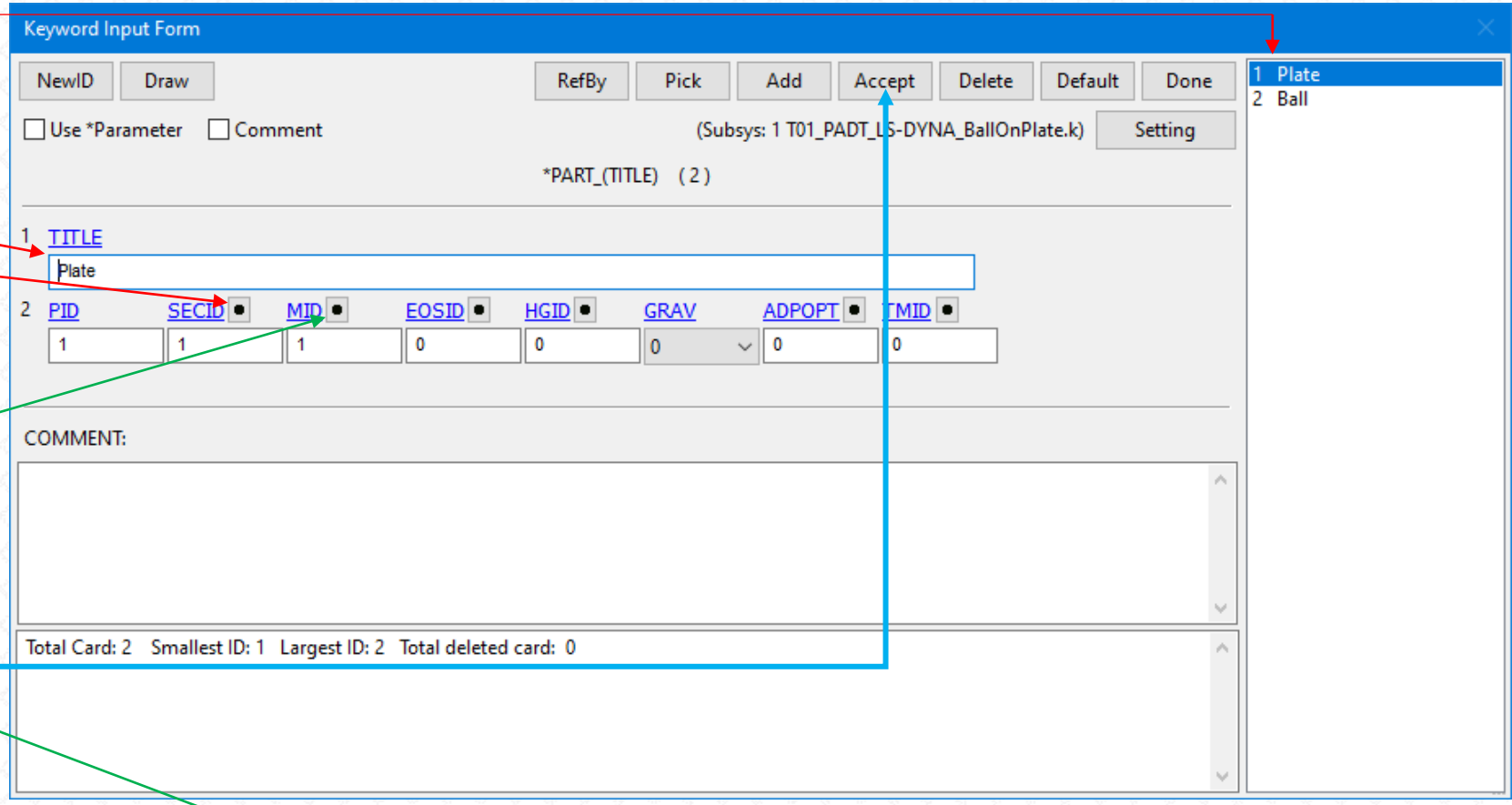
Assign Material and Section Properties to Parts

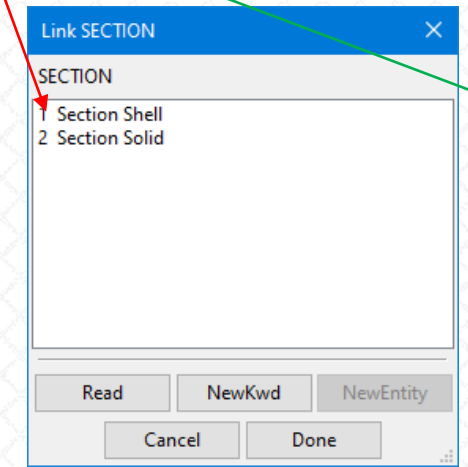
1. Switch radio button from “All” to “Model”
2. Expand [PART]
3. Double click on [PART]

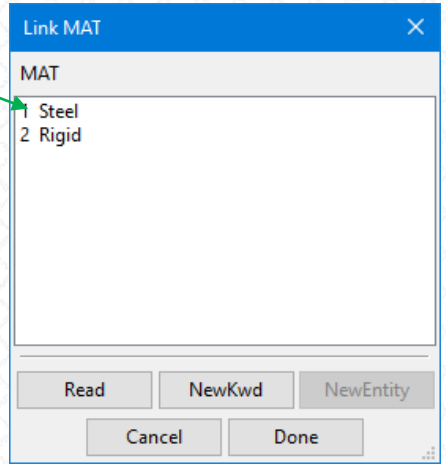


Assign Material and Section Properties to Parts

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

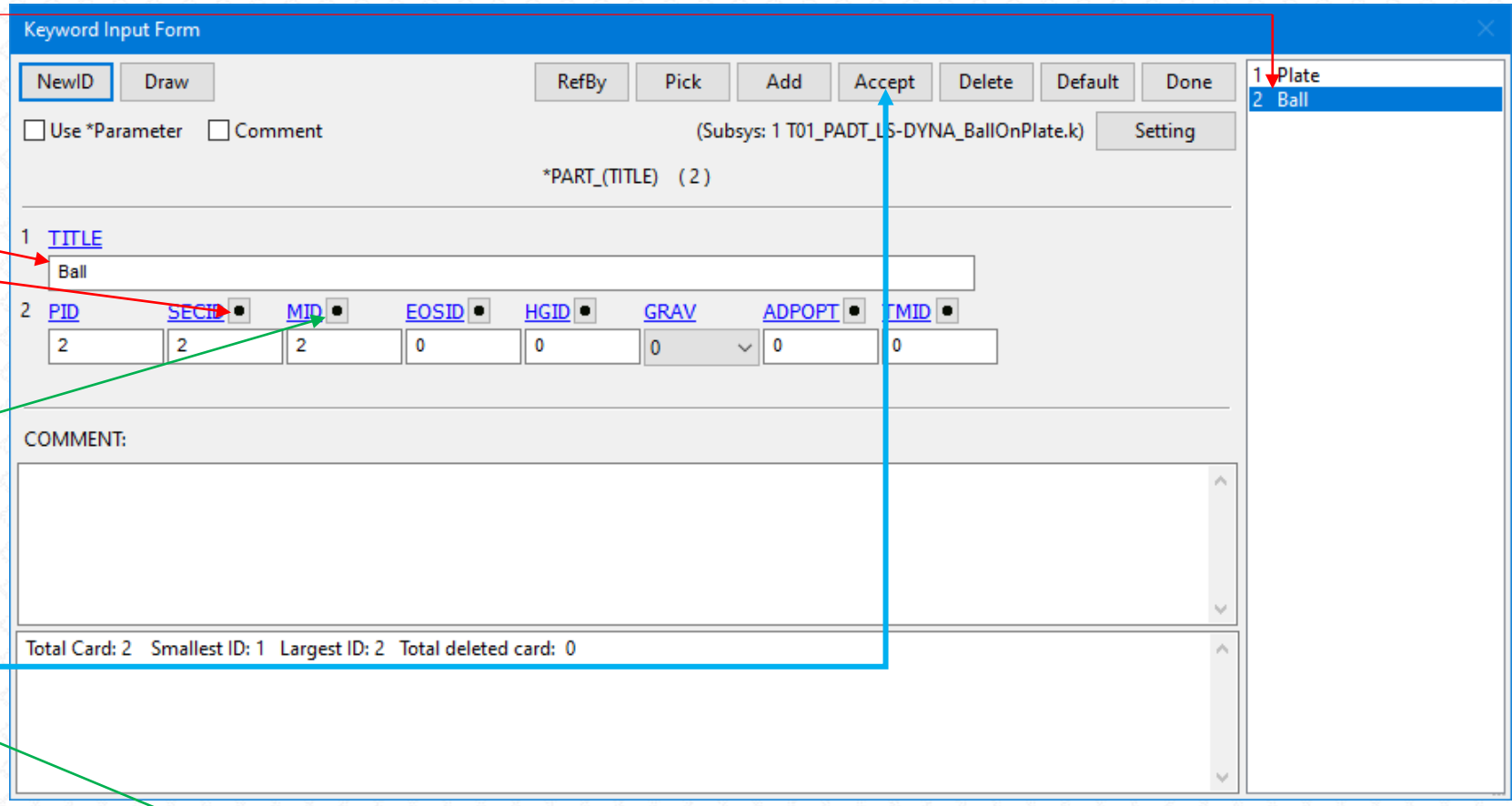


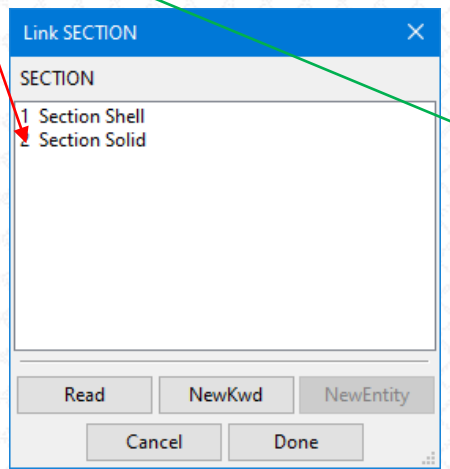


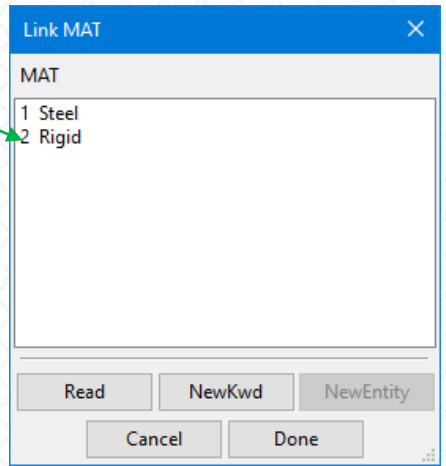


Assign Material and Section Properties to Parts

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





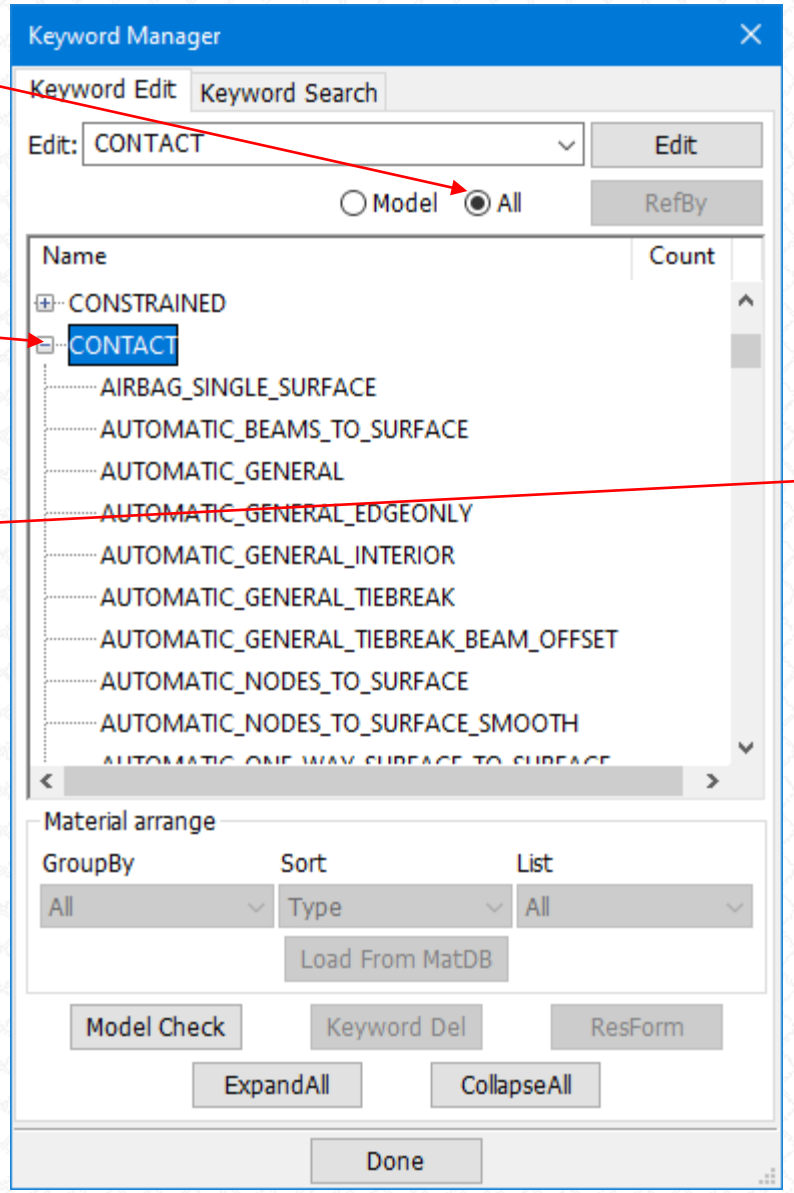


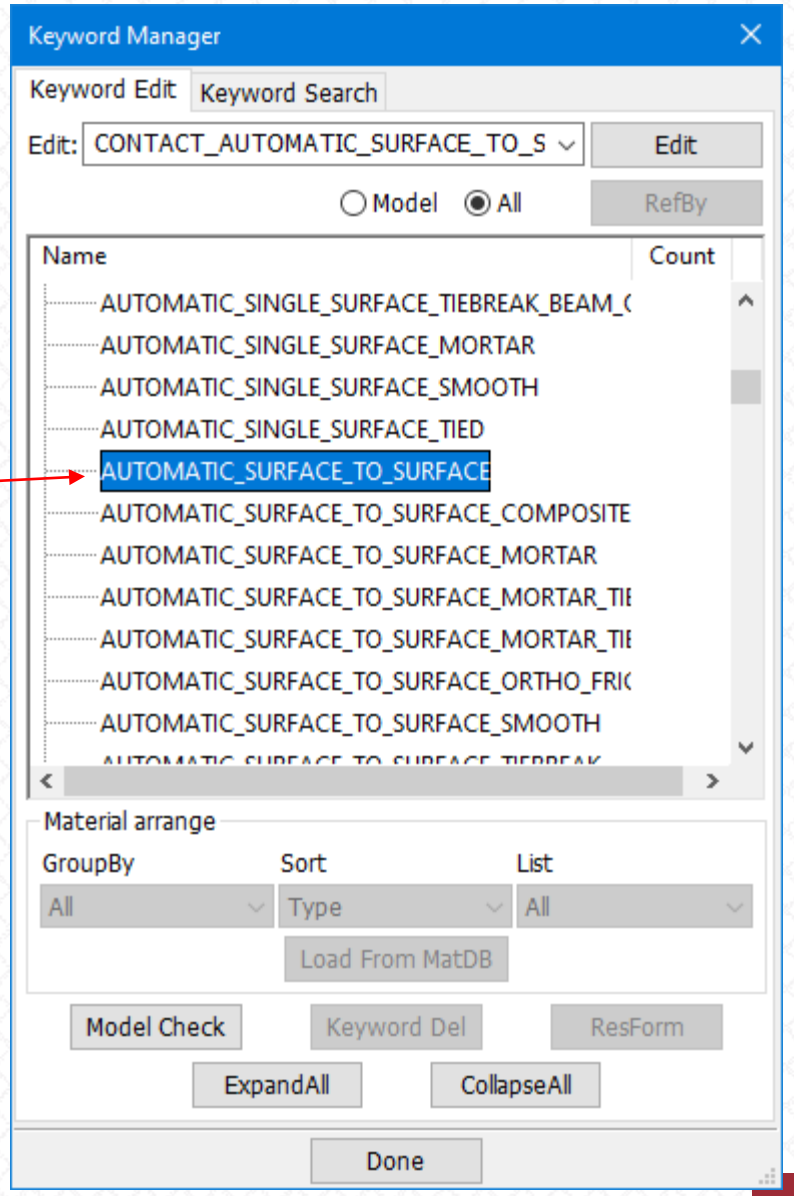
Steps

Step #	Desciption	
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

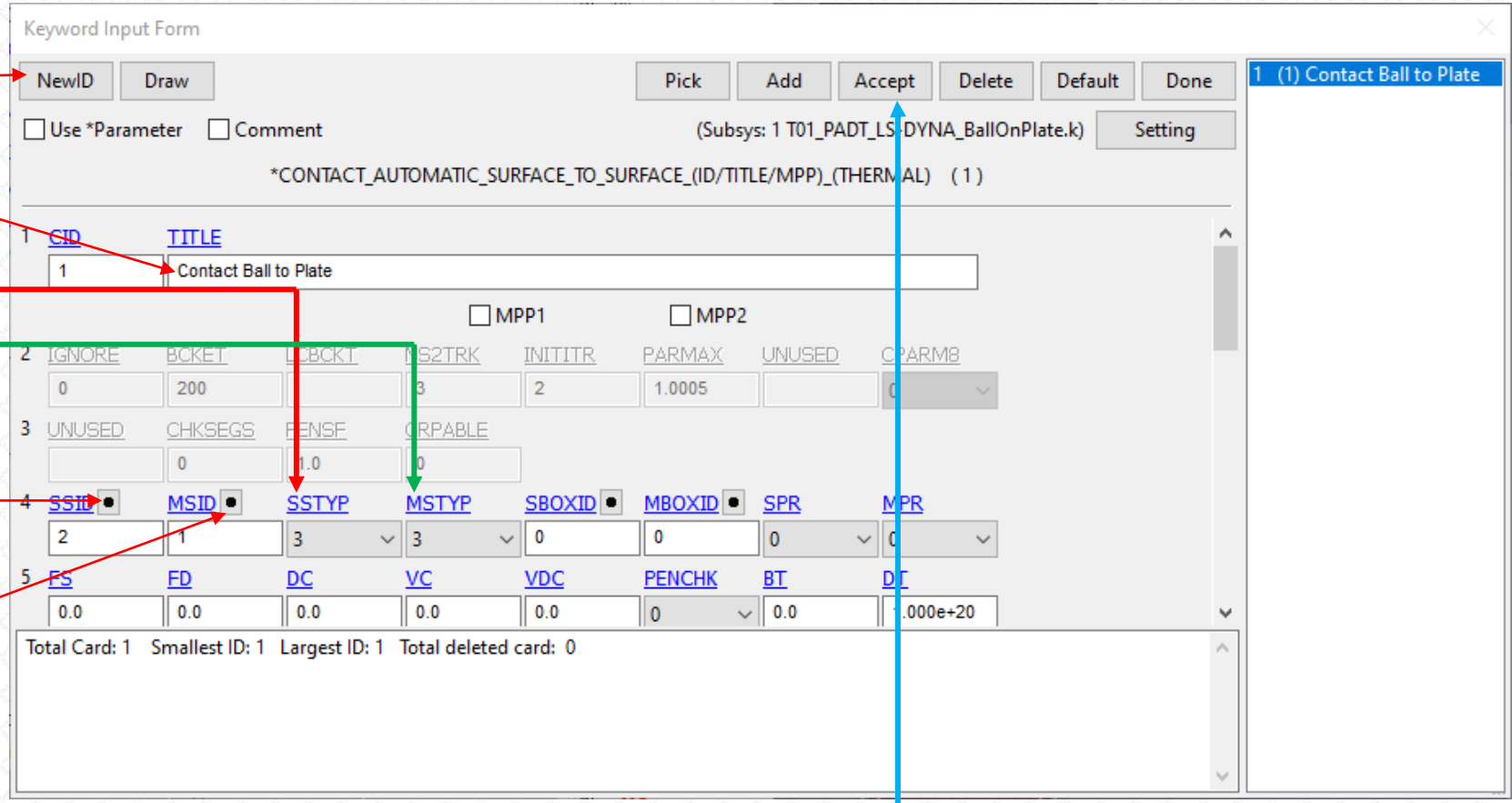
1. Switch radio button from “Model” to “All”
2. Expand [CONTACT]
3. Scroll down and double click on [AUTOMATIC_SURFACE_TO_SURFACE]

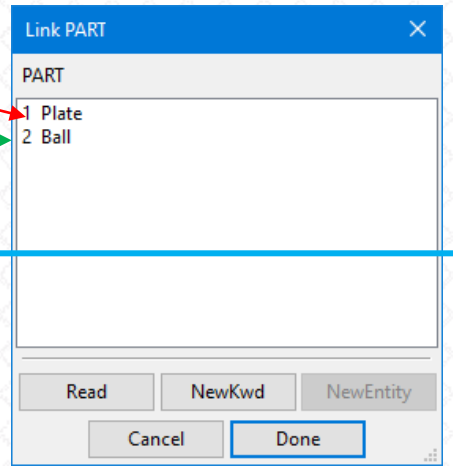




Contact

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



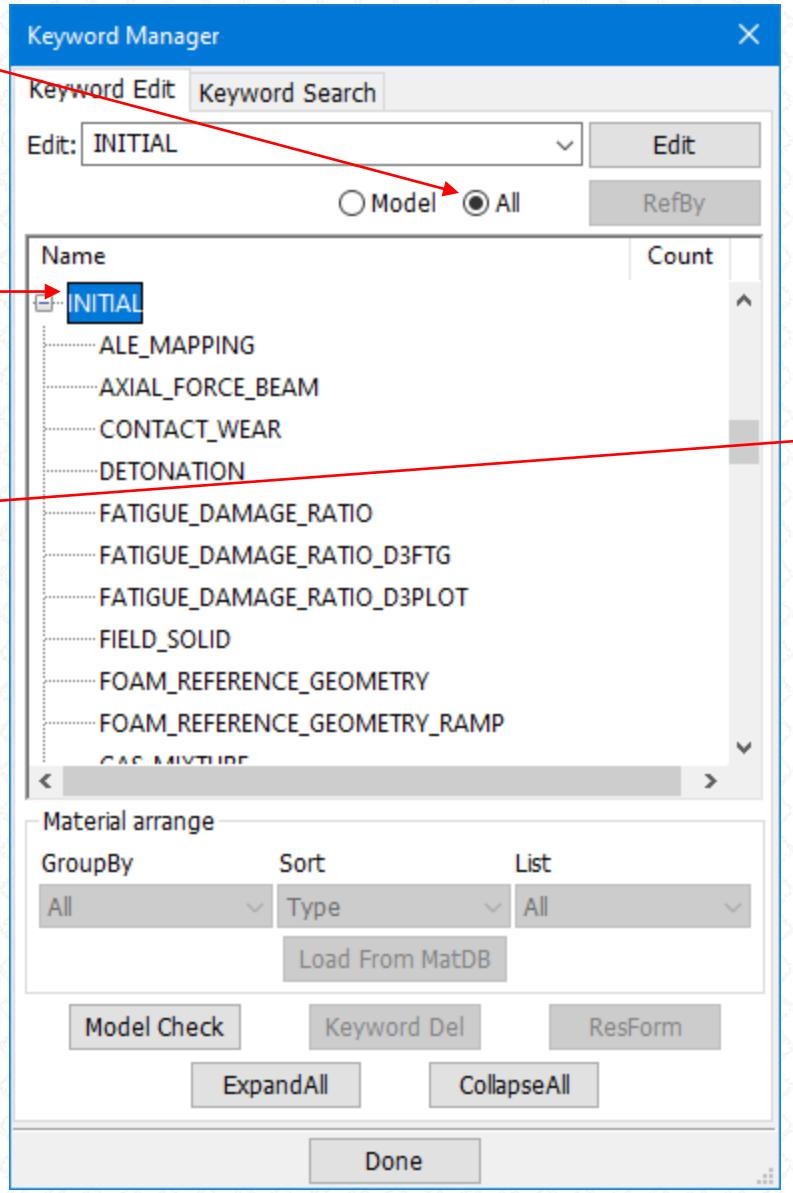


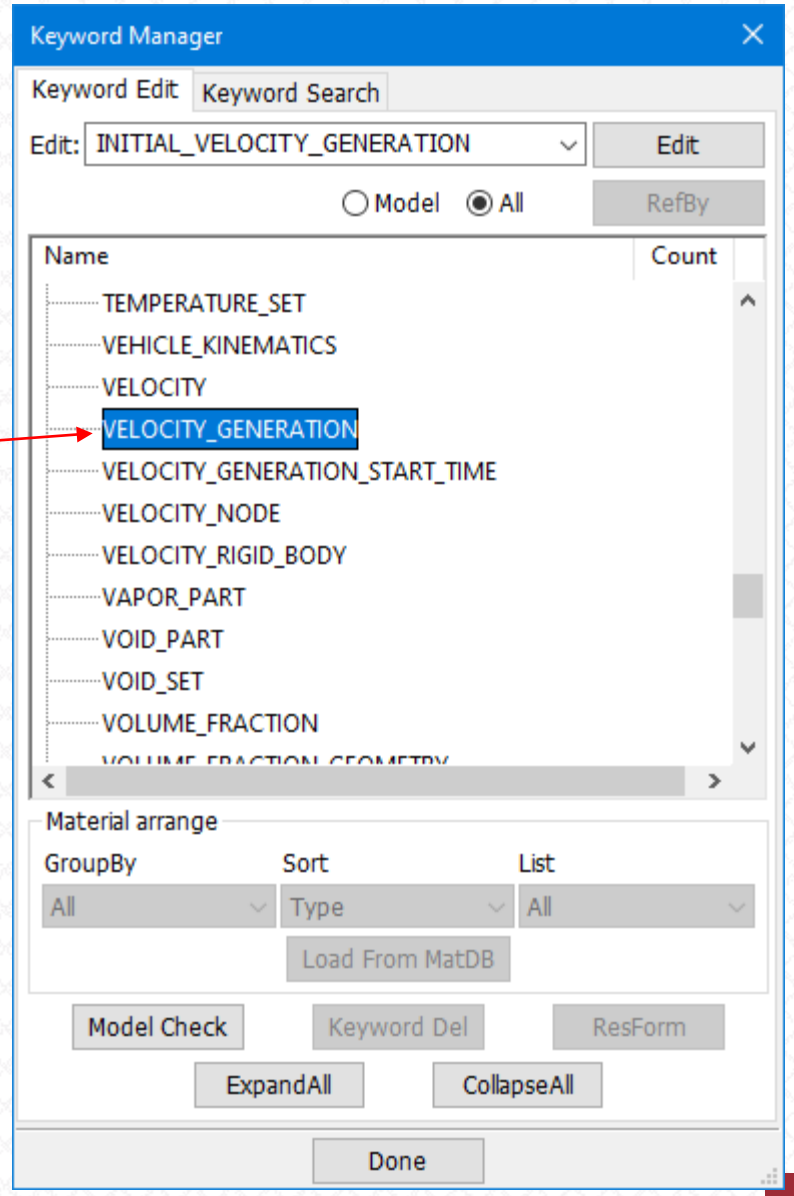
Steps

Step #	Desciption	
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

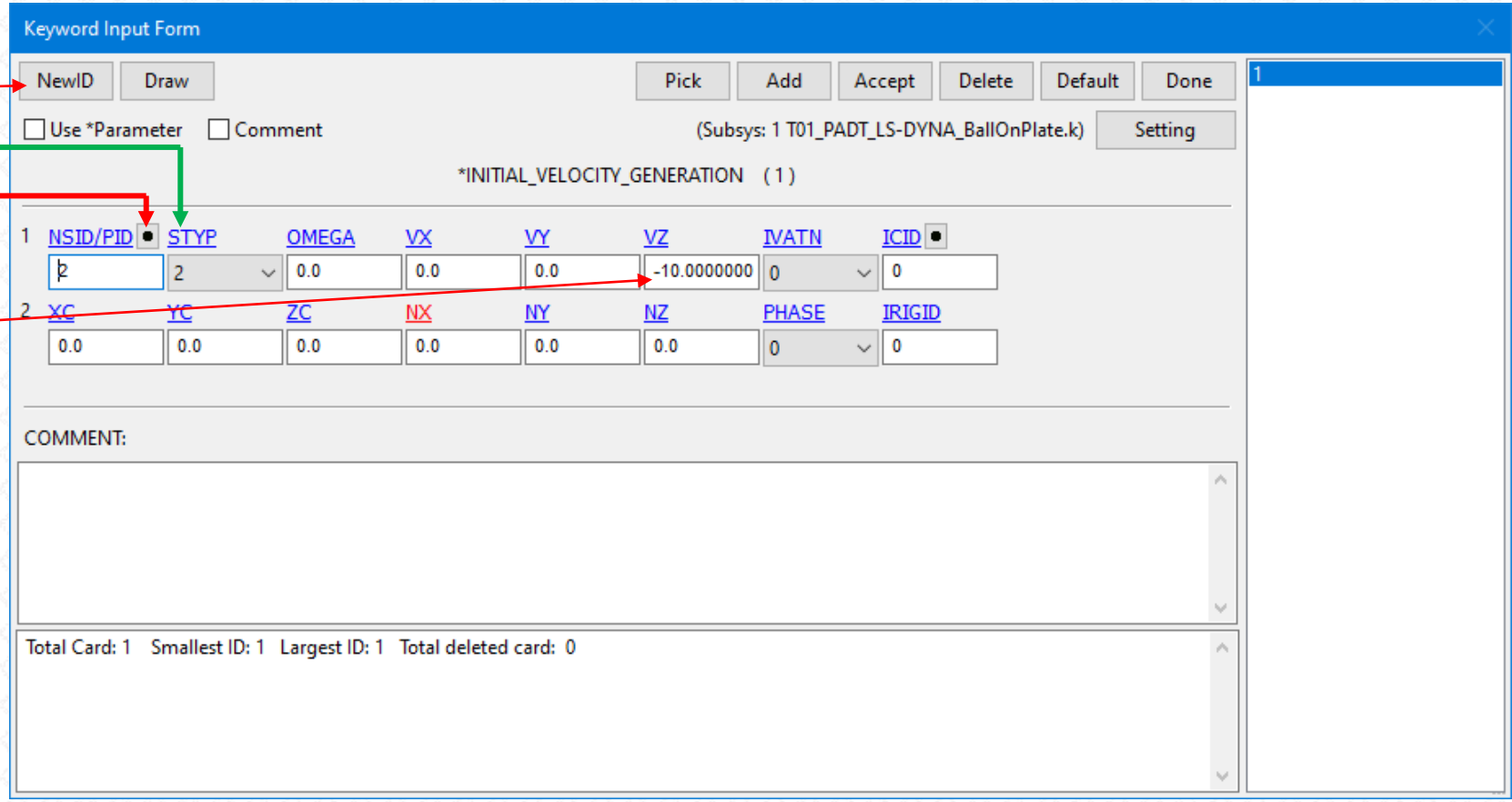
1. Select radio button “All” if it’s not selected
2. Expand [INITIAL]
3. Scroll down and double click on [VELOCITY_GENERATION]

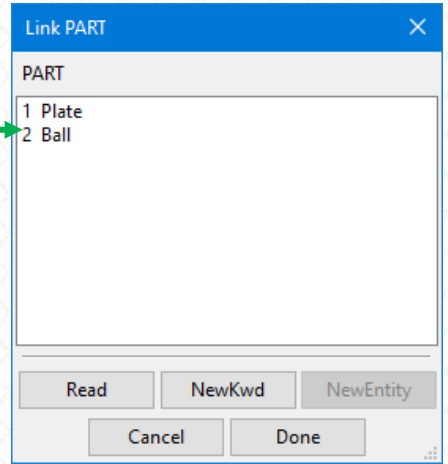




Initial Velocity

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



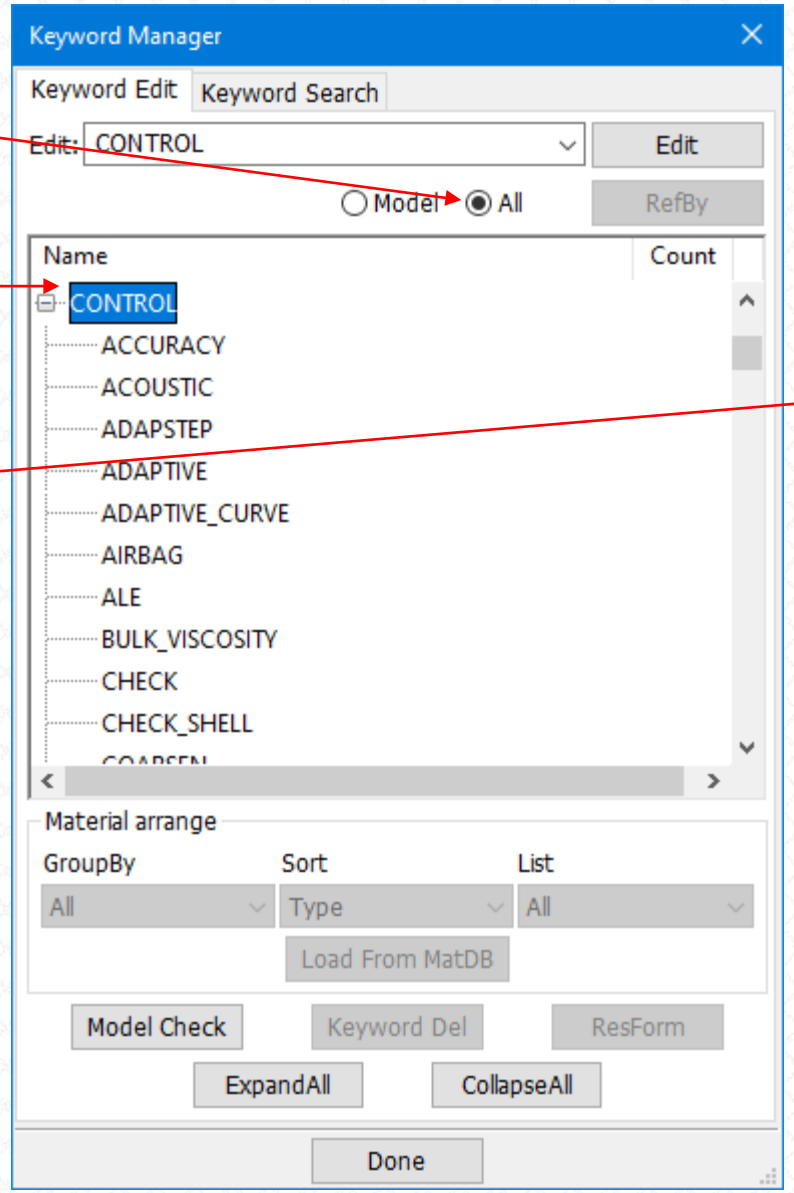


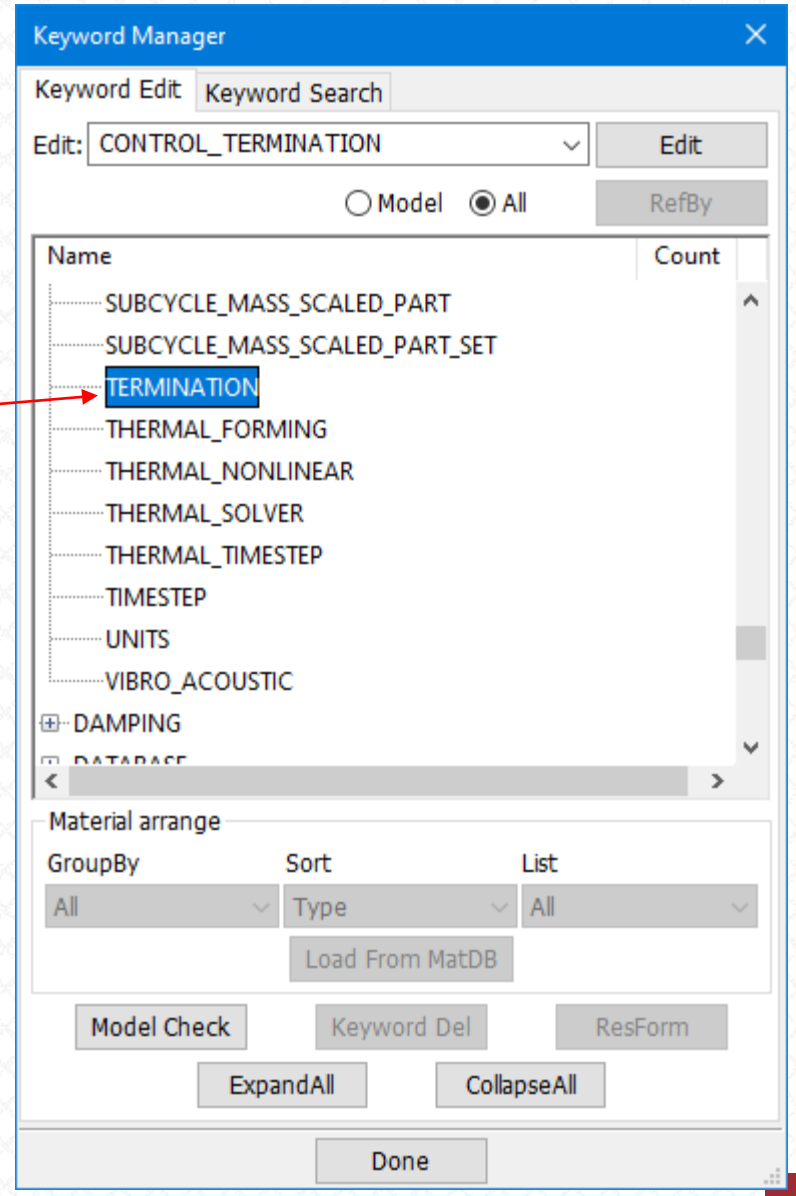
Steps

Step #	Desciption	
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]



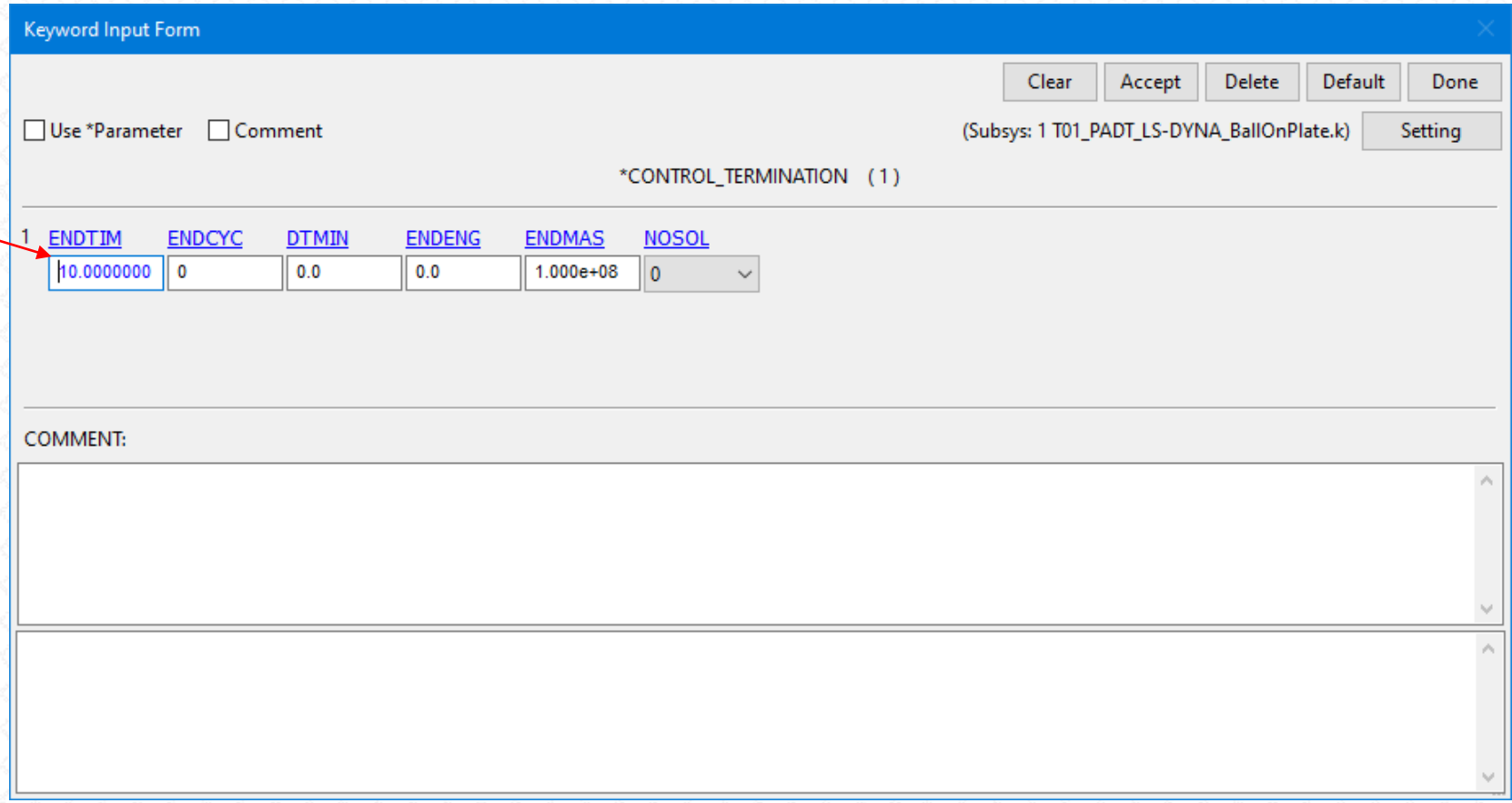


Analysis Time

1. ENDTIM: 10

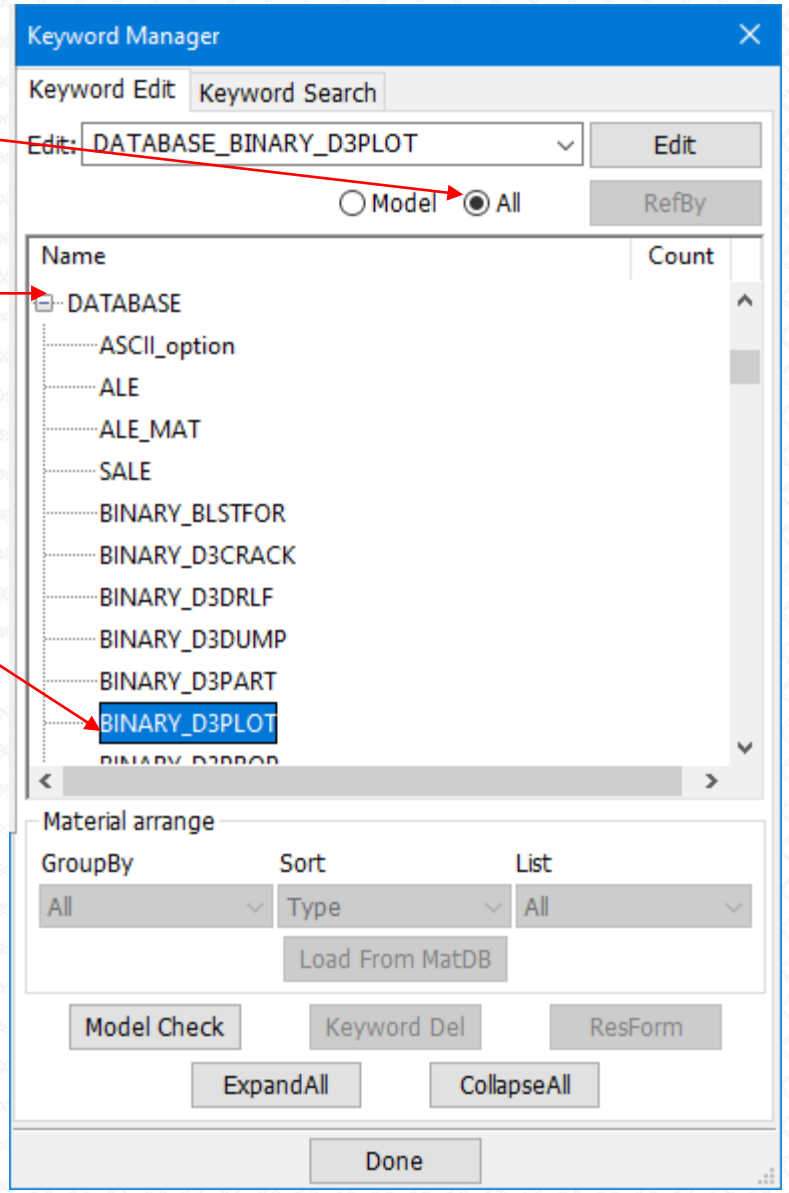
Note:
ENDTIM = 10 means that we are modeling 10 ms of simulation where Ball impacts a Plate, and bounces back (we'll see that 10 ms is sufficient for that)

2. Accept
3. Done



Output Controls

1. Select radio button “All” if it’s not selected
2. Expand [DATABASE]
3. Double click on [BINARY_D3PLOT]

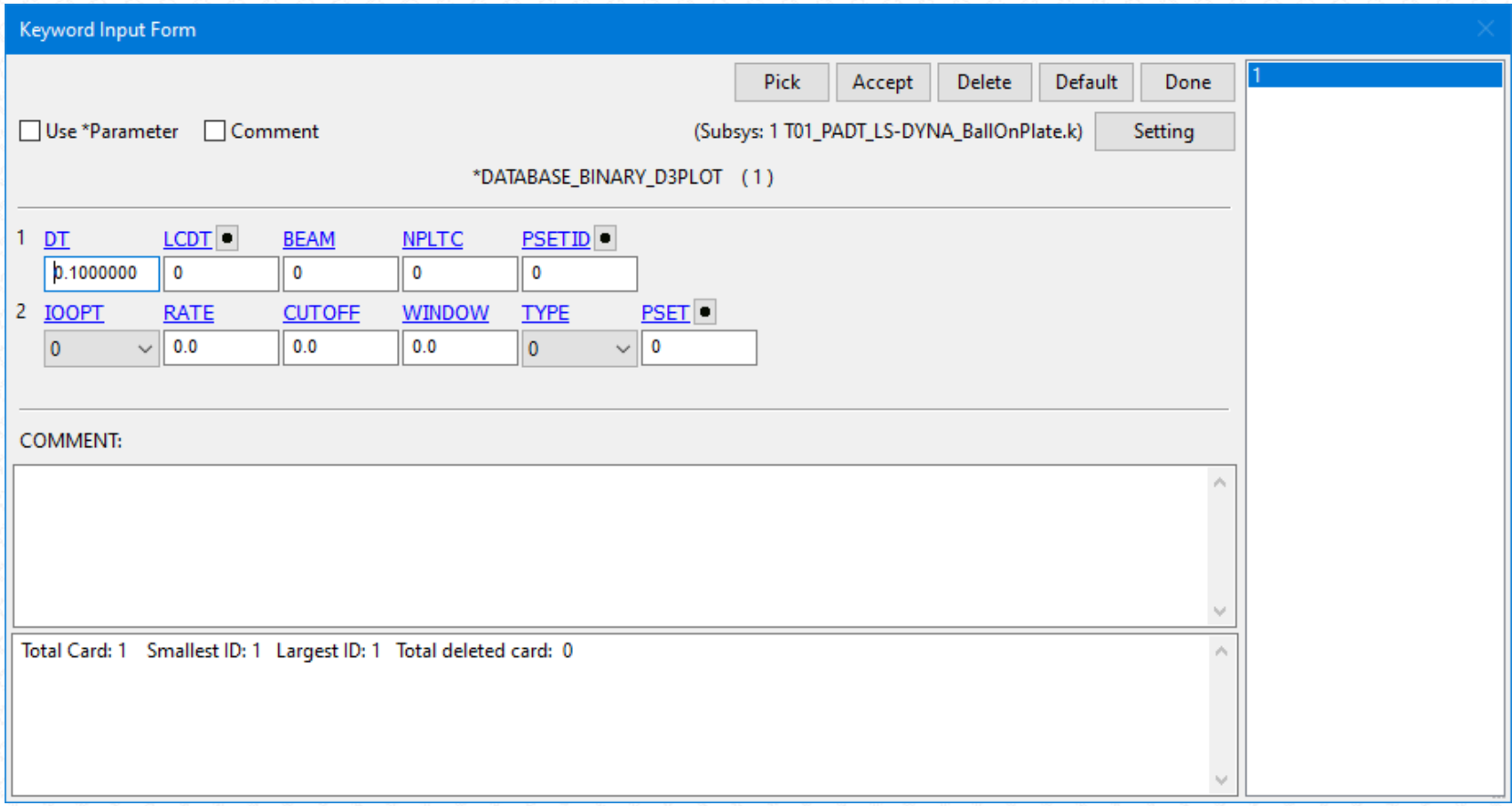


Output Controls

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 ms/ 0.1 ms = 100) with results written to disc.

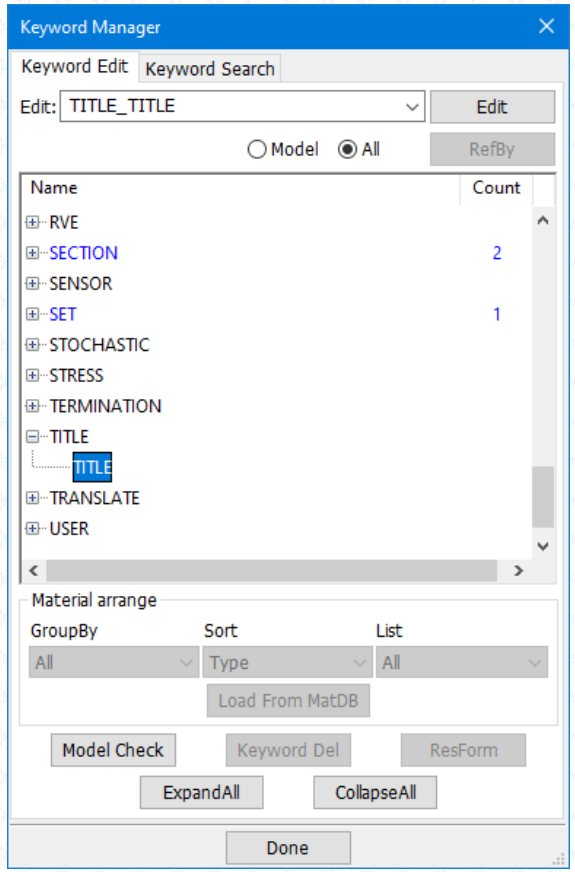
2. Accept
3. Done



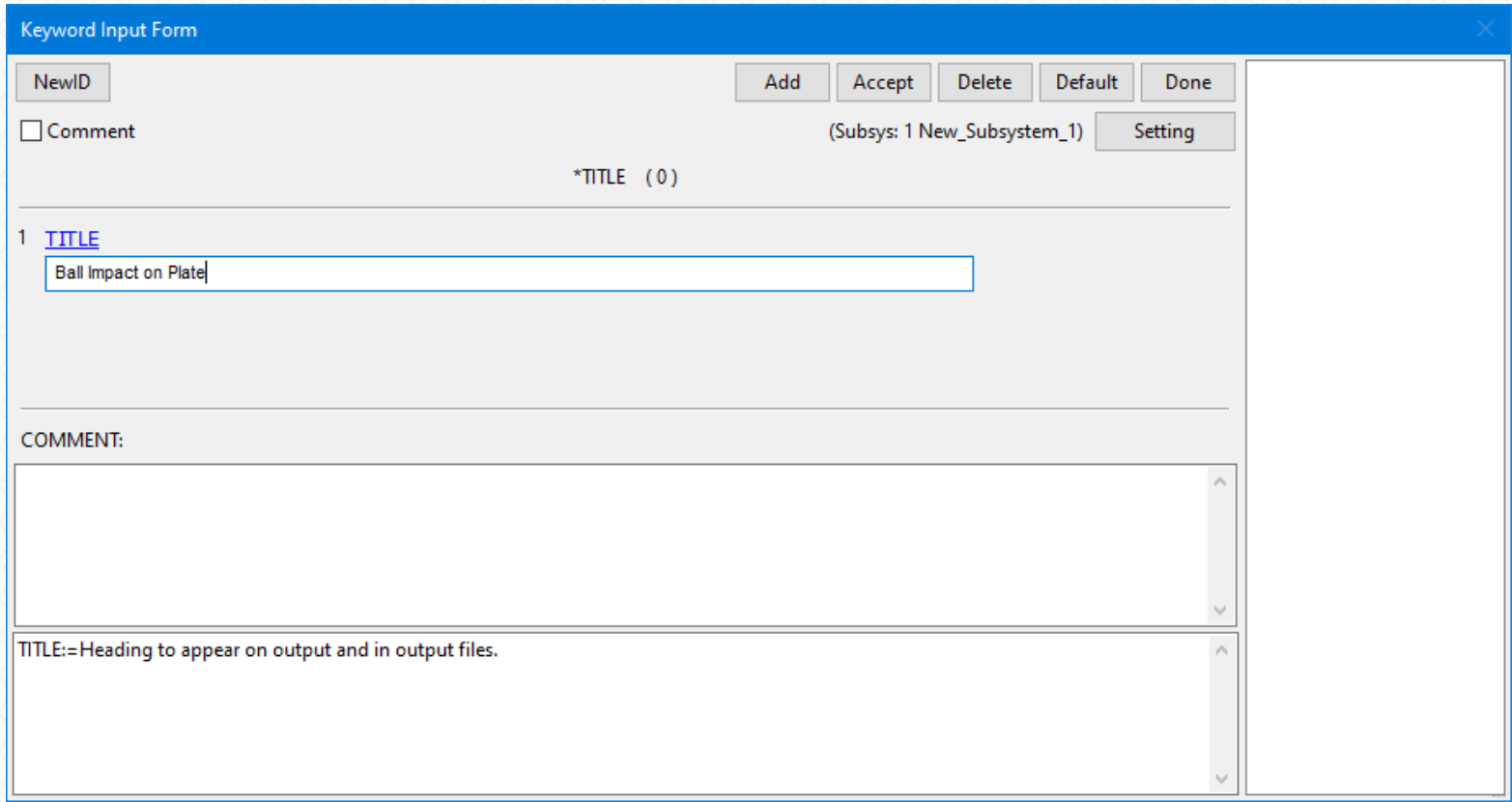
Title

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]



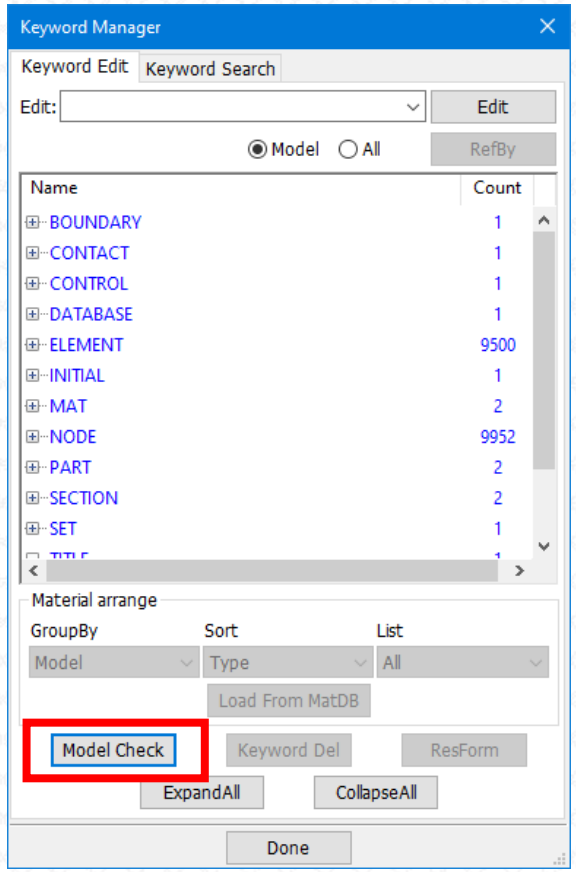
4. TITLE: Ball Impact on Plate
5. Accept
6. Done



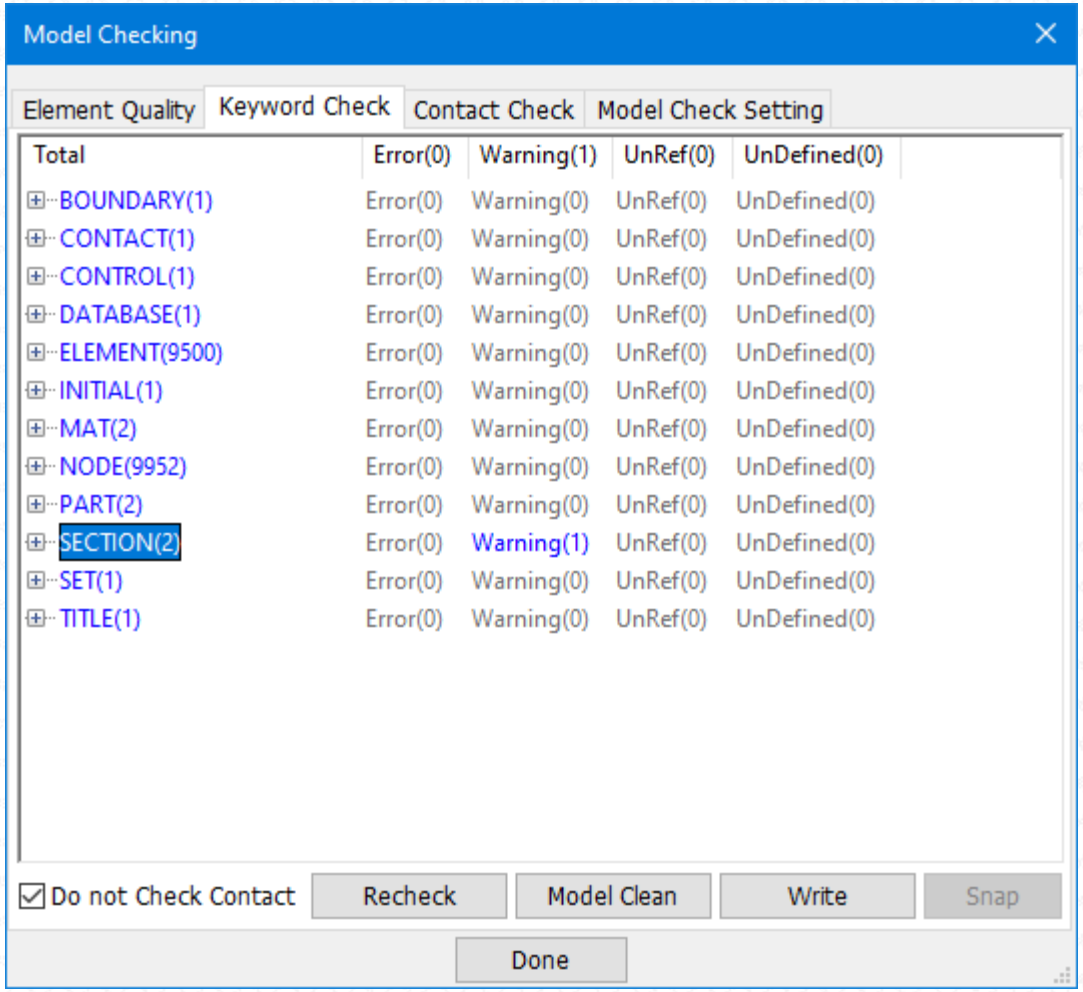
Model Check

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

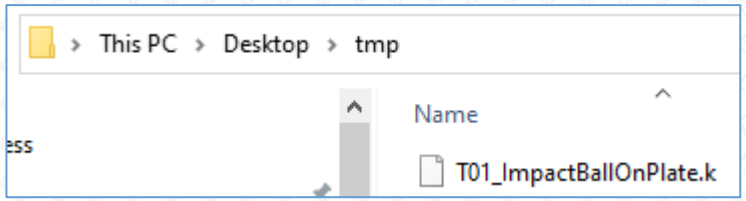


2. “Model Checking” popup window will appear
3. Make sure there are no Errors
4. Read Warnings, UnRef, UnDefined messages
5. Done



Save

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

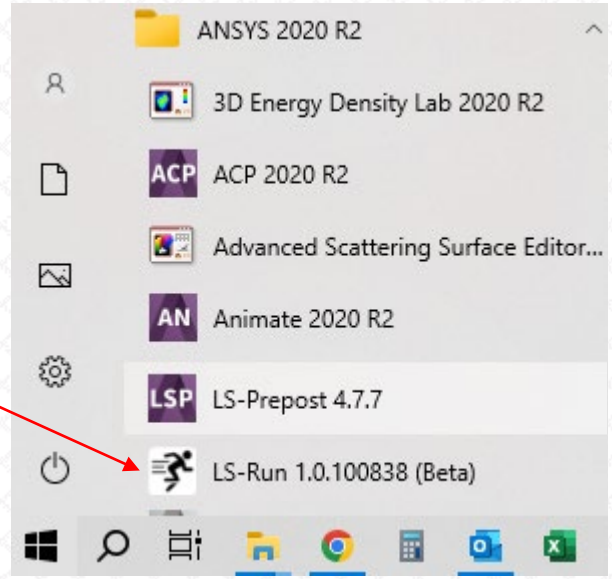


Steps

Step #	Desciption	
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	

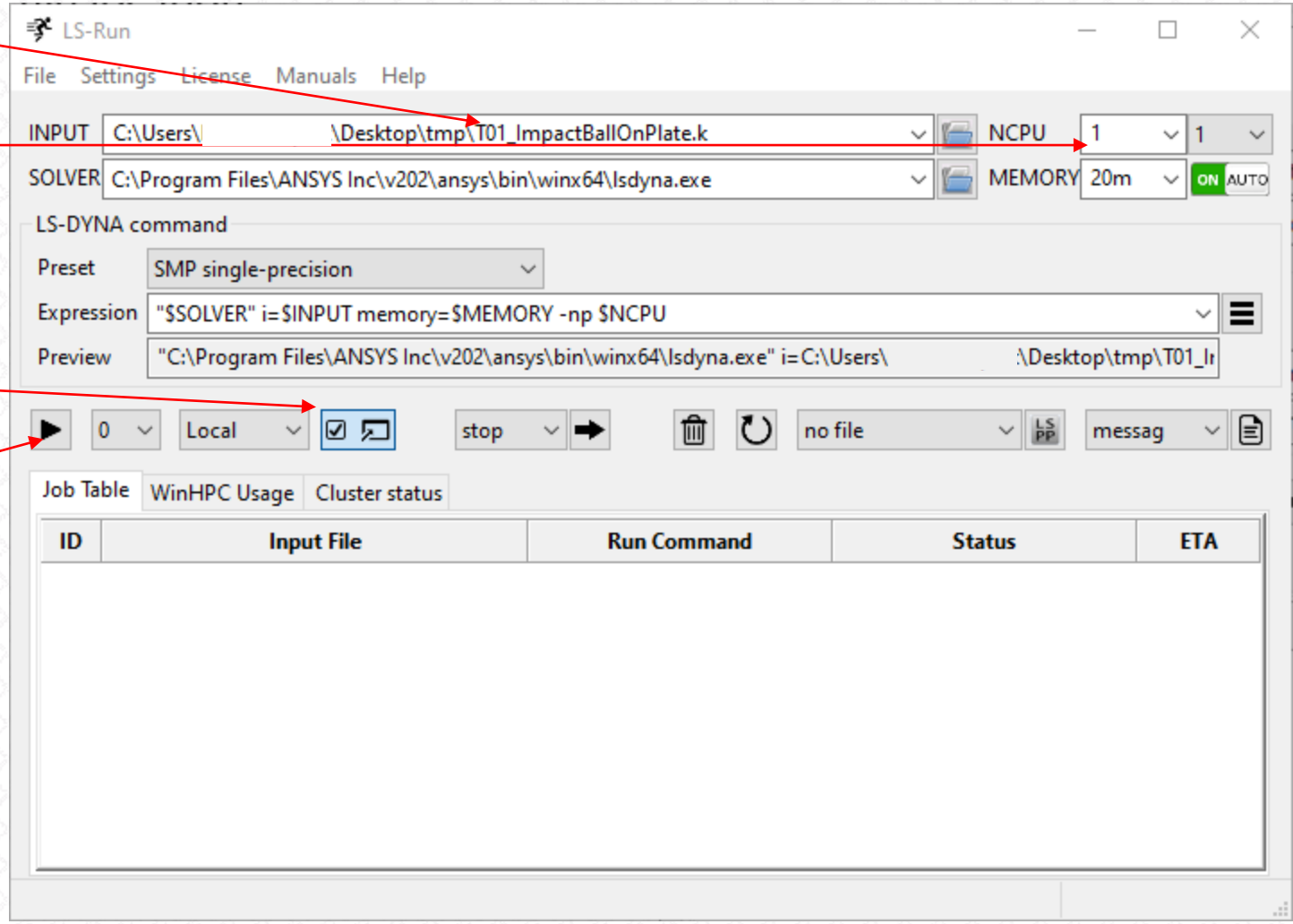
Submit Analysis in LS-Run

1. Start > ANSYS 2020 R2 > LS-Run



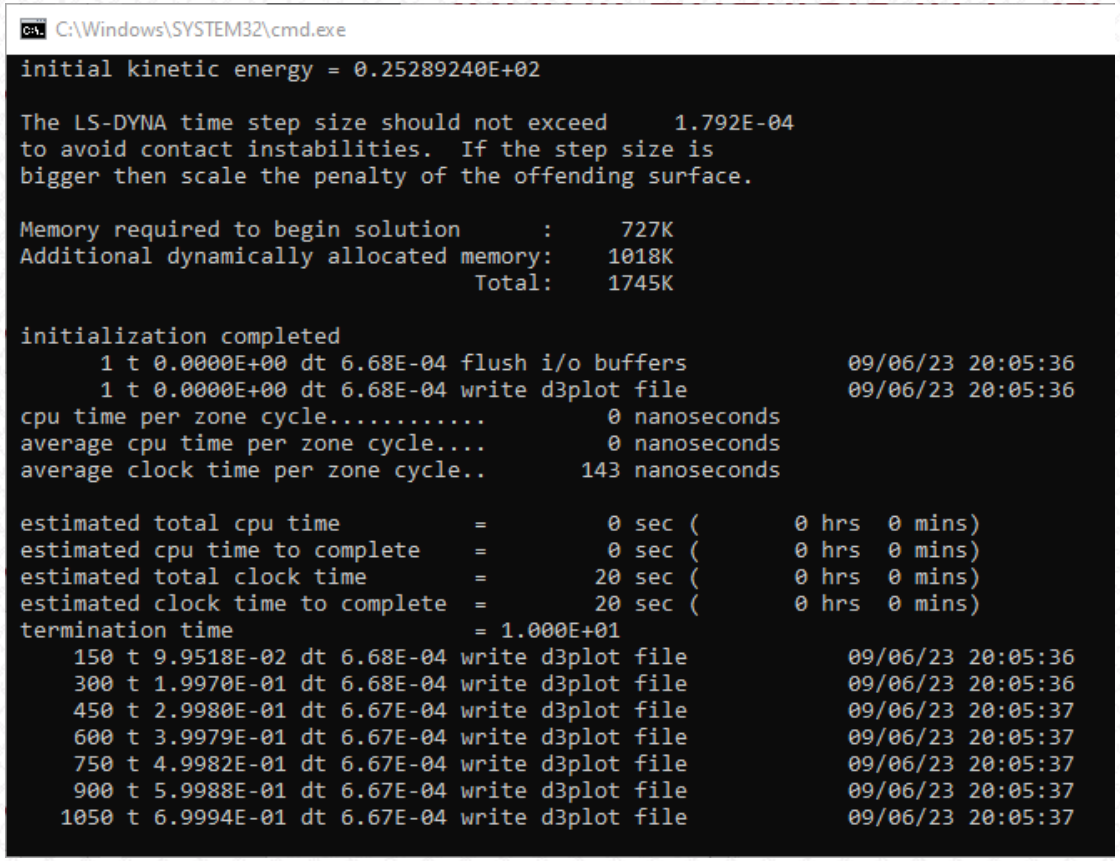
Submit Analysis in LS-Run

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"

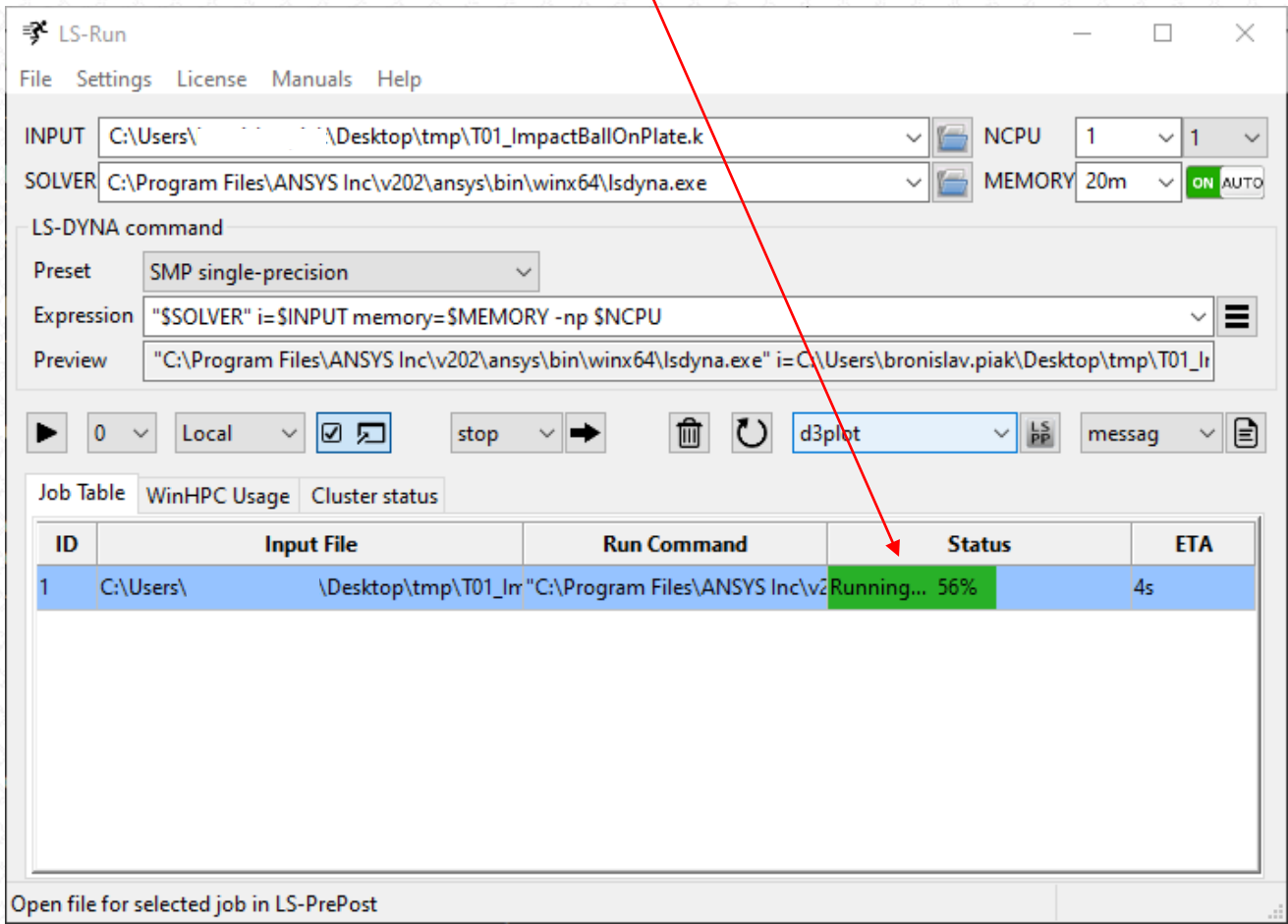


Submit Analysis in LS-Run

1. Command prompt window will display miscellaneous information



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

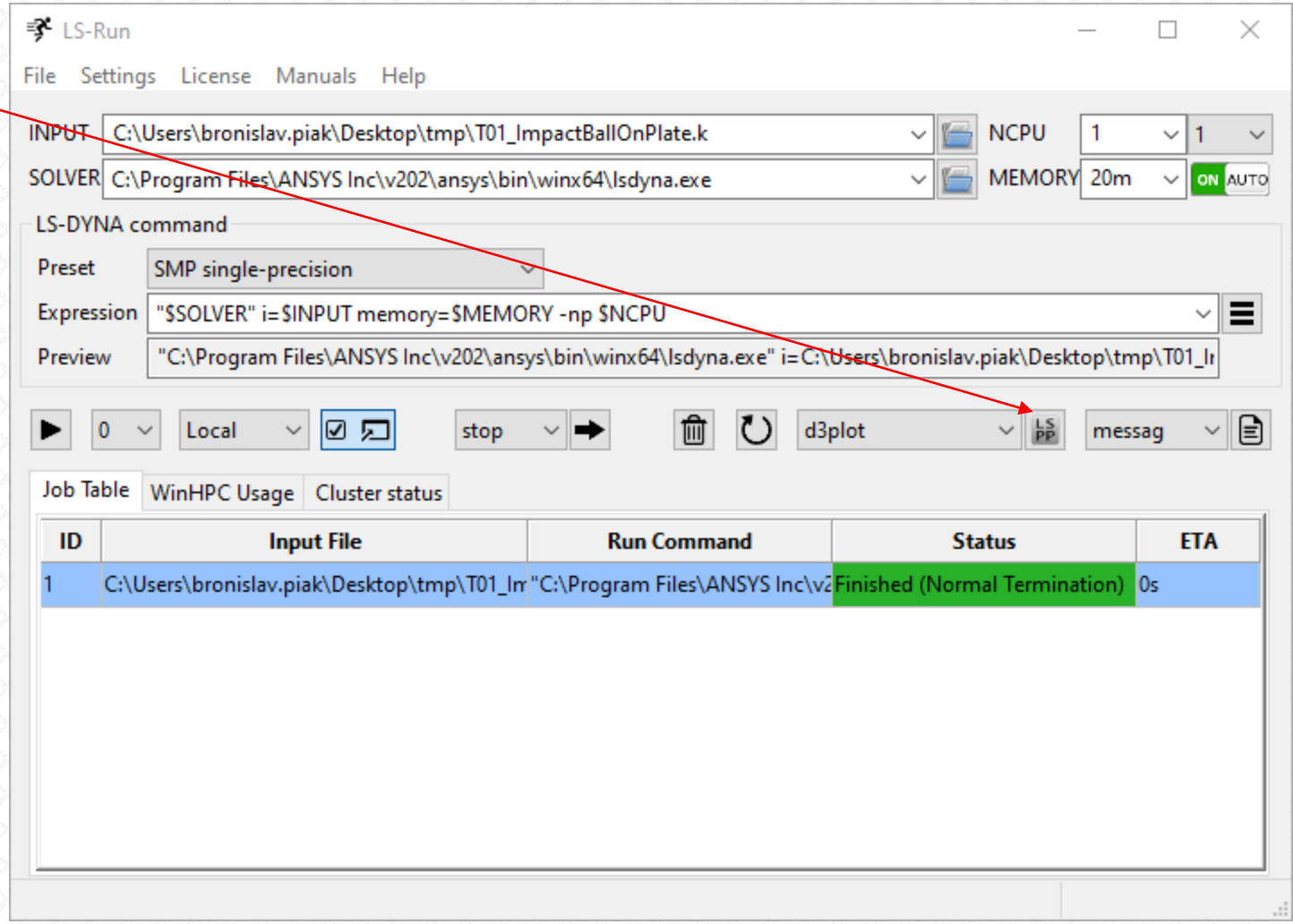


Steps

Step #	Desciption	
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

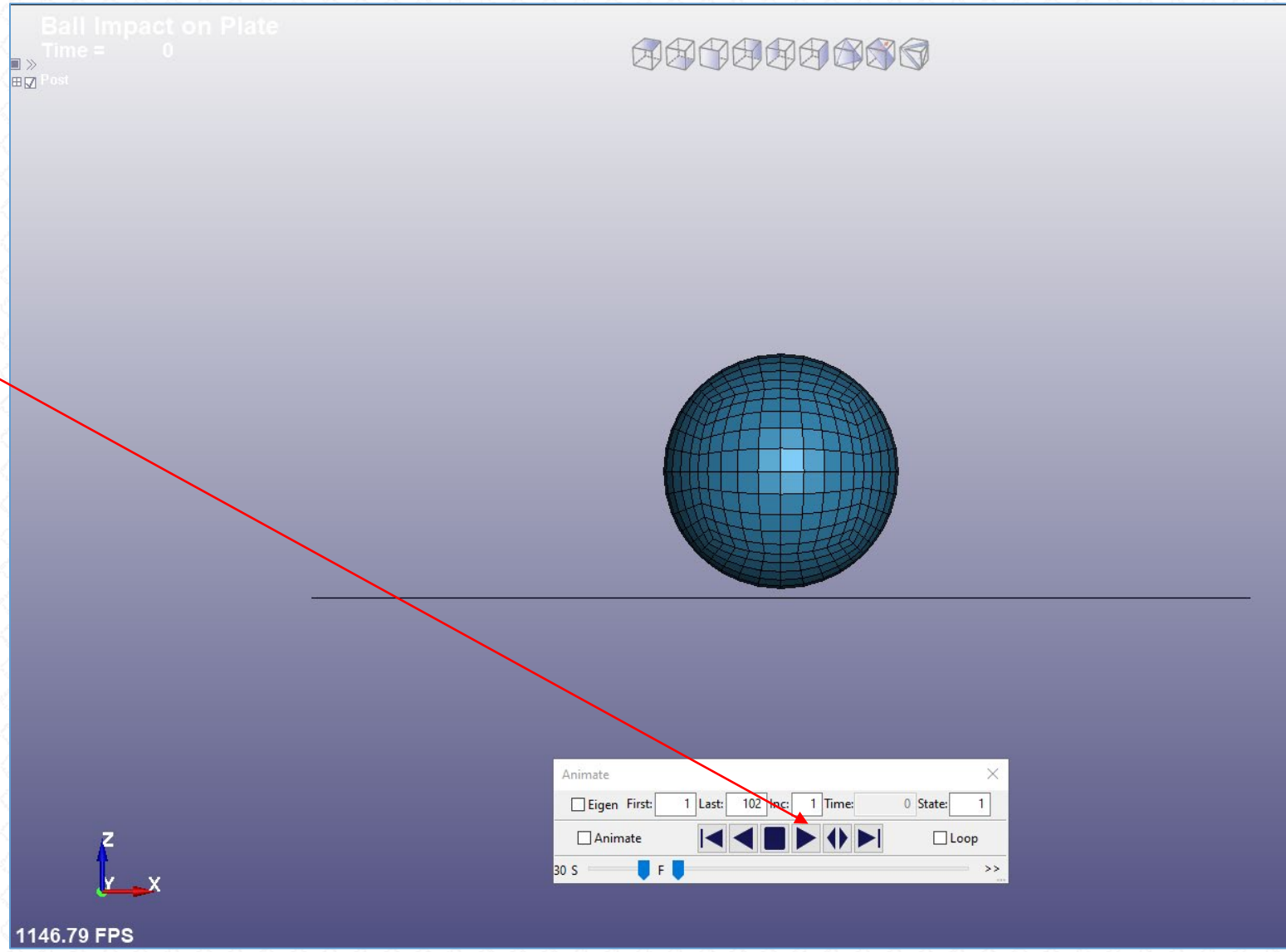
Postprocess Results in LS-Prepost

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

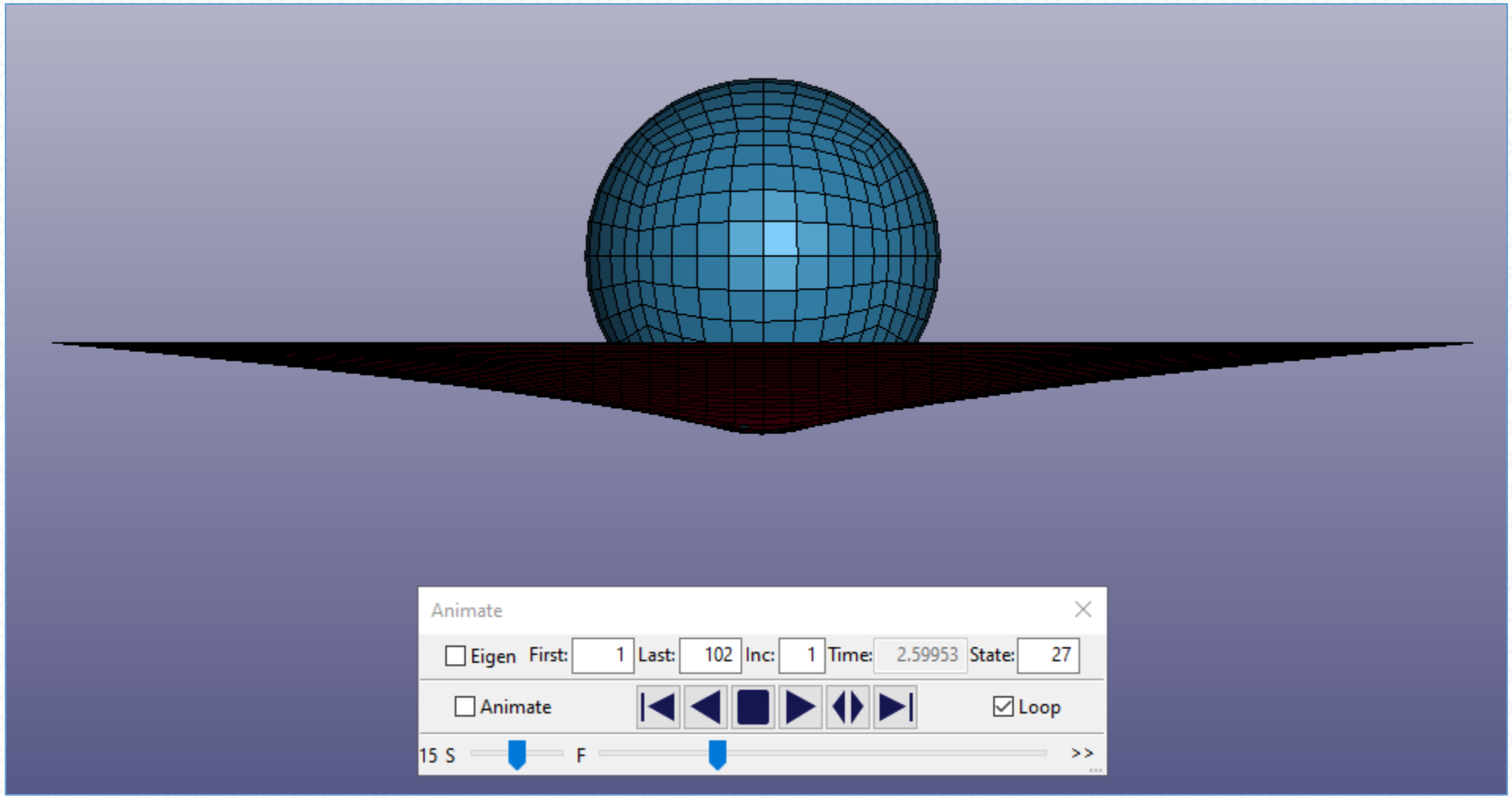


Postprocess Results in LS-Prepost

1. Click on “Play” button to animate the results

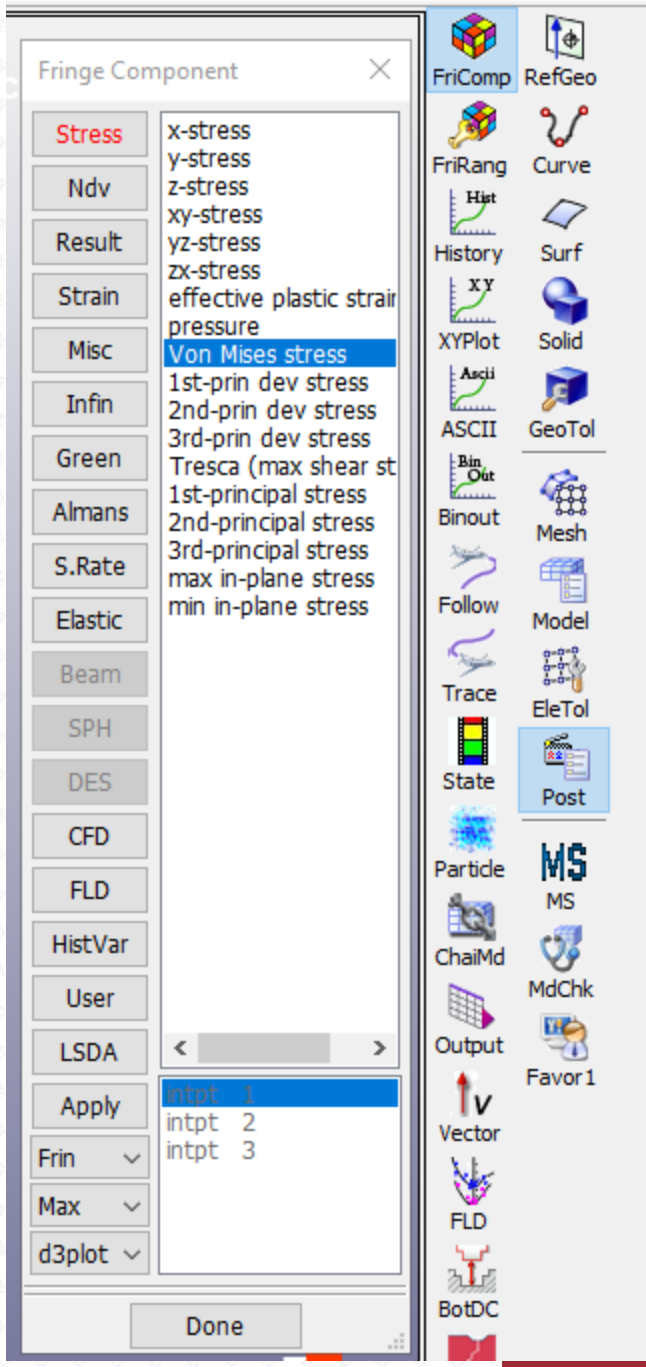


Postprocess Results in LS-Prepost

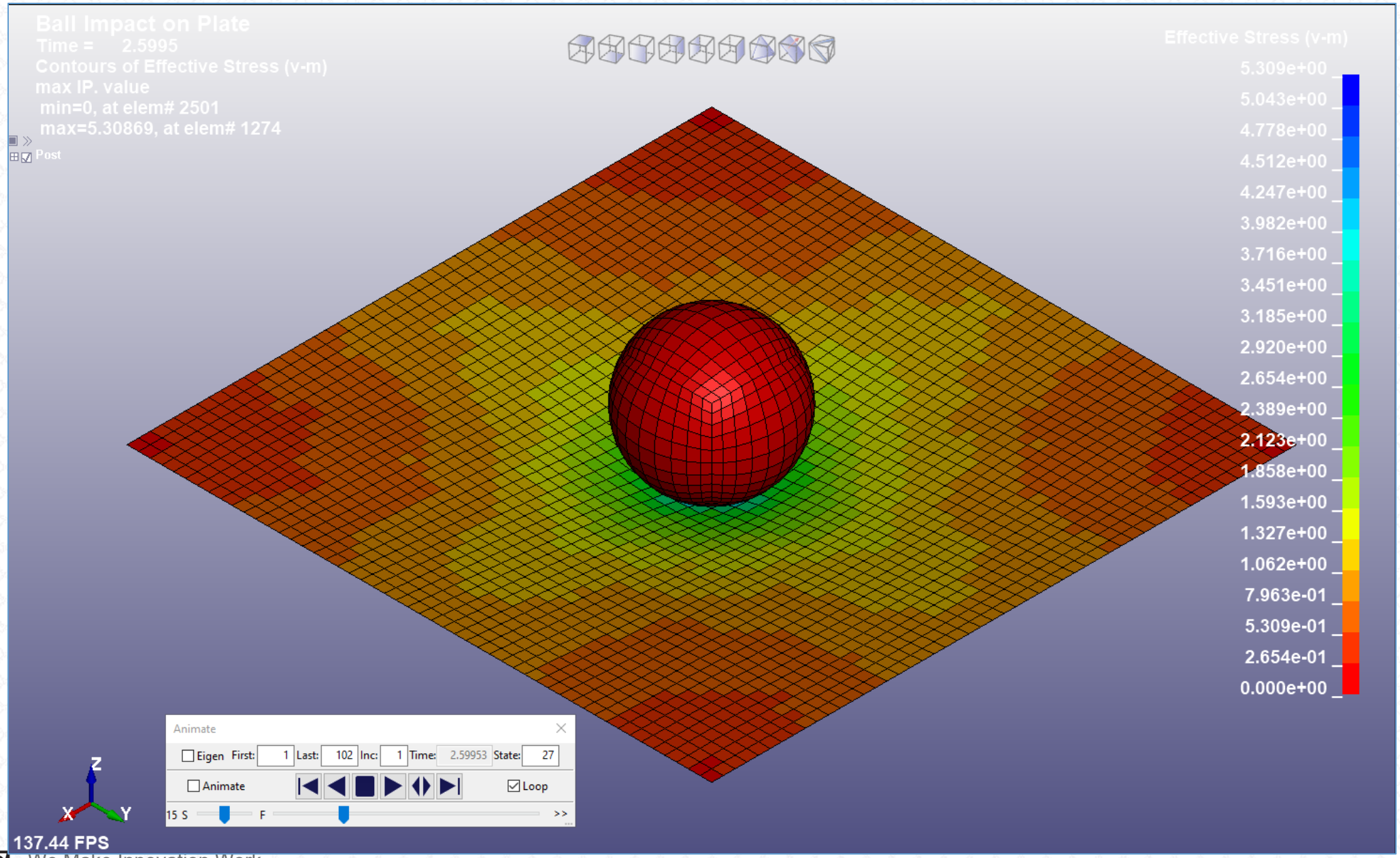


Postprocess Results in LS-Prepost

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!

```
$# LS-DYNA Keyword file created by LS-PrePost(R) V4.7.7 - 17Feb2020
$# Created on Sep-11-2023 (08:23:52)
*KEYWORD
*TITLE
$#                                                                         title
Ball Impact on Plate
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas      nosol
     10.0         0       0.0       0.01.000000E8         0
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam      npltc    psetid
       0.1         0         0          0         0
$#   ioopt      rate    cutoff    window      type      pset
         0       0.0       0.0       0.0         0         0
*BOUNDARY_SPC_SET
$#     nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         0         0         0
*SET_NODE_LIST_TITLE
NODESET(SPC) 1
$#      sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0MECH
$#     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      2586      2587      2588      2589      2590      2591      2592      2593
      2594      2595      2596      2597      2598      2599      2600      2601
```