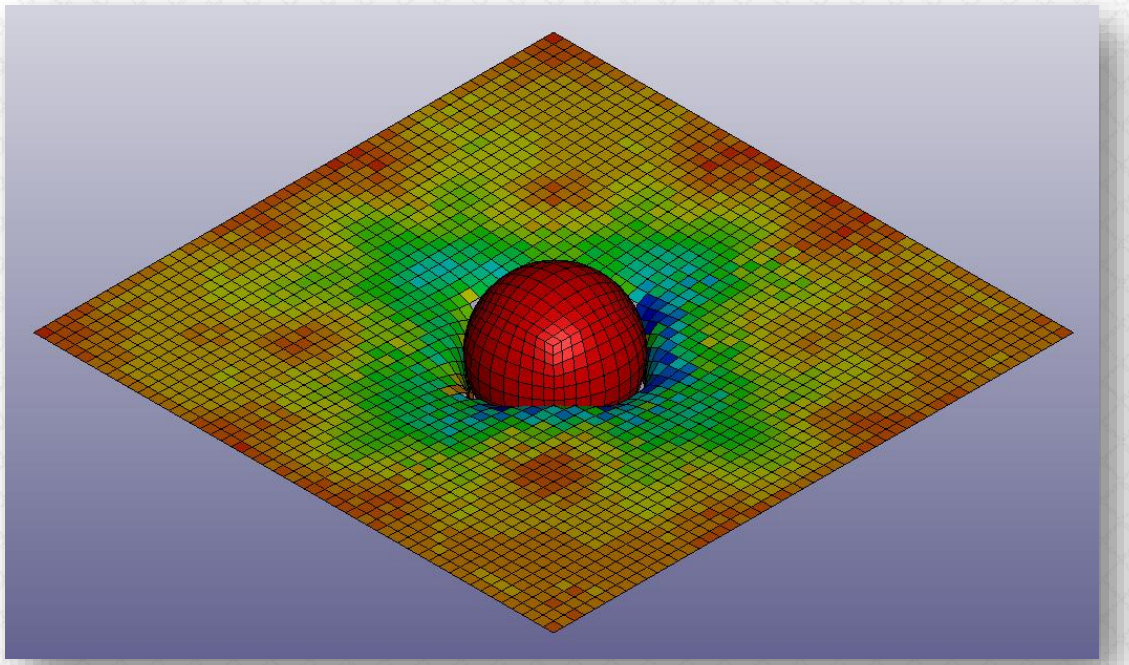
Introduction to LS-DYNA
Tutorial #2
Ball Impacting a Plate
(LS-DYNA keyword edition)

```
$# 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
      2585      2584      2583      2582      2581      2580      2579      2578
      2577      2576      2575      2574      2573      2572      2571      2570
      2569      2568      2567      2566      2565      2564      2563      2562
      2561      2560      2559      2558      2557      2556      2555      2554
      2553      2552      2551      2193      2142      2091      2040      1989
      1938      1887      1836      1785      1734      1683      1632      1581
      1530      1479      1428      1377      1326      1275      1224      1173
      1122      1071      1020       969       918       867       816       765
       714       663       612       561       510       459       408       357
       306       255       204       153       102        51      2244      2295
      2346      2397      2448      2499      2550        42        41        40
        39        38        37        36        35        34        33        32
        31        30        29        28        27        26        25        24
        23        22        21        20        19        18        17        16
        15        14        13        12        11        10         9         8
         7         6         5         4         3         2         1        43
        44        45        46        47        48        49        50       970
      1021      1072      1123      1174      1225      1276      1327      1378
      1429      1480      1531      1582      1633      1684      1735      1786
      1837      1888      1939      1990      2041      2092      2143      2194
      2245      2296      2347      2398      2449      2500       919       868
       817       766       715       664       613       562       511       460
       409       358       307       256       205       154       103        52
```



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)
- Notepad++

Goal

- Learn basics of the syntax of LS-DYNA keyword file (.k)
- Configure Notepad++ to highlight LS-DYNA keyword syntax

```
$# 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       0.0 1.000000E8         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
      2585      2584      2583      2582      2581      2580      2579      2578
      2577      2576      2575      2574      2573      2572      2571      2570
      2569      2568      2567      2566      2565      2564      2563      2562
      2561      2560      2559      2558      2557      2556      2555      2554
      2553      2552      2551      2193      2142      2091      2040      1989
      1938      1887      1836      1785      1734      1683      1632      1581
      1530      1479      1428      1377      1326      1275      1224      1173
      1122      1071      1020       969       918       867       816       765
       714       663       612       561       510       459       408       357
       306       255       204       153       102        51      2244      2295
      2346      2397      2448      2499      2550        42        41        40
        39        38        37        36        35        34        33        32
        31        30        29        28        27        26        25        24
        23        22        21        20        19        18        17        16
        15        14        13        12        11        10         9         8
         7         6         5         4         3         2         1        43
        44        45        46        47        48        49        50       970
      1021      1072      1123      1174      1225      1276      1327      1378
      1429      1480      1531      1582      1633      1684      1735      1786
      1837      1888      1939      1990      2041      2092      2143      2194
      2245      2296      2347      2398      2449      2500       919       868
       817       766       715       664       613       562       511       460
       409       358       307       256       205       154       103        52
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID
$#     cid                                                                 title
         0Contact Ball to Plate
$#    ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         2         1         3         3         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.0       0.0       0.0       0.0       0.0         0 0.01.00000E20
$#     sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
       1.0       1.0       0.0       0.0       1.0       1.0       1.0       1.0
```

Tutorial #2

We will explore and modify LS-DYNA keywork file .k created in Tutorial #1
(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

Specifically, we will modify:
Plate thickness
Plate material

Ball

Dimensions : radius = 25mm

Material: Steel

density RO = 7.83e-6 kg/mm3

Elastic Modulus E = 207 GPa

Poisson's ratio PR = 0.3

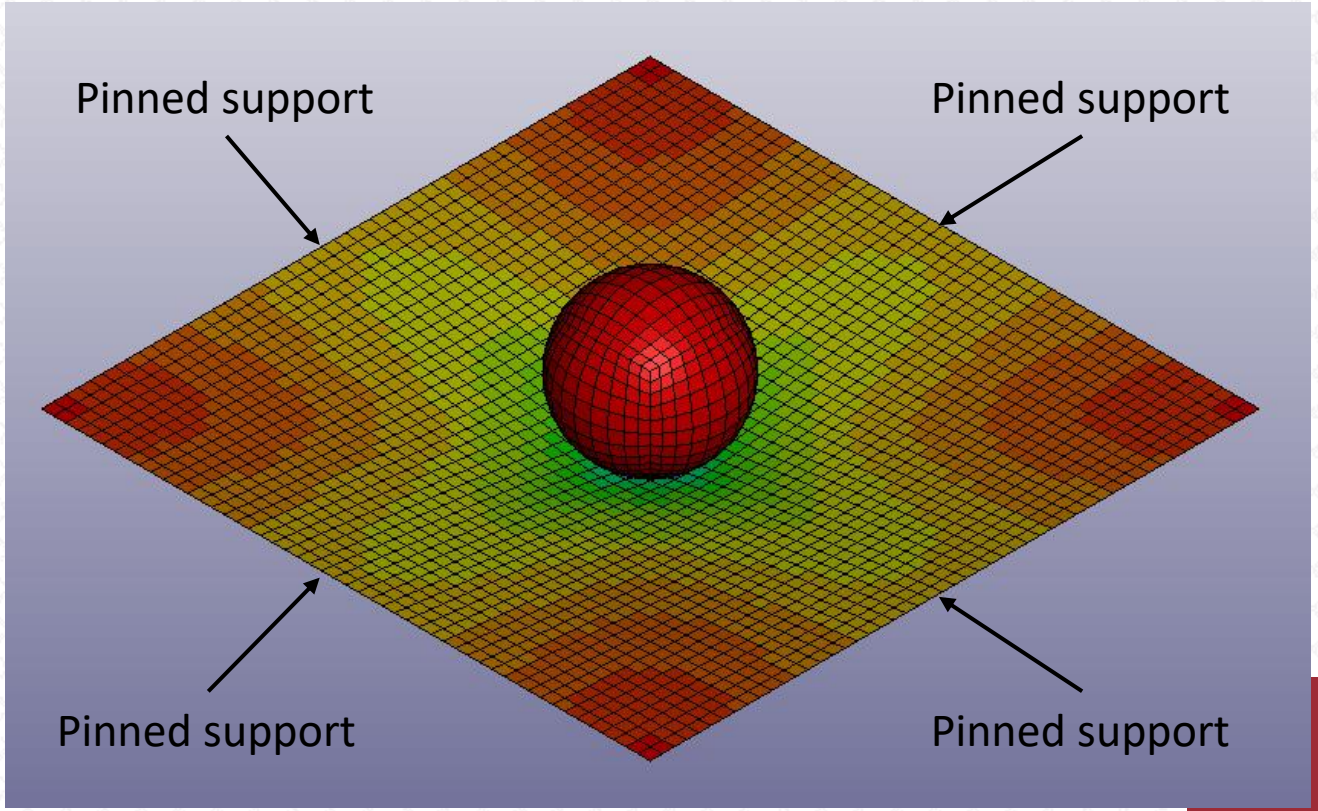
Ball will be modeled as a Rigid Body

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

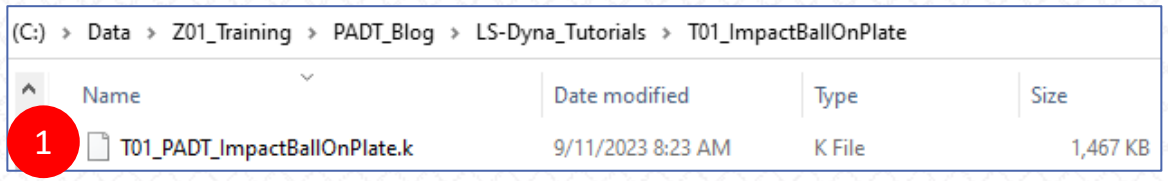
We will use the [kg-mm-ms-kN] units set in this tutorial.

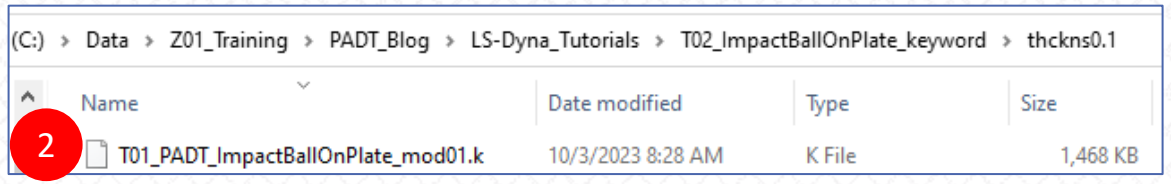
Consistent set of units for Steel

							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

LS-DYNA keyword file

1. Navigate to your Tutorial #1 folder and find file with extension .k.
2. Copy it to another folder (for example, T02_ImpactBallOnPlate_keyword), and rename to T01_PADT_ImpactBallOnPlate_mod01.k
3. Open the file in Notepad++





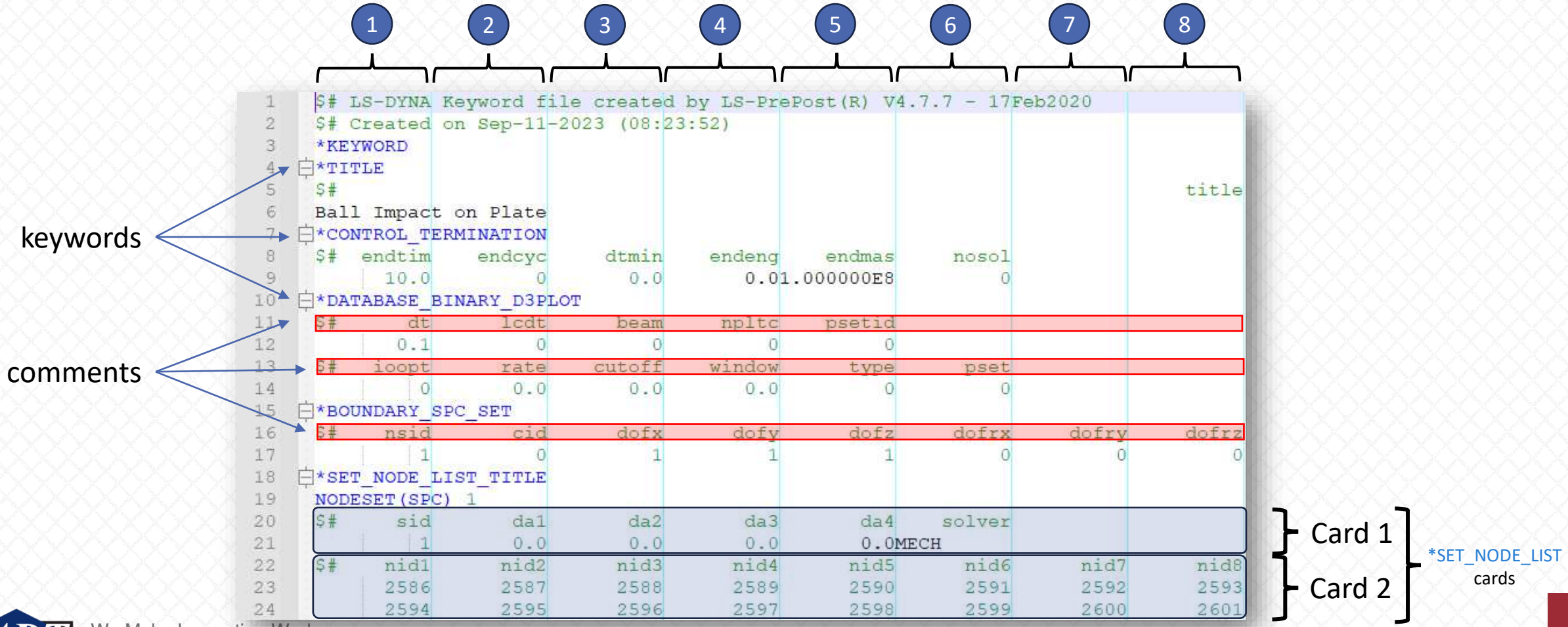
3

```
$# 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
```

Note:
See Appendix 2 to configure Notepad++ to color-code LS-DYNA keyword file

LS-DYNA keyword file

1. Keywords start with * (for example, *KEYWORD, *CONTROL_TERMINATION)
2. 8 fields of 10 characters each (or 10 fields of 8 characters each, e.g. *NODE keyword)
3. Use $ for comments
4. Keywords, materials and solvers are extensively described in LS-DYNA Keyword User's Manual Volume I, II and III respectively. See Appendix 1 on how to access them.



Steps

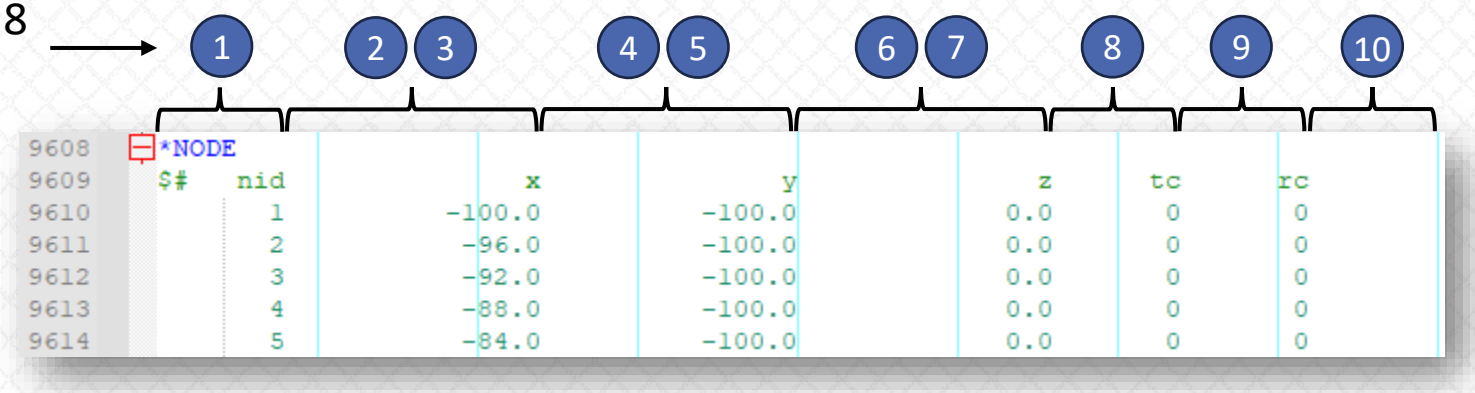
Step #	Desciption	
1	Create Geometry and Mesh	LS-DYNA keyword file from Tutorial #1 already contains geometry/FE mesh of a Plate and a Ball
2	Boundary Conditions	
3	Material Properties	
4	Section/Element Properties	
5	Assign Material and Section Properties to Parts	
6	Contact	
7	Initial Velocity	
8	Analysis Time and Output Controls	
9	Submit Analysis in LS-Run	
10	Postprocess results in LS-Prepost	

FE Mesh / Nodes / Elements

1. Search (CTRL+F) in Notepad++ to find *NODE keyword. No changes needed here. Let's just learn about the syntax of *NODE keyword.

*NODE: define a node and its coordinates in the global coordinate system

There are 10 fields of 8 characters each



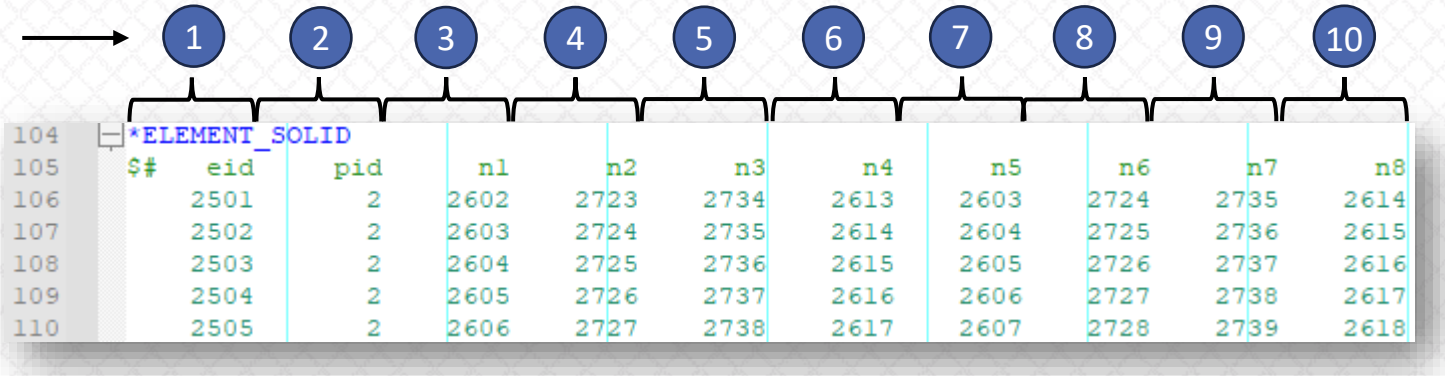
Variable	Description (Card 1 | include as many cards as desired)
nid	Node number / ID
x	X coordinate
y	Y coordinate
z	Z coordinate
tc	Translational Constraint
rc	Rotational Constraint

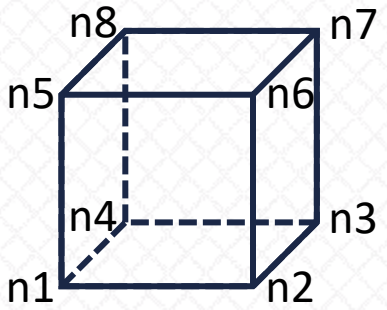
FE Mesh / Nodes / Elements

1. Search (CTRL+F) in Notepad++ to find *ELEMENT_SOLID keyword. No changes needed here.

*ELEMENT_SOLID: define 3D solid elements

There are 10 fields of 8 characters each





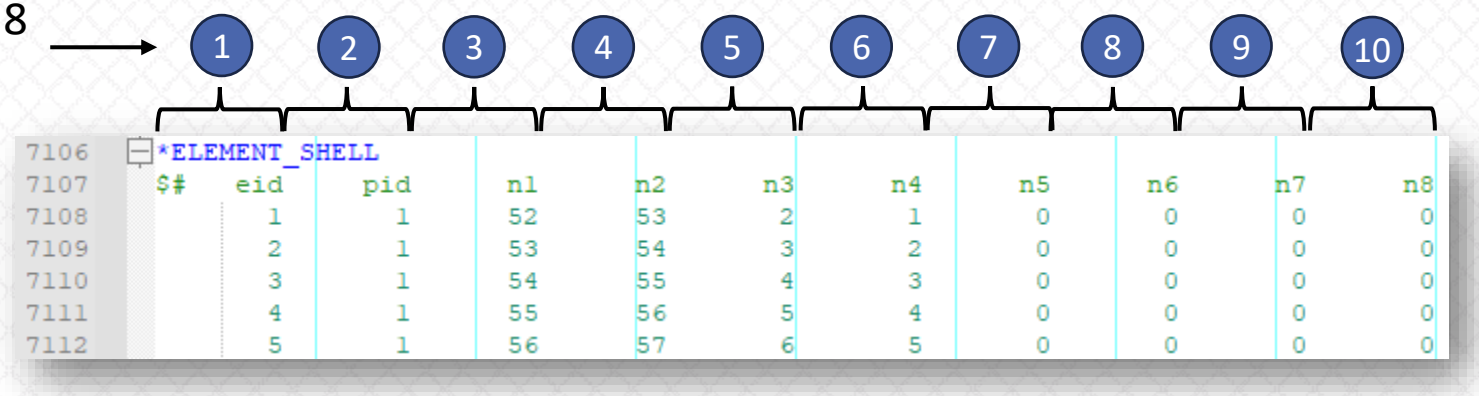
Variable	Description (Card 1 | For elements with 4-8 nodes the cards in the format of LS-DYNA v 940-970 are still supported. The older format doesn't include Card 2)
eid	Element ID
pid	Part ID
n1	Nodal point 1
n2	Nodal point 2
n3	Nodal point 3
…	…
n8	Nodal point 8

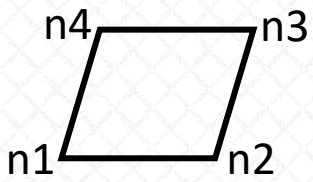
FE Mesh / Nodes / Elements

1. Search (CTRL+F) in Notepad++ to find *ELEMENT_SHELL keyword. No changes needed here.

*ELEMENT_SHELL: define 3, 4, 6, and 8 node elements including 3D shells, membranes, 2D plane stress, plane strain, and axisymmetric solids.

There are 10 fields of 8 characters each





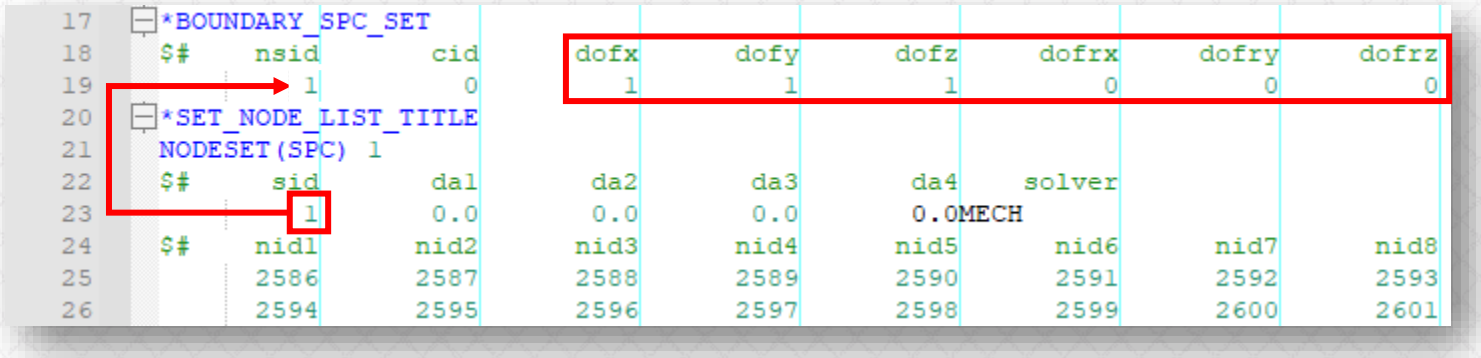
Variable	Description (Card 1)
eid	Element ID
pid	Part ID
n1	Nodal point 1
n2	Nodal point 2
n3	Nodal point 3
n4	Nodal point 4
n5 - n8	Mid-side nodes for eight node shells

Steps

Step #	Desciption	
1	Create Geometry and Mesh	
2	Boundary Conditions	LS-DYNA keyword file from Tutorial #1 already contains 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. Search (CTRL+F) in Notepad++ to find *BOUNDARY_SPC_SET and *SET_NODE_LIST_TITLE keywords. No changes needed here.



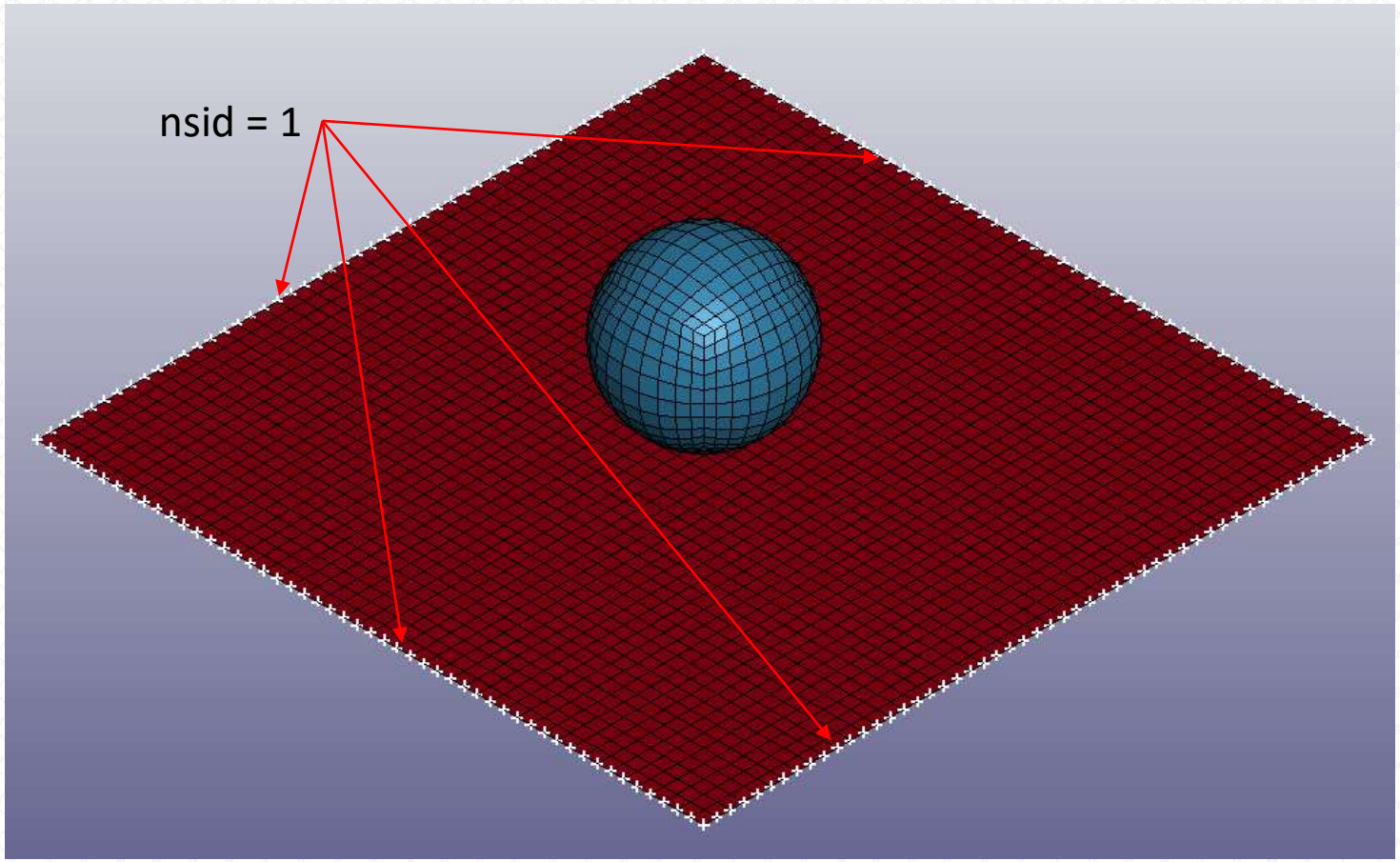
*BOUNDARY_SPC_SET define nodal single point constraints applied to a nodal set	
Variable	**Description (Card 1)**
nsid	**Node ID or nodal set ID**
cid	Coordinate system ID
dofx	**Insert 1 for translational constraint in local x-direction**
dofy	**Insert 1 for translational constraint in local y-direction**
dofz	**Insert 1 for translational constraint in local z-direction**
dofrx	**Insert 1 for rotational constraint about local x-axis**
dofry	**Insert 1 for rotational constraint about local y-axis**
dofrz	**Insert 1 for rotational constraint about local z-axis**

*SET_NODE_LIST_TITLE define a nodal set with some identical or unique attributes	
Variable	**Description (Card 1 | this card is required)**
sid	**Set ID**
da1	First nodal attribute default value
da2	Second nodal attribute default value
da3	Third nodal attribute default value
da4	Fourth nodal attribute default value
solver	Name of solver using this set (MECH, CESE, etc.)

Variable	Description (Card 2a | included if and only if the keyword option is unset, LIST, or LIST_SMOOTH. Include as many cards as needed)
nid1	Node ID 1
…	…
nid8	Node ID 8

Boundary Conditions

*BOUNDARY_SPC_SET and *SET_NODE_LIST_TITLE keywords allowed to specify boundary conditions on all outer edges as shown below.



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 modify material property for the Plate, and keep Rigid material property for the Ball as is
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 | Plate

1. Search (CTRL+F) in Notepad++ to find *MAT_ELASTIC_TITLE keyword.

```
*MAT_ELASTIC_TITLE
Steel
$#     mid        ro         e        pr        da        db  not used
         1 7.83000E-6     207.0       0.3       0.0       0.0       0.0
```

2. Replace all 4 lines with the following:

```
*MAT_POWER_LAW_PLASTICITY
$#     MID       RHO         E        PR         K         N       src       srp
$                Kg/mm^3     GPa                 GPa
         1 7.7126E-06 8.8948E+01     0.330 1.2463E-01 0.070747       0.0       0.0
$#   sigy        vp      epfs
       0.0       0.0    0.0972
```

*MAT_ELASTIC (also known as *MAT_001) This is an isotropic hypoelastic material	
Variable	**Description (Card 1)**
mid	Material ID
ro	Mass density
e	Young's modulus
pr	Poisson's ratio
da	Axial damping factor
db	Bending damping factor
k (not used)	Bulk modulus

*MAT_POWER_LAW_PLASTICITY (also known as *MAT_018) This is an isotropic plasticity model with rate effects which uses a power law hardening rule	
Variable	**Description (Card 1)**
mid	Material ID
ro	**Mass density**
e	**Young's modulus**
pr	**Poisson's ratio**
k	**Strength coefficient**
n	**Hardening exponent**
src	Strain rate parameter, C. If zero, rate effects are ignored
srp	Strain rate parameter, P. If zero, rate effects are ignored

Variable	**Description (Card 2)**
sigy	Optional input parameter for defining the initial yield stress
vp	Formulation for rate effects
epsf	**Plastic failure strain for element deletion**

Material Properties | Plate (cont.)

Therefore, we changed perfectly elastic material properties (*MAT_ELASTIC_TITLE) to elastoplastic behavior with isotropic hardening (*MAT_POWER_LAW_PLASTICITY).
Further, you'll see that this Material ID is assigned to the Plate.

You can read more about *MAT_POWER_LAW_PLASTICITY in LS-DYNA Manual Volume II.

```
*MAT_POWER_LAW_PLASTICITY
$#     MID       RHO         E        PR         K         N       src       srp
$              Kg/mm^3       GPa                GPa
        1 2.7126E-06 6.8948E+01       0.33 1.2463E-01  0.070747       0.0       0.0
$#   sigy        vp      epfs
       0.0       0.0    0.0972
```

Remarks:

Elastoplastic behavior with isotropic hardening is provided by this model. The yield stress, σ_y, is a function of plastic strain and obeys the equation:

$$\sigma_y = k\varepsilon^n = k(\varepsilon_{yp} + \bar{\varepsilon}^p)^n$$

where ε_{yp} is the elastic strain to yield and $\bar{\varepsilon}^p$ is the effective plastic strain (logarithmic). If

Material: Aluminum Pure-H12 Sheet
density RO = 2.71e-6 kg/mm3
Elastic Modulus E = 68.948 GPa
Poisson's ratio PR = 0.33
Plastic failure strain for element deletion EPFS = 0.0972

Material Properties | Ball

1. Search (CTRL+F) in Notepad++ to find *MAT_RIGID_TITLE keyword. No changes needed here.

*MAT_RIGID_TITLE: This is *MAT_020. Parts made from this material are considered to belong to a rigid body.

```
*MAT_RIGID_TITLE
Rigid
$#      mid          ro           e          pr           n     couple           m       alias
         2 7.83000E-6       207.0          0.3         0.0        0.0         0.0
$#      cmo         con1         con2
        0.0            0            0
$#lco or a1           a2           a3           v1           v2           v3
        0.0          0.0          0.0          0.0          0.0          0.0
```

Material: Steel

density RO = 7.83e-6 kg/mm3

Elastic Modulus E = 207 GPa

Poisson's ratio PR = 0.3

Variable	Description (Card 1 | this card is required)
mid	Material ID
ro	**Mass density**
e	**Young's modulus. Reasonable values must be chosen for contact analysis**
pr	**Poisson's ratio. Reasonable values must be chosen for contact analysis**
n	MADYMO3D 5.4 coupling flag
couple	coupling option if applicable
m	MADYMO3D 5.4 coupling flag
alias	VDA surface alias name

Variable	Description (Card 2b | included if CMO = 0.0)
cmo	Center of mass constraint option (cmo = 0: no constraints)

Variable	Description (Card 3 | must be included but may be left blank)
lco	Local coordinate system for local output to rbdout
a1 – v3	Alternative method for specifying local system

Further, you'll see that this Material ID is assigned to the Ball

Steps

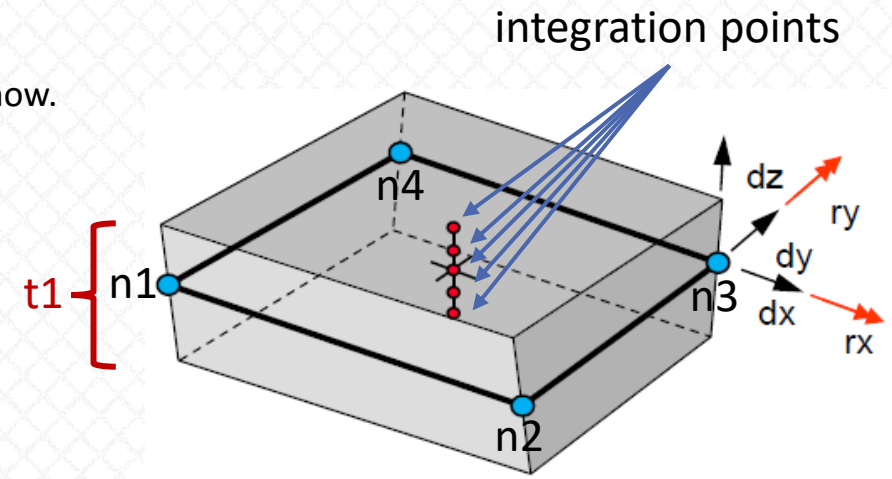
Step #	Desciption	
1	Create Geometry and Mesh	
2	Boundary Conditions	
3	Material Properties	
4	Section/Element Properties	LS-DYNA keyword file from Tutorial #1 already contains SHELL section property for the Plate, and SOLID section property for the Ball
5	Assign Material and Section Properties to Parts	
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 | Plate

1. Search (CTRL+F) in Notepad++ to find *SECTION_SHELL_TITLE keyword. No changes needed here for now.

*SECTION_SHELL_TITLE : Define section properties for shell elements.

```
*SECTION_SHELL_TITLE
Section Shell
$#    secid    elform      shrf       nip     propt    qr/irid     icomp     setyp
         1         2        1.0         2       1.0          0         0         1
$#        t1        t2        t3        t4      nloc     marea      idof   edgset
       0.1       0.1       0.1       0.1       0.0       0.0       0.0         0
```



Variable	Description (Card 1 | mandatory)
secid	Section ID
elform	**Element formulation options (elform = 2: Belytschko-Tsay)**
shrf	Shear correction factor which scales the transverse shear stress
nip	**Number of through thickness integration points**
propt	Printout option (propt =1: Average resultants and fiber lengths)
qr / irid	Quadrature rule or Integration rule ID (qr / irid = 0: Gauss/Lobatto)
icomp	Flag for orthotropic / anisotropic layered composite material model
setyp	Not used (obsolete)

Variable	Description (Card 2 | mandatory)
t1	**Shell thickness at node n1**
t2	**Shell thickness at node n2**
t3	**Shell thickness at node n3**
t4	**Shell thickness at node n4**
nloc	Location of reference surface (shell mid-thickness) for 3D shell elements (nloc = 0: Nodes are located at mid-thickness of shell (default))
marea	Non-structural mass per unit area
idof	Treatment of through thickness strain
edgset	Edge node set required for shell type seatbelts

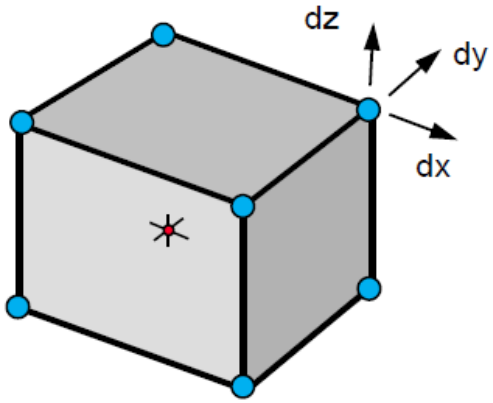
Further, you'll see that this Section ID is assigned to the Plate

Section Properties / Element Formulation | Ball

1. Search (CTRL+F) in Notepad++ to find *SECTION_SOLID_TITLE keyword. No changes needed here.

*SECTION_SOLID_TITLE : Define section properties for solid continuum and fluid elements.

```
*SECTION_SOLID_TITLE
Section Solid
$#    secid     elform       aet
         2          1         0
```



Variable	Description (Card 1 | mandatory)
secid	**Section ID**
elform	**Element formulation options (elform = 1: Constant stress solid element (default))**
aet	Ambient element type (aet = 0: Non-ambient)

Further, you'll see that this Section ID is assigned to the Ball

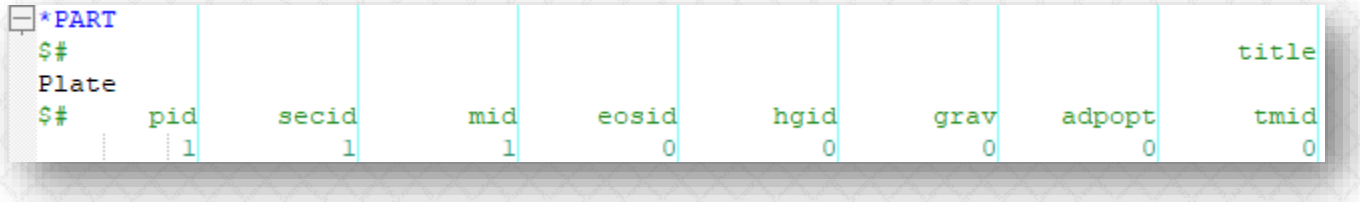
Steps

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	LS-DYNA keyword file from Tutorial #1 already contains Material and Section properties assignments 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 | Plate

1. Search (CTRL+F) in Notepad++ to find *PART keyword. No changes needed here.

*PART : Define parts, that is, combine material information, section properties, hourglass type, thermal properties, and a flag for part adaptivity.



Variable	Description (Card 1 | required)
title	**Heading for the part**

Variable	Description (Card 2 | required)
pid	**Part ID**
secid	**Section ID defined in *SECTION keyword**
mid	**Material ID defined in *MAT keyword**
eosid	Equation of state ID defined in *EOS keyword
hgid	Hourglass / bulk viscosity ID defined in *HOURGLASS keyword
grav	Flag to turn on gravity initialization according to *LOAD_DENSITY_DEPTH
adpopt	Indicate if this part is adapted or not (adpopt = 0: Adaptive remeshing is inactive for this part ID)
tmid	Thermal material property ID defined in *MAT_THERMAL keyword

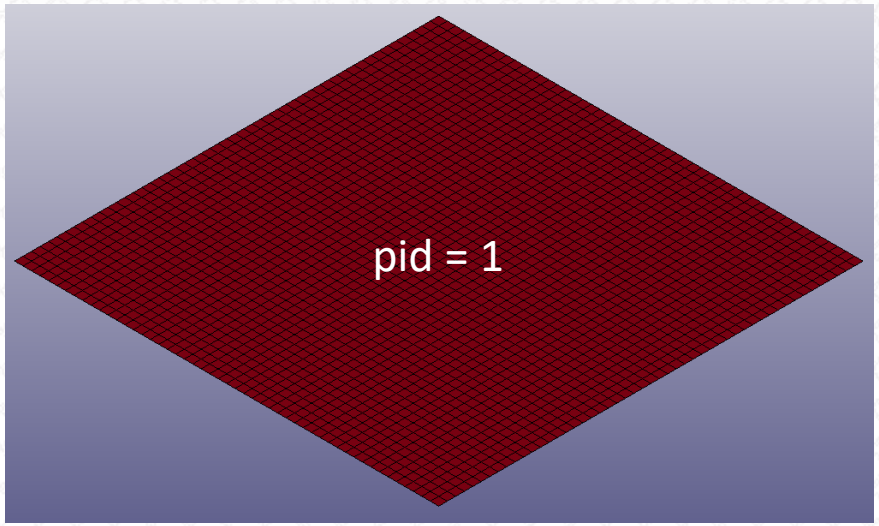
Assign Material and Section Properties to Parts | Plate

This is how *PART, *SECTION, and *MAT keyword relate to each other for the Plate:

```
*PART
$#                                                                         title
Plate
$#     pid      secid        mid      eosid       hgid       grav     adpopt       tmid
         1          1          1          0          0          0          0          0
```

```
*SECTION_SHELL_TITLE
Section Shell
$#    secid    elform      shrf       nip     propt    qr/irid     icomp     setyp
         1          2       1.0         2       1.0          0         0         1
$#       t1         t2        t3        t4      nloc      marea      idof    edgset
        0.1        0.1       0.1       0.1       0.0        0.0       0.0         0
```

```
*MAT_POWER_LAW_PLASTICITY
$#     MID        RHO          E         PR          K          N        src        srp
$                Kg/mm^3      GPa                   GPa
         1 2.7126E-066.8948E+01       0.3301.2463E-01   0.070747        0.0        0.0
$#    sigy         vp       epfs
        0.0        0.0     0.0972
```



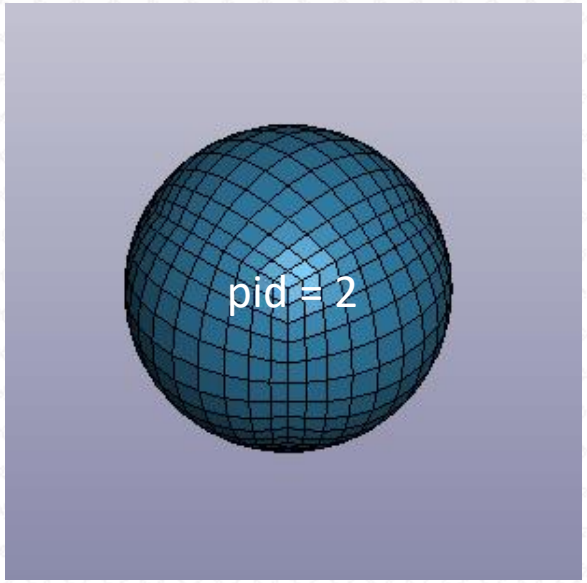
Assign Material and Section Properties to Parts | Ball

This is how *PART, *SECTION, and *MAT keyword relate to each other for the Ball:

```
*PART
$#                                                                         title
Ball
$#     pid      secid        mid      eosid       hgid       grav     adpopt       tmid
         2          2          2          0          0          0          0          0
```

```
*SECTION_SOLID_TITLE
Section Solid
$#    secid    elform        aet
         2          1          0
```

```
*MAT_RIGID_TITLE
Rigid
$#      mid         ro          e         pr          n     couple          m      alias
         2  7.83000E-6      207.0        0.3        0.0        0.0        0.0
$#      cmo       con1       con2
        0.0          0          0
$#lco or a1         a2         a3         v1         v2         v3
        0.0        0.0        0.0        0.0        0.0        0.0
```



Steps

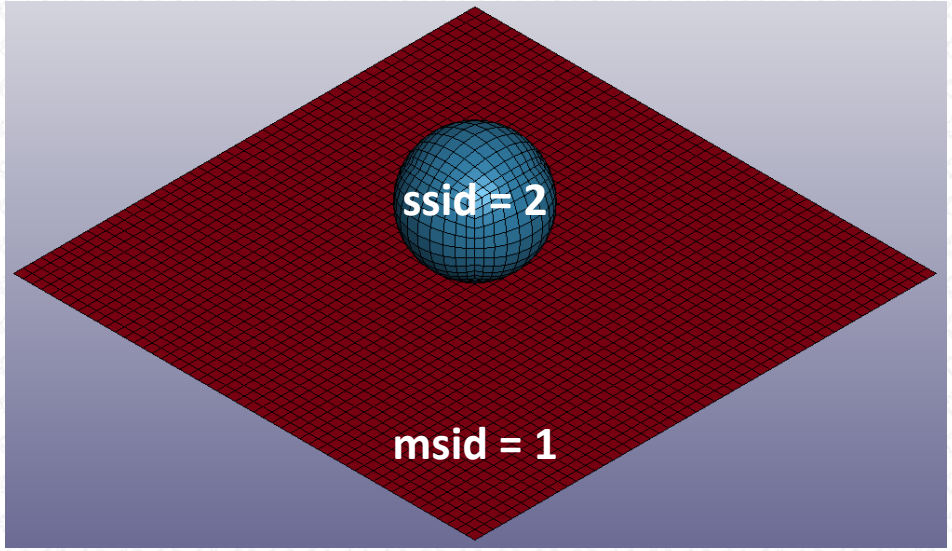
Step #	Desciption	
1	Create Geometry and Mesh	
2	Boundary Conditions	
3	Material Properties	
4	Section/Element Properties	
5	Assign Material and Section Properties to Parts	
6	Contact	LS-DYNA keyword file from Tutorial #1 already contains keywords defining contact between the Ball and Plate
7	Initial Velocity	
8	Analysis Time and Output Controls	
9	Submit Analysis in LS-Run	
10	Postprocess results in LS-Prepost	

Contact

1. Search (CTRL+F) in Notepad++ to find *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID keyword

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID: Define a contact interface in a 3D model.

```
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID
$#     cid                                                         title
         0Contact Ball to Plate
$#    ssid       msid      sstyp      mstyp     sboxid     mboxid       spr       mpr
         2          1          3          3          0          0          0         0
$#      fs         fd         dc         vc        vdc     penchk         bt        dt
       0.0        0.0        0.0        0.0        0.0          0 0.01.00000E20
$#     sfs        sfm        sst        mst       sfst       sfmt        fsf       vsf
       1.0        1.0        0.0        0.0        1.0        1.0        1.0       1.0
```



Variable	Description (Additional Card for ID keyword option)
cid	Contact interface ID
title	**Interface descriptor**

Variable	Description (Card 1 | always required)
ssid	**Slave segment set ID, node set ID, part set ID, part ID, or shell element set ID**
msid	**Master segment set ID, node set ID, part set ID, part ID, or shell element set ID**
sstyp	**ID type of SSID (sstyp = 3: part ID)**
mstyp	**ID type of MSID (mstyp = 3: part ID)**
sboxid	Include in contact definition only those slave nodes/segments withing SBOXID
mboxid	Include in contact definition only those master segments within box MBOXID
spr	Include the slave side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files, and optionally in the dynain file for wear (spr = 0: do not include)
mpr	Include the master side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files, and optionally in the dynain file for wear (spr = 0: do not include)

Variable	Description (Card 2 | always required)
fs	Static coefficient of friction
fd	Dynamic coefficient of friction
dc	Exponential decay coefficient
vc	Coefficient for viscous friction
vdc	Viscous damping coefficient in percent of critical or the coefficient of restitution expressed as percentage
penchk	Small penetration in contact search option
bt	Birth time (contact surface becomes active at this time) (bt = 0: Birth time is inactive, meaning contact is always active)
dt	Death time (contact surface is deactivated at this time) (dt = 0: dt defaults to 1E20)

Contact (cont.)

```
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID
$#     cid                                                         title
         0Contact Ball to Plate
$#    ssid       msid      sstyp      mstyp     sboxid     mboxid       spr       mpr
         2          1          3          3          0          0         0         0
$#      fs         fd         dc         vc        vdc     penchk        bt        dt
       0.0        0.0        0.0        0.0        0.0          0      0.01.00000E20
$#     sfs        sfm        sst        mst       sfst       sfmt       fsf       vsf
       1.0        1.0        0.0        0.0        1.0        1.0       1.0       1.0
```

Variable	Description (Card 3 | always required)
sfs	Scale factor on default slave penalty stiffness when SOFT = 0 or SOFT = 2
sfm	Scale factor on default master penalty stiffness when SOFT = 0 or SOFT = 2
sst	Optional contact thickness for slave surface (overrides default contact thickness). This option applies to contact with shell and beam elements. SST has no bearing on the actual thickness of the elements; it only affects the location of the contact surface
mst	Optional contact thickness for master surface (overrides default contact thickness). This option applies only to contact with shell elements.
sfst	Scale factor applied to contact thickness of slave surface. This option applies to contact with shell and beam elements. SFST has no bearing on the actual thickness of the elements; it only affects the location of the contact surface
sfmt	Scale factor applied to contact thickness of master surface. This option applies only to contact with shell elements. SFMT has no bearing on the actual thickness of the elements; it only affects the location of the contact surface
fsf	Coulomb friction scale factor
vsf	Viscous friction scale factor

Steps

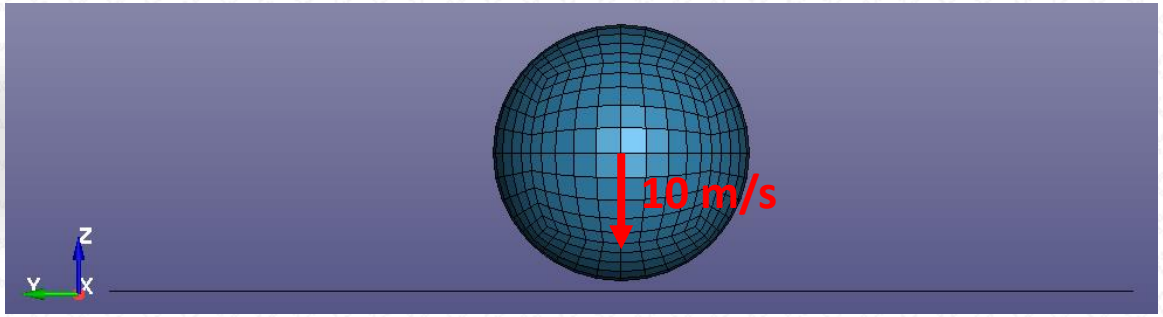
Step #	Desciption	
1	Create Geometry and Mesh	
2	Boundary Conditions	
3	Material Properties	
4	Section/Element Properties	
5	Assign Material and Section Properties to Parts	
6	Contact	
7	Initial Velocity	LS-DYNA keyword file from Tutorial #1 already contains keywords defining initial velocity of the Ball
8	Analysis Time and Output Controls	
9	Submit Analysis in LS-Run	
10	Postprocess results in LS-Prepost	

Initial Velocity

1. Search (CTRL+F) in Notepad++ to find *INITIAL_VELOCITY_GENERATION keyword. No changes needed here.

*INITIAL_VELOCITY_GENERATION : Define initial velocities for rotating and translating bodies.

```
*INITIAL_VELOCITY_GENERATION
$#nsid/pid      styp     omega        vx        vy        vz     ivatn      icid
         2         2       0.0       0.0       0.0    -10.0         0         0
$#        xc        yc        zc        nx        ny        nz     phase    irigid
       0.0       0.0       0.0       0.0       0.0       0.0         0         0
```



Variable	Description (Card 1)
nsid/pid	**Part ID, part set ID, or node set ID**
styp	**Set type (styp = 2: Part ID)**
omega	Angular velocity about the rotational axis
vx	**Initial translational velocity in x-direction**
vy	**Initial translational velocity in y-direction**
vz	**Initial translational velocity in z-direction**
ivatn	Flag for setting the initial velocities of slave nodes and parts (ivatn = 0: Slave parts are ignored)
icid	Local coordinate system ID (icid = 0: the specified translational velocities (VX,VY,VZ) are in the global system)

Variable	Description (Card 2)
xc	Global x-coordinate on rotational axis
yc	Global y-coordinate on rotational axis
zc	Global z-coordinate on rotational axis
nx	x-direction cosine
ny	y-direction cosine
nz	z-direction cosine
phase	Flag determining basis for initialization of velocity
irigid	Controls hierarchy of initial velocities set with *INITIAL_VELOCITY_GENERATION versus those set with *PART_INERTIA / *CONSTRAINED_NODAL_RIGID_BODY_INERTIA when the commands conflict.

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	LS-DYNA keyword file from Tutorial #1 already contains total duration of simulation and output controls
9	Submit Analysis in LS-Run	
10	Postprocess results in LS-Prepost	

Analysis Time

1. Search (CTRL+F) in Notepad++ to find *CONTROL_TERMINATION keyword. No changes needed here.

*CONTROL_TERMINATION : Stop the job.

```
*CONTROL_TERMINATION
$#    endtim    endcyc     dtmin     endeng     endmas      nosol
      10.0         0       0.0        0.0 1.000000E8          0
```

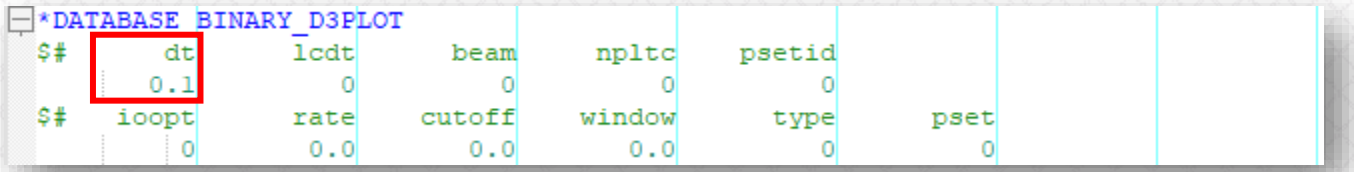
Variable	Description (Card 1)
endtim	**Termination time. Mandatory**
endcyc	Termination cycle. The termination cycle is optional and will be used if the specified cycle is reached before the termination time
dtmin	Reduction (or scale) factor to determine minimum time step, tsmin
ending	Percent change in energy ratio for termination of calculation. If undefined, this option is inactive
endmas	Percent change in the total mass for termination of calculation. This option is relevant if and only if mass scaling is used to limit the minimum time step size; see *CONTROL_TIMESTEP field DT2MS.
nosol	Flag for a non-solution run, that is, normal termination directly after initialization (nosol = 0: off (default))

Note:
ENDTIM = 10 means that we are modeling 10 ms of simulation where Ball impacts a Plate

Output Controls

1. Search (CTRL+F) in Notepad++ to find *DATABASE_BINARY_D3PLOT keyword. No changes needed here.

*DATABASE_BINARY_D3PLOT : Request binary output. Database for entire model.



Variable	Description (Card 1)
dt	**This field defines the time interval between output states, DT**
lcdt	Optional load curve ID specifying time interval between dumps
beam	Discrete element option flag (beam = 0: Discrete spring and damper elements are added to the d3plot database where they are displayed as beam elements)
npltc	DT = ENDTIM/NPLTC This overrides the DT specified in the first field
psetid	Part set ID for D3PART and D3PLOT options only

Variable	Description (Card 2 | additional card for D3PLOT option)
ioopt	This variable applies to the D3PLOT option only and governs how the plot state frequency is determined from curve LCDT
rate	Time interval T between filter sampling
cutoff	Frequency cut-off C in Hz
window	The width of the window W in units of time for storing the single, forward filtering required for the TYPE = 2 filter option
type	Flag for filtering options (type = 0: No filtering (default))
pset	Part set ID for filtering. If no set is specified, all parts are included.

Title

1. Search (CTRL+F) in Notepad++ to find *TITLE keyword. No changes needed here.

*TITLE : Define the job title.

```
*TITLE
$#                                                                  title
Ball Impact on Plate
```

Variable	Description (Card 1)
title	Heading to appear on output and in output files

Save

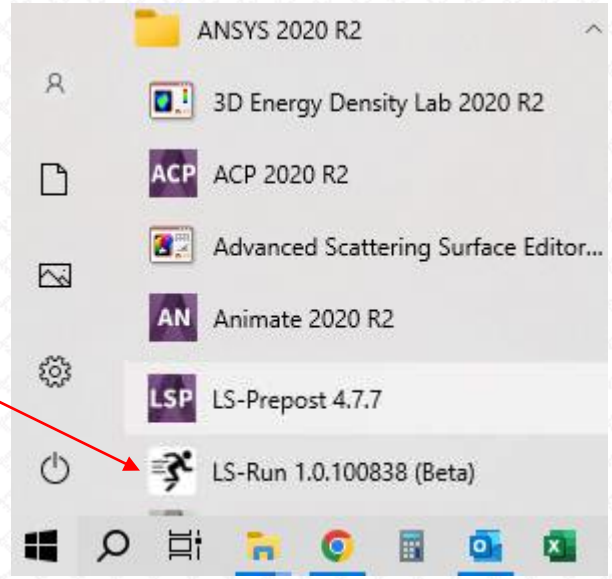
1. [Ctrl + S] to save .k file

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	

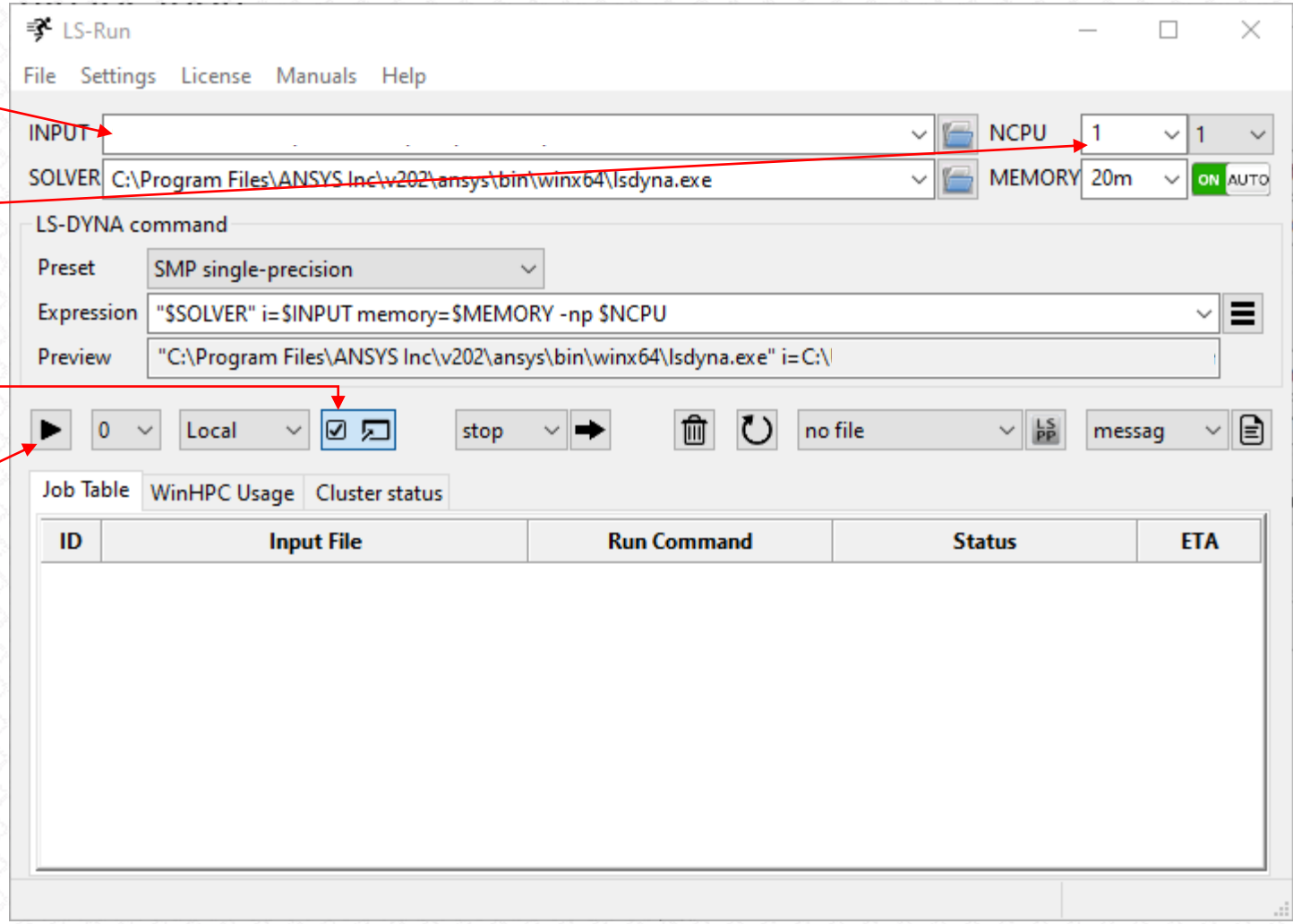
Submit Analysis in LS-Run

1. Start > ANSYS 2020 R2 (or newer) > LS-Run



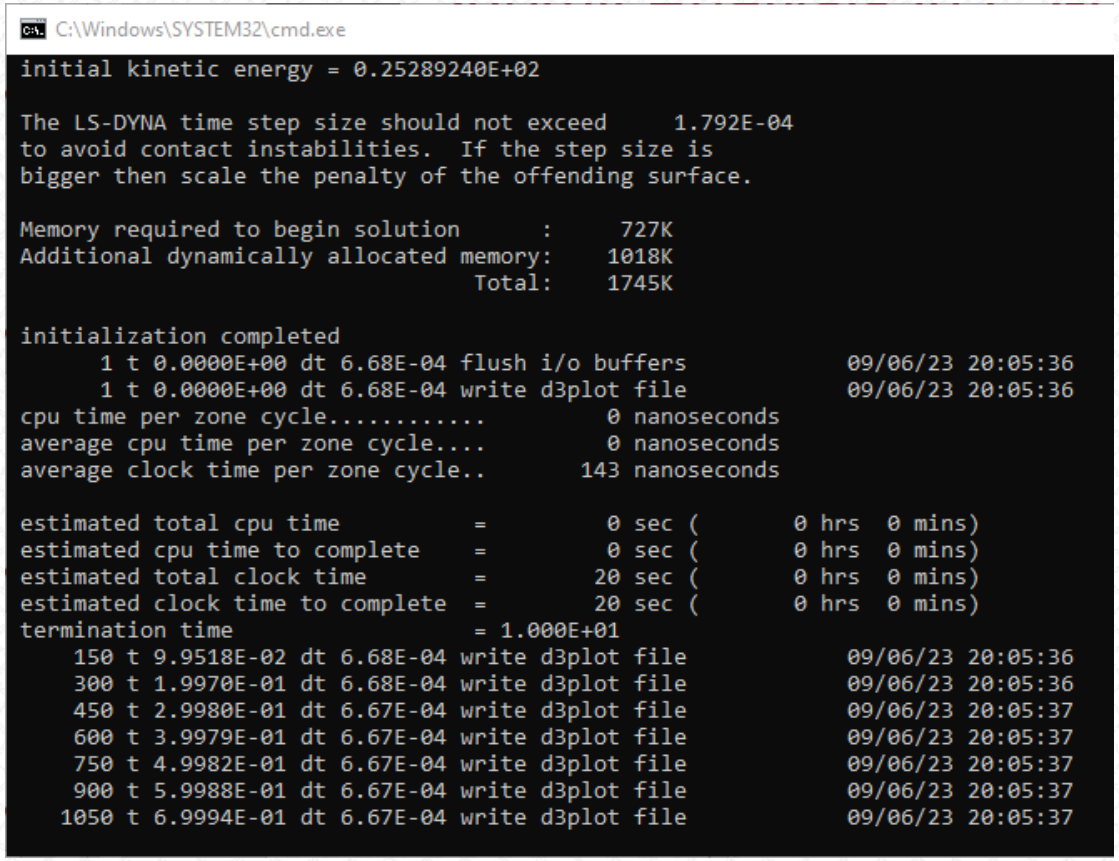
Submit Analysis in LS-Run

1. INPUT: browse to T01_PADT_ImpactBallOnPlate_mod01.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"

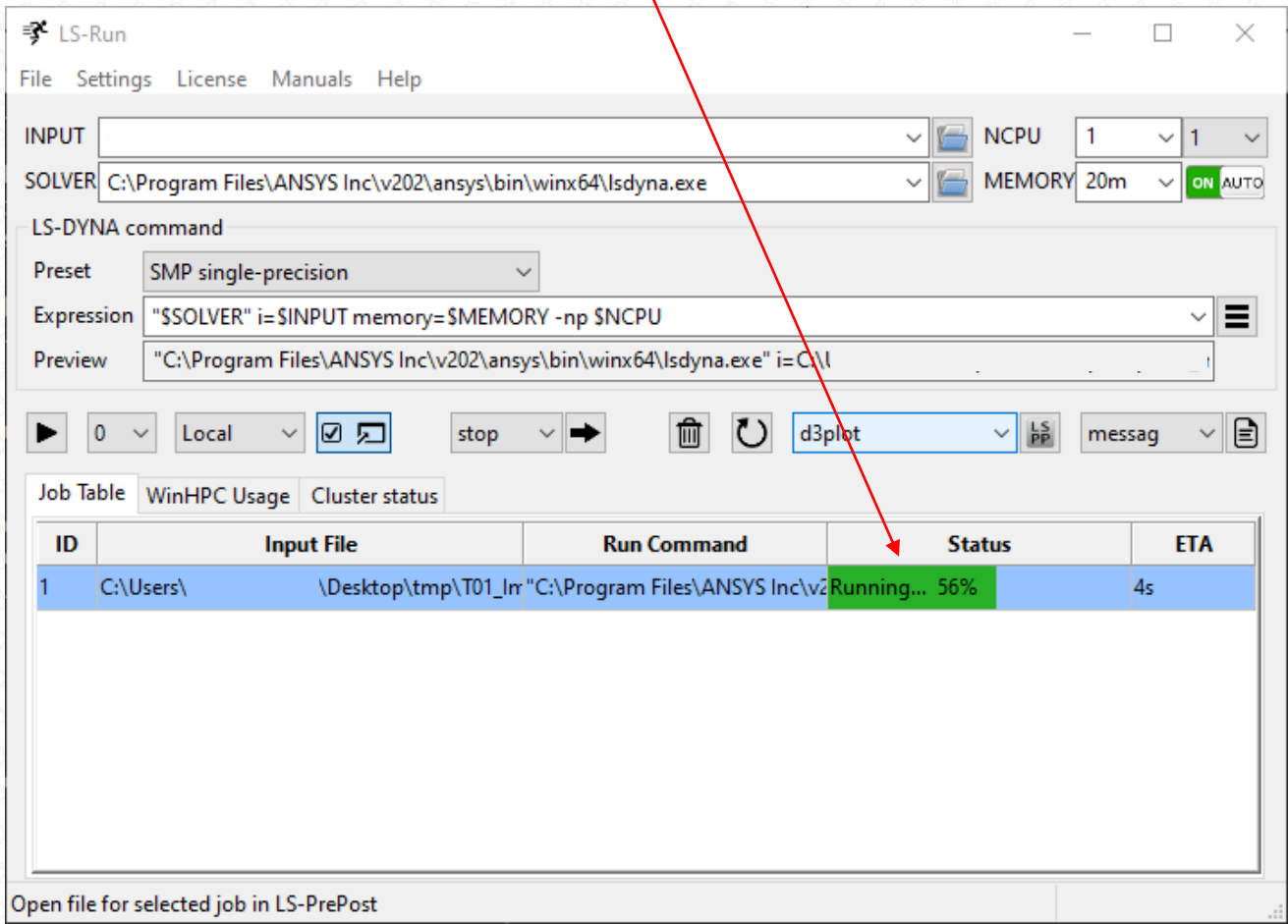


Submit Analysis in LS-Run

1. Command prompt window will display miscellaneous information



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

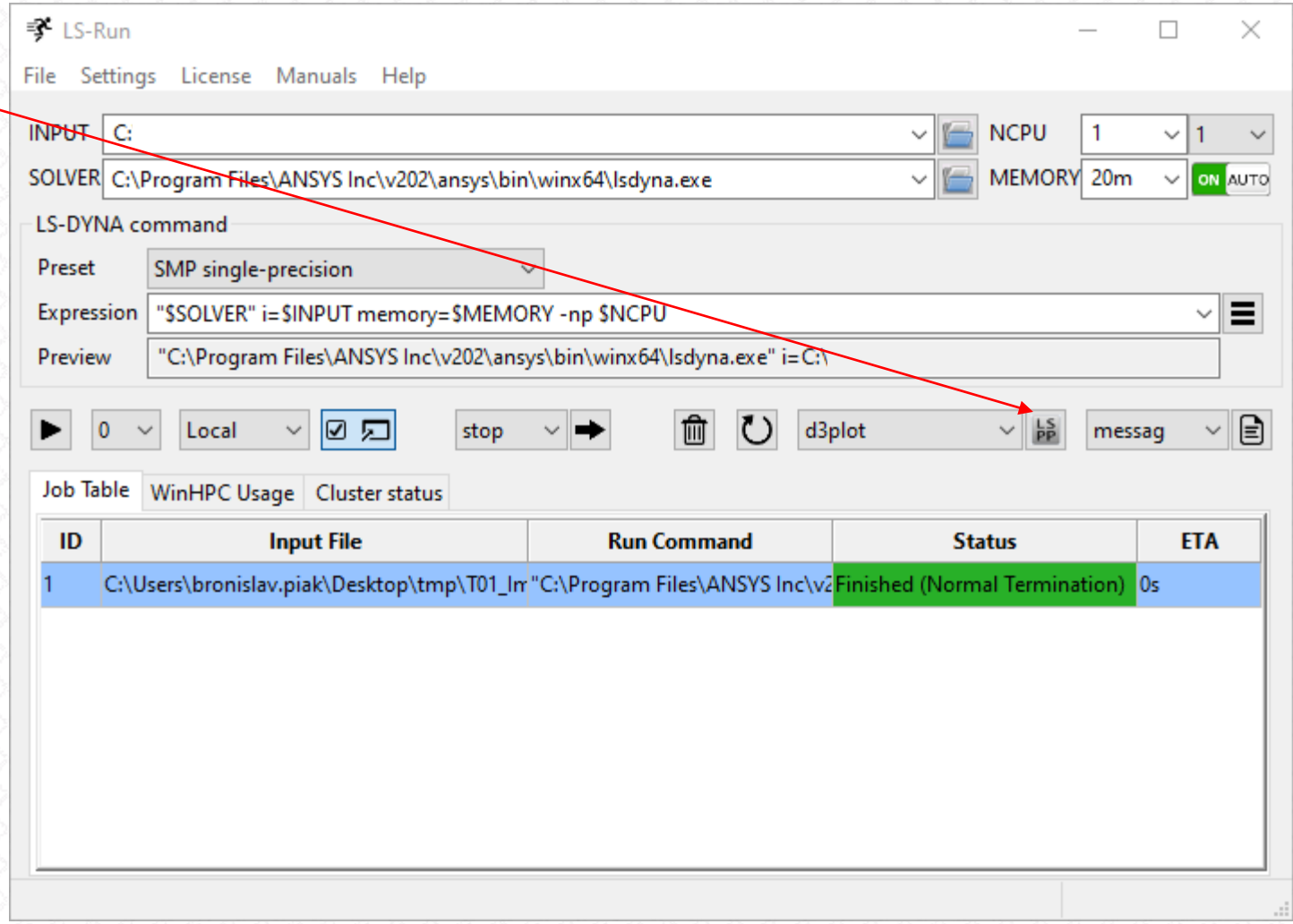


Steps

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

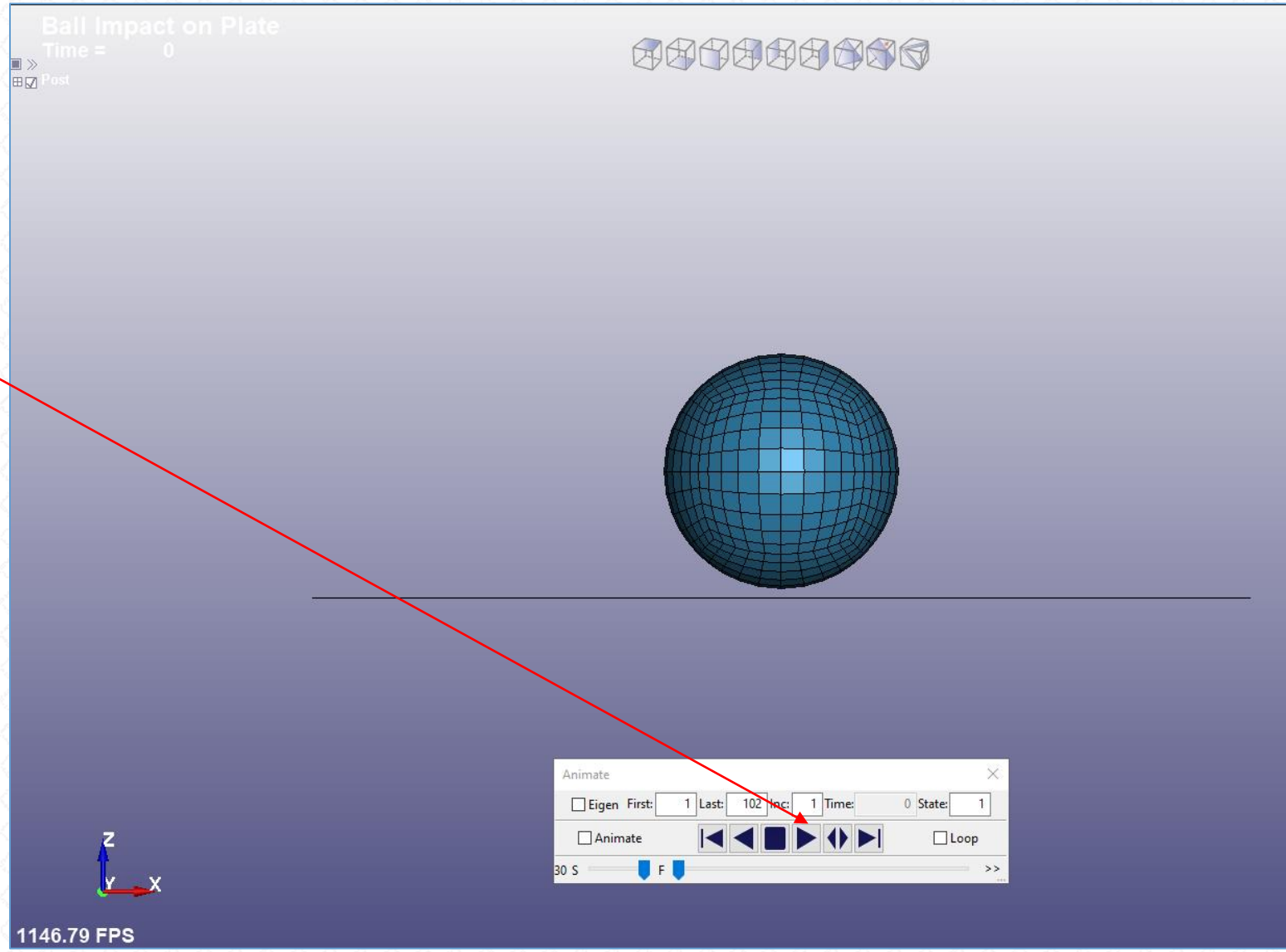
Postprocess Results in LS-Prepost

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

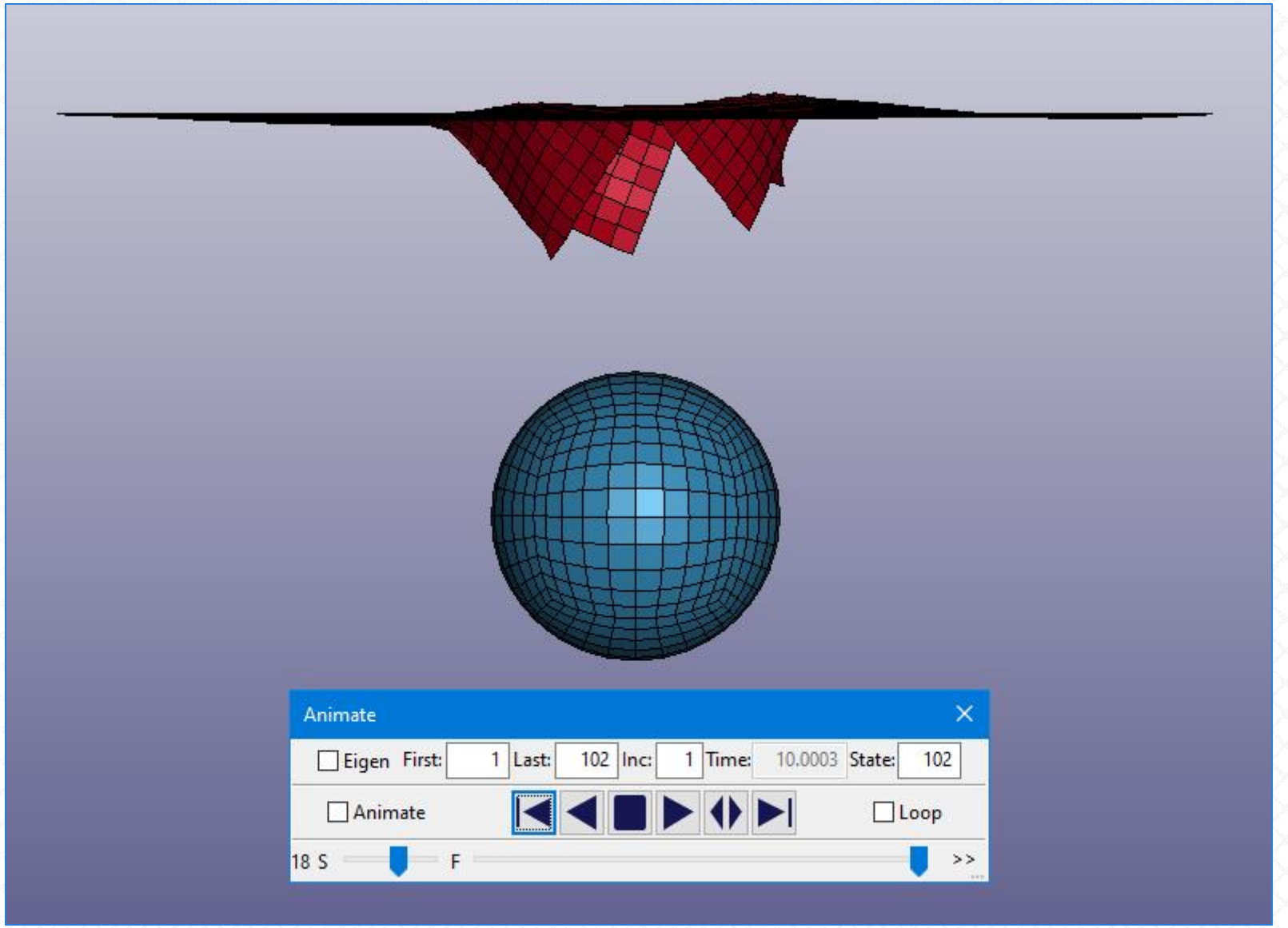


Postprocess Results in LS-Prepost

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

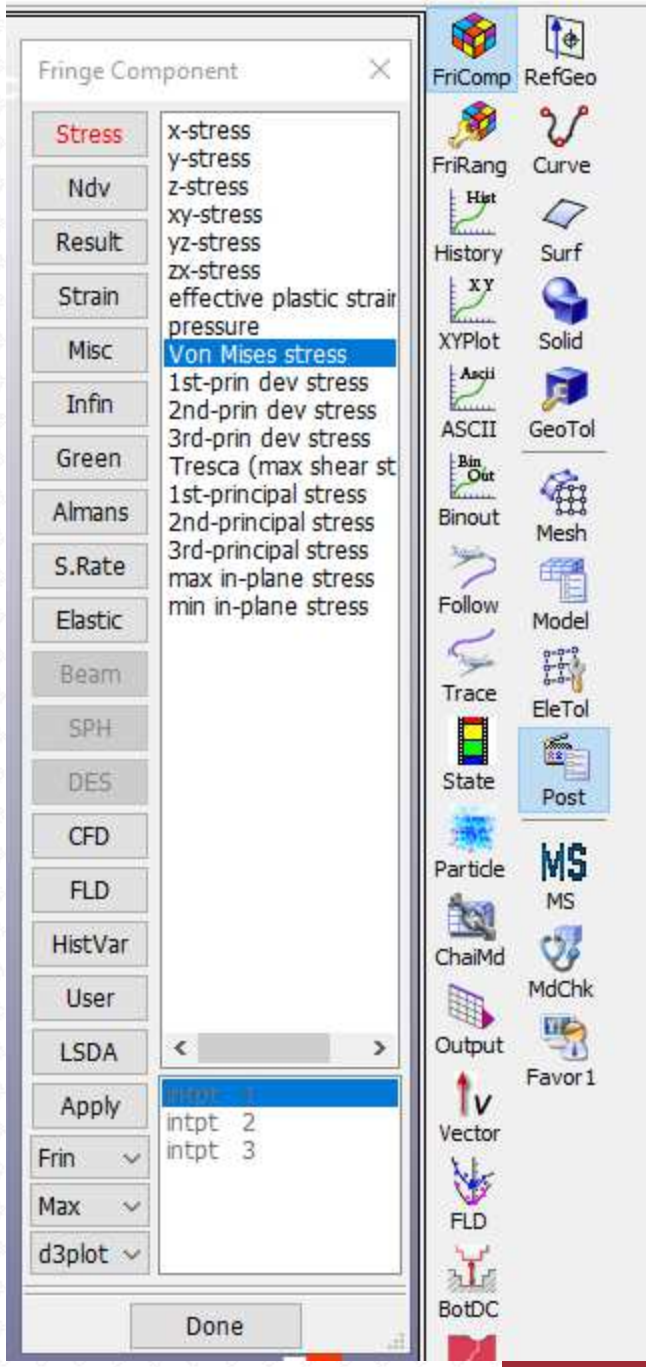


Postprocess Results in LS-Prepost

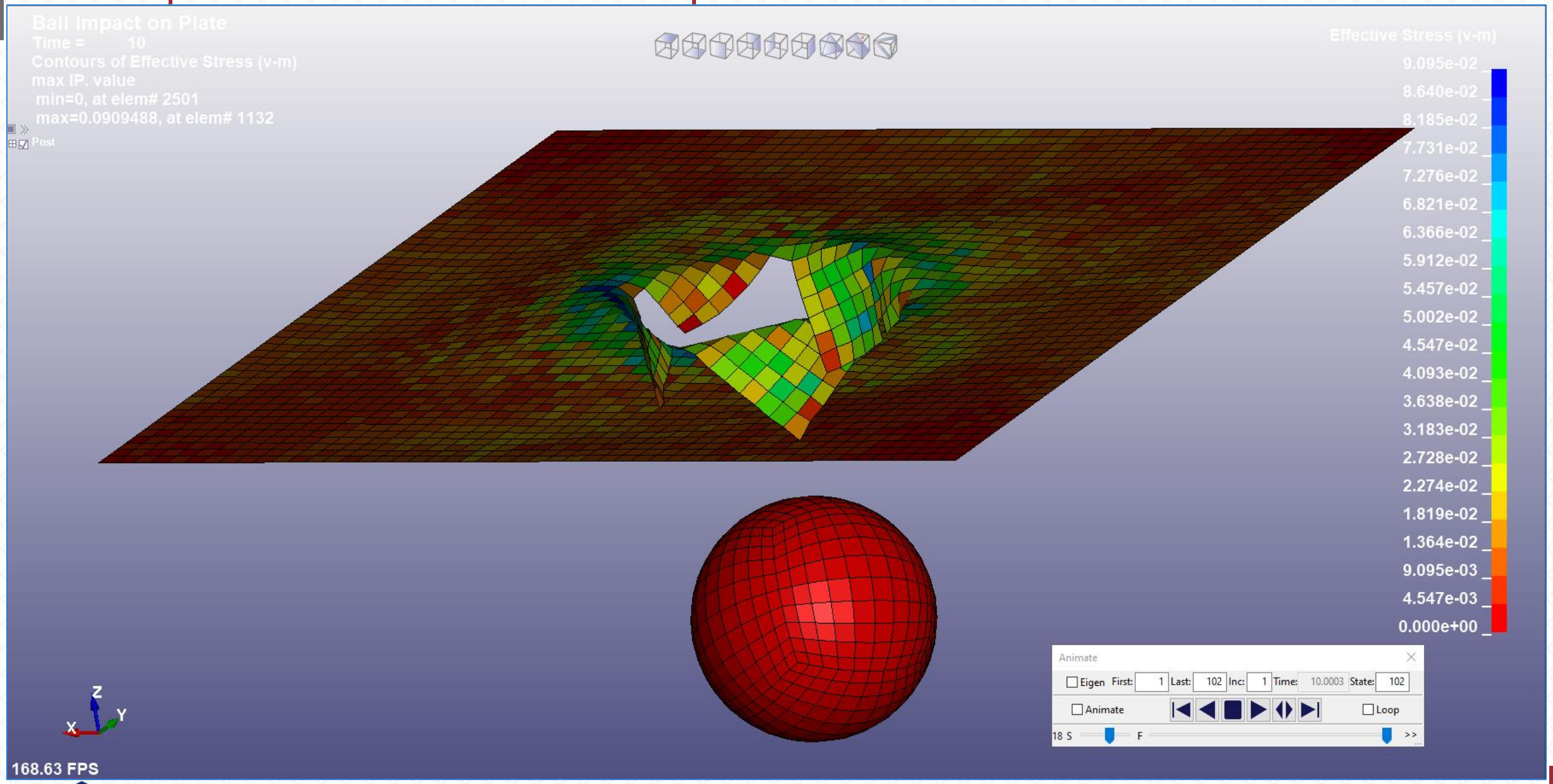


Postprocess Results in LS-Prepost

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



Postprocess Results in LS-Prepost



Steps

Step #	Desciption	
11	Modify Plate thickness	We will use *PARAMETER keyword to modify Plate thickness in *SECTION_SHELL keyword

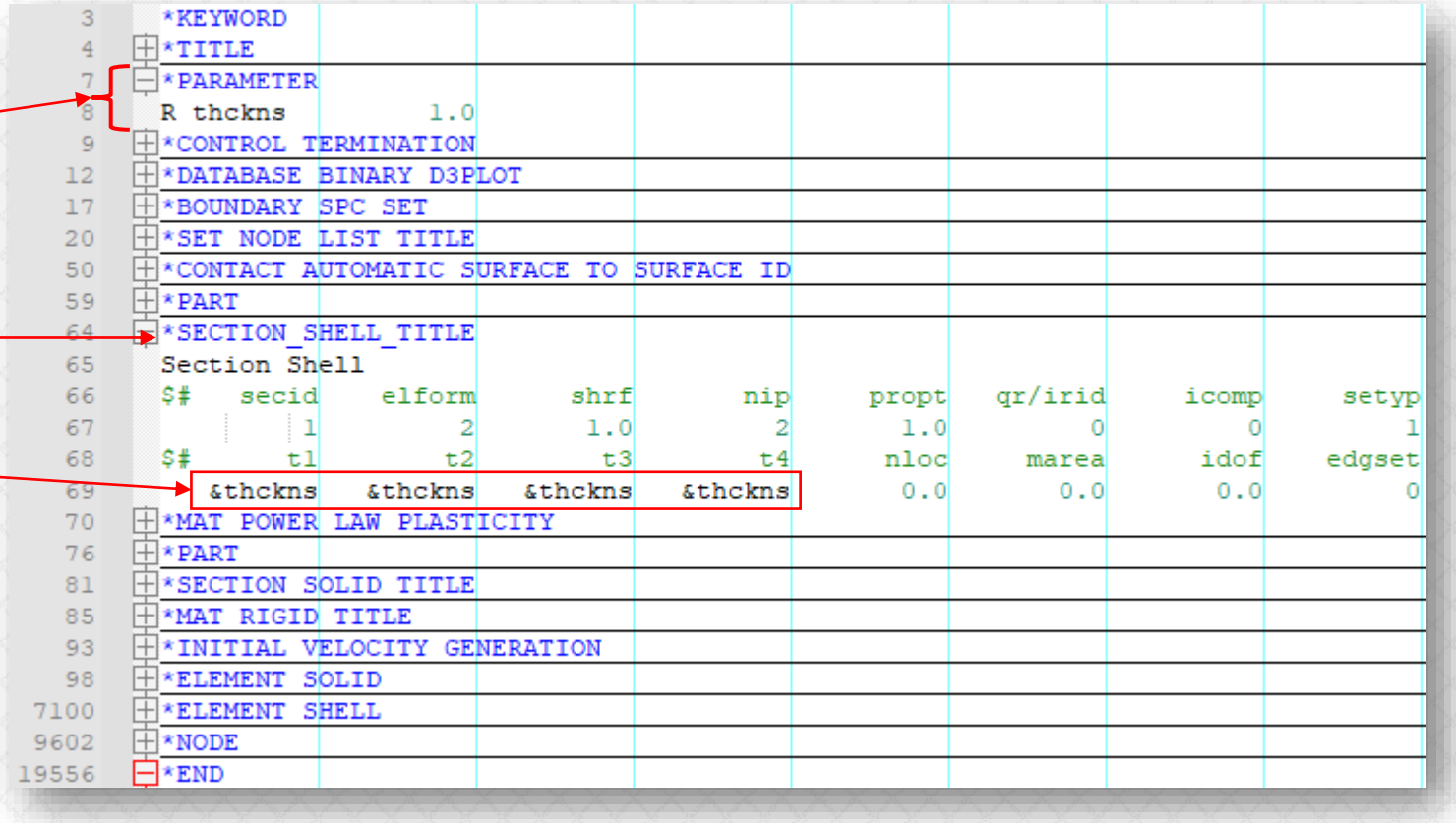
*PARAMETER

1. Copy T01_PADT_ImpactBallOnPlate_mod01.k to a new folder
2. Rename to T01_PADT_ImpactBallOnPlate_mod02.k
3. Open T01_PADT_ImpactBallOnPlate_mod02.k
4. Insert *PARAMETER keyword as shown
   Here we created parameter *thckns = 1.0*
   R is to specify that *thckns* is a real number

5. Search (Ctrl+F) in Notepad++ to find *SECTION_SHELL_TITLE keyword

6. Replace numerical values for t1, t2, t3 and t4 with $thckns

7. Save (Ctrl+S) .k file



Note:
*PARAMETER allowed us to parameterize Plate thickness.
We changed it from 0.1 mm to 1.0 mm

*PARAMETER

Purpose:
Define the numerical values of parameter names referenced throughout the input file. The parameter definitions, if used, should be placed at the beginning of the input file following *KEYWORD or at the beginning of an include file if the LOCAL option is specified.
Read more about *PARAMETER in LS-DYNA Keyword User's Manual Volume I.

Parameter Cards. Include as many cards as necessary.

Card 1	1	2	3	4	5	6	7	8
Variable	PRMR1	VAL1	PRMR2	VAL2	PRMR3	VAL3	PRMR4	VAL4

VARIABLE	DESCRIPTION
PRMR*n*	PRMR*n* sets both the n^{th} parameter and its storage type. PRMR = T xxxxxxxxx 9 character name The first character, "T", is decoded as follows: T.EQ."R": Parameter is a real number T.EQ."I": Parameter is an integer T.EQ."C": Parameter is a character The remaining 9 characters specifiy the name of the parameter. A parameter name "time" (case insensitive) is disallowed. For example, to define a shell thickness named, "SHLTHK", the input "RSHLTHK", "R␣␣␣SHLTHK", or "R␣␣SHLTHK␣" are all equivalent 10 character strings ("␣" is space). For instructions regard how to use the variable "SHLTHK" see Remark 1.

VARIABLE	DESCRIPTION
VAL*n*	Define the value of the n^{th} parameter as either a real or integer number, or a character string consistent with preceding definition for PRMR*n*.

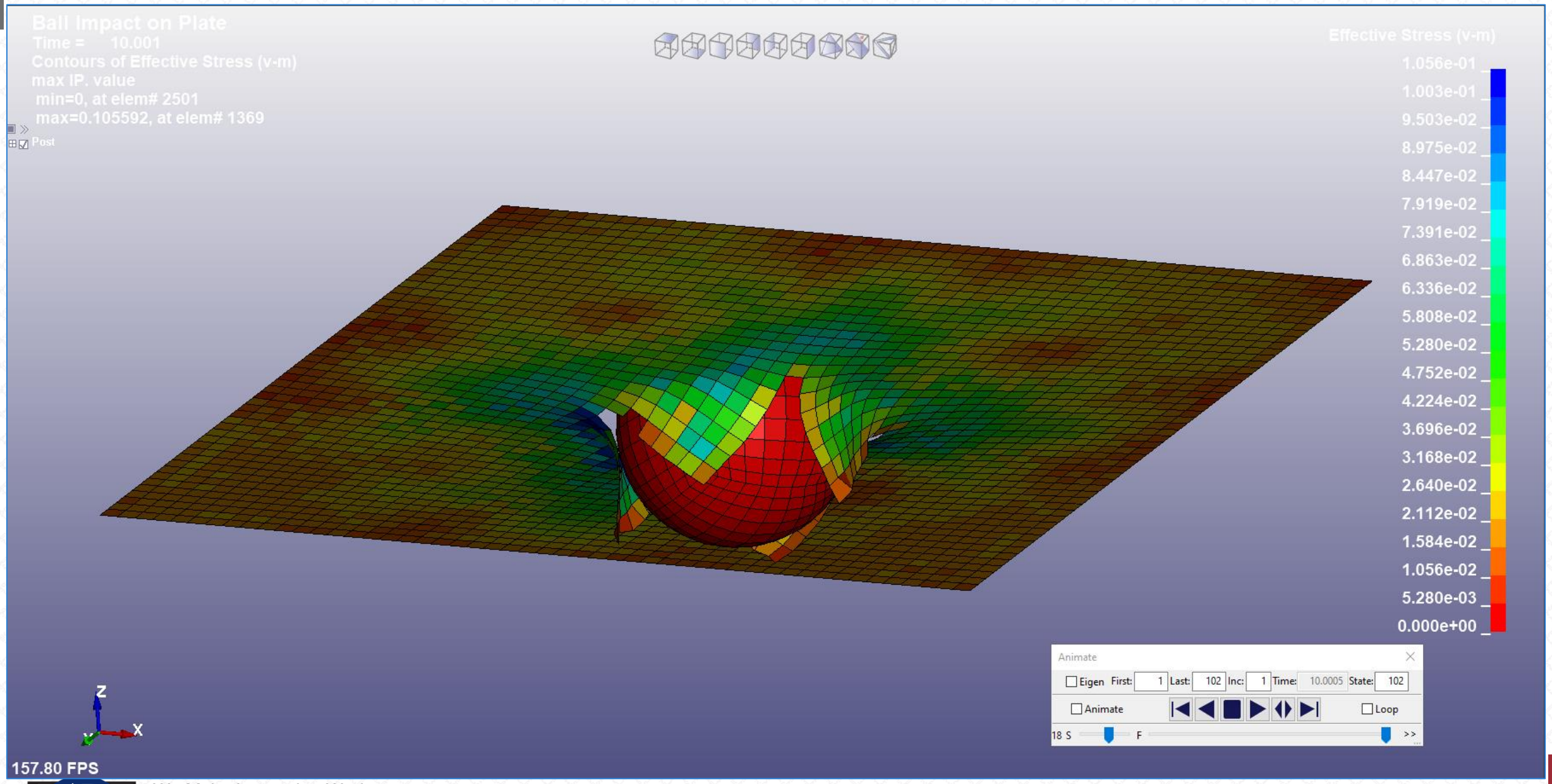
Remarks:

1. **Syntax for Using Parameters.** Parameters can be referenced anywhere in the input by placing an "&" immediately preceding the parameter name. If a minus sign "-" is placed directly before "&", i.e., "-&", with no space the sign of the numerical value will be switched.

Steps

Step #	Desciption	
11	Modify Plate thickness	
12	Submit Analysis in LS-Run	Perform this step similarly to Step 9
13	Postprocess results in LS-Prepost	Perform this step similarly to Step 10

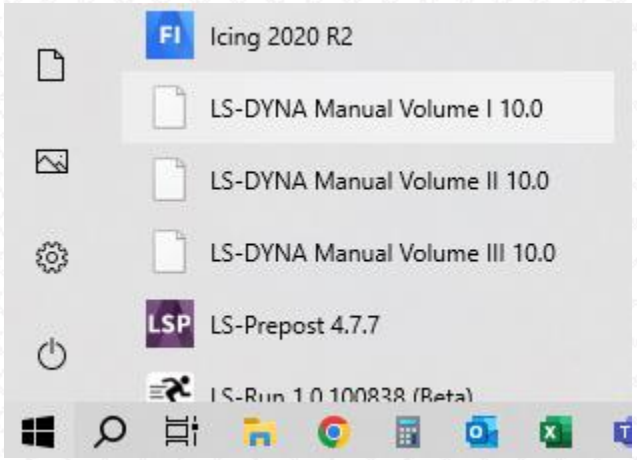
Postprocess Results in LS-Prepost



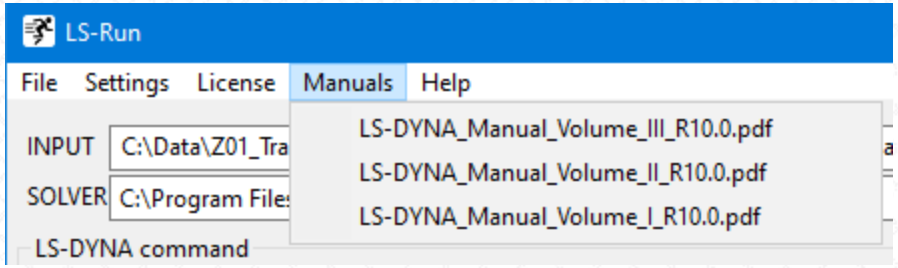
Appendix 1

How to access LS-DYNA Keyword User's Manuals

1. LS-DYNA Keyword User's Manuals come with Ansys LS-DYNA installation.
2. Navigate:
   Start menu -> ANSYS 2020 R2 (or newer) -> Scroll until you see "LS-DYNA Manual Volume I", "LS-DYNA Manual Volume II" and "LS-DYNA Manual Volume III" -> Open

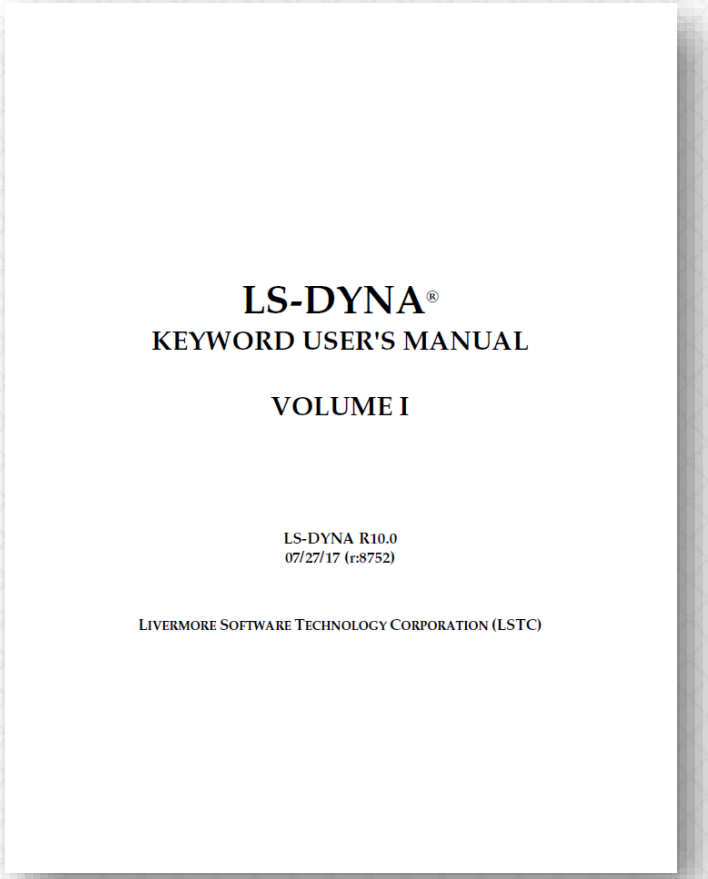


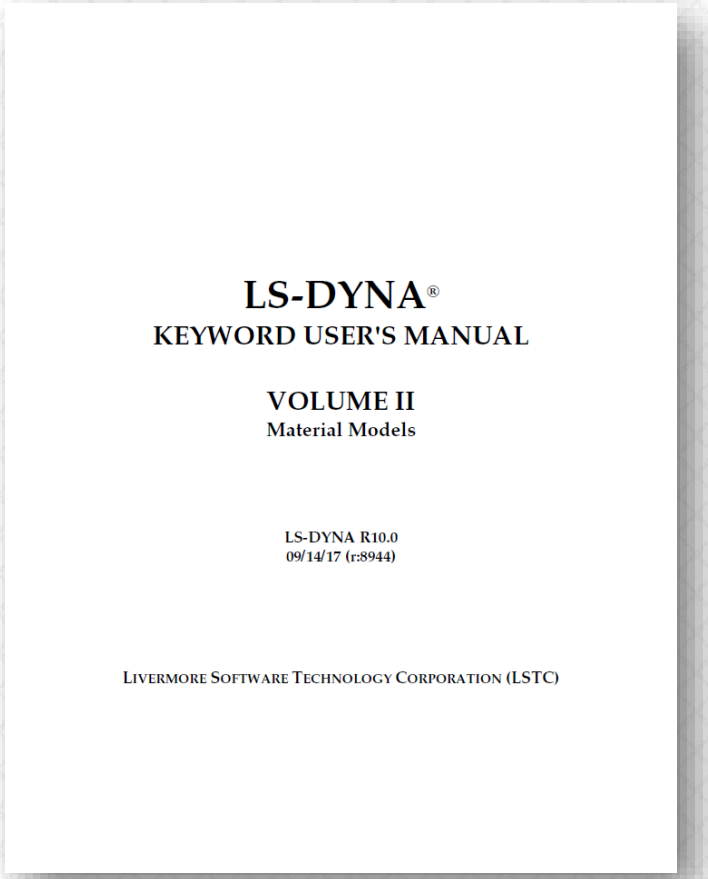
3. Second option:
   a) Start menu -> ANSYS 2020 R2 (or newer) -> LS-Run -> launch
   b) Top menu bar of LS-Run -> Manuals

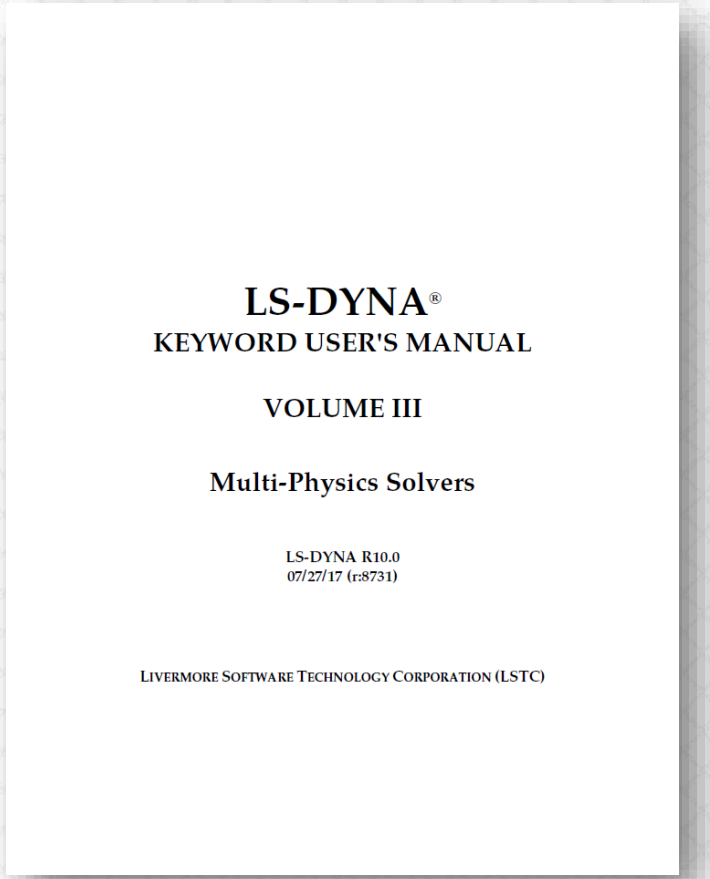


LS-DYNA Keyword User's Manuals

1. LS-DYNA Keyword User's Manuals are a go-to resource describing keywords, materials, EOS (Equations of State), Multi-Physics Solvers, their syntax, use cases, theory and many more.
2. There are 3 Volumes:
   a) LS-DYNA Keyword User's Manual Volume I: describing keywords
   b) LS-DYNA Keyword User's Manual Volume II: describing material models and EOS
   c) LS-DYNA Keyword User's Manual Volume III: describing Multi-Physincs Solvers







Appendix 2

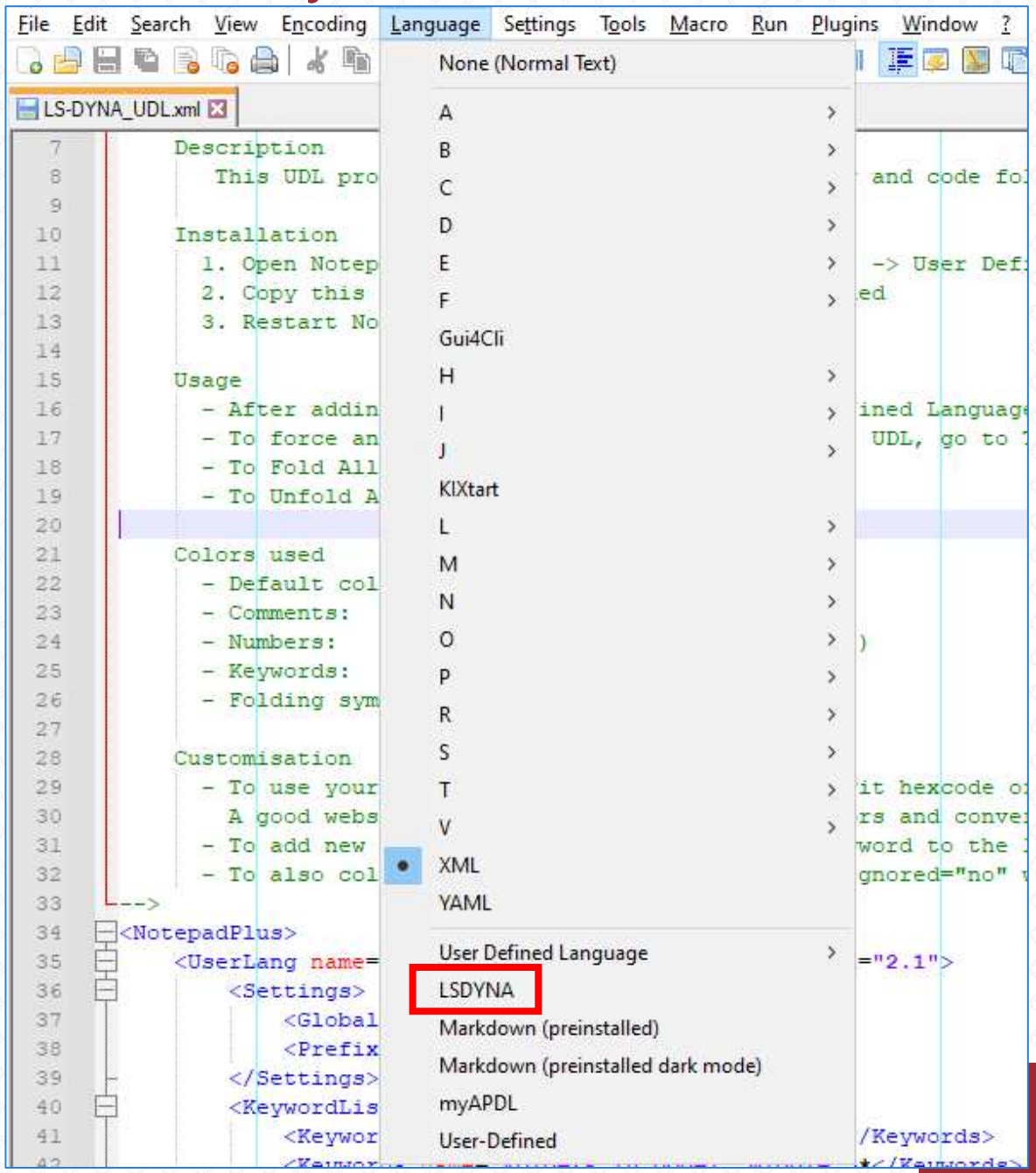
Configure Notepad++ to color-code LS-DYNA keyword file

Installation:

1. Download “LS-DYNA_UDL.xml”
2. Open Notepad++:
3. Go to Top Menu -> Language -> User Defined Language -> Open User Defined Language Folder
4. Copy “LS-DYNA_UDL.xml” into the folder which opened
5. Restart Notepad++

Usage:

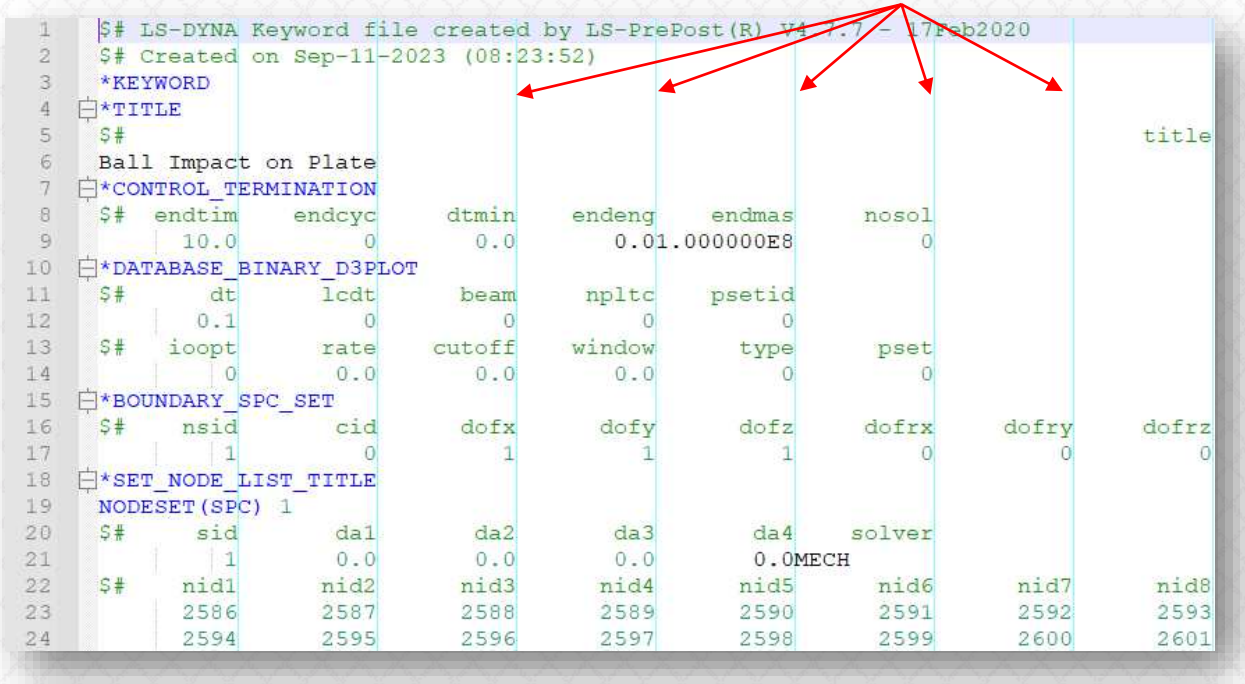
1. After adding this file to Notepad++'s User Defined Languages Folder, any .k, .key, or .dyn files opened in Notepad++ will automatically be formatted using this UDL
2. To force an open file to be formatted use this UDL, go to Top Menu -> Language -> LSDYNA
3. To Fold All lines press ***Alt+0***
4. To Unfold All lines press ***Alt+Shift+0***



Configure Notepad++ to display column markers

Column markers make it easier to read and write keyword's 8 character fields:

column markers



```
$# 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
```

Let's configure Notepad++ to display them.

Configure Notepad++ to display column markers

1. Notepad++ -> Settings -> Preferences -> Margins/Border/Edge
2. Under “Vertical Edge Settings” type:
   10 20 30 40 50 60 70 80