# 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

						Consisten	t set of un	its for Steel	
MASS	LENGTH	TIME	FORCE	STRESS	ENERGY	DENSITY	YOUNG's	35MPH	GRAVITY
								56.33KMPH	
kg	m	s	Ν	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/cm²	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	Ν	MPa	N-mm	7.83E-03	2.07E+05	15.65	9.81E-03
ton	mm	s	Ν	MPa	N-mm	7.83E-09	2.07E+05	1.56E+04	9.81E+03
lbf-s²/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-s²/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 #	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**



www.padtinc.com

#### Create Plate

1. Shell Plate will be created









1. Ball mesh will be shown:



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



1. Change Density to 10

- 2. Create
- 3. Accept
- 4. Done



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"



-	<u>M</u> isc. <u>V</u> iew	<u>G</u> eometry	F <u>E</u> M	Application Settings Help							
	New										
	Open	;	>								
	Import	;	>								
	Recent	;	>	L <sub>m 2</sub> Pa							
	Save	;	>								
	Save As	>	>	Save Keyword As	Ctrl+Shift+S						
	Update	Ctrl+U		Save Active Keyword As	Ctrl+Shift+						
Run LS-DYNA				Save Project As	Ctrl+Shift+						
	During	Chilly D	-	Save Post.db As							
	Mauia	Ctrl+P		Save Reduced D3plot Files As.	Ctrl+Shift+[						
Movie		Ctri+M	_	Save Geom As	Ctrl+Shift+0						
E	Exit	Ctrl+X		Save Keyword and Project As							
	Save and Exit			Save Solution As							
XÔ	<u>XXXXXX</u>	0000	<u> </u>	<u>X66666666</u> X	<u>)00000</u> 0						
Save	e Keyword				- × 🚫						
		9	Save I	Keyword File	X						
ile I	Name:										
					🛛						
C:	\Users\	Desktop	\tmp	\T01_ImpactBallOnPlate	Browse						
	tput Version:	V971 R10	· ·	LongFmt I10Fmt A	dvanced						
Out											
Out	Save	Renur	mber	Offset Cano	cel						
Out	Save	Renu	mber	Offset Cano	:el .::						
	Save	Renur Desktop > tn	mber np	Offset Cano							

T01\_ImpactBallOnPlate.k

# 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	



- 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.



Sel. Nodes(200)		
Pick Box In	Adjacent	ByNode
Area Prox Out	Attach	ByElem
Poly Circ Add	Clear	ByPart
Saha Dlan ORm	Sava	ByGPart
	Jure	BySubsys
ID Type any	Load	BySet/Grp
Label selection 3DSurf Entire	Deselect	ByEdge
Prop Adap Ang 5	Whole	ByPath
	Active	BySegm

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



**33933333** 

- 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



	Entity Creation ×
Apply	Show O Cre O Mod O Del
Apply	E-Boundary
Done	Spc     Prescribed Motion(BPM)     Set     Node
000010	Spc Symmetry Plane Auto Merge Convert NodeTR
000000	Contact
000000	Camping Sym plane All Fix[11111]
	x y z RX RY RZ
	Death 1.0E+20
	Ber Section NSID 2 NewId
	All None Rev ALSL
	< Apply Cancel Write
	NSet 1 (111000) (sub:1)
	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	

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

Karana Marana	~	SelPart	PefCeo
Reyword Manager	^	SciPart	s P
Keyword Edit Keyword Search		Keywrd	Curve
Edit: MAT_ELASTIC	<ul> <li>✓ Edit</li> </ul>	-	
◯ Mode► () All	RefBy	CreEnt	Surf
Name	Count	Ø.	
I ⊡ MAT	^	PartD	Solid
000-ADD COHESIVE			
000-ADD DAMAGE DIEM		Display	GeoTol
000-ADD_DAMAGE_GISSMO		AR	12.
000-ADD_DAMAGE_GISSMO_STOCHASTIC		RefChk	*##
000-ADD_EROSION		Renum	Mesh
000-ADD_FATIGUE		Renum	E
000-ADD_GENERALIZED_DAMAGE			Model
000-ADD_INELASTICITY		Section	6-6-Y
000-ADD_PERMEABILITY			EleTol
000-ADD_PORE_AIR		MSelect	
000-ADD_THERMAL_EXPANSION		MSEIECT	Post
000-NONLOCAL		667	MC
000-ELASTIC_PERI		Subsys	MS
000-ELASTIC_PERI_LAMINATE		ě	c57
001-ELASTIC		Groups	MdChk
001_FLUID-ELASTIC_FLUID			
002-ORTHOTROPIC_ELASTIC		Views	Eavor 1
002_ANIS-ANISOTROPIC_ELASTIC			10/011
002_SUNIL-COMPOSITE_FAILURE_SUNIL		PtColor	
003-PLASTIC_KINEMATIC		5	
004-ELASTIC_PLASTIC_THERMAL		Appear	
005-SOIL_AND_FOAM			
006-VISCOELASTIC		Annotat	
007-BLATZ-KO_RUBBER	~		
<	>	SplitW	
Material arrange		<b>*</b>	
GroupBy Sort List	t	Explod	
All $\checkmark$ Type $\checkmark$ All	× ا	1 X X	
Load From MatDB		Light	
Model Check Keyword Del	ResForm	— — — — — — — — — — — — — — — — — — —	
		Reflect	
ExpandAll Collapse	All	8888	
Done			



1. Click "NewID"———	Keyword Input Form								
2. TITLE: Steel	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)								
288888888888888888888888888888888888888	TITLE								
	Steel								
3. Type values for RO, E, PR as shown –	MID         RO         E         PR         DA         DB         NOT USED           h         7.830e-06         207.00000         0.3000000         0.0         0.0         0.0								
a) Density $RO = 7.83e-6 \text{ kg/mm}^3$									
b) Elastic Modulus $E = 207 GPa$	COMMENT:								
c) Poisson's ratio $PR = 0.3$									
$C_{j} = 0.5$									
- Dono									
5. Done	Total Card: 1 Smallest ID: 1 Largest ID: 1 Total deleted card: 0								
	×								



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

Keyword Manager		×
Keyword Edit Keyword Search		
Edit: MAT_RIGID ~	Edit	
🔿 Model 💿 All	RefBy	
Name	Count	
018-POWER_LAW_PLASTICITY		^
019-STRAIN_RATE_DEPENDENT_PLASTICITY		
020-RIGID		
021-ORTHOTROPIC_THERMAL		
022-COMPOSITE_DAMAGE		



5							$ \land \land \land$	$\sim \sim \sim$					
1.	Click "NewID"	Keyword Ir	put Form										×
2.	TITI F: Rigid	NewID			MatDB	RefBy	Pick	Add	Accept	Delete	Default	Done	2 Rigid
3		🗌 Use *Pa	rameter 🗌 Cor	nment			(Sul	bsys: 1 T01_	PADT_LS-DYN	IA_BallOnP	late.k)	Setting	
					*№	1AT_RIGID_(1	TITLE) (020)	(1)					
		TITLE										^	
		Rigid											
2	Type values for BO E PR as shown		RO 7.830e-06	<u>E</u> 207.00000	0.3000000	<u>N</u> 0.0		<u>M</u> √ 0.0					
<b>.</b>	a) Donsity $PO = 7.820.6 kg/mm^2$	2 <u>CMO</u>	CON1	CON2					IL				
	a) Defisity $RO = 7.85e-6$ kg/fillins	0.0	~ 0	0									
	b) Elastic Modulus $E = 207 \text{ GPa}$	3 LCO OF	<u>A1 A2</u>	<u>A3</u>	<u>V1</u>	<u>V2</u>	<u>V3</u>	_					
	c) Poisson's ratio PR = 0.3		0.0	0.0	0.0	0.0	0.0						
4.	Accept	COMMENT	Г:										
5.	Done											$^{\sim}$ $\checkmark$	
		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 #	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	

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





		Ke	yword Inpu	ut Form										
1.	Click "NewID"	1	VewID	Draw			RefBy	Sort/T1	Add	Accept	Delete	Default	Done	1 Section Shell
			Use *Parar	meter 🗌 Co	nment	*S	ECTION_SH	(Sub ELL_(TITLE) (	sys: 1 T01_F [1]	PADT_LS-DYI	NA_BallOnP	late.k)	Setting	
2.	TITLE: Section Shell		TITLE Section S	hell									^	
3.	Leave ELFORM = 2 (default, Belytschko-	1	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID	ICOMP	SETY	P			
	Tsay element formulation)		h	2	1.0000000	2	1	~ 0	0	~ 1	$\sim$			
4.	Type 0.1 in [T1] field	2	<u>T1</u> 0.1000000	<u>T2</u> 0 0.1000000	<u>T3</u> 0.1000000	<u>T4</u> 0.1000000	0.0	0.0	1DOF 0.0	<u>EDGS</u> 0	ET			
5.	Press Enter		Repeated I	Data by Button	and List									
6.	T2, T3, T4 fields should auto update								Data	Pt.		_		
	with 0.1 value								Repla	ace	Insert			
7.	Accept	_							Dele	ete	Help		*	
8.	Done	То	tal Card: 1	Smallest ID: 1	Largest ID: 1	Total deleted	card: 0						^	
		L		0000	<u>xovov</u>	X X X X	000	000	<u></u>	000	000		v	



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





5		Keyword	Input For	rm								
1.	Click "NewID"	NewID	Drav	N		RefBy	Add	Accept	Delete	Default	Done	2 Section Solid
		Use *	arameter	r [	Comment	(Sub	osys: 1 T01_	PADT_LS-DY	NA_BallOnP	late.k)	Setting	
		9			*SECTION_SOLI	D_(TITLE) (	(1)					
		тпи									^	
2	TITLE: Section Solid	Sect	on Solid									
 २	Leave ELEOBM = 1 (default_constant	1 <u>SECI</u>		<u>LFOF</u>	<u>RM</u> <u>AET</u>							
32	stress solid element formulation)	Repea	ted Data	ı by Bı	utton and List							
Л	Accent	의 📩		<u> </u>			Data	a Pt.				
Ţ.	Dono	2					Rep	lace	Insert			
5.	Done	<u> </u>					Del	lete	Help			
		Кереа	ted Data I	by Ві	utton and List						¥	
		Total Car	d: 1 Sma	allest	t ID: 2 Largest ID: 2 Total deleted card: 0						^	
		2										
		2									~	
			<u>v</u> vv		<u></u>		XXXX		XXX	XXXX	XXX	YAXAAAAA



# 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





#### Assign Material and Section Properties to Parts

LQ.	+++++++++++++++++++++++++++++++++++++++	Keyword I	nput Form									×
1.	[1 Plate] part should be pre-selected	NewID	Draw		RefBy	Pick	Add	Accept	Delete	Default	Done	1 Plate 2 Ball
2.	TITLE: Plate	Use *Pa	arameter 🗌 Comment			(Su	ubsys: 1 T01_PA	NDT_LS-DYN	IA_BallOnP	late.k)	Setting	
52					*PART_(T	ITLE) (2)						
3	Press [•] next to [SECID]	1 TITLE										
л. Л	"Link SECTION" nonun window will	Plate						_				
4.		2 <u>PID</u>	SECID  MID	EOSID	HGID •	GRAV	ADPOPT		•			
28.	appear					0	~ 0					
5.	Double click on [1 Section Shell]							_				
		COMMEN	11:									
6.	Press [•] next to [MID]											
7.	"Link MAT" popup window will appear											
8.	Double click on [1 Steel]										~	
9	Accept	Total Card	l: 2 Smallest ID: 1 Largest ID: 2	2 Total deleted	I card: 0						^	
Ò												
					x x x z	5 X X I	<u></u>	XXX	2. 2. 2		×	
	55555555555555555555555555555	Link SECTIO	N X	Link V	IAT		×					
		SECTION	~	MAT								
		2 Section Sh	hell blid	2 Rigio	l d							
				53			2					
				20								
				82								
												XXXXXX
	PADT We Make Innovation Work	Read	NewKwd NewEntity	R	ead N	lewKwd	NewEntity					8888888
	www.padune.com		Cancel Done		Cancel	Done	e					22222

#### Assign Material and Section Properties to Parts

666666666666666666666666666666666666666	Keyword Input Form			
1. Switch to [2 Ball] part	NewID Draw	RefBy Pick Add A	Accept Delete Default	Done 1 Plate
2. TITLE: Ball	Use *Parameter Comment	(Subsys: 1 T01_PAD	T_LS-DYNA_BallOnPlate.k) S	etting
		*PART_(TITLE) (2)		
3. Press [•] next to [SECID]	1 <u>TITLE</u>			
4. "Link SECTION" popup window will				
appear			0	
5. Double click on [2 Section Solid]				
	COMMENT:			
6. Press [•] next to [MID]				^
7. "Link MAT" popup window will appear				
8. Double click on [2 Rigid]				~
9. Accept	Total Card: 2 Smallest ID: 1 Largest ID: 2 Total deleted o	card: 0		^
10. Done				
				~
	Link SECTION X Link MA	л ×	2000000000	
	SECTION	5		
	1 Section Shell 2 Section Solid 2 Rigid			
		5		
	l 🕺	S		
We Make Innovation Work				2222222222222
www.padtinc.com	Kead NewKwd NewEntity Rea	Cancel Done		
}\$	Cancel Done	Cancel Done		20000000

# 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

Keyword Manager Keyword Manager × 1. Switch radio button from "Model" to "All". Keyword Edit Keyword Search Keyword Edit Keyword Search Edit: CONTACT Edit: CONTACT\_AUTOMATIC\_SURFACE\_TO\_S ~ Edit Edit O Model O All RefBy ○ Model ● All RefBy Name Count Name Count 2. Expand [CONTACT] ⊞ CONSTRAINED AUTOMATIC\_SINGLE\_SURFACE\_TIEBREAK\_BEAM\_( CONTACT AUTOMATIC\_SINGLE\_SURFACE\_MORTAR AIRBAG\_SINGLE\_SURFACE AUTOMATIC\_SINGLE\_SURFACE\_SMOOTH AUTOMATIC\_BEAMS\_TO\_SURFACE AUTOMATIC\_SINGLE\_SURFACE\_TIED 3. Scroll down and double click on AUTOMATIC\_SURFACE\_TO\_SURFACE AUTOMATIC\_GENERAL AUTOMATIC\_GENERAL\_EDGEONLY AUTOMATIC\_SURFACE\_TO\_SURFACE\_COMPOSITE [AUTOMATIC SURFACE TO SURFACE] AUTOMATIC\_GENERAL\_INTERIOR AUTOMATIC\_SURFACE\_TO\_SURFACE\_MORTAR AUTOMATIC\_GENERAL\_TIEBREAK AUTOMATIC\_SURFACE\_TO\_SURFACE\_MORTAR\_TIL AUTOMATIC\_GENERAL\_TIEBREAK\_BEAM\_OFFSET AUTOMATIC\_SURFACE\_TO\_SURFACE\_MORTAR\_TIE AUTOMATIC\_NODES\_TO\_SURFACE -AUTOMATIC\_SURFACE\_TO\_SURFACE\_ORTHO\_FRI( AUTOMATIC\_NODES\_TO\_SURFACE\_SMOOTH AUTOMATIC SURFACE TO SURFACE SMOOTH AUTOMATIC ONE WAY SUBFACE TO SUBFACE AUTOMATIC CUREACE TO CUREACE TIERREAK > ٠ ₹ > -Material arrange Material arrange GroupBy List GroupBy Sort Sort List All All All All Type Type Load From MatDB Load From MatDB Keyword Del Model Check Keyword Del Model Check ResForm ResForm ExpandAll CollapseAll ExpandAll CollapseAll

Done

Done



#### Contact



# 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**

Keyword Manager Keyword Manager × 1. Select radio button "All" if it's not selected-Reyword Edit Keyword Search Keyword Edit Keyword Search Edit: INITIAL\_VELOCITY\_GENERATION Edit: INITIAL Edit Edit  $\sim$ ○ Model ● All RefBy RefBy Name Name Count Count 2. Expand [INITIAL] TEMPERATURE\_SET ALE\_MAPPING VEHICLE\_KINEMATICS AXIAL\_FORCE\_BEAM VELOCITY VELOCITY\_GENERATION CONTACT\_WEAR 3. Scroll down and double click on DETONATION VELOCITY\_GENERATION\_START\_TIME FATIGUE\_DAMAGE\_RATIO VELOCITY NODE [VELOCITY GENERATION] FATIGUE\_DAMAGE\_RATIO\_D3FTG VELOCITY\_RIGID\_BODY FATIGUE\_DAMAGE\_RATIO\_D3PLOT VAPOR\_PART FIELD SOLID VOID\_PART FOAM\_REFERENCE\_GEOMETRY VOID\_SET FOAM\_REFERENCE\_GEOMETRY\_RAMP VOLUME FRACTION CAC MIVTURE VOLUME EDACTION CEOMETRY > < < > Material arrange Material arrange GroupBy Sort List GroupBy Sort List All Туре All All Type All Load From MatDB Load From MatDB Model Check Keyword Del ResForm Model Check Keyword Del ResForm ExpandAll ExpandAll CollapseAll CollapseAll Done Done



# Initial Velocity

-6666666666666666666666666666	Keyword Input Form	
1. Click "NewID"————	NewID Draw Pick Add Accept Delete Default Done	
2. STYP: 2	Use *Parameter Comment (Subsys: 1 T01_PADT_LS-DYNA_BallOnPlate.k) Setting	
<ol> <li>Press [•] next to [NSID/PID]</li> <li>"Link PART" popup window will appear</li> <li>Double click on [2 Ball]</li> <li>VZ = -10</li> </ol>	$\begin{array}{c c c c c c c c c c c c c c c c c c c $	
7. Accept 8. Done	COMMENT:	
	Total Card: 1 Smallest ID: 1 Largest ID: 1 Total deleted card: 0	
We Make Innovation Work	Link PARI X PART 1 Plate 2 Ball Read NewKwd NewEntity Cancel 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

Keyword Manager  $\times$ Keyword Manager 1. Select radio button "All" if it's not selected Keyword Edit Keyword Search Keyword Edit Keyword Search Edit: CONTROL Edit: CONTROL\_TERMINATION Edit Edit  $\sim$  $\sim$ ○ Model ● ● All RefBy RefBy Name Name Count Count 2. Expand [CONTROL] CONTROL SUBCYCLE\_MASS\_SCALED\_PART  $\mathbf{A}$ ACCURACY "SUBCYCLE\_MASS\_SCALED\_PART\_SET TERMINATION ACOUSTIC ADAPSTEP THERMAL\_FORMING 3. Scroll down and double click on ADAPTIVE THERMAL\_NONLINEAR ADAPTIVE\_CURVE "THERMAL\_SOLVER [TERMINATION] THERMAL\_TIMESTEP AIRBAG ALE TIMESTEP BULK\_VISCOSITY UNITS CHECK VIBRO\_ACOUSTIC CHECK\_SHELL DAMPING ¥ COADCENT D DATADACE < > < > Material arrange -Material arrange List GroupBy Sort GroupBy Sort List  $\sim$  All All All Туре All Туре Load From MatDB Load From MatDB Model Check Keyword Del Model Check Keyword Del ResForm ResForm ExpandAll ExpandAll CollapseAll CollapseAll Done Done



### 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	
	Clear Accept Delete Default Done
Use *Parameter Comment	(Subsys: 1 T01_PADT_LS-DYNA_BallOnPlate.k) Setting
*CONTROL_TERMINATION (1)	
1         ENDTIM         ENDCYC         DTMIN         ENDENG         ENDMAS         NOSOL           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 ms/ 0.1 ms = 100) with results written to disc.

2. Accept

3. Done

Ke	yword Input I	orm										
							Pick	Accept	Delete	Default	Done	1
	] Use *Parame	ter Cor	nment			(Sub	sys: 1 T01_	PADT_LS-DYN	NA_BallOnP	late.k)	Setting	
				*DA	TABASE_BINA	RY_D3PLOT	(1)					
	DT	LCDT •	BEAM	NPLTC	PSETID •							
	p.1000000	0	0	0	0							
	<u>IOOPT</u>	RATE	CUTOFF	WINDOW	TYPE	PSET •						
	0 ~	0.0	0.0	0.0	0 .	~ 0						
											\$	
Γo	otal Card: 1 S	mallest 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]

Keyword Manager	×
Keyword Edit Keyword Search	
Edit: TITLE_TITLE	Edit
🔿 Model 💿 All	RefBy
Name	Count
⊕RVE	^
<b>⊞</b> SECTION	2
⊕ SENSOR	
SET	1
⊕ STOCHASTIC	
⊞STRESS	
⊡TITLE	
TRANSLATE	
⊕. · USER	~
<	>
Material arrange	
GroupBy Sort List	
All $\sim$ Type $\sim$ All	$\sim$
Load From MatDB	
Model Check Keyword Del	ResForm
ExpandAll CollapseA	I
Done	



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

Keyword Input Form				
NewID		Add Accept	Delete Defa	ult Done
Comment		(Subsys: 1 N	lew_Subsystem_1)	Setting
*1	TITLE (0)			
1 <u>TITLE</u>				
Ball Impact on Plate				
COMMENT:				
				~
				~
TITLE:=Heading to appear on output and in output files.				^
				~

#### 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

Keyword Mana	ger	×
Keyword Edit	Keyword Search	
Edit:	\ \	Edit
	● Model O All	RefBy
Name		Count
	(	1 ^
		1
		1
⊡ ··· DATABASE		1
<b>⊞</b> ELEMENT		9500
⊞…INITIAL		1
⊞ MAT		2
⊞…NODE		9952
⊞ PART		2
■SECTION		2
⊞ SET		1
<		
Material arran	ge	
GroupBy	Sort List	
Model	$\sim$ Type $\sim$ All	$\sim$
	Load From MatDB	
Model Ch	eck Keyword Del	ResForm
	ExpandAll CollapseAl	I
	Done	

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

Element Quality Keyword	Check Cont	tact Check	Model Chec	k Setting	
Total	Error(0)	Warning(1)	UnRef(0)	UnDefined(0)	
⊞…BOUNDARY(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
⊞ ·· CONTACT(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
⊞…CONTROL(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
⊞ DATABASE(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
ELEMENT(9500)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
⊞ INITIAL(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
MAT(2)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
⊞ NODE(9952)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
⊞…PART(2)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
ECTION(2)     SECTION(2)	Error(0)	Warning(1)	UnRef(0)	UnDefined(0)	
⊞SET(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
⊞TITLE(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)	
☑ Do not Check Contact	Recheck	Mode	el Clean	Write	Snap



#### 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 —

	28888888888888888888888888888888888888	💕 LS-F	Run	n															_		$\times$	
		File Se	ettin	ings	Licen	se l	Manua	als H	elp													
2.	NCPU: 1 (default)	INPUT	C:	C:\Us	Jsers\		N	Deskto	op∖tm	p\T01	Impact	BallOn	Plate.k			~	🔒 N	CPU	1	~ 1	1 \	/
	a) Change to desired/available number of CPUs, if you'd like to speed up computation time	SOLVER LS-DYN Preset	R C: NA (	C:\Pr Con	rogram f mmand SMP sing	Files\/ gle-p	ANSYS	S Inc∖va	202\ar	nsys\ł	in\winx6	54\Isdy	na.exe			~		IEMOR	20m	~ <		0
3.	Check box to start LS-DYNA in a command prompt window	Expres Previev	ssior w	on [	"\$SOLVE "C:\Prog	ER" i= gram	\$INPU Files\/	JT men ANSYS	nory= Inc\v/	\$MEN 202\a	IORY -np nsys\bin\	sNCF winx6	U 4∖Isdyna	.exe" i=	:C:\Use	rs\	-	:\Desk	top\tm	p\T01_	✓ ■	
	a) Command prompt will display useful information during computations	Job Ta	0 able	∨ e V	Uoca WinHPC	l Usag Ir	ye Cli nput F	uster st	tatus	stop		Run	Comma	۲ nd	no file		Statu	× Ls pp	mess	ag ET	∼ E	]
4.	Hit "Run"																					



#### Submit Analysis in LS-Run

#### 1. Command prompt window will display miscellaneous information

#### C:\Windows\SYSTEM32\cmd.exe

initial kinetic energy = 0.252892	240E+02		
The LS-DYNA time step size should to avoid contact instabilities. bigger then scale the penalty of	d not excee If the ste the offend	d 1.792E-0 p size is ing surface.	94
Memory required to begin solution Additional dynamically allocated	n : memory: Total:	727K 1018K 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 d3pl	ot file	09/06/23 20:05:36
cpu time per zone cycle		0 nanoseconds	5
average cpu time per zone cycle.		0 nanoseconds	5
average clock time per zone cycl	e 1	43 nanosecond	5
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 -+ (:)-	00/06/02 00:05:26
150 t 9.9518E-02 dt 6.08E-04	write dapi	ot file	09/06/23 20:05:30
460 t 1.9970E-01 dt 0.08E-04	write dapi	ot file	09/00/23 20:05:30
600 + 3 0070E-01 dt 6 67E-04	white dapl	ot file	09/00/25 20.05.57
750 ± 4 0082E-01 dt 6 67E-04	write d3pl	ot file	09/06/23 20:05:37
900 + 5 9982E-01 dt 6 67E-04	write d3pl	ot file	09/06/23 20:05:37
1050 t 6.9994F-01 dt 6.67E-04	write d3pl	ot file	09/06/23 20:05:37
	mile appr		00,00,20 20.00.07

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

LS-DYNA command         Preset       SMP single-precision         Expression       "\$SOLVER" i=\$INPUT memory=\$MEMORY -np \$NCPU         Preview       "C:\Program Files\ANSYS Inc\v202\ansys\bin\winx64\lsdyna.exe" i= C\Users\bronislav.piak\Desktop\tmp\T0         Image: Down and the status       Image: Down and the status         Image: Down and the status       Image: Down and the status		Program Files\ANSYS Inc\v202\ansys	s\bin\winx64\lsdyna.exe	- E MEMORY	20m ~ 🛛
Expression       "\$SOLVER" i=\$INPUT memory=\$MEMORY -np \$NCPU         Preview       "C:\Program Files\ANSYS Inc\v202\ansys\bin\winx64\lsdyna.exe" i= C\Users\bronislav.piak\Desktop\tmp\T0         Image: Clocal interval       Image: Clocal interval         Job Table       WinHPC Usage         Cluster status       Image: Cluster status	reset	ommand SMP single-precision	~		
Preview       "C:\Program Files\ANSYS Inc\v202\ansys\bin\winx64\Isdyna.exe" i= C\Users\bronislav.piak\Desktop\tmp\T0         Image: Clocal index of the status       Image: Clocal index of the status         Image: Clocal index of the status       Image: Clocal index of the status	xpression	"\$SOLVER" i=\$INPUT memory=\$M	EMORY -np \$NCPU		~
ID Input File Run Command Status	ob Table	WinHPC Usage Cluster status			
	ID	Input File	Run Command	Status	ETA
1 C:\Users\ \Desktop\tmp\T01_Im"C:\Program Files\ANSYS Inc\v2 <mark>Running 56%</mark> 4s		Jsers\ \Desktop\tmp\T0	)1_Im <sup>=</sup> "C:\Program Files\ANSYS Inc\v2 <mark>Ru</mark>	nning 56%	4s



Open file for selected job in LS-PrePost

# 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

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

ile Settin	ngs License Manuals Help			
NPUT C:	:\Users\bronislav.piak\Desktop\tmp\]	T01_ImpactBallOnPlate.k	✓	~ 1
SOLVER C:	\Program Files\ANSYS Inc\v202\ansy	s\bin\winx64\lsdyna.exe	V MEMORY 20r	n v on Au
LS-DYNA	command			
Preset	SMP single-precision	V		
Expressior	n "\$SOLVER" i=\$INPUT memory=\$M	IEMORY -np \$NCPU		~
Preview	"C:\Program Files\ANSYS Inc\v202	?\ansys\bin\winx64\lsdyna.exe" i=C:\	Users\bronislav.piak\Desktop\t	mp\T01_lr
Job Table	WINHPL Usage Cluster status			
Job Table	Input File	Run Command	Status	ETA
ID 1 C:\	Input File	Run Command	Status Finished (Normal Termination)	ETA Os
ID 1 C:\	Input File	Run Command 01_In <sup>•</sup> "C:\Program Files\ANSYS Inc\v/	Status Finished (Normal Termination)	ETA Os
ID 1 C:\	Input File	Run Command 01_Im "C:\Program Files\ANSYS Inc\v/	Status Finished (Normal Termination)	ETA Os
ID 1 C:\	Input File	Run Command 01_Im "C:\Program Files\ANSYS Inc\v/	Status Finished (Normal Termination)	ETA Os
ID 1 C:\	Input File	Run Command 01_Im "C:\Program Files\ANSYS Inc\vi	Status Finished (Normal Termination)	ETA Os
ID 1 C:\	Input File	Run Command 01_In <sup>-</sup> "C:\Program Files\ANSYS Inc\v.	Status Finished (Normal Termination)	ETA Os



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









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







#### 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	\$# 3	LS-DYNA	Keyword f	ile create	d by LS-Pr	ePost(R) V	4.7.7 - 17	Feb2020	
2	\$ <b>#</b> (	Created	on Sep-11-	-2023 (08: <mark>2</mark>	23:52)				2
3	*KE	YWORD							
4		TLE							2
5	\$ <b>#</b>								title
6	Bal	l Impac	t on Plate						2
7	¦‡*CO	NTROL_T	ERMINATION						2
8	\$#	endtim	endcyc	dtmin	endeng	endmas	nosol		2
9		10.0	0	0.0	0.0	1.000000E8	0		2
10		TABASE	BINARY_D3P	LOT					1. 2
11	\$#	dt	lcdt	beam	npltc	psetid			2
12		0.1	0	0	0	0			
13	\$#	ioopt	rate	cutoff	window	type	pset		
14		0	0.0	0.0	0.0	0	0		
15	⊨*BO	UNDARY_	SPC_SET						
16	\$#	nsid	cid	dofx	dofy	dofz	dofrx	dofry	dofrz
17		1	0	1	1	1	0	0	0
18		T_NODE_	LIST_TITLE						
19	NOD	ESET (SP	C) 1						
20	\$#	sid	da1	da2	da3	da4	solver		5
21		1	0.0	0.0	0.0	0.0	MECH		6
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

