First Dyna/Primer Tutorial

Laurence Marks

January 2022



In this first tutorial we use Primer and LS-Dyna to model a steel ball impacting an aluminium plate. The mesh is available as a Nastran BDF file (actually created in Abaqus). By completing this exercise you should learn about the program structure of Primer and Dyna, how to define a basic explicit solution and how to look at the results.



Problem Description

In this problem a steel ball (25mm diameter) hits an aluminium disc at 10m/s. The high speed, short duration of the event, with the ball free to move, makes it an ideal application of an explicit solver.



The model is solved twice, initially with elastic material properties, then with plastic properties. In the second case we can observe plastic deformation and hence a permanent shape change. We keep the steel properties linear but use a bi-linear material definition for the aluminium plate.

The ball is meshed using lower order tetrahedral elements, the aluminium plate with lower order, thin shell elements.







Step 1: Read meshed model from Abaqus via Nastran BDF file

Question 1: Where do you switch on the element boundaries?

Translate abaqus inp file as shown below



Step 2: Fix periphery of target

Create node set for definition of constraint (SPC) (much easier than doing this as a single operation)



Switch on display of nodes – "ent" on display menu region, followed by select drawing all nodes, as shown below.



-			ENTITIES			?	i×							
Dismiss		U	pdate	H	slp									
Туре	Name	Labe	i Draw	Туре	Name	Label	Draw		F	,	PART TREE			
ALL TYPES	× ×	1>	< < ×	ALL NODES			V	- -						
ELEMENTS				ATTACHED 8			~		Onte - B	iacă: Lic	winder One	u Sketc	the Include	í l
AFBAG				UNATTO			~		Tune Bet	trach C	These Parts	E Cala	at Lease	
ALE				01070	_				Type + Rei	iresri C	Cation. 2011	3000	or Nasonia	3
BOUNDARY				PARTS	_	_			Contents	ina				
CONNECTION					_		_		80	M1 ()				
CONSTRAINED.				All elements	⊻ ×	⊻ ×	⊻ ×			a 🗖 1	IPSOLD 1	5		
CONTACT				BEAM			×			a 🗖 2	IPSHELL	'n		
DAMPING				BEAMPULLEY		_	~		-					- 61
DATABASE				BEAM SOURCE		1	×						_	
DEFINE				DISCRETÉ			1	k-						
DEF_TO_RIG				DISCRETÉ SPH		-	1	- 1 K						
INTIAL				INERTIA			×.	1						
INTERFACE				MASS		-	-	* *						
LOAD				MASS MATRIX			~	**						
PARTICLE				MASS PART		_	-	7*						
RIGIDAVALL				SBELT		_	M	* *						
SET				ACCEL		-	1	1.4						
TARGET				PRETENS		_	×.	- 3						
				RETRACT			×.	× 0						
MESH			~	SENSOR		-	1	* 7						
			-	SUP			÷.	* *						
GEOMETRY			~	SHELL SHELL MURRED		-	č.	*						
				SPL NURBS P		-	÷.	15						
TARGET MARKER				SUL RURDS F		-		<i>k</i>						
				SOUD		-	7	č.						
MOROPH		_	_	5000		=	÷							
				TSHELL			÷.							
						_								
													1.	
													<	- v
Label limit	Automa	dically o	display labs	lied/harried entity type:	5									
1000	Ignore I	,int an	d Generate	al labels										_
Labelled with			Draw	associated data				Manual	CT SI	Node pla	ti dal	មភាទ	hr-Fil Save P	Lock
V Lobel	_ EQ S	tote	0	vect	2			Stop	Timestep P nd	t Vels (Tr	DE AC	Zoom	CN -	Al
Model	Secti	on	Tr	iad Triad / Local)	(elem			Tidy P	AYY AYZ	V7 al	80		R	Rev
Pert	Hour	2000	Lo	cel X AXES: Elemen	t Ax 🔻				-XV -V7	N7 .8	50			Ect
Material	Therr	n Met	Di	aw Beam End Release	18				-X1 - 172 -	- AL				0.5

Now create a set, call it something like "fixed_periphery_nodes"



Select "add items" and create set. Now we need to create the boundary condition, called an SPC in Primer.



Page | 4



The keyword editor dialog is used to define the actual nature of the SPC. We are going to fix all degrees of freedom.

- 1. Change the zeros to 1's in the DOF's that you want to fix that's all of them.
- 2. Select nodes (NSID select the set we created) then hit create and update.



You should now have fixed the periphery of the plate. That was simple wasn't it? Its one command in Abaqus CAE.





Step 3: Apply initial velocity to ball.

Create a node set for the initial velocities. Call it "initial_velocity" and select all the nodes in the ball. (Click add node then drag a window round the ball.)



We now define an initial velocity for the set "initial_velocity



Keyword: M1/INITIAL_VELOCITY																
CANCEL RESET				ALL		HELP		Key- = word = format =								
UPDATE CHECK_ALL SKETCH_ALL									LL	Single ≡		=				
Keyword M1 INITIAL_VELOCITY (1/1 mod)										layout E						
															√ ×	A
	#	Options.	7	NSID	S_NO	NSIDEX	s_NO	BOXID	вох	IRIGID	Var	ICID	CSYS			1
				VX	F	VY	F	٧Z	F	VXR	F	VYR	F	VZR	F	2
				VXE	F	VYE	F	VZE	F	VXRE	F	VYRE	F	VZRE	F	3
		Create	►	2	v	0	T	0	v	0		0	v			
				0.0		10000.0		0.0		0.0		0.0		0.0		
		1	⊳	2	T	0	T	0	Ŧ	0		0	Ŧ			
				0.0		40000.	.0	0.0		0.0		0.0		0.0		
1																
4																

Set the initial velocity to be 40e3 mm/s2 in the y direction – if this has worked the screen updates with some arrows in the nodes on the ball.

Page | 6



Step 4: Define the contact

We have to define the nature of the interaction of the ball and plate. Rather than selecting "keyword" this time we select create



Give the interaction the original name "contact" and select "automatic single surface". There are many contact options – this isn't the place to look at the differences. Hit create contact and the contact is defined.





The screen is now an awful mess.

Step 5: Review the material definitions

We have defined material properties in the Abaqus model. These are linear, and in fact the first runs we'll do will use a linear material definition.



These properties are fine – we'll change them to a simple non-linear model after the first run. At this stage "abort modify".



Step 6: Define some controls for the Dyna Solver

Obviously there are lots of parameters that the solver uses. In this initial example we are only going to modify a couple of the defaults.



So we select control, we go to modify, then select standard. We set the termination time of the solution to 4e-3s.

Step 7: Write the solver file

The role of Primer is to write a file for the Dyna solver. You'll probably have to close loads of dialog boxes in the bottom left hand side of the screen to get back to this one.



Give the file a name like "tutorial_1.key" Don't close primer.



Step 8: Run the model in LS Dyna



When its running Dyna reports progress to a text window. When it has finished "hit any key to continue"



Step 9: Post Processing (displaying the results)

If we go back to the LS-Dyna environment we can fire up the D3Plot post processor. If you chose the results file with the same name as the key file with a PTF extension rather than key.



We'll now see the familiar plate and ball. Hit the play button and the ball will bounce off the plate.



Contour

To display the stresses push the contour button. What you'll get is essentially nothing as the default plot type is plastic strain and because the properties in this first run are linear you haven't got any.



	X_DIRECT_STRESS
	Y_DIRECT_STRESS
Page Number : Page Number :	Z_DIRECT_STRESS
D3PLOT T/HIS R Primer Tune Memory D3PLOT T/HIS R Primer Tune Memory	XY_SHEAR_STRESS
Attached Deform Measure Utilities Attached Deform Measure Utilities	YZ_SHEAR_STRESS
Blank Disp opt PropTies Vol Clip Blank Disp opt PropTies Vol Clip	ZX_SHEAR_STRESS
Colour Entity Trace Write Colour Entity Trace Write	Derived stresses
Cut Sect Groups User Data XY Data Cut Sect Groups User Data XY Data	PRESSURE
Q Data Part Tree JavaScript Layout Q Data Part Tree JavaScript Layout	VON_MISES_STRESS
	SIGNED_VON_MISES_STRESS
Scalar 1 Scalar 2 Vector "Vel" Scalar 1 Scalar 2 Vector "Vel"	MAX_PRINC_STRESS
Scalar 1 Active Scalar 1 Options	MID_PRINC_STRESS
Colonaria Statio	MIN_PRINC_STRESS
Category: Stress	MAX_SHEAR_STRESS
Component: PLASTIC_STRAIN T Component: VON_MISES_STRESS	MAX_DEV_PRINC_STRESS
Contours : 13 Auto all Medium Options Contours : 13 Auto all Medium Options	MID_DEV_PRINC_STRESS
Max & Min : Show max & min only v Options Max & Min : Show max & min only v Options	MIN_DEV_PRINC_STRESS
Envelope : OFF Options	2D in-plane stresses
Intot: MIDDLE surface = ALL intots =	S2MAX_2D_PRINC_STRESS
Inf pt: MDDLE surface T ALL int pts T	S2MIN_2D_PRINC_STRESS
Ref frame : GLOBAL V Options	S2MAX_2D_SHEAR_STRESS
Magnitude : Magnitude x cos[phase+phi] - Magnitude x cos[phase+phi] -	Other
Averaging: ON T Attributes: Options Averaging: ON T Attributes: Options	TRIAXIALITY
Contour Levels for "Scalar 1" Strain	LODE_ANGLE
	LODE_PARAMETER

Stress tensor

So select "stress" as the category and select "von_mises_stress". There is no such thing as Signed_von_mises_stress.



We can plot element boundaries by selecting Disp_opt at the top right hand side of the screen. Choose "all edges" at the bottom of the dialog box. We then get a series of stress plots through the history of the impact.



Step 10: Updating the material to allow for yielding

If we go back to Primer we can update the aluminium properties to allow yielding to occur. Select MAT from the menu and select MAT1.



Change the material type to MAT_03_PLASTIC_KINEMATIC and define a yield of 75MPa (SIGY) and a tangent modulus of 1000MPa (ETAN). Update material.

STEP 11: Write a new file and re-run the model

Use a new model file – something like tutorial_1_plastic and re-run it – repeat step 8. You can repeat stage 9 to view the results. The ball should have put a big dent in the ally disc.



