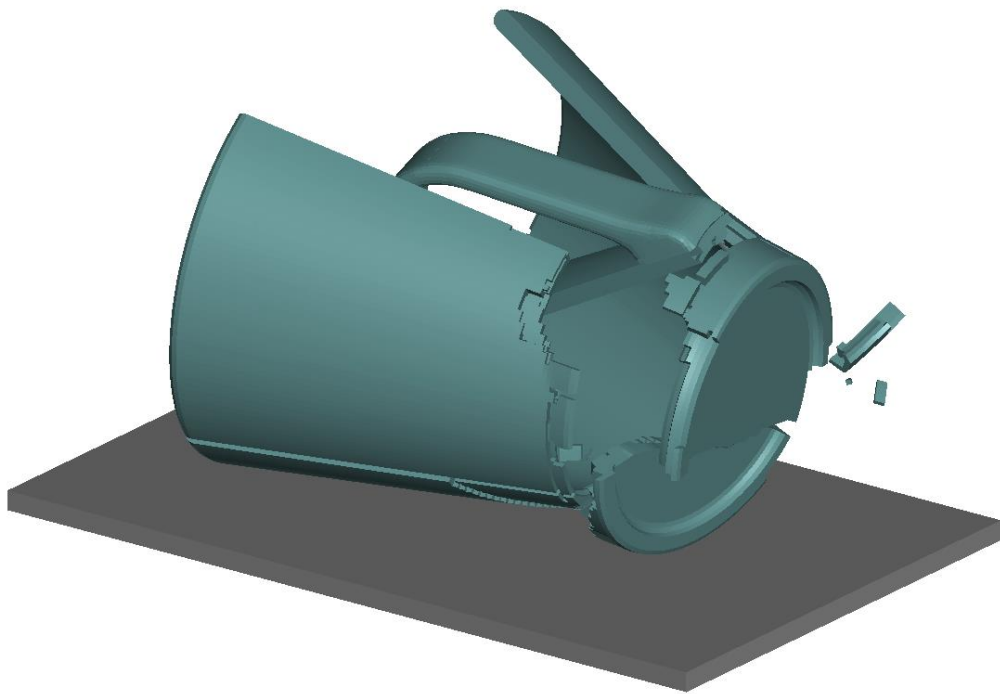


Technical Guide for Explicit Analyses using Ansys LS-DYNA



2025-06-17 v1.6

Table of content

Introduction.....	4
Support and training	4
System of units	4
References to useful information.....	5
Control cards.....	6
Database cards	12
Contact cards.....	15
Element formulations.....	23
Simulation topics.....	25
Example 1: Drop test	28
Example 2: Crash box	32
Example 3: Deep drawing.....	34
Example 4: Post-buckling strength	37
Example 5: Bolted connection	40
Example 6: Interference fit.....	43
Example 7: Rubber sealing.....	46
Example 8: Vehicle initialization for crash.....	49
Record of revisions.....	53
Copyright and Trademark Notice	53

Abstract

The purpose of this document is to provide a Technical Guide on how to build, run and analyze LS-DYNA models for explicit simulations. The idea is that the methods and settings described in this document can serve as a good starting point when analyzing similar engineering problems using the explicit solver in Ansys LS-DYNA.

This document is under continuous development and future improved revisions are expected.

Disclaimer

By using this Technical Guide, you hereby consent to this disclaimer and agree to its terms.

All the information in this Technical Guide, comprised of this document and the accompanying simulation models, is published in good faith and for general information purposes only. Neither Ansys, DYNAmore Nordic AB nor the authors make any warranties about the completeness, reliability, and accuracy of the information in this Technical Guide. Any action you take upon the information you find in this Technical Guide is strictly at your own risk. Neither Ansys, DYNAmore Nordic AB nor the authors will be liable for any losses and/or damages in connection with the use of the Technical Guide. It is always up to the user of this Technical Guide to verify the results.

Technical Guide

Introduction

The purpose of this document is to provide a Technical Guide on how to build, run and analyze Ansys LS-DYNA models for explicit simulations. The idea is that the methods and settings described in this document can serve as a good starting point when analyzing similar engineering problems using the explicit solver in LS-DYNA. The content is based on the experience of the authors and an effort has been made to use generally accepted best methods and settings, i.e. as known to the authors.

This Technical Guide is accompanied by some models that exemplify some common applications when using the LS-DYNA explicit solver. The idea is that the models can be used/modified by the reader to test ideas/functionality in LS-DYNA.

The Technical Guide is developed with MPP/ LS-DYNA in mind. This version of the Technical Guide is prepared for LS-DYNA version R16. Output files such as D3PLOT and BINOUT can be studied in any suitable LS-DYNA post-processor, such as for instance Ansys LS-PrePost.

In v1.6 of the Technical Guide a general update of the content has been made. New sections include “References to useful information” and “Simulation topics”.

Support and training

For thorough details regarding LS-DYNA keywords and material models, the reader is referred to the LS-DYNA User's Manuals¹. For help with using LS-DYNA and LS-PrePost contact your local LS-DYNA distributor. Useful web resources for general support, conference papers, example models etc can be found at lsdyna.ansys.com.

The document is under continuous development and future improved revisions are expected.

If you find errors in this document, you are welcome to contact support@dynamore.se.

System of units

The system of units used in the models is mm, ms, kg, kN, GPa

¹ ANSYS, LS-DYNA® keyword user's manual Volume 1, 2 & 3. See also <https://lsdyna.ansys.com/manuals>.

References to useful information

Manuals can be found at <https://lsdyna.ansys.com/manuals/>. Volume I is the primary source for information about functionality in LS-DYNA. It may be useful to also check the latest draft/dev manual for updated descriptions.

ANSYS, “LS-DYNA® Keyword User’s Manual Volume I”, LS-DYNA R16, (2025).

ANSYS, “LS-DYNA® Keyword User’s Manual Volume II, Material Models”, LS-DYNA R16, (2025).

ANSYS, “LS-DYNA® Keyword User’s Manual Volume III, Multiphysics Solvers”, LS-DYNA R16, (2025).

ANSYS, “LS-DYNA® Theory Manual”, (2025).

Some extra links to frequently used web pages.

- General information about LS-DYNA functionality.
<https://lsdyna.ansys.com/knowledge-base/general>
- History variables in LS-DYNA.
<https://lsdyna.ansys.com/history-variables-for-certain-material-models>
- The implicit Best Practice Guide.
<https://lsdyna.ansys.com/knowledge-base/implicit>

References to documents and papers.

- Conference papers.
<https://lsdyna.ansys.com/conference-papers>
- Useful presentations.
<https://lsdyna.ansys.com/presentations>
<https://lsdyna.ansys.com/past-dynamore-nordic-webinar-presentations>
- Haufe, A., Schweizerhof, K., DuBois, P., “Properties & Limits: Review of Shell Element Formulations”, Developer Forum, (2013).
<https://www.dynamore.de/de/download/papers/dynamore/de/download/papers/2013-ls-dyna-forum/documents/review-of-shell-element-formulations-in-ls-dyna-properties-limits-advantages-disadvantages>
- Erhart, T., “Review of Solid Element Formulations in LS-DYNA – Properties, Limits, Advantages, Disadvantages” Solid Erhart, LS-DYNA Forum, (2011).
<https://www.dynamore.de/de/download/papers/forum11/entwicklerforum-2011/erhart.pdf>
- Schmied, C., “Solid element formulations in LS-DYNA”, DYNAMore Express Webinar, (2022).
https://lsdyna.ansys.com/wp-content/uploads/2025/02/2022_solids.pdf
- Främby, J., “Review of LS-DYNA Cohesive Elements”, 2025.
<https://lsdyna.ansys.com/wp-content/uploads/2025/05/LS-DYNA-Cohesive-Elements-v1.1.pdf>
- Andrade, F., Erhart, T., “Good old *MAT_024: A review of LS-DYNA’s most popular material model”, 2020.
https://lsdyna.ansys.com/wp-content/uploads/2025/02/dynamore-express-good-old-mat_024-a-review-of-ls-dyna2019s-most-popular-material-model.pdf

Control cards

Most of the control parameters are left to their default settings. The main approach from the authors has been to rather define parameters further down in the model than on a global level on *CONTROL_OPTION, if possible. The control cards, and database cards, can be found in the include file "control_database_v1.6.k".

The parameter settings used in this Technical Guide are shown in the control cards below. The parameters that differ from defaults are marked with a blue rectangle. Dashed rectangles indicate additional parameters that are further discussed. The parameter settings may differ slightly in some of the accompanied example models to comply with the specific problem at hand.

Read the manual for more information and details regarding control cards, parameters and settings.

*CONTROL_ACCURACY

1	OSU	INN	PIDOSU	IACC	EXACC	SRTFLG
	1	4	0	0	0.0	0

Comment: Objective stress update and invariant node numbering is switched on. PIDOSU can be used to limit objective stress update to a part set.

*CONTROL_BULK_VISCOSITY

1	Q1	Q2	TYPE	BTYPE	TSTYPE
	1.5000000	0.0600000	-2	0	0

Comment: Bulk viscosity is activated for some shell element formulations in addition to the standard bulk viscosity for solids.

*CONTROL_CONSTRAINED

1	SPRCHK	SPRSMD	SPRSRCH
	1	0	0

Comment: SPR2/SPR3 initialization check. SPRCHK=1 to automatically increase search radius to find enough nodes (default) and to write a warning to stdout.

*CONTROL_CONTACT

1	SLSFAC 0.1000000	RWPNAL 1.0000000	ISLCHK 2	SHLTHK 0	PENOPT 1	THKCHG 1	ORIEN 1	ENMASS 0
2	USRSTR 0	USRFRC 0	NSBCS 0	INTERM 0	XPENE 4.0000000	SSTHK 1	ECDT 0	TIEDPRJ 0
Active optional cards <input type="radio"/> None <input type="radio"/> Opt1 <input type="radio"/> Opt12 <input type="radio"/> Opt123 <input type="radio"/> Opt1234 <input checked="" type="radio"/> Opt12345								
3	SFRIC 0.0	DFRIC 0.0	EDC 0.0	VFC 0.0	TH 0.0	TH_SF 0.0	PEN_SF 0.0	PTSCL 1.0000000
4	IGNORE 0	FRCENG 1	SKIPRWG 0	OUTSEG 0	SPOTSTP 2	SPOTDEL 1	SPOTHIN 0.0	DIR_TIE 0
5	ISYM 0	NSEROD 0	RWGAPS 1	RWGDTH 0.0	RWKSE 1.0000000	ICOV 0	SWRADE 0.0	ITHOFF 0
6	SHLEDG 0	PSTIFF 0	ITHCNT 0	TDCNOF 0	FTALL 1	UNUSED	SHLTRW 0.0	IGACTC 0
7	IREVSPT 0	UNUSED	COHTIEM 0	TIEOPT 0	STROBJ 0			

Comment: With RWPNAL=1 rigid entities coming into contact with rigid walls are treated with a penalty formulation. Shell thickness changes are considered in single surface contact, see THKCHG and ISTUPD on *CONTROL_SHELL. Actual shell thickness is used in single surface contact, see SSTHK. IGNORE, SHLEDG and PSTIFF can be activated on a global level here or locally on *CONTACT (which the authors recommend). SPOTDEL is activated so that the spot weld is deleted when an attached shell fails. SPOTSTP=2, to report deleted spot welds due to SURFB in the contact definition not found. STROBJ=1 to turn on strong objectivity in non-groupable single surface contacts using SOFT=0 or 1. It may increase robustness but comes with an extra cost. COHTIEM=1 is a flag to treat how the mass from SURFB of a tied contact affects the time step estimation of cohesive elements. May in some cases increase robustness. FRCENG to get more output and FTALL is to get correct output. ISLCHK initial penetration check is hard coded on, i.e. at time zero.

*CONTROL_DYNAMIC_RELAXATION

1	NRCYCK 250	DRTOL 0.0010000	DRFCTR 0.9950000	DRTERM 0.0	TSSFDR 0.0	IRELAL 0	EDTTL 0.0400000	IDRFLG 0
---	---------------	--------------------	---------------------	---------------	---------------	-------------	--------------------	-------------

Comment: Dynamic Relaxation (DR) is activated by SIDR≠0 on any *DEFINE_CURVE in the model. DR can be fully disabled in the simulation by IDRFLG=-999. There are different ways to apply a preload during the DR-phase. Using a ramp up to a constant level is often preferred since it introduces less impulsive loads to the system compared to applying the full load directly. However, for small loads, such as gravity, a steeper or constant load curve might be more appropriate. By DRTERM a time limit can be set, which is typically suitable. Check the convergence with ASCII file relax. Check animations and fringe plots of the DR-phase with D3DRLF, see *DATABASE_BINARY_D3DRLF. Also check so that the correct

preload level is attained and carried over to the transient analysis. If the solution converges too early, then tightening the tolerance criterion DRTOL may improve. It can be mentioned that resultant contact forces during DR can be output using *DATABASE_RCFORC_DR.

*CONTROL_ENERGY

1	<u>HGEN</u> 2	<u>RWEN</u> 2	<u>SLNTEN</u> 2	<u>RYLEN</u> 2	<u>IRGEN</u> 2	<u>MATEN</u> 2	<u>DRLEN</u> 2	<u>DISEN</u> 1
---	------------------	------------------	--------------------	-------------------	-------------------	-------------------	-------------------	-------------------

Comment: Energy dissipation calculation is switched on for all relevant parameters.

*CONTROL_OUTPUT

1	<u>NPOPT</u> 1	<u>NEECHO</u> 3	<u>NREFUP</u> 1	<u>IACCOP</u> 1	<u>OPIFS</u> 0.0	<u>IPNINT</u> 0	<u>IKEDIT</u> 100	<u>IFLUSH</u> 5000
Active optional cards <input type="radio"/> None <input type="radio"/> Opt1 <input type="radio"/> Opt12 <input type="radio"/> Opt123 <input type="radio"/> Opt1234 <input checked="" type="radio"/> Opt12345								
2	<u>IPRTE</u> 0	<u>IERODE</u> 1	<u>TET10S8</u> 2	<u>MSGMAX</u> 50	<u>IPCURV</u> 0	<u>GMDT</u> 0.0	<u>IP1DBLT</u> 0	<u>EOCS</u> 0
3	<u>TOLEV</u> 2	<u>NEWLEG</u> 0	<u>FRFREQ</u> 10000	<u>MINFO</u> 0	<u>SOLSIG</u> 0	<u>MSGFLG</u> 0	<u>CDETOL</u> 10.0000000	<u>IGEOM</u> 1
4	<u>PHSCHNG</u> 0	<u>DEMEND</u> 0	<u>ICRFILE</u> 0	<u>SPC2BND</u> 0	<u>PENOUT</u> 0	<u>SHLSIG</u> 0	<u>HISNOUT</u> 1	<u>ENGOUT</u> 0
5	<u>INSE</u> 0	<u>ISOLSE</u> 0	<u>IBSF</u> 0	<u>ISSF</u> 0	<u>MLKBAG</u> 0	<u>KINENG</u> 0	<u>ISFCNT</u> 0	
6	<u>IELOGKEY</u> 0	<u>IELOGINI</u> 0	<u>IELOGSOL</u> 0					

Comment: IACCOP=1 to average nodal accelerations between output intervals for output to ASCII file nodout and the time history database d3thdt. Typically, this allows for a lower output frequency to attain a similar level of accuracy for the acceleration signal. See "Example 1: Drop test". HISNOUT=1 activates output of more information for history variables in d3hsp. NPOPT and NEECHO limit the amount of data written to d3hsp and standard out. Output of eroded internal and kinetic energy into the matsum file, see IERODE. Update of the reference node coordinate is turned on for beams for visualization purposes, see NREFUP. This requires a unique third node to be defined for each beam element. Note that TET10S8=1 can be used to output full nodal connectivity for higher order elements. Set ICRFILE to 1 or 2 to output node and element sets used to calculate secforc data for visualization in e.g. LS-PrePost. SPC2BND=1 to convert constraints defined on *MAT_RIGID and *CONSTRAINED_NODAL_RIGID_BODY_SPC to equivalent *BOUNDARY_PRESCRIBED_MOTION_RIGID. Needed for force extraction from bndout. PENOUT=1 or 2 to output information about contact penetrations for MORTAR contacts to d3plot and sleout. ENGOUT=1 to output information about minimum sliding energy density for MORTAR contacts to d3plot. OPIFS and ISFCNT both relate to interface files. I.e. OPIFS=DT is the output time interval for the interface file, see

*INTERFACE_COMPONENT_OPTION. ISFCNT is the continuity level in applying interface linking data. Applying the motion with a higher degree of continuity may reduce the level of noise that is introduced through the interface linking. In addition, the results will be less sensitive for output frequency, see ISFCNT=2 and 3. Set IPCURVE=1 to output digitized curve data to message and d3hsp files, see LCINT on *CONTROL_SOLUTION and *DEFINE_CURVE. FRFREQ is the output frequency for the failed element report in cycles in d3hsp and message files.

*CONTROL_SHELL

1	<u>WRPANG</u> 30.000000	<u>ESORT</u> 1	<u>IRNXX</u> -1	<u>ISTUPD</u> 4	<u>THEORY</u> 1	<u>BWC</u> 2	<u>MITER</u> 1	<u>PROJ</u> 1
Active optional cards <input type="radio"/> None <input type="radio"/> Opt1 <input type="radio"/> Opt12 <input type="radio"/> Opt123 <input checked="" type="radio"/> Opt1234								
2	<u>ROTASCL</u> 1.0000000	<u>INTGRD</u> 0	<u>LAMSHT</u> 0	<u>CSTYP6</u> 1	<u>THSHEL</u> 0			
3	<u>PSSTUPD</u> 0	<u>SIDT4TU</u> <input checked="" type="checkbox"/>	<u>CNTCO</u> 1	<u>ITSFLG</u> 0	<u>IRQUAD</u> 3	<u>W-MODE</u> 0.0	<u>STRETCH</u> 0.0	<u>ICRQ</u> 0
4	<u>NFAIL1</u> 0	<u>NFAIL4</u> 1	<u>PSNFAIL</u> <input checked="" type="checkbox"/>	<u>KEEPCS</u> 0	<u>DELFR</u> 3	<u>DRCPSID</u> <input checked="" type="checkbox"/>	<u>DRCPRM</u> 1.0000000	<u>INTPER</u> 0
5	<u>DRCMTH</u> 0	<u>LISPSID</u> <input checked="" type="checkbox"/>	<u>NLOCDT</u> 0	<u>ISWSHL</u> 0				

Comment: Automatic sorting of degenerated quadrilateral shell elements, see ESORT. Shell thickness change due to membrane straining is activated, see ISTUPD. ISTUPD=4 means that elastic strains are neglected for thickness update (ISTUPD=1 includes elastic strains). For a double precision implementation see ISTUPD= 6/5, i.e. for some element types. The application of ISTUPD can be restricted to a part set using PSTUPD. THEORY=1 is recommended when having non-uniform-thickness shells in contacts. LAMSHT=5 is recommended when working with layered composites, see *PART_COMPOSITE and *INTEGRATION_SHELL. Shell thickness offset, for instance by NLOC, affects the contact reference plane, see CNTCO. Instead of using the global parameter CNTCO=1 this can be controlled locally on a selected *CONTACT, see parameter SHLOFF (then set CNTCO=0). Note: moving the reference surface of a contact will come with a cost and might affect the stable time step. NFAIL4=1 to delete highly distorted fully integrated elements and print a message. DRCPSID can be used to activate drilling stiffness of a shell part set. DELFR to delete isolated or badly connected shell elements.

*CONTROL_SOLID

1	ESORT 11	FMATRIX 0	NIPTETS 4	SWLOCL 1	PSFAIL 0	T10JTOL 0.0	ICOH 101	TET13K 0		
Active optional cards <input type="radio"/> None <input type="radio"/> Opt1 <input checked="" type="radio"/> Opt12										
2	PM1 0	PM2 0	PM3 0	PM4 0	PM5 0	PM6 0	PM7 0	PM8 0	PM9 0	PM10 0
3	TET13V 1	RINRT 0	COHEQC 11							

Comment: Automatic sorting of degenerated tetrahedrons (to type 13) and pentahedrons (to type 15) as well as cohesive pentahedron types 19 and 20 to types 21 and 22, respectively, is activated with ESORT=11. Sometimes FMATRIX=2 will improve accuracy for prestressed rubber solids using reference geometry or when explicit/implicit switching is performed. When using an ERODING contact PSFAIL will limit the check for negative volume to the solid elements in the part set. ICOH=101 to delete badly connected cohesive elements and to use the higher-accuracy implicit cohesive element variants in both implicit and explicit calculations. The more accurate type 13 solid implementation is activated using parameter TET13V, highly recommended. COHEQC=11 for cohesive element quality checks with option to "Issue a warning and continue".

*CONTROL_SOLUTION

1	SOLN 0	NLQ 0	ISNAN 1	LCINT 1001	LCACC 0	NCDCE 1	NOCOPY 0	CRVP 0
---	-----------	----------	------------	---------------	------------	------------	-------------	-----------

Comment: ISNAN=1 to check for NaN in force and moment arrays. Mainly used for debugging purposes, though, we suggest to always use it. LCINT is set to 1001, default is 100. An odd number is useful when having curves that are both in the negative and the positive region of the x-axis.

*CONTROL_HOURLGLASS

1	IHQ 4	QH 0.05
---	----------	------------

Comment: This can be considered a backup hourglass formulation if *HOURLGLASS happens to be neglected on *PART, i.e. for a part where *HOURLGLASS applies. For a robustness purpose a general and popular option is IHQ=4 (QH=0.05). The authors encourage to always set the hourglass control on *PART and define it by *HOURLGLASS.

Other useful control settings

Additional control cards used in this Technical Guide are *CONTROL_MPP_IO_LSTC_REDUCED, *CONTROL_MPP_IO_NODUMP and CONTROL_MPP_DECOMPOSITION_AUTOMATIC. There are no input parameters for these cards.

The decomposition of a model can be studied in e.g. LS-PrePost by adding the keyword *CONTROL_MPP_DECOMPOSITION_OUTDECOMP. In the accompanied examples *CONTROL_MPP_DECOMPOSITION_AUTOMATIC has been used, which often makes a more efficient simulation compared to default. More advanced options are available for the user to improve efficiency further, e.g. *CONTROL_MPP_DECOMPOSITION_TRANSFORMATION or *CONTROL_MPP_DECOMPOSITION_ARRANGE_PARTS. For instance, if the deformation is mainly in the global X-direction then *CONTROL_MPP_DECOMPOSITION_TRANSFORMATION with TYPE SX=0 may be a good choice.

*CONTROL_MPP_IO_NODUMP suppresses the output of all possible restart files. Remove this card if restart files are needed.

Note that *CONTROL_TERMINATION and *CONTROL_TIMESTEP can be found in the main file *run.key*.

*CONTROL_TERMINATION

1	ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS	<input checked="" type="checkbox"/>	NOSOL
	10.0000000	0	0.0	0.0	1.000e+08		0

Comment: Termination time is set by ENDTIM.

*CONTROL_TIMESTEP

1	DTINIT	TSSFAC	ISDO	TSLIMIT	DT2MS	LCTM	ERODE	MS1ST
	0.0	0.9	0	0.0	-1.111e-04	0	0	0
Active optional cards <input type="radio"/> None <input type="radio"/> Opt1 <input checked="" type="radio"/> Opt12								
2	DT2MSF	DT2MSLC	IMSCL	UNUSED	UNUSED	RMSCL	EMSCL	IHDO
	0.0	0	0			0.0		1
3	IGADO	DTUSR	DTDYNV					
		0						

Comment: For most cases the default value of the time step scale factor TSSFAC is 0.9, but there are exceptions, see manual. Further reduction of TSSFAC can be needed to assure numerical stability. Conventional mass scaling is activated using a negative time step size for DT2MS. IHDO=1 is used for a more consistent time step calculation for solid elements. A negative number on IMSCL references a part set for selective mass scaling, see manual for details. LCTM can be used to limit the maximum timestep with a load curve.

Database cards

Output files containing results information are requested and specified in the input file using *DATABASE, see *DATABASE_BINARY_OPTION1_{OPTION2} and *DATABASE_OPTION1_{OPTION2}. There is hardly any default output data generated by LS-DYNA. The user should always decide what data to output. The database output given in this Technical Guide can be considered as a baseline to start with.

The parameters that differ from defaults are marked with a blue rectangle. Dashed rectangles indicate additional parameters that are further discussed.

Read the manual for more information and details regarding database cards, parameters and settings.

*DATABASE_BINARY_D3PLOT

1	DT 0.5000000	LCDT <input type="checkbox"/>	BEAM 0	NPLTC 0	PSETID <input type="checkbox"/>	0
Active optional cards <input type="radio"/> None <input checked="" type="radio"/> Opt1						
2	IOOPT 0	RATE 0.0	CUTOFF 0.0	WINDOW 0.0	TYPE 0	PSET <input type="checkbox"/> 0

Comment: Time interval DT between d3plot output states. Note that *DATABASE_BINARY_D3PLOT has been positioned in the main file run.key.

*DATABASE_EXTENT_BINARY

1	NEIPH 0	NEIPS 0	MAXINT 3	STRFLG 1	SIGFLG 1	EPSFLG 1	RLTFLG 1	ENGFLG 1
2	CMPFLG 0	IEVERP 1	BEAMIP 0	DCOMP 1	SHGE 2	STSSZ 1	N3THDT 2	IALEMAT 1
3	NINTSLD 0	PKP_SEN 0	SCLP 1.0000000	HYDRO 0	MSSCL 1	THERM 0	INTOUT	NODOUT
4	DTDT 0	RESPLT 0	NEIPB 0	QUADSLD 0	CUBSLD 0	DELERES 0		

Comment: Some options to control the content of the binary output databases (typically d3plot, d3thdt and d3part). NEIPH, NEIPS and NEIPB, number of additional history variables to output for solids, shells, tshells and beams. See also *DEFINE_MATERIAL_HISTORIES and HISNOUT on *CONTROL_OUTPUT for additional options. MAXINT, number of shell (and tshell) through-thickness integration points to output. BEAMIP, number of integration points to output for beams. NINTSLD, number of solid element integration points to output. To obtain values for individual integration points, set NINTSLD to 8. STRFLG=1, write strain tensor data to d3plot, elout and dynain. IEVERP=1 to output each d3plot state to a separate plot file. SHGE=2 for including shell hourglass energy density. HYDRO to output extra data

(history variables) for solids, e.g. pressure from bulk viscosity, hourglass energy per unit initial volume and internal energy per reference volume. MSSCL=1 to output incremental (added) nodal mass due to mass scaling.

*DATABASE_FORMAT



Comment: IBINARY=1, 32bit IEEE format for binary output files (e.g. d3plot) to reduce output volume for simulations made with double precision LS-DYNA by a factor of two.

*DATABASE_OPTION1_{OPTION2}

*DATABASE_BNDOUT								
\$#	dt	binary	lcur	ioopt	option1	option2	option3	option4
	0.01	0	0	1	0	0	0	0
*DATABASE_ELOUT								
\$#	dt	binary	lcur	ioopt	option1	option2	option3	option4
	0.01	0	0	1	0	0	0	0
*DATABASE_GLSTAT								
\$#	dt	binary	lcur	ioopt				
	0.01	0	0	1				
*DATABASE_MATSUM								
\$#	dt	binary	lcur	ioopt				
	0.01	0	0	1				
*DATABASE_NODOUT								
\$#	dt	binary	lcur	ioopt	option1	option2		
	0.01	0	0	1	0.0	0		
*DATABASE_RCFORC								
\$#	dt	binary	lcur	ioopt				
	0.01	0	0	1				
*DATABASE_RWFORC								
\$#	dt	binary	lcur	ioopt				
	0.01	0	0	1				
*DATABASE_SECFORC								
\$#	dt	binary	lcur	ioopt				
	0.01	0	0	1				
*DATABASE_SLEOUT								
\$#	dt	binary	lcur	ioopt				
	0.01	0	0	1				
*DATABASE_SSSTAT								
\$#	dt	binary	lcur	ioopt				
	0.01	0	0	1				

Comment: Request and control the output to different LS-DYNA ASCII databases using *DATABASE_OPTION1_{OPTION2}. Time interval between outputs DT. BINOUT is the default binary container for ASCII output using MPP. To convert BINOUT to separate ASCII files the I2a tool can be used. BNDOUT, boundary condition forces and energy. ELOUT, element data (see *DATABASE_HISTORY_OPTION). MATSUM, part energy. NODOUT, nodal motion (see *DATABASE_HISTORY_NODE_OPTION). RCFORC, resultant contact interface forces. RWFORC, rigid wall forces. SECFORC, cross section forces (see *DATABASE_CROSS_SECTION_OPTION), SLEOUT, contact interface energy. GLSTAT, global statistics and energy.

Other useful databases

D3PART, INTFOR and D3DRLF are examples of commonly used binary databases. Sometimes specific data and/or a specific output frequency are needed for a set of parts. A part set can be selected for this purpose and referenced by *DATABASE_BINARY_D3PART. The content/output to D3PART can be further specialized using *DATABASE_EXTENT_D3PART. The content of the contact interface database, the INTFOR file, is specified using *DATABASE_BINARY_INTFOR[_FILE] and *DATABASE_EXTENT_INTFOR. Observe that you need to set SAPR and SBPR directly on *CONTACT as well. The output frequency of the Dynamic relaxation database, D3DRLF, is specified using *DATABASE_BINARY_D3DRLF.

*DATABASE_BINARY_D3MAX activates the output of binary plot database D3MAX. Elements to be included in D3MAX are chosen with *DATABASE_MAX_OPTION. The parameter "OPTION" determines if maximum or minimum values are to be recorded.

*DEFINE_MATERIAL_HISTORIES offer extensive possibilities to control output to history variables such as to record max/min values over time, stress ranges, points in time and to apply operators.

*DATABASE_GLSTAT_MASS_PROPERTIES to include also mass and inertial properties in GLSTAT.

*DATABASE_SSSTAT or *DATABASE_SSSTAT_MASS_PROPERTIES for output of subsystem data to ascii file SSSTAT. Subsystems are defined by the means of part sets using *DATABASE_EXTENT_SSSTAT. "MASS_PROPERTIES" allows for additional output of mass and inertial properties

Contact cards

The main contact types that are used in the examples are *CONTACT_AUTOMATIC_SINGLE_SURFACE with SOFT=1 or SOFT=2 for shells and solids and *CONTACT_AUTOMATIC_GENERAL for beams. *CONTACT_ERODING_SINGLE_SURFACE is used for the cases where erosion of solids is expected. For some contact situations a MORTAR contact variant can be useful. It is highly recommended for implicit and implicit-explicit switching. Observe that parameter settings may be different in different contact cards shown below pending these main contact categories, SOFT=1/SOFT=2/MORTAR.

The contact settings are mostly left to their defaults. Common for these contacts is that friction coefficients are set with FS/FD/DC and contact damping is added with VDC. Using option “_ID” is recommended for many reasons.

The contact settings are shown in the cards that follow. The parameters that differ from defaults are marked with a blue rectangle. Dashed rectangles indicate additional parameters that are further discussed.

Read the manual for more information and details regarding contact cards, parameters and settings.

*CONTACT_AUTOMATIC_SINGLE_SURFACE_ID								
\$#	cid						title	
1General contact								
\$#	surfa	surfb	surfatyp	surfbtyp	saboxid	sbboxid	sapr	sbpr
	1	0	2	0	0	0	0	0
\$#	fs	fd	dc	vc	vdc	penchk	bt	dt
	0.2	0.2	0.0	0.0	20.0	0	0.01	0.00000E20
\$#	sfsa	sfsb	sast	sbst	sfsat	sfsbt	fsf	vsf
	1.0	1.0	0.0	0.0	1.0	1.0	1.0	1.0
\$#	soft	sofscl	lcidab	maxpar	sbopt	depth	bsort	frcfreq
	1	0.1	0	1.025	2	2	0	1
\$#	permax	thkopt	shlthk	snlog	isym	i2d3d	sldthk	sldstf
	0.0	0	0	0	0	0	0.0	0.0
\$#	igap	ignore	dprfac	dtstif	edgek	unused	flangl	cid_rcf
	1	2	0.0	0.0	0.0	0.0	0.0	0
\$#	q2tri	dtpchk	sfnbr	fnlscl	dnlscl	tcs0	tiedid	shledg
	0	0.0	0.0	0.0	0.0	0	0	0
\$#	sharec	cparm8	ipback	srnde	fricsf	icor	ftorq	region
	0	0	0	0	1.0	0	0	0
\$#	pstiff	ignroff	-	fstol	2dbinr	ssftyp	swtpr	tetfac
	0	0	-	2.0	0	0	0	0.0
\$#	-	shloff						
	-	0.0						

Comment: SOFT=1. Choose type of SURFA with SURFATYP, e.g. part set. The part set ID is given as SURFA. Set FS=-2 to reference a friction table, see *DEFINE_FRICTION. If the model has more than one friction table, then FD should reference a friction table ID. IGNORE=2 so that initial penetrations are tracked, which means that nodes are not moved. Information about initial penetrations at t=0.0 can be found in message files. Set FTORQ=2 to account for moments across the contact interface. Typically, it is recommended to set DTSTIF to the initial timestep size for consistency or to a specific timestep size needed for a validated component. SHLOFF is the local version of CNTCO on *CONTROL_SHELL. If used, set CNTCO=0 and SHLOFF=1 to define a local definition (on the contact at hand) instead of a global definition.

```

*CONTACT AUTOMATIC_SINGLE_SURFACE_ID
$#      cid                                     title
      1General contact
$#  surfa  surfb  surfatyp  surfbtyp  saboxid  sbboxid  sapr  sbpr
      1      0      2          0          0          0      0      0
$#  fs      fd      dc      vc      vdc      penchk      bt      dt
      0.2    0.2      0.0      0.0    20.0          0    0.01.00000E20
$#  sfsa  sfsb  sast  sbst  stsas  sfsbt  fsf  vsf
      1.0    1.0      0.0      0.0      1.0      1.0      1.0      1.0
$#  soft  sofsc1  lc1dab  maxpar  sbopt  depth  bsort  frcfrq
      2      0.1      0      1.025    3      35      0      1
$#  permax  thkopt  shlthk  snlog  isym  i2d3d  sldthk  sldstf
      0.0      0      0      0      0      0      0.0      0.0
$#  igap  ignore  dprfac  dtstif  edgek  unused  flangl  cid_rcf
      1      2      0.0      0.0      0.0          0      0.0      0
$#  q2tri  dtpchk  sfnbr  fnlscl  dnlscl  tcs0  tiedid  shledg
      0      0.0      0.0      0.0      0.0      0      0      1
$#  sharec  cparm8  ipback  srnde  fricsf  icor  ftorq  region
      0      0      0      0      1.0      0      0      0
$#  pstiff  ignroff  -  fstol  2dbinr  ssftyp  swtpr  tetfac
      1      0      -  2.0      0      0      0      1.0
$#  -  shloff
      0.0

```

Comment: SOFT=2 with SBOPT=3 and DEPTH=35. SHLEDG=1 for square shell edges flush with the nodes. Note that for DEPTH=35 the setting of SHLEDG is ignored and the behaviour is similar to SHLEDG=1. Set FS=-2 to reference a friction table, see *DEFINE_FRICTION. If the model has more than one friction table, then FD should reference a friction table ID. IGNORE=2. TETFAC can be used to scale penalty stiffness for tetrahedral solid elements. The authors recommend setting TETFAC to at least 1.0. Set FTORQ=2 to account for moments across the contact interface. Typically, it is recommended to set DTSTIF to the initial timestep size for consistency or to a specific timestep size needed for a validated component. EDGEK>0.0 to scale the penalty stiffness for edge-to-edge by EDGEK. EDGEK=0.0 to use the default penalty stiffness. The PSTIFF parameter determines the method for calculating the penalty stiffness. SHLOFF is the local version of CNTCO on *CONTROL_SHELL. If used, set CNTCO=0 and SHLOFF=1 to define a local definition (on the contact at hand) instead of a global definition. If sliding is prevalent for SOFT=2, then DPRFAC=0.001 on Optional Card C is suggested.


```

*CONTACT_ERODING_SINGLE_SURFACE_ID
$#      cid                                     title
      1General contact
$#  surfa  surfb  surfatyp  surfbtyp  saboxid  sbboxid  sapr  sbpr
      1      0      2          0          0          0      0      0
$#  fs      fd      dc      vc      vdc      penchk      bt      dt
      0.2    0.2      0.0      0.0    20.0          0      0.01.00000E20
$#  sfsa  sfsb  sast  sbst  sfsat  sfsbt  fsf  vsf
      1.0    1.0      0.0      0.0      1.0      1.0      1.0      1.0
$#  isym  erosop  iadj
      0          1          1
$#  soft  sofsc1  lcidab  maxpar  sbopt  depth  bsort  frcfrq
      2      0.1      0      1.025    3      35          0          1
$#  penmax  thkopt  shlthk  snlog  isym  i2d3d  sldthk  sldstf
      0.0          2          1          0          0          0      0.0
$#  igap  ignore  dprfac  dtstif  edgek  unused  flangl  cid_rcf
      1          2      0.0      0.0      0.0          0      0.0      0
$#  q2tri  dtpchk  sfnbr  fnlscl  dnlscl  tcso  tiedid  shledg
      0          0.0      0.0      0.0      0.0          0          0      1
$#  sharec  cparm8  ipback  srnde  fricsf  icor  ftorq  region
      0          2          0          0      1.0          0          0      0
$#  pstiff  ignroff  -  fstol  2dbinr  ssftyp  swtpr  tetfac
      1          0          -  2.0          0          0          0      1.0
$#  -  shloff
      0.0

```

Comment: The eroding contact has the same settings as the ordinary SOFT=2 contact presented above except that one extra card has been added specific for the ERODING functionality. Both EROSOP and IADJ are hard coded 1 for MPP. Note that the ERODING contact will automatically activate a negative volume failure criterion for all solid elements in the model, except if PSFAIL (*CONTROL_SOLID) is defined. Thereby limiting the criterion to solid elements in a part set. Note that if the ERODING functionality is used together with SOFT=1 then the parameter settings used for the SOFT=1 contact above should be applied.

```

*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR_ID
$#      cid                                     title
      1General contact
$#  surfa  surfb  surfatyp  surfbtyp  saboxid  sbboxid  sapr  sbpr
      1      0      2          0          0          0      0      0
$#  fs      fd      dc      vc      vdc      penchk      bt      dt
      0.2    0.2      0.0      0.0    20.0          0      0.01.00000E20
$#  sfsa  sfsb  sast  sbst  sfsat  sfsbt  fsf  vsf
      1.0    1.0      0.0      0.0      1.0      1.0      1.0      1.0
$#  soft  sofsc1  lcidab  maxpar  sbopt  depth  bsort  frcfrq
      0      0.1      0      1.025    2          2          0          1
$#  penmax  thkopt  shlthk  snlog  isym  i2d3d  sldthk  sldstf
      0.0          0          0          0          0          0      0.0
$#  igap  ignore  mpar1  mpar2  edgek  unused  flangl  cid_rcf
      1          2      0.0      0.0      0.0          0          0      0

```

Comment: The contact settings for the MORTAR-contact are mostly left to defaults. The contact stiffness can be scaled by SFS/IGAP if penetrations are unacceptable. In case of solids or tshells, setting the release depth with PENMAX may also help. PENMAX should then be set to a typical element thickness found in the model. Set FS=-2 to reference a friction table, see *DEFINE_FRICTION. If the model has more than one friction table, then FD should reference a friction table ID. For MORTAR, IGNORE=2 by default, although it behaves a bit differently compared to other contacts, i.e. no tracking of initial penetrations. If a tracked surface is desired, then IGNORE=1 can be activated. If IGNORE is set to a negative value, then contact within a part is omitted.

```

*CONTACT_AUTOMATIC_GENERAL_ID
$#      cid                      title
      1Contact for beams
$#  surfa  surfb surfatyp  surfbtyp  saboxid  sbboxid  sapr  sbpr
      1      0      2      0      0      0      0      0
$#  fs      fd      dc      vc      vdc      penchk      bt      dt
      0.2      0.2      0.0      0.0      20.0      0      0.01.00000E20
$#  sfsa  sfsb  sast  sbst  sfsat  sfsbt  fsf  vsf
      1.0      1.0      0.0      0.0      1.0      1.0      1.0      1.0
$#  soft  sofsc1  lc1dab  maxpar  sbopt  depth  bsort  frcfrq
      1      0.1      0      1.025      2      2      0      1
$#  permax  thkopt  shlthk  snlog  isym  i2d3d  sldthk  sldstf
      0.0      0      0      0      0      0      0.0      0.0
$#  igap  ignore  dprfac  dtstif  edgek  unused  flangl  cid_rcf
      1      2      0.0      0.0      0.0      0.0      0.0      0

```

Comment: *CONTACT_AUTOMATIC_GENERAL is only used for beam-to-beam contact by the authors. Typically, it is recommended to set DTSTIF to the initial timestep size for consistency or to a specific timestep size needed for a validated component. SOFT=1 and IGNORE=2. Note: There is an option to exclude contact with beams in the same part (PID) using CPARM8 (1 or 2) in the additional _MPP card. Set CPARM8=2 to consider also spotweld beams (type 9) in the contact.

Tied contacts

There are many options in LS-DYNA for modelling of tied contacts. See the LS-DYNA manual for the complete functionality that is available as well as for more details. In this section some commonly used tied contact variants are presented. The intention is not to cover all functionality but to highlight some tied contacts that are widely used, and which may serve as a good starting point. For best control it is recommended to use node and segment sets rather than part or part sets. Thereby making an effort to limit that, unintentionally, wrong nodes are being tied to the wrong surface. It may sometimes be advantageous to use `CONSTRAINED_EXTRA_NODE` instead for tying deformable nodes to rigid bodies since the nodes can be an arbitrary distance away from the rigid body.

Constraint or penalty

There are two main categories of tied contacts, i.e. constraint based and penalty based. The offset options on `OPTION4` on `*CONTACT` determine the category. If `OPTION4` is left blank the contact will be constraint based. If “`OFFSET`” or “`BEAM_OFFSET`” is used, then the contact will be penalty based. “`CONSTRAINT_OFFSET`” is constraint based. Penalty based variants will work with rigid bodies but not constraint based. That is a fundamental difference. Another functionality that is affected by `OPTION4` is the treatment of `SURFA` nodes. Whether they are moved to `SURFB` at initialization or not (allowing offsets). Hence if `OPTION4` is left blank (constraint based), then `SURFA` nodes will be projected and moved to `SURFB` at initialization. For the other `OPTION4` variants `SURFA` nodes will not be moved thus allowing offsets/play between `SURFA` and `SURFB` to remain. A table for some contacts is given below.

Contact type	Allow rigid bodies	Allow offset/play
<code>TIED_SHELL_EDGE_TO_SURFACE</code>	No	No
<code>TIED_SHELL_EDGE_TO_SURFACE_OFFSET</code>	Yes	Yes
<code>TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET</code>	Yes	Yes
<code>TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET</code>	No	Yes

By `IPBACK≠0` on Optional Card E a “backup” penalty contact will be created for `SURFA` nodes that fail to tie due to multiple constraints. The `IPBACK` flag is only applicable to constraint-based tied contacts (TIED with no options, or with `CONSTRAINED_OFFSET`).

Furthermore, if using `OFFSET/CONSTRAINED_OFFSET` it is possible to make use the `TIEDID=1` parameter to improve accuracy.

Tying tolerance

If the default tying tolerance is not appropriate then it can be set by the user, i.e., with negative numbers on SAST and SBST. For numerical robustness aim to keep the tying distance low.

For example, SAST = SBST = -1.0 will give a tolerance of $0.6 \cdot (1.0 + 1.0) = 1.2$ mm.

Highlighted contacts

The contact cards below are presented with OPTION4 (offset options) left blank but could equally been presented with other choices of OPTION4. The choice of contact typically depends on the type of elements being connected. It primarily depends on whether the nodes have rotational DOF or not. The intended use of the contacts below is revealed in their titles. Negative SAST/SBST may be used to set tying tolerance, see dashed rectangles.

```
*CONTACT_TIED_NODES_TO_SURFACE_ID
$#      cid                                     title
      1Tied solids-to-solids and solids-to-shells
$#  surfa  surfb  surfatyp  surfbtyp  saboxid  sbboxid  sapr  sbpr
      1      2          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
$#   sfsa   sfsb   sast   sbst   sfsat   sfsbt   fsf   vsf
      1.0    1.0      0.0      0.0      1.0      1.0      1.0      1.0
```

Comment: OPTION4 can be "blank", "OFFSET" or "CONSTRAINED_OFFSET". Connects translational DOF:s.

```
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_ID
$#      cid                                     title
      1Tied shells-to-shells and beams-to-shells
$#  surfa  surfb  surfatyp  surfbtyp  saboxid  sbboxid  sapr  sbpr
      1      2          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
$#   sfsa   sfsb   sast   sbst   sfsat   sfsbt   fsf   vsf
      1.0    1.0      0.0      0.0      1.0      1.0      1.0      1.0
```

Comment: OPTION4 can be "blank", "OFFSET", "BEAM_OFFSET" or "CONSTRAINED_OFFSET".

"BEAM_OFFSET" and "CONSTRAINED_OFFSET" enforces equilibrium since transmitted forces and moments are coupled. The drilling rotation constraint method that is default in implicit can be activated for a part set also in explicit, see DRCP SID on *CONTROL_SHELL. Thereby tying also torsion for beams tied to shells. Connects translational and rotational DOF:s if both sides have rotational DOF:s.

```
*CONTACT_TIED_SHELL_EDGE_TO_SOLID_ID
$#      cid                                     title
      1Tied shell edges-to-solids and beam ends-to-solids
$#  surfa  surfb  surfatyp  surfbtyp  saboxid  sbboxid  sapr  sbpr
      1      2          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
$#   sfsa   sfsb   sast   sbst   sfsat   sfsbt   fsf   vsf
      1.0    1.0      0.0      0.0      1.0      1.0      1.0      1.0
```

Comment: This contact has the capability to transfer the moment even if SURFB does not have rotational DOF:s.

Other useful information about contacts

Remember, the solution timestep size will determine how often the contact situation is evaluated and will influence the contact penetration level, penalty stiffness, robustness, and accuracy.

*CONTACT_FORCE_TRANSDUCER is useful for output of contact forces in cases with a single surface contact. A SURFB definition is optional for force transducers. If the contact forces acting between two surfaces are needed, a SURFB should be defined. In this case, only the contact force measured between SURFA and SURFB is recorded. Otherwise, the contact forces between SURFA and any surface are output.

*DATABASE_RCFORC for output of contact interface forces.

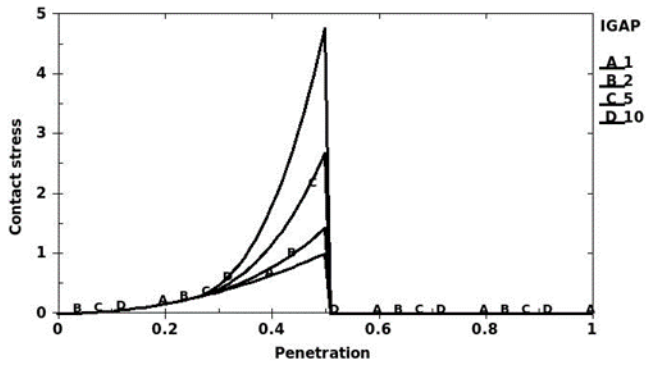
*DATABASE_BINARY_INTFOR[_FILE] is used to generate a binary contact interface force file (INTFOR) which can be used for fringe plot of e.g. contact pressure. Observe that the INTFOR file name is given either using the _FILE option or s=<INTFOR> on the command line. On the contact card at hand, SAPR=1 and/or SBPR=1 must be set. SAPR/SBPR are also needed for output of nodal interface force in the ASCII file NCFORC, see *DATABASE_NCFORC. *DATABASE_NCFORC is not recommended for large contact definitions.

Some comments about penalty stiffness

The recommendation is to begin with default settings for penalty stiffness. The penalty stiffness can be altered, if needed, with several different scaling parameters. Sometimes a reduction is needed to assure numerical stability. In other cases, an increase is needed to reduce the size of the penetrations. Note that stiffness scaling through different means will multiply.

For SOFT=1 and SOFT=2 contacts, the (global) penalty stiffness for the contact at hand can be scaled using SFSA and SFSB directly on the contact definition. The base level of the penalty stiffness for this contact can/should be set using the DTSTIF parameter. If used, DTSTIF is often set to the initial time step dt of the simulation at hand. Furthermore, for individual parts the penalty stiffness can be scaled (locally) using SSF on *PART_CONTACT. For SOFT=2 contact the penalty stiffness scale factor for edge-to-edge can be set by EDGEK, i.e. EDGEK>0.0. EDGEK=0.0 to use the default penalty stiffness. The penalty stiffness for tetrahedron elements can be scaled using TETFAC.

For MORTAR contact there are parameters to affect the penalty stiffness behaviour. The base level of penalty stiffness can be scaled with SFSA. If locally penetrations become too large then IGAP can be increased, see figure below.



PENMAX is typically used to set a consistent release depth for solid parts. However, for solids and tshells the PENMAX parameter also affects the base level of penalty stiffness for MORTAR contacts.

For tied contacts using penalty formulation (Option 4: OFFSET/BEAM_OFFSET) it has been observed that reducing the penalty stiffness, e.g. SFSA=SFSA=0.1, may increase the numerical stability. This type of stability issues may be observed through strange contact energies in sleout/glstat.

Element formulations

The element formulation for a part (PID) is set on the *SECTION card, e.g., the ELFORM parameter on *SECTION_SHELL, *SECTION_SOLID or *SECTION_BEAM.

General

The application at hand, as well as previous experiences, often contribute to the choices that are made. In explicit simulation, linear element formulations are typically favoured due to the higher cost of using higher order elements. The first choice is often whether to use an under-integrated element formulation or a more expensive fully integrated variant. There are no strict rules on how to make this choice, and it is often application based. Generally, under-integrated elements tend to be a bit more robust in terms of surviving harsh conditions but also a bit too soft compared to an equivalent fully integrated element. Given a good mesh quality and conditions that are not too harsh a fully integrated element is typically the more accurate element but more costly. Other circumstances that may indicate a favour for the fully integrated element may be the application of point loads or the build-up of excessive hourglass energy. The ability to capture, e.g. bending modes, forming of plastic hinges etc, with the level of accuracy needed will require enough integration points over the interesting region. Keep in mind that the choice that is made is not only the element formulation, i.e., the material, the element formulation and the hourglass control should be seen as a unit.

All under integrated elements are sensitive for hourglass deformations, which LS-DYNA automatically will try to restrict by adding hourglass forces. Hence, make your choice of hourglass formulation for your choice of element type and selected material properties. Preferably by referencing *HOURGLASS on *PART. For the under-integrated element, the choice of hourglass formulation and hourglass coefficient may not be trivial. Though, some guidance is given in the LS-DYNA manual. Often in automotive crash the hourglass coefficient for shells elform 2 (typically IHQ = 4, stiffness form) is decreased from the default 0.1 to 0.05 (or 0.03). For example, for solids elform 1 an hourglass formulation often used is IHQ=6. Where in the case of elastic materials, the hourglass coefficient may in some cases be adjusted (e.g. QM=1.0).

A general recommendation is to set ESORT on *CONTROL_SHELL and *CONTROL_SOLID so that degenerated quadrilateral shells are automatically switched to a more suitable triangular formulation and degenerated solids are automatically switched to a tetrahedron or pentahedron formulation. It is recommended to minimize the use of pentahedron elements when creating structured meshes.

Some valuable sources for information regarding elements in LS-DYNA are, other than the LS-DYNA manual, “Properties & Limits: Review of Shell Element Formulations” by André Haufe, Karl Schweizerhof and Paul DuBois, “Solid element formulations in LS-DYNA” by Christoph Schmied and “Review of Solid Element Formulations in LS-DYNA” by Tobias Erhart.

Elements by selection

There are many element formulations in the LS-DYNA library, and it is not the intention to give an overview of all of them in this document. The table below highlight some elements that are often used in explicit simulations and that can serve as a good starting point in many cases.

Element type and formulation	Hourglass control	Comment
Shell, elform = 2	IHQ = 4 (or 2 for viscous)	Under-integrated, very effective
Shell, elform = -16	IHQ = 8	Fully integrated, enhanced version of elform 16
Solid, elform = 1	IHQ = 6 (or 3 for viscous)	Under-integrated, very effective
Solid, elform = -1	NA	Fully integrated more efficient version of elform -2
Solid, elform = 13	NA	1-point tetrahedron element, TET13V = 1 on *CONTROL_SOLID
Solid, elform = 16	NA	4 or 5 point 10-noded tetrahedron
Solid, elform = -2	NA	Fully integrated, Like elform 2, but can handle poor aspect ratio better.
Solid, elform 19	ICOH=101 Higher accuracy implicit variant	ICOH=101 can be used both for zero volume and finite volume cohesive elements. That is connecting shells and/or solids.
Beam, elform 1	NA	Hughes-Liu with cross section integration (plastic deformation)
Beam, elform 2	NA	Belytschko-Schwer resultant beam (elastic deformation)
Beam, elform 3	NA	truss (resultant)
Beam, elform 6	NA	Discrete beam/cable (spring/damper)
Beam, elform 9	NA	Spot weld beam

Note that the more accurate type 13 solid implementation is activated on *CONTROL_SOLID with parameter TET13V. This is highly recommended by the authors. When using elform 13 one should avoid common nodes between different parts. Worth mentioning is that there are cases where the tetrahedron elform 10 element may be preferred to elform 13, e.g., when the material model requires it (typically foams) or when problem arises with negative volumes (severe deformations).

The modelling of e.g. adhesive connections, using cohesive elements will depend on the application and the choice of modelling (merge nodes?/tied nodes?/tied nodes with or without offset to shells?). The choices that are made for modelling will affect the optimal choice of cohesive element. However, the authors thinks that elform 19 with ICOH=101 provides the most flexibility between usability, robustness and accuracy. Though, it may not be the optimal choice for any given situation.

Simulation topics

Using LS-DYNA there are often multiple approaches to solve/study engineering problems. In this chapter of the Technical Guide the intention is to discuss subjects that the authors found relevant and interesting when performing simulations in LS-DYNA.

There is often a balance between efficiency, accuracy and robustness when performing simulations. As discussed earlier, control settings can be one way to address this balance. However, this balance may also be affected by choices that are made when building the model. Some of these aspects are discussed in this section.

Model discretization

Ultimately the discretization of the simulation model will influence the accuracy in the results, robustness of the simulation and the efficiency/throughput. The number of elements, their size and their formulation should be able to resolve the problem to the needed accuracy level. E.g. the FE-discretization should resolve the initial and deformed geometry.

Material modelling - Elasto-plastic metals for crash/impact

*MAT_LINEAR_PIECEWISE_PLASTIC (MAT_024) is a frequently used material model in LS-DYNA. It will serve as an example here when discussing some aspects to consider when setting up such a material model. A curve/table is frequently used to describe effective stress vs plastic strain for metals.

This curve should:

- Start at σ_{yield} at $\text{effective_plastic_strain}=0.0$.
- No segment in the curve definition should have an inclination exceeding the young's modulus.
- The last segment of the curve will be extrapolated by LS-DYNA if the plastic strains in the simulation exceed the value found at the last input point of the curve.
- The curve should be monotonically increasing with increasing plastic strains.
- Both the curve and the first derivative should be 'smooth'.
- This curve will be re-discretized by LS-DYNA with LCINT number of points. Set LCINT to an odd and sufficiently large number (e.g. 1001). Preferably, make all your input curves so that LCINT has no effect.

If a table of curves, i.e. at different strain rates, is given (instead of a single curve) to capture viscoplasticity there are some additional points that can be noted:

- The curves given should never cross each other (this is true also in the extrapolated region).
- LS-DYNA will interpolate between two curves for intermediate strain rates.
- VP=1 may be set to activate a more accurate update of the yield surface (note: can be costly).
- For strain-rate interpolation accuracy you may make use of the LOG_INTERPOLATION feature.

Material models to avoid

The authors would like to highlight two material models in LS-DYNA that should be treated with care. The first material model is *MAT_ELASTIC. This material model can carry all loads, i.e. with no plastic deformation and no failure. Rather, the recommendation is to use an elasto-plastic material model, e.g.

*MAT_024. If elastic properties are desired a reasonable high value of yield stress (an upper limit) can be set. Then possible plastic strains can be found and visualized in a post-processor. In this way, elements and regions where the yield stress has been reached can be found and identified.

The second material model is *MAT_RIGID. For deformable entities in the model/simulation the information travels with the speed of sound. However, for parts that are modelled with rigid material all information is available to the complete rigid part the next cycle (unrealistic). Furthermore, rigid entities may introduce unwanted noise in the simulation.

Adding damage/failure for robustness

Adding material failure may make the simulation more robust. Rather than keeping highly deformed/distorted elements that ruin numerical stability a failure criterion can be used to remove problematic highly distorted elements. Therefore, it is recommended to add a failure criterion that addresses those elements. Failure can, preferably, be accompanied by an added damage model so that the loads in the region are decreased over a small time-period, i.e. before the element is deleted. Adding some damage, for robust purposes, also holds true when modelling for brittle failure. In addition, keeping highly loaded elements (possibly distorted) that should have failed may open up for unrealistic load paths.

Energy dissipation in material models

Energy dissipation in material models normally calms the simulation down. For many elasto-plastic material models that are typically used for metals there is no energy dissipation when in the elastic region. This can be addressed by adding *DAMPING_PART_STIFFNESS. For foams and rubbers, it may be beneficial, from a robustness point of view, to add some hysteresis or damping to dissipate some energy.

Trouble shooting – Errors and Warnings

- Scan through all the warnings and errors in stdout, d3hsp and message files. Does it give an indication of what went wrong? Often this search is done manually. However, there are tools designed to help with this (as well as providing a means of automatization). For instance, the DYNAmore Eco System tools sets are worth mentioning.
- Initializing the model using the latest version of LS-DYNA may provide more and better information.
- Increase output frequency, e.g. with a load curve, so that more data is available (typically d3plots) to resolve and study the problem.
- Investigate the energies found in GLSTAT. Follow the energies into the MATSUM and SLEOUT. Typically, unrealistic behavior should be explained.
- Investigate, search for the unexpected, by fringe plots of the d3plot:
 - Plastic strains and stresses
 - Nodal velocity field
 - Mass scaling
 - Eroded elements
 - Etc.

- If the simulation hangs/stop during initialization or at the beginning – try to start the MPP simulation on one core. Hopefully, this may provide some more information about the issue.

Load imbalance and decomposition

There are two main variants of load imbalance that may occur in a simulation, i.e. coming from the cost of either element and material or the contact work.

In the beginning of each simulation, MPP LS-DYNA will estimate the cost of the material and element in the model and try to divide it into equally computationally 'heavy' parts for each thread. This is to decompose the model. The default decomposition is rectangular boxes. A "french fries" decomposition in the impacting direction by `*CONTROL_MPP_DECOMPOSITION_TRANSFORMATION` may improve efficiency. For example, for a case where the impact happens in the global x-direction it would be natural to use `SX=0`. An option for the user to influence the decomposition behavior for certain parts is by `*CONTROL_MPP_DECOMPOSITION_ARRANGE_PART`.

The cost estimation at the beginning of the simulation will use the default factors in LS-DYNA. However, it is possible to make a more targeted cost estimate that is based on knowledge from previous, similar, simulations. That is by using a TIMINGS-file. The procedure is as follows. First measure the cost in a simulation over a number of cycles. Typically, during a timeframe of interest. The information is output to a file called `DECOMP_TIMINGS.OUT`. Later when running a "similar" model LS-DYNA can now make use of the TIMINGS-file to make a better estimation of the cost. Observe, this only changes the cost estimation. It is still possible to combine it with for instance the "french fries" decomposition strategy.

To produce TIMINGS-file:

```
*CONTROL_MPP_PFILE
decomposition {timing_start <cycle> timing_end <cycle>}
```

To use TIMINGS-file:

```
*CONTROL_MPP_PFILE
decomposition {timing_file DECOMP_TIMINGS.OUT}
```

Example 1: Drop test

A ceramic mug is dropped from a given height, freefall until it contacts a hard surface. This example is intended to exemplify a parameter definition, an accelerometer, a strain gauge load cell and the contact treatment of eroding elements.

Note: The idea of the provided example model is to show ideas/functionality in LS-DYNA. It is not intended to demonstrate current state-of-the-art modelling technique. However, it can be used /modified by the reader to test their ideas and understanding of different functionality in LS-DYNA.



Simulation data

# nodes	184k
# elements	146k
Timestep size (ms)	0.1e-3 (constant)
Termination time (ms)	10.0

Elements and material

The mug and the surface are modelled with solid, hexahedral ELFORM -1, elements. Four elements through the thickness. The element size is 1 mm. Material type *MAT_110² is used for the mug. Elements

² Parameters collected from the paper “Implementation and Validation of the Johnson-Holmquist Ceramic Material Model in LS-DYNA”, by D. S. Cronin et al, and modified to fit the purpose of the example.

erode when the negative pressure reaches 0.04 GPa. This can be confirmed by, for instance, plotting the pressure for solid EID 817 or 19649 (depending on which element erode) in ELOUT.

Contact definitions

Contact type `*CONTACT_ERODING_SINGLE_SURFACE` with `SOFT=2` is used for the model. An eroding contact is needed so that contact is treated correctly. This means an updated contact surface after elements have been eroded. The handle is attached to the mug with a tied contact, i.e.

`*CONTACT_TIED_NODES_TO_SURFACE`.

Control cards

No mass scaling or very little mass scaling is recommended when performing drop test simulations so that the kinetics is not significantly affected due to additional, artificial, added mass. In this case mass scaling is applied, see `DT2MS` on `*CONTROL_TIMESTEP`. It adds 4 extra grams to the mug. This is an increase of 2% of the total weight of the mug. It may be acceptable given that the time stepsize was increased by a factor 2 and therefore shortening the simulation time considerably. Since the parameter `MSSCL` is set to 1 on `*DATABASE_EXTENT_BINARY` it is possible to visualize the added mass.

Loads and constraints

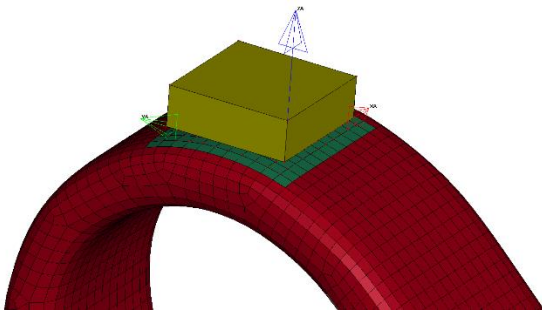
Gravity is applied with `*LOAD_BODY_Z`. The bottom of the impacted surface is fully constrained with `SPC`.

Initial conditions

The impact velocity is set with a combination of `*INITIAL_VELOCITY`, `*PARAMETER` and `*PARAMETER_EXPRESSION` so that only the drop height is required as input. The model is thereby prepared for quick configuration when performing, for example, parameter studies.

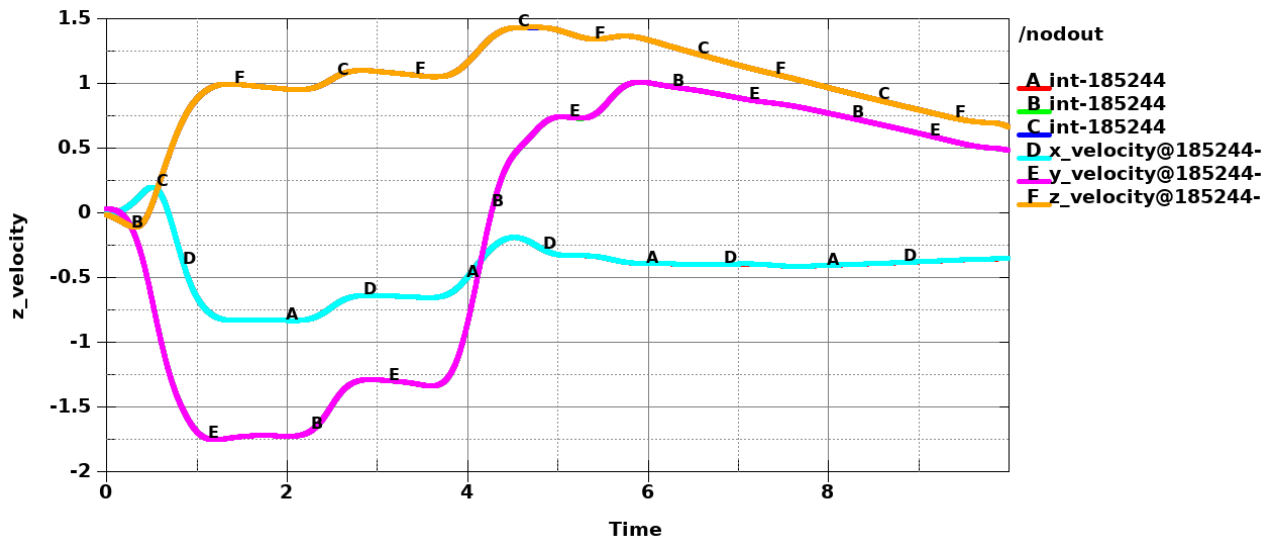
Accelerometer

An accelerometer is installed on the handle to output accelerations in a local coordinate system in `NODOUT` (NID 185243). An additional node (NID 185244) on the same accelerometer outputs accelerations in the global coordinate system.



Comment: An accelerometer is mounted on the handle.

The global node can be used to indicate if the output frequency of the accelerometer is sufficiently high. A check can be made by integrating each acceleration component (x, y, z) and compare them to the corresponding measured velocities on the global node. These curves should match when offsets are removed (due to initial velocity), see figure below.



Comment: The integrated accelerations and the velocities match (nodout, global node). A Butterworth filter with a cut-off frequency 1000 Hz is applied.

Note that the problem is very sensitive to small variations. One contributing factor to this is the modelling of the brittle failure.

Strain gauge load cells

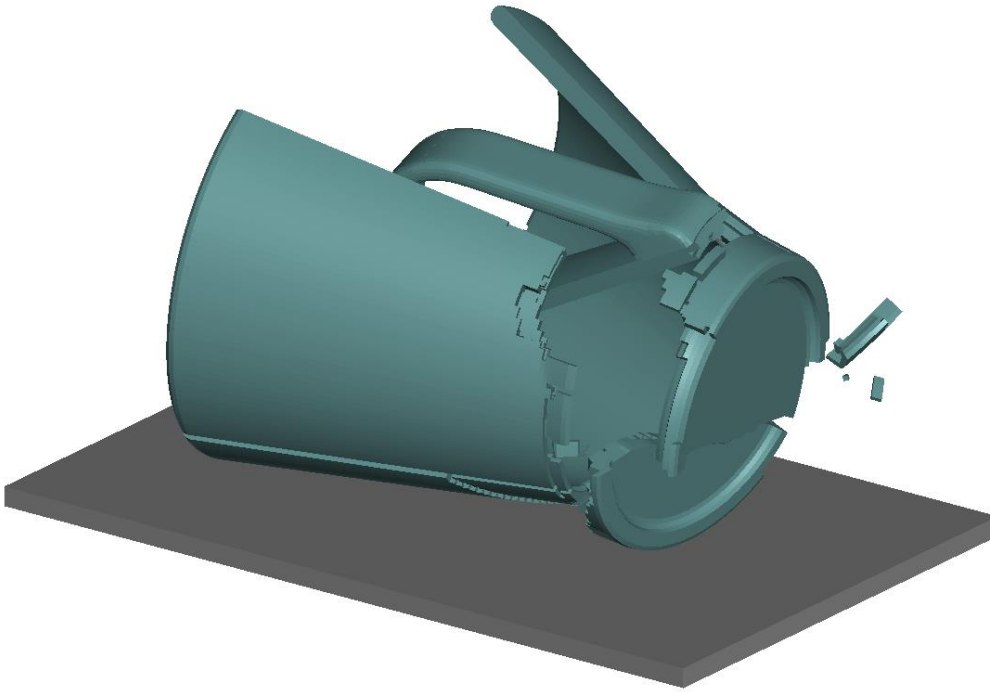
Two strain gauge load cells (shell EID 200001 and EID 200002) are installed on the inner and outer surface of the mug, respectively. The strain gauge results, given in the local coordinate system of the shell element, can be plotted from ELOUT. A Butterworth filter with a cut-off frequency 3000 Hz may be appropriate in this example. The results show (before failure) tension on the outer surface (positive x-strain) and compression on the inner surface (negative x-strain) as expected (by looking at the deformation). Note that STRFLG=1 must be set on *DATABASE_EXTENT_BINARY to get strain data in ELOUT and D3PLOT.



Comment: Strain gauge load cell (yellow shell element).

Results

In this example the brittle ceramic mug breaks. The crack propagation can be studied through D3PLOT. Evaluation of accelerations will depend highly on the chosen filter and frequency.

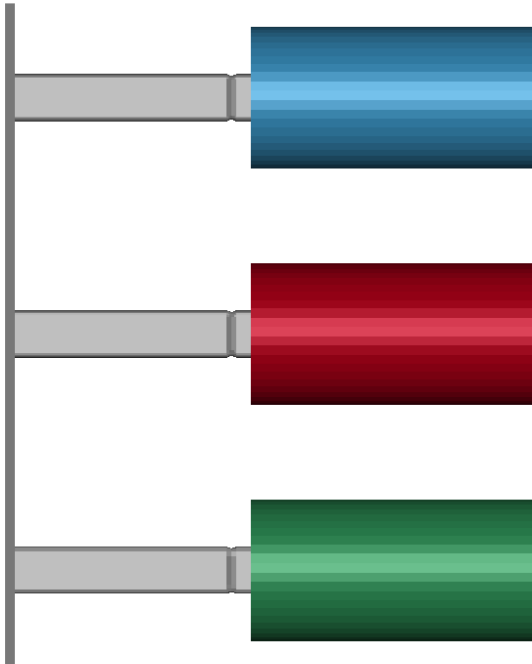


Comment: Drop test at 10 milliseconds.

Example 2: Crash box

Three identical, except for the element size, crash boxes are impacted by rigid cylinders. Each cylinder has a weight of 700 kg. The impact speed is 30 km/h. This example is intended to exemplify *MAT_024 with strain rate effects, the use of material history variables, application of contact force transducers and the concept of mesh convergence.

Note: The idea of the provided example model is to show ideas/functionality in LS-DYNA. It is not intended to demonstrate current state-of-the-art modelling technique. However, it can be used /modified by the reader to test their ideas and understanding of different functionality in LS-DYNA.



Simulation data

# nodes	81k
# elements	72k
Timestep size (ms)	0.3e-3 (constant)
Termination time (ms)	100.0

Elements and material

The three crash boxes (100x100x500 mm, thickness 2.0 mm) are modelled with shell elements ELFORM -16, which is a fully integrated element type. Hourglass control type 8 is applied for improved behaviour of warped shells. The mesh size is, for the three boxes, 10/5/2.5 mm respectively. The material model used for the steel is *MAT_024 where strain rate effects are accounted for by adding a strain rate table definition. The table includes two hardening curves corresponding to static load (strain rate=0) and strain rate 1000/s. Intermediate strain rate values in LS-DYNA are interpolated between the curves. By setting VP=1 on the material card, the numerical noise caused by strain rate effects can be minimized.

Contact definitions

The contact, for the complete model, is treated by one `*CONTACT_AUTOMATIC_SINGLE_SURFACE` contact with `SOFT=1`. A key parameter when evaluating the performance of a crash box is the force vs. displacement curve. By adding `*FORCE_TRANSDUCER_PENALTY`, one for each impactor, the force between each impactor and its corresponding crash box can be measured.

Control cards

By setting `NEIPS=1` on `*DATABASE_EXTENT_BINARY` the effective plastic strain rate (`VP=1`) for `*MAT_024` is output to the `D3PLOT` as history variable #1. Using LS-PrePost this can be visualized under “FComp-Misc-History var#1”. Note that with `HISNOUT=1` on `*CONTROL_OUTPUT` the content of each history variable is printed in `d3hsp`, search for “history variables listing”.

Loads and constraints

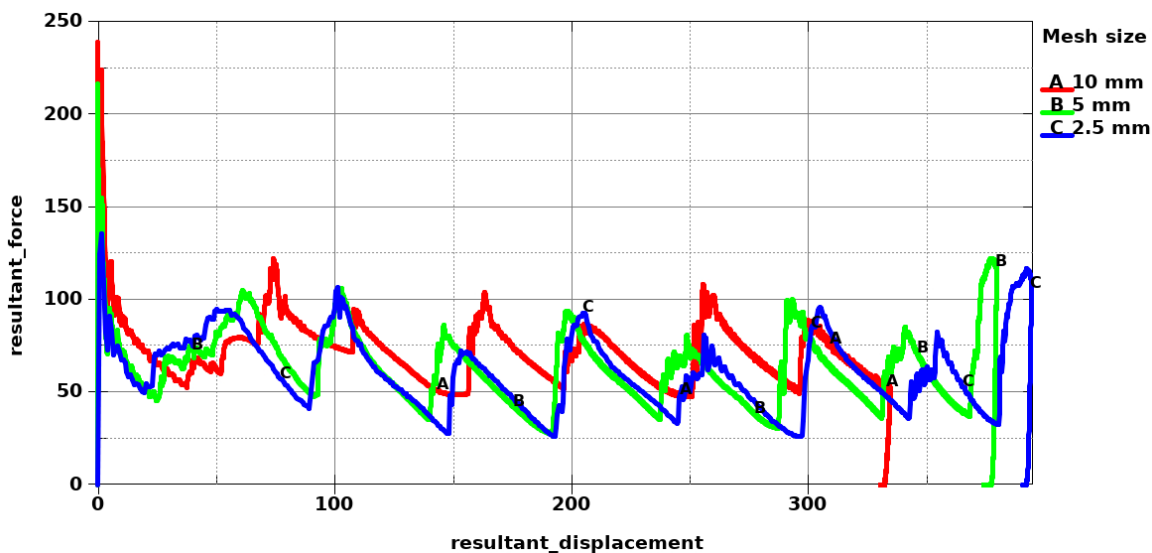
The rear end of the crash box is fixed with `*BOUNDARY_SPC_SET`.

Initial conditions

The initial velocity of the impactors is applied with `*INITIAL_VELOCITY_RIGID_BODY`.

Results

By specifying the same output frequency for `RCFORC` and `NODOUT` the force versus displacement curve can easily and accurately be plotted in a post-processor. The force curves, and stop displacements, for the tubes with 2.5 and 5 mm mesh sizes are similar. Whereas the 10 mm mesh produce a significantly stiffer response.



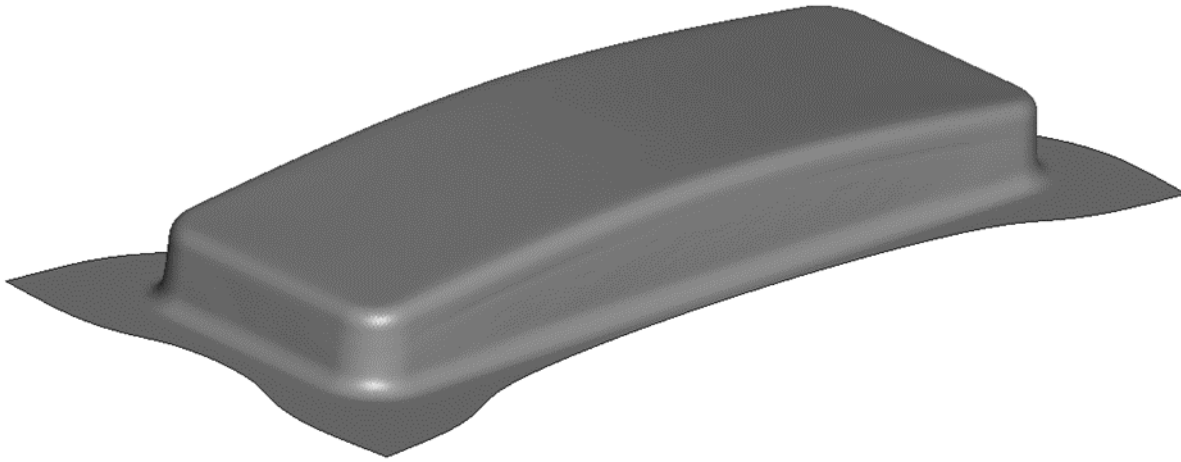
Comment: Force vs. displacement curves for the three tubes.

Example 3: Deep drawing

A rectangular shape with curved top is formed with a single action die process. This example is intended to exemplify a typical setup for a deep drawing simulation with a blank and tool definition, process boundary conditions, material model and some settings (control cards/contact) for metal forming simulations.

The model uses *PARAMETER and *PARAMETER_EXPRESSION to define the process. The user needs to define the parts ID:s for the blank and tools, distances between the tools, blank thickness, tool velocities and binder force. The load curves defining the motions of the tools and the end time are calculated from the user input.

Note: The idea of the provided example model is to show ideas/functionality in LS-DYNA. It is not intended to demonstrate current state-of-the-art modelling technique. However, it can be used /modified by the reader to test their ideas and understanding of different functionality in LS-DYNA.



Simulation data

# nodes	11k (initial)
# elements	11k (initial)
Timestep size (ms)	0.54e-3 (constant)
Termination time (ms)	8.8

Elements and material

The blank and tools are modelled with shell elements. The tools are rigid and meshed with deviation mesh mode which allows the elements to better follow the tool geometry. The blank is meshed with an initial element length of 4 mm. Adaptive meshing is applied to the blank which enables the elements to be refined in two steps. A stiffness based hourglass control, type 4, is applied (QH=0.1).

MAT_BARLAT_YLD2000 is used for the blank with material parameters defining the anisotropic behavior of the material. The hardening is defined with a load curve and the material direction is defined by the AOPT flag.

Contact definitions

*FORMING_ONE_WAY_SURFACE_TO_SURFACE is used between the blank and all the tools. This contact type is recommended for sheet metal forming simulations since it has some useful properties, e.g. the tool thickness is not considered and it can be used with look-ahead adaptivity. Note that this is a non-symmetric contact type and the blank should be set as tracked (SURFA) surface. The contact is also sensitive for orientation of the mesh and the normal of the tool elements should be directed toward the blank.

Control cards

The mesh adaptivity of the blank is defined in *CONTROL_ADAPTIVE. Here the adaptivity frequency, tolerance, refinement levels etc. are defined. ADPOPT is set to 1 for the blank in PART to activate the adaptivity for the part. In *CONTROL_CONTACT it is recommended to set PENOPT=4 for sheet metal forming simulations. Thereby the stiffness in the contact will be based on the blank. Shell thickness changes due to membrane stretching is one of the main results from this type of simulation, so the ISTUPD in *CONTROL_SHELL must be set to 1 or 4.

In deep drawing simulations, when the blank is clamped between the die and binder, the dynamic effects are usually low. Therefore, this type of simulations can be mass scaled to quite a high ratio. The timestep used for this simulation is $6e-4$ ms, see DT2MS in *CONTROL_TIMESTEP, which increases the blank mass by approximately 1050 %.

Loads and constraints

The motion of the die is defined by a trapezoidal load curve with a ramping time of 1 ms. The forming velocity is prescribed with *BOUNDARY_PRESCRIBED_MOTION_RIGID. The binder has a load applied to it using *LOAD_RIGID_BODY. The binder also has a vertical constraint applied to it during the forming by *CONSTRAINED_RIGID_BODY_STOPPERS. The punch is fixed during the whole process.

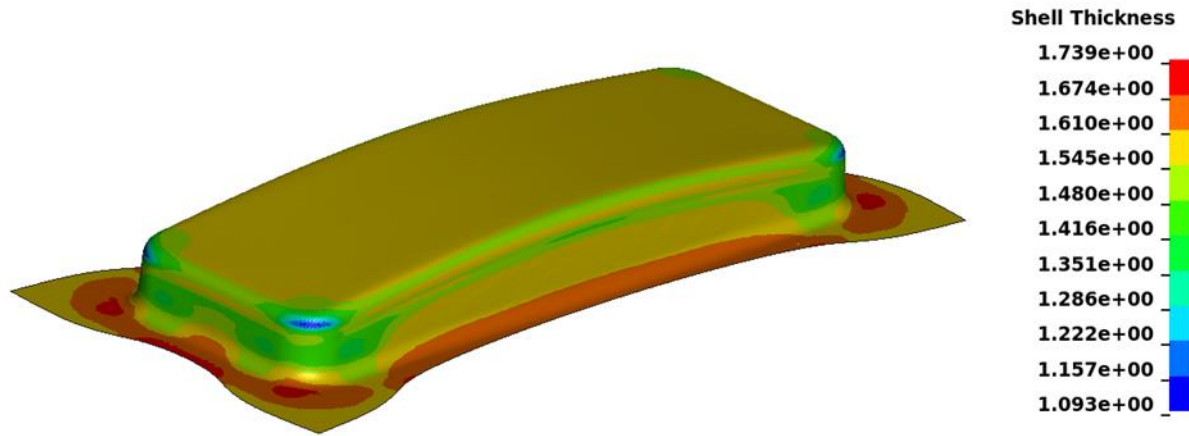
Output

Sheet metal forming simulations are typically evaluated based on output from the ASCII databases BNDOUT, GLSTAT, MATSUM, RCFORC and SLEOUT. In *DATABASE_EXTENT_BINARY, the STRFLG option must be set to 1 to get the strain data. The strain output is needed when doing post-processing with a Forming Limit Diagram (FLD).

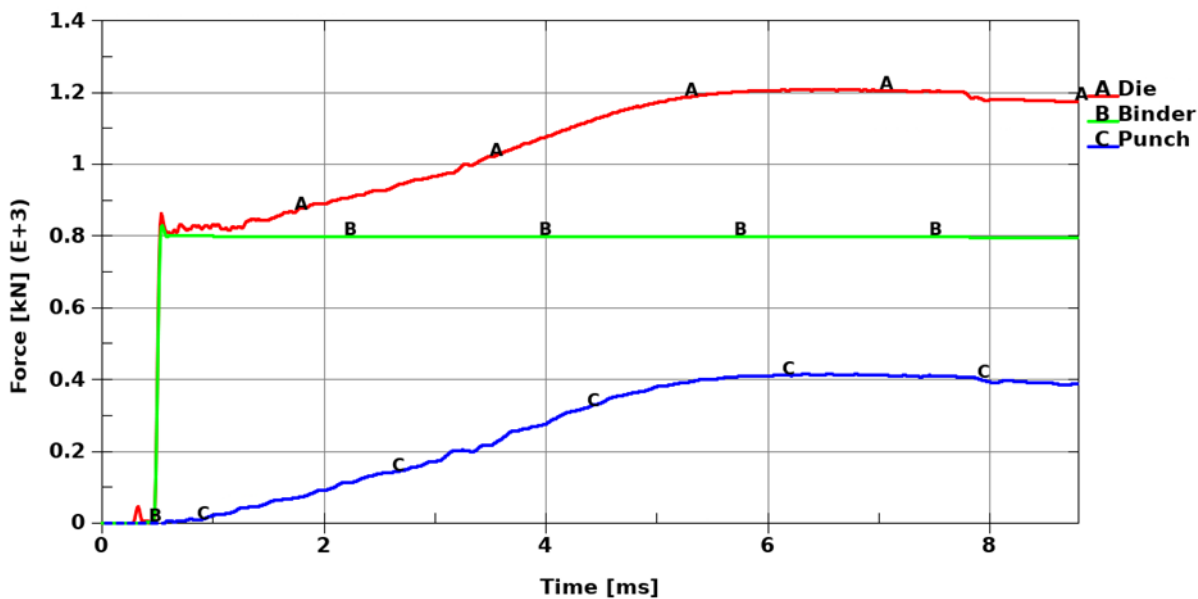
*INTERFACE_SPRINGBACK_LSDYNA is used to extract a dynain file at the end of the simulation containing thickness, strain, and stress in the blank.

Results

There are no hot spots of plastic strain in the blank that could indicate tool geometry errors. The contact forces show some oscillations due to the mesh adaptivity. Apart from that, the curves look smooth.



Comment: Thickness distribution in the blank at the end of the forming process.



Comment: Tool forces during the forming simulation.

Springback

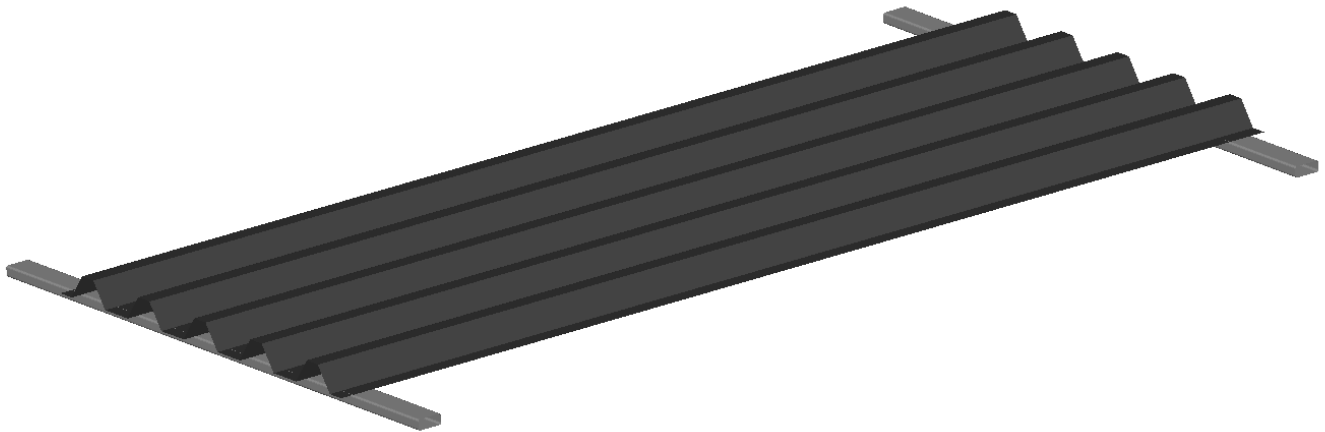
A springback analysis is performed to calculate the resultant deformation of the blank when it has been removed from the forming tools. The analysis is performed using static implicit. The element formulation is switched from ELFORM 2 to ELFORM 16. The number of integration points (NIP) is increased from five to seven to improve the accuracy³. Some constraint points need to be added to prevent the rigid body motions. When adaptivity is used for the blank, it is convenient to use *CONSTRAINED_COORDINATE rather than SPCs as this keyword is not dependant on the node numbering. Note that the implicit springback simulation require double precision LS-DYNA.

³ The stresses, through the thickness, are automatically handled in LS-DYNA.

Example 4: Post-buckling strength

This example is intended to exemplify a quasi-static buckling analysis using LS-DYNA. The target of this simulation is not only to find the ultimate strength of the profile but also to find the residual post-buckling strength. This example includes a test rig with an applied uniform pressure load. The concept of a PI-regulator as well as how to apply geometry perturbations are also demonstrated. The use of selective mass scaling is briefly discussed.

Note: The idea of the provided example model is to show ideas/functionality in LS-DYNA. It is not intended to demonstrate current state-of-the-art modelling technique. However, it can be used /modified by the reader to test their ideas and understanding of different functionality in LS-DYNA.



Simulation data

# nodes	136k
# elements	136k
Timestep size (ms)	0.664e-3 (constant)
Termination time (ms)	150.0

Elements and material

The steel profile (750x45x2000 mm, thickness 0.5 mm) is modelled with shell elements ELFORM -16, which is a fully integrated element type. Hourglass control type 8 is applied. A uniform mesh with element size 4-5 mm is used. Element thickness perturbation (± 0.003 mm) is introduced with *PERTURBATION_SHELL_THICKNESS. The material model used for the steel is *MAT_024. Strain-rate effects are not considered since the event is supposed to be quasi-static.

Contact definitions

The contact between the profile and the supports is handled by an *CONTACT_AUTOMATIC_SINGLE_SURFACE contact with SOFT=1.

Control cards

By adding *DATABASE_CURVOUT, the curve produced by *DEFINE_CURVE_FUNCTION is output. This curve is used as the steer signal for the pressure as applied by the PI-controller.

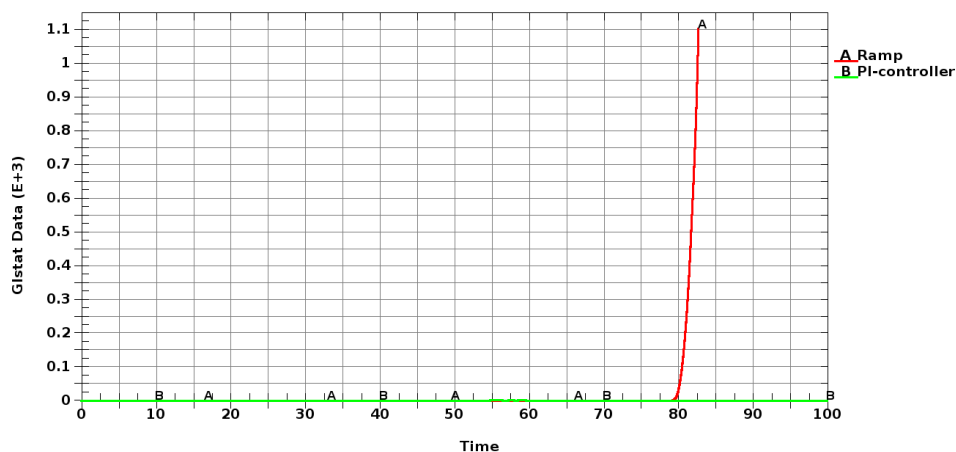
Loads and constraints

Boundary symmetry conditions are set in the transverse direction by *BOUNDARY_SPC_SET. Gravity is accounted for by *LOAD_BODY_Z. The profile is loaded with a uniform pressure. The pressure can be applied in different ways where a relatively slow ramp load is the most straight forward approach. This works fine until the ultimate strength is reached. After that the kinetic energy will increase rapidly as the profile collapses due to buckling. A ramp load can therefore not be used to evaluate the post buckling strength of this profile. For this reason, a PI-controller is used as defined by *DEFINE_CURVE_FUNCTION with function PIDCTL. The controller is set to keep a steady load velocity of 0.4 m/s by controlling the applied pressure. NID 108355 is used by the PI-regulator to monitor the velocity of the applied load. This node is the dependent node in a *CONSTRAINED_INTERPOLATION definition, which follows the motion of the profile. By this arrangement a more robust measurement of the “average” cross section z-velocity is achieved.

Results

Adjusting the PI controller parameters is a bit tricky. In this case the initial settings of parameters, i.e., the proportional gain k_p and the integral gain k_i , gave a relatively fast prediction of ultimate load as well as post-buckling strength. However, the elastic response before buckling was oscillating a lot. This was improved by decreasing k_p and increasing k_i . Even slower loading rates would probably improve the accuracy. However, that would also increase the cost.

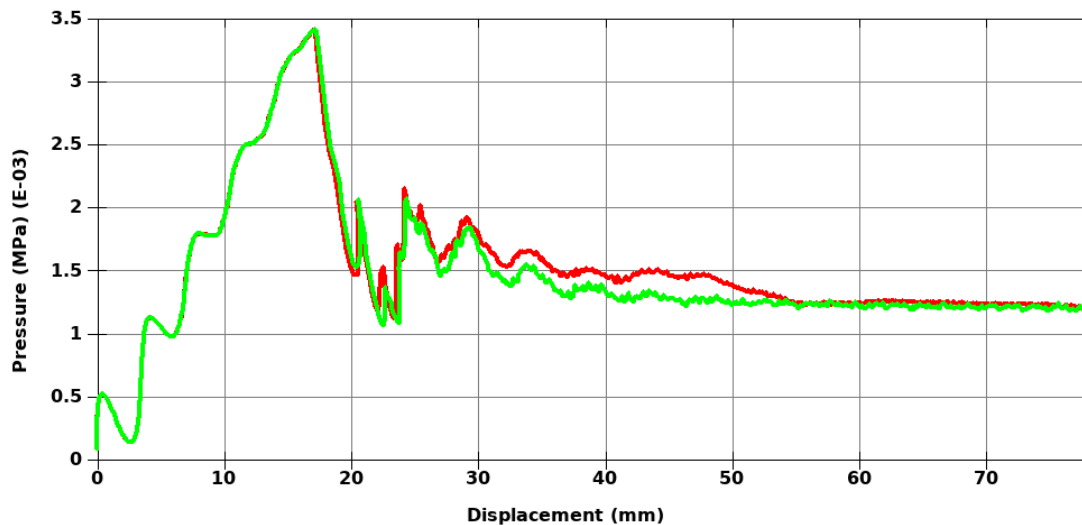
The kinetic energy is held to a low level throughout the simulation. The peak kinetic energy was about 7% of the total energy just after buckling and then rapidly decreased to a level <1%. The figure below shows the kinetic energy in the simulations with a ramp load compared to a PI-controller. It is evident that a ramp cannot be used to evaluate the post-buckling strength in this case. Note that in the figure below, the time for the RAMP-curve has been scaled down a factor 10 for a clearer comparison.



Comment: Kinetic energy with a ramp load (red curve) and with a PI-controller (green curve).

By scaling CURVOUT by a factor of $1.0\text{e-}2$ the applied pressure (MPa) can be plotted. BNDOUT gives the reaction forces in the supports, which can be compared to the applied pressure. In addition, the z-displacement can be plotted from NODOUT.

It was possible to decrease the simulation time about 40% by using Selective Mass Scaling (SMS). SMS can be an efficient way to speed up a simulation. Though, care must be taken so that the mass scaling does not significantly affect the simulation results. To activate selective mass scaling set `IMSCL=1`, or preferably refer to a `*SET_PART`, on `*CONTROL_TIMESTEP`. `DT2MS=-1.4756e-3` was found to give a good speed-up without compromising the results too much (x2 the timestep size).



Comment: Pressure vs. displacement. Conventional mass scaling (red curve) and selective mass scaling (green curve).

Example 5: Bolted connection

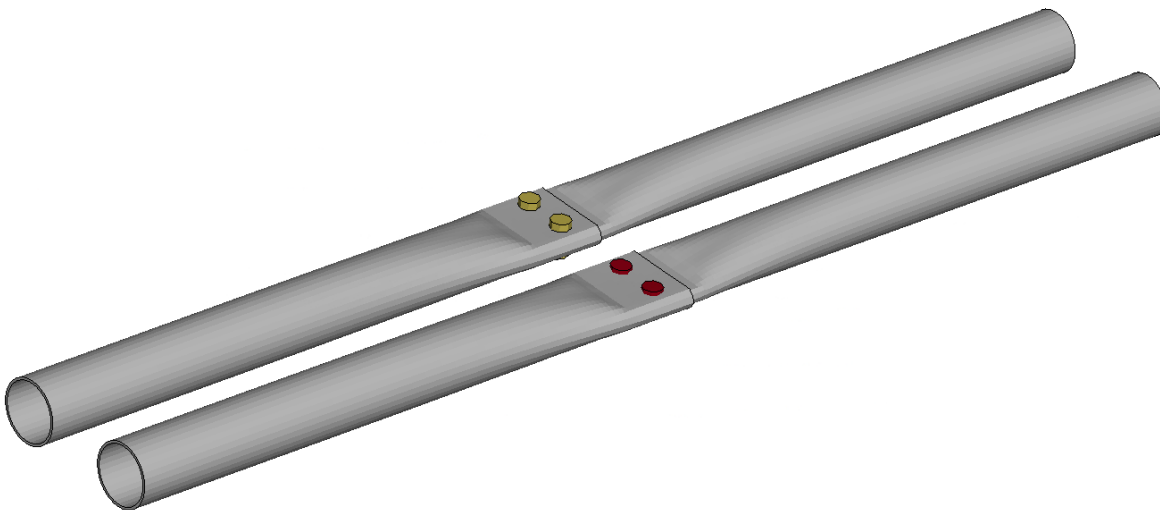
Bolted connections can be modelled in many ways depending on the purpose. This connection include friction between bolt head/washer and underlying surface, and it includes pretension. It allows for:

- slip due to play between the bolt shank and the hole edge.
- tear-out of the hole.
- the bolt to deform/bend to some degree.
- the bolt to break.

The bolt shank is, in this example, modelled in two ways, which is with beam elements and with solid elements, respectively.

This example is intended to exemplify the use of `*INITIAL_AXIAL_FORCE_BEAM`, `*INITIAL_STRESS_SECTION`, `*CONTACT_AUTOMATIC_GENERAL` and `*CONTROL_DYNAMIC_RELAXATION`.

Note: The idea of the provided example model is to show ideas/functionality in LS-DYNA. It is not intended to demonstrate current state-of-the-art modelling technique. However, it can be used /modified by the reader to test their ideas and understanding of different functionality in LS-DYNA.



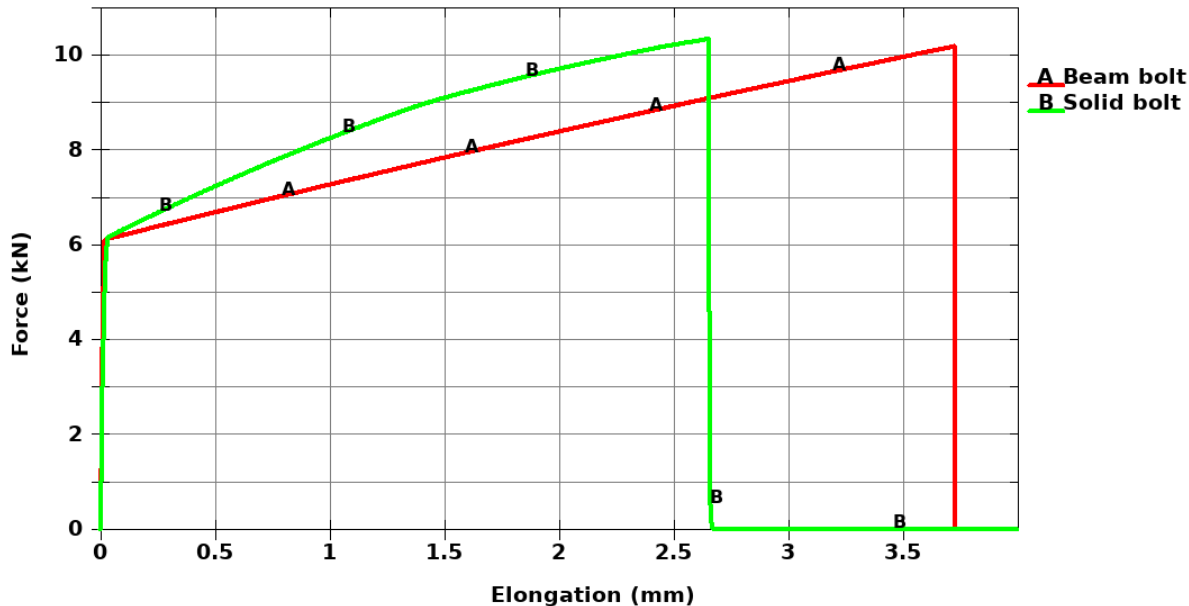
Simulation data

# nodes	19k
# elements	18k
Timestep size (ms)	0.4e-3 (constant)
Termination time (ms)	40.0

Elements and material

The bolts in the first connection are modelled with beam elements (ELFORM 1) for the shank and shell elements ELFORM -16 for the bolt head/washer. The bolts in the second connection are modelled completely with solid elements ELFORM 1. Hourglass control type 6 and type 8 is applied to the solids and shells, respectively. The bolt shank cross section area is the same for both types of connections and

both are discretized with 3 elements along the length. The same material model, *MAT_024 with a bilinear plastic hardening curve, is used for all bolts. The yield strength is 240 MPa and the ultimate strength is 400 MPa. The elements are set to fail at 20% effective plastic strain. *MAT_ADD_EROSION was added with NUMFIP=1.0 to gain better control of the failure settings. In this way the beam element is deleted when the first integration point fails. As shown in the figure below, a simulated tension test (not included) gave differences in results.



Comment: Tension test results for the beam bolt and the solid bolt, respectively.

Contact definitions

*CONTACT_ERODING_SINGLE_SURFACE with SOFT=2 is used for shells and solids.

*CONTACT_AUTOMATIC_GENERAL is used between the beam bolt shank and the null beams added around the hole edges.

Control cards

DRCPSID on *CONTROL_SHELL activates a drilling rotation constraint for shell elements. Without this the beams of the bolt shank would be unconstrained in torsion/drilling. *DATABASE_EXTENT_BINARY, the parameter BEAMIP=4 is set to obtain data for all integration points of the beam.

Prestress of bolts

The bolts are prestressed during dynamic relaxation, which is activated by SIDR=1 on any curve.

*INITIAL_AXIAL_FORCE_BEAM and *INITIAL_STRESS_SECTION with ramped load curves is used to prestress the beams and solids, respectively. IZSHEAR=2 on *INITIAL_STRESS_SECTION to allow bending/shear stresses to develop during the prestress of the solid bolts. Observe that there is a similar option for pretension of beam elements, see KBEND=1 (KBEND not used in this example).

*CONTROL_DYNAMIC_REALAXTION, DRTERM=10.0 will limit the relaxation phase to a maximum of 10ms. Since DRTOL is set to a very low number (1.e-10), 10.0ms for the prestress is expected. The

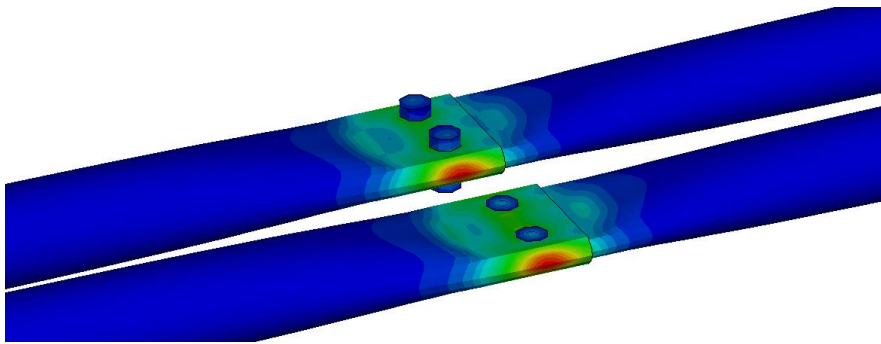
prestress process can be studied in detail through D3DRLF (*DATABASE_BINARY_D3DRLF) and the convergence from ASCII output file RELAX. The end of the stress initialization is confirmed in d3hsp. The resulting prestress load level can be checked in ELOUT and/or SECFORC.

Loads and constraints

The tubes are fully constrained at one end and pulled in the other end using rigid grips in combination with *BOUNDARY_PRESCRIBED_MOTION_RIGID.

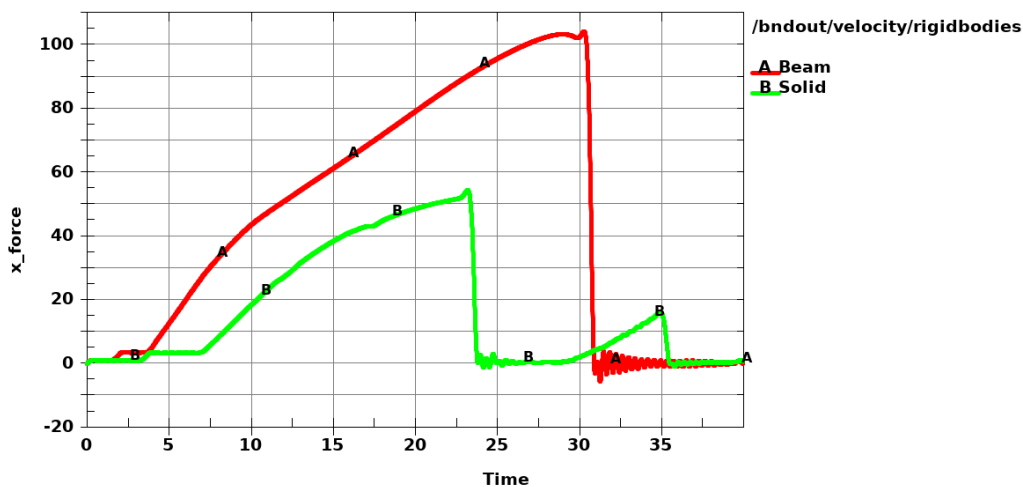
Results

A fringe plot of the prestressed connections show that the resulting stress distributions are similar, see figure below. Studying stress and force levels in ELOUT and SECFORC confirms that the desired prestress level (3 kN, 120 MPa) has been achieved.



Comment: Prestressed connections.

The ultimate load of the beam element connection is about a factor 2 larger than for the solid bolt variant in this example. The reasons for this can be multiple. For complex loading situations such as this, calibration of the modelling at hand is needed. Especially in situations when a very detailed model may not be convenient. Note that the force levels in the tubes can be attained from the cross-section definitions in the model.



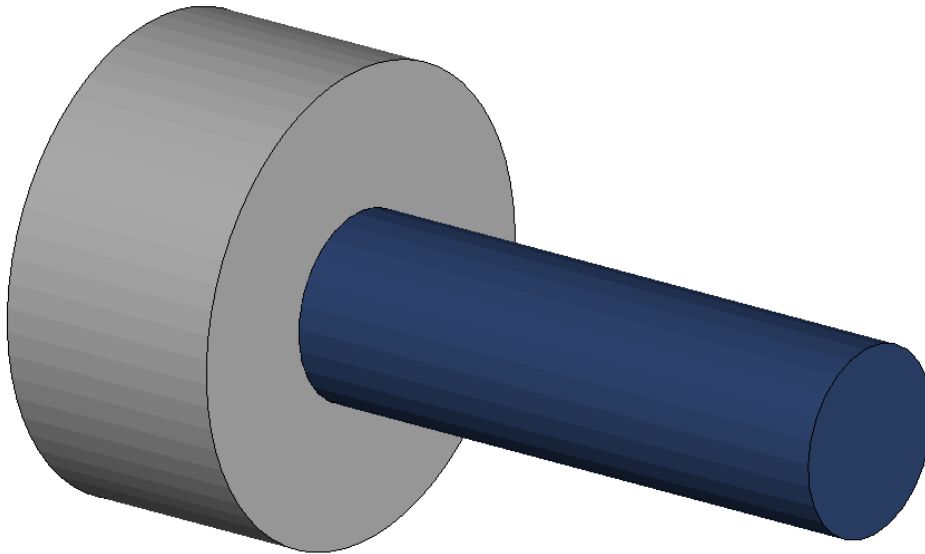
Comment: Ultimate load (x-force) for the loading case where the connections are pulled apart. BW 1000Hz filter applied. This can be cross-checked from SECFORC output.

Example 6: Interference fit

This example consists of the inner shaft and the outer hub. The hub is thought to have been heated and is set to be cooled 250°C in the simulation, thereby introducing the interference pressure. The applied temperature corresponds to an allowance of about 0.1 mm (0.25% of the diameter). Firstly, the shaft is loaded with a prescribed rotation and, secondly, the shaft is pulled thus testing the performance of the friction fit for torque and axial load. In this case the simulation results can be compared to an analytical solution.

This example is intended to exemplify the use of `*LOAD_THERMAL_VARIABLE`, `*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH` and the `INTFOR` file.

Note: The idea of the provided example model is to show ideas/functionality in LS-DYNA. It is not intended to demonstrate current state-of-the-art modelling technique. However, it can be used /modified by the reader to test their ideas and understanding of different functionality in LS-DYNA.



Simulation data

# nodes	7k
# elements	17k
Timestep size (ms)	0.3e-3 (constant)
Termination time (ms)	100.0

Elements and material

The shaft is modelled with ELFORM 1 hexahedron (Hourglass control type 6) and ELFORM 15 pentahedron elements (through automatic sorting). The hub is modelled with ELFORM 13 tetrahedron elements. Both the shaft and the hub are made of steel, `*MAT_024`, where the yield stress is set high to aim at only elastic deformation. The thermal property of the hub is applied with `*MAT_ADD_THERMAL_EXPANSION`.

Contact definitions

The contact between shaft and hub is treated by

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH. The SMOOTH option is necessary here to transfer the torque more correctly. The faceted surface that would be the case for a non-SMOOTH contact is replaced with a fitted smooth approximation based on the mesh. The interface force file is requested by setting SAPR=SBPR=1 on contact card 1 in combination with using S=iff on the execution command line. The output frequency is set with *DATABASE_BINARY_INTFOR. The static and dynamic friction coefficient is 0.1 in the model.

Control cards

*CONTROL_DYNAMIC_RELAXATION is defined to apply the temperature load during the DR-phase.

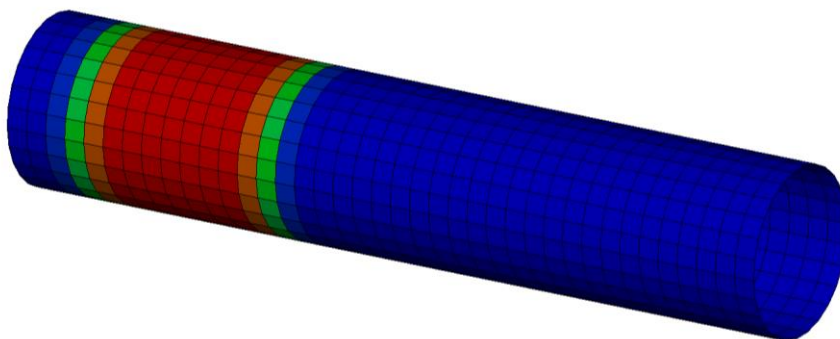
Loads and constraints

The outer surface of the hub is fixed through *CONSTRAINED_INTERPOLATION, which allows only radial movement. The beam element is added for the purpose of fixing the dependent node. This boundary condition allows for comparing the results with a known analytical solution. The loading, which are x-moment and x-force, is applied to the rigid end of the shaft with

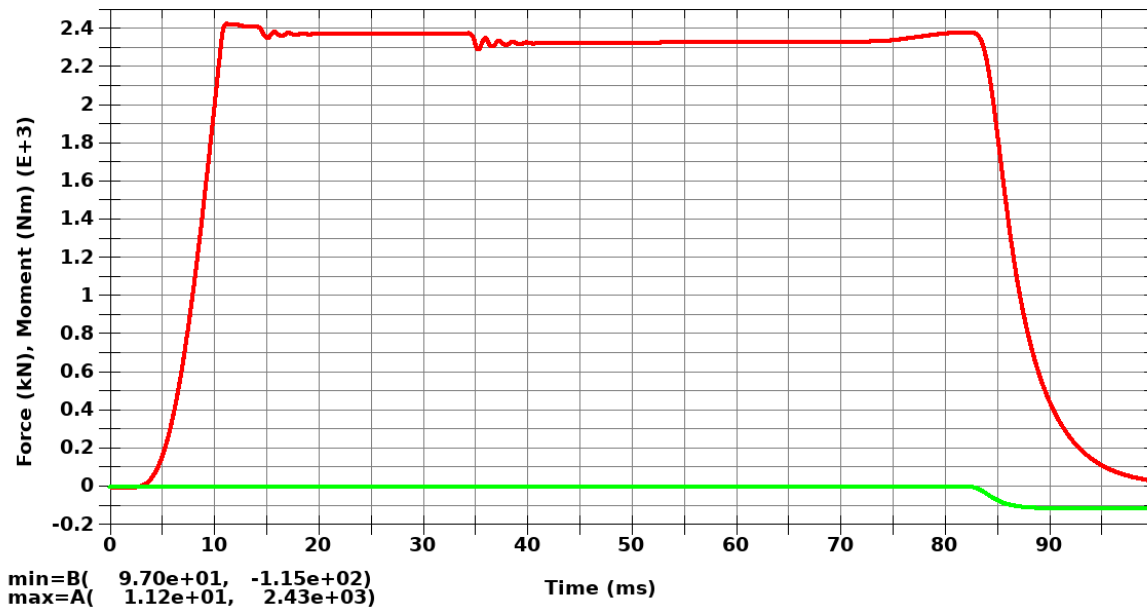
*BOUNDARY_PRESCRIBED_MOTION_RIGID.

Results

The interference pressure can be fringe plotted from the interface force file. The INTFOR file can be requested on the command line using s=<INTFOR>. The axial friction force and the transmission torque can be plotted from BNDOUT. The measured allowance, which is after compensation for the contact penetration, is about 0.088 mm. Analytically this would yield an interference pressure of 194 MPa. The maximum/ultimate analytical transmission torque is 2438 Nm. The analytical transmission axial force is 121.9 kN. The simulation is within 5% of these values. Note that the analytical estimate is sensitive to the input allowance. Hence, the accuracy in the comparison is estimated to be within a few percent.



Comment: Interference pressure. Red colour is range 190-200 MPa.

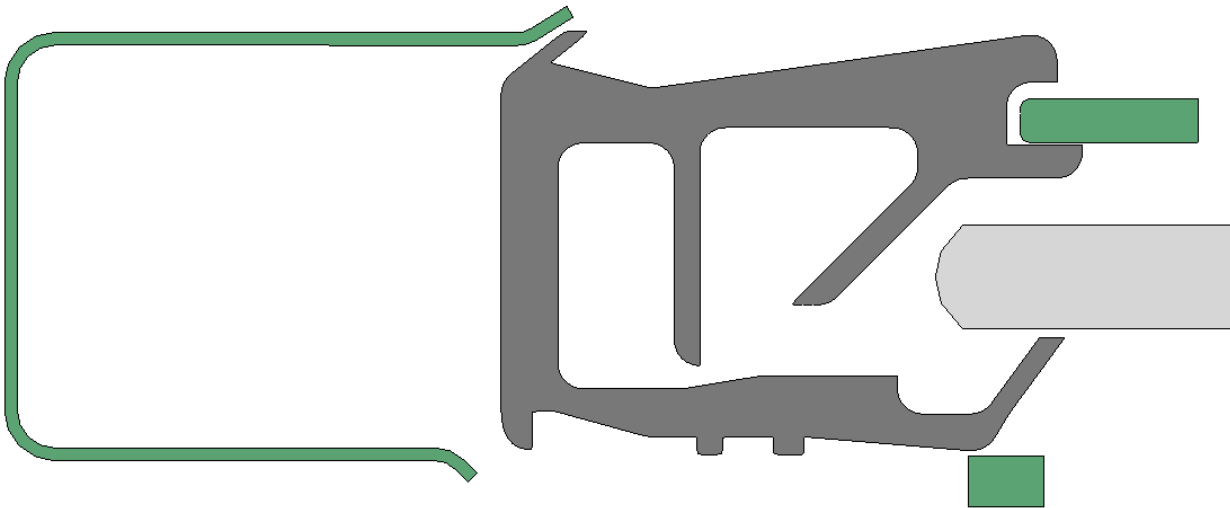


Comment: Transmission torque (red curve) and transmission axial force (green curve).
BW 1000Hz filter applied.

Example 7: Rubber sealing

This example is intended to exemplify the modelling of a rubber sealing. In this case a sealing between the door frame and the side window of a car. The purpose of this 3D quasi-static simulation is to test the ability of the sealing to prevent water and air leakage. The model consists of the deformable rubber sealing as well as a rigid door frame and a window. This example is intended to exemplify how to improve robustness by, in this case, decreasing TSSFAC, adding energy dissipations and using an appropriate contact definition (MORTAR in this case).

Note: The idea of the provided example model is to show ideas/functionality in LS-DYNA. It is not intended to demonstrate current state-of-the-art modelling technique. However, it can be used /modified by the reader to test their ideas and understanding of different functionality in LS-DYNA.

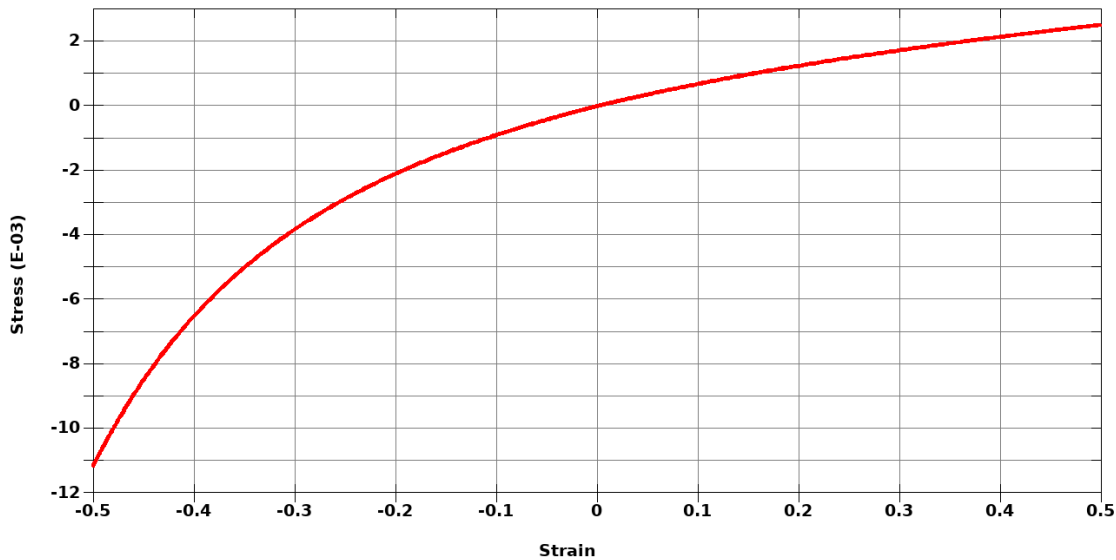


Simulation data

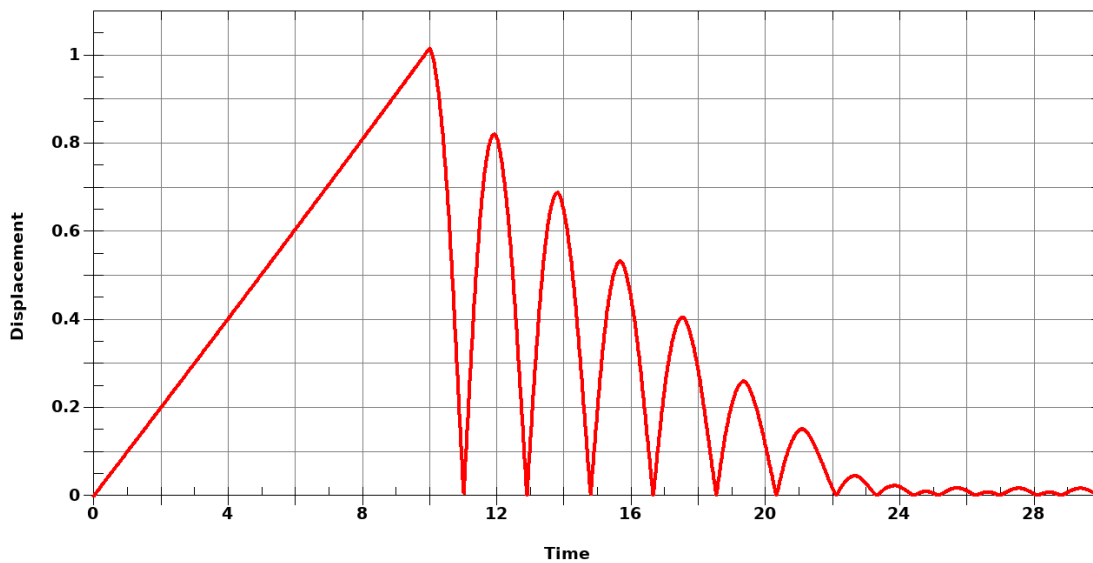
# nodes	30k
# elements	23k
Timestep size (ms)	0.311e-3 (constant)
Termination time (ms)	540.0

Elements and material

The rubber is modelled with the solid hexahedron (mostly) ELFORM 1 elements in combination with hourglass formulation IHQ=6 and QM=1.0. The material model is *MAT_077 with a poisson's ratio of 0.495. The hyperelastic constants (C10, C01) are set directly. Frequency independent damping is accounted for by G and SIGF. It was needed to decrease TSSFAC from default 0.9 to 0.7 to achieve numerical stability of the material.



Comment: The stress-strain curve was produced through axial tension and compression of a one-element model.



Comment: The damping curve was produced by simulating the damped oscillations of a solid element beam.

Contact definitions

The contact is treated by `*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR`. The only parameters that are changed from defaults are the friction behaviour and the contact damping (`VDC=20`). A `*DEFINE_FRICTION` table is referenced by setting `FS=-2`.

Control cards

ISYM on *CONTROL_CONTACT references to a node set on the symmetry plane. The MORTAR contact picks this up and uses it in the contact algorithm.

Loads and constraints

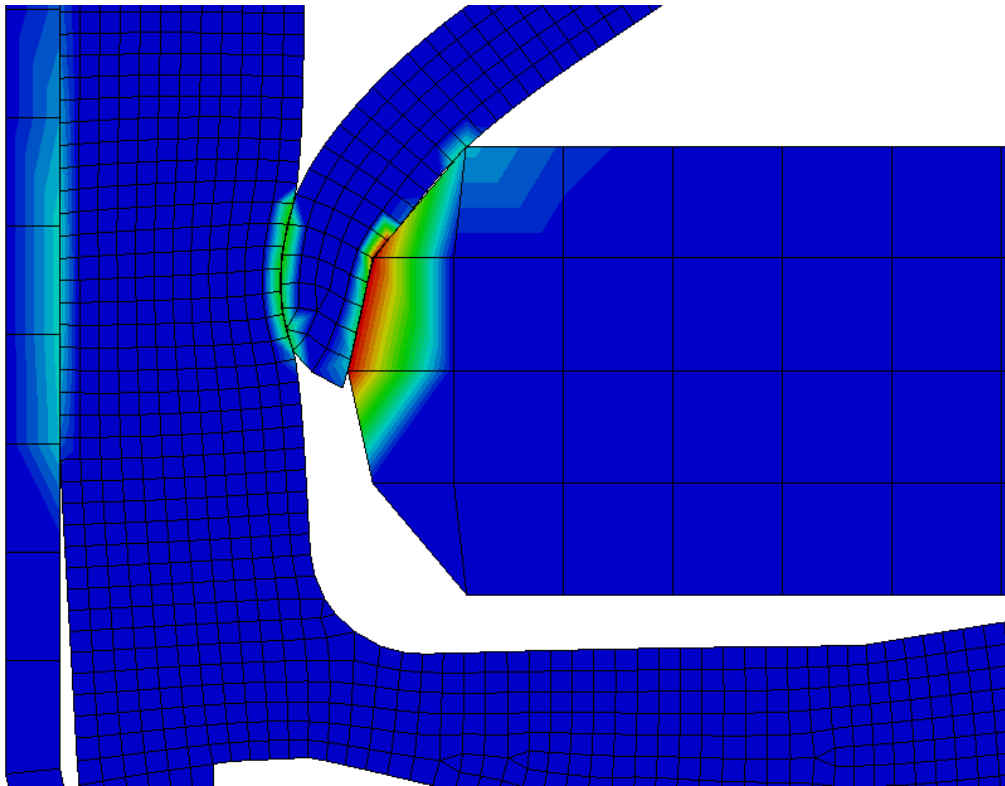
The movements of the rigid parts, which are defined by smooth velocity curves, are controlled by *BOUNDARY_PRESCRIBED_MOTION_RIGID. The symmetry constraint (in z-direction) is defined with *BOUNDARY_SPC_SET.

Output

*CONTROL_OUTPUT, PENOUT=1 for output of contact (absolute) penetrations into D3PLOT. This feature is handy when evaluating if contact is established or not. The maximum (absolute) penetration is also output to SLEOUT.

Results

The contact performs well, which is crucial when evaluating potential leakage.



Comment: Visualization of contact penetration in the MORTAR-contact.

Note: For demonstrative purposes, this simulation is performed in 3D. However, this example can with advantage be run in 2D. A test simulation in 2D (not documented in this document) was 10 times faster than the 3D simulation.

Example 8: Vehicle initialization for crash

In this chapter a procedure of how to initialize a vehicle for crash analysis is shown. This example is intended to exemplify a way to find equilibrium before the actual event happens. Furthermore, how to apply initial velocity (including rotational velocity of the wheels) using

*INITIAL_VELOCITY_GENERATION[_START_TIME] and the use of *PART_INERTIA.

Note: The idea of the provided example model is to show ideas/functionality in LS-DYNA. It is not intended to demonstrate current state-of-the-art modelling technique. However, it can be used /modified by the reader to test their ideas and understanding of different functionality in LS-DYNA.



Simulation data

# nodes	20k
# elements	18k
Timestep size (ms)	1.0e-3 (constant)
Termination time (ms)	200.0

Barrier model

The original barrier model was developed by Livermore Software Technology Corporation (LSTC)⁴. The model has been modified by the authors to fit the purpose of this example. The mass and inertia properties of the carriage are defined through *PART_INERTIA. The wheels are inflated by airbags using *AIRBAG_SIMPLE_PRESSURE_VOLUME. The barrier has been positioned, in the vertical direction, where it should be when static equilibrium is attained. This means that the tires are initially sunk below ground level, see the figure below. Note that for a general suspension model the springs are assumed to be prestressed, e.g., with initial offset, according to the vehicle weight acting upon them.

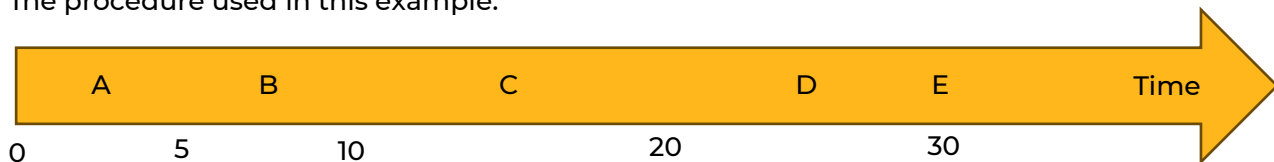
⁴ D. Bhalsod, R. Chivukula, H. Devaraj; “214 Shell Barrier”, LSTC, 2018.



Comment: The initial position of the vehicle/tire showing ground level and floor plate.

Vehicle initialization procedure

The procedure used in this example:



- A. The tires are inflated by `*AIRBAG_SIMPLE_PRESSURE_VOLUME`.
- B. The rigid floor is moved to ground level (pushing the tires) with `*BOUNDARY_PRESCRIBED_MOTION_RIGID`. Gravity is ramped up using `*LOAD_BODY_Z`.
- C. The contact between the floor plate and the tire is replaced by a `*RIGIDWALL_PLANAR`.
- D. Initial velocity is applied using `*INITIAL_VELOCITY_GENERATION[START_TIME]`. Note that `IRIGID=1` to overrule any velocities set on `*PART_INERTIA`.

Mass weighted damping is applied to the tires during the complete initialization and ramped down to zero before the event. Note that no damping is applied in the airbag definition (MWD). Such airbag damping would potentially prevent the wheels from rotating correctly.

Preparations

The preparations needed when setting up a model as shown in this procedure is

1. to determine the damping coefficient of `*DAMPING_PART_MASS_SET` for the tires. For example, by inflating a free wheel and measure the period of the tire oscillations.
2. to determine the vertical position of the barrier when in static equilibrium and thereby determine the z-direction motion of the rigid plate to reach ground level. For example, by leaving the vehicle on ground under the influence of gravity and measure the motion of the vehicle in z-direction.

If a wheel suspension is present, then the suspension springs must be prestressed accordingly to achieve static equilibrium. Note that the vertical motion of the rigid floor described earlier is mainly to initiate the deformation of the tires, i.e., not to prestress the suspension springs.

Contact definitions

The contact between tires and ground is treated by *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE with SOFT=1. The contact is replaced by *RIGIDWALL_PLANAR using the BIRTH/DEATH functionality.

Control cards

No special cards added.

Loads and constraints

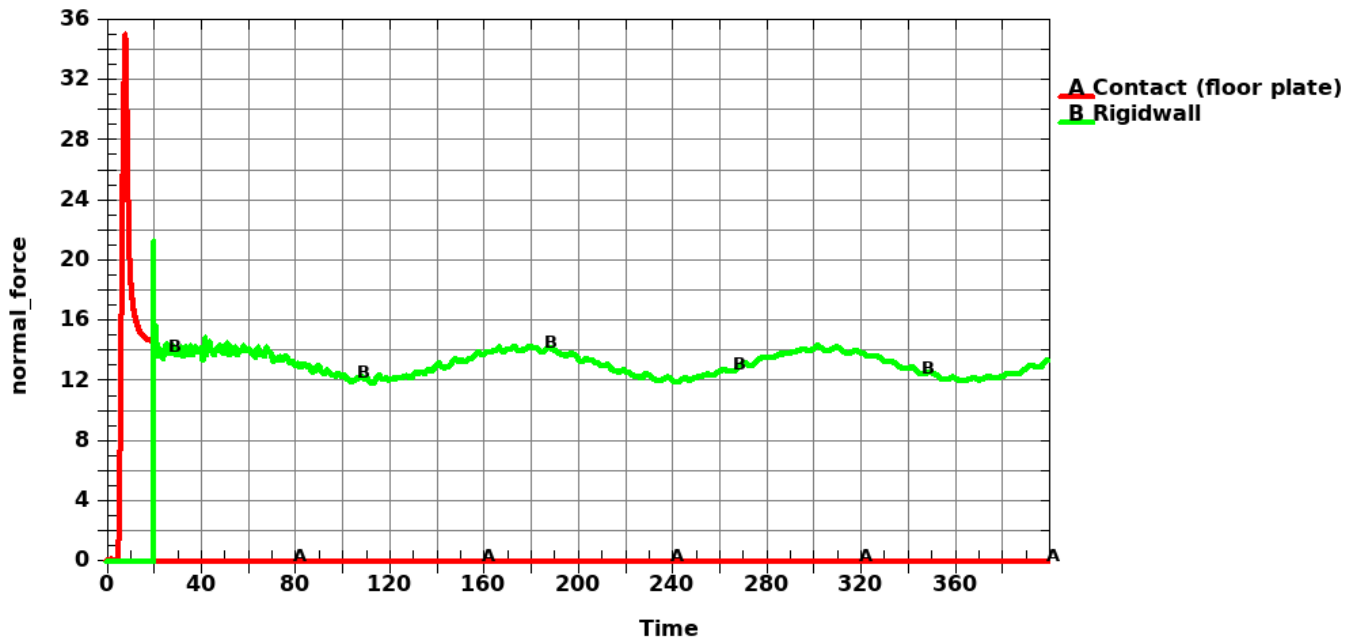
*LOAD_BODY_Z, *BOUNDARY_PRESCRIBED_MOTION_RIGID and *INITIAL_VELOCITY_GENERATION as described earlier. The use of *PARAMATER and *PARAMETER_EXPRESSION makes it easy to change the velocity.

Output

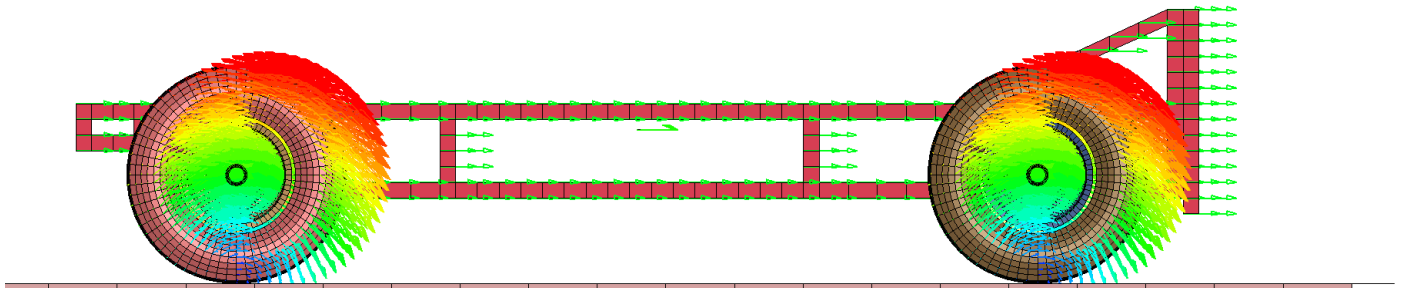
*DATABASE_ABSTAT has been added for monitoring tire pressure.

Results

A “steady state” analysis showed that the initiated carriage drops <1 mm under the influence of gravity. The initial velocity, applied after the initialization procedure, may be studied by a vector plot from D3PLOT. The kinetic energy in GLSTAT can be checked analytically.



Comment: A “steady state” analysis (no velocity applied) to check static equilibrium. A switch of any contact or rigid wall will introduce noise, see spike in green curve.



Comment: Initial velocity applied.

There are alternatives to this method, for example:

1. Run the preload in pseudo time using `*CONTROL_DYNAMIC_RELAXATION`.
A preferred choice by many users is to make most/all preparations in the DR-phase.
2. Run the preload using the implicit solver in LS-DYNA, in transient or pseudo time.

All methods mentioned above is performed in one LS-DYNA simulation. Another approach would be to find the equilibrium state in a first simulation and then start a secondary simulation, of the actual event, from this state.

Record of revisions

Rev. no	Release date	Author
1.0	2019-03-21	Klas Engstrand Anders Bernhardsson
1.1	2019-10-21	Klas Engstrand
1.2	2021-08-20	Klas Engstrand David Aspenberg
1.3	2023-02-01	Klas Engstrand
1.4	2023-02-15	Klas Engstrand
1.5	2024-03-27	Klas Engstrand Jimmy Forsberg
1.6	2025-06-17	Klas Engstrand Jimmy Forsberg

Copyright and Trademark Notice

All brands, trademarks and images mentioned are property of ANSYS, Inc.

Keywords: Ansys LS-DYNA; explicit;

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
www.ansys.com

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks of Ansys, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2025 ANSYS, Inc. All rights Reserved.