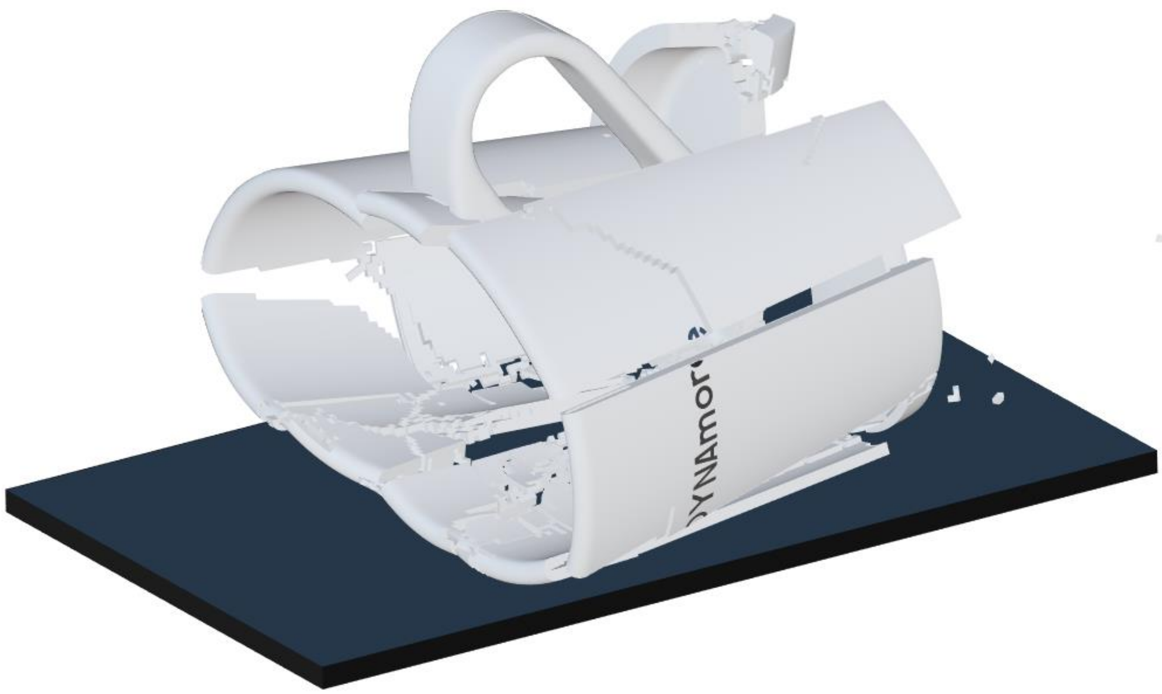
Technical Guide

Technical Guide for Explicit Analyses using Ansys LS-DYNA



2024-03-27 v1.5

Table of content

Abstract

The purpose of this document is to provide a hands-on technical guide, by providing LS-DYNA models of typical applications, on how to make well-conditioned 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 building similar models for simulation using the explicit solver in Ansys LS-DYNA.

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

Disclaimer

/ Technical Guide

Introduction

This document includes examples of typical applications for simulation using the explicit solver in Ansys LS-DYNA. The document is to be used in conjunction with running/studying the accompanied FE-models. The intention of the authors is to provide a hands-on technical guide, by providing LS-DYNA models of typical applications, on how to make well-conditioned LS-DYNA models for explicit simulations. The modeling is based on the experience from the authors and an effort has been made to use generally accepted best methods and settings. The idea is that the methods and settings described in this document can serve as a good starting point when building similar models for explicit simulation.

The models have been built and post processed using Ansys LS-PrePost. The examples are developed with MPP/ LS-DYNA in mind and tested with version R14.1 in single precision. To provide a relevant measurement of how computationally demanding each example is, the total elapsed time of each simulation/example is noted. For this purpose, all examples have been benchmarked on 4 cores on a Linux machine equipped with dual Xeon SP 6148 CPU`s (launch date Q3'17). Output files such as D3PLOT and BINOUT can be studied in any suitable LS-DYNA post-processor, such as for instance LS-PrePost.

Support and training

For thorough details regarding LS-DYNA keywords and material models, the reader is referred to the LS-DYNA User's Manuals[1]. For help with using Ansys LS-DYNA and Ansys LS-PrePost contact your local Ansys LS-DYNA distributor. Useful web resources include www.dynasupport.com for general support and www.dynalook.com for conference papers. Example keyword files can be found at www.dynaexamples.com. Basically, the same information can be found at lsdyna.ansys.com under Resources and Knowledge base. If you find errors in this document, you are welcome to contact support@dynamore.se.

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

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.

Control cards

Most of the control parameters are left to their default settings. The main approach from the authors have 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.k.

The control settings used in this technical guide are shown in the cards below. The parameters that differ from defaults are marked with a blue rectangle. Dashed rectangles indicate additional parameters that are discussed. The parameter settings may differ slightly in some examples to comply with the specific problem at hand.

*CONTROL_ACCURACY

1	OSU	INN	PIDOSU	IACC	EXACC
	1	4	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, read the manual for details.

*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_CONTACT

1	SLSFAC	RWPNAL	ISLCHK	SHLTHK	PENOPT	THKCHG	ORIEN	ENMASS
	0.1000000	1.0000000	2	0	1	1	1	0
2	USRSTR	USRFRC	NSBCS	INTERM	XPENE	SSTHK	ECDT	TIEDPRJ
	0	0	0	0	4.0000000	1	0	0
3	SFRIC	DFRIC	EDC	VFC	TH	TH_SF	PEN_SF	PTSCL
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	1.0000000
4	IGNORE	FRCENG	SKIPRWG	OUTSEG	SPOTSTP	SPOTDEL	SPOTHIN	
	0	1	0	0	2	1	0.0	
5	ISYM	NSEROD	RWGAPS	RWGDTH	RWKSF	ICOV	SWRADF	ITHOFF
	0	0	1	0.0	1.0000000	0	0.0	0
6	SHLEDG	PSTIFF	ITHCNT	TDCNOF	FTALL	UNUSED	SHLTRW	IGACTC
	0	0	0	0	1		0.0	0
7	IREVSPT	UNUSED	COHTIEM					
	0		0					

Active optional cards: None, Opt1, Opt12, Opt123, Opt1234, Opt12345 (selected)

*Comment: Shell thickness changes are considered in single surface contact, see THKCHG and ISTUPD in *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 (recommended). 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.*

*CONTROL_DYNAMIC_RELAXATION

Card	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	IDRFLG
1	250	1.0E-03	9.95E-01		0.0	0	4.0E-02	0

*Comment: DR is activated by SIDR≠0 on any *DEFINE_CURVE in the model. DR can be fully disabled in the simulation by IDRFLG=-999, see manual for details.*

*CONTROL_ENERGY

Card	HGEN	RWEN	SLNTEN	RYLEN	IRGEN	MATEN	DRLEN	DISEN
1	2	2	2	2	2	2	2	1

Comment: Energy dissipation calculation is switched on for most parameters.

*CONTROL_OUTPUT

Card	NPOPT	NEECHO	NREFUP	IACCOP	OPIFS	IPNINT	IKEDIT	IFLUSH
1	1	3	1	0	0.0	0	100	5000

Active optional cards: None Opt1 Opt12 Opt123 Opt1234

Card	IPRTF	IERODE	TET10S8	MSGMAX	IPCURV	GMDT	IP1DBLT	EOCS
2	0	1	2	50	0	0.0	0	0

Card	TOLEV	NEWLEG	FRFREQ	MINFO	SOLSIG	MSGFLG	CDETOL	IGEOM
3	2	0	10000	0	0	0	10.0000000	1

Card	PHSCHNG	DEMDEN	ICRFILE	SPC2BND	PENOUT	SHLSIG	HISNOUT	ENGOUT
4	0	0	0	0	0	0	1	0

Card	INSF	ISOLSF	IBSF	ISSF	MLKBAG	KINENG
5	0	0	0	0	0	0

*Comment: NPOPT and NEECHO limit the amount of data written to d3hsp and standard out. 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. Output of eroded internal and kinetic energy into the matsum file, see IERODE. Note that TET10S8=1 can be used to output full nodal connectivity for higher order elements. ICRFILE=2 to visualize (in e.g. LS-PrePost) cross sections defined by *DATABASE_CROSS_SECTION. SPC2BND 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≠0 to output information about contact penetrations for MORTAR contacts to d3plot and sleout. HISNOUT=1 activates output of more information for history variables in d3hsp. ENGOUT≠0 to output information about minimum sliding energy density for MORTAR contacts to d3plot.*

*CONTROL_SHELL

Card								
1	WRPANG	ESORT	IRNXX	ISTUPD	THEORY	BWC	MITER	PROJ
	30.000000	1	-1	4	1	2	1	1
2	ROTASCL	INTGRD	LAMSHT	CSTYP6	THSHEL			
	1.0000000	0	0	1	0			
3	PSSTUPD	SIDT4TU	CNTCO	ITSFLG	IRQUAD	W-MODE	STRETCH	ICRQ
	0	0	1	0	3	0.0	0.0	0
4	NFAIL1	NFAIL4	PSNFAIL	KEEPCS	DELFR	DRCPSID	DRCPRM	INTPERR
	0	1	0	0	3	0	1.0000000	0
5	DRCMTH	LISPSID	NLOCDT					
	0	0	0					

Active optional cards: None, Opt1, Opt12, Opt123, Opt1234

*Comment: Automatic sorting of degenerated quadrilateral shell elements, see ESORT. Shell thickness change due to membrane straining is activated, see ISTUPD. 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. 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

Card										
1	ESORT	FMATRIX	NIPTETS	SWLOCL	PSFAIL	T10JTOL	ICOH	TET13K		
	1	0	4	1	0	0.0	1	0		
2	PM1	PM2	PM3	PM4	PM5	PM6	PM7	PM8	PM9	PM10
	0	0	0	0	0	0	0	0	0	0
3	TET13V	RINRT	COHEQC							
	1	0	0							

Active optional cards: None, Opt1, Opt12

Comment: Automatic sorting of degenerated tetrahedron and pentahedral solid elements is activated with ESORT. Note that optional card 2 is not supported in LS-DYNA versions before R11.2.2. Sometimes FMATRX=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=1 to delete badly connected cohesive elements. The more accurate type 13 solid implementation with TET13V.

*CONTROL_SOLUTION

Card							
1	SOLN	NLQ	ISNAN	LCINT	LCACC	NCDCF	NOCOP
	0	0	1	1001	0	1	0

Comment: ISNAN=1 for check of NaN in force and moment arrays (debugging purposes). 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. NOCOPY=1 may speed-up element processing for certain materials.

Additional control settings

Additional control cards 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 Ansys LS-PrePost by adding the keyword *CONTROL_MPP_DECOMPOSITION_OUTDECOMP. In the examples presented in this guide *CONTROL_MPP_DECOMPOSITION_AUTOMATIC has been used, which often makes a more efficient solution compared to default. When running simulations on many cores other settings might be more suitable through, e.g., *CONTROL_MPP_DECOMPOSITION_TRANSFORMATION or *CONTROL_MPP_DECOMPOSITION_ARRANGE_PARTS.

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

*CONTROL_HOURGLASS is omitted. In this guideline the hourglass control is referenced on *PART and defined by *HOURGLASS.

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

*CONTROL_TERMINATION

	ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS	NOSOL
1	10.0000000	0	0.0	0.0	1.000e+08	0

Comment: Termination time is set by ENDTIM.

*CONTROL_TIMESTEP

	DTINIT	TSSFAC	ISDO	TSLIMT	DT2MS	LCTM	ERODE	MS1ST
1	0.0	0.9	0	0.0	-1.111e-04	0	0	0

Active optional cards: None / Opt1 / Opt12

	DT2MSF	DT2MSLC	IMSCL	UNUSED	UNUSED	RMSCL	EMSCL	IHDO
2	0.0	0	0			0.0		1

	-	IGADO	IDTUSR
3		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. Mass scaling setting in 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.

Database cards

Output files containing results information are requested and specified in the input file using *DATABASE. There are many options available, and the reader is advised to the *DATABASE section in the manual. In the comments below only a brief and limited explanation is given for the parameters. Please read the manual for the full description. The following, baseline, database settings are used in this guideline. The parameters that differ from defaults are marked with a blue rectangle. Dashed rectangles indicate additional parameters that are discussed.

*DATABASE_BINARY_D3PLOT

1	DT	LCDT	BEAM	NPLTC	PSETID	
	0.5000000	0	0	0	0	
2	IOOPT	RATE	CUTOFF	WINDOW	TYPE	PSET
	0	0.0	0.0	0.0	0	0

*Comment: Time interval DT between d3plot output states. There are many binary output databases that can be requested and controlled using *DATABASE_BINARY_OPTION1_{OPTION2}. E.g. *DATABASE_BINARY_D3DRLF, *DATABASE_BINARY_D3PART and DATABASE_BINARY_ INTFOR are commonly used.*

*DATABASE_EXTENT_BINARY

1	NEIPH	NEIPS	MAXINT	STRFLG	SIGFLG	EPSFLG	RLTFLG	ENGFLG
	0	0	3	1	1	1	1	1
2	CMPFLG	IEVERP	BEAMIP	DCOMP	SHGE	STSSZ	N3THDT	IALEMAT
	0	1	0	1	2	1	2	1
3	NINTSLD	PKP_SEN	SCLP	HYDRO	MSSCL	THERM	INTOUT	NODOUT
	0	0	1.0000000	0	1	0		
4	DTDT	RESPLT	NEIPB	QUADSLD	CUBSLD	DELERES		
	0	0	0	0	0	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 for additional options to control the output of history variables. MAXINT, number of shell (and tshell) through-thickness integration points to output. NINTSLD, number of solid element integration points to output. BEAMIP, number of integration points to output for beams. STRFLG=1, write strain tensor data to d3plot, elout and dynain. IEVERP=1 to output each d3plot state to a separate plot file. SHGE=1 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 valume and internal energy per reference volume. MSSCL=1 to output incremental (added) nodal mass due to mass scaling. The content can be further specified for a part set using *DATABASE_BINARY_D3PART and *DATABASE_EXTENT_D3PART for output in d3part. The content of the contact interface database intfor is specified using *DATABASE_BINARY_INTFOR and *DATABASE_EXTENT_INTFOR.*

*DATABASE_FORMAT

1	IFORM	IBINARY
	0	1

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
```

*Comment: Request and control output of different LS-DYNA ASCII databases using *DATBASE_OPTION1_{OPTION2}. Time interval between outputs DT. BINARY=0, data written to the binary database binout is default for MPP. BNDOUT, boundary condition forces and energy. ELOUT, element data (see *DATABASE_HISTORY_OPTION). GLSTAT, global statistics and energy. MATSUM, part energies. NODOUT, nodal motion (see *DATABASE_HISTORY_NODE_OPTION). RCFORC, resultant contact interface forces. RWFORC, rigidwall forces. SECFORC, cross section forces (see *DATABASE_CROSS_SECTION_OPTION), SLEOUT, contact interface energy.*

Record maximum, and/or minimum, stress/strain

*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.

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_GENERAL for beams. *CONTACT_ERODING_SINGLE_SURFACE is used for the cases where erosion of solids is expected. For tricky contact situations, often with high contact pressures, a MORTAR contact variant can be useful.

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. The soft constraint option is chosen with SOFT. Using option "_ID" is recommended for many reasons.

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

```
*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     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    frcfrq
         1       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       0.0
$#    igap    ignore    dprfac    dtstif    edgek    unused    flangl   cid_rcf
         1         2       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         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
```

Comment: SOFT=1. 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.

```
*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     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    frcfrq
         2       0.1         0     1.025         3        35         0         1
$#  penmax    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
$#   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         0         0         0       1.0         0         0         0
$#  pstiff   ignroff         -     fstol    2dbinr    ssftyp     swtpr    tetfac
         1         0                 2.0         0         0         0       1.0
```

Comment: SOFT=2 with SBOPT=3 and DEPTH=35. SHLEDG=1 for square shell edges flush with the nodes. IGNORE=2. TETFAC can be used to make the contact stiffness for tetrahedral solid elements more comparable with hex meshes, see manual. 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. EDGEK can be used to increase edge-to-edge penalty stiffness.

```
*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    sofscl    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       0.0
$#    igap    ignore    dprfac    dtstif     edgek    unused    flangl   cid_rcf
         1         2       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
```

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.

```
*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    sofscl    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       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 SFSA/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. 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. See manual for details.

```
*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    sofscl    lcidab    maxpar     sbopt     depth     bsort    frcfrq
         1       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       0.0
$#    igap    ignore    dprfac    dtstif     edgek    unused    flangl   cid_rcf
         1         2       0.0       0.0       0.0                 0.0         0
```

*Comment: *CONTACT_AUTOMATIC_GENERAL is only used for beam-to-beam contact in this technical guide. 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. 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.*

Tied contacts

There are many options in LS-DYNA for the 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 contact variants are presented. The intention is not to cover all functionality but to highlight some tied contacts that are widely used and may serve as a good starting point when modelling tied connections. For best control it is recommended to use node and segment sets rather than part or part sets. Thereby preventing that, unintentionally, wrong nodes are being tied. 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. I.e., if they are moved to SURFB at initialization or not (i.e., allowing offsets). If OPTION4 is left blank, 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. An example is given below.

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

By IPBACK≠0 on Optional Card E a "backup" penalty contact will created for rigid nodes that fail to tie. This feature is available for some constraint based tied contacts, see below.

TIED_NODES_TO_SURFACE
TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET
TIED_SURFACE_TO_SURFACE
TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET
TIED_SHELL_EDGE_TO_SURFACE
TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET

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.

E.g., SAST = SBST = -1.0 will give a tolerance of 0.6*(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, which is primarily due to if the nodes have rotational DOF or not. The intended use of the contacts below is revealed in their titles. 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”.

```
*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 DRCPSID on *CONTROL_SHELL. Thereby tying also torsion for beams tied to shells.*

```
*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: OPTION4 can be “blank”, “OFFSET”, “BEAM_OFFSET” or “CONSTRAINED_OFFSET”. The OFFSET variants do only tie rotational DOF for solid elements with rotational DOF, i.e., ELFORMs 3, 4, 20 or 22, or when the solid part is rigid.

Additional contact information

Other contact types are occasionally used in the examples, such as for instance FORMING contacts and *FORCE_TRANSDUCER. Settings used for these contacts can be found in the models.

Resultant contact interface force is output in rcforc, see *DATABASE_RCFORC.

SPR/MPR=1 is needed when a contact interface force file (intfor) is requested, see also *DATABASE_BINARY_INTFOR. SPR/MPR=1 is also needed for output of nodal interface force in ncforc, see *DATABASE_NCFORC.

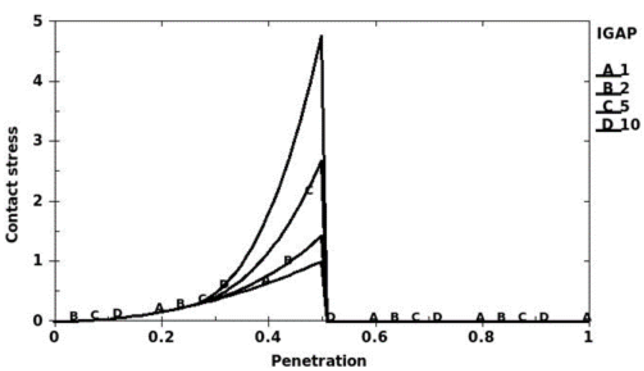
Some comments about penalty stiffness

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.

For SOFT=1 and SOFT=2 contacts, the (global) penalty stiffness for the contact at hand can be scaled using SFSA/SFSB directly on the contact definition. The base level of the penalty stiffness for this contact can 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 edge-to-edge penalty stiffness can be scaled using EDGEK. The penalty stiffness for tetrahedron elements can be scaled using TETFAC. Also, if sliding is prevalent for SOFT=2, then DPRFAC=0.001 on Optional Card C is recommended. Note: for SOFT=2 when using the scaling parameters SFSA/SFSB then they should both be set, and to the same value.

For MORTAR there are some special parameters. The base level of penalty stiffness can be scaled with SFSA. For solids and tshells the PENMAX parameter also affects the base level of penalty stiffness. Though, PENMAX is typically used to set a consistent release depth for solid parts. If locally penetrations become too large then IGAP can be increased, see figure below.



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

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

Element formulations

The element formulation for a part (PID) is set on the *SECTION card, e.g., the ELFORM 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. 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. For solids, the ability to capture bending modes accurately may also favour the fully integrated element. 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.

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 elform 2 shells (IHQ = 4, stiffness form) is decreased from the default 0.1 to 0.05 or 0.03. The hourglass coefficient for elform 1 solids may in some cases be adjusted as well, e.g., QM=1.0 for elastic materials.

A general recommendation is to set ESORT=1 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 is to 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 = -2	NA	Fully integrated, Like elform 2, but can handle poor aspect ratio better.
Solid, elform = 16	NA	4 or 5 point 10-noded tetrahedron
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

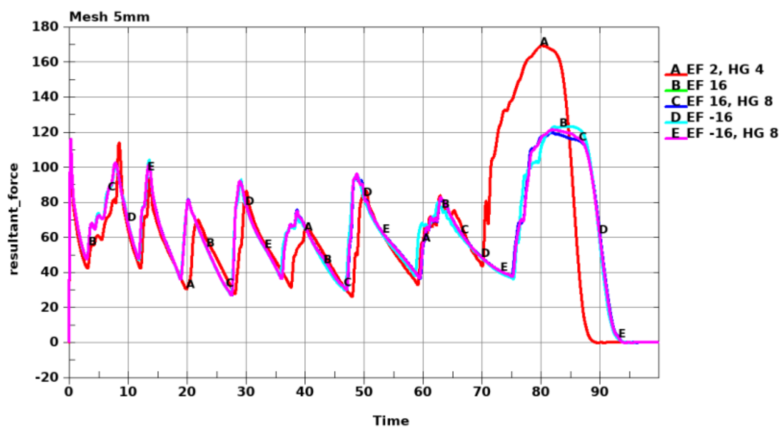
Worth mentioning is that there are cases where the tetrahedron elform 10 element may be preferred to elform 13, e.g., when the material model requires it (typically foams), when problem arises with negative volumes (severe deformations) or when two tetrahedron parts share nodes. Note: the more accurate type 13 solid implementation is activated on *CONTROL_SOLID with parameter TET13V (recommended).

In addition, hourglass control IHQ=10 in combination with solid elform 1 or 16 activates the Cosserat Point Element, which has demonstrated good performance when modelling for large deformations, e.g., foams and hyperelastic materials such as rubber.

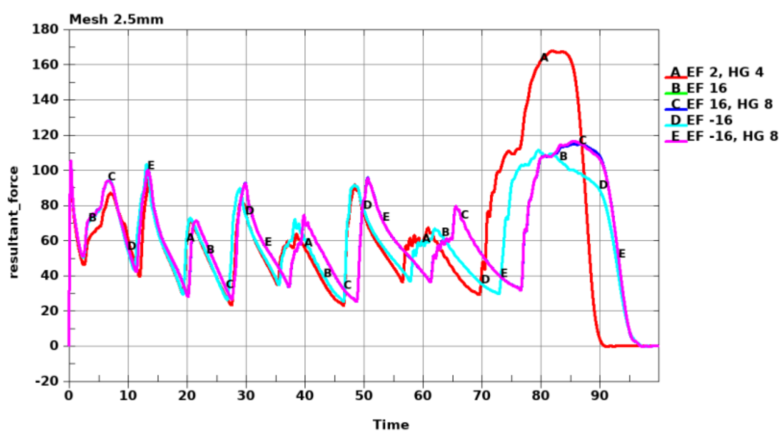
Benchmark example: Crash box

The examples in this guide can serve as a platform for quick benchmarks or testing of different settings. In this section the "Crash box" (see Example 2 for more details) was used to do comparative testing of shell elform 2 and 16/-16. Five simulations were performed. As shown in the table below the runtime was about 2.4 times larger for elform 16/-16 compared to elform 2, which was expected. Adding hourglass control 8 to elform 16/-16 did, in this example, not increase the runtime significantly. As expected, a difference in response can be seen comparing elform 2 with elform 16/-16. In this example the results using 16/-16 were practically identical. The effect of adding hourglass form 8 was only of significance for the tube with the finest mesh (2.5 mm).

Simulation	Shell elform	Hourglass control	Runtime (minutes)	Runtime (normalized)
1	2	4 (QM = 0.05)	43	1.00
2	16	None	101	2.32
3	16	8	103	2.37
4	-16	None	102	2.35
5	-16	8	105	2.42



Comment: Tube with mesh size 5.0 mm.



Comment: Tube with mesh size 2.5 mm.

Example 1: Drop test

This is an example of a drop test simulation using Ansys LS-DYNA. A ceramic mug is dropped from a height of 1.5 m, allowing it to freefall until it hits a hard surface. The model exemplifies a parameter definition, an accelerometer, a strain gauge load cell and the contact treatment of eroding elements.

Simulation data

# nodes	184k
# elements	146k
Timestep size (ms)	0.1e-3 (constant)
Termination time (ms)	10.0
Solution time (minutes)	114

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_110[2] is used for the mug. Elements 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.

---

[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.

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. An ERODING contact automatically invokes a negative volume failure criterion for all solid elements in the model. The use of PSFAIL on *CONTROL_SOLID limits this criterion to a partset, which is recommended. 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. This 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 fringe plot of the added mass is possible.

Loads and constraints

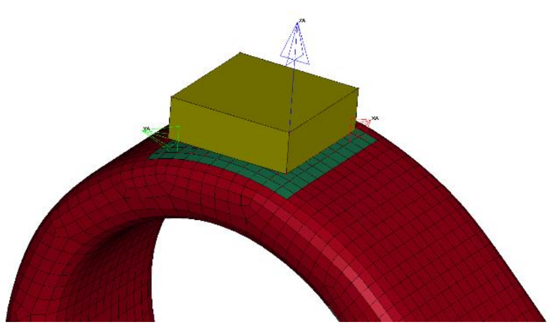
Gravity is applied with *LOAD_BODY_Z. The bottom of the impacted surface is fully constrained with SPC.

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. Thereby preparing the model for quick configuration when performing, for example, parameter studies.

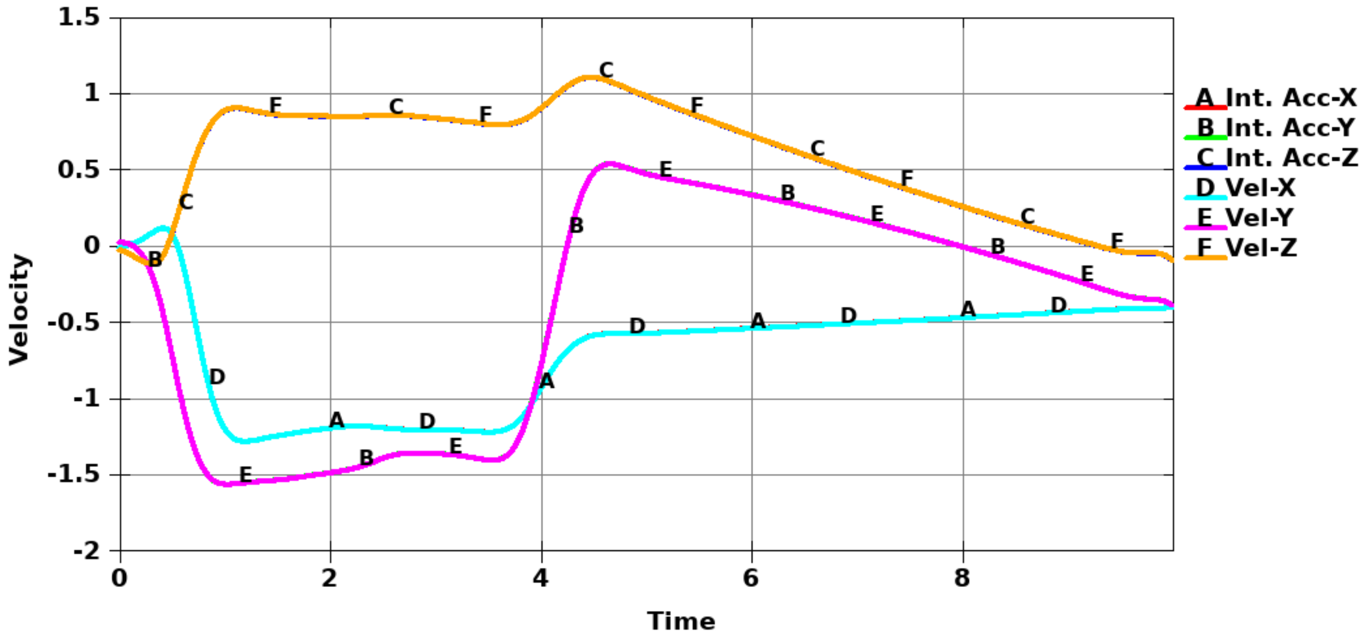
Accelerometer

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.

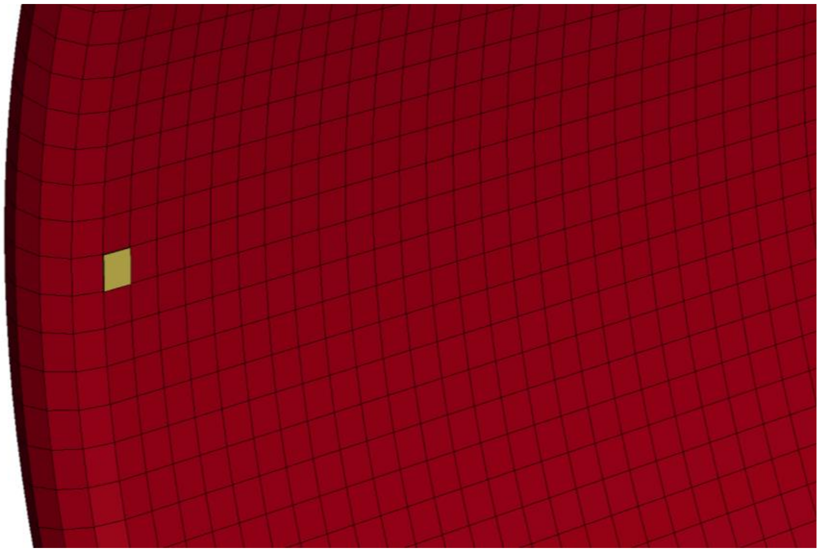
The global node can be used to check that the output frequency of the accelerometer is sufficiently high, which is by integrating each acceleration component (x, y, z) and compare the results to the corresponding measured velocities. These curves should match when offsets are removed, see figure below.



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

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. The results show (initially in the simulation) tension on the outer surface (positive x-strain) and compression on the inner surface (negative x-strain) as expected (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).

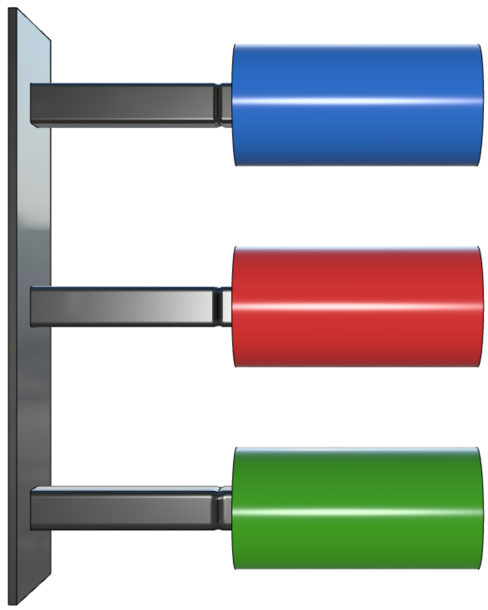
Results

Well, the purpose of a drop test is often to see if something breaks or not and the answer here is obvious. It can also be said that this crash event is rapid and brittle, which is shown by the acceleration curves. Evaluation of acceleration levels would therefore depend highly on the chosen filter and frequency.

Comment: Drop test at 10 milliseconds.

Example 2: Crash box

This is an example of a crash box buckling analysis using Ansys LS-DYNA. Three identical, except for the mesh 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 case exemplifies *MAT_024 with strain rate effects, history variables, contact force transducer and the concept of mesh convergence.



Simulation data

# nodes	81k
# elements	72k
Timestep size (ms)	0.3e-3 (constant)
Termination time (ms)	100.0
Solution time (minutes)	109

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. The mesh size is for the three boxes 10 mm, 5 mm and 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, which can be fringe plotted in a post-processor. In Ansys LS-PrePost this can be done 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 "h i s t o r y v a r i a b l e s l i s t i n g".

Loads and constraints

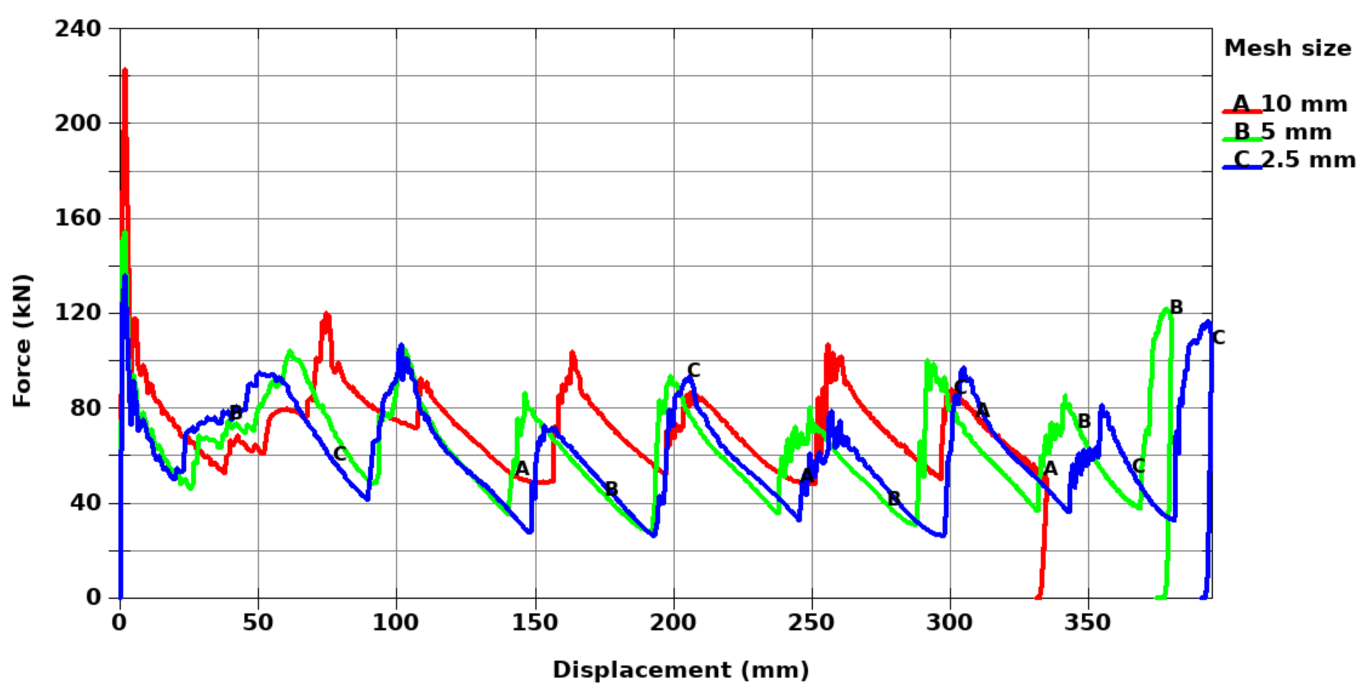
The rear end of the crash box is fixed with *BOUNDARY_SPC_SET.

Initial conditions

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

Results

By specifying the same out 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 tube with 5 mm mesh and 2.5 mm are similar whereas the 10 mm mesh produce a significantly stiffer response.

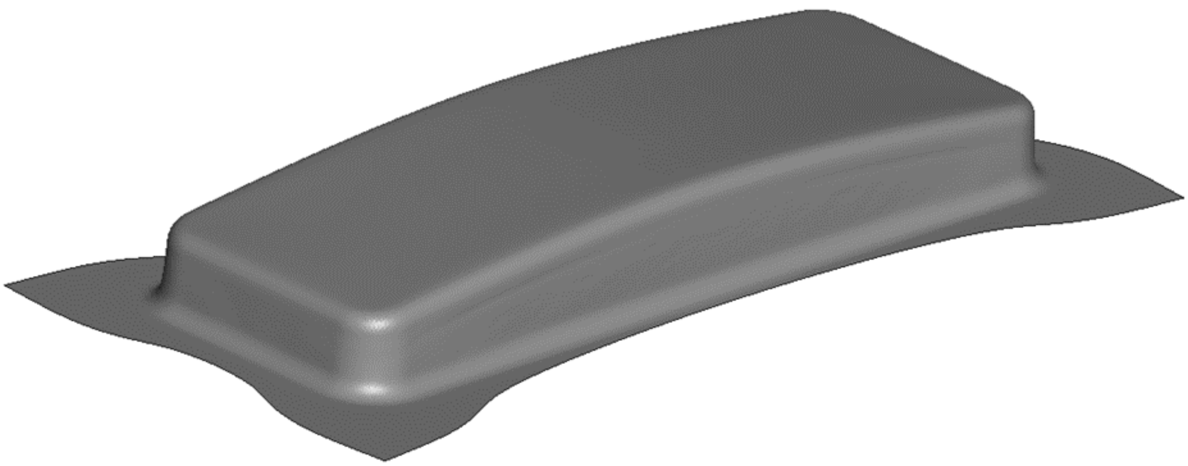


Comment: Force verses displacement curves for the three tubes.

Example 3: Deep drawing

This is an example of a deep drawing simulation using Ansys LS-DYNA. A rectangular shape with curved top is formed with a single action die process. This example shows a typical setup for a deep drawing simulation with blank and tool definition, process boundary conditions, material model and some recommended settings for metal forming simulations.

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



Simulation data

# nodes	11k (initial)
# elements	11k (initial)
Timestep size (ms)	0.54e-3 (constant)
Termination time (ms)	8.8
Solution time (minutes)	4

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 follow the correct tool geometry. The blank is meshed with an initial element length of 4 mm, but also has adaptive mesh applied 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

Contact type 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 with some parameters setting defined for this purpose, e.g. the tool thickness is not considered. Note that this 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. Also set ADPOPT 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. The shell thickness change 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.

Deep drawing simulations when the blank is clamped between the die and binder usually have low dynamic effects and can therefore 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 360 %.

Loads and constraints

The die is translated with a trapezoidal load curve defining the velocity with a ramping time of 1 ms. The die is translated with the forming velocity (BOUNDARY_PRESCRIBED_MOTION_RIGID), and the binder has a load applied to it with LOAD_RIGID_BODY. The binder also has a vertical constraint (CONSTRAINED_RIGID_BODY_STOPPERS) applied to it during the forming. The punch is fixed during the whole process.

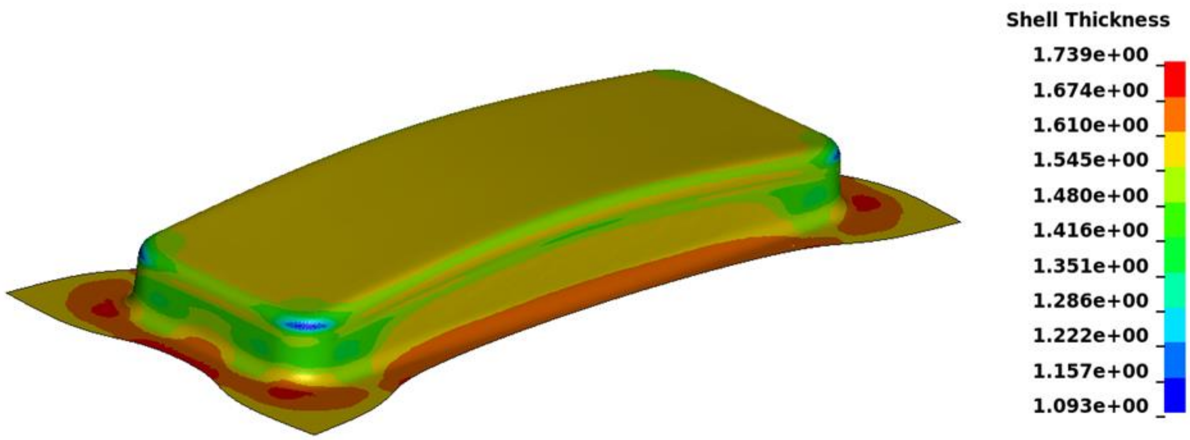
Output

The ASCII files usually needed for a sheet metal forming simulation is bndout, glstat, matsum, rcforc and sleout. In the 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 Forming Limit Diagram (FLD).

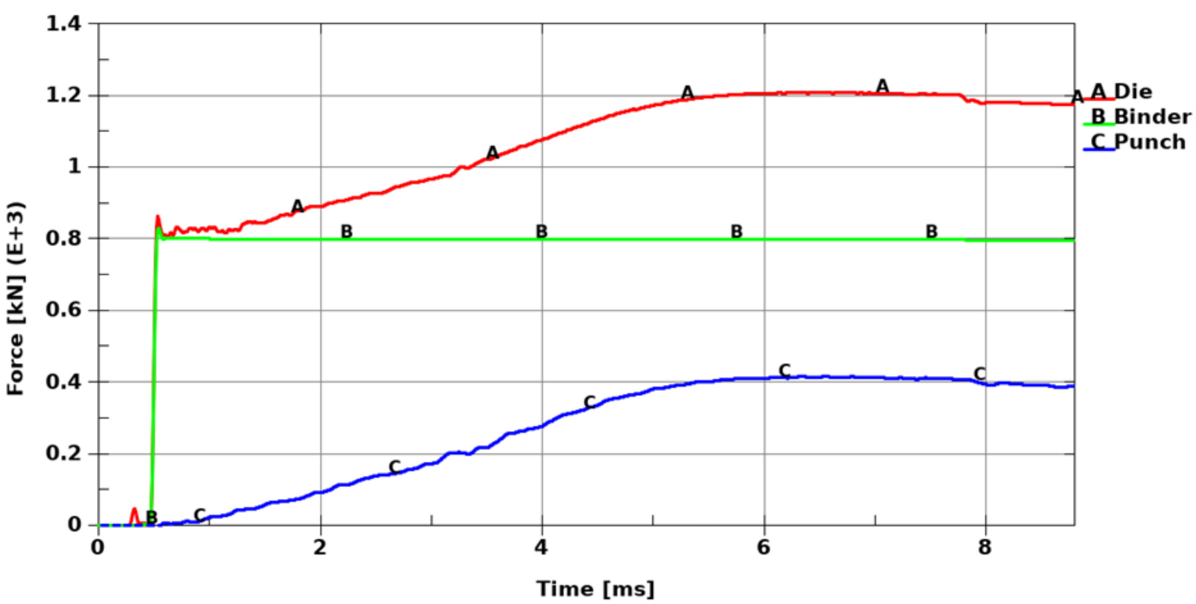
INTERFACE_SPRINGBACK_LSDYNA is used to extract a dynain file at the end of the simulation containing the thickness, strain, and stress in the blank.

Results

A search in the d3hsp and message files shows no errors or significant warnings. 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, but apart from that the curves looks smooth.



Comment: Thickness distribution of blank at end of forming.



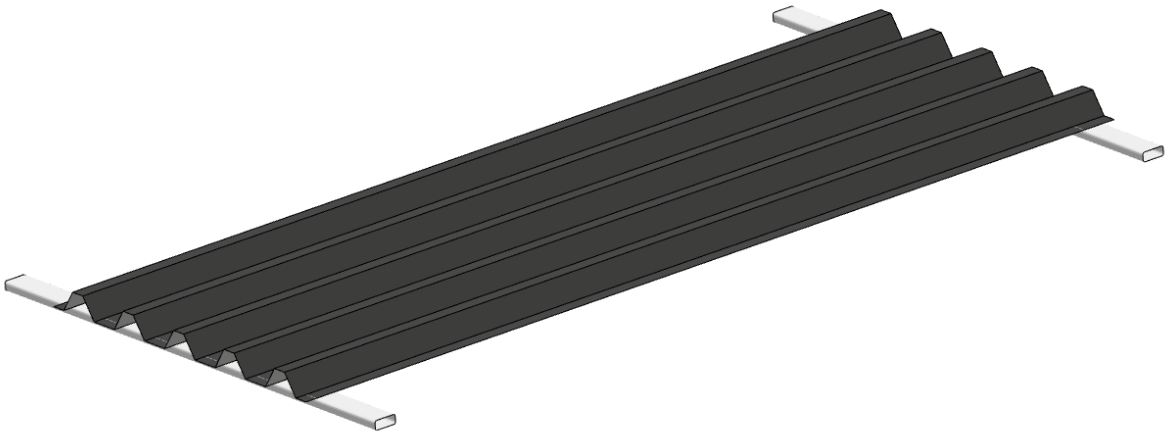
Comment: Tool forces during forming.

Springback

A springback analysis gives the resultant deformation of the blank when it is 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 accuracy. 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.

Example 4: Post-buckling strength

This is an example of a quasi-static buckling analysis using Ansys LS-DYNA. The target of the simulation is not only to find the ultimate strength of the profile but also to find the residual post-buckling strength. The model demonstrates a testing rig with an applied uniform pressure load, the use of a PI-regulator as well as the inclusion of geometry perturbations. The use of selective mass-scaling is briefly discussed.



Simulation data

# nodes	136k
# elements	136k
Timestep size (ms)	0.664e-3 (constant)
Termination time (ms)	150.0
Solution time (minutes)	176[3]

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 simulation is to be quasi-static. It is important to have a realistic description of the hardening curve, also in the necking region, when simulating post-buckling behaviour.

[3] The solution time may be reduced by about 40% with Selective Mass Scaling (SMS). The main file run.key is prepared for such a test, see lines that are commented out.

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 defined by *DEFINE_CURVE_FUNCTION is output. The curve is in this case the steer signal for the pressure as applied by the PI-controller.

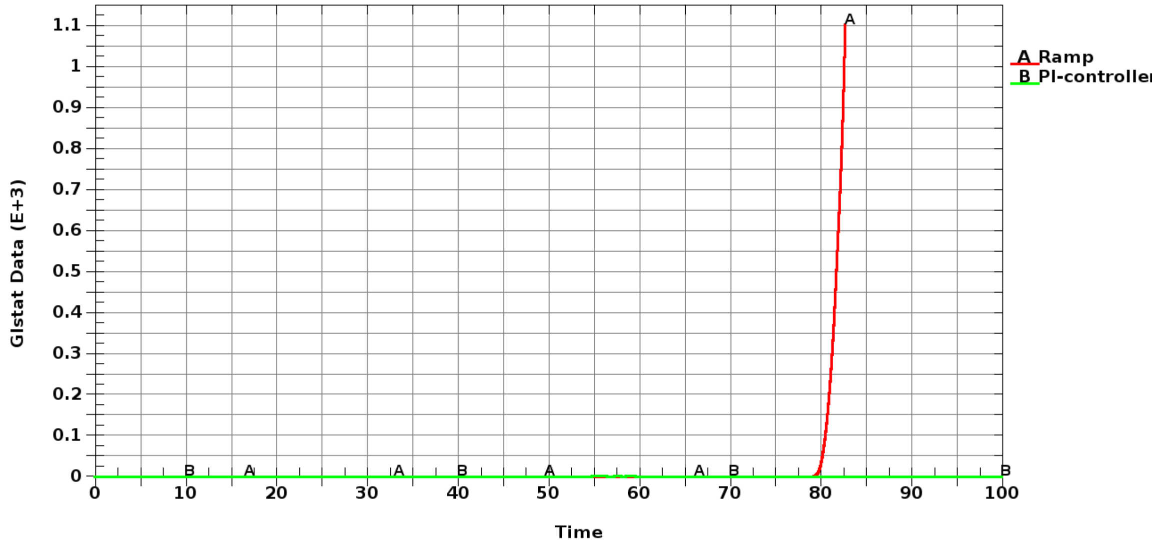
Loads and constraints

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 up till ultimate strength is reached. After that the kinetic energy will increase rapidly as the profile collapses due to buckling. A ramp load cannot 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. As with all PI controllers adjusting the parameters can be a bit tricky. In this case the initial settings of parameters, i.e., the proportional gain kp and the integral gain ki, gave a fast prediction of ultimate load as well as post-buckling strength. However, the elastic response before buckling was oscillating too much. This was improved by decreasing kp and increasing ki.

Results

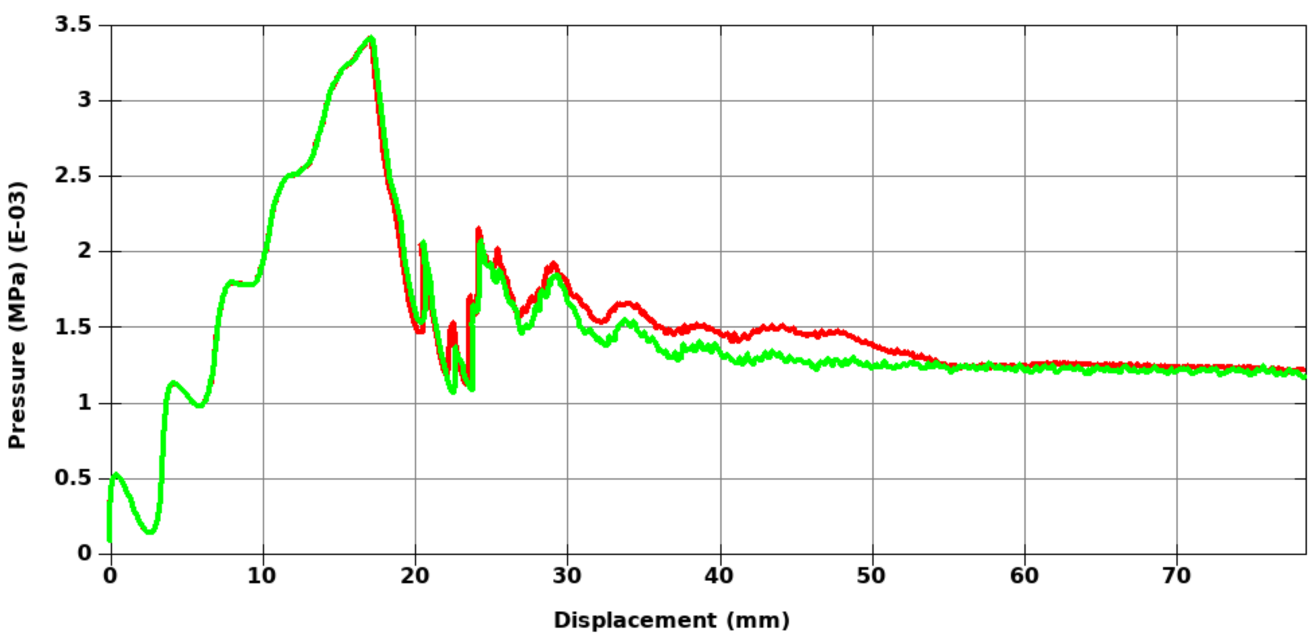
By scaling CURVOUT by a factor of 1.0e-2 the applied pressure (MPa) can be plotted. BNDOUT gives the reaction forces in the supports. In addition, the z-displacement can be plotted from NODOUT. The node at hand, NID 108355, is used by the PI-regulator to measure the velocity of the applied load. This node is the dependent node in a *CONSTRAINED_INTERPOLATION definition and follows the motion of the profile. By this arrangement, a more robust set point (in this case the velocity) can be monitored.

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 post-buckling strength. 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).

It is possible to decrease the simulation time about 40% by using selective mass scaling. This can be an efficient way to speed up for instance an optimization loop. 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, it allows slip due to play between the bolt shank and the hole edge, it allows tear-out of the hole, it allows the bolt to deform/bend to some degree, it allows the bolt to break, and it includes pretension. The bolt shank is in this example modelled in two ways, which is with beam elements and with solid elements, respectively.

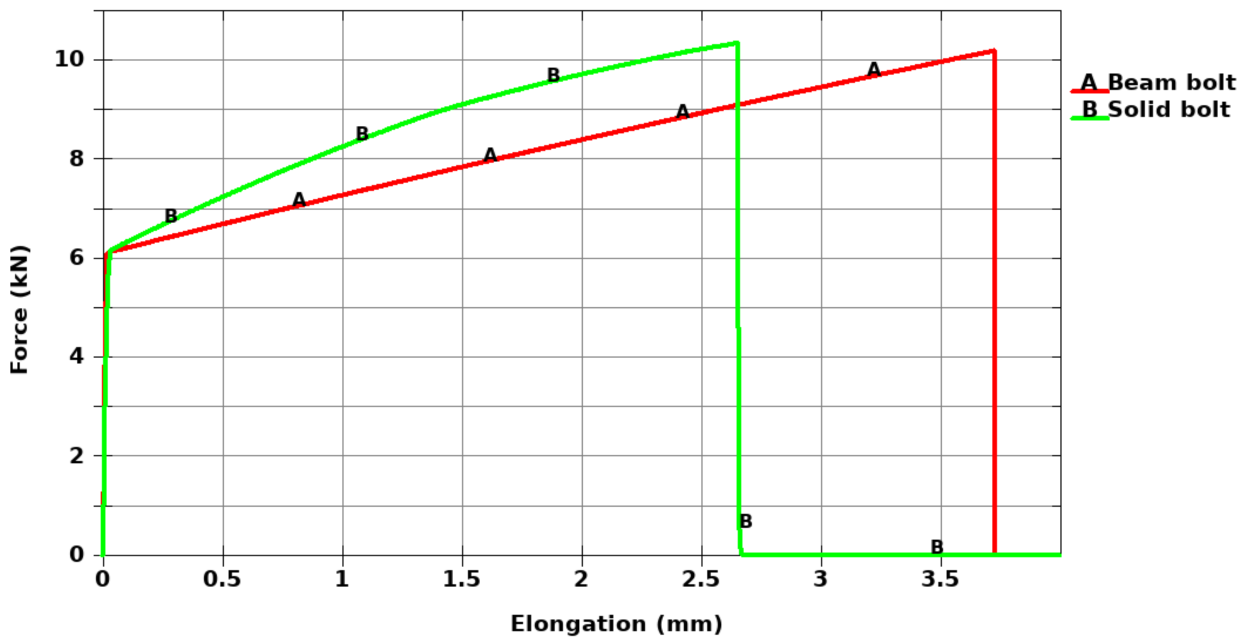
This example demonstrates, among others, the use of *INITIAL_AXIAL_FORCE_BEAM, *INITIAL_STRESS_SECTION, *CONTACT_AUTOMATIC_GENERAL and *CONTROL-_DYNAMIC_RELAXATION.

Simulation data

# nodes	19k
# elements	18k
Timestep size (ms)	0.4e-3 (constant)
Termination time (ms)	40.0
Solution time (minutes)	14

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 shanks 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. By defaults the beam element is deleted when all 4 integration points fail. To make the comparison more interesting *MAT_ADD_EROSION was added with NUMFIP=1.0. This way the beam element is deleted when the first integration point fails. As shown in the figure a simulated tension tests give differences in results, as expected due to the difference in element types.



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 for the beam bolt shank and for the null beams that are added for 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. On *DATABASE_EXTENT_BINARY, BEAMIP=4 for beam integration point output.

Prestress of bolts

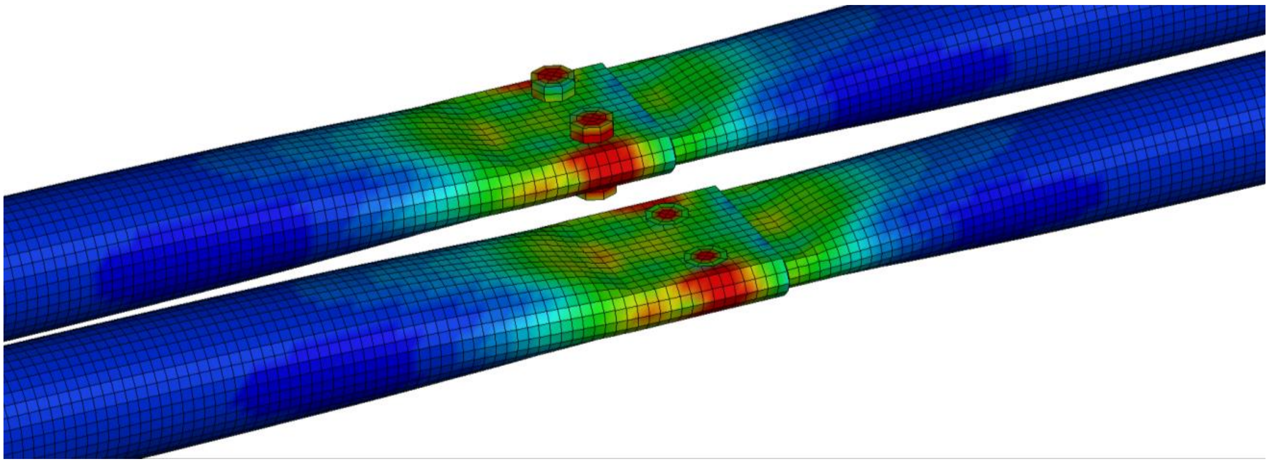
The bolts are prestressed during dynamic relaxation which is activated by SIDR=1 on the loading curves. *INITIAL_AXIAL_FORCE_BEAM and *INITIAL_STRESS_SECTION with ramped load curves prestress the beams and solids, respectively. IZSHEAR=2 on *INITIAL_STRESS_SECTION allows bending/shear stresses to develop during the prestress of the solid bolts. *CONTROL_DYNAMIC_REALAXTION, DRTERM=10.0 limits the relaxation phase to a maximum of 10 milliseconds. Since DRTOL is set to a very low number (1.e-10) 10.0 ms for the prestress is expected. The prestress process can be studied in detail through D3DRLF (see *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.

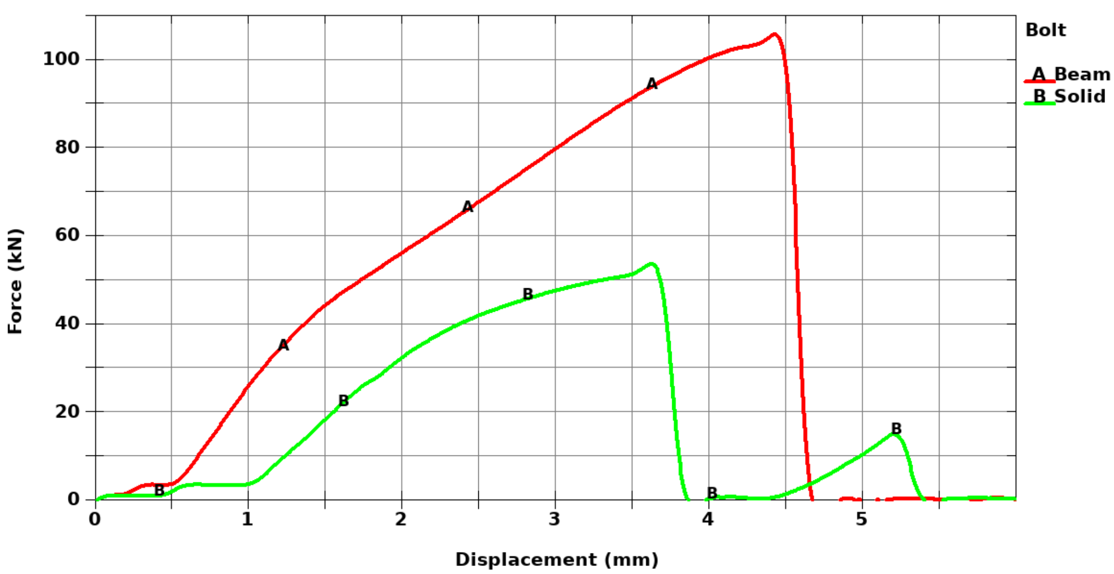
Results

A fringe plot of the prestressed connections show that the resulting stress distributions are the same. Looking at stresses and axial force in ELOUT and SECFORC confirm 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. For complex loading situations such as this calibration of the model at hand is needed. Especially in situations when a very detailed model is not convenient, which the modelling approach presented here is an example of.

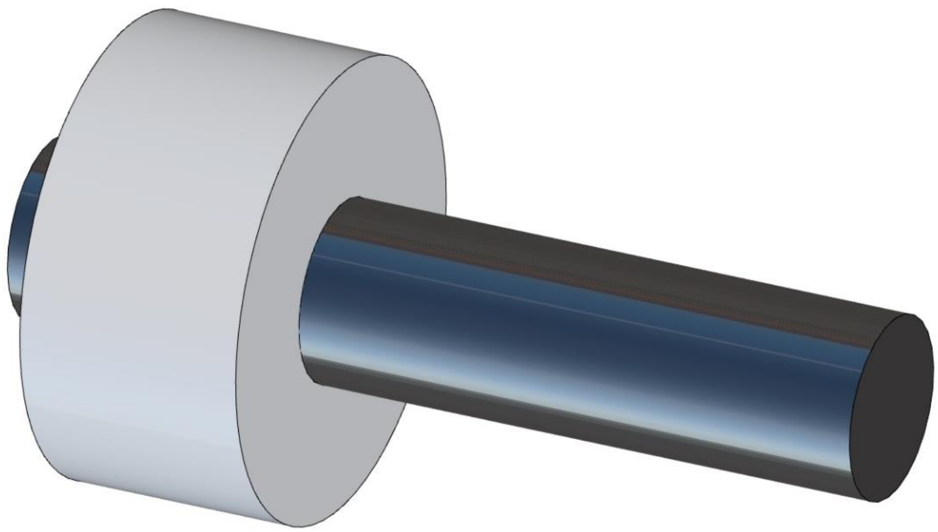


Comment: Ultimate load (x-force) for the loading case where the connections are pulled apart. Beam element variant in red and solid element variant in green. BW 3000Hz filter applied.

Example 6: Interference fit

This is an example of an interference fit analysis in Ansys LS-DYNA. The model 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 demonstrates, among others, the use of *LOAD_THERMAL_VARIABLE, *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH and the INTFOR file.



Simulation data

# nodes	7k
# elements	17k
Timestep size (ms)	0.3e-3 (constant)
Termination time (ms)	100.0
Solution time (minutes)	18

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 ensure 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 correctly, i.e., the faceted surface that would be the case for a non-SMOOTH contact would yield an incorrect torque. 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

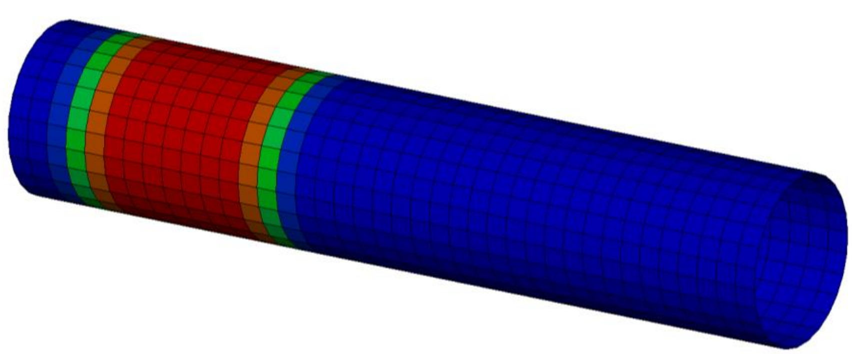
*CONTROL_DYNAMIC_RELAXATION is added for the temperature load, which is applied during the DR-phase.

Loads and constraints

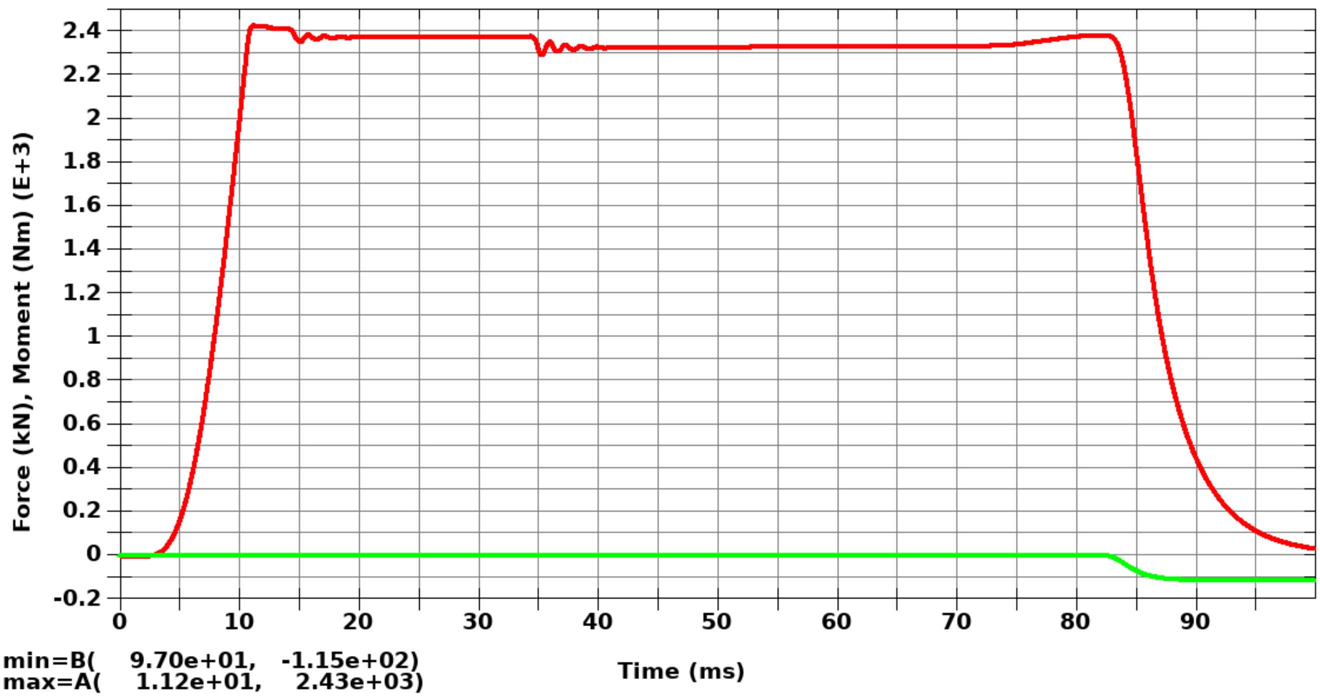
The outer surface of the hub is fixed through *CONSTRAINED_INTERPOLATION, which allows only radial movement. The beam element is added for the purpose of fixing the dependent node. This boundary condition is chosen to make comparing of the results with a known analytical solution applicable. The loading,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

Results

The main results are the interference pressure, which can be fringe plotted from the interface force file, and the axial friction force and transmission torque. The latter two 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. The analytical estimate is sensitive to the input allowance. Hence, the accuracy in the comparison is estimated to be within a few percent.



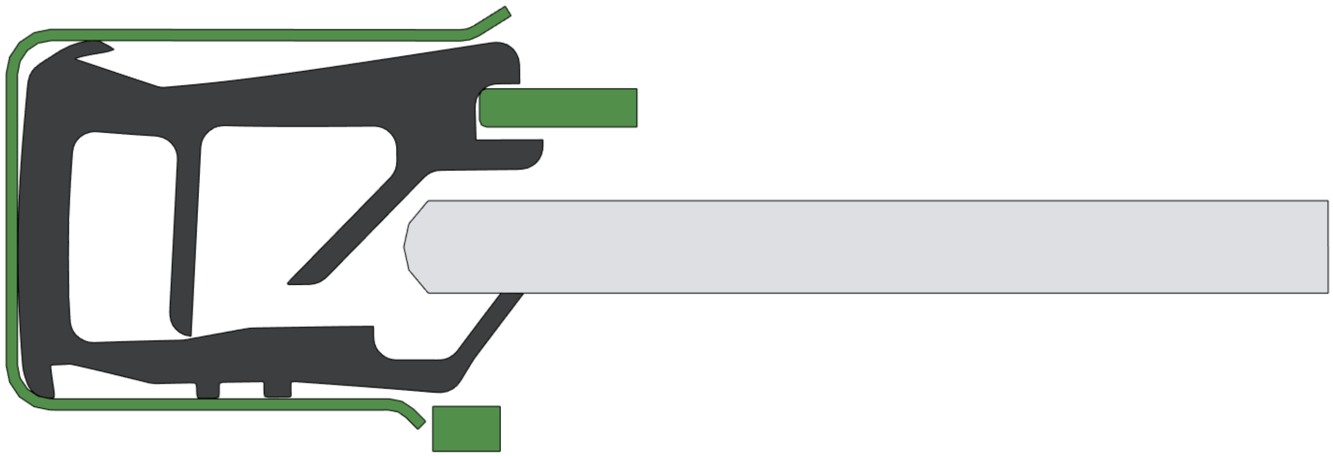
Comment: Interference pressure. Red colour is range 190-200 MPa.



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

Example 7: Rubber seal

This example exemplifies the modelling of a rubber seal. In this case a seal 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 seal to prevent water and air leakage. The model consists of the deformable rubber seal as well as a rigid door frame and a window. This example demonstrates, among others, the use of *MAT_077 (*MAT_HYPERELASTIC_RUBBER) in combination with a MORTAR-contact to handle the complicated contact situation.

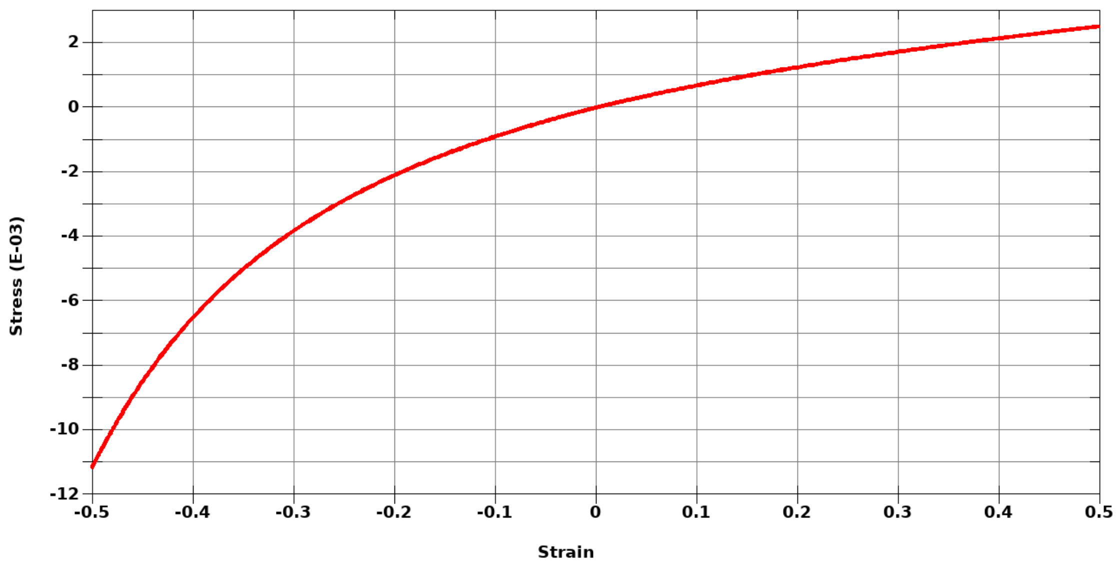


Simulation data

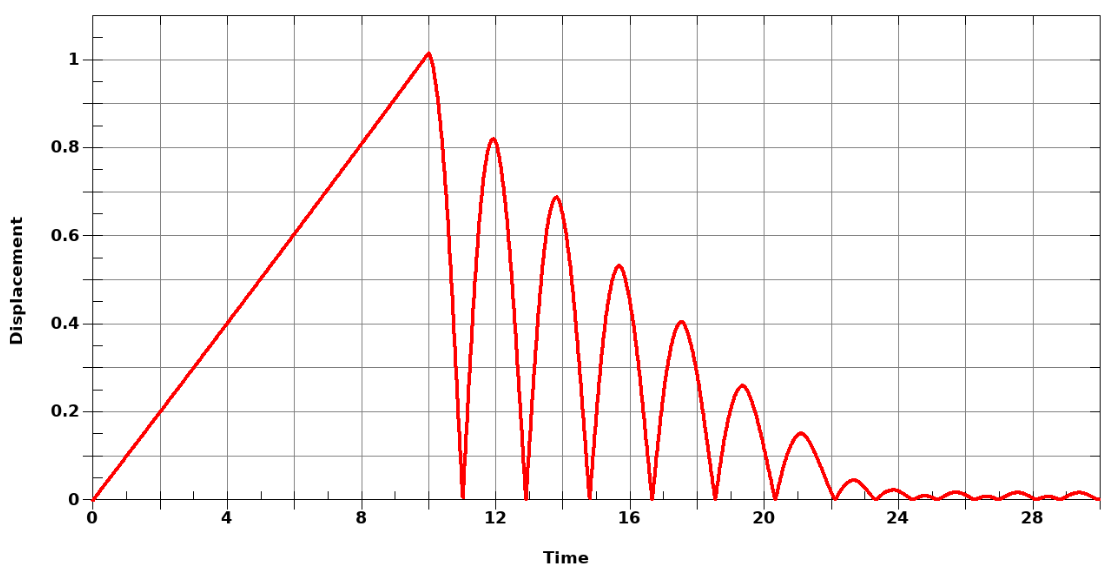
# nodes	30k
# elements	23k
Timestep size (ms)	0.311e-3 (constant)
Termination time (ms)	540.0
Solution time (minutes)	171

Elements and material

The rubber is modelled with the solid hexahedron (mostly) ELFORM 1 element in combination with hourglass formulation IHQ=6 and QM=1.0. The material model is *MAT_077 with a poison'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 a *CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR contact. The only parameters that are changed from defaults are contact damping, VDC=20, and friction behaviour. A *DEFINE_FRICTION 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

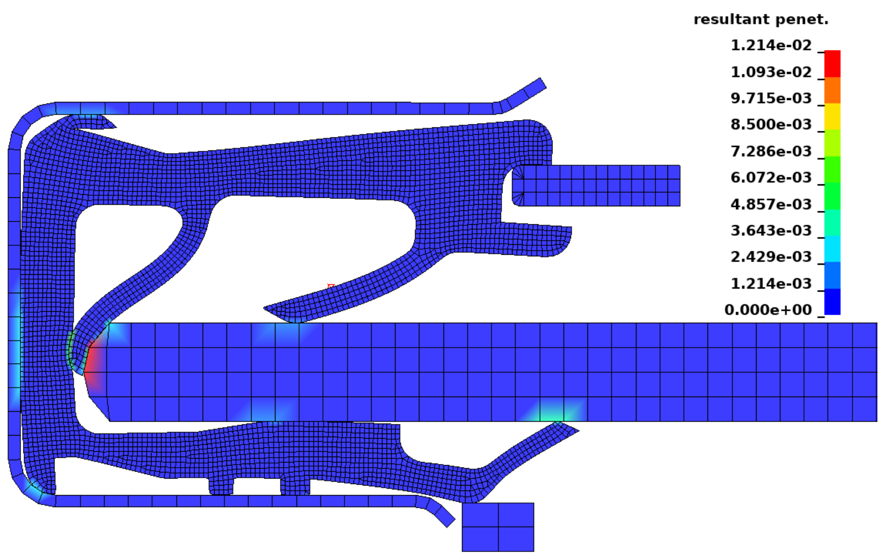
The movements of the rigid parts, which are defined by smooth velocity curves, are controlled by *BOUNDARY_PRESCRIBED_MOTION_RIGID. The symmetry constraint (in z-direction) is defined with *BOUNDARY_SPC_SET.

Output

*CONTROL_OUTPUT, PENOUT=1 for output of contact (absolute) penetrations in 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. It needs to be accurate.

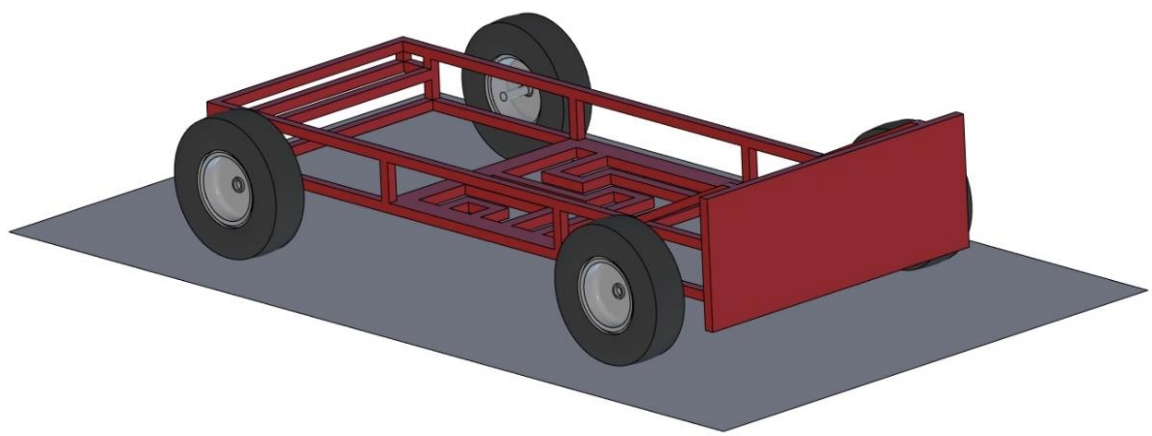


Comment: Fringe plot of contact penetration in the MORTAR-contact.

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

This example shows a procedure to initialize a vehicle for crash analysis. The aim is to first set the vehicle in balance with gravity and then to apply initial velocity, which also includes rotational velocity of the wheels. This example demonstrates, among others, the use of *INITIAL_VELOCITY_GENERATION for a situation that includes *PART_INERTIA, and *INITIAL_VELOCITY_GENERATION_START_TIME.



Simulation data

# nodes	20k
# elements	18k
Timestep size (ms)	1.0e-3 (constant)
Termination time (ms)	200.0
Solution time (minutes)	4

Barrier model

The original barrier model was developed by Livermore Software Technology Corporation (LSTC)[4]. The model has been modified by the author to fit the purpose of this example e.g., the bumper parts have been removed and the initial velocity cards have been updated. Worth mentioning is that the mass and inertia properties of the carriage is defined through *PART_INERTIA and that the wheels are inflated by airbags with *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 by the author is as follows:

1. The tires are inflated during a span of 5 milliseconds.
2. A rigid floor is pushing up the tires to ground level between time 5 ms and 10 ms. The vertical motion of the floor is prescribed with *BOUNDARY_PRESCRIBED_MOTION_RIGID. Gravity, see *LOAD_BODY_Z, is ramped up during the same time-period. In addition, mass weighted damping is applied to the tires. The damping is applied until time 20 ms and then ramped down to zero. This means no damping after time 20 ms. Note that no damping is applied by the airbag model itself, see parameter MWD in *AIRBAG_SIMPLE_PRESSURE_VOLUME. The reason for this is that such airbag damping would prevent the wheels from rotating correctly.
3. The contact between the floor plate and the tire is then removed and replaced by *RIGIDWALL_PLANAR. This step is not necessary if using the floor plate through the entire analysis.
4. Initial velocity is applied at time 30 ms with *INITIAL_VELOCITY_GENERATION and *INITIAL_VELOCITY_GENERATION_START_TIME. The procedure of point 1 to 3 may in some cases be completed faster, and if so the start time for the initial velocity can be set earlier. The *INITIAL_VELOCITY_GENERATION card is defined twice, which means once for PHASE=0 and once again for PHASE=1. Finally, IRIGID=1 for the carriage to overrule initial velocity set on *PART_INERTIA.

An alternative to this method would be to use *CONTROL_DYNAMIC_RELAXATION. This would be easier to set up and it would probably work, but it would be more time consuming (i.e. longer simulation time) compared to this relatively fast procedure. This is even more true in the case of a more complicated car. A second alternative would be to do it with the implicit solver, but it would potentially be time consuming to do the necessary changes to make the model run efficiently also using the implicit solver.

Preparations

The preparations needed when setting up a model as shown in this procedure is a) to determine the damping coefficient of *DAMPING_PART_MASS_SET for the tires and b) 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.

Point a is done by inflating a free wheel and measure the period of the tire oscillations. Point b can be determined by leaving the vehicle on ground under the influence of gravity and measure the motion of the vehicle in z-direction. Note that in this model there is no wheel suspension. If wheel suspension is present, then the suspension springs must be prestressed accordingly. This means given that the vehicle is modelled as if static equilibrium is fulfilled. 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 later replaced by *RIGIDWALL_PLANAR using the BIRTH/DEATH functionality.

Control cards

No special cards added.

Loads and constraints

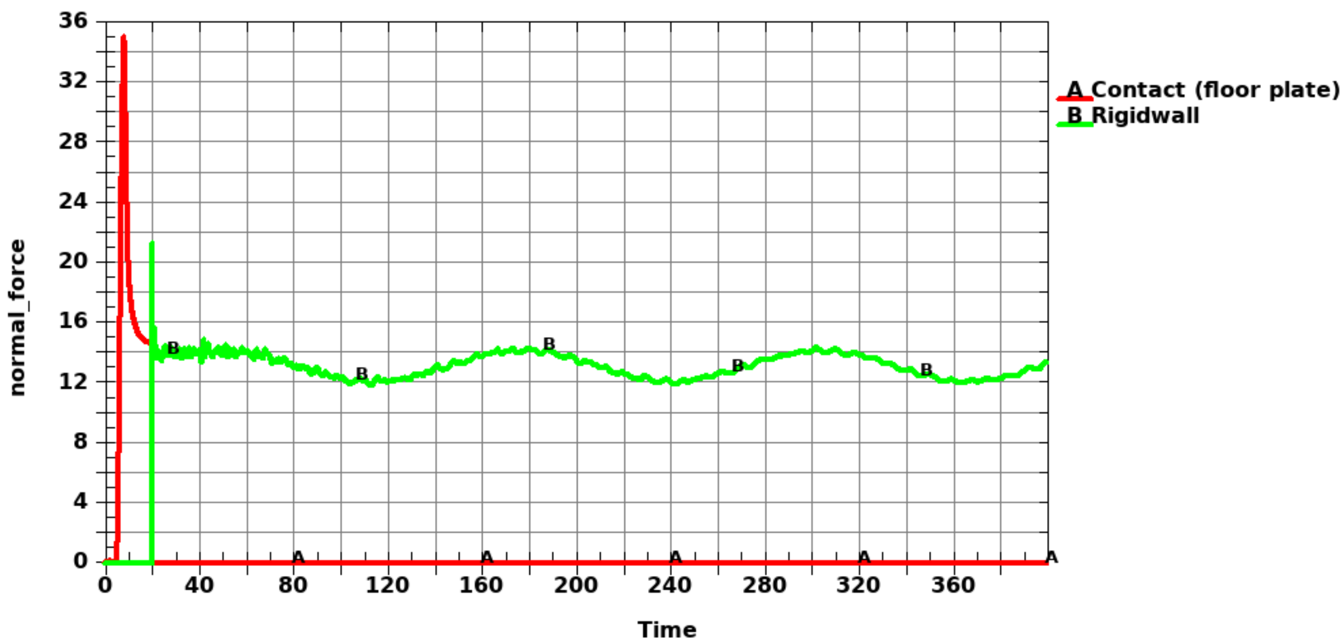
*LOAD_BODY_Z, *BOUNDARY_PRESCRIBED_MOTION_RIGID and *INITIAL_VELOCITY_GENERATION as described earlier. The use of *PARAMATER and *PARAMETER_EXPRESSION makes it easy to change the velocity.

Output

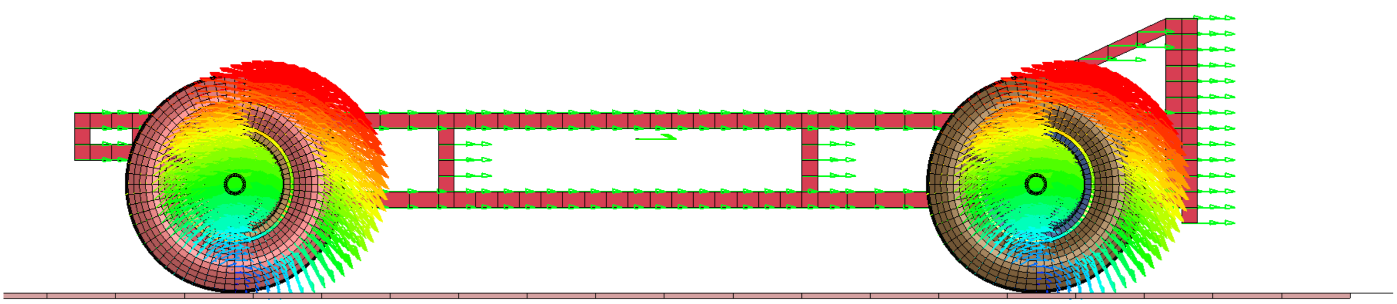
*DATABASE_ABSTAT has been added for monitoring tire pressure.

Simulation check

A “steady state” analyses shows that the initiated vehicle/carriage drops <1 mm under the influence of gravity, which proof that the vehicle initialization procedure has been successful. The initial velocity at time 30 ms is confirmed by a vector plot of the velocity from D3PLOT. Checking the kinetic energy in GLSTAT is also a good measure.



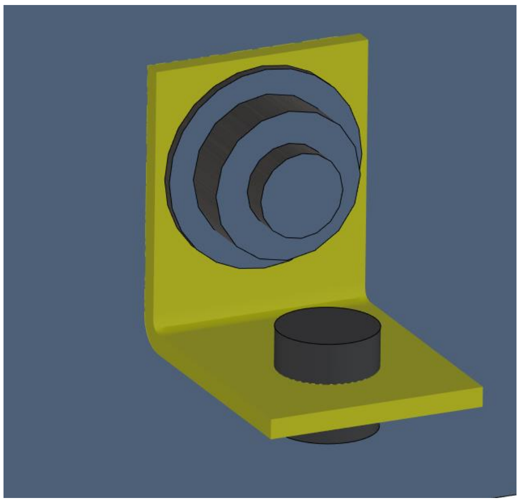
Comment: A “steady state” analysis (no velocity applied) to check static equilibrium. A switch of any contact or rigid wall will introduce noise, see spike in green curve.



Comment: Initial velocity applied.

Example 9: Short Fiber reinforced plastic bracket

This is an example of a Short Fiber Reinforced Plastic (SFRP) bracket. The mechanical properties of SFRP are greatly dependent on the fiber orientations with respect to the loading direction. Within a component, the orientation of fibers differs due to the manufacturing process. The manufacturing process, often Injection Molding, of the component at hand must therefore be considered when simulating short fiber thermoplastics. A process of how to find out the fiber directions and make a LS-DYNA material card is described here. For further information about the simulation of plastics, please contact your local distributor of Ansys LS-DYNA.



Simulation data

# nodes	80k
# elements	54k
Timestep size (s)	0.2e-6 (constant)
Termination time (s)	0.02
Solution time (minutes)	34

Note that the unit system in this example is mm, s, tonne, N, MPa.

Elements and material

The plastic bracket is modelled with 10-noded tetrahedron elements type 16. The number of integration points is set by NIPTETS=4 on CONTROL_SOLID. Three suitable LS-DYNA material model alternatives for SFRP are the composite model *MAT_ANISOTROPIC_ELASTIC_PLASTIC (*MAT_157), the micromechanical model *MAT_4A_MICROMEC (MAT_215) and the machine learning-based multiscale material model *MAT_DMN_COMPOSITE_FRC. In this example *MAT_157 is chosen. A brittle failure can be considered in *MAT_157 by either the Tsai-Wu or the Tsai-Hill criterion, although it is not included in this example.

Procedure

The procedure to create the input data for *MAT_157 to LS-DYNA is in short:

1. Perform an Injection Molding simulation of the component.
2. Map the results to LS-DYNA format (*INITIAL_STRESS_SOLID).
3. Characterize the hardening properties of the material (LCSS).

Fiber orientation and anisotropic stiffness

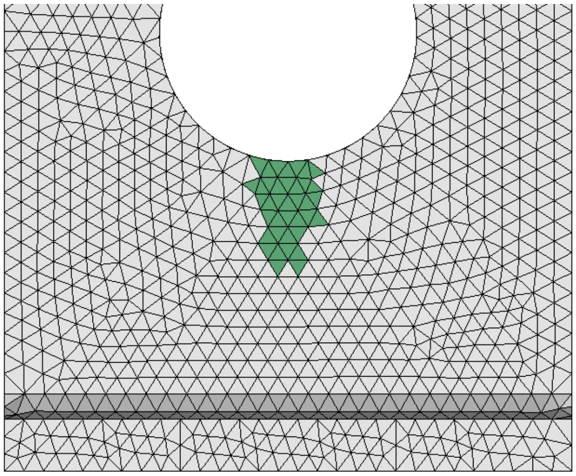
To consider the fiber orientations in the LS-DYNA model, one must first perform the Injection Molding simulation. There are different software on the market that can be used for this purpose. In this example, the Ansys Technology Partner Moldex3D, by CoreTech System Co., Ltd, was used. The setup was straightforward since the material, including fiber content, was chosen from a database of material data from many suppliers. The component was read in as a 3D-CAD, in this case a STEP-file. A gate and runner were defined where the material was injected. The software suggested a mesh size, that was generated automatically. Process parameters such as flow rate profile and temperatures can be adjusted before starting the filling analysis.

The results from the Injection Molding analysis must then be mapped to a format that can be read by LS-DYNA and the mesh one intends to use for the structural evaluation, which probably differs from the mesh used for the Injection Molding analysis. There is a limited mapping functionality within Moldex3D. In this case, Envyo, by DYNAmore GmbH an Ansys Company, was used for the mapping. The results from the filling simulation were exported, including weld lines information and fiber orientations. This means that a total of three files were exported, i.e. nodes and elements of the injection molding mesh, fiber orientations and location of weld lines. There is a functionality in Envyo to make separate parts on areas around weld lines, where the material properties typically can be set lower if desired.

In Envyo, the target material model was set to *MAT_157. The parameter IHIS was set to 3, which means that both material directions and an anisotropic elastic stiffness matrix is output to *INITIAL_STRESS_SOLID for each integration point of the element. The elastic properties of the polymer matrix and fibers are in the mapping combined to give the complete elastic stiffness matrix in what is referred to as a homogenization process. To perform this homogenization with Envyo, the elastic material properties, density etc, of the polymer matrix and the individual fibers are part of the input to Envyo. Also note that the local element coordinate systems, in which the fiber orientations are given, are calculated differently depending on the value of the INN parameter on *CONTROL_ACCURACY. The chosen setting in the simulation model, in this case INN=1, must therefore also be set in Envyo.

Currently, the mapped material directions cannot be shown directly from the keyword file in LS-PrePost. The model must first be initialized by LS-DYNA and then the first vector (the main direction) can be displayed from the d3plot results. For LS-PrePost to know which value of the INN parameter that has been used, one must also open the keyword file. After that, the visualization is done through "Post" - "Vector Plot". For solids choose "Hist. Var. cosine" and set "H.VAR X" = -4, "H.VAR Y" = -5 and "H.VAR Z" = - 6, which is where you find the appropriate history variables for *MAT_157. The corresponding settings for shells are "H.VAR X" = -1, "H.VAR Y" = -2.

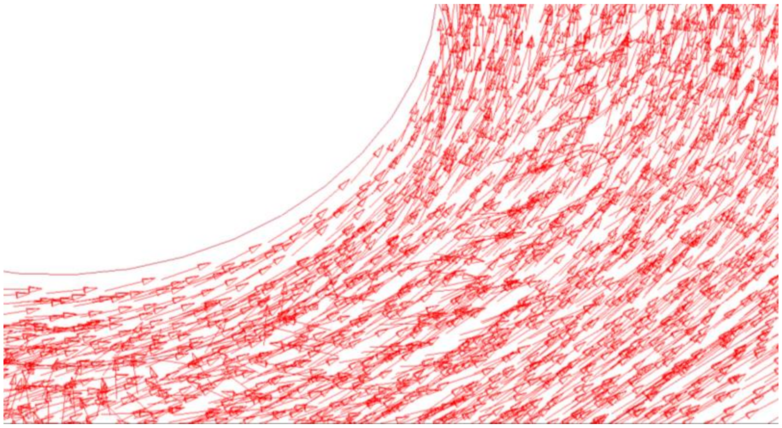
Note: Even though the elastic stiffness tensors are input on an integration point level, which overwrites the defined stiffness tensor on the material card, a reasonable value of a typical stiffness must be given on the material card. LS-DYNA uses this information to e.g. calculate an initial time step size.



Comment: The weld line is given a separate PID by Envyo (green in figure).

```
*MAT_ANISOTROPIC_ELASTIC_PLASTIC
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$#     MID        RO      SIGY      LCSS       QR1       CR1       QR2       CR2
         1
$#     C11       C12       C13       C14       C15       C16       C22       C23

$#     C24       C25       C26       C33       C34       C35       C36       C44

$#     C45       C46       C55       C56       C66         F         G         H

$#       L         M         M                AOPT        VP                MACF
                                               0.0
$#      XP        YP        ZP        A1        A2        A3               EXTRA

$#      V1        V2        V3        D1        D2        D3      BETA      IHIS
                                                                               3
*INITIAL_STRESS_SOLID
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$# EID/SID      NINT     NHISV     LARGE    IVEFLG    IALEGP    NTHINT    NTHHSV
         1         1        27         1
$#           SIGXX           SIGYY           SIGZZ           SIGXY           SIGYZ

$#           SIGZX             EPS           HISV1           HISV2           HISV3
                                                q1              q2              q3
$#           HISV4           HISV5           HISV6           HISV7           HISV8
              q4              q5              q6             c11             c12
$#           HISV9          HISV10          HISV11          HISV12          HISV13
             c13             c14             c15             c16             c22
$#          HISV14          HISV15          HISV16          HISV17          HISV18
             c23             c24             c25             c26             c33
$#          HISV19          HISV20          HISV21          HISV22          HISV23
             c34             c35             c36             c44             c45
$#          HISV24          HISV25          HISV26          HISV27          HISV28
             c46             c55             c56             c66
```

Comment: Orientation vectors (red) and stiffness matrix (blue) for IHIS=3.



Comment: First vector of material direction plotted in Ansys LS-PrePost.

Material characterization

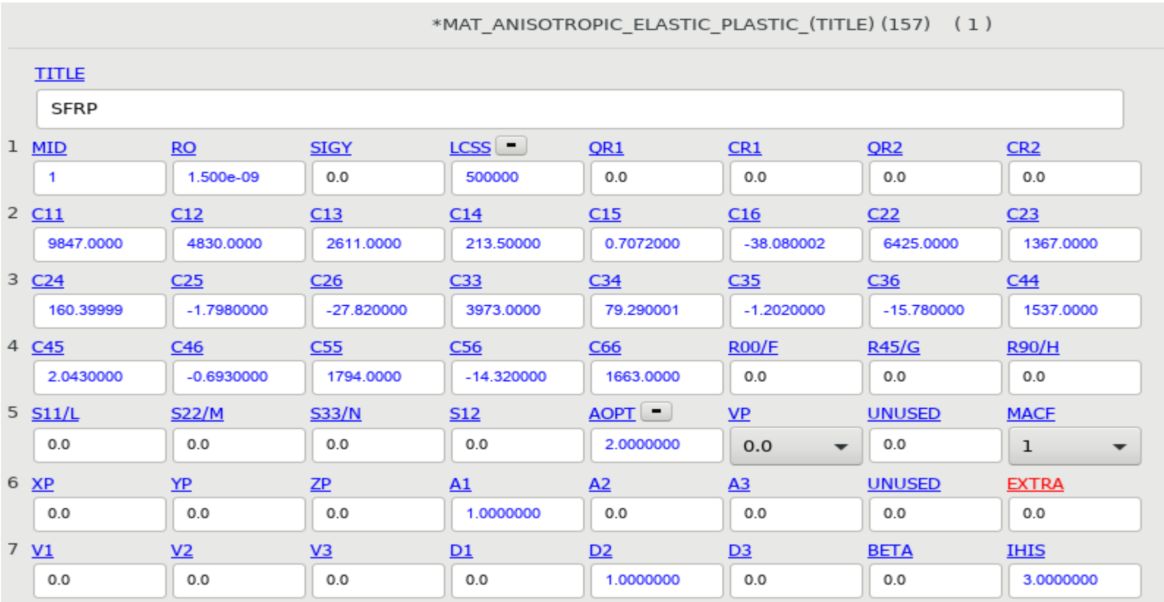
The hardening parameters on the material card are based on physical tension tests of specimens, which means 0° specimen (fibers aligned with specimen), 45° specimen and 90° specimen (fibers oriented transverse specimen). A CT-scan can be used to measure the detailed distribution of fibers to form an average orientation degree. The testing can also be done at different strain rates to calibrate strain rate dependency, but only static data is used in this example. If CT-scan results of the fiber orientation degree are to be used in the calibration, they need to be either mapped from three-dimensional scan data using a mapping tool, such as Envyo, or assumed as non-varying through the thickness of the specimen. Both options require a homogenization step of the elastic properties though. Fiber orientation distribution through the thickness can also be accomplished by performing an Injection Molding simulation of the specimen. For the homogenization in this example, in which the fiber orientations were based on measurements, the Homogenization Tool in Ansa by BETA CAE, another Ansys Technology Partner, was used. Apart from the measured (and assumed constant over the specimen) orientation degree, remaining input parameters were the same as mentioned earlier for Envyo, i.e. the elastic properties of the individual fibers and the polymer matrix and density. Ansa outputs a homogenized stiffness tensor, which can be used directly in *MAT_157 when performing the tensile test simulations. The material hardening curve LCSS, which includes contributions from both matrix and fibers, can be calibrated against results from tensile tests in three directions. Since the material model is based on Hill's yield criteria, in the case of shells, the R-values may also be calibrated and for solids the parameters F/G/H/L/M/N may be calibrated. This part of the procedure can be viewed as a homogenization of the plastic properties. The software chosen for doing the optimization of the hardening curve was Ansys LS-OPT.

Control cards

*CONTROL_ACCURACY, INN=1, which complies with the setting that was chosen in Envyo when doing the mapping. *CONTROL_OUTPUT, TET10S8=1 so that all 10 nodes of the tetrahedral solids are output to d3plot. *DATABASE_EXTENT_BINARY, NEIPH=6 for the possibility to plot the material directions in LS-PrePost.

Results

The produced input cards for LS-DYNA are shown below together with a fringe plot of Von Mises stress.



*MAT_ANISOTROPIC_ELASTIC_PLASTIC_(TITLE) (157) (1)

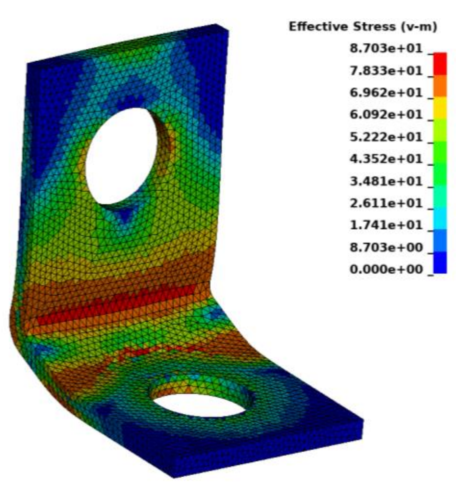
TITLE: SFRP

1	MID	RO	SIGY	LCSS	QR1	CR1	QR2	CR2
	1	1.500e-09	0.0	500000	0.0	0.0	0.0	0.0
2	C11	C12	C13	C14	C15	C16	C22	C23
	9847.0000	4830.0000	2611.0000	213.50000	0.7072000	-38.080002	6425.0000	1367.0000
3	C24	C25	C26	C33	C34	C35	C36	C44
	160.39999	-1.7980000	-27.820000	3973.0000	79.290001	-1.2020000	-15.780000	1537.0000
4	C45	C46	C55	C56	C66	R00/F	R45/G	R90/H
	2.0430000	-0.6930000	1794.0000	-14.320000	1663.0000	0.0	0.0	0.0
5	S11/L	S22/M	S33/N	S12	AOPT	VP	UNUSED	MACF
	0.0	0.0	0.0	0.0	2.0000000	0.0	0.0	1
6	XP	YP	ZP	A1	A2	A3	UNUSED	EXTRA
	0.0	0.0	0.0	1.0000000	0.0	0.0	0.0	0.0
7	V1	V2	V3	D1	D2	D3	BETA	IHIS
	0.0	0.0	0.0	0.0	1.0000000	0.0	0.0	3.0000000

*Comment: Material card (*MAT_157) for the SFRP.*

```
      9195           1          27           1           0           0           0           0
               0.0                   0.0                   0.0                   0.0                   0.0
               0.0                   0.0             0.5698915            -0.7453539            -0.3459353
         -0.086142              -0.47286             0.8769168              6649.625              2055.398
          1870.337                   0.0                   0.0                   0.0              4068.352
          1430.028                   0.0                   0.0                   0.0              2983.448
               0.0                   0.0                   0.0              1651.795                   0.0
               0.0              1047.936                   0.0                1473.1                   0.0
```

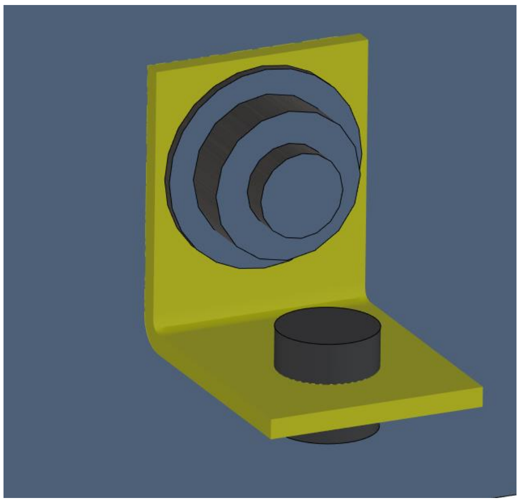
*Comment: *INITIAL_STRESS_SOLID for EID 9195, as an example.*



Comment: Fringe plot of Von Mises stress.

Example 10: Plastic bracket

This is an example of a thermoplastic bracket. Thermoplastics can be modelled on different levels in LS-DYNA depending on the complexity needed for the simulation at hand. Typically, the start point is *MAT_024 where only tensile test data is needed to calibrate the material model and from there the complexity can be increased. In this example *MAT_SAMP_LIGHT (*MAT_187L) is used, which can be considered a compromise choice between *MAT_024 and *MAT_SAMP-1 (*MAT_187) in terms of complexity and cost. For further information about simulation of plastics please contact your local distributor of Ansys LS-DYNA.



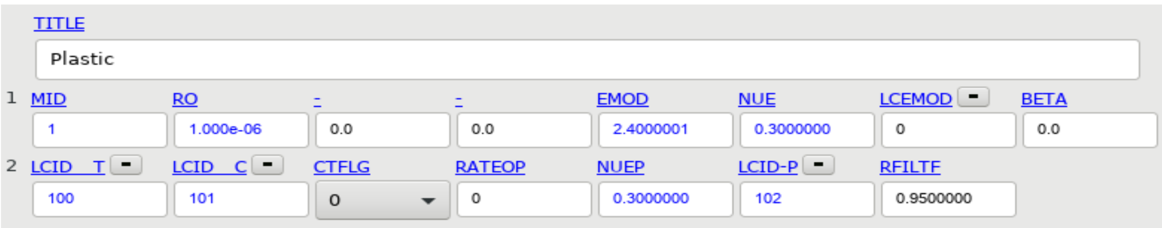
Simulation data

# nodes	80k
# elements	54k
Timestep size (ms)	0.2e-3 (constant)
Termination time (ms)	20.0
Solution time (minutes)	20

Elements and material

The plastic bracket is modelled with 10-noded tetrahedron elements type 16. The number of integration points is set by NIPTETS=4 on CONTROL_SOLID. Commonly used material models for plastics are *MAT_024, *MAT_124, *MAT_187 and *MAT_187L. *MAT_024 can account for strain rate dependent plastic hardening (viscoplasticity) with symmetry in terms of tension/compression. *MAT_124 is like *MAT_024 but also offers the choice to have asymmetric viscoplasticity in compression and tension. Both these material models use a constant plastic Poisson's ratio of 0.5, which is typically not true for plastics. *MAT_187 and *MAT_187L can also handle asymmetric viscoplasticity, but in this case with a constant or variable plastic Poisson's ratio to allow for the material to be compressible during plastic deformation. Both these models also include functionality for viscoelasticity. To utilize the full capacity of *MAT_187 test data for tension, compression, shear and biaxial tests is needed for the calibration. *MAT_187L is a simplified and faster version of *MAT_187 in terms of functionality and CPU cost. For instance, only test data for tension and compression is needed for the calibration. It is worth mentioning that *MAT_187 does include options for damage and failure, which is not the case with *MAT_187L. However, it can be added through *MAT_ADD_EROSION or *MAT_ADD_DAMAGE. Due to differences between *MAT_187 and *MAT_187L material curves can typically not be directly transferred between the material models, which means a recalibration is needed.

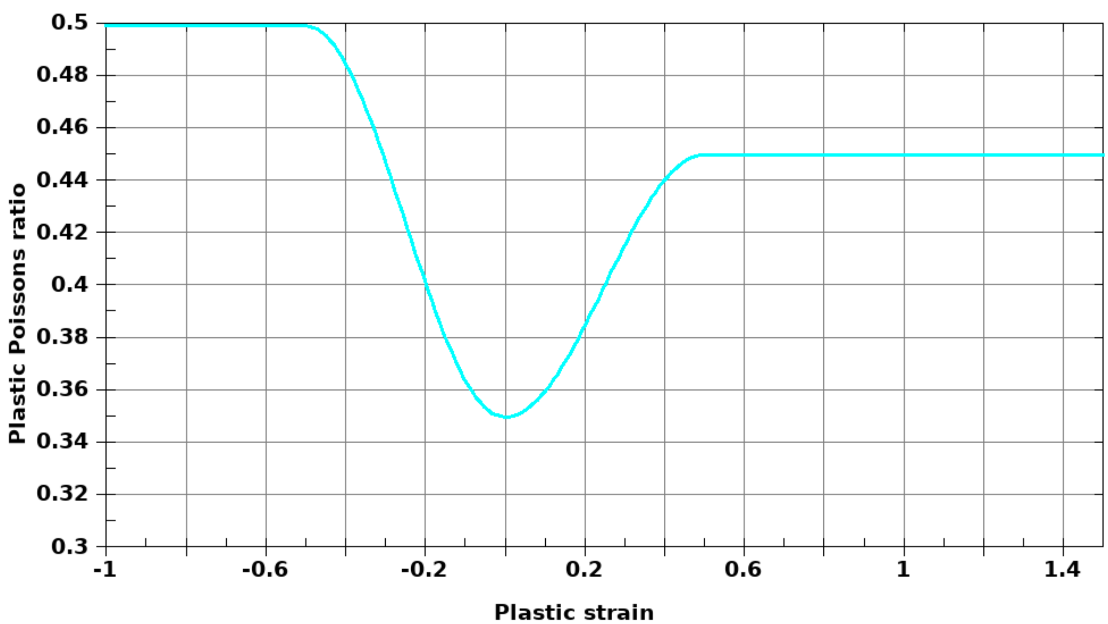
The material card used for *MAT_187L is shown below. LCID-T/LCID are the plastic hardening curves in tension and compression, respectively. LCID-P is the plastic Poisson's ratio as function of plastic strain, and it covers both compression and tension.


TITLE

Plastic

1	MID	RO	-	-	EMOD	NUE	LCEMOD	BETA
	1	1.000e-06	0.0	0.0	2.4000001	0.3000000	0	0.0
2	LCID_T	LCID_C	CTFLG	RATEOP	NUEP	LCID-P	RFILTF	
	100	101	0	0	0.3000000	102	0.9500000	

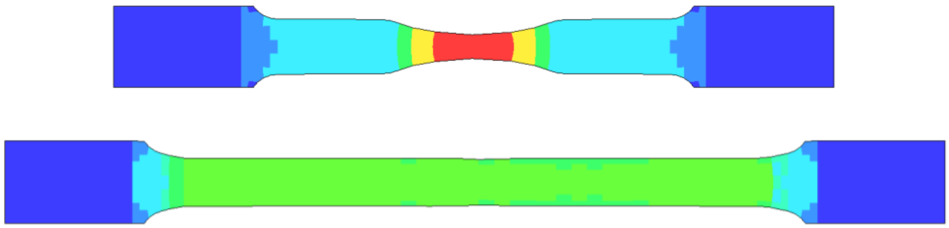
*Comment: The material card for *MAT_SAMP_LIGHT used in this example.*



Comment: Poisson's ratio curve, LCID-P.

*MAT_187L versus *MAT_024

The plastic Poisson's ratio is constant for *MAT_024, whereas it may vary for *MAT_187L if given by a load curve. This is a significant difference, which is illustrated by comparing the results from a tension test. When the test specimen reaches the necking point for *MAT_024 the necking is local, whereas for *MAT_187L the deformation is stable and extended over the gauge length. When local necking happens for *MAT_024 the load bearing capacity decreases rapidly, whereas *MAT_187L may continue to carry load. By using a steep hardening curve, the behavior of *MAT_024 can be made more like *MAT_187L. The steep curve is then, artificially, compensating for the constant plastic Poisson's ratio.



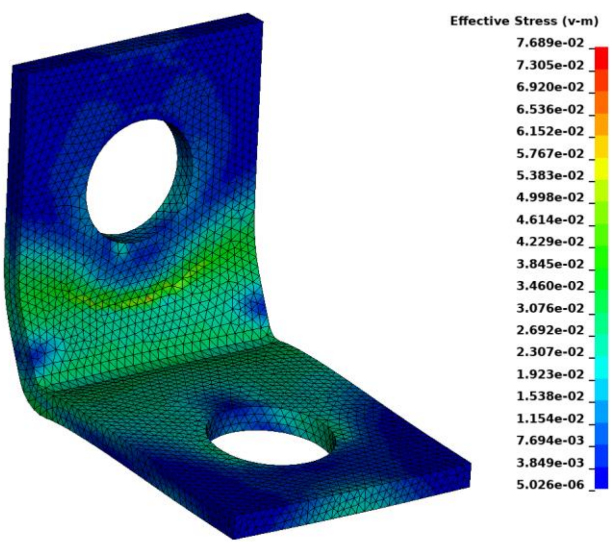
*Comment: Fringe plot of Von Mises stress. Upper: *MAT_024, Lower: MAT_187L.*

Control cards

*CONTROL_OUTPUT, TET10S8=1 so that all 10 nodes of the tetrahedral solids are output to d3plot.
*DATABASE_EXTENT_BINARY, NEIPH=7 for the possibility to plot additional data for *MAT_187L.

Results

A fringe plot of the Von Mises stress is shown below.



Comment: Fringe plot of Von Mises stress.

Record of revisions

Rev. no	Release date	Author	Comment
1.0	2019-03-21	Klas Engstrand Anders Bernhardsson	Release of document.
1.1	2019-10-21	Klas Engstrand	Example 8 added.
1.2	2021-08-20	Klas Engstrand David Aspenberg	General update. Example 9 added. Example 10 added.
1.3	2023-02-01	Klas Engstrand	“Element formulations” added.
1.4	2023-02-15	Klas Engstrand	Tied contacts added in section “Contact cards”
1.5	2024-03-27	Klas Engstrand Jimmy Forsberg	General update. “Database cards” added.

Copyright and Trademark Notice

Keywords: Ansys LS-DYNA; explicit;

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