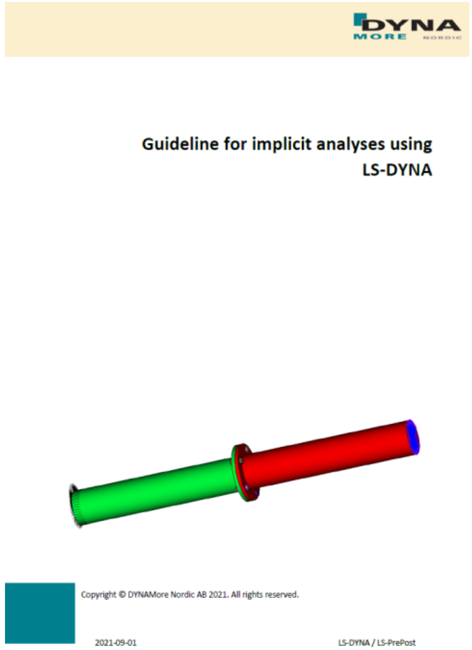
Guideline for implicit analyses using LS-DYNA –
Latest revision and some details on control card settings


Guideline for implicit analyses using
LS-DYNA

Copyright © DYNAMore Nordic AB 2021. All rights reserved.

2021-09-01

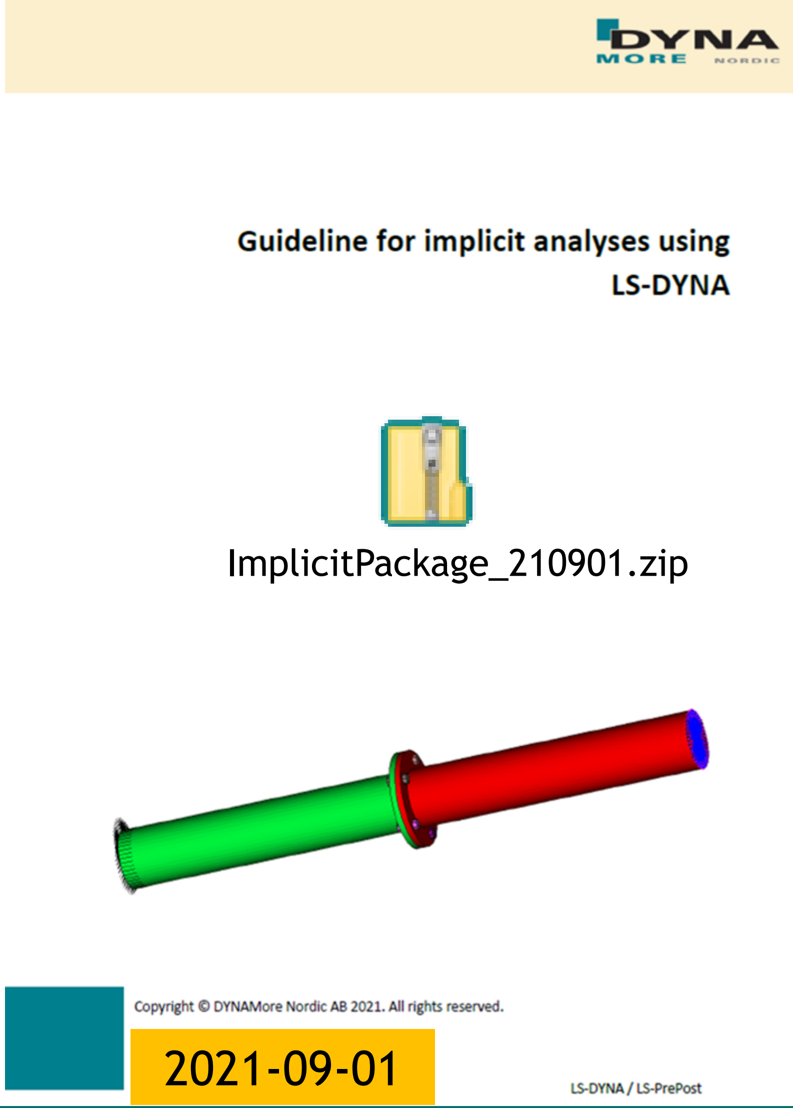
LS-DYNA / LS-PrePost

Anders Jonsson, DYNAmore Nordic AB

Overview

- Introduction to the Guideline
- Implicit set up
- News in the latest revision
- Some details on control card settings
- Damping in implicit dynamics
- Load history management using `*CASE` and `dynain.lsda`
- Clarifications on elements for linear analyses

Guideline for implicit analyses in LS-DYNA


Guideline for implicit analyses using LS-DYNA

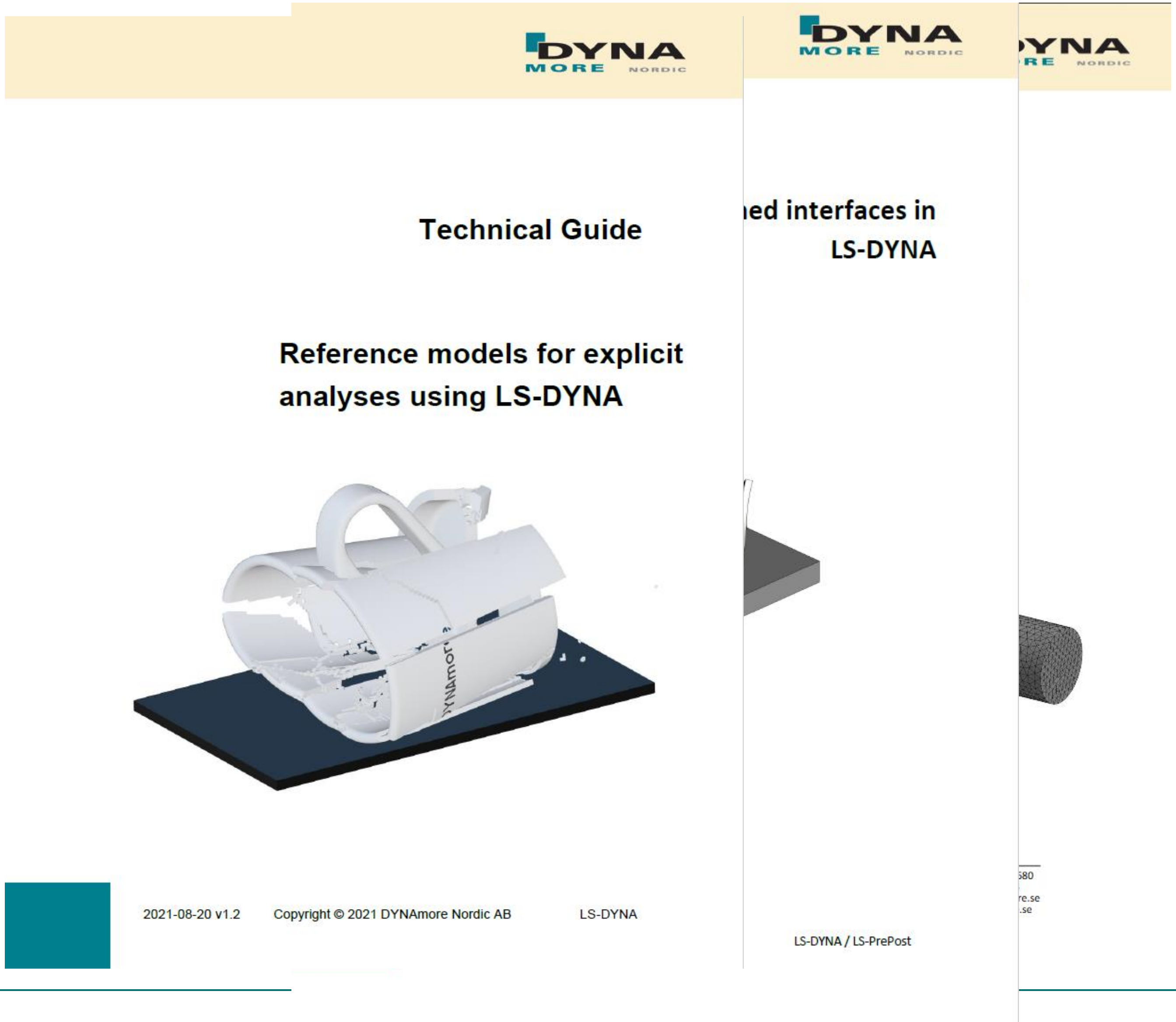
ImplicitPackage_210901.zip

2021-09-01

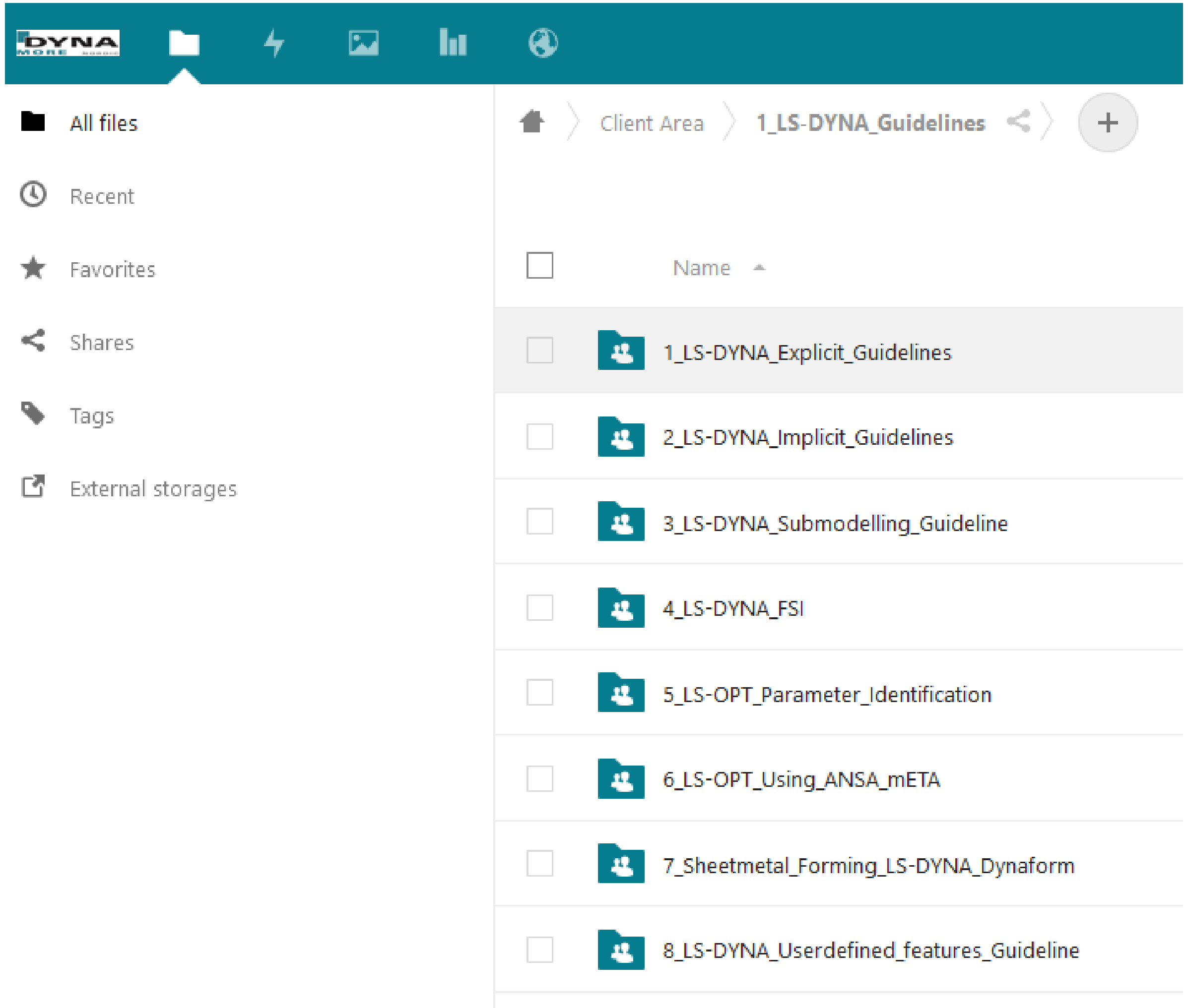
LS-DYNA / LS-PrePost

- Help for getting started with LS-DYNA Implicit
  - Reference guide in everyday work
- How to set up common analysis types
- Recommended element types, contacts etc.
- Small example models
- Recommended control cards prepared in keyword files for direct use via `*INCLUDE`

Other guidelines ...



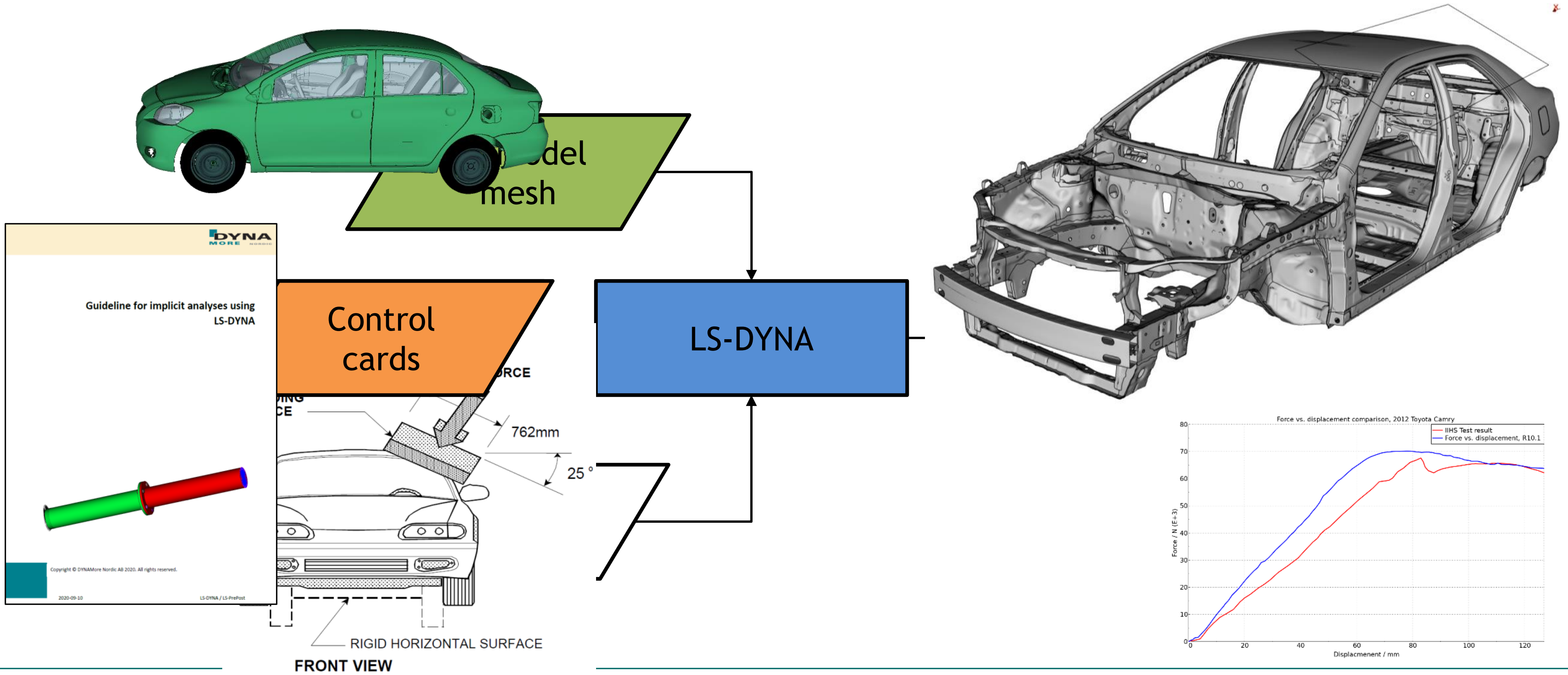
Download guidelines from files.dynamore.se > Client Area



- A living document
- Updates 1 – 2 times / year
- New revisions announced in DYNAmore Nordic News

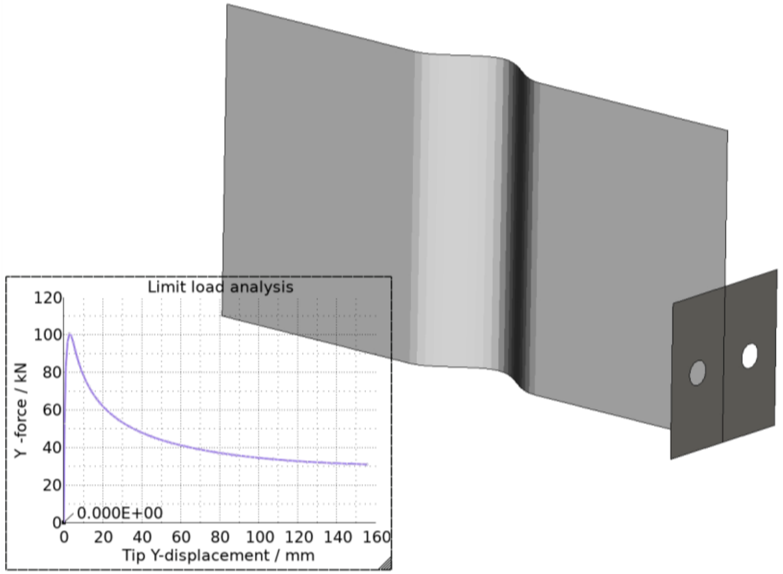
Implicit set-up in LS-DYNA

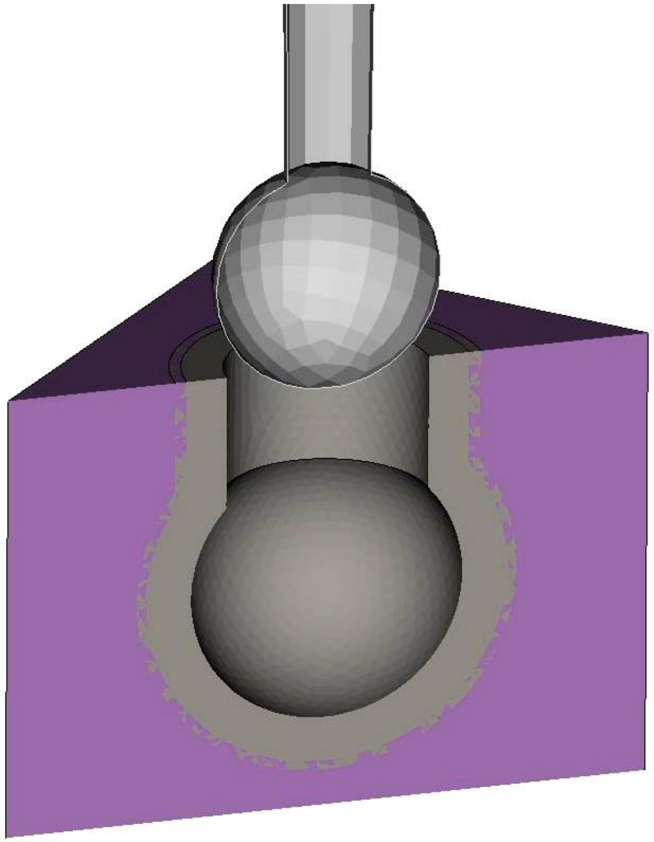
- The guideline helps set-up implicit analyses with minimal effort



The “Guideline for implicit analyses in LS-DYNA” provides ...

- A starting point for implicit analyses
- Ready-to-use keywordfiles containing suggested control card settings
  - Plug-and-play by `*INCLUDE`
- Description of how to set up common implicit analysis types
  - Linear static
  - Non-linear static
  - Implicit dynamics
  - Frequency domain
  - etc.
- About 20 small example models
- Recommendations regarding
  - contacts
  - element formulations
  - material models
- Troubleshooting guide for solving convergence issues
- Topics regarding Implicit / Explicit





Implicit set-up in LS-DYNA

- Other sources of useful information regarding implicit analyses in LS-DYNA:
  - www.dynasupport.com/howtos/implicit
  - http://www.dynaexamples.com/implicit
  - www.dynalook.com – search European conference papers
- Dynamore / Ansys LST also gives courses in implicit analyses:
  - Non-linear implicit analysis in LS-DYNA
  - NVH & Frequency domain analysis in LS-DYNA
  - From explicit to implicit
  - Implicit analysis using LS-DYNA (DYNAmore Germany)
- see also: https://www.dynamore.se/en/training/seminars and https://www.dynamore.de/en/training/seminars

- A very educational Webinar from Christoph Schmied, DYNAmore Germany:
  YouTube https://www.youtube.com/watch?v=7SL321fO7_4&t=781s

- Please report bugs and errors in the Guideline package! To support@dynamore.se

Implicit set-up in LS-DYNA – Control cards

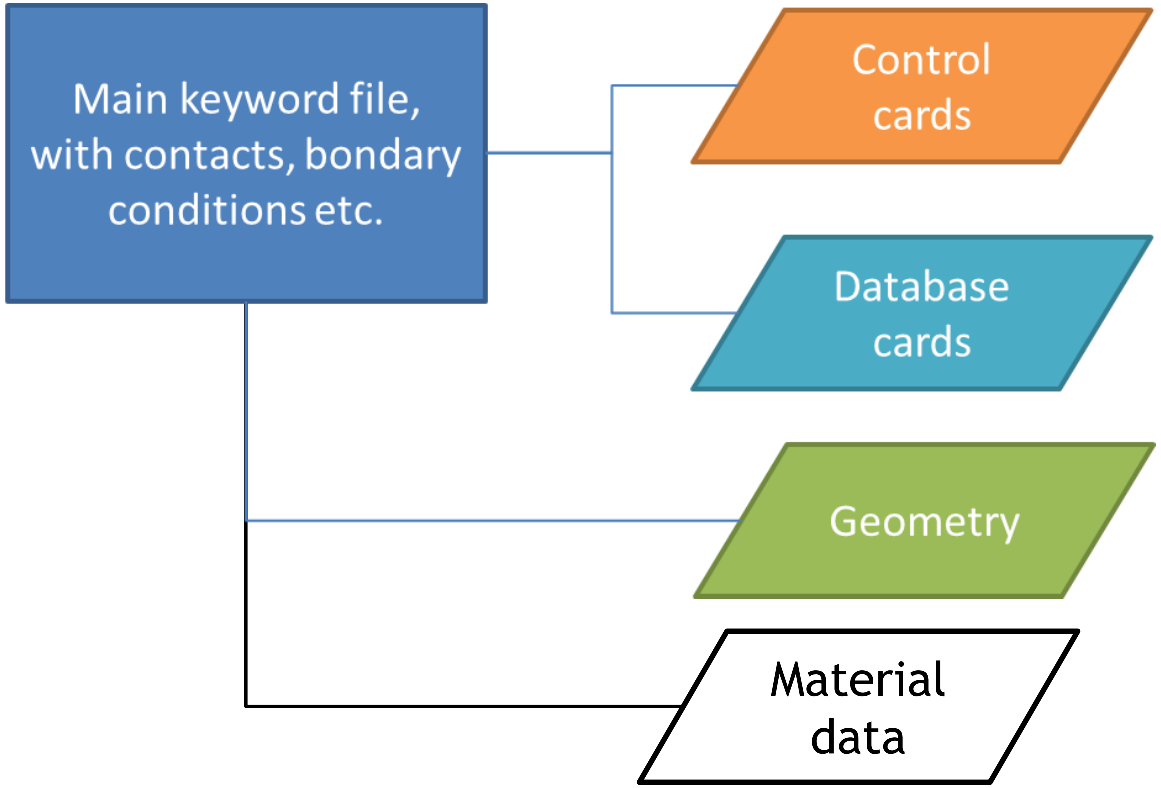
- Identify analysis type and select appropriate control card include file.
  - In many cases, *CONTROL_TERMINATION is the only required additional control card.
- Use an include file structure! Then the control card files can be used directly.

Analysis type	Include files		Add keywords:(*)
	control_cards_...	database_cards_...	*CONTROL_IMPLICIT_...
Linear static	linear.key	static.key	
Non-linear static	nonlin.key	static.key	
Linear buckling	(1)	static.key	BUCKLE
Non-linear postbuckling	arc.key	static.key	
Eigenfrequency analysis	(1)	static.key	EIGENVALUE
Linear transient modal dynamics	linear.key	dynamic.key	MODAL_DYNAMIC
Frequency domain analyses(2)	linear.key		
Non-linear implicit dynamics	nonlin.key	dynamic.key	DYNAMICS

Notes: (*) *CONTROL_TERMINATION must always be added. (1) Can be part of both linear and non-linear analysis. (2) Frequency response functions, steady state dynamics etc.

Implicit set-up in LS-DYNA – Control cards

- Identify analysis type and select appropriate control card include file.
  - In many cases, `*CONTROL_TERMINATION` is the only required additional control card.
- Use an include file structure! Then the control card include files can be used directly.



Keyword-based (bottom-up) approach

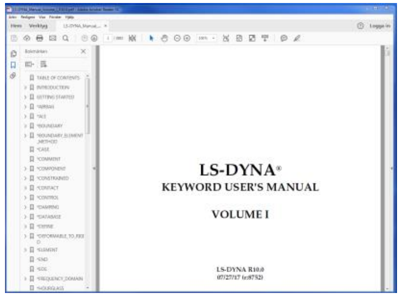
```
$ 190402, test linear stiffness analysis
*KEYWORD
$===============================================================================
$ Database and control cards
$===============================================================================
*INCLUDE
/disk/appo/Implicit/andersebj/Cards/control_cards_linear.key
*INCLUDE
/disk/appo/Implicit/andersebj/Cards/database_cards_static.key
*CONTROL_MPP_IO_NODUMP
*CONTROL_TERMINATION
       8.0
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
        -1         1
                                        1

$===============================================================================
$ Geometry
$===============================================================================
*INCLUDE
Camry_V1_NoSusAndPowerTrain_impl9.k
*INCLUDE
../ORG01/download_implicit_roofcrush/xtra_sw.k
*INCLUDE
../ORG01/download_implicit_roofcrush/weld7.k
*INCLUDE
../ORG01/download_implicit_roofcrush/roof_welds.k
$
$===============================================================================
$ Loads and boundary conditions
$===============================================================================
*DEFINE_CURVE_TITLE
Zero cure
  19000001
0.,0.
10.,0.
*DEFINE_CURVE_TITLE
Ramp front left
  19000002
0.,0.
1.,1.
2.,0.
```

LS-DYNA Implicit

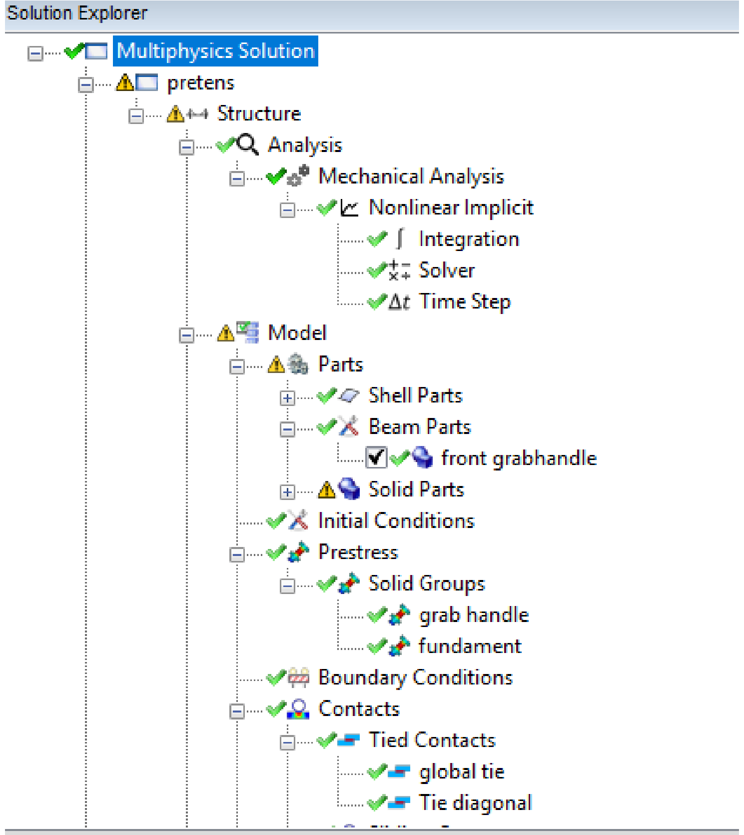
Keyword approach

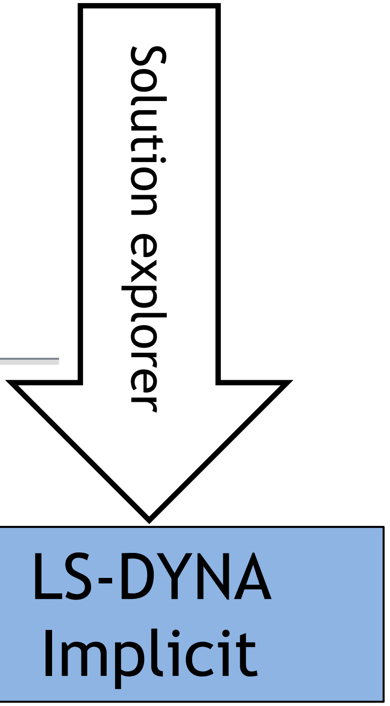
Text editor / (vi, notepad++)



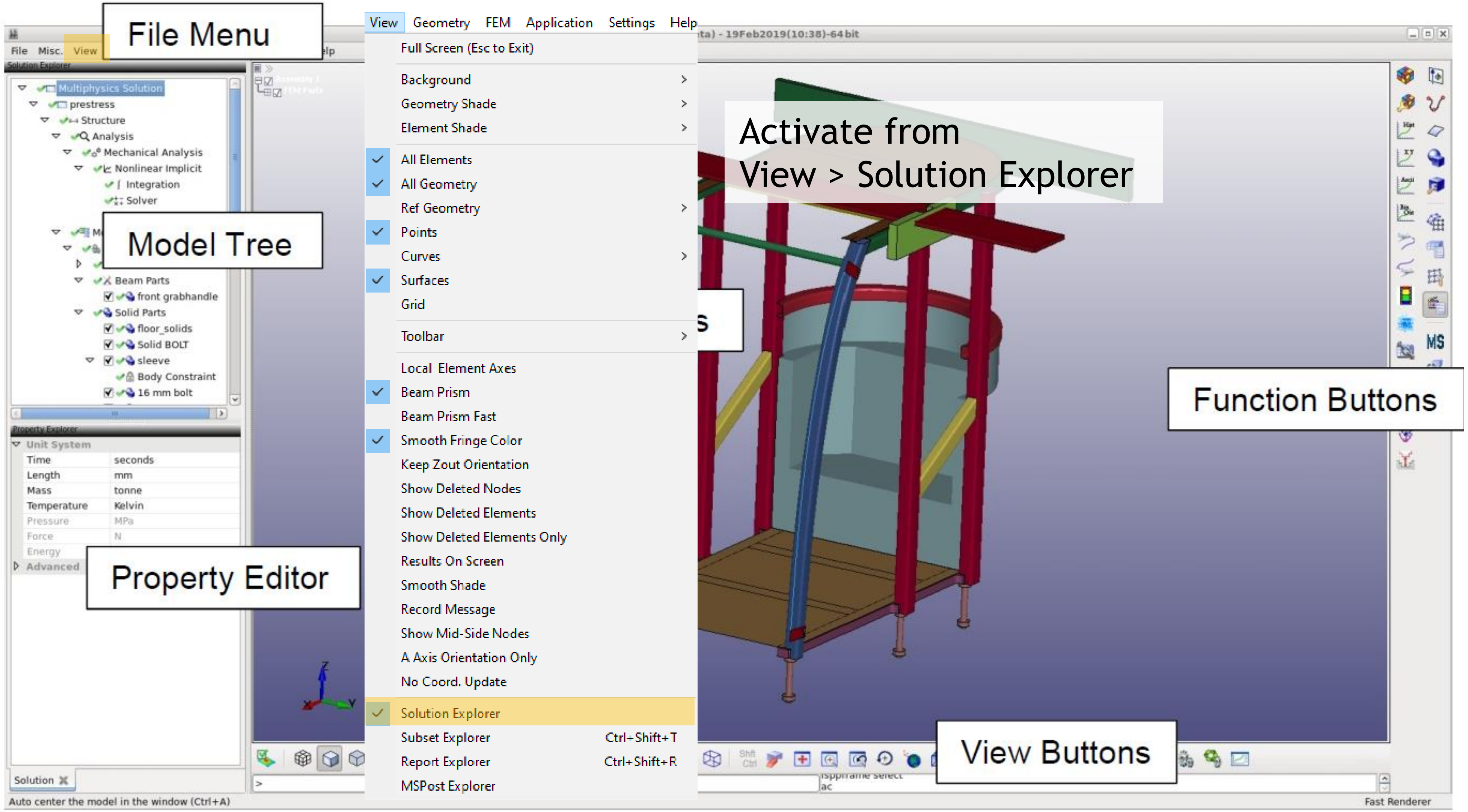
The Solution Explorer in LS-PrePost

- Top-down approach
- Work with general engineering terms
  - Not so much use of specific LS-DYNA keywords
- GUI-based
- Will give recommended
  - control card settings
  - contacts
  - element types
  - materials



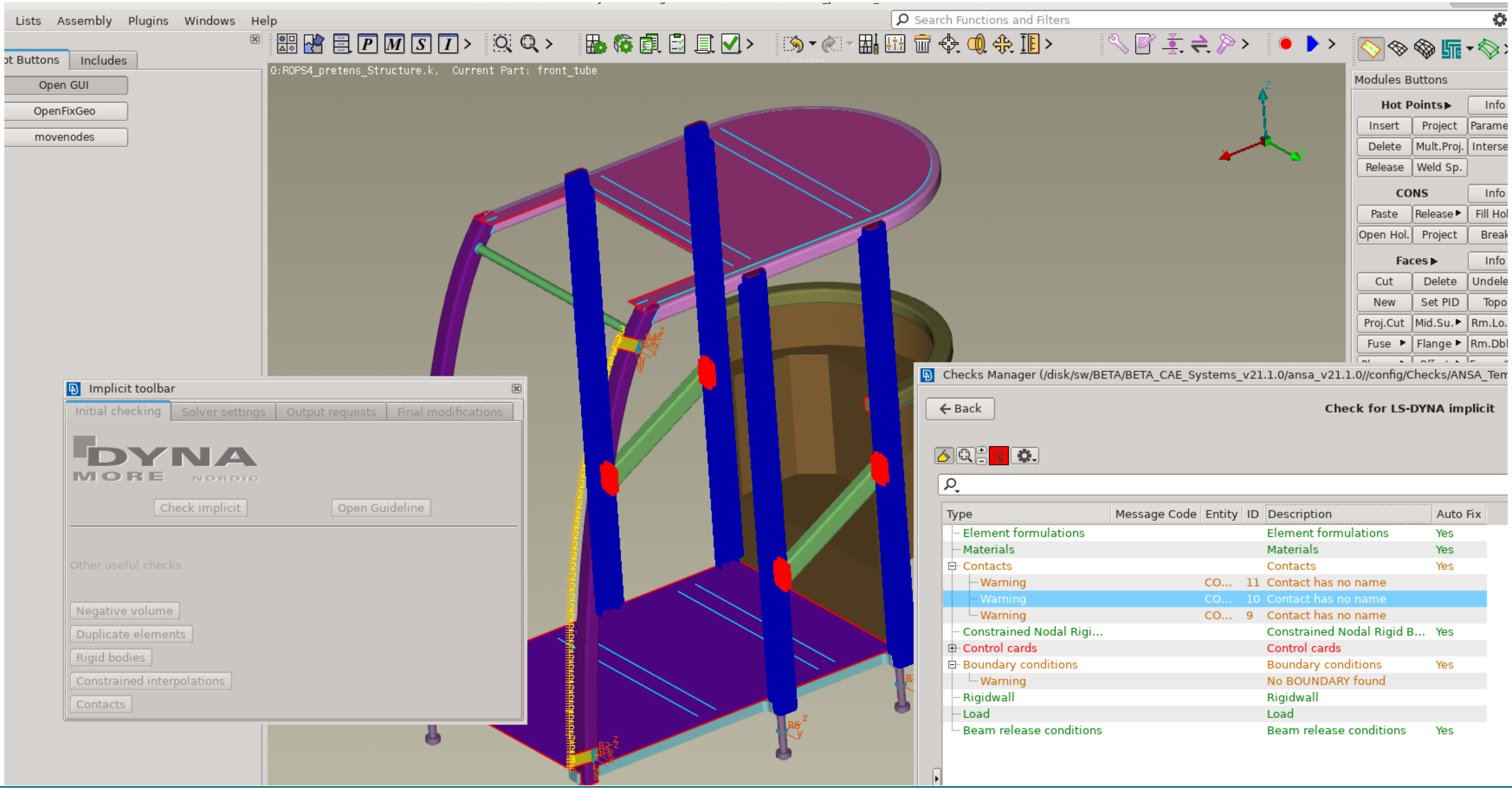


The Solution Explorer in LS-PrePost



DYNAmore Implicit toolbar for ANSA

- Help with model check and set-up



News in Revision 2021-09-01

- Updated control card files
  - *DNORM* = 1 on `*CONTROL_IMPLICIT_SOLUTION`
  - *IGS* = 2 (default) on `*CONTROL_IMPLICIT_GENERAL`
- Added section on damping in implicit dynamics
- Updated Appendix C
  - Improved and extended troubleshooting
- Added section on `*CASE` and `dynain.lsda`
  - New approach to load history management
- Minor corrections and improvements
  - Clarifications on elements for linear analyses
- This revision is intended for use with LS-DYNA R10.1, R10.2, R11.1, R12.0.0

Implicit set-up “under the hood”

- *CONTROL_IMPLICIT_GENERAL
  - IGS for the Geometric stiffness
- *CONTROL_IMPLICIT_SOLUTION
  - Linear or non-linear solution
  - Convergence measures
  - Full Newton vs. quasi-Newton (BFGS)
  - Optionally activate Arc-length solver
- *CONTROL_IMPLICIT_AUTO
  - Time step size control
  - Load curve specifying key points
- *CONTROL_IMPLICIT_SOLVER
  - Activating additional printout from the linear solver
  - Use the non-symmetric linear solver

Modifications of “factory defaults” are recommended

The stiffness matrix – geometric contribution

position 5, **IGS=1** includes the geometric stiffness

```
*CONTROL_IMPLICIT_GENERAL
$    IMFLAG       DT0                                  IGS
                                                         1
```

- The stiffness matrix is composed of a material and geometric part
  - $\boldsymbol{K} = \boldsymbol{K}_{\text{mat}} + \boldsymbol{K}_{\text{geo}} + \cdots = \int \boldsymbol{B}^T \boldsymbol{C} \boldsymbol{B} + \int \boldsymbol{\beta}^T \boldsymbol{\sigma} \boldsymbol{\beta} + \cdots$
- The material part depends on the material properties $\boldsymbol{C}$
  - fairly insensitive to small geometric changes
  - always included in the computations
  - often positive definite (positive eigenvalues)
- The geometric part depends on the stress $\boldsymbol{\sigma}$
  - small geometric changes impacts the matrix
  - optionally included in the computations because
  - compressive stress gives negative contributions
- The full stiffness matrix senses buckling, bifurcations, instabilities by an indefinite or singular matrix, but for a nonlinear simulation this may deteriorate convergence

The stiffness matrix – geometric contribution

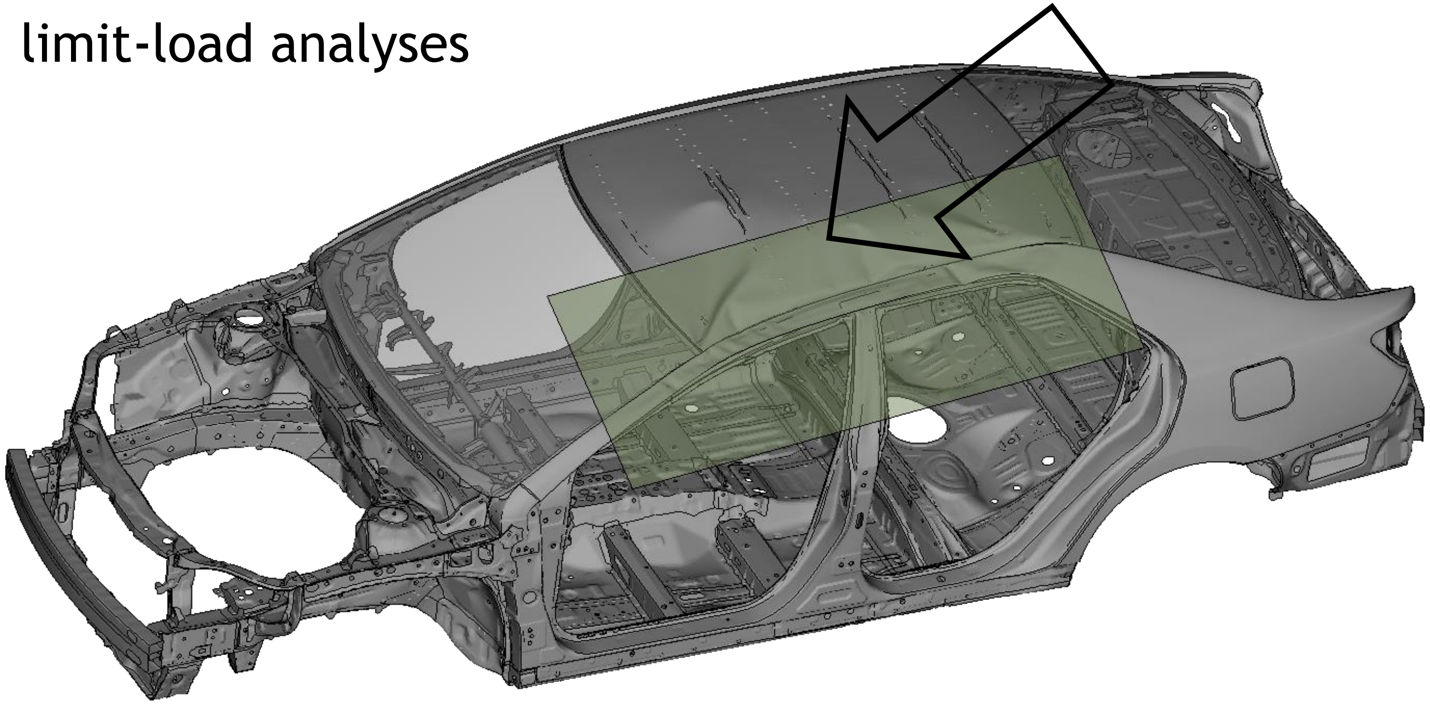
position 5, **IGS=-PSID** includes the geometric stiffness for a part set

```
*CONTROL_IMPLICIT_GENERAL
$    IMFLAG        DT0                                   IGS
                                                        -123
```

- The geometric part depends on the stress $\boldsymbol{\sigma}$
- New options to deal with the geometric stiffness, from R12.0.0
  - Set **DT0<0** to eliminate negative principal stresses in geometric (initial stress) stiffness. Initial time step is |DT0|
  - Apply IGS to a part set only, by setting **IGS=-PSID** May be useful for bolt pre-tensioning

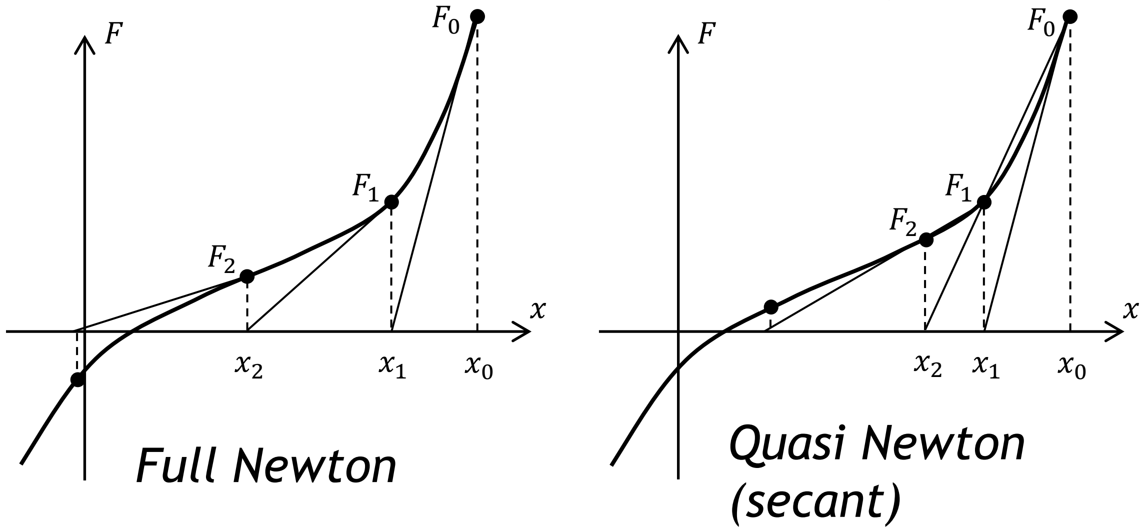
Newton method – BFGS parameters

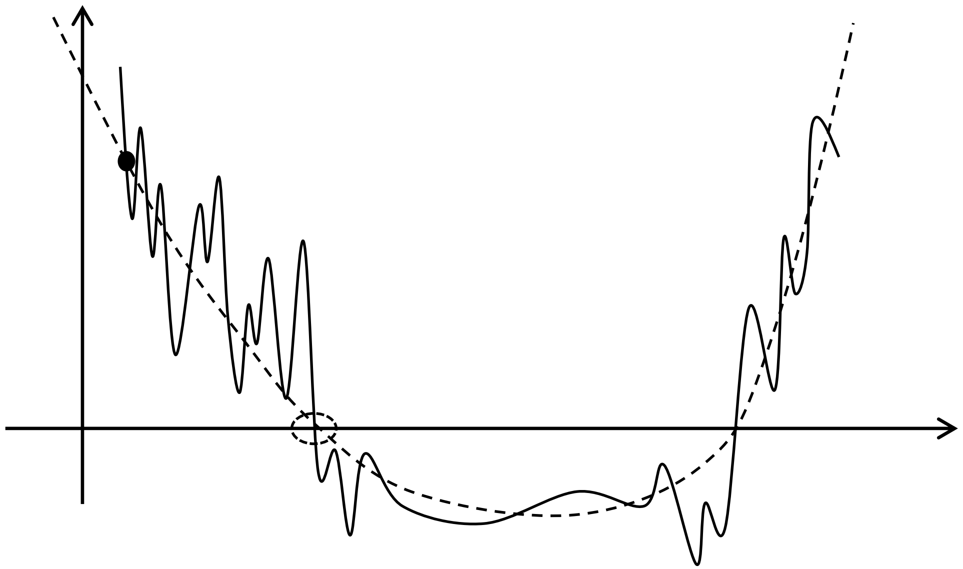
- In some cases, when the model response is highly non-linear, switching to a full-Newton solution may be beneficial for convergence. The complete stiffness matrix is re-formed in each iteration.
  - Large changes in contact state in each iteration
  - Buckling, limit-load analyses



from www.dynaexamples.com/implicit

Newton method – the 1D analogue





- Full Newton converges faster in theory
  - Quadratic convergence if $\boldsymbol{K}$ is exact and $\boldsymbol{x}$ is inside ”convergence radius”
- In practice problems are far from academic
  - $\boldsymbol{K}$ is never exact and $\boldsymbol{x}$ is rarely inside ”convergence radius”, due to
    - Complex materials
    - Contact with friction
    - Geometric (initial stress) contribution
    - Geometry dependent loads
- Quasi Newton methods tends to correct *small* errors in the stiffness matrix and assumes it does not change much between iterations
  - Constant change of contact state might require full Newton
- Outside the convergence radius, an approximate stiffness matrix might do good

Newton method – BFGS parameters

```
*CONTROL_IMPLICIT_SOLUTION
$    NSOLVR    ILIMIT    MAXREF     DCTOL     ECTOL     RCTOL     LSTOL    ABSTOL
         12        11        15     0.001      0.01      1.E10       0.9    1.E-20
$     DNORM    DIVERG     ISTIF   NLPRINT    NLNORM    D3ICTL
          1         1         1         3         4         1
```

- Default method is BFGS
  - Approximate update to stiffness matrix factors
- Stiffness matrix is reformed every **`ILIMIT`** step
  - Stiffness matrix is also reformed at divergence, energy explosions and zero line search
  - When not reformed, a rank-2 correction (BFGS) is applied to the existing factored matrix
  - Set **`ILIMIT=1`** if full Newton is desired
- If more than **`MAXREF`** reformations are needed the step is abandoned
  - RETRY: reduce step size and try again
  - Choose **`MAXREF`** with care to avoid spending too much time on failed steps

Newton method – BFGS parameters

```
*CONTROL_IMPLICIT_SOLUTION
$   NSOLVR    ILIMIT    MAXREF     DCTOL     ECTOL     RCTOL     LSTOL    ABSTOL
        12         1        55     0.001      0.01     1.E10       0.9    1.E-20
$    DNORM    DIVERG     ISTIF   NLPRINT    NLNORM    D3ICTL
         1         1         1         3         4         1
```

- Stiffness matrix is reformed every **ILIMIT** step
  - Set **ILIMIT=1** if full Newton is desired
- If more than **MAXREF** reformations are needed the step is abandoned
  - RETRY: reduce step size and try again
  - Choose **MAXREF** with care to avoid spending too much time on failed steps
- Also adjust ITEOPT and ITEWIN of *CONTROL_IMPLICIT_AUTO accordingly

Newton method – displacement tolerance parameters

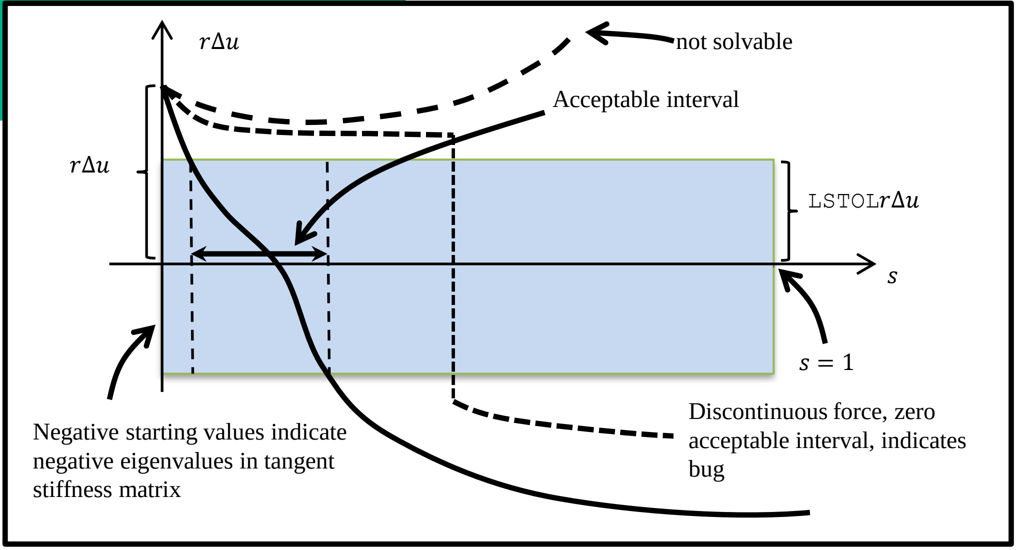
```
*CONTROL_IMPLICIT_SOLUTION
$   NSOLVR    ILIMIT    MAXREF     DCTOL     ECTOL     RCTOL     LSTOL    ABSTOL
        12        11        15     0.001      0.01      1.E10       0.9    1.E-20
$    DNORM    DIVERG     ISTIF   NLPRINT    NLNORM    D3ICTL
         1         1         1         3         4         1
```

- Norm of incremental displacment, $\|\Delta\boldsymbol{u}\|$, is the important convergence measure
  - Others are in practice disabled by default
- Two variants of relative displacement tolerance parameters
  - For **DNORM=2** (default), $\|\Delta\boldsymbol{u}\| \leq \mathrm{DCTOL}\,\left\|\boldsymbol{x}_i - \boldsymbol{x}_0 + \boldsymbol{u}_{j+1}\right\|$, relative to accumulated displacement so far
  - For **DNORM=1** (recommended), $\|\Delta\boldsymbol{u}\| \leq \mathrm{DCTOL}\,\left\|\boldsymbol{u}_{j+1}\right\|$, relative to displacment during step
  - Latter is more strict and generally recommended, mandatory for cyclic problems
- Usually recommended to *not* increase this value
  - Even though it is tempting to believe it will speed up convergence, it may give the opposite effect since each step may start from a ”bad” equilibrium point

Newton method – line search

```
*CONTROL_IMPLICIT_SOLUTION
$   NSOLVR    ILIMIT    MAXREF     DCTOL     ECTOL     RCTOL     LSTOL    ABSTOL
        12        11        15     0.001      0.01     1.E10       0.9    1.E-20
$    DNORM    DIVERG     ISTIF   NLPRINT    NLNORM    D3ICTL
         1         1         1         3         4         1
$

$    LSMTD
         5
```



- A smooth force will provide a large confidence interval
- If line search is not solvable, or confidence interval is small, this indicates a poor response from structure
  - Worst case, a bug
- **LSMTD=5** invokes a robust line search
  - Limits the residual force in each step

Newton method – more on convergence

```
*CONTROL_IMPLICIT_SOLUTION
$   NSOLVR    ILIMIT    MAXREF     DCTOL     ECTOL     RCTOL     LSTOL    ABSTOL
        12        11        15     0.001      0.01     1.E10       0.9    1.E-20
$    DNORM    DIVERG     ISTIF   NLPRINT    NLNORM    D3ICTL
         2         1         1         0         4         0
```

- **`ABSTOL`** is used internally to detect convergence when forces are small
  - Generally recommended to use **`ABSTOL=1.e-20`** to avoid premature convergence
  - **`ABSTOL`** negative means a check on the residual norm
- **`NLNORM`** governs how inner products are computed
  - Default is to only consider translational degrees of freedom
  - Use **`NLNORM=4`** which weighs translational and rotational degrees of freedom using a length scale computed from the element sizes
  - **`NLNORM`** negative means that its absolute value overrides the length scale
- *All* convergence criteria must be satisfied simultaneously, *except* the absolute criteria which overrides the others
  - Convergence on maximum norms are optional

Additional convergence control based on maximum norms

- From R10.0, an optional line for strict convergence criteria, based on the max norm, has been added on `*CONTROL_IMPLICIT_SOLUTION`
- Activated by setting *DNORM* = -1 or -2
- Strict criteria must be met in addition to the standard convergence criteria *DCTOL*, *ECTOL*, *RCTOL*

Optional 2b	1	2	3	4	5	6	7	8
Variable	DMTOL	EMTOL	RMTOL		NTTOL	NRTOL	RTTOL	RRTOL
Type	Relative (unitless)				F	Absolute values		F
Default	0.0	0.0	0.0		0.0	0.0	0.0	0.0

Additional convergence control based on maximum norms

- From R10.0, an optional line for strict convergence criteria, based on the max norm, has been added on `*CONTROL_IMPLICIT_SOLUTION`
- Activated by setting `dnorm` = -1 or -2
- Strict criteria must be met in addition to the standard convergence criteria *DCTOL*, *ECTOL*, *RCTOL*
  - *DMTOL* - relative displacement max norm criteria
  - *EMTOL* - relative energy max norm criteria
  - *RMTOL* - relative residual max norm criteria
  - *NTTOL* - absolute nodal translational displacement max norm criteria
  - *NRTOL* - absolute nodal rotational displacement max norm criteria
  - *RTTOL* - absolute rigid body translational displacement max norm criteria
  - *RRTOL* - absolute rigid body rotational displacement max norm criteria

The stiffness matrix – nonsymmetric contribution

line 2, **LCPACK=3** includes the nonsymmetric contribution

```
*CONTROL_IMPLICIT_SOLVER
$   LSOLVR     LPRINT

$   LCPACK
         3
```

- Default is to use a symmetric stiffness matrix
  - Faster solution of the linear system
  - Saves memory
  - Most often sufficient
  - Requires symmetrization or neglection of non-symmetric contribution
- Nonsymmetric solver is optional and might help for
  - Contact when high (μ > 0.3) coefficients of friction are used
  - Geometric dependent loads
  - Anisotropic matierials ...

Advanced set up – summary of control card modifications

- *CONTROL_IMPLICIT_GENERAL
  - (Set *IGS* = 1, in some cases for example bolt pre-tension or tensile tests)
  - In most cases, no modification needed
- *CONTROL_IMPLICIT_SOLUTION
  - For highly non-linear problems, switch to full-Newton. Set *ILIMIT* = 1 and *MAXREF* = a reasonable value (30 - 100). Update *CONTROL_IMPLICIT_AUTO - settings accordingly
  - (Setting DNORM = 2 gives a more relaxed convergence criteria which may be useful for troubleshooting convergence problems)
- *CONTROL_IMPLICIT_SOLVER
  - In most cases, no modification needed
  - *LCPACK* = 3 for cases with high friction and stick/slip
  - (*LPRINT* = 3 for additional print out)

Damping in implicit dynamics

- Rayleigh damping – mass and stiffness damping
  - *DAMPING_PART_MASS, *DAMPING_PART_STIFFNESS
  - *DAMPING_GLOBAL
  - Leave *IRATE* = 0 on *CONTROL_IMPLICIT_DYNAMICS
- Frequency independent damping
  - *DAMPING_FREQUENCY_RANGE_DEFORM
- Numerical damping
  - From the time integration scheme

Rayleigh damping

- Rayleigh damping – mass and stiffness damping

$$M\ddot{x} + C\dot{x} + Kx = F$$

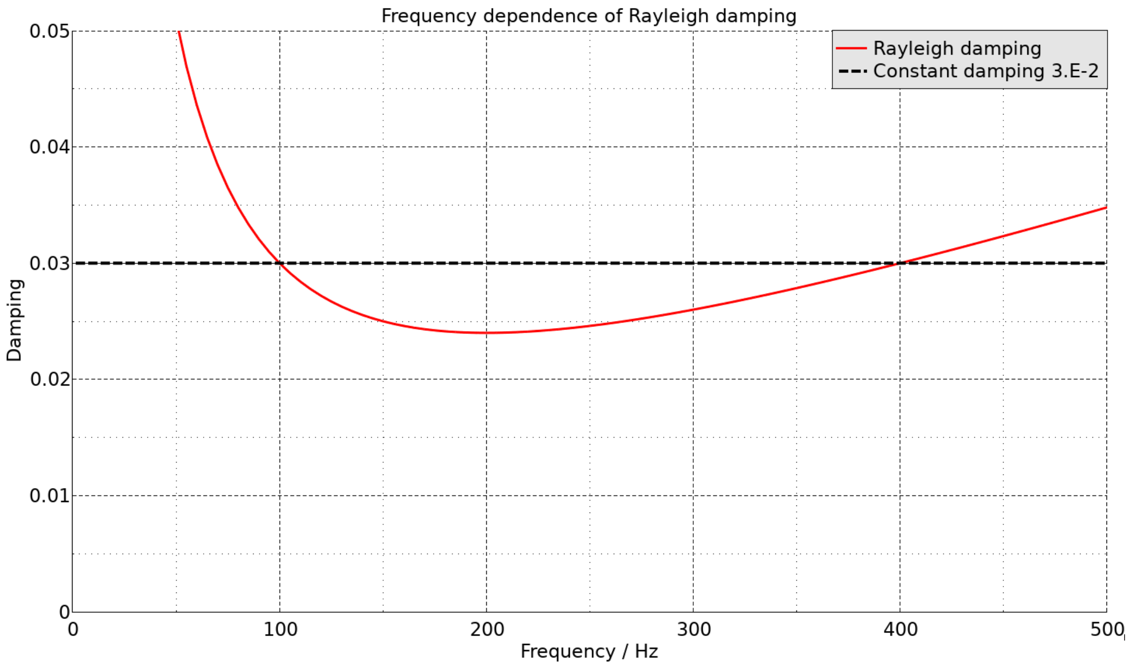
- Damping matrix is a linear combination of the stiffness and mass matrix

$$C = \alpha M + \beta K$$

  - NOTE: α, β are NOT unitless!

- Gives frequency dependent damping

$$2\xi = \frac{\alpha}{\omega} + \beta\omega$$



Rayleigh damping – parameter input

*DAMPING_PART_STIFFNESS, $COEFF = -\beta$

Rayleigh damping: $C = \alpha M + \beta K$

*DAMPING_GLOBAL, $VALDMP = \alpha$

*DAMPING_PART_MASS, $LCID*SF = \alpha$

- To apply mass damping to all parts, use *DAMPING_GLOBAL with $VALDMP = \alpha$
  - To apply mass damping to some parts in a model, use *DAMPING_PART_MASS
- To apply the classical (Rayleigh) stiffness damping, use *DAMPING_PART_STIFFNESS with $COEFF = -\beta$
  - $COEFF > 0$ is optimized for explicit analyses
- Note: α, β are NOT unitless!

Rayleigh damping – the stiffness part ...

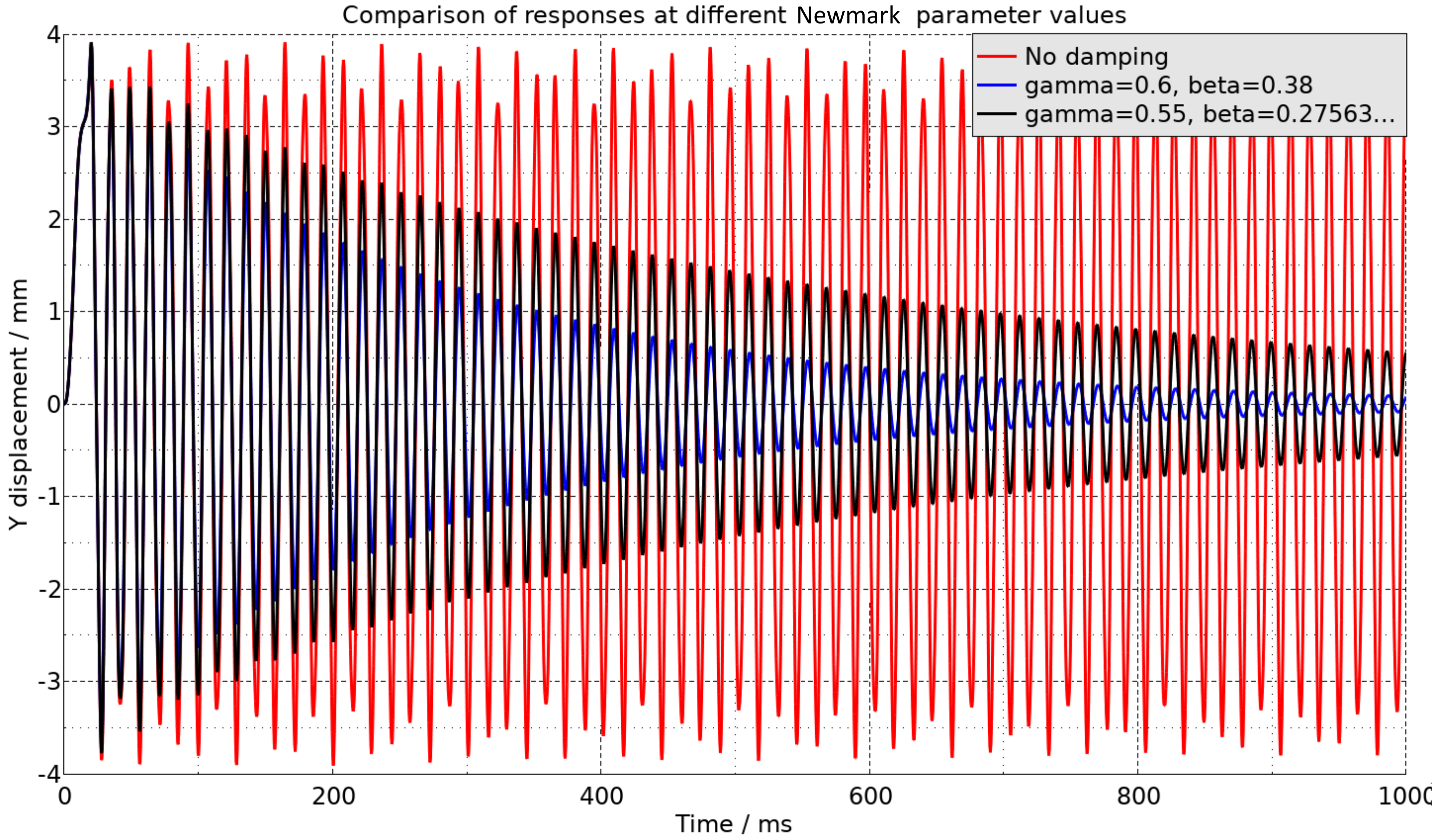
- Stiffness damping is implemented as a rate effect from the materials
- If rate effects are disabled by *IRATE* = 1 or 2 on `*CONTROL_IMPLICIT_DYNAMIC`, the stiffness damping, `*DAMPING_PART_STIFFNESS`, will also be disabled!
- To include the stiffness damping, leave *IRATE* blank on `*CONTROL_IMPLICIT_DYNAMIC`

Frequency indepdenent damping

- Approximately frequency independent damping can be specified by *DAMPING_FREQUENCY_RANGE_DEFORM
  - Input fraction of critical damping and frequency range F_{low} F_{high}
  - Best suited for low damping values (< 0.05)
  - Frequency ranges such that $10 \leq F_{\mathrm{high}}/F_{\mathrm{low}} \leq 300$.
- Will alter the dynamic stiffness of the structure slightly
  - Adjust Youngs modulus of involved materials to compensate
- May impose limit on the max time step in versions prior to R12.

Numerical damping

- Numerical damping
  - From the time integration scheme, *GAMMA* and *BETA* parameters
  - Default values *GAMMA* = 0.5 and *BETA* = *0.25* give very little numerical damping
  - Will also depend on time step size, and lowest eigenfrequency of the structure
  - Artificial dissipation, but may in some cases be beneficial for convergence (solution stability)

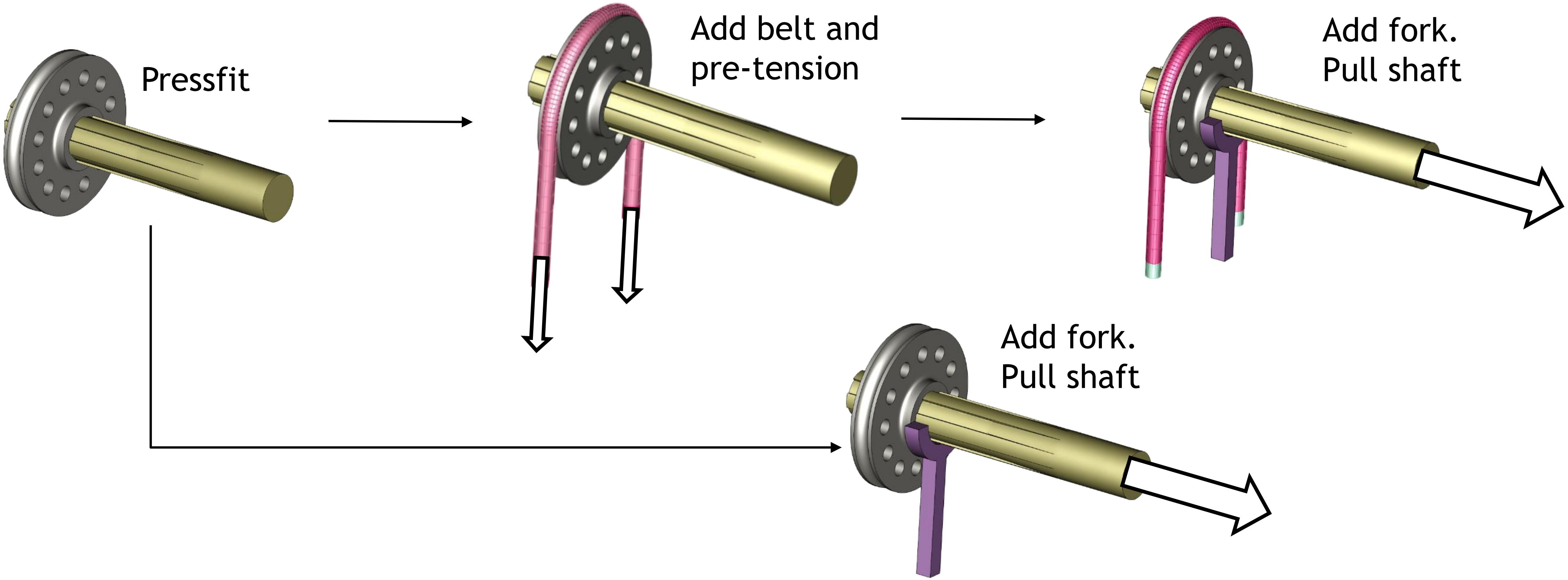


Load history management using *CASE and dynain.lsda

- An analysis consisting of a sequence of separate loadings or load steps can conveniently be divided into a sequence of LS-DYNA analyses
- Information is propagated via dynain.lsda - files from one load step to the next
  - *INTERFACE_SPRINGBACK_LSDYNA, *FTYPE* = 3
  - Binary file containing deformed geometry, initial stresses, history variables, contact state from **Mortar contacts**, etc.
  - A bit similar to a restart file
  - Full flexibility to modify control cards settings, add parts etc. between cases
- The analyses are run automatically in a sequence
  - *CASE - functionality

Load history management using *CASE and dynain.lsda

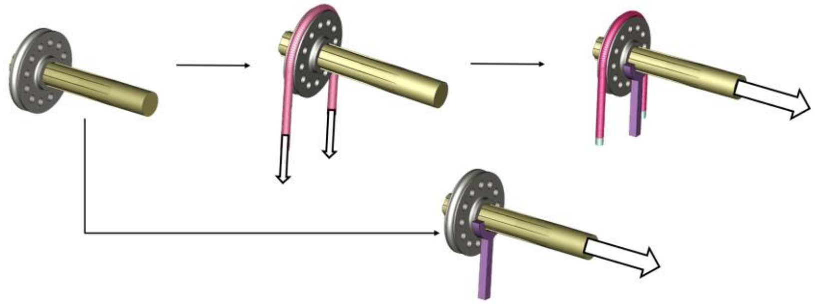
- Create branched load histories



- Duplicate pre-loaded substructures

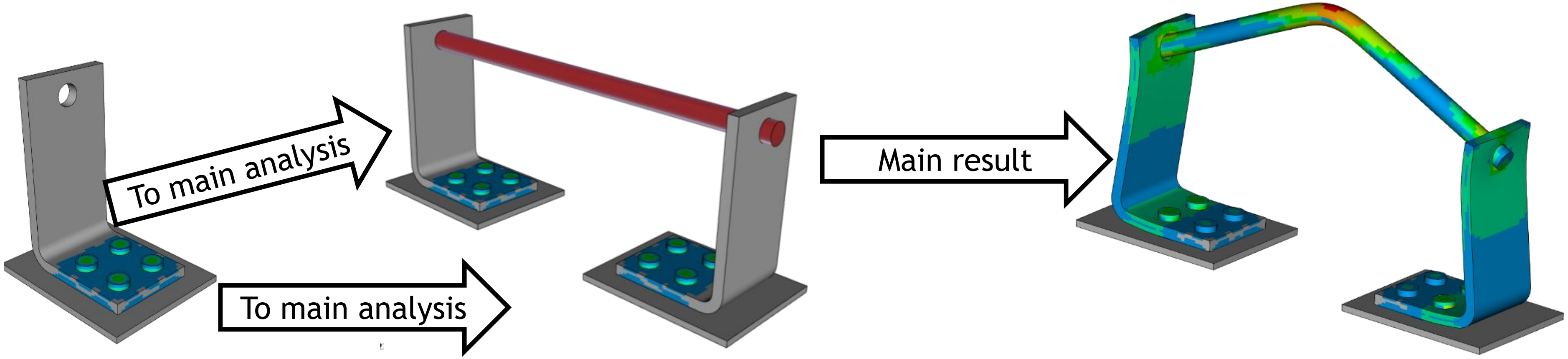
Load history management using *CASE and dynain.lsda

- Create branched load histories



- Duplicate pre-loaded substructures

Pre-loading of substructure



Recommended element types

- For non-linear analyses

Element type	Comment	LS-DYNA keyword	Element formulation
Beam		*SECTION_BEAM	1
	For bolts w. pre-tension		9
	For springs, dampers etc.		6
Shell	1st order	*SECTION_SHELL	16
	2nd order		23
Solid	1st order hex	*SECTION_SOLID	-2
	(1st order tet)		(13)
	2nd order tet		16 (17)
	2nd order hex		23

Recommended element types

- For linear analyses

Element type	Comment	LS-DYNA keyword	Element formulation
Beam	Structural beams for linear static analyses	*SECTION_BEAM	13[(1)]
	For springs, dampers etc.		6
Shell	1st order	*SECTION_SHELL	18 (20, 21, 99)
Solid	1st order hex	*SECTION_SOLID	18
	(1st order tet)		(10)
	2nd order tet		16 (17)
	2nd order hex		23

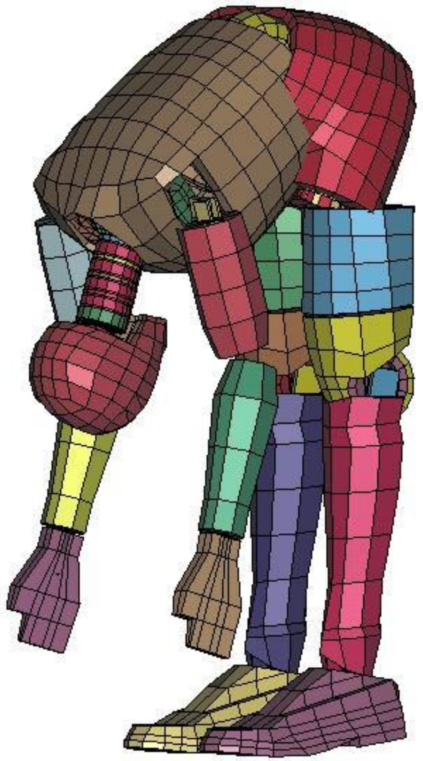
Notes: (1) Beam elforms 1, 2 or 11 are not allowed for linear static analyses in versions prior to R11

Recommended element types for linear analyses

- Structural beams
  - elform 13 recommended for linear statics
  - spotweld beams elform 9 are allowed
  - 1, 2, and 11 are allowed from R11
- Solid elements
  - 1st order tets may give an overly stiff result
  - Should the use of 1st order tets be unavoidable, use elform 10 for linear analyses

Thank you!

Your LS-DYNA distributor and more

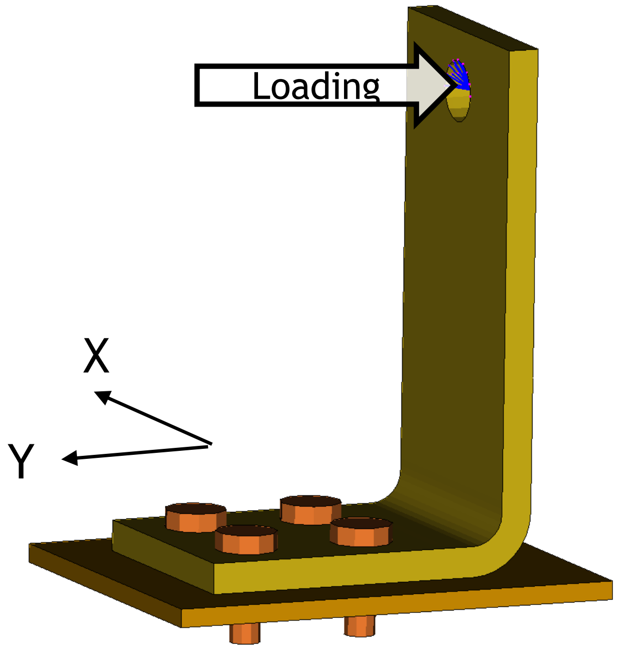


Load history management using *CASE and dynain.lsda

- How to treat analysis of a series of loadings / operations
  - For example bolt pre-tensioning, gravity loading and finally the target loading
- Traditionally as a time history
  - Many load curves
  - Activation/de-activation of features based on time
- A new approach using `*CASE` and `dynain.lsda`
  - Each load step is a separate analysis
  - Analyses run in a sequence by the `*CASE` keyword
  - Information propagated by dynain.lsda

Example: Loading of a L-bracket

- Load history:
  - A structure is subjected to 1 kN loading/unloading in the X-direction, followed by 1 kN loading/unloading in the Y-direction.
  - But first, bolt pre-tensioning is applied
  - This can be seen as three “load stages” or `*CASE`s
- Case 1: Bolt pre-tensioning
  - Use initial implicit dynamics to overcome rigid-body modes
- Case 2: X-loading and un-loading
  - Start from pre-loaded configuration: case1.dynain.lsda
- Case 3: Y-loading and un-loading
  - Start from previous case: case2.dynain.lsda
- Run the cases in a sequence using LS-DYNA’s case driver



Background

- *CASE in LS-DYNA
  - Functionality for setting up many analyses in one keyword file
  - Each analysis is called a case
  - Cases are run sequentially
  - Each case "has its own time", t = 0 .. ENDTIM
  - Separate families of results: case1.d3plot, case1.d3hsp, case2.d3plot ...

- dynain files
  - A keyword file containing the final state of an analysis
  - Deformed geometry (*NODE, *ELEMENT_SHELL_THICKNESS, ...)
  - Elements(*)
  - Residual stress state (*INITIAL_STRESS_...) and history variables

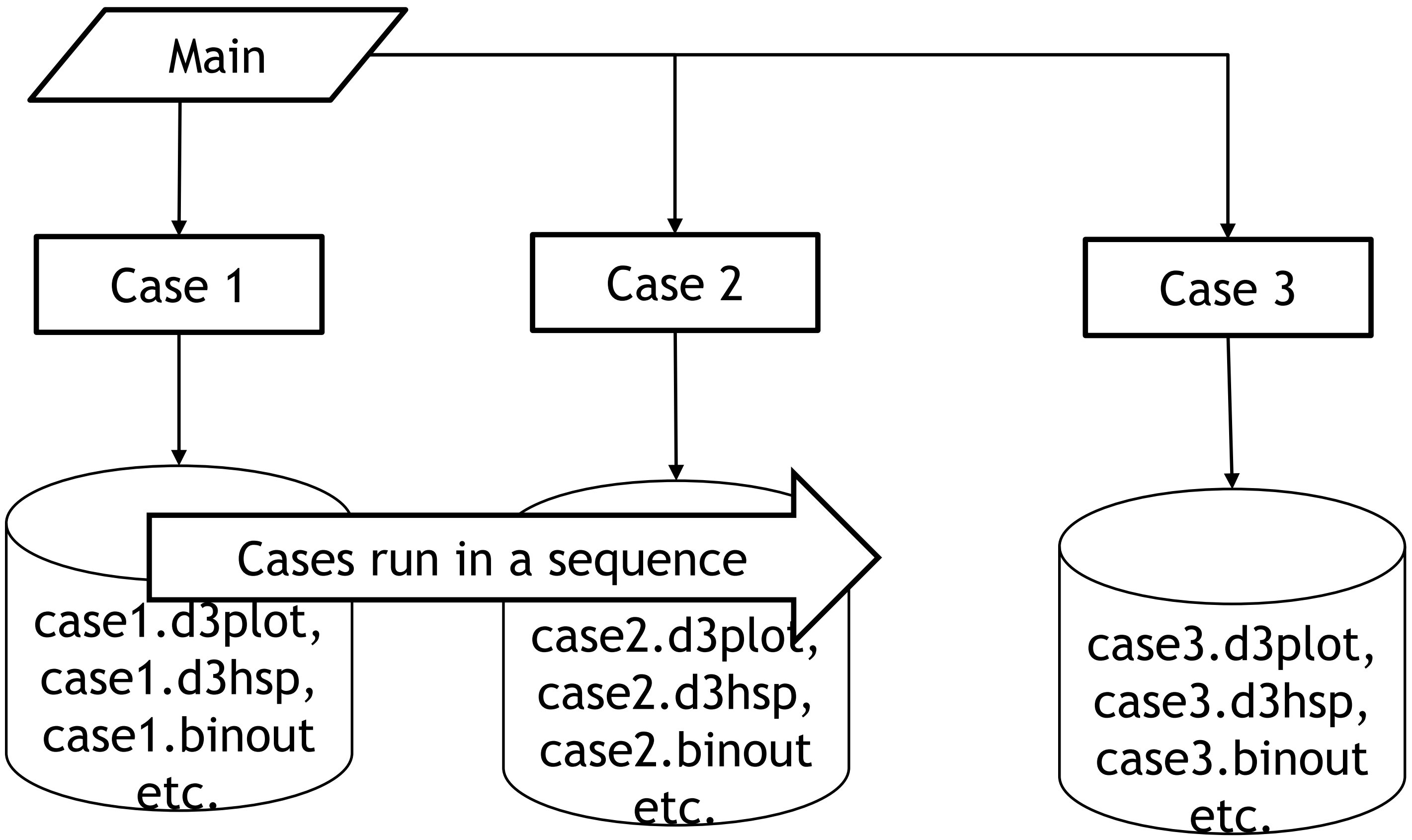
- Combining *CASE and dynain files: a new approach for multi-step (implicit) analyses in LS-DYNA

Background

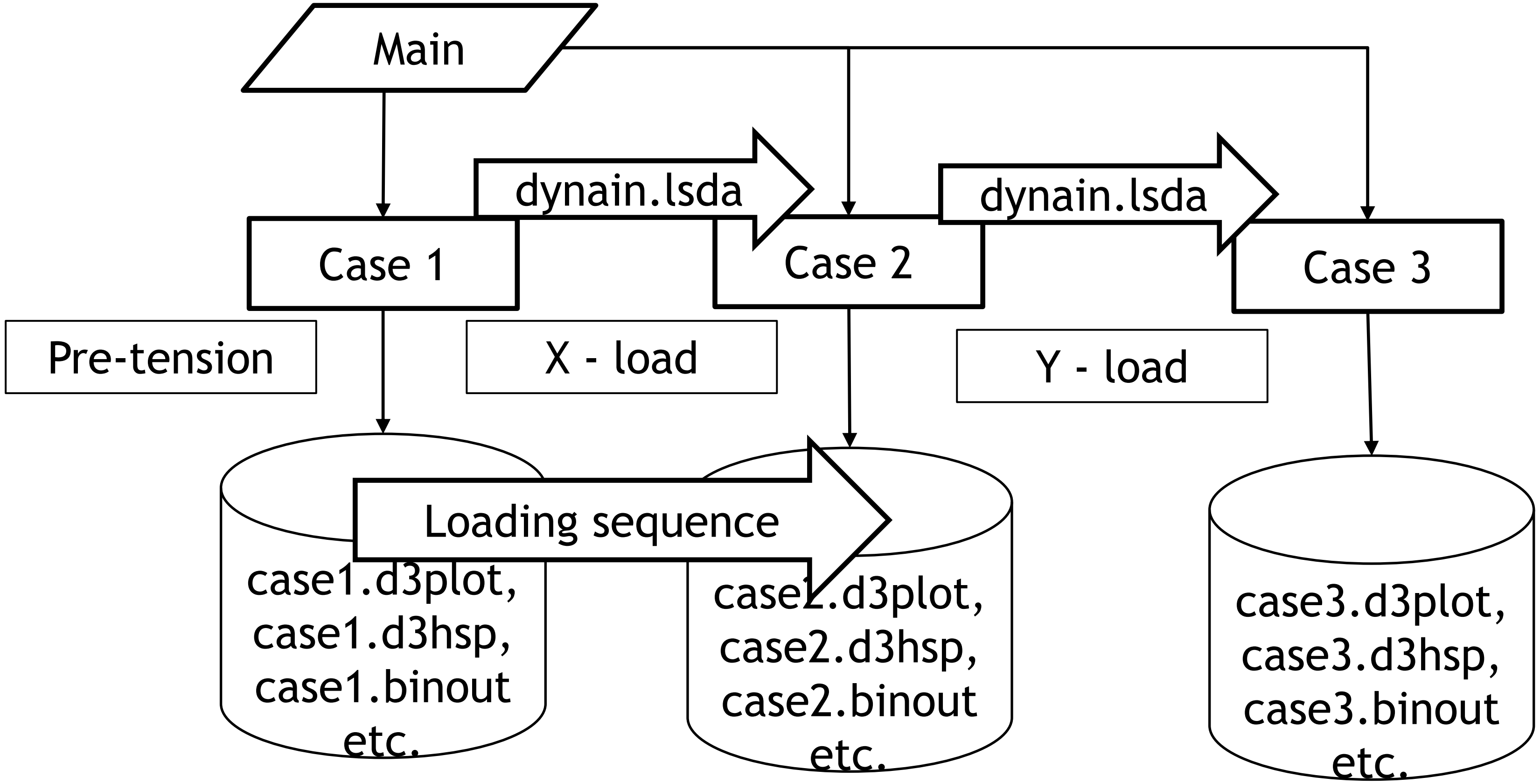
- `*CASE` in LS-DYNA
  - Functionality for setting up many analyses in one keyword file
  - Each analysis is a called a case
  - Cases are run sequentially
- `dynain` files
  - A keyword file containing the final state of an analysis
  - Deformed geometry (`*NODE`, `*ELEMENT_SHELL_THICKNESS`, ...)
  - Elements(*)
  - Residual stress state (`*INITIAL_STRESS_`...) and history variables

- Combining `*CASE` and `dynain` files: a new approach for multi-step (implicit) analyses in LS-DYNA
  - For a background, see
    Borrvall, T., [Solution explorer in LS-PrePost- a GUI for Nonlinear Implicit FE](), 12th European
    LS-DYNA conference, Koblenz 2019.
  - Appendix in coming revisions of the keyword manual

The *CASE - concept



The *CASE / dynain.lsda - concept



The *CASE / dynain.lsda - concept

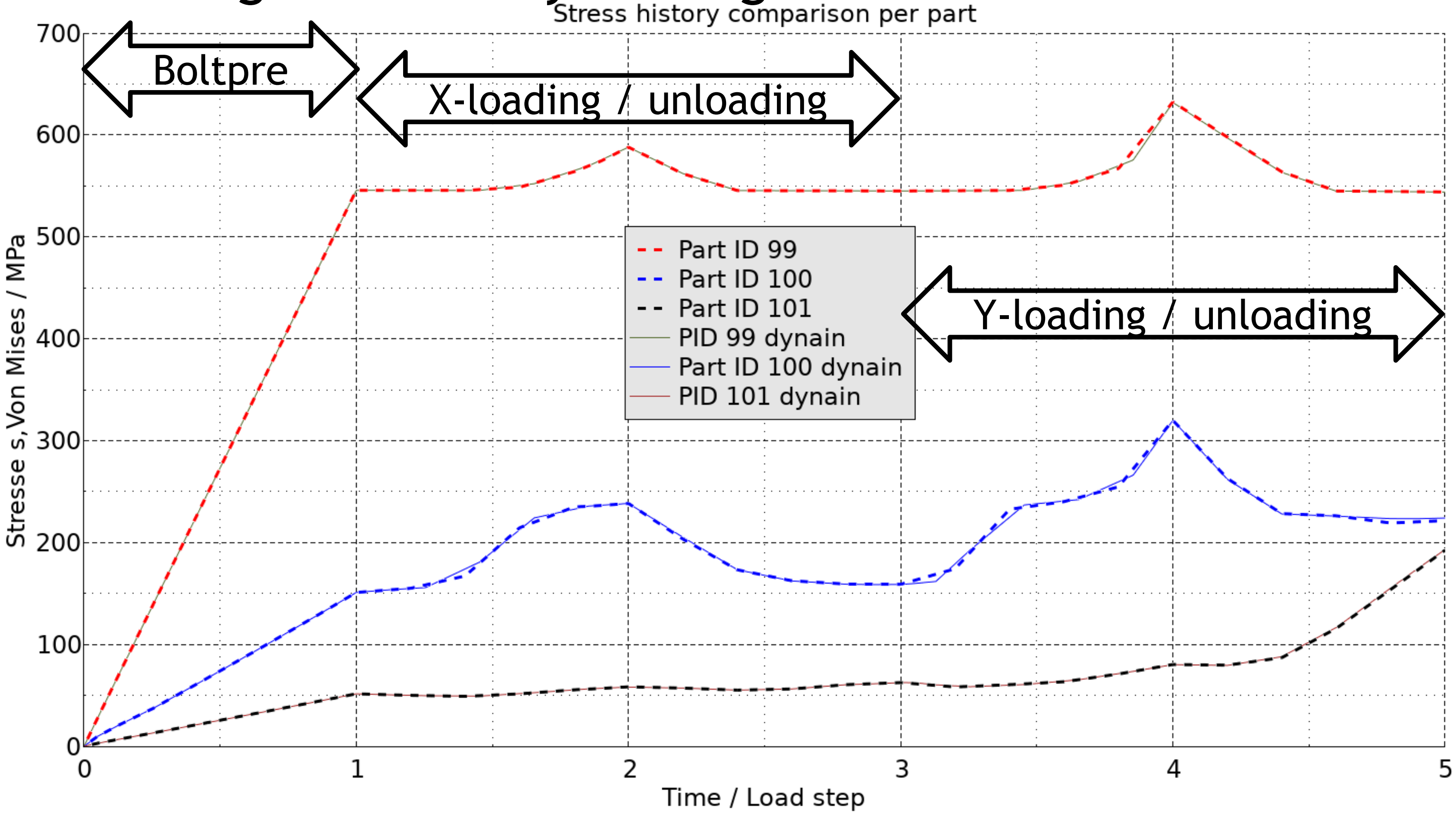
- Analysis of a sequence of load steps is run as a sequence of analyses
  - A more intuitive way of setting up a load history using load steps
  - Clear definition of which features that are active in each step
  - Reduce need for birth/death times of contacts, boundary conditions etc.

- Information is propagated from step to step using a binary dynain file
  - *INTERFACE_SPRINGBACK_LSDYNA
  - *ftype* = 3, *cflag* = 1, ... ➔ **dynain.lsda**
  - Contains deformations, stresses, history variables, contact state etc.
  - **NOTE! Must use Mortar contacts!**
  - Can be opened for 3d visualization in LS-PrePost (from version 4.8)
- *CASE to describe the sequence of steps

- Easy to re-use pre-tensioned configuration is obtained after each step

Loading of L-bracket: Results comparison

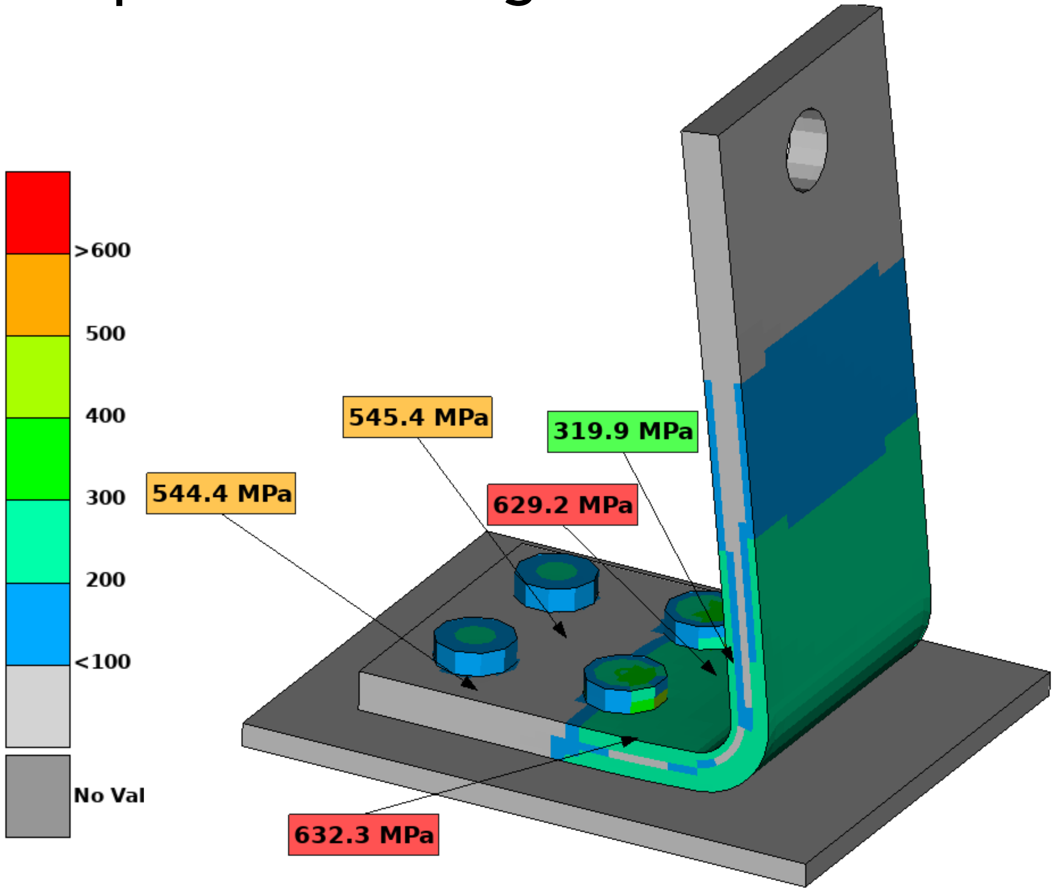
- Bolt pre-tensioning followed by loading



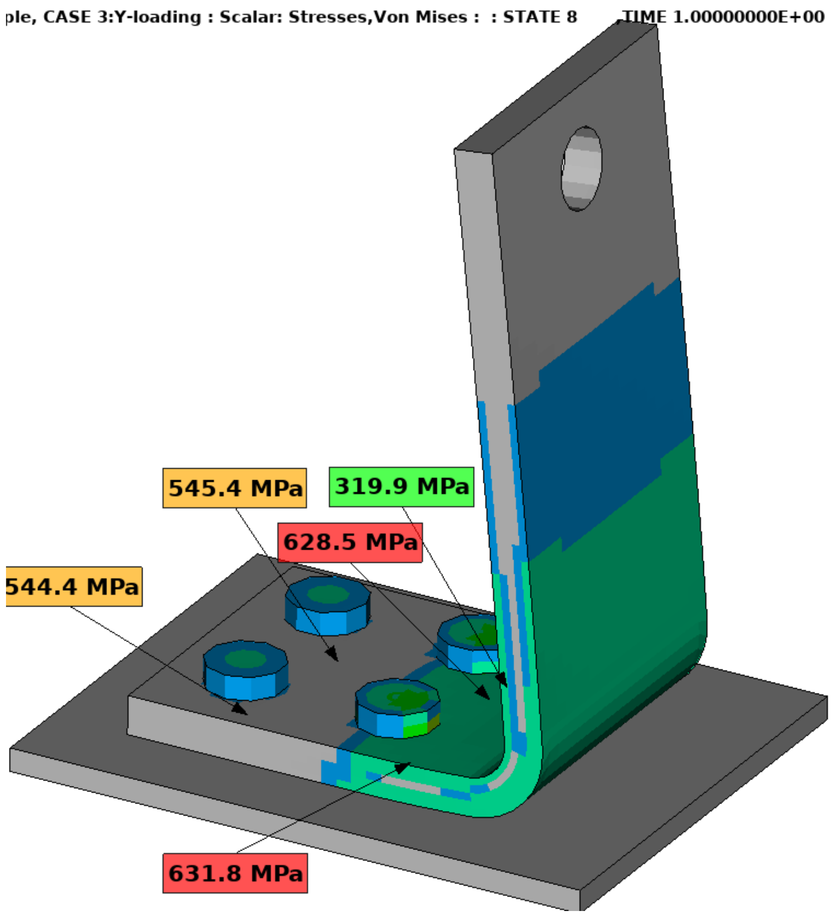
Stress history comparison: dynain.lsda w. solid lines, all-in-one with dashed

Loading of L-bracket: Results comparison

- Stress at peak Y-loading



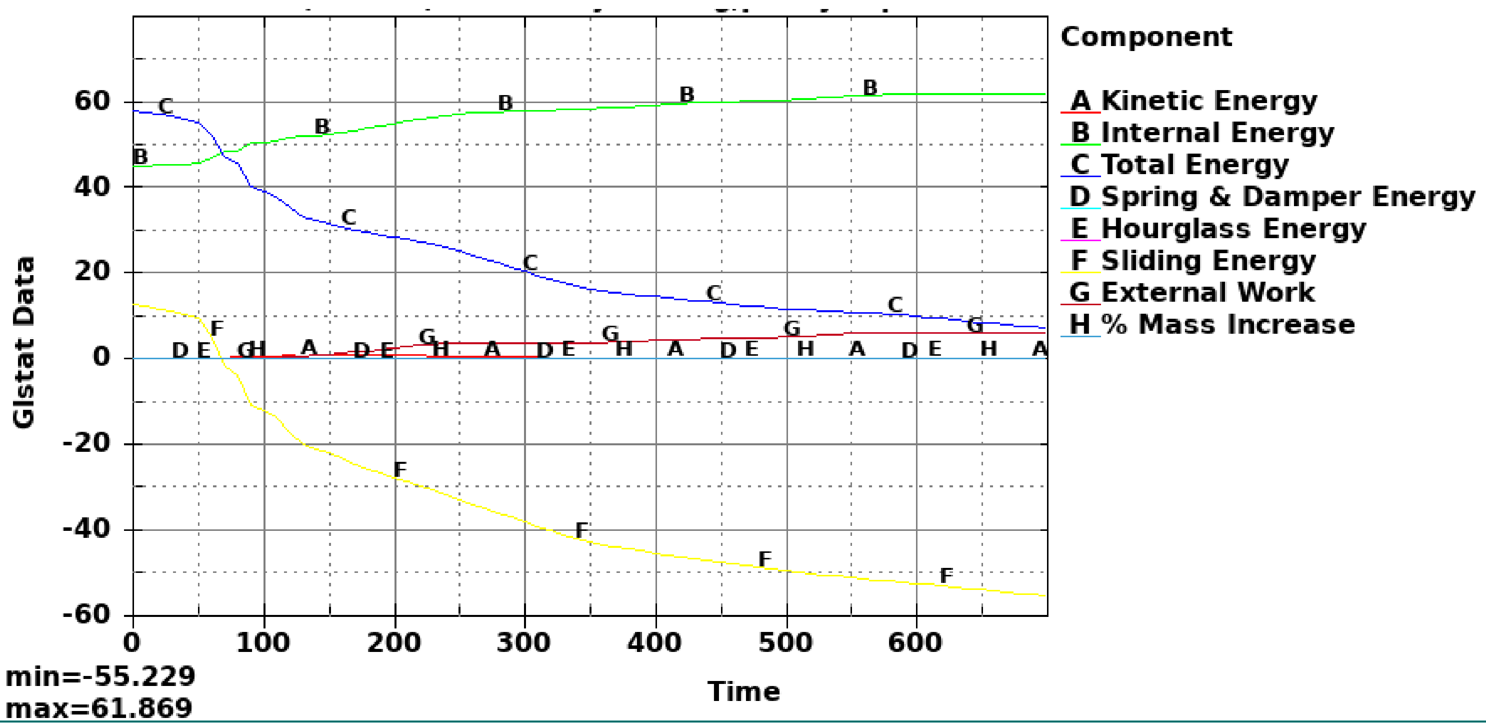
Peak stress time history



Peak stresses using dynain.lsda

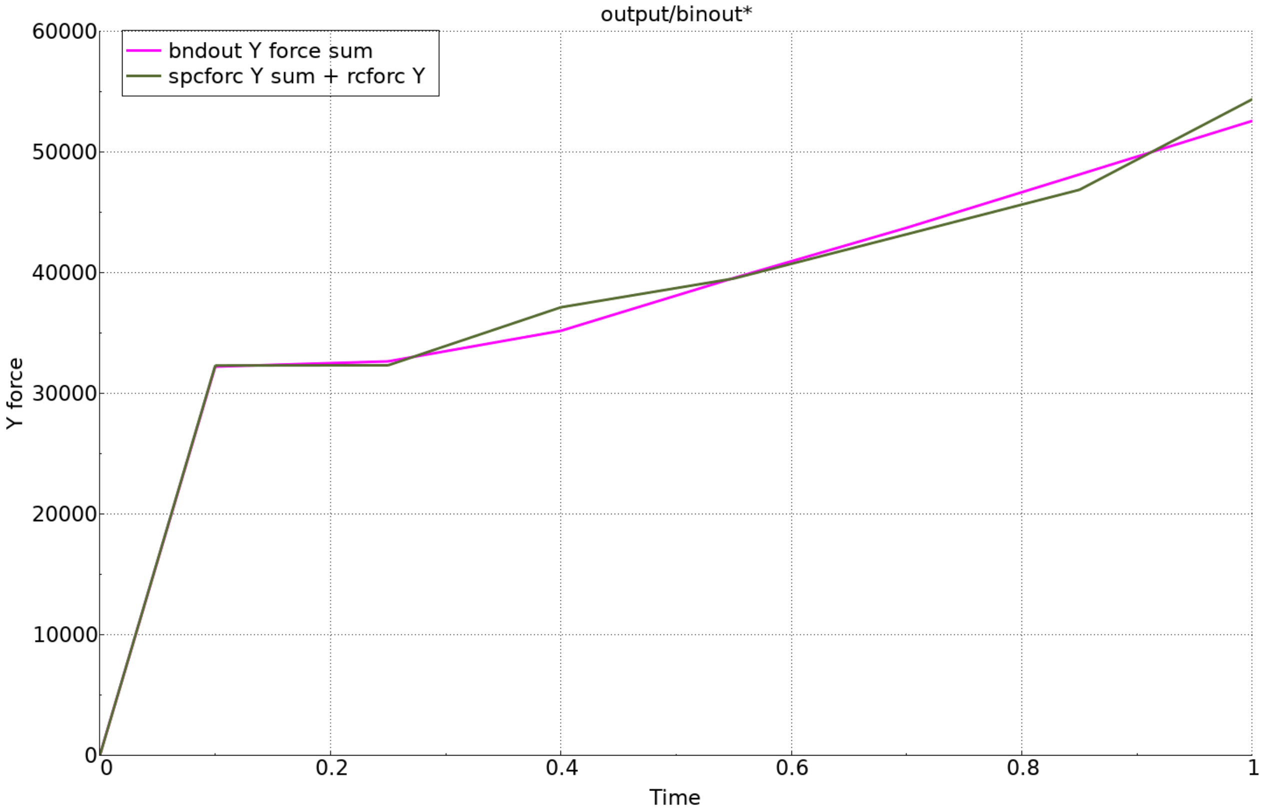
Checking your results in LS-DYNA

- Explicit: check energy balance
  - Optional: check force equilibrium etc
- Implicit: check force equilibrium etc
  - Also check energy balance



Checking your results in LS-DYNA

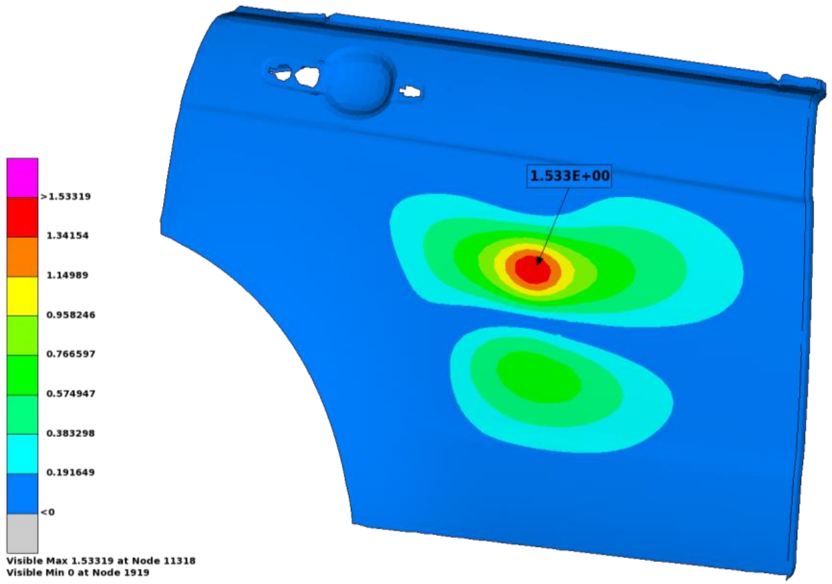
- Explicit: check energy balance
  - Optional: check force equilibrium etc
- Implicit: check force equilibrium etc
  - Also check energy balance



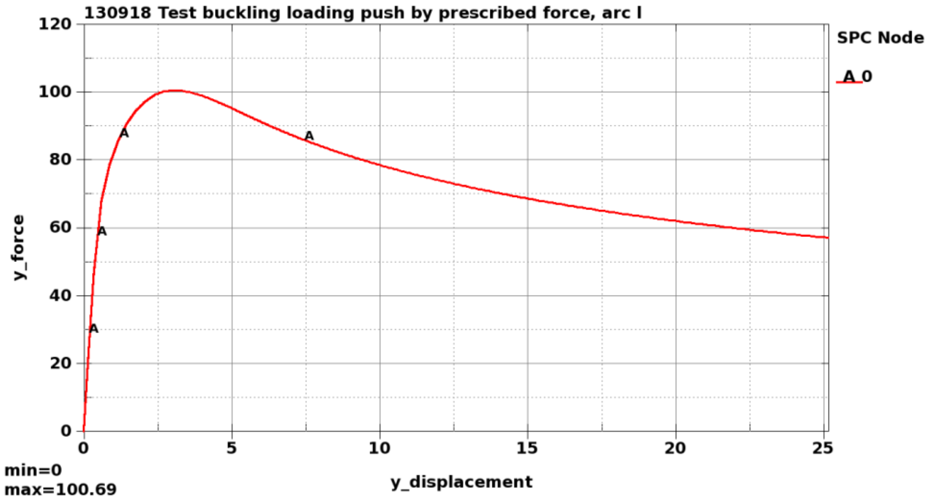
Some of the attached examples

Filename	Example of
doorstiff001.key	Linear static analysis
bend001.key	Non-linear static analysis
bolts001.key	Non-linear dynamic/static analysis. Bolt pre-tensioning
buckle001.key	Linear buckling analysis
limitload001.key	Non-linear limit load analysis
eigen001.key	Eigenfrequency analysis
eigen002.key	Intermittent eigenfrequency analysis
transient001.key	Linear transient modal dynamics
transient_nonlin001.key	Non-linear transient dynamics
ssd_fatigue001.key	Steady state dynamics with fatigue evaluation
Set_4.key	Random vibration analysis with fatigue evaluation
run_2d.key	Axisymmetric 2D analysis, large deformation
TPU.key	Rubber
Insert_and_interference.key	Rubber, press-fit, contacts
Large_deformation.key	Rubber, contacts, large deformations
pretens001.key	Bolt pre-tensioning, first step of restart analysis
full_restart.key	Example of full restart, from pretens001.key
small_restart.key	Small restart from pretens001.key

Attached examples



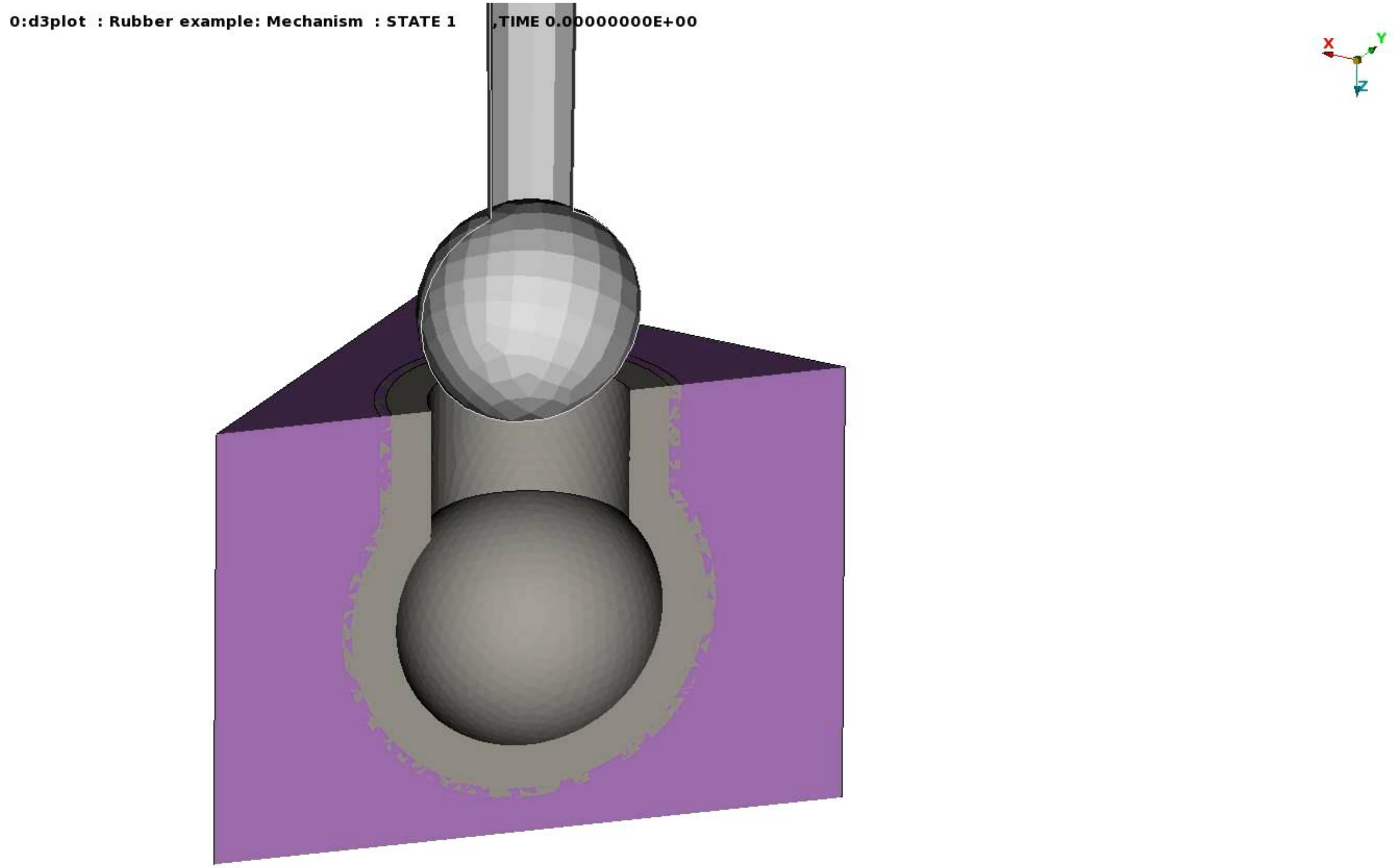
doorstiff001.key



limitload001.key

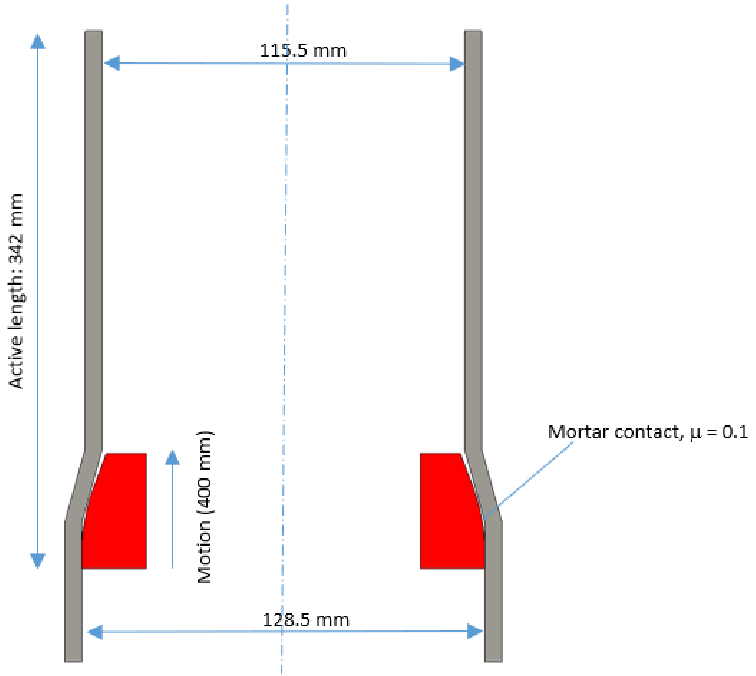
Attached examples - Rubber

- A rubber bushing is press fit into a steel casing, and a steel ball is inserted into the bushing

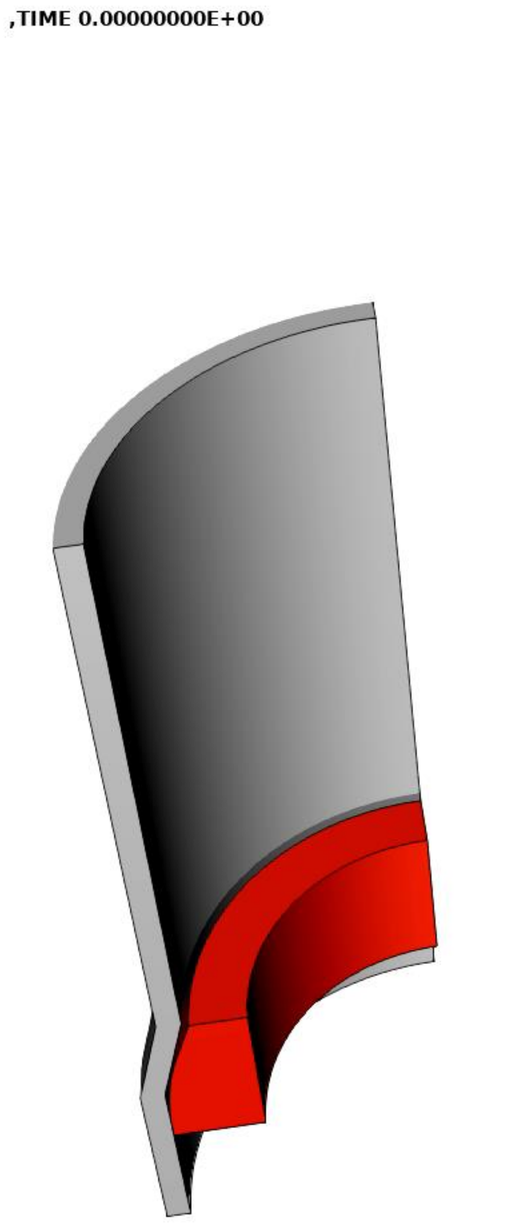


Attached examples

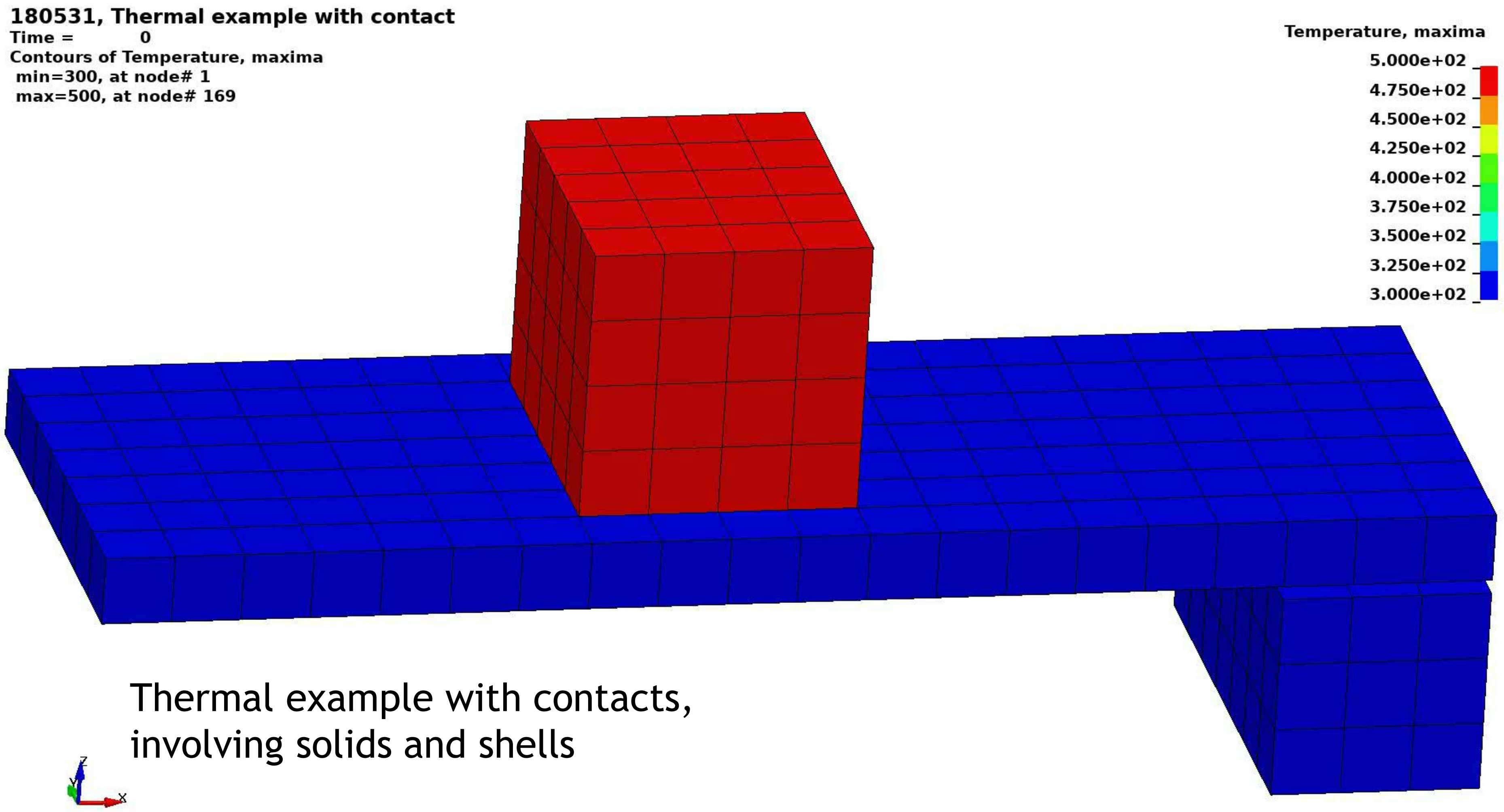
- 2D, thermal ...



2D Axi-symmetry: A conical insert (red) is pushed into a tube (gray).



Attached examples



Thermal example with contacts, involving solids and shells

Tied contacts

- For tied contact, there are many options available

	*CONTACT_TIED_	Constraint formulation	DOFs	Move nodes[1]	Moment transferred[2]
For solids	NODES_TO_SURFACE	Kinematic	1 - 3	Yes	
	NODES_TO_SURFACE_OFFSET	Penalty	1 - 3	No	No
	NODES_TO_SURFACE_CONSTRAINED_OFFSET	Kinematic	1 - 3	No	Yes
For structural elements	SHELL_EDGE_TO_SURFACE	Kinematic	All[3]	Yes	
	SHELL_EDGE_TO_SURFACE_OFFSET	Penalty	All[3]	No	No
	SHELL_EDGE_TO_SURFACE_BEAM_OFFSET	Penalty	All[3]	No	Yes
	SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET	Kinematic	All[3]	No	Yes

Notes: (1) Only slave nodes within the search distance will be moved. (2) Due to separation between secondary nodes and main segments. Only applies to the `_OFFSET_` option. (3) All available degrees of freedom, depending on element type of the slave node.

- If beams or shells are involved, use the `SHELL_EDGE` type to constrain all DOFs
- It is strongly recommended to check which nodes get tied and not! Use the pre-processor's contact check functionality. Also check messages printed in `d3hsp`. Nodes that are not tied will be reported.

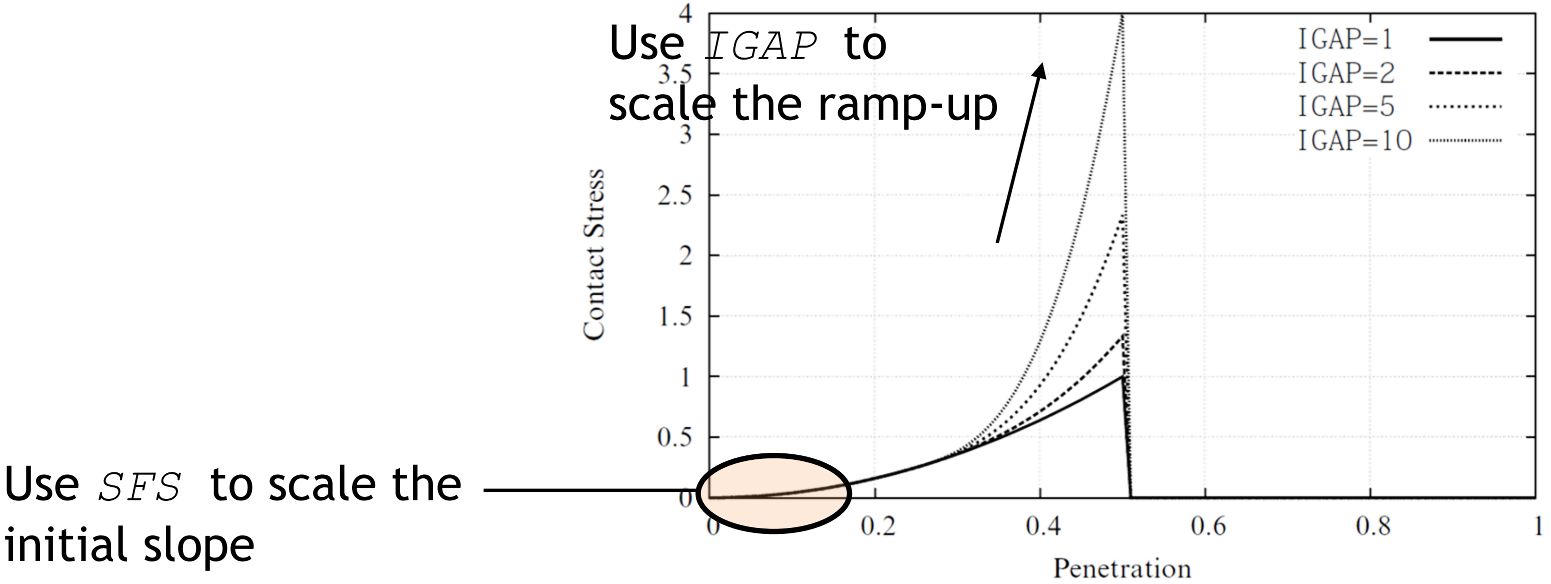
Contacts

- For sliding contact in implicit analyses, the recommendation is to use the Mortar contacts:
  - `*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_ID`,
  - `*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR_ID`.
- Normally requires a minimum of input:
  - define what is in contact using parts or part sets, and
  - specify coefficient of friction.

```
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR_ID
$#     cid                                                                 title
      1010Global contact all
$#    ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         0         0         5         0         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
      0.15
$#     sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
```

- Penetration information will be reported in mes0* - files.
  - If “too large” penetrations are noted during the simulation, contact stiffness can be increased using the *SFS*- and *IGAP*-parameters.

Mortar Contact – dealing with locally high penetrations

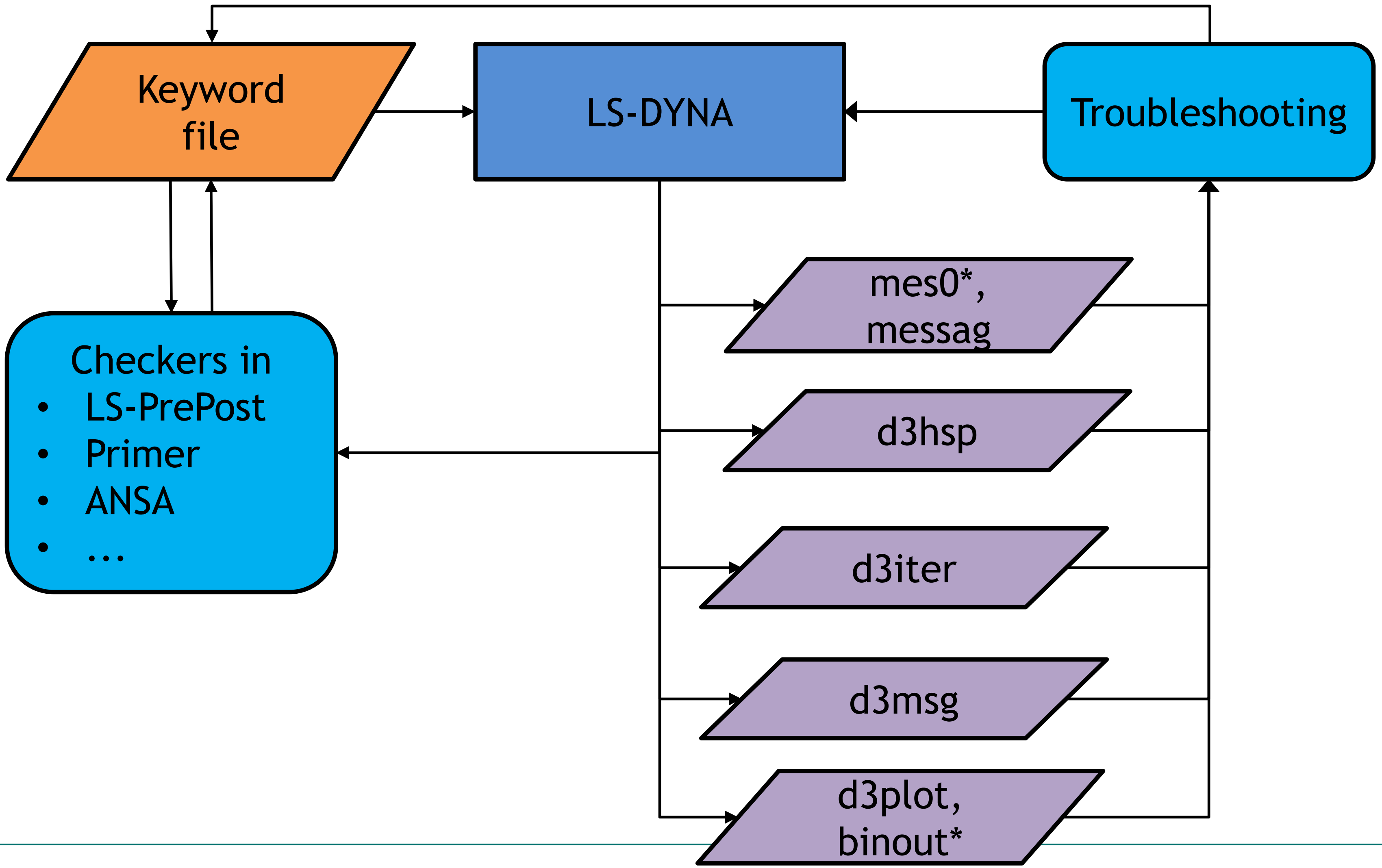


- **IGAP>1** can be used to increase stiffness for large penetrations, or to deal with large contact pressures
- Better for convergence in general compared to increasing **SFS**

Troubleshooting convergence problems

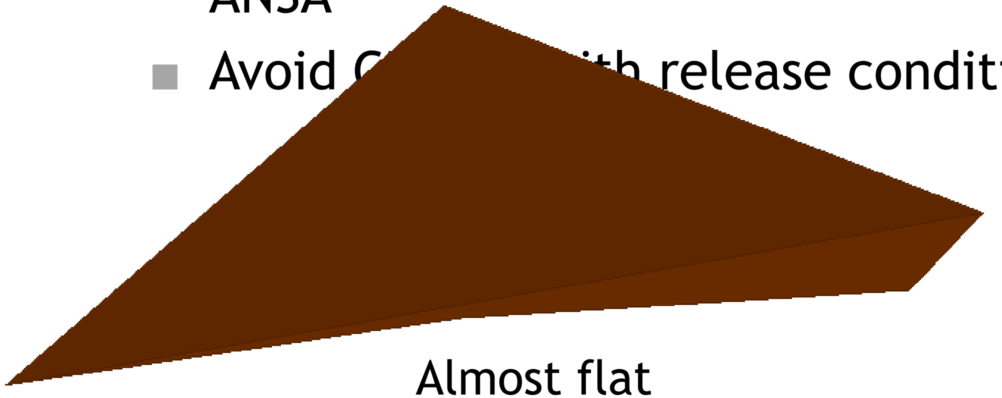
- Included as Appendix C of the Guideline
  - See also previous Webinars “Troubleshooting convergence problems in LS-DYNA implicit” and “Model checking in LS-DYNA – from pre to post simulation”
  - Available from files.dynamore.se > Client Area > 7_Webinars
  - A brief summary follows ...

Troubleshooting convergence problems

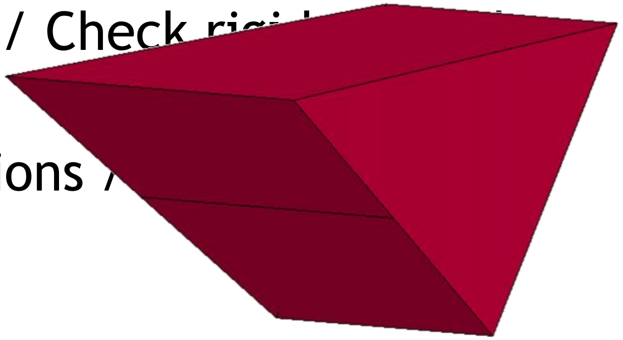


Troubleshooting in LS-DYNA implicit

- First step – basic model quality checking ...
  - Check mesh quality, avoid “4-noded trias” poorly shaped “pentas”
  - Negative / small volume for 2nd order tets
  - Poor aspect ratio of elements
  - Unintended cracks in the mesh / Check DupNodes in LS-PrePost, “red cons” in ANSA
  - Avoid CNRBs with common nodes / Check rig… in ANSA
  - Avoid C… ith release conditions /



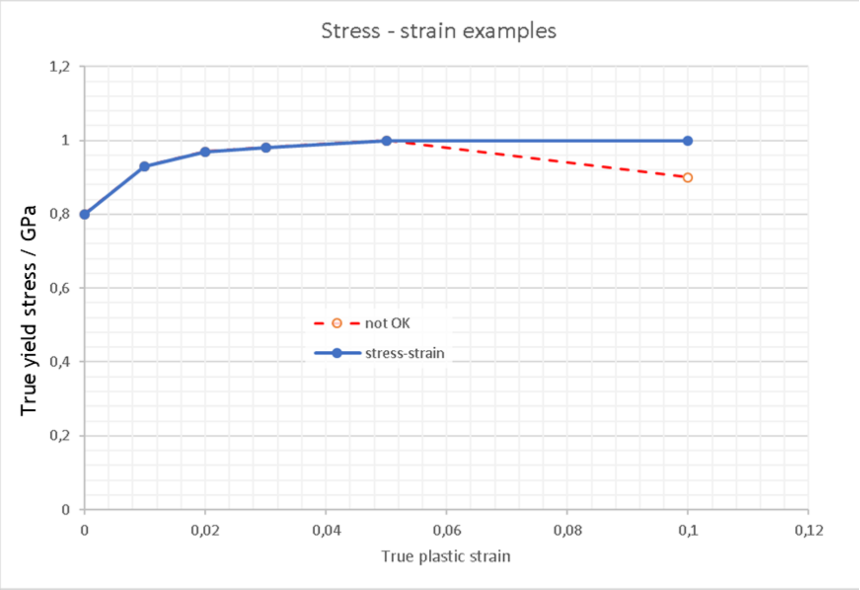
Almost flat tet-10



Degenerated hex

Troubleshooting convergence problems

- Check material models
  - Avoid use of *MAT_ELASTIC for rubbers (at least for finite deformations)
  - Use MAT_24 instead of MAT_3 with *BETA* = 1 for isotropic hardening
  - For non-linear elastic springs, try replace *ELEMENT_BEAM elform 6 and *MAT_67 with *ELEMENT_DISCRETE and *MAT_S04.
  - Check last segment of hardening curve! LS-DYNA will extrapolate, negative last segment slope is not recommended.
  - For *MAT_FABRIC, use FORM = -14, and ECOAT, TCOAT. Also use elform 9 (meml formulation) on *SECTION_SHELL



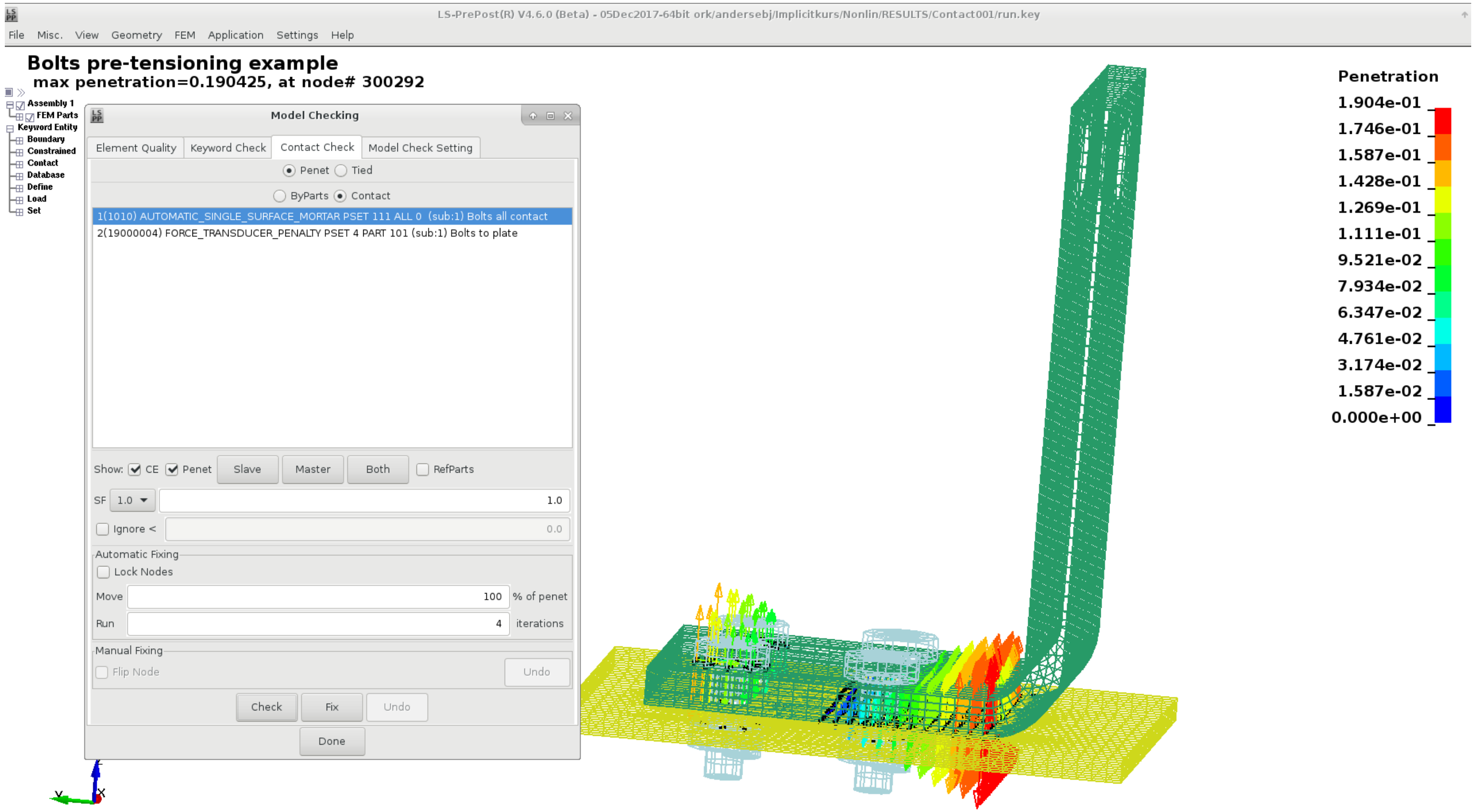
  - Do not involve parts with *MAT_NULL in
  - Note! Some material models are currently not supported in implicit (e.g. *MAT_SAMP-1).

Troubleshooting convergence problems

- Check tied contacts. Ensure that the intended slave nodes are tied (not too many, not too few) Inspect d3hsp/mes0* files to confirm check results from pre-processors.
  - ANSA: Check > Contacts > Contacts from solver, input the d3hsp - file. Untied nodes will be highlighted in the model.
  - Define a slave node set if full control is desired
  - Check for mesh distortion due to nodal projections `*** Warning 41240 (SOL+1240)`
  - Use minimum number of tied contacts, due to risk of over-occupying master segments and consequential failure of following tied contacts
- When using constraint-based tied contact, nodes subjected to other constraints (`*CONSTRAINED_NODAL_RIGID_BODY`, `*BOUNDARY_SPC`, ...) can not be tied. One remedy may be to set *IPBACK* > 0 on Optional Card E of the `*CONTACT_TIED_` - definition. By this, LS-DYNA will automatically create a penalty-based tied contact for nodes that are subjected to other constraints.

Troubleshooting convergence problems

- Check (and correct) unintended initial penetrations in



In LS-PrePost: Application>Model Checking>Contact Check

Troubleshooting convergence problems

- Negative eigenvalue warnings are most likely due to rigid body modes or elements that gets severely distorted (for example rubber) during deformation.

```
*** Warning 60124 (IMP+124)
        XX negative eigenvalues detected
```

  - Check for rigid body modes by eigenvalue analysis (`*CONTROL_IMPLICIT_EIGENVALUE`)
  - Also look in the d3iter - file
  - Use implicit dynamics (`*CONTROL_IMPLICIT_DYNAMICS`) or in some cases inertia relief to properly handle rigid body modes. Beware of time scale (ms, s)
- Use the d3iter – file! Looking at the non-converged states can give a lot of information, if for example loose parts in the structure are "flying away". Scale up the displacement, or fringe plot it, in order to identify areas where large changes in displacement take place.
  - Set *RESPLT* = 1 on Card 4 of `*DATABASE_EXTENT_BINARY` to be able to fringe plot the residual in the d3iter – files. This can pinpoint where in the model convergence is the hardest.

Troubleshooting convergence problems

- Check model connectivity
  - For example, is the list of tie slave parts complete?
  - Look for AUTOSPC warnings in the d3hsp – file.

```
*** Warning 60055 (IMP+55)
     *********************************************************
     *                                                       *
     *                  -  WARNING  -                        *
     *                                                       *
     *           6 Unconstrained degrees of freedom have     *
     * been detected and removed from this IMPLICIT model.   *
     *                                                       *
     *********************************************************
```

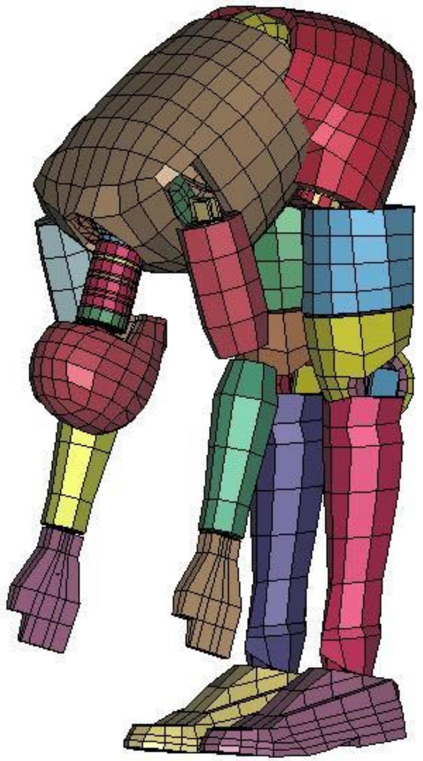
Set *LPRINT* = 3 on `*CONTROL_IMPLICIT_SOLVER` to get a list of all nodes (or rigid parts) subjected to AUTOSPC:s

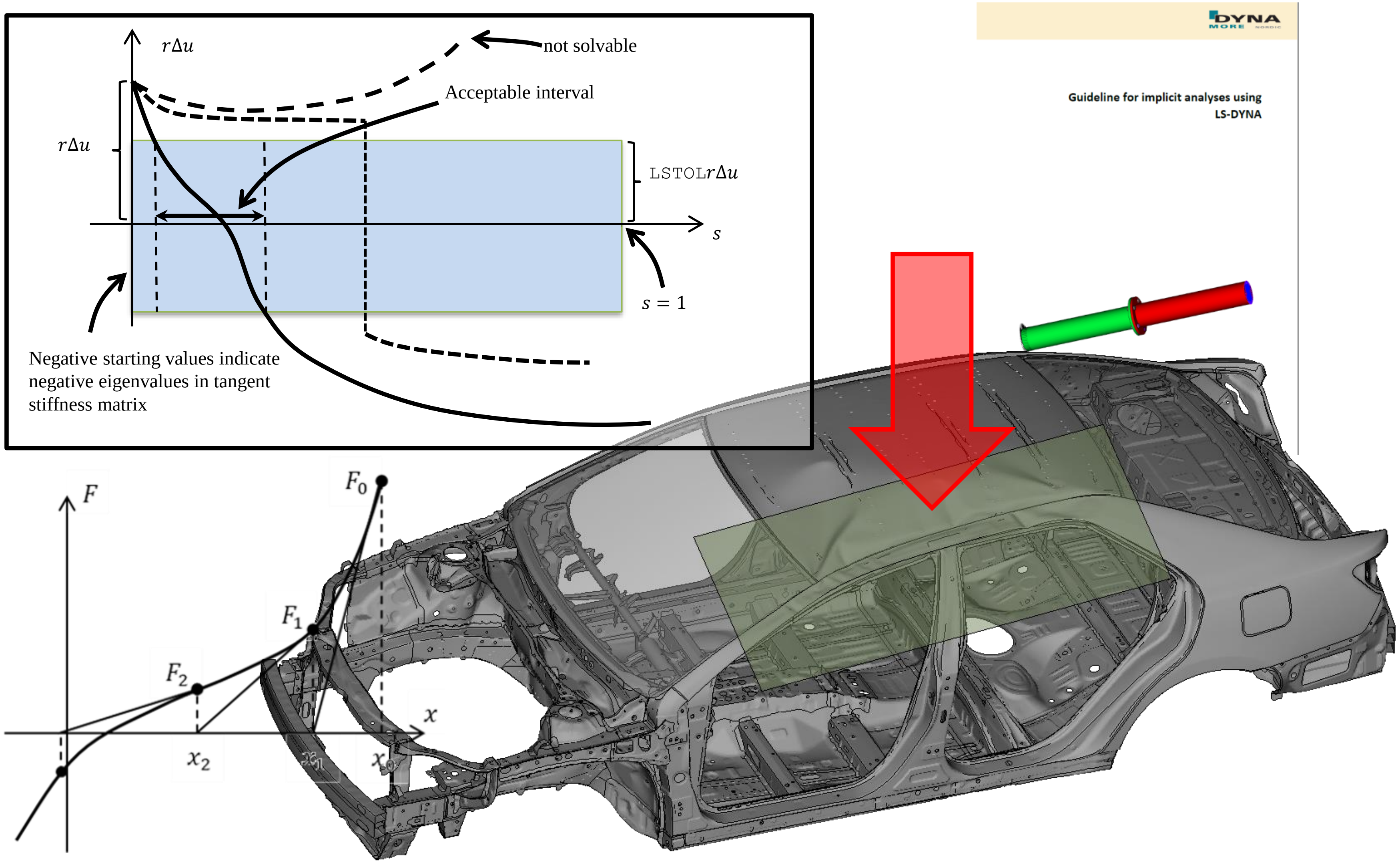
Appendices on special topics

- Appendix A: Analyses of rubber
  - Material models
  - Element formulations (including a short discussion of EFG)
  - Examples
- Appendix B: Restart
  - Small restart and full deck restart
  - Examples of analyses of a pre-loaded structure subjected to multiple load cases
  - Analyses of multiple load steps using the `*CASE` / `dynain.lsda` concept
- Appendix D - F: Issues regarding implicit ⇔ explicit and (automatic) switching
- Appendix G: Comments on the control card settings for implicit analyses

Thank you!

Your LS-DYNA distributor and more





$r\Delta u$
not solvable
Acceptable interval
$r\Delta u$
LSTOL$r\Delta u$
s
$s = 1$
Negative starting values indicate negative eigenvalues in tangent stiffness matrix
F
F_0
F_1
F_2
x
x_2
x_0
Guideline for implicit analyses using LS-DYNA

Document information

Doc. no.:
Revision: 1
Prepared for:
Project no.:
Approved by:
Release date:
Distribution: Approved for public release / Confidential

DYNAmore Nordic AB
Brigadgatan 5
SE-587 58 LINKÖPING
Sweden

Org. no. 556819-8997
EC VAT: SE556819899701

Phone: +46 (0)13 236680
Fax: +46 (0)13 214104
E-mail: info@dynamore.se
Web: www.dynamore.se