Short Introduction to LS-DYNA and LS-PrePost

Jimmy Forsberg

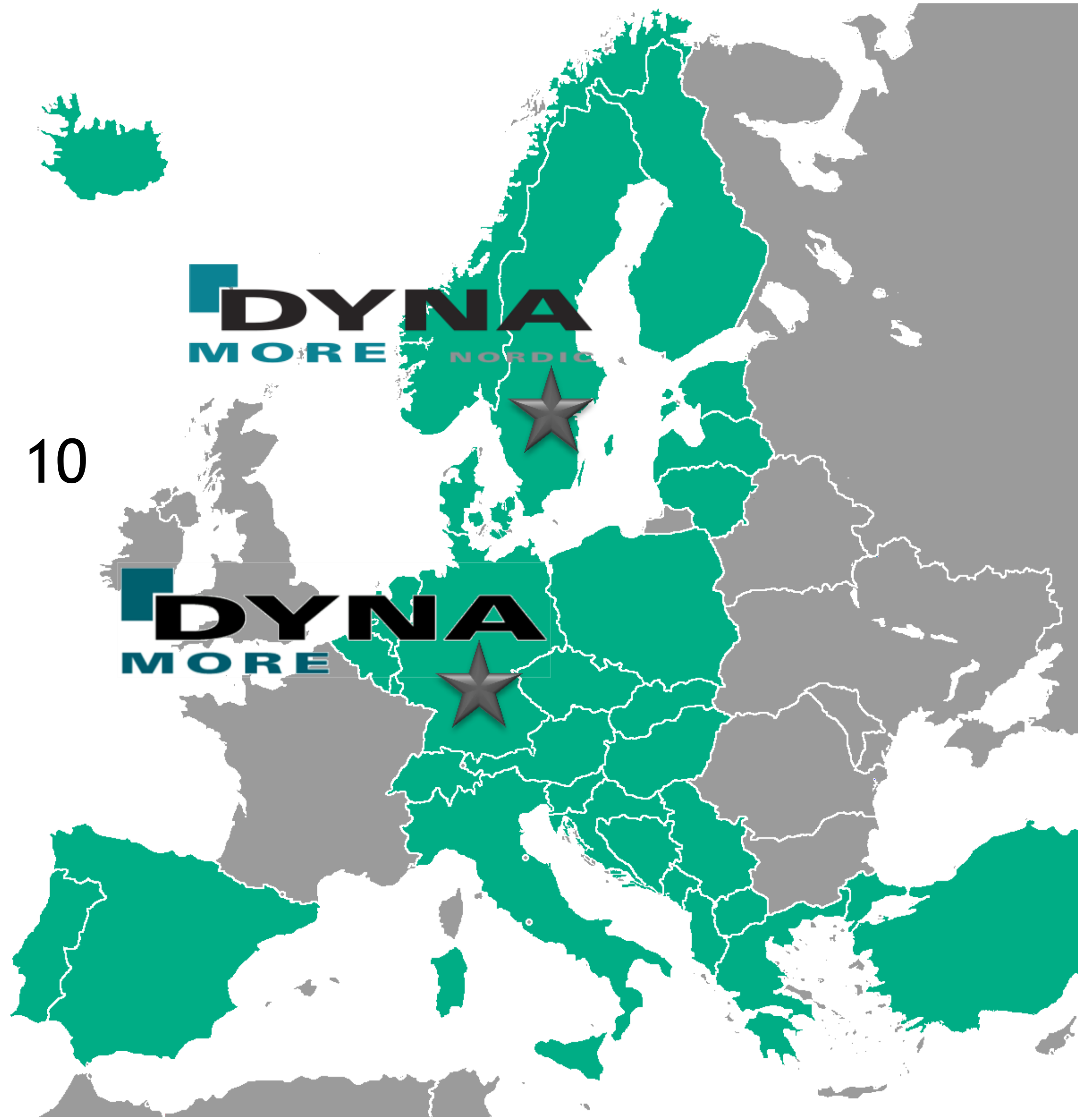
Content

- DYNAmore Nordic presentation
- Introduction to LS-DYNA
  - General work with different solvers.
  - LS-DYNA capabilities
  - Keywordfile structure

- Introduction to LS-PrePost
  - Layout
  - Pre-processing
  - Post-processing
  - Special features
    - Composite tool

DYNAmore Group

- CAE Software
- Engineering services
- Distributor for LSTC
- Personnel: 70
- LSTC code developer: 10
- Head office in Stuttgart, Germany



LSTC
Livermore Software Technology Corp.

DYNAmore Group

- Sweden
  - 17 Employees
  - 37 years in average
  - 9 Ph.D.
  - 8 M.Sc.
  - 1 Economics/Adm
- Office in Linköping
- Office in Göteborg

DYNAmore GmbH

Germany

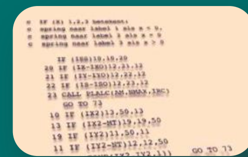
- ~60 Employees
- Headquarters in Stuttgart-Vaihingen
- Offices
  - Ingolstadt
  - Dresden
  - Wolfsburg
  - Fürstenwalde (Berlin)
- On-site Offices
  - Sindelfingen
  - Untertürkheim
  - Weissach
  - Ingolstadt

Stuttgart [Headquarters]

Business model

Technical Software

- Sales
- Support
- Training



Software development

- Development
- Research
- Implementation
- Improvement
- Support

Consultancy work

- Non-linear analysis
- Linear analysis
- Dynamic analysis
- Static analysis
- Optimization

DYNAmore Nordic AB

- LS-DYNA
- LS-OPT
- Ansa
- Crash dummies
- Crash barriers
- Oasys Primer
- DynaForm
- FormingSuite
- Femzip

- Material modeling
- Contacts
- Element technology
- Training
- GUI development
- HPC Cluster

- Vehicle safety
- Explosion analysis
- Metal forming
- Offshore
- Energy
- Roadside safety
- Accident reconstruction
- Vibration and NVH
- Thermo-mechanical
- On-site

DYNAmore Nordic - Selected customers

DYNAmore Group – Selected customers

LSTC
Livermore Software Technology Corp.

VOLVO SAAB PORSCHE FIAT GM Autoliv BENTELER AUTOMOTIVE

HYUNDAI Drive your way Ford KIA HOLDEN DRIVE ON BMW Audi

GESTAMP HARDTECH MAGNA STEYR more value • more car

Contact

- Software Products
  - Dr. Marcus Redhe
  - E-mail: marcus.redhe@dynamore.se
  - Mobile: +46 – (0)70 55 131 42
- Engineering Service and Support
  - Dr. Daniel Hilding
  - E-mail: daniel.hilding@dynamore.se
  - Mobile: +46 – (0)70 65 366 85
- Address:

  DYNAmore Nordic
  Brigadgatan 14
  587 58 Linköping
  Sweden
- Web: http://www.dynamore.se
- Phone: +46 – (0)13 23 66 80

LS-DYNA

One code strategy

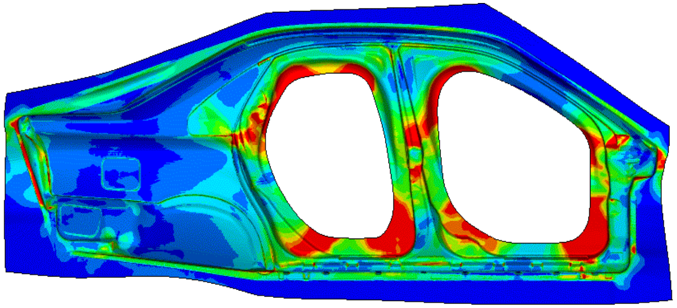
"Combine the multi-physics capabilities into one scalable code for solving highly nonlinear transient problems to enable the solution of coupled multi-physics and multi-stage problems"

Explicit/Implicit	✓
Heat Transfer	✓
Mesh Free EFG,SPH,Airbag Particle	✓
User Interface Elements, Materials, Loads	✓
Acoustics Frequency Response, Modal Methods	✓
Discrete Element Method	✓

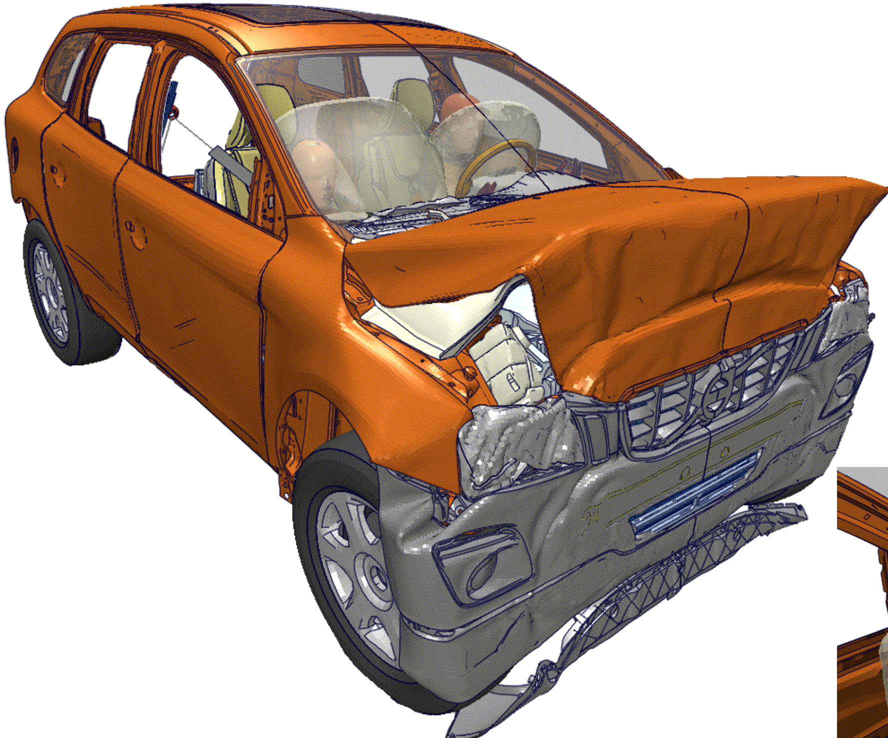
Incompressible Fluids	R7
CESE Compressible Fluid Solver	R7
Electromagnetism	R7

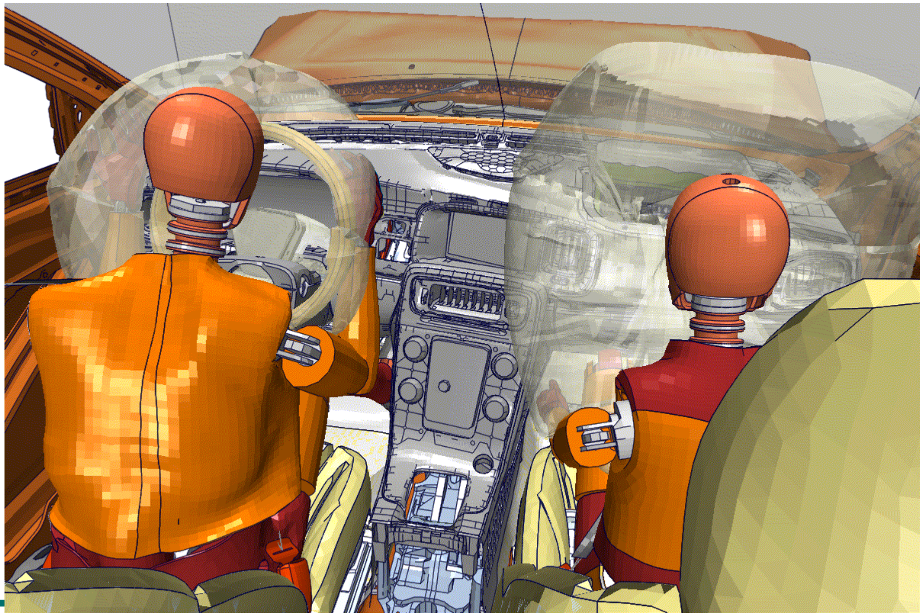
SBD – Simulation Based Design

- Instead of a physical prototype, a virtual model is created. The purpose of the model is to resemble the behaviour of the physical product.
- All development/testing is made in the virtual product. Thus, you treat the model as you would if it was a physical product.
- The benefits are several:
  - Shorter time to market
  - Reduce number of costly prototypes
  - Increased innovation
  - Lower development costs
  - Higher quality
- … but also the challenges
  - Rethink development process
  - Trust the results
  - Educate personnel, new partners..



Volvo XC60





DYNA MORE NORDIC

What do you need?

PRE-PROCESSOR
Generates the FE-model
Applies boundary conditions etc

LSTC Livermore Software Technology Corp.
LS-PrePost

← History?
← Geometry
← Material
← Process

SOLVER
Solves the numerical model

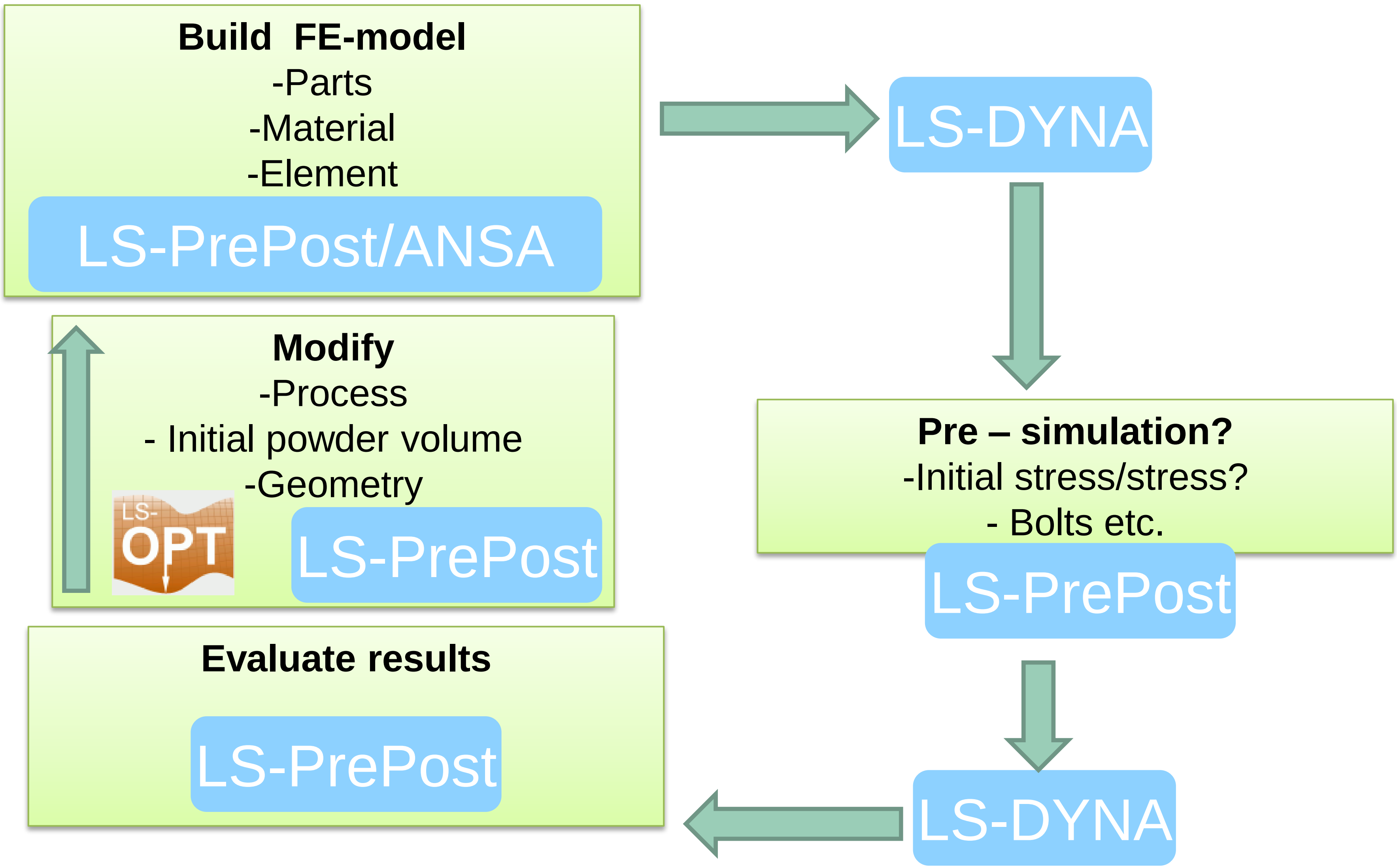
LSTC Livermore Software Technology Corp.
LS-DYNA

POST-PROCESSOR
View the results

LSTC Livermore Software Technology Corp.
LS-PrePost

← Dependence on analysis

Simulation process



Keywords and Elements

Keywords - Define Geometry

Input file (.k)

Newton's second law, F=ma, requires consistent units

	S1	S2	S3
length	meter	millimeter	millimeter
time	second	second	millisecond
mass	kilogram	tonne	kilogram
force	Newton	Newton	kiloNewton
Young's modulus of steel	210.0E+09	210.0E+03	210.0
density of steel	7.85E+03	7.85E-09	7.85E-06
gravitation	9.81	9.81E+03	9.81E-03

Keyword User's manual

LS-DYNA
KEYWORD USER'S MANUAL

August 2006
Version 971

GENERAL CARD FORMAT

The following sections specify for each keyword the cards that have to be defined. Each card is defined in its rigid format form and is shown as a number of fields in an 80 character string. **Most cards are 8 fields with a length of 10 and a sample card is shown below.**

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	PSID	A1	A2	A3	HAZ		
Type	I	I	F	F	F	I		
Default	none	none	1.0	1.0	0	1		
Remarks	1			2		3		

The type is the variable type and is either F, for floating point or I, for an integer. The default gives the value set if zero is specified, the field is left blank or the card is not defined. The remarks refer to comments at the end of the section. The card format is given above the card if it is other than eight fields of 10. Free formats may be used with the data separated by commas.

Input file - Keywords

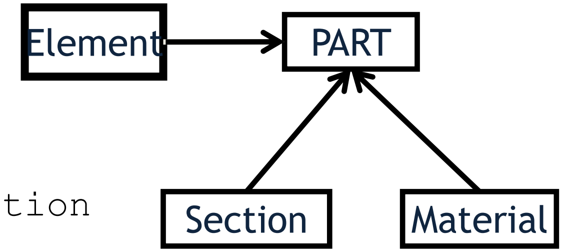
Input file (.k)

Comment card begins with $

```
*KEYWORD
*TITLE
Test example
$ Control cards govern entire model / simulation
*CONTROL_TERMINATION
*CONTROL_TIMESTEP
$ Define output of results
*DATABASE_BINARY_D3PLOT
*DATABASE_GLSTAT
$ Define section and material
*PART
$ Define element types and integration
*SECTION_SHELL
$ Define material properties
*MAT_ELASTIC
*MAT_FIBER
$ Define nodes and elements
*NODE
*ELEMENT_SHELL
$ Define loads and BC
*LOAD_NODE
*END
```

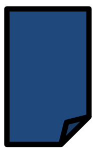
Mandatory (*KEYWORD)

Mandatory (*END)



Keyword Format

Input file (.k)



- Similar functions are grouped together under the same keyword
- A data block begins with a keyword and ends with the next keyword
- Keywords are left justified
- No distinction between lower and upper case letters
- Variables are right justified in their fields
- A ‘0’ or blank means that the variable will get the default value
- The decimal point is always written out for floating point variables

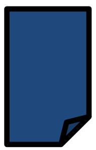
Keyword Format

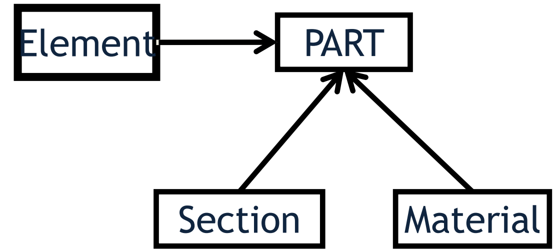
Input file (.k)

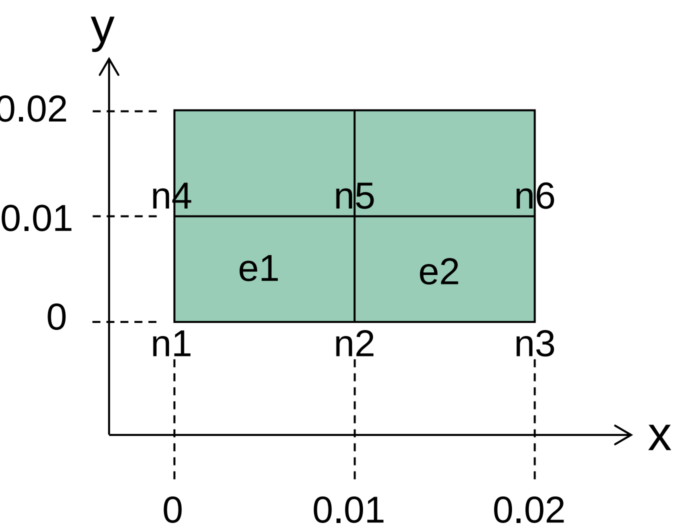
- Comments rows are written after a dollar sign in the first position
- *COMMENT keyword exist
- Do not use ‘tabs’ when editing or creating your file
- Line feed signs may cause problems when transferring files from Dos to Unix

Keywords - Define Geometry

Input file (.k)



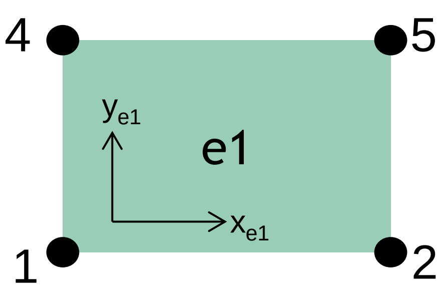





```
*NODE
$ NID   *     x        *      y       *      z       *
        1          0.00          0.00          0.00
        2        1.0E-2             0             0
        3          0.02             0             0
        4        2.0E-2        1.0E-2             0
 5, 0.01, 0.01, 0.0
 6, 0.02, 0.01, 0
$
$
*ELEMENT_SHELL
$ID, PID, n1, n2, n3, n4
  1,   1,  1,  2,  5,  4
  2,   1,  2,  3,  6,  5
```

Fixed format

Free format



Local coordinate system:

X_{e1}: from n1 to n2

y_{e1}: perpendicular to x_{e1} directed towards n3

- Some element formulations are more costly than others

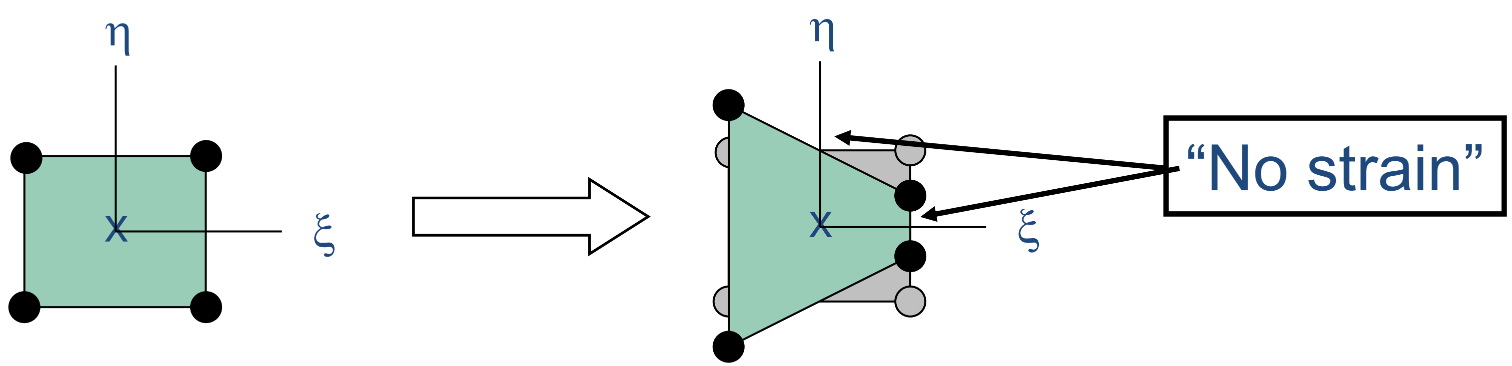
- Stresses and strains are calculated at the integration points
- Accelerations, velocities and displacements are evaluated at the nodes

Under Integrated and Fully Integrated Elements

- Most element formulations in LS-DYNA are under-integrated, i.e. the stresses and strains are only calculated in the mid-point of each element.
- Advantage: Computational efficiency. The material model is called once per integration point and time step.
- Disadvantage: The element formulation contains zero-energy modes (hourglass modes)

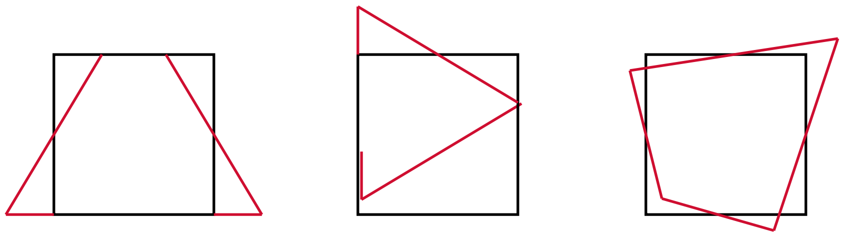
Under Integrated and Fully Integrated Elements

- The following element deformation does not yield any strains in the integration point, and thus no stress
- There is deformation, but no associated internal energy, hence the name zero-energy modes.
- These modes have to be suppressed using ”hourglass control”

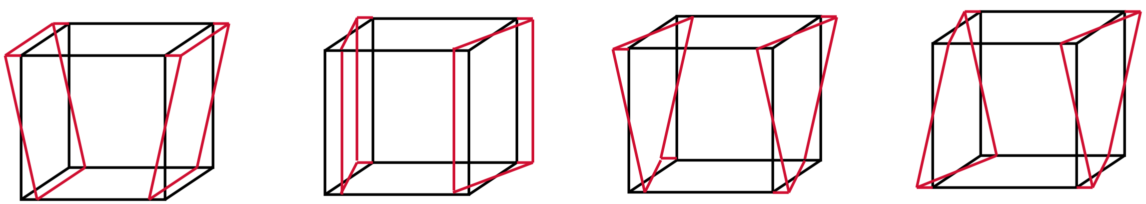


Hourglass Control

- Zero energy modes = Hourglass modes
- Hourglass controlled by *CONTROL_HOURGLASS and *HOURGLASS
- Hourglass modes for 1 point integration Q4 shell elements:



- Hourglass modes for 1 point integration solid elements:



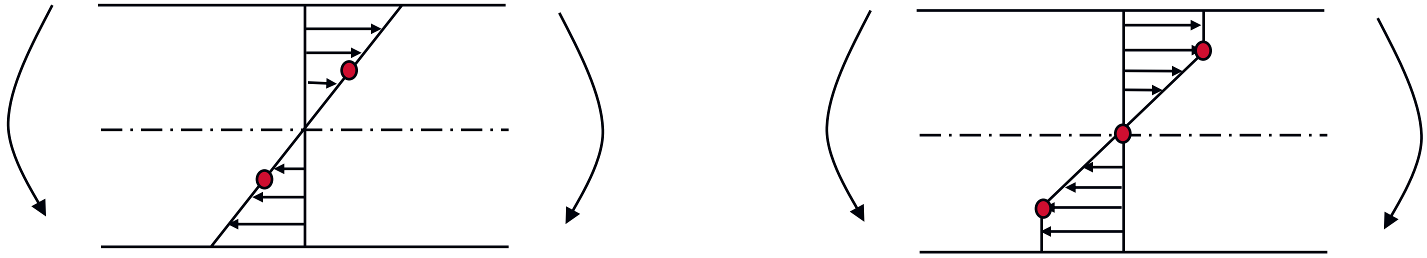
+ 8 more!

SECTION_SHELL

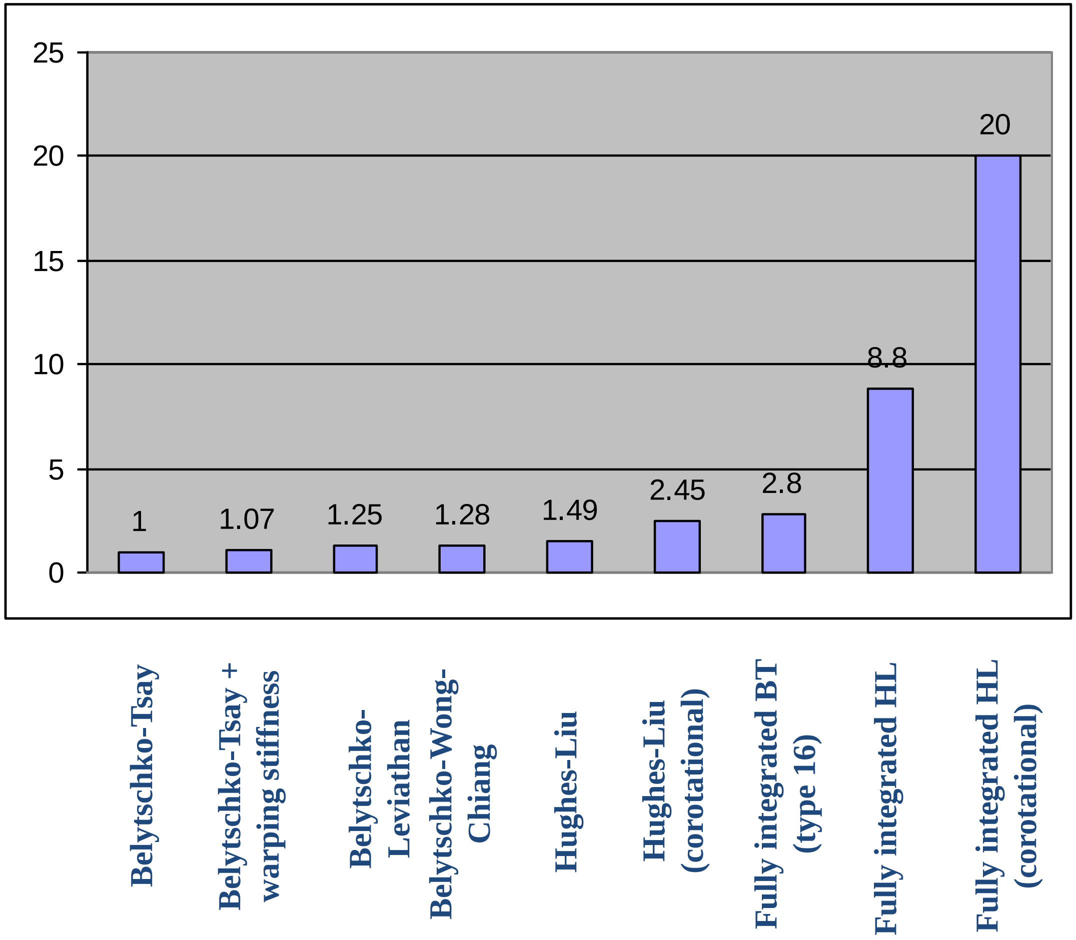
- Element formulation
  - Belytschko-Tsay
  - Belytschko-Wong-Chiang
  - Hughes-Liu
  - Belytschko-Leviathan
  - Fully integrated shells
  - Higher order shells 6/8 noded tria/quad
  - ……
- Element thickness
- Number of integration points through shell thickness

Elements (shell) - NIP

- 1 point integration through the thickness gives a membrane element
- 2 point integration through the thickness is the default (sufficient for a linearly elastic material)
- For plastic bending behaviour, at least 3 points are needed through the thickness
- 5 points recommended for sheet metal stamping.
  7 points for springback
- Use odd numbers to include the neutral axis



Element Performance



*CONTROL_ACCURACY

- Invariant node numbering
  - particularly important when large shear forces are present in an element
- 2nd order stress update
  - spinning bodies such as turbine blades, rotating tires
  - sometimes for stiffness hourglass control
  - implicit solutions with large strains in each step

Material Models

- Over 200 models for various applications exists in LS-DYNA.
- Determine the stress based on strain, strain-rate, temp etc.
- Not materials, but models subject to restrictions:
  - Load magnitude
  - Deformation speed (strain rate)
  - Temperature
- The models are defined by material parameters
  - E, ν, ρ, etc.

Hypoelasticity

Hypoelasticity relates a strain rate to a corresponding stress rate

$$\dot{\boldsymbol{\sigma}} = \mathbf{C} : \mathbf{D}$$

Hooke's law:

$$\sigma = E\varepsilon$$

Stress is incrementally updated from the strain rate with aid of the constitutive tensor C

$$\Delta\sigma = \dot{\sigma}\Delta t$$

$$\Delta\varepsilon = D\Delta t$$

Most of the materials in LS-DYNA are based on this formulation for the elastic response.

Merits and drawbacks (theoretical)

+

- It is fairly straightforward to use and easy to implement in a finite element code

-

- The response is path-dependent, the stress for a closed strain cycle can be nonzero, it should be used when the elastic deformation is relatively small
- It is difficult to deal with anisotropic constitutive models because the constitutive tensor C is restricted to be isotropic for nonlinear analysis. This is however solved in LS-DYNA with a co-rotational update.

Hyperelasticity - definition

- A material is hyperelastic if the internal work is independent of the deformation path.
- It is characterized by the existence of a strain energy function that is a potential for the stress.

$$\mathbf{S} = 2\frac{\partial \psi(\mathbf{C})}{\partial \mathbf{C}} = \frac{\partial W(\mathbf{E})}{\partial \mathbf{E}}$$

$\mathbf{S}$ Second Piola Kirchhoff stress tensor

$\mathbf{E}$ Green strain tensor

$\mathbf{C}$ Right Cauchy-Green tensor

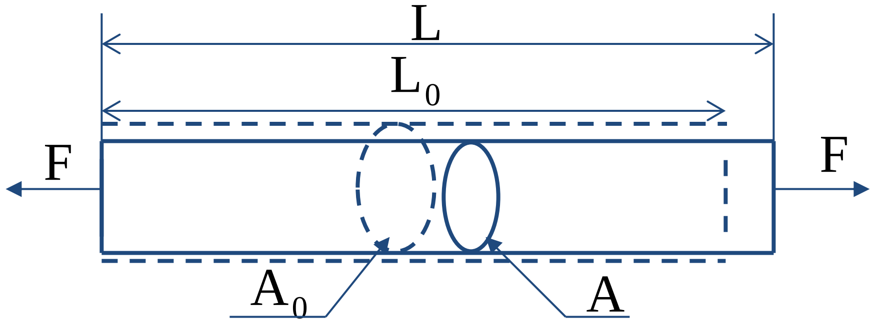
- Typically used when elastic deformation is substantial, e.g. rubber.

Stress and strain – uni-axial deformation

<u>Tensile test:</u>

Engineering stress $\sigma_E = F / A_0$

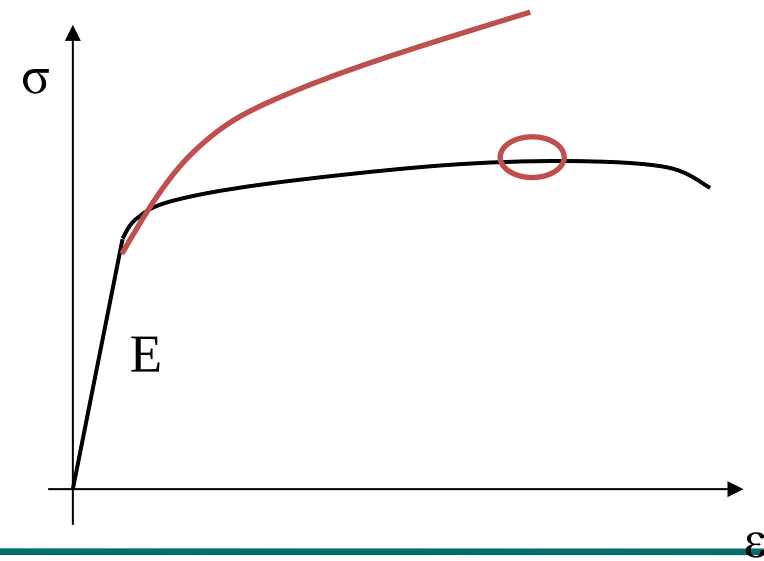
Engineering strain $\varepsilon_E = (L - L_0) / L_0$



<u>In LS-DYNA:</u>

True stress $\sigma = F / A$

True strain $\varepsilon = \ln(L / L_0)$



<u>Elastic response:</u>

Hooks law: $\sigma = E\varepsilon$

Area reduction: $A = A_0(1 - 2\nu\varepsilon)$

Elasto-plasticity in 3-D – multi-axial deformation

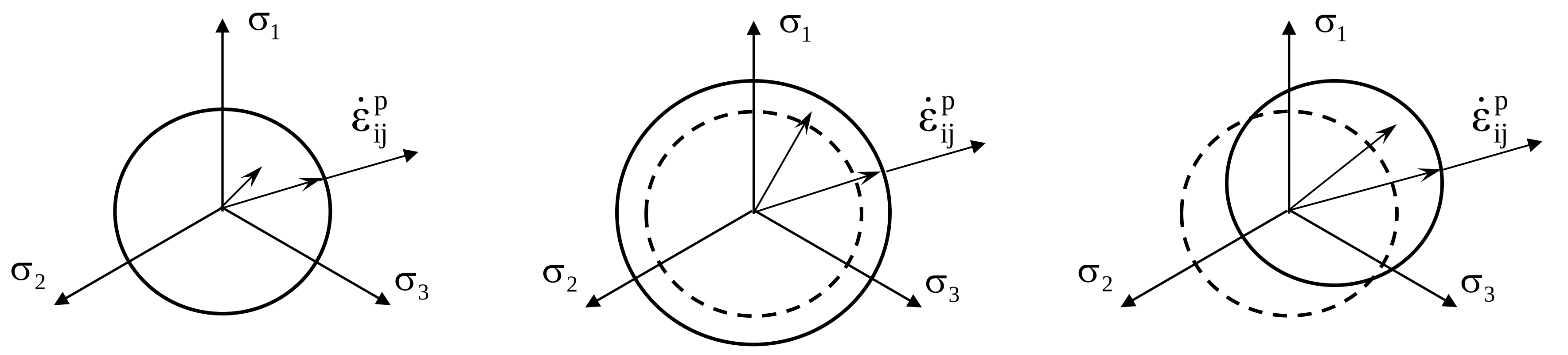
Deviatoric stress

Volumetric stress

Stress decomposition……….: $\sigma_{ij} = \overbrace{S_{ij}} + \overbrace{\delta_{ij}\sigma_{kk}/3}$

Von Mises yield criterion….: $f = \sqrt{\frac{3}{2} s_{ij} s_{ij}} - \sigma_y(\varepsilon_p)$

Plastic strain…………………: $\dot{\varepsilon}_{ij}^{p} = \lambda \dfrac{\partial f}{\partial S_{ij}}$



Perfect plasticity | Isotropic hardening | Kinematic hardening

Elastic-visco-plastic material

*MAT_PIECEWISE_LINEAR_PLASTIC

MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
C	*P*	LCSS	*LCSR*	*VP*			
EPS1	EPS2	. . . .					
EP1	ES2	. . . .					

For: Metals, loading exceeding yielding stress, rate effects
In: All element types

Theory: Isotropic plasticity model with visco-plasticity option

E	Young's Modulus	*C,P*	Strain-rate parameters
RO	Density	LCSS	Load curve for $\sigma_y(\varepsilon_p)$
PR	Poisson's Ratio	*LCSR*	Load curve for strain-rate scaling
SIGY	Yield stress	*VP*	Visco-plastic flag
ETAN	Tangent modulus	EPS1…	Piecewise linear def.

Elastic-visco-plastic material

Activating visco-plasticity:
(σ_y^s =static yield stress)

$\mathrm{C,P} = 0 \Rightarrow$ No visco-plastic effects

$\mathrm{C,P} \neq 0, \mathrm{VP} = -1 \Rightarrow$ Scale σ_y^s by: $1+\left(\frac{\dot{\mathrm{e}}}{\mathrm{C}}\right)^{1/\mathrm{P}}, \mathrm{e}_{\mathrm{ij}} = \varepsilon_{\mathrm{ij}} - \delta_{\mathrm{ij}}\varepsilon_{\mathrm{kk}}$

$\mathrm{C,P} \neq 0, \mathrm{VP} = 0 \Rightarrow$ Scale σ_y^s by: $1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/P}, \dot{\varepsilon} = \sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$

$\mathrm{C,P} \neq 0, \mathrm{VP} = 1 \Rightarrow$ Yield stress is given by: $\sigma_y = \sigma_y^s \left\{ 1+\left(\frac{\dot{\varepsilon}_{eff}^{p}}{C}\right)^{1/P} \right\}$

VP=1 is recommended as it uses a consistent visco-plastic theory

Elastic-plastic material with Bauschinger efftect

***MAT_PLASTIC_KINEMATIC**

MID	RO	E	PR	SIGY	ETAN	BETA
SRC	SRP	FS	VP			

For: Metals under large loading
In: All element types

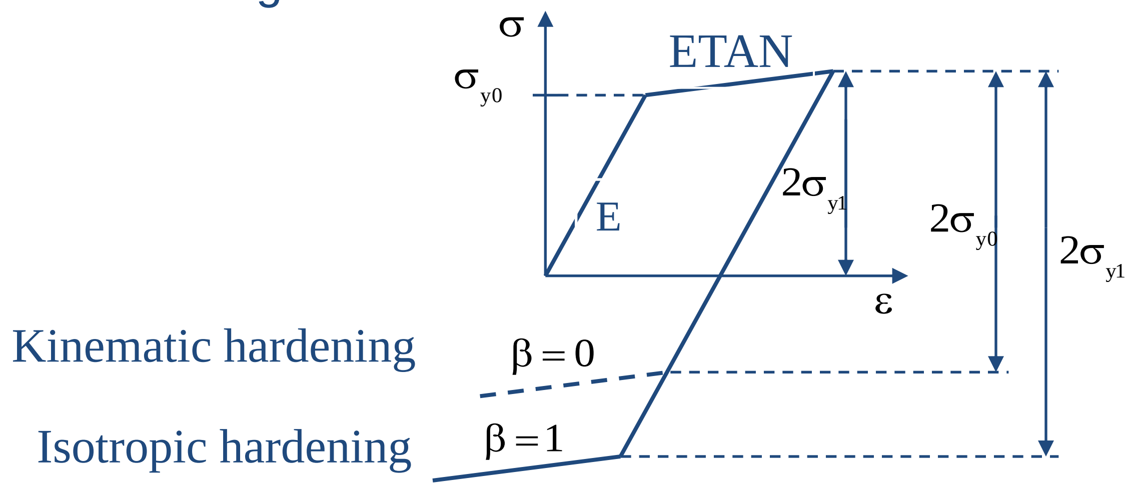
Theory: Isotropic and kinematic hardening plasticity, viscoplastic

E	Young's Modulus
RO	Density
PR	Poisson's Ratio
SIGY	Yield stress
ETAN	Tangent modulus

BETA	Hardening parameter	FS	Failure strain
SRC	Strain rate parameter C	VP	Rate formulation flag
SRP	Strain rate parameter P		

Elastic-plastic material with Bauschinger efftect

Definition of material hardening:



Other models with kinematic hardening:

***MAT_PLASTIC_GREEN-NAGHDI_RATE**

***MAT_ANISOTROPIC_VISCOPLASTIC**

*EOS

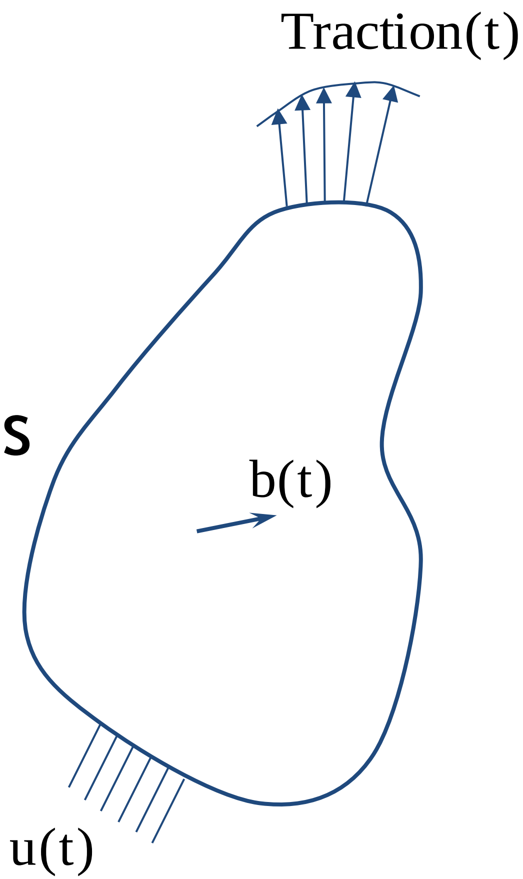
- Certain material models only solve for the deviatoric part of the stress tensor
- An Equation of State (EOS) is required to find the pressure part of the stress tensor
- Mostly used in conjunction with fluid-like behaviour (high explosives, airbag inflation …)
- Solid elements only

Boundary/Initial Conditions

Initial and Boundary Conditions

- Variation in time using load curves
- Variation in space
- Arbitrary directions using
  - Local coordinate systems
  - Vectors

  But limited to cartesian coordinates



*LOAD_NODE[_SET|_POINT]

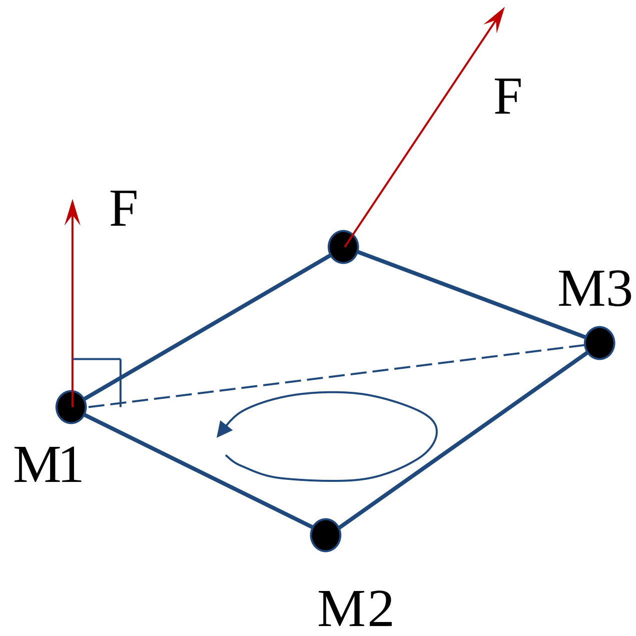
NODE/NSID DOF LCID SF CID M1 M2 M3

Nodal loads for one node or a set of nodes

DOF	Direction of load in current coordinate system
LCID	Load curve ID for variation in time
SF	Scale load curve amplitude
CID	Define a local coordinate system
M1-M3	Follower force definition

Singularities at point loads may be a problem.

Multiple load cards are accumulated.



*INITIAL_STRESS[_BEAM|_SHELL|_SOLID]
*INITIAL_STRAIN[_SHELL|_SOLID]

Initialise the state of stress and strain in elements

Normally used to carry results obtained in one simulation to another.
- Multistage forming
- Forming -> Crash

Keyword data normally generated automatically by preprocessors.

Kinematic Conditions

- Prescribe motion in the model

- *BOUNDARY: w.r.t cartesian coordinates
  - Fixed supports
  - Symmetric boundaries
- *CONSTRAINED: internal definitions
  - Mechanical Joints
  - Merging shell-brick elements
  - Define rigid bodies

*BOUNDARY_PRESCRIBED_MOTION[_NODE|_SET|_RIGID]

ID	DOF	VAD	LCID	SF	VID	DEATH	BIRTH

Apply nodal displacement, velocity, or acceleration to the model; translations or rotations

DOF	Direction of load, global or local direction, see manual!
VAD	Type of load
LCID	Load curve ID for variation in time
SF	Scale amplitude of the loadcurve
VID	Vector ID for vector to be used if DOF=4 or 8
DEATH/BIRTH	Active range of time for this boundary condition

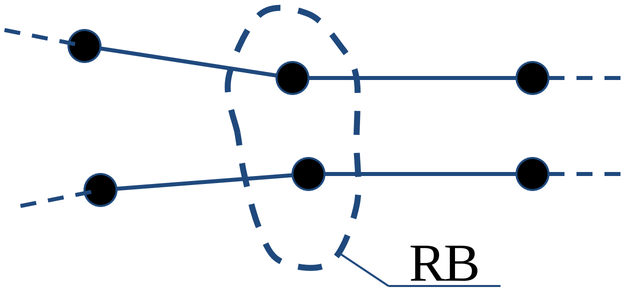
Use the _RIGID option for rigid bodies.
For local directions with rigid bodies see the MAT_RIGID keyword.

*CONSTRAINED_NODAL_RIGID_BODY

PID CID NSID PNODE IPRT

Create a new rigid body using existing nodes

PID	Part id req. is a unique one
CID	Coordinate system for output
NSID	Node set
PNODE	Optional centre node
IPRT	Print flag



For spot-welds and other types of rigid connections.

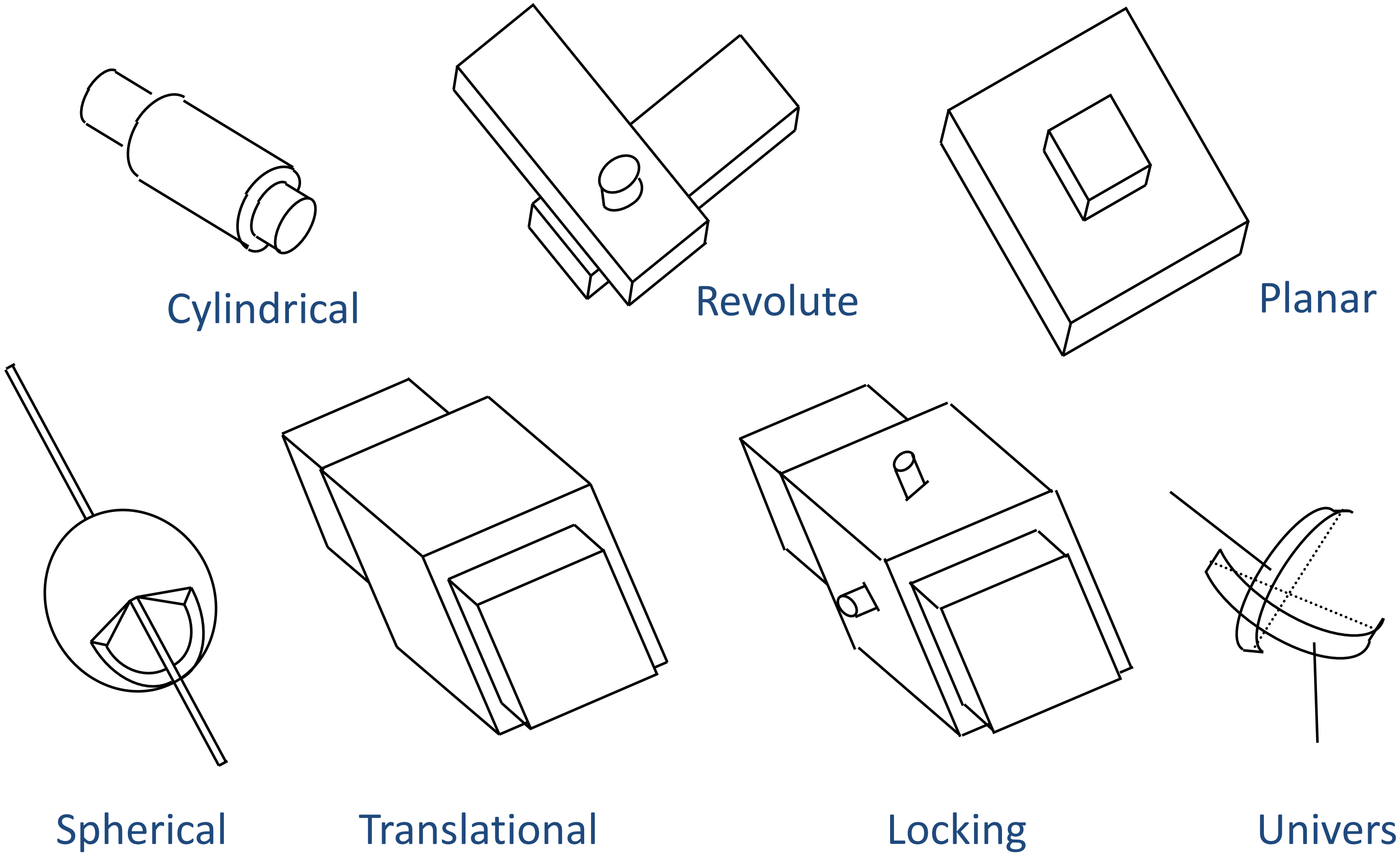
*CONSTRAINED_JOINT_"JOINTTYPE"

N1 N2 N3 N4 N5 N6 RPS DAMP

Define mechanical joints between rigid bodies
N1-N6 Nodes in the rigid bodies
RPS Scale the penalty stiffness
DAMP Dynamic damping

N1,N3,N6 in RB1. N2,N4,N6 in RB2.
Place the nodes in one RB far apart.
(N1,N2) etc. initially coincident, except universal joint, read the manual!
Motor and gear joints are available for advanced mechanisms.

*CONSTRAINED



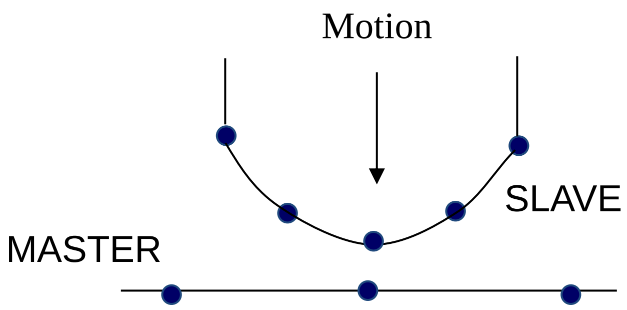
Contacts

Some of the available contacts *CONTACT_option_option_…

AIRBAG_SINGLE_SURFACE
AUTOMATIC_GENERAL
AUTOMATIC_GENERAL_INTERIOR
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_NODES_TO_SURFACE_TIEBREAK
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK
CONSTRAINT_NODES_TO_SURFACE
CONSTRAINT_SURFACE_TO_SURFACE
DRAWBEAD
ERODING_NODES_TO_SURFACE
ERODING_SURFACE_TO_SURFACE
FORCE_TRANSDUCER_CONSTRAINT
FORCE_TRANSDUCER_PENALTY
FORMING_NODES_TO_SURFACE_TIEBREAK
FORMING _ONE_WAY_SURFACE_TO_SURFACE
FORMING _SURFACE_TO_SURFACE
NODES_TO_SURFACE
NODES_TO_SURFACE_INTERFERENCE
ONE_WAY_SURFACE_TO_SURFACE
RIGID_NODES_TO_RIGID_BODY
RIGID_BODY_ ONE_WAY_TO_RIGID_BODY
RIGID_BODY_TWO_WAY_TO_RIGID_BODY
SINGLE_EDGE
SINGLE_SURFACE
SLIDING_ONLY
SLIDING_ONLY_PENALTY
SURFACE_TO_SURFACE
SURFACE _TO_SURFACE_INTERFERENCE
TIEBREAK_NODES_TO_SURFACE
TIEBREAK_ SURFACE _TO_SURFACE
TIED_NODES_TO_SURFACE
TIED_NODES_TO_SURFACE_OFFSET
TIED_SHELL_EDGE_TO_SURFACE
SPOTWEALD
SPOTWEALD_WITH_TORSION
TIED_ SURFACE _TO_SURFACE
TIED_ SURFACE _TO_SURFACE_OFFSET

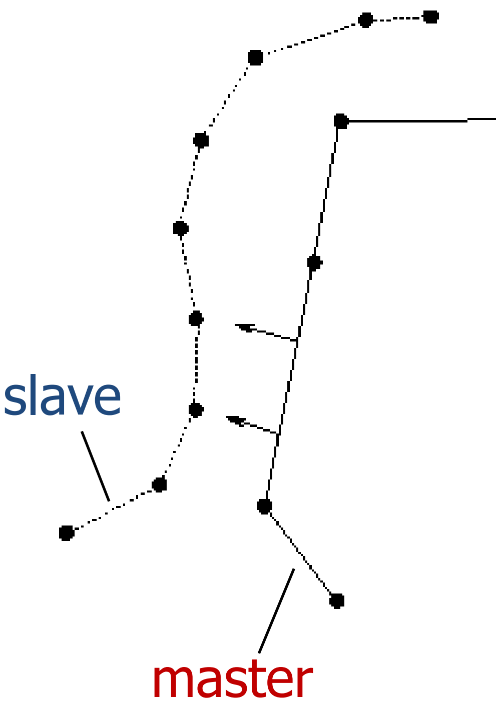
Contact

- A way of treating interaction between different parts
- Contacts are defined by sets (node/part/segments) or a box
- Generally there is a master side and a slave side of the contact
- The master side can be a mathematically described with a geometrical surface (rigid)
- The thickness of shells are normally taken into account
- Most recommended contacts are based on the penalty method
- Several contacts treating special applications exists
- Old contact types kept for compatibility reasons



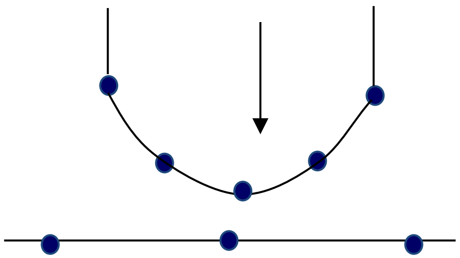
Interesting Keywords for Contacts

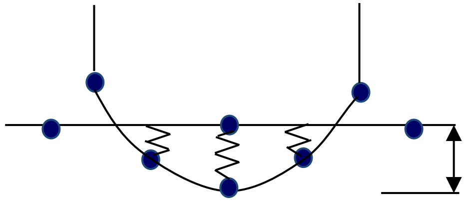
- **Contacts in LS-DYNA is affected by many different keywords**
  - *SECTION_SHELL (Shell thicknesses, middle/top/bottom surface meshed)
  - *MAT_xxx (Penalty stiffness, E, pr, dens)
  - *DEFINE_FRICTION (Friction behavior between parts)
  - *PART_CONTACT (Contact behavior for parts)
  - *CONTROL_CONTACT (Overall contact behavior)
  - *CONTACT_xxx (Contact definition)



- The different parameters on different keywords might be used, depending on contact type.
- The parameter might have a different meaning depending on contact type use.
- Makes contact definition tricky in LS-DYNA! Some of the most interesting parameters found on different cards will be examined in this presentation.

Important contact parameters: penalty method (default)





Contact force

$F_i = \delta_i\, k$

k= interface spring stiffness

Solid elements

$$k = \frac{cKA}{V/A}$$

Shell elements

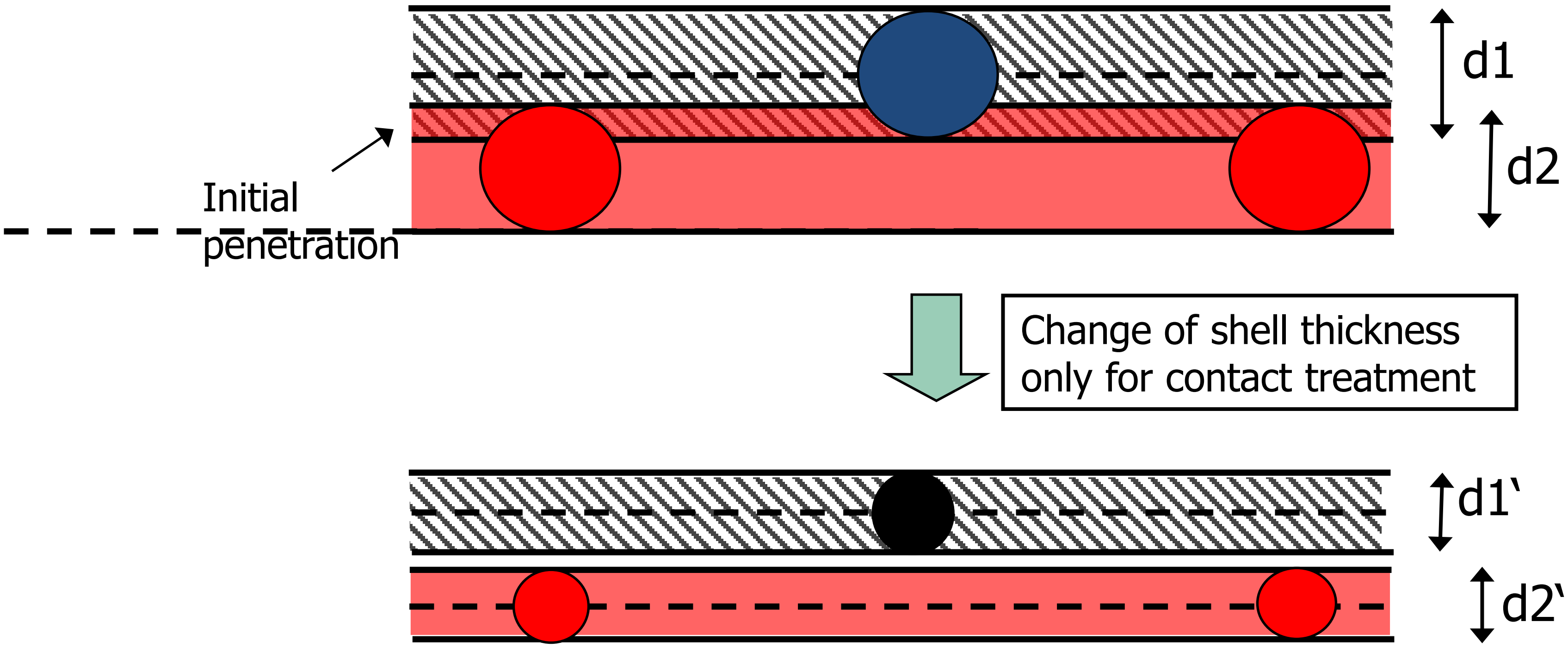
$$k = \frac{cKA}{diagonal}$$

K= bulk modulus

c = penalty factor

The time step of the analysis is determined by LS-DYNA from the elements of the FE-mesh without considering the contact interfaces!

Contact Thickness and Initial Penetrations



Important contact parameters: friction

- Sliding friction – FS, FD, DC and VC
  - Defined in keyword *CONTACT
  - Based on Coulomb friction
  - Default values gives no friction

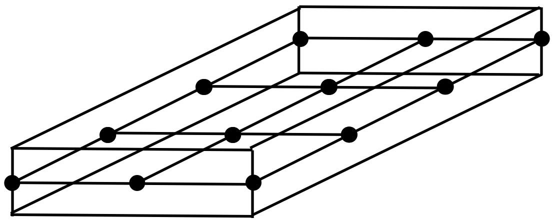
$$\mu_c = FD + (FS - FD)e^{-DC\left|V_{rel}\right|}$$

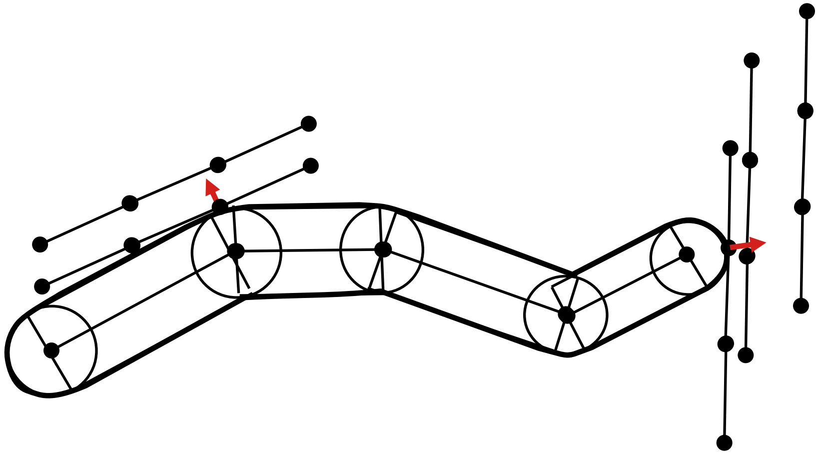
  - FS and FD are static respectively dynamic friction coefficient
  - DC - decay coefficient
  - If FD and FS not are equal, then FD should be less than FS and DC nonzero
  - VC is the coefficient for viscous friction and limits the friction force (typically $3^{-½}$ of yield stress)
  - Viscous damping VDC improves stability. For metal contacts use 20% and for soft material 40-60%

Automatic contacts without self contact

- *CONTACT_AUTOMATIC_NODES_TO_SURFACE
- *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE

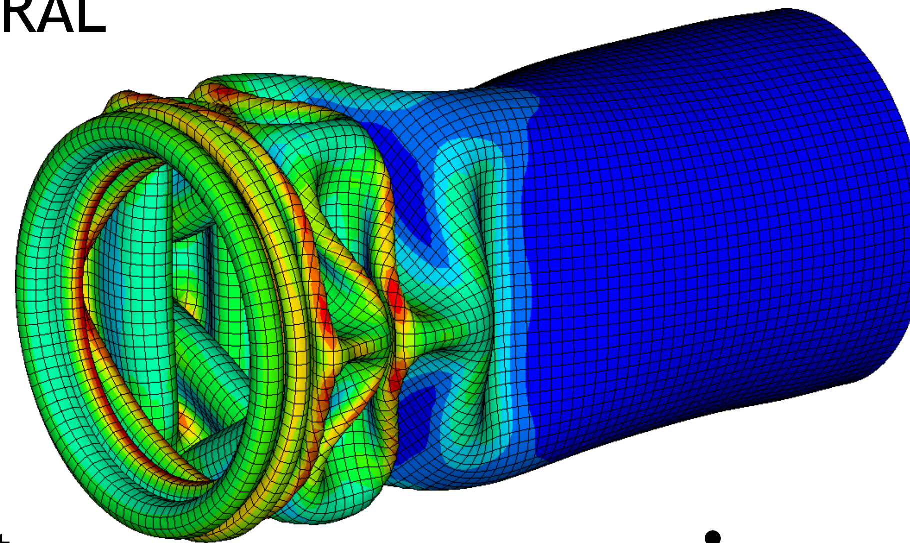
  - Thickness taken into account
  - Contact surface is offset by half thickness from mid-plane
  - Orientation of segments not needed
  - Contact from both sides
  - Handles disjoint meshes
  - Applies a smooth surface based on a radii at the edges (including free edges)
  - Initial penetrations are detected
  - Possible to change or scale contact thickness
  - Friction and damping available



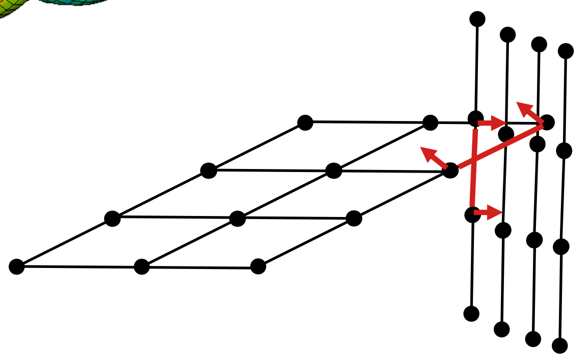


Single-surface contacts (self contact)

- *CONTACT_AUTOMATIC_SINGLE_SURFACE
- *CONTACT_AUTOMATIC_GENERAL

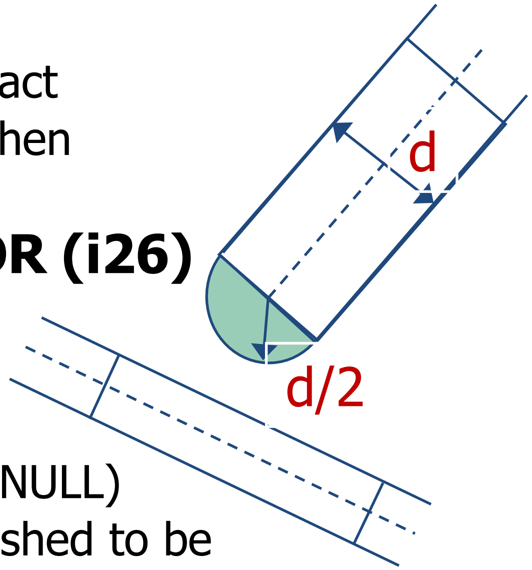


  - Same features as the automatic contacts
  - Only require definition of the slave surface
  - Include self contact
  - Sensitive to initial penetrations
  - Possible to use only one contact definition forthe complete model
  - Beam and edge to edge contacts are included *CONTACT_AUTOMATIC_GENERAL



Edge/Beam Contacts

- ***CONTACT_AUTOMATIC_GENERAL (26)**
  - exclude interior edges
  - entire length of each **exterior** edge is checked for contact
  - OBS, the edge cylinder is not affected by OPTT or TH when using part_contact.
- ***CONTACT_AUTOMATIC_GENERAL_INTERIOR (i26)**
  - like *CONTACT_AUTOMATIC_GENERAL,
  - but interior edges are treated like exterior edges
  - Alternative way to treat edge contact:
  - creating null beam elements (*ELEMENT_BEAM,*MAT_NULL) approximately 1mm in diameter along every edge wished to be considered for edge-to-edge contact and including these null beams in a separate AUTOMATIC_GENERAL contact



- ***CONTACT_SINGLE_EDGE (22)**
  - Treats only edge-to-edge contact
  - no thickness offset at the contact edge
- ***CONTACT_xxx_MORTAR ()**
  - edge-to-edge contact
  - no thickness offset at the contact edge

Tied contacts

- CONTACT_TIED_NODES_TO_SURFACE
- *CONTACT_TIED_SURFACE_TO_SURFACE
- *CONTACT_TIED_SHELL_EDGE_TO_SURFACE
  ….._OFFSET

  - Possibility to "tie" nodes to a surface (segment)
  - NODES_... and SURFACE_... ties translational d.o.f
  - SHELL_EDGE_.. ties translational and rotational d.o.f
  - Constraint based. Thus, will not work with rigid bodies.
  - …_OFFSET allows for a segment thickness and is penalty based.
  - …_TIEBREAK_... has failure options.
  - Can be used to model glue, spotwelds etc.

Control Cards & Execution

- The purpose is:
  - Activate solution options;
    implicit solution, adaptive remeshing, mass scaling …
  - Change default values on options and parameters

- Remember that:
  - Ordering between them and position are arbitrary
    Good practise is to put them first in your input file
  - Do not use more then one control card of each type
  - All control cards are optional except
    *CONTROL_TERMINATION

Control Card Default Values

- Default values exist for all options and most parameters
- Control cards change default values globally
- Default values are defined hierarchically
  The order between them are:
  - LS-DYNA defaults
  - Control card input
  - Individual Keyword input
- Set your defaults with the control cards and change the keyword input where default values not should be used
- Input of ‘0’ will normally give the default value which is shown in the manual

- Always consider the following control cards since they can strongly affect your results or output
  - *CONTROL_ACCURACY
  - *CONTROL_CONTACT
  - *CONTROL_ENERGY
  - *CONTROL_HOURGLASS
  - *CONTROL_SHELL
  - *CONTROL_SOLID
  - *CONTROL_TERMINATION
  - *CONTROL_TIMESTEP

Implicit Solution Types

- **Linear Analysis**
  - static or dynamic
  - single, multi-step
- **Eigenvalue Analysis**
  - frequencies and mode shapes
  - linear buckling loads and modes
  - modal analysis: extraction and superposition
  - Dynamic analysis by modal superposition (971)
- **Nonlinear Analysis**
  - Newton, Quasi-Newton, Arclength solution
  - static or dynamic

- default LS-DYNA: **static and nonlinear!**

Output Files

- Binary files (can be viewed in LS-PrePost)
  *DATABASE_BINARY_Option
- ASCII files for more detailed output
  (graphs can be shown in LS-PrePost)
  *DATABASE_Option
- Data in the binary and ASCII files is controlled by
  *DATABASE_EXTENT_Option
  *DATABASE_HISTORY_Option
- Control files (d3hsp)
- Message files (messag)

Output Files

- **D3PLOT (database for complete output states)**
- **D3DUMP (complete database for restart)**
- **RUNRSF (running restart file, overwritten)**
- D3PART (as D3PLOT but includes just specified parts)
- D3THDT (database for time history data of element subsets)
- D3DRLF (dynamic relaxation database)
- D3MEAN (CFD database)
- INTFOR (database for output of contact interface data)
- XTFILE (extra time history data)
- D3EIGV (modal data from eigenvalue analysis)
- D3CRCK (crack data from Winfrith concrete model)

ASCII Output Files

- **GLSTAT (global data)**
- **MATSUM (material energies)**
- **RCFORC (resultant interface forces)**
- **SLEOUT (sliding interface energy)**
- **NODOUT (nodal point data)**
- **ELOUT (element data)**
- SECFORC (cross section forces)
- RWFORC (rigid wall forces)
- SSSTAT (subsystem data)
- DEFORC (discrete elements)
- NCFORC (nodal interface forces)
- DEFGEO (deformed geometry)
- SPCFORC (SPC reaction forces)
- NODFOR (nodal force groups)
- ABSTAT (airbag statistics)
- BNDOUT (boundary condition force/ energy)
- RBDOUT (rigid body data)
- GCEOUT (geometric contact entities)
- JNTFORC (joint force)
- SBTOUT (seat belt output)
- AVSFLT (AVS database)
- SWFORC (nodal constraint reaction forces)
- MOVIE
- MPGS
- TRHIST (trace particle history)
- TPRINT (thermal output)
- SPHOUT (SPH data)

Demonstrate LS-PrePost

- PreProcessing
- PostProcessing

Thank you!