Die große Elementbibliothek in LS-DYNA

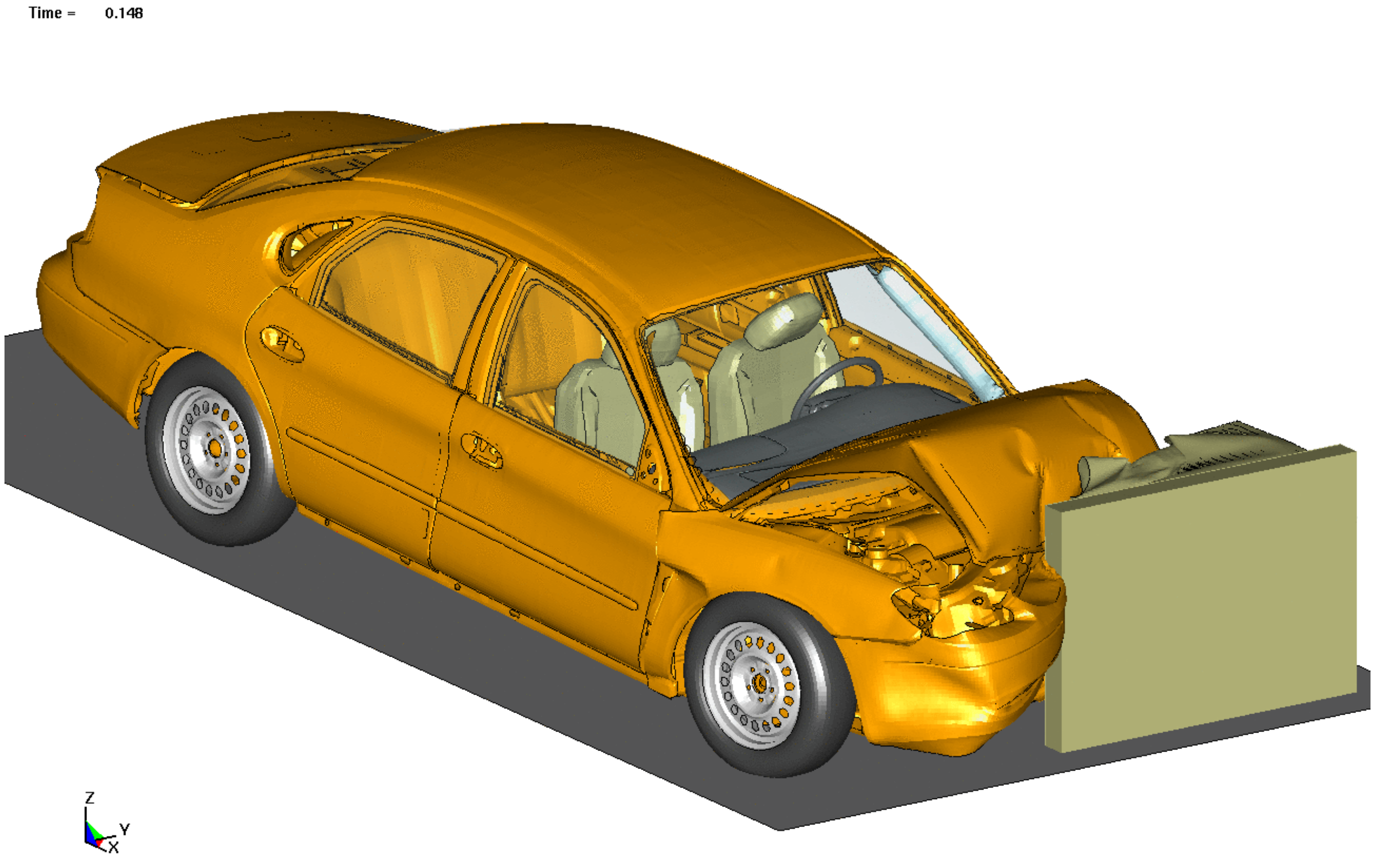
-

Wann nimmt man was?

Dr.-Ing. Ulrich Stelzmann
CADFEM Service

A typical model for LS-DYNA may have Millions of elements.

But which element typ should be used for what?



*ELEMENT

Elementdefinition in LS_DYNA:

*ELEMENT_BEAM

*ELEMENT_DIRECT_MATRIX_INPUT

*ELEMENT_DISCRETE

*ELEMENT_INERTIA

*ELEMENT_MASS

*ELEMENT_PLOTEL

*ELEMENT_SEATBELT

*ELEMENT_SHELL

*ELEMENT_SOLID

*ELEMENT_SPH

*ELEMENT_TSHELL

Shell Elements
(thin shells)

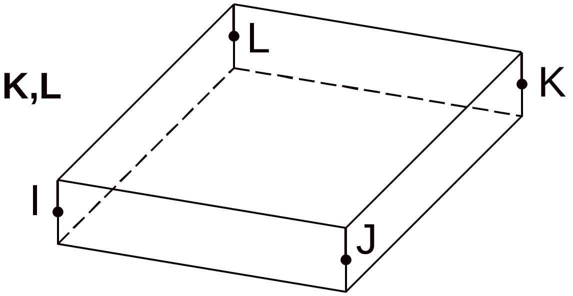
*ELEMENT_SHELL
*SECTION_SHELL

*ELEMENT_SHELL

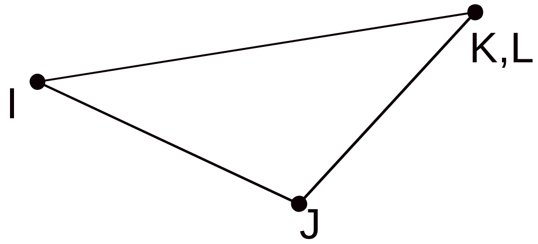
*SECTION_SHELL:

Shell element:

with the nodes I,J,K,L



triangular Shell element:



à ***Shell thickness***

- the shell thickness is defined in *SECTION_SHELL, t1 until t4 (commonly four times the same value)

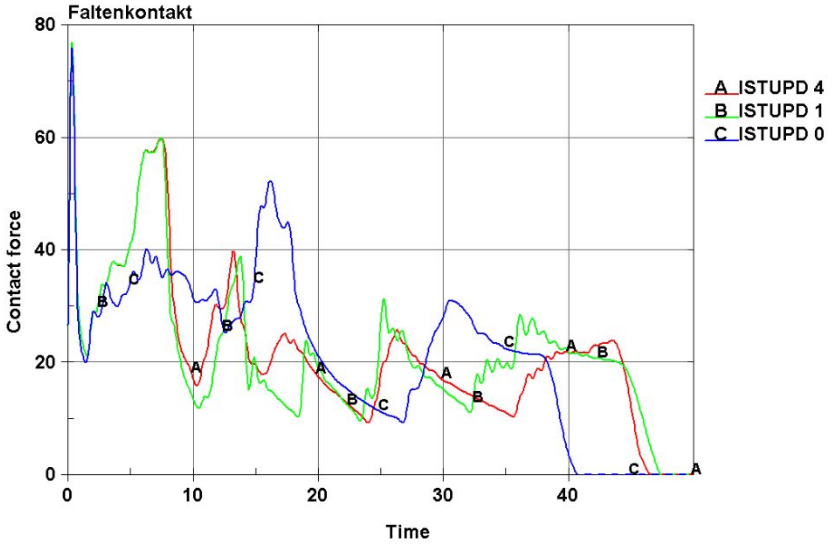
- additional input is possible in the element card, with *ELEMENT_SHELL_THICKNESS; this overwrites the thickness from section definition

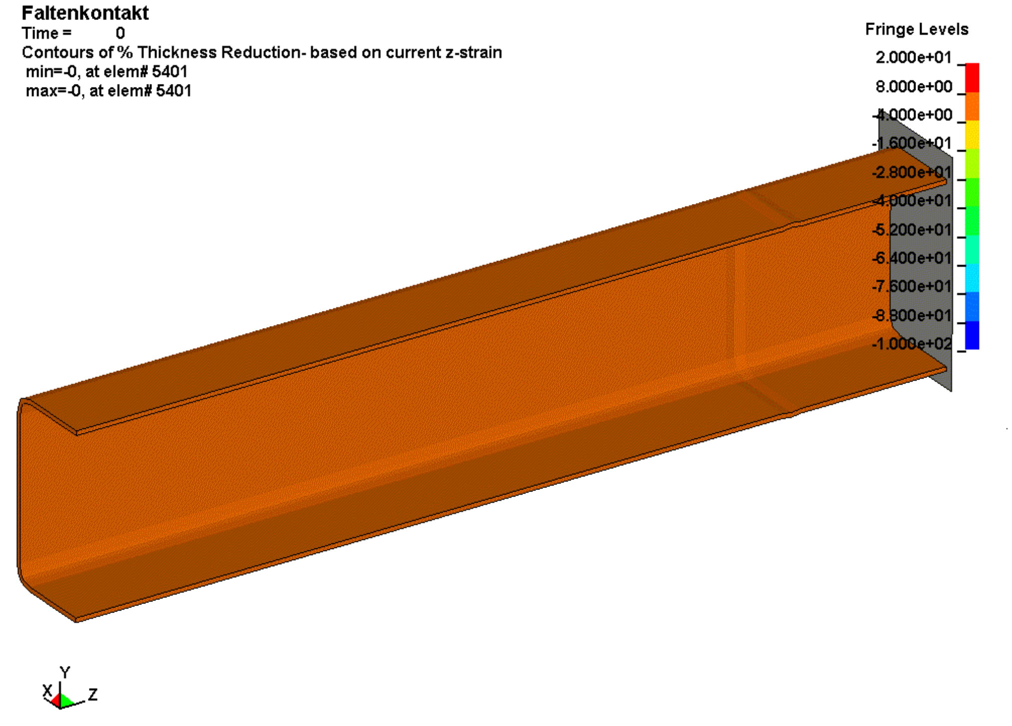
- in order to consider thickness change of the shell due to membrane straining one has to set *CONTROL_SHELL, istupd (important for large strains)

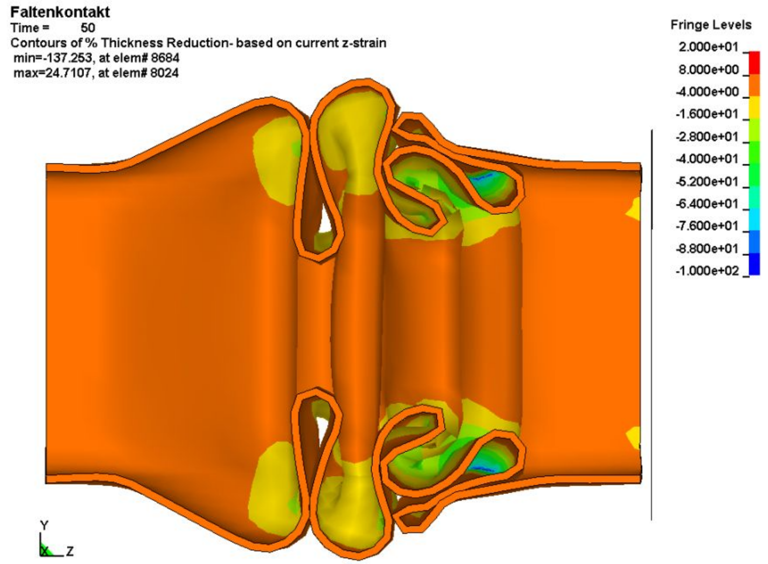
recommended: **istupd=4** (for elastic-plastic material models, only the plastic part of membrane strain will change the thickness)

*ELEMENT_SHELL

The change of shell thickness may have big influence on stiffness if large membran strain exist.
The simplest example is a tensile test.

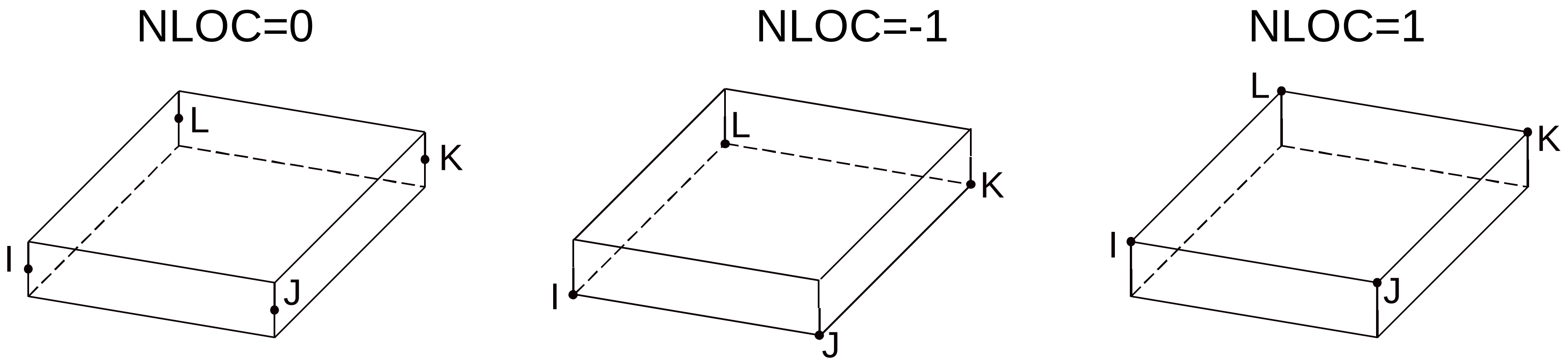






Shell Offset

Offset in shell elements: ***SECTION_SHELL, NLOC**



Since LS971 the reference surface for all shell elements can be set to an arbitrary location, not only between NLOC=-1 and 1 (lower and upper shell surface) but also outside the shell with |NLOC|>1.

Since R3.2 this is considered in contact (normal and AUTOMATIC) by setting:
***CONTROL_SHELL**, third card, third field=**1**

LS-PREPOST can not display this offset.

*ELEMENT_SHELL

Number of integration points

- most shell elements (other than type 6, 7 and 16) have 1 integration point in plane,
  shell element types 6, 7 and 16 have 4 integration points in plane
- The number of the integration points across the thickness is variable and must be defined in
  *SECTION_SHELL, NIP
  Default is nip=2, which is **NOT** sufficient for most applications.
- use the following rules to define the number of integration points throughout thickness:
  - for membranes: 1 integration point
  - for linear material: 2 integration points sufficient

    à Attention: Stress output not accurate on shell top- and bottom surface

    If stress output at top and bottom is important we recommend:

    NIP=5 (*section_shell)
    INTGRD=1 (*control_shell, Lobatto integration)
  - in case of non-linear material 3 to 5 (or more) integration points are needed
  - for high accuracy in springback simulations, use 7 to 9 through thickness integ. Points

à Attention: If NIP is larger than 10, trapezoidal integration rule is used, which is much less accurate than Gauss integration. Use this only for composite shells.

*ELEMENT_SHELL

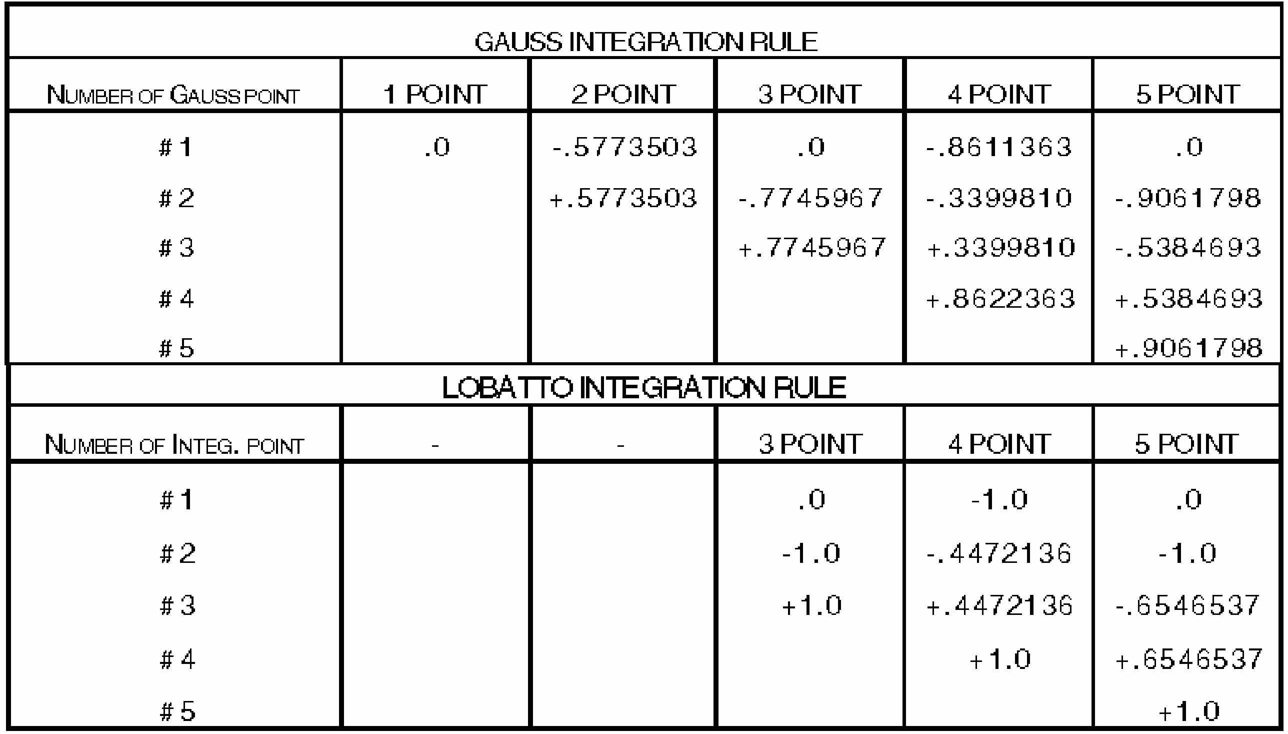
integration rule across the thickness

- usually the Gauss integration rule is used for thickness integration
- although the outer integration points are not located on the surface, this method gives accurate results and is commonly used.
- thickness integration can be switched from Gauss to Lobatto integration by setting *CONTROL_SHELL, intgrd=1

In this case the inner and outer integration points are located on the shell surface. This feature is only available for 3-10 integration points throughout the thickness

Especially with nip=3 Lobatto integration is not as good as Gauss.

Recommend: intgrd=0 and nip=5



GAUSS INTEGRATION RULE					
NUMBER OF GAUSS POINT	1 POINT	2 POINT	3 POINT	4 POINT	5 POINT
# 1	.0	-.5773503	.0	-.8611363	.0
# 2		+.5773503	-.7745967	-.3399810	-.9061798
# 3			+.7745967	+.3399810	-.5384693
# 4				+.8622363	+.5384693
# 5					+.9061798

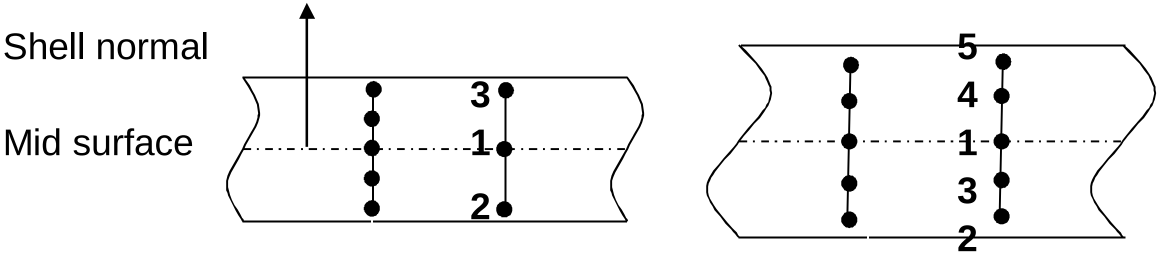
LOBATTO INTEGRATION RULE					
NUMBER OF INTEG. POINT	-	-	3 POINT	4 POINT	5 POINT
# 1			.0	-1.0	.0
# 2			-1.0	-.4472136	-1.0
# 3			+1.0	+.4472136	-.6546537
# 4				+1.0	+.6546537
# 5					+1.0

*ELEMENT_SHELL

Output of integration point results

- with *DATABASE_EXTENT_BINARY; *maxint*, declare the number of integration points, for which LS-DYNA writes results to the binary database d3plot

à for *maxint* =*3* (default) the results are written for the middle and the two outermost integration points, available as middle, lower and upper surface



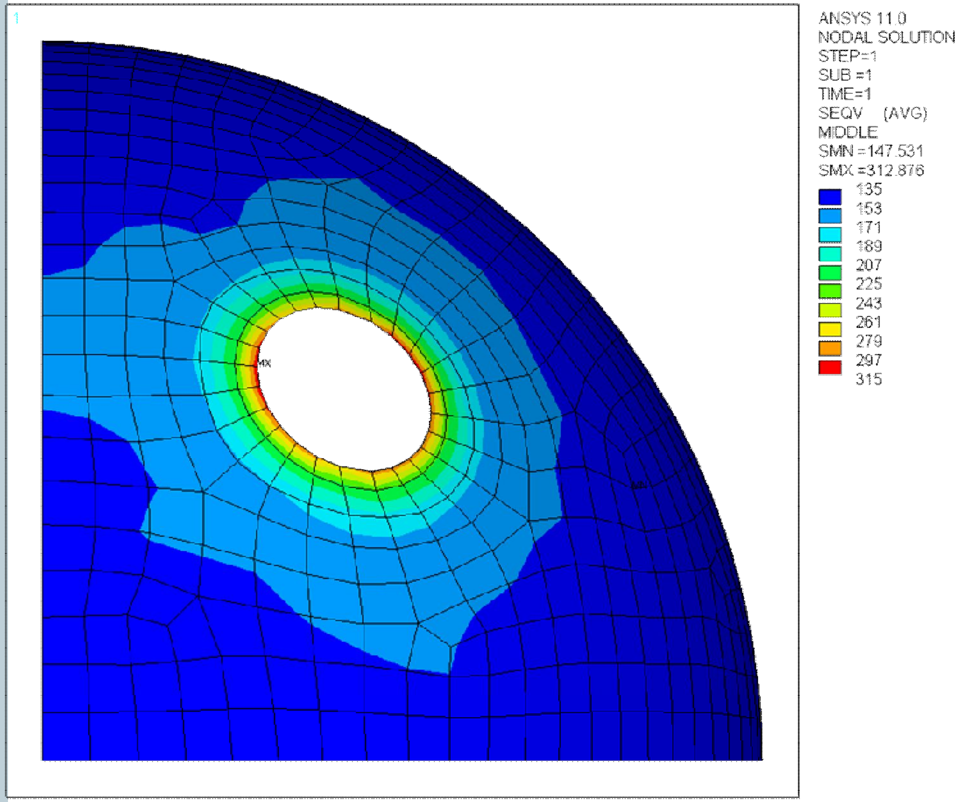
NIP=5 ; MAXINT=3 NIP=5 ; MAXINT=5

Note: LS-DYNA does not extrapolate stresses from integration points to nodes and write them to the binary database d3plot.
Extrapolated stresses can be written to the ASCII file ELOUTDET for selected elements, controlled by *DATABASE_EXTENT_BINARY; INTOUT and NODOUT

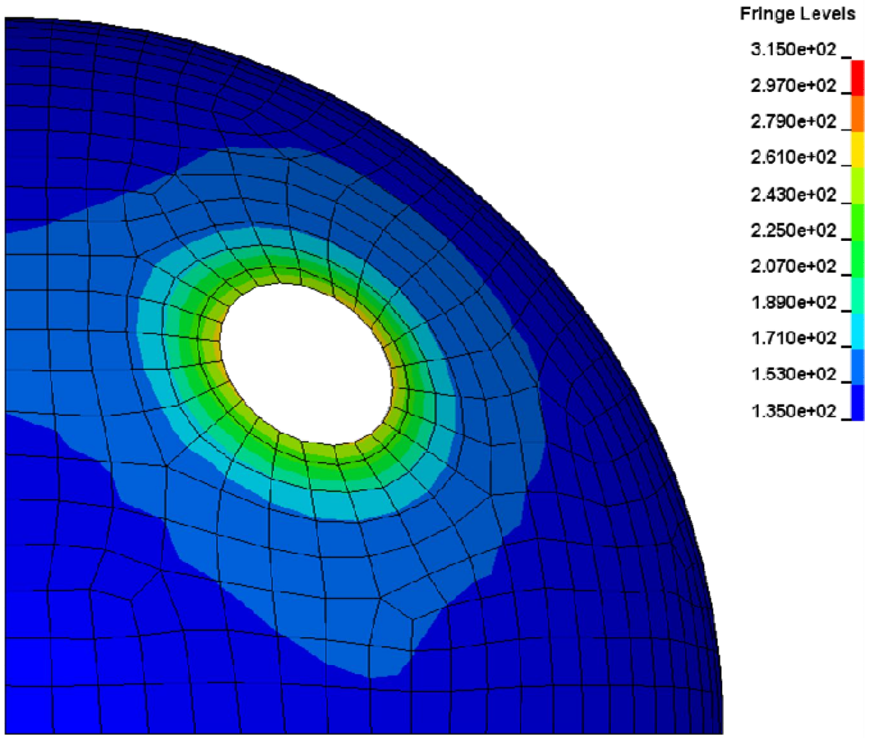
Stress Output

As default for shell elements LS-DYNA writes out only one inplane integration point to the d3plot database. For fully integrated shells an averaged value is written. Because of this stress gradients can not exactly be displayed in the LS-PREPOST.

ANSYS/POST1



LS-PREPOST

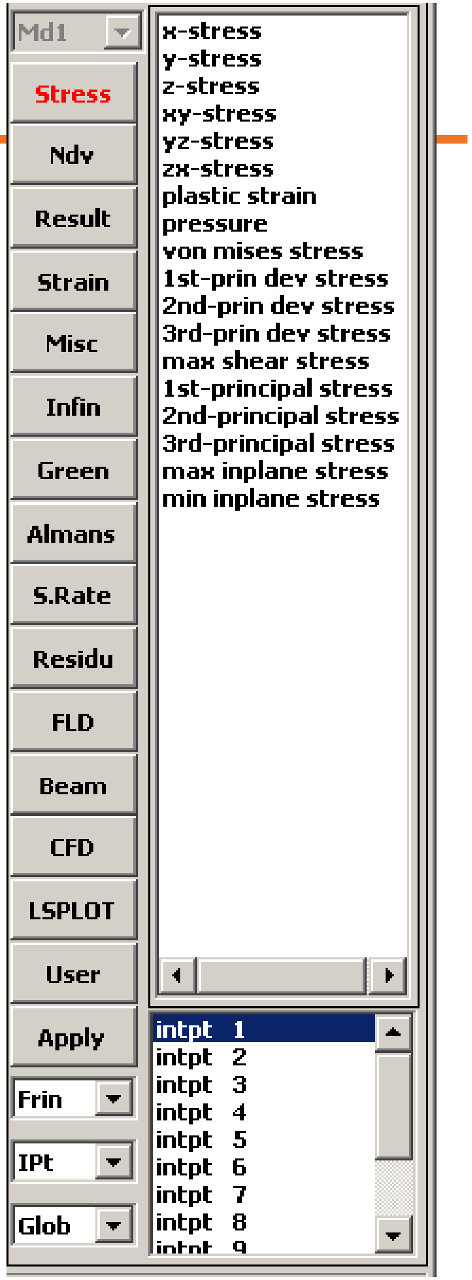


Stress Output

Since LS971_R3.2.1 four inplane integration points for shells can be written to the d3plot database by setting *DATABASE_EXTENT_BINARY, MAXINT as a negative number. The absolute value of MAXINT describes the number of through thickness integration points.

For a fully integrated shell element with five through thickness integration points and MAXINT=-3, 12 integration points are written. Current version 2.4 and 3.0 of LS-PREPOST is now able to use all four inplane integration points for averaged nodal stresses.
But without averaging again only a constant value is displayed for each element.

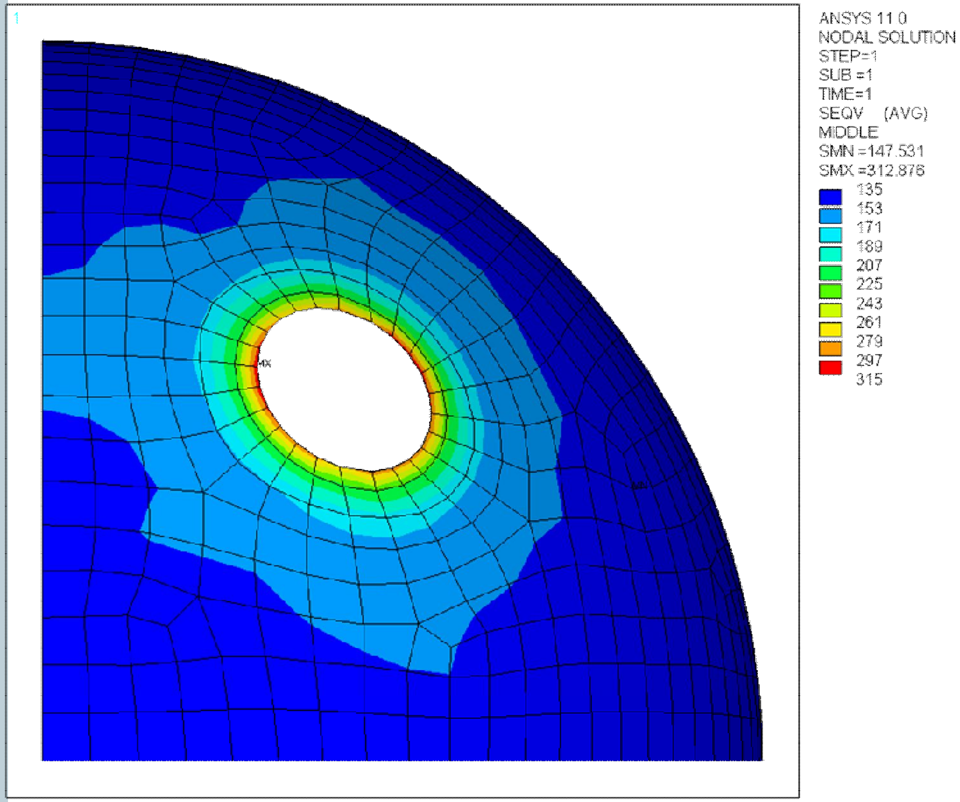
Third party Postprocessors may show 12 thickness integration points in this case if they are not updated – like LSPP 2.3 …



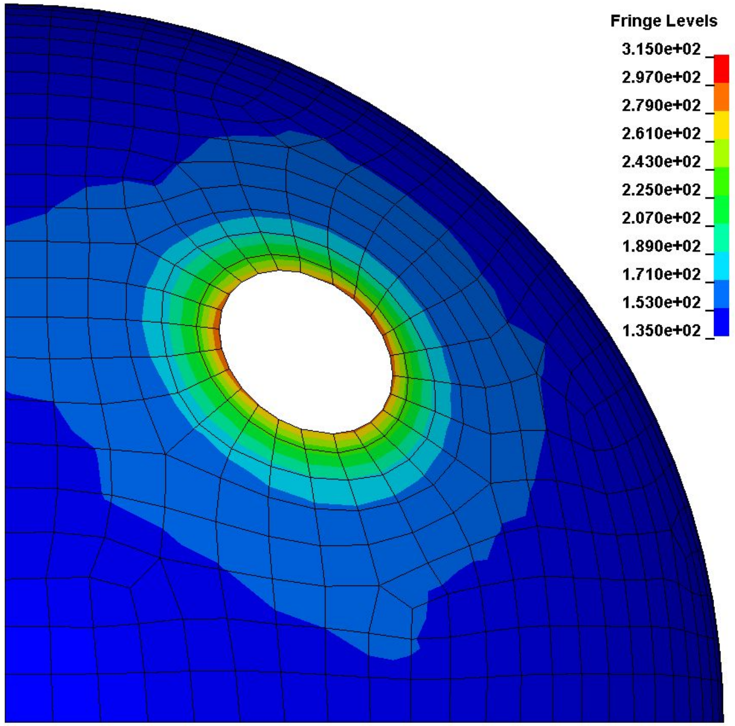
Stress Output

Now stress gradients and stress spots can be visualized more accurately in the LS-PREPOST.

ANSYS/POST1



LS-PREPOST



*ELEMENT_SHELL

*SECTION_SHELL: ELFORM

EQ.1:	**Hughes-Liu**	**à expensive**
EQ.2:	**Belytschko-Tsay**	**à default, recommended**
EQ.3:	BCIZ triangular shell	
EQ.4:	co-rotational C0, triangular shell	**à indirect use**
EQ.5:	**Belytschko-Tsay membrane**	**à FABRIC only**
EQ.6:	**S/R Hughes-Liu**	} Very expensive
EQ.7:	**S/R co-rotational Hughes-Liu**	
EQ.8:	Belytschko-Leviathan shell	
EQ.9:	**fully integrated Belytschko-Tsay membrane**	**à FABRIC only**
EQ.10:	Belytschko-Wong-Chiang	
EQ.11:	**fast (co-rotational) Hughes-Liu**	**à expensive**
EQ.12:	plane stress (x-y plane)	} Only 2D
EQ.13:	plane strain (x-y plane)	
EQ.14:	axisymmetric solid (y-axis of symmetry) – area weighted	
EQ.15:	axisymmetric solid (y-axis of symmetry) – volume weighted	
EQ.16:	**fully integrated shell element with EAS-formulation**	**à recommended**
EQ.17:	fully integrated DKT, triangular shell element	**à indirect use**
EQ.18:	fully integrated linear DK quadrilateral/triangular shell	} Only for linear implicit!
EQ.20:	fully integrated linear assumed strain C0 shell	
EQ.21:	fully integrated linear assumed strain C0 shell	
EQ.25:	Belytschko-Tsay shell with thickness stretch	} Under development
EQ.26:	fully interated shell (EAS formulation) with thickness stretch	
EQ.27:	C0 triangular shell with thickness stretch	

*ELEMENT_SHELL

Belytschko-Tsay-Shell (Type 2)

- standard element with one point integration
- very fast
- problems in case of warping and large shear deformation
- very efficient: moderate accuracy (often sufficient) in combination with high speed
- Quality can be improved by:
  Belytschko-Wong-Chiang warping stiffness: *CONTROL_SHELL, bwc=1, proj=1
  (ca. 20% more CPU time)

à Recommended shell element if speed is desired

Fully integrated shell (Type 16)

- fully integrated element with EAS-formulation and without Hourglass modes
- very fast for a fully integrated element (2.5 times more expensive than type 2)
- new standard element of Belytschko-Tsay group for increased accuracy
- Bathe/Dvorkin method for improvement of transversal shear
- behaviour of warped elements can be improved by
  *HOURGLASS, IHQ=8 (15% speed penalty)

à Recommended shell element if accuracy is desired

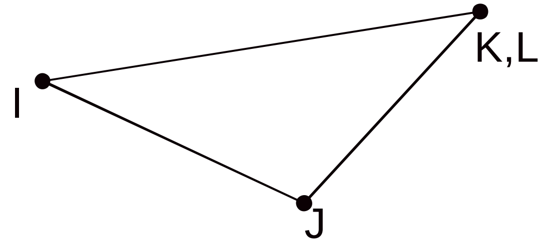
*ELEMENT_SHELL

C0 Triangular shell (Type 4)

- one point integrated special triangular element, no hourglass forms
- by setting *CONTROL_SHELL, esort=1, all triads use this formulation automatically
- only a small number of triads recommended in a quad dominated mesh (too stiff)
- 0.85 times less expensive than type 2, but two times more elements

DKT Triangular Shell (Typ 17)

- fully integrated Discrete Kirchhoff Triangular-Element
- sometimes better than triads type 4, particularly in bending, but needs twice calculation time
- by setting *CONTROL_SHELL, esort=2, all triads use this formulation automatically
- may become unstable for thick elements à not generally recommended



*ELEMENT_SHELL

Belytschko-Wong-Chiang (Type 10)

- slightly slower than type 2 (1.2 times more expensive than type 2)
- nearly identical to type 2 if *CONTROL_SHELL, bwc=1, proj=1 is set

Belytschko-Leviathan (Type 8)

- calculation time and accuracy comparable to type 10 (1.4 times more expensive than type2)
- physical Hourglass control, i.e. no input of Hourglass parameters needed
- for linear material it should be as accurate as an fully integrated element

Hughes-Liu-Shell (Type 1)

- first shell element in LSDYNA (DYNA3D)
- developed from continuum model, one point integration
- high accuracy (also in case of twisted elements)
- highly expensive (2.5 times more expensive than type 2)

selective reduced Hughes-Liu-Shell (Type 6,7)

- most costly shell element (10–20 times more expensive than type 2) **à** never recommended
- only shear part with reduced integration, otherwise 4 integration points in plane
à thus only one Hourglass mode
- use of *CONTROL_SHELL; CSTYP=2 (unique normal orientation) is approved

*ELEMENT_SHELL

Belytschko-Tsay- Membrane (Type 5)

- membrane element without bending stiffness
- only 1 integration point throughout the thickness
- one integration point in the element plane (Hourglass modes possible)
- Use only with *MAT_FABRIC **à** not generally recomended

Fully integrated Belytschko-Tsay- Membrane (Type 9)

- same as Type 5, but 4 integration points in the element plane (no Hourglass modes)
- Use only with *MAT_FABRIC **à** not generally recomended

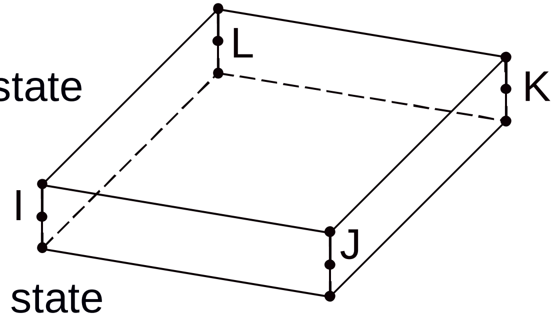
Recommendation for general membrane element: Typ 2 or Typ 16 with NIP=1 !

*ELEMENT_SHELL

Newest development: thin shell elements with thickness stress

Classical shell elements have the assumption that the stress in thickness direction is zero. The new element formulations 25-27 consider a fully three dimensional stress state, like in solid elements. Therefore 3d material models are necessary for these elements.
To apply appropriate forces – currently by contact only - these elements have 8 additional nodes: four nodes at the top surface and four nodes at the bottom surface. These nodes are generated automatically and are unvisible.

Belytschko-Tsay Shell with thickness strech (Typ 25)



- extended formulation of type 2 shell with 3 dimensional stress state
- one inplane integration point

Fully integrated Shell with thickness strech (Typ 26)

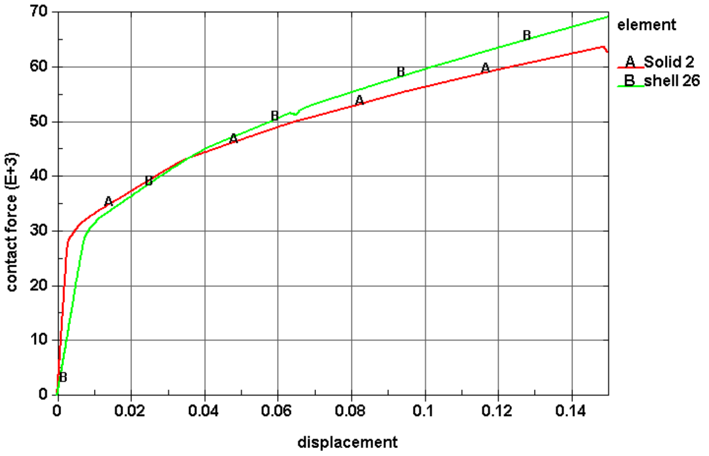
- extended formulation of type 16 shell with 3 dimensional stress state
- four inplane integration points, no hourglass modes

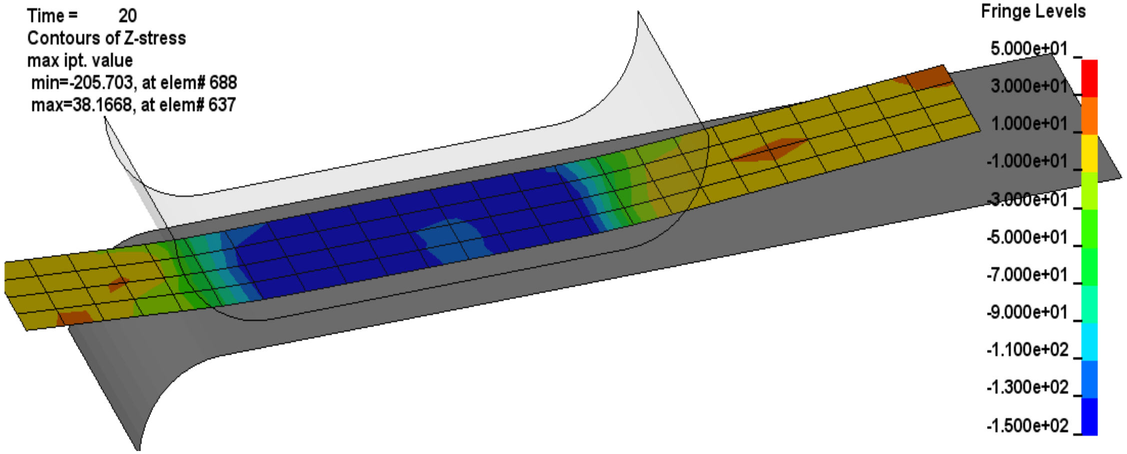
C0 Triangular Shell with thickness strech (Typ 27)

- extended formulation of type 4 triad shell with 3 dimensional stress state
- one inplane integration point

Shell elements with thickness stretch

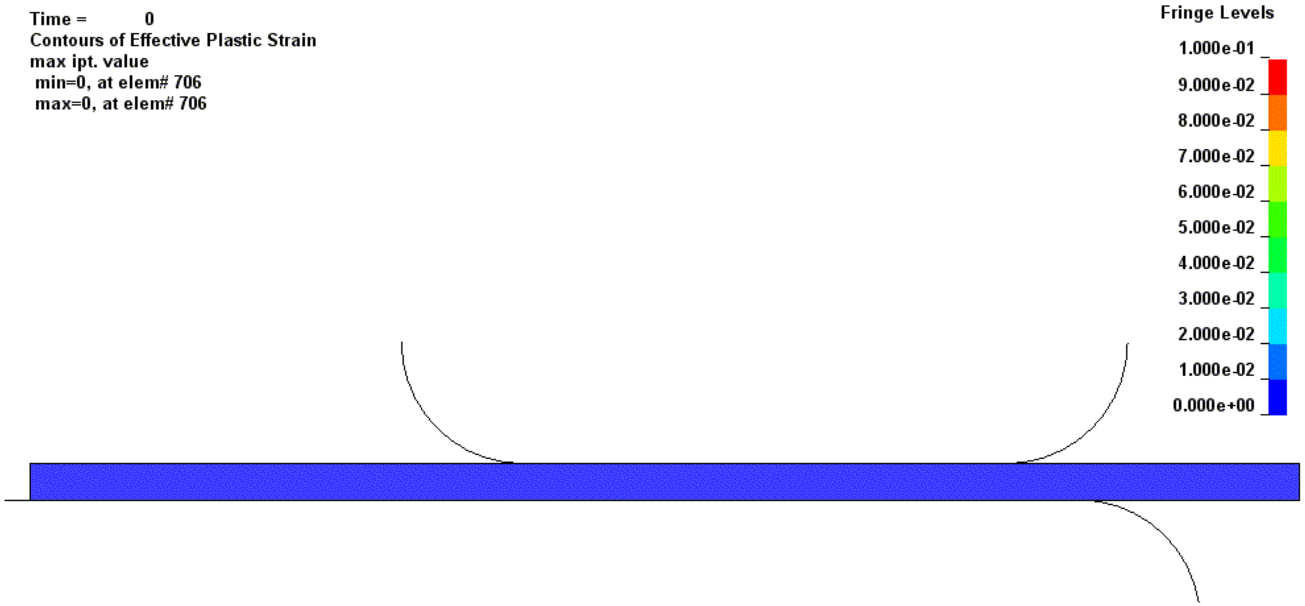
- LS-DYNA shell element types 25-27 are available starting R4 version. These elements are locking like 4 noded shells. Internally additional nodes (scalar nodes) are generated to represent the upper and lower surface. With IDOF in *SECTION_SHELL it can be decided if each element has its unique scalar nodes (IDOF=2: discontinuous thickness field) or connected by the neighbor scalar nodes (IDOF=1: continuous thickness field).
- In R5 the standard shell type 16 gets an similar option by setting IDOF=3 in *SECTION_SHELL. This implementation is much easier than elform 25-27. Only contact forces between shell lower and upper surface can cause thickness stress here.
- In opposite to solid elements: time step size is not based on thickness!



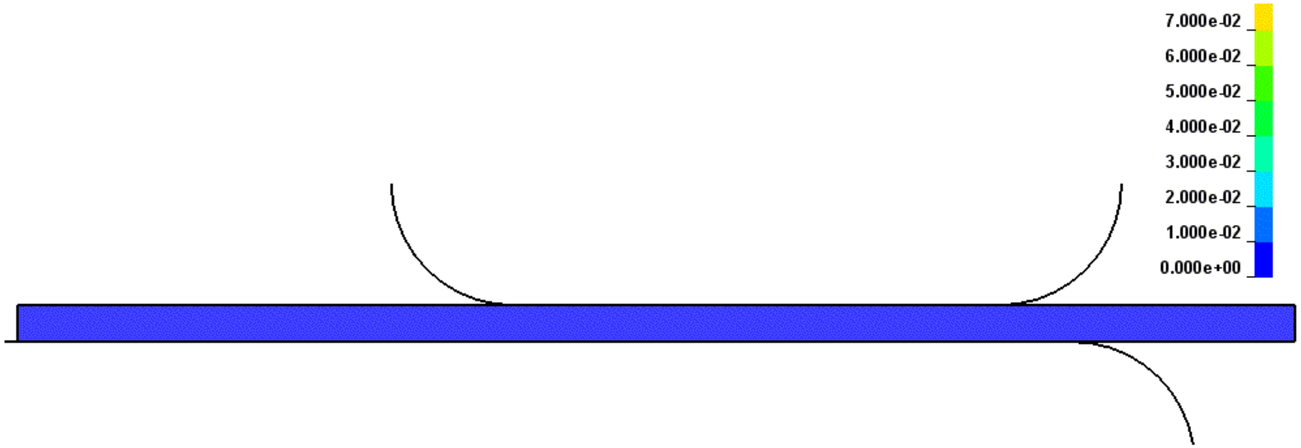


Shell elements with thickness stretch

2 Solid elements
over thickness
elform=1,
IHQ=6



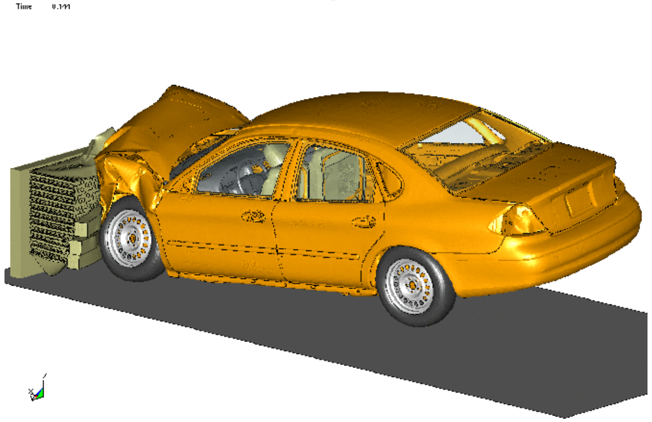
Shell element
elform=26



*ELEMENT_SHELL

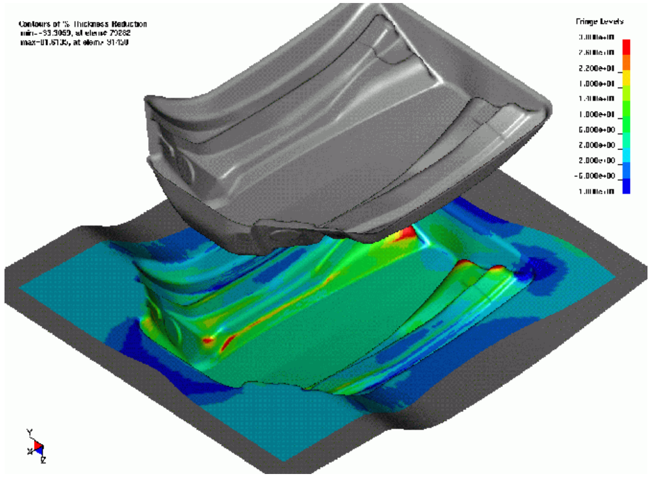
Conclusion:

For most structural applications only two element formulations are important:



Prefer speed:

-*SECTION_SHELL; ELFORM=2, NIP=3
-*HOURGLASS; IHQ=4
-*CONTROL_SHELL; ISTUPD=4, BCW=1, PROJ=1



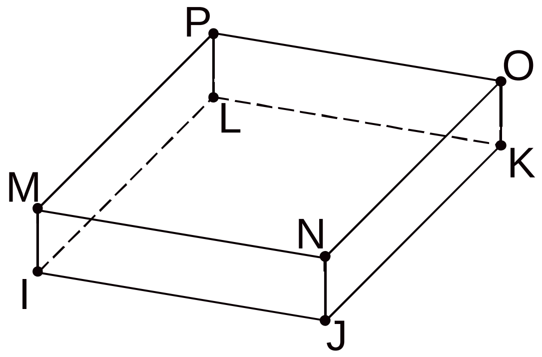
Prefer accuracy:

-*SECTION_SHELL; ELFORM=16, NIP=5
-*HOURGLASS; IHQ=8
-*CONTROL_SHELL; ISTUPD=4

Thick Shell Elements

*ELEMENT_TSHELL
*SECTION_TSHELL

*ELEMENT_TSHELL



***SECTION_TSHELL: ELFORM**

EQ.1: one point reduced integration
EQ.2: selective reduced 2 x 2 in plane integration
EQ.3: assumed strain 2 x 2 in plane integration
EQ.5: assumed strain reduced integration

8 Node Shell elements are rarely used in LS-DYNA

*ELEMENT_TSHELL

One point integrated thick shell (Type 1)

- 8 node shell with 2d stress state (like thin shell)
- The thickness of the element is constrained by a penalty function between top and bottom nodes, thickness can only change by membran strain (like thin shell)
- Hourglass modes possible, mostly much to soft à not recommended

Fully integrated thick shell (Type 2)

- 8 node shell with 2d stress state (like thin shell)
- The thickness of the element is constrained by a penalty function between top and bottom nodes, thickness can only change by membran strain (like thin shell)
- No Hourglass modes
- Accuracy comparable to thin shell but much slower: 7-8 times more than type 2 shell
- Time step size is based on all three dimensions

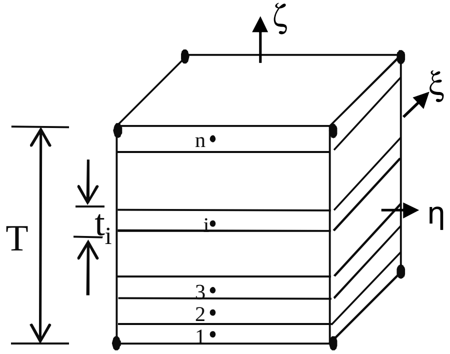
à Typically no advantages over thin shells

Assumed strain thick shell (Type 3)

- 8 node shell/solid with 3d stress state
- thickness can change by thickness stress
- extremely slow: 65 times more than type 2 shell
- not really a shell: at least two elements over thickness are necessary so solve for bending

à Not meaningful in explicit

*ELEMENT_TSHELL



Layered solid (Type 5)

- 8 node layered solid with 3d stress state
- new Feature in LS971 R5
- access to *MAT_ELASTIC and composite material models *MAT_22, *MAT_59
- shear locking and hourglass stabilization by build in assumed strain method, *HOURGLASS setting has no meaning

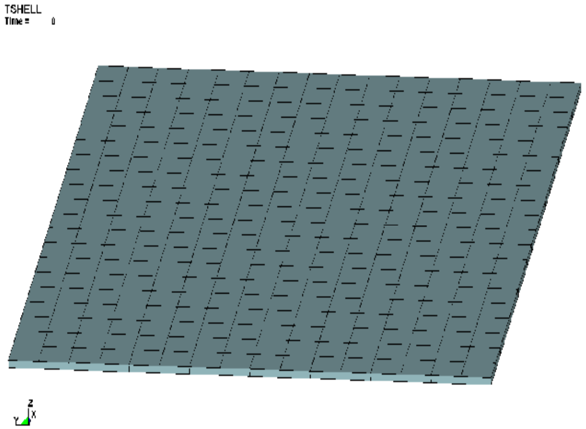
- efficient in computation: only 1.5 times more than type 2 shell

- solves bending problem with only one element over thickness, also for bad aspect ratio, of course more elements are possible
- laminated shell theory available (*CONTROL_SHELL, lamsht=4), but makes only sense if only one element over the thickness
- developed to model thick composite structures

*ELEMENT_TSHELL

Element type	**Time step**	**Number of cycles**	**Time (sec)**	**Element cost factor**	**Reaction force (N)**
Shell type 2	1.66e-3	18135	5	1	0.20
Shell type 16	1.66e-3	18133	12	2.4	0.20
Tshell type 1	3.3e-4	90760	43	1.7	2.8
Tshell type 2	3.3e-4	90705	193	7.7	0.20
Tshell type 3	2.82e-4	106540	1950	66.4	0.14
Solid 1, IHQ=6	2.82e4	106570	37	1.25	0.203
Tshell type 5 (MLS)	2.82e-4	106570	41	1.5	0.206

Rectangular plate under concentrated load



Dimension: 200*160*2
Element length: 10mm
Thickness: 2mm

Solid Elements

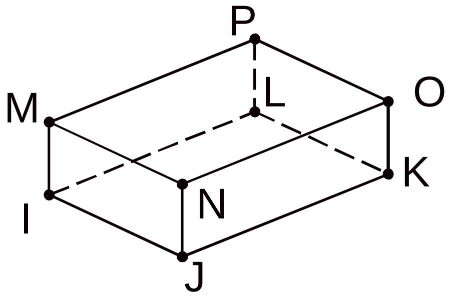
*ELEMENT_SOLID
*SECTION_SOLID

*ELEMENT_SOLID

Element shapes:

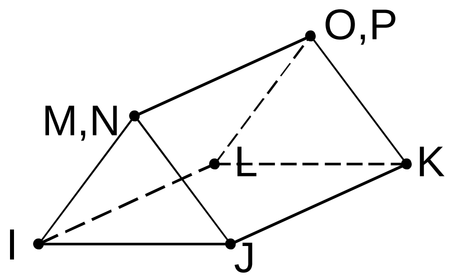
Hexahedron:

(favoured solid element)



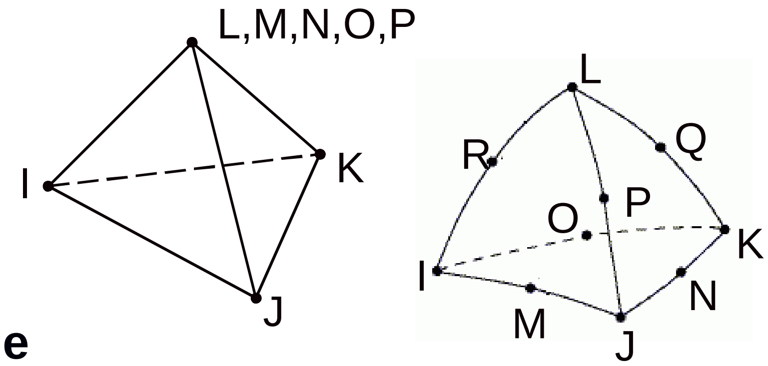
Pentahedron:

(extrusion in depth of triangles)



Tetrahedron:

- **4-noded without rotation: very stiff, only used for foams**
- **4-noded with rotation: compromise between effort and accuracy**
- **10-noded very accurate but also very costly in terms of computation time**



*ELEMENT_SOLID

*SECTION_SOLID: ELFORM

EQ.1: constant stress hexahedron element à default

EQ.2: fully integrated S/R hexahedron

EQ.-1: fully integrated S/R hexahedron without shear locking (simplified)

EQ.-2: fully integrated S/R hexahedron without shear locking

EQ.3: fully integrated quadratic 8 node hexahedron with nodal rotations

EQ.4: S/R quadratic tetrahedron element with nodal rotations

EQ.5: 1 point ALE hexahedron

EQ.6: 1 point Eulerian hexahedron

EQ.7: 1 point Eulerian ambient hexahedron

EQ.8: acoustic hexahedron

EQ.9: 1 point corotational hexahedron for *MAT_MODIFIED_HONEYCOMB

EQ.10: 1 point tetrahedron

EQ.11: 1 point ALE multi-material element, hexahedron

EQ.12: 1 point integration with single material and void, hexahedron

EQ.13: 1 point tetrahedron with nodal pressure

EQ.14: 8 point acoustic hexahedron

EQ.15: 2 point pentahedron element

EQ.16: 5 point 10 noded quadratic tetrahedron with mid side nodes

EQ.17: 10 noded composite tetrahedron with mid side nodes (new in ls971)

EQ.18: 8 point enhanced strain hexahedron element for linear statics only

EQ.19: 4 point cohesive element

EQ.20: 4 point cohesive element with offsets for use with shells

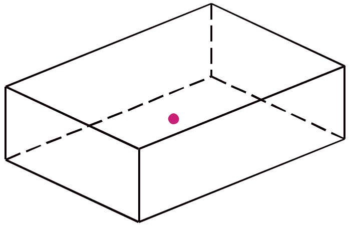
EQ.41: Mesh-free hexahedron - EFG

EQ.42: Mesh-free tetrahedron - EFG

*ELEMENT_SOLID

standard element (Type 1)

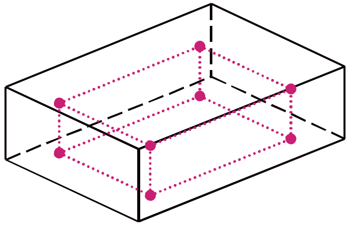
- 8-node hexahedron solid element with tri-linear shape functions
- reduced integration, i.e. stresses are calculated only in one integration point in the middle of the element
- Hourglass modes possible



1 integration point

fully integrated element (Type 2)

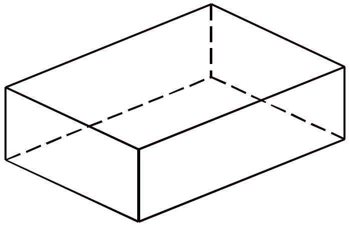
- 8-node hexahedron solid element with tri-linear shape functions
- fully integrated with 8 integration points
  à no Hourglass modes
- 2-3 times more expensive than type 1
- helpful, if Hourglass modes are a problem
- handicap: lower deformations obtained as with type 1
- handicap: shear locking if bad aspect ratio



8 integration points

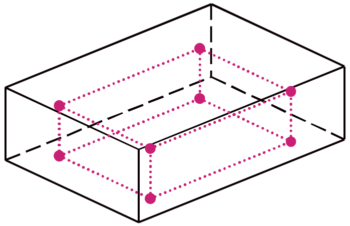
fully integrated quadratic 8 node element with nodal rotations (Type 3)

- 8-node hexahedron solid element with quadratic shape function
- 6 degrees-of-freedom per node: translations and rotations
- 14 integration points
- not useful for plasticity or material with Poisson ratio close to 0.5
- very expensive in cpu time (3 times more expensive than type 2)
- **à** not generally recomended



14 integration points

*ELEMENT_SOLID



8 integration points

fully integrated S/R hexahedron without shear locking (Typ -2)

- extension of type 2 hexahedron: accurate formulation
- selectively reduced integration, 8 integration points
  à no Hourglass forms
- 5 times more expensive than type 2
- Advantage over type 2: no shear locking, also in bad aspect ratios

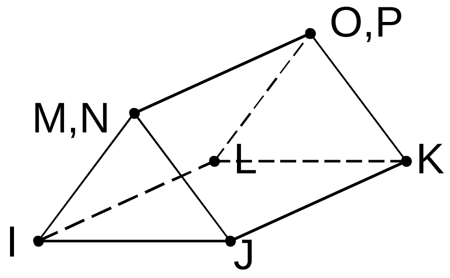
fully integrated S/R hexahedron without shear locking (Typ -1)

- extension of type 2 hexahedron: simplified, fast formulation
- selectively reduced integration, 8 integration points
  à may have some remaining hourglass forms
- 1.2 times more expensive than type 2
- Advantage over type 2: no shear locking, also in bad aspect ratios
- Advantage over type -2: much less expensive and in most cases sufficient

*ELEMENT_SOLID

Pentahedron element (Type 15)

- 6-noded element with trlinear displacement ansatz and 2 integration points
- element typically is generate when triangular surface element is extruded into the depth
- with the input:
  *CONTROL_SOLID, esort=1
  all 6-noded solid elements get automatically this
  element formulation



Solid element - (type 8 and type 14)

- for acoustic simulation (sound distribution within fluids)
- nodes only have a pressure degree of freedom
- One point (type 8) and fully integrated (type 14)

1 point corotational for *MAT_MODIFIED_HONEYCOMB (Type 9):

- special hexahedron element for extra large deformations in combination with material law 126 only (*MAT_MODIFIED_HONEYCOMB)
- keeps stable in large element distorsion, prevent “negative volume”, but limited accuracy

*ELEMENT_SOLID

Tetrahedron element (Type 10)

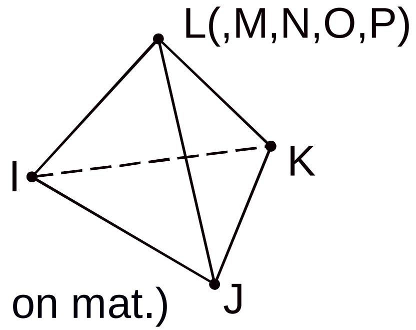
- 4-nodes tetrahedron element with tri-linear shape functions and 1 integration point
- in general much too stiff
  à often used in combination with foam material, where compression dominates

Tetrahedron element (Type 13)

- like tetrahedron element type 10, but with additional pressure degree of freedom at nodes
- 10-40% more expensive than type 10 – based on material law
- nearly no volumetric locking
  à recommended for incompressible material like rubber or plastic flow in bulk metal forming
- Shear locking is not much better than tet type 10
- only available for a few material models, for all other materials it switch back type 10

S/R quadratic tetrahedron element with nodal rotations (Type 4)

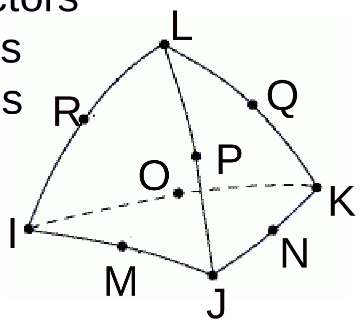
- 4-node tetrahedron solid element with quadratic shape functions
- 6 degrees-of-freedom per node: translations and rotations
- 5 integration points
- expensive in cpu time (1.5-5 times more expensive than type 10 – based on mat.)
- accuracy better than tetrahedron type 10, but less than hexahedron type 2
- sometimes double precision is needed if element rotates



*ELEMENT_SOLID

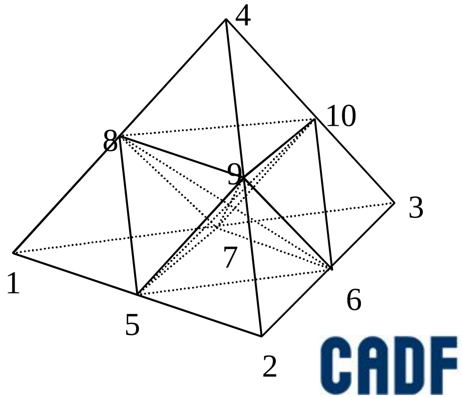
10-noded tetrahedron element (Type 16)

- tetrahedron element with midside nodes and quadratic shape function
- 4+1 integration points
- needs approx. the same computation time as type 4 but time step size is halfed
- because of shape function the midside nodes have different weighting factors than the corner nodes **à** midside nodes should not be used to apply loads
- midside nodes must stay “near” the straight line between the corner nodes
  à not applicable for large deformation



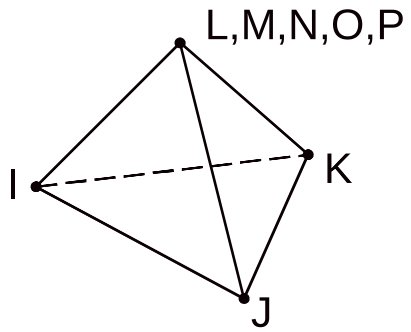
10-noded composite tetrahedron element (Type 17)

- tetrahedron element with midside nodes and quadratic shape function
- the word “composite” means, that one element internally consists of a lot of smaller 4 noded elements
- because of this, also midside nodes have the same weighting factors than the corner nodes
- 4+1 integration points
- same time step size as type 16 but much more cpu time
- still under development, not always stable



TET13

***SECTION_SOLID, elform=13**



4 noded, 1 point tetrahedron with linear shape function and additional pressure degree of freedom.

- Less volumetric locking, therefore recommended for applications where incompressible material behavior is important:

hyper-elastic material (Rubber)
isochoric plastic deformation (Forging)

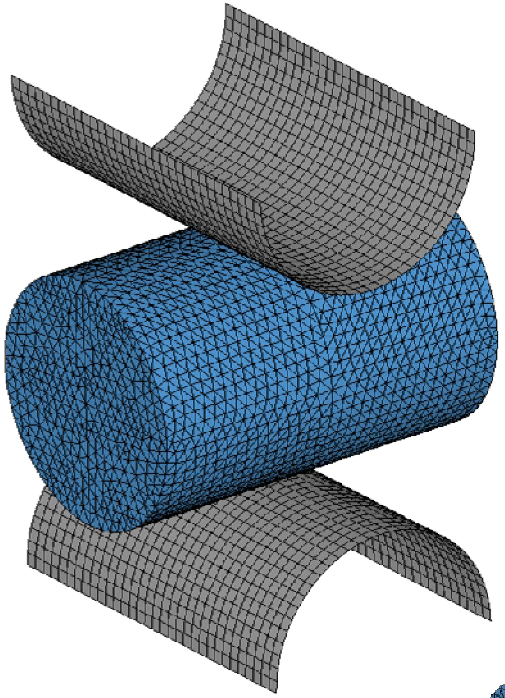
- Only less improvement in shear locking in comparison to standard linear tetrahedron elform=10

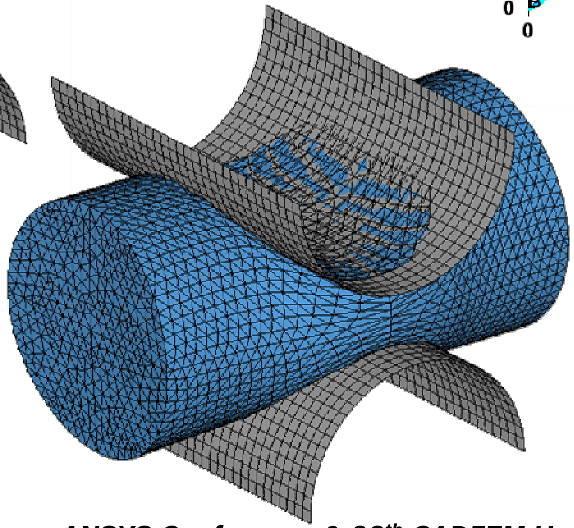
Only available for material models:
1,3,6,24,27,77,81,82,91,92,106,120,123,124,128,129,181,183,225,244

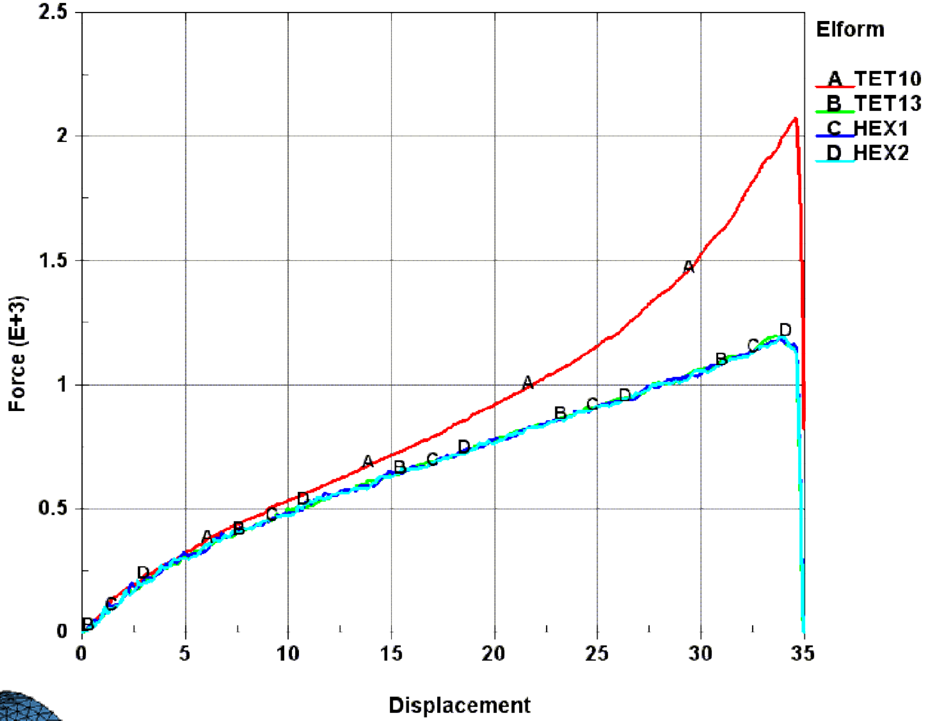
For all other material models this element is identical to elform=10

TET13

Example: Bulk metal forming



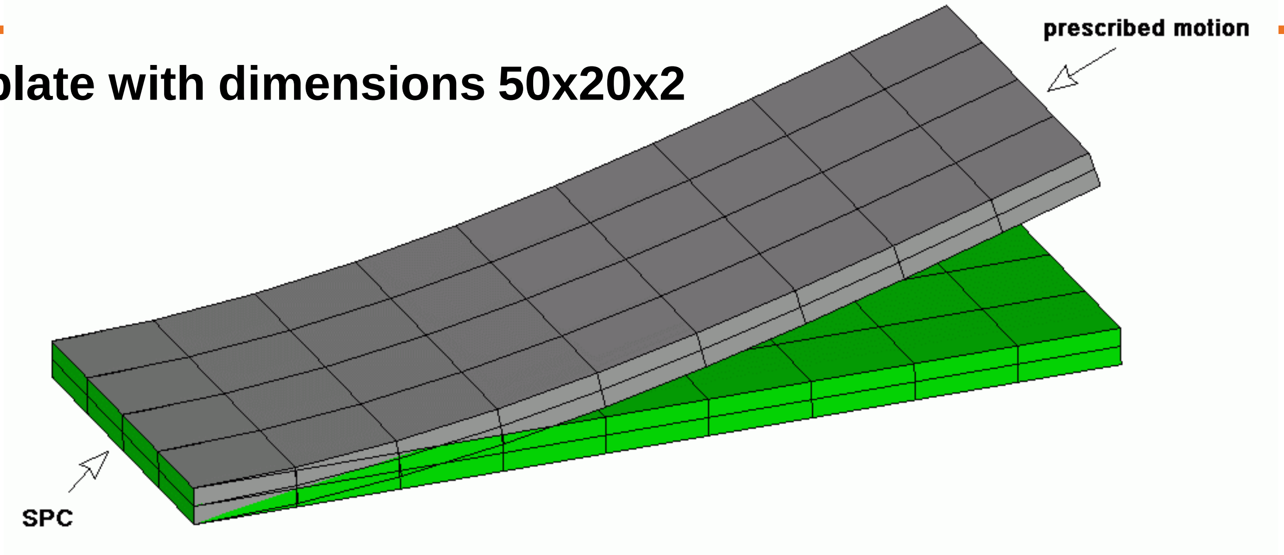


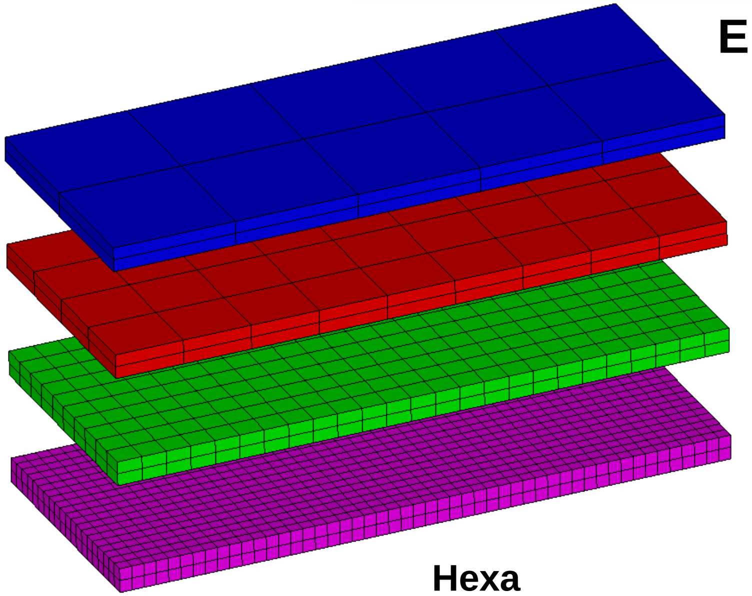


Elform	TET10	TET13	HEX1	HEX2
CPU time	46s	65s	13s	36s

Mesh for thin walled structures

Test model: plate with dimensions 50x20x2





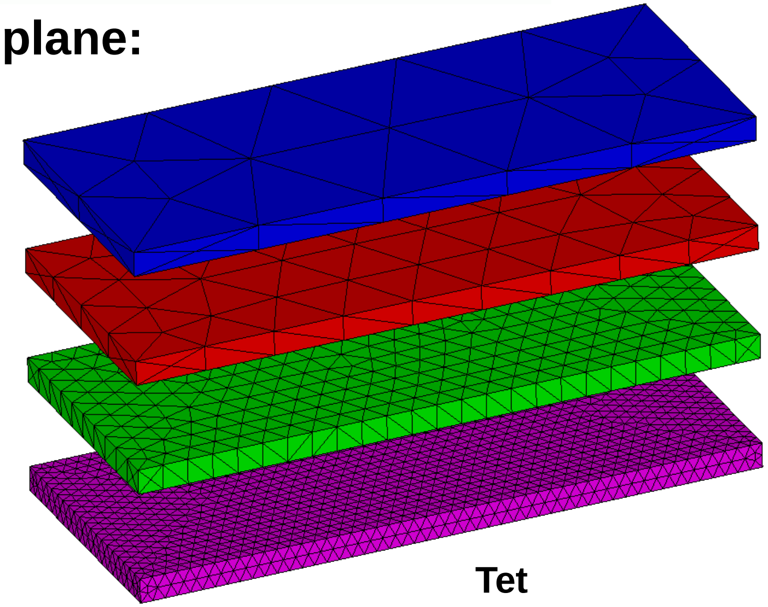
Element size in plane:

10 mm

5 mm

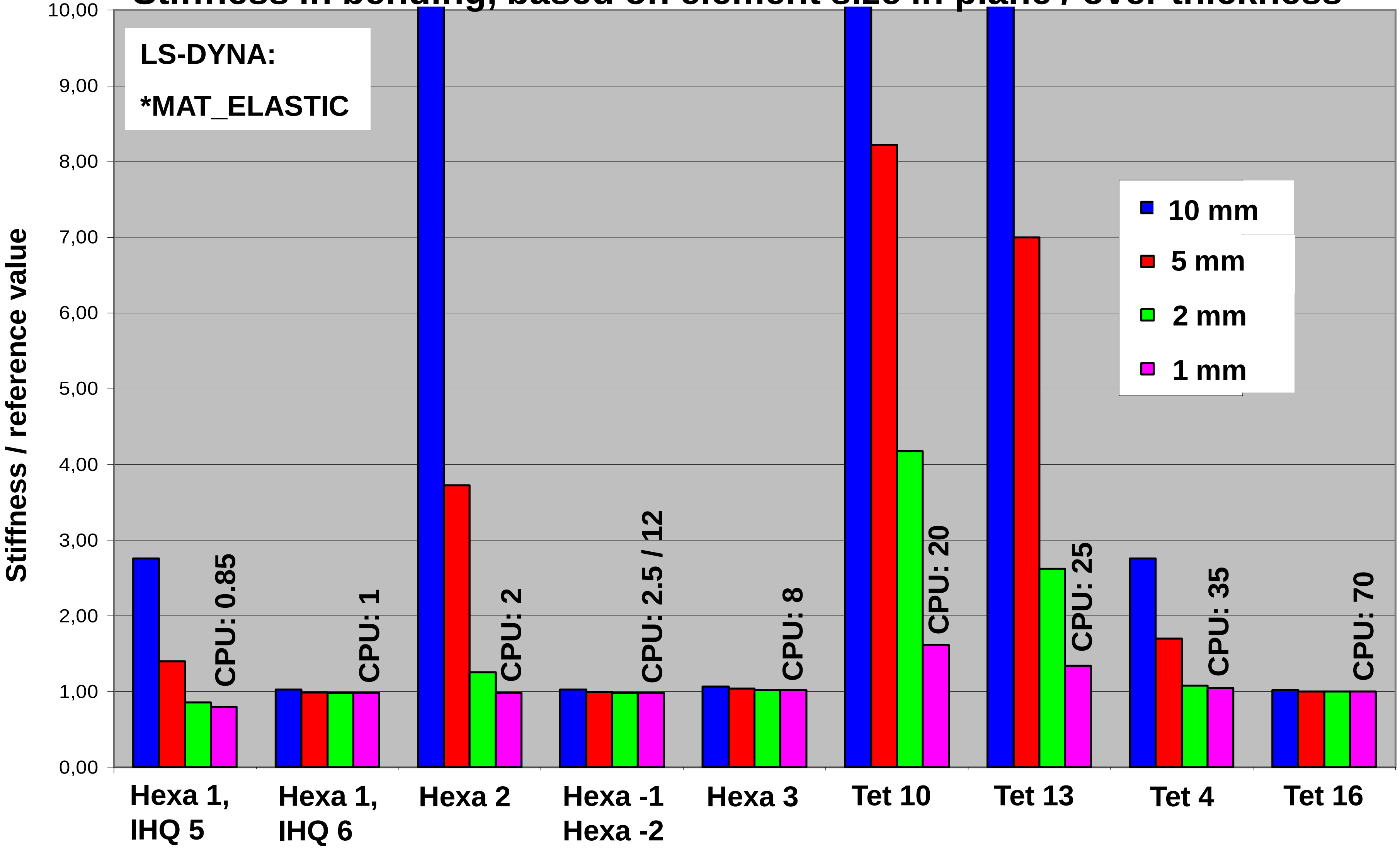
2 mm

1 mm



Mesh for thin walled structures

Stiffness in bending, based on element size in plane / over thickness



Mesh for thin walled structures

Conclusions:

- Solid type 2 is very sensitive in aspect ratio for bending problems - shear locking
- Solid type 1 in combination with Hourglass control type 6 (IHQ=6, QM=1) solves linear bending problem very well but is to stiff for plasticity with this setting.
- Solid types -1 and -2 do not show shear locking effects. Type -1 is approx. 30% more expensive than type 2 and type -2 is 5 times more expensive than type 2.
- Linear tetrahedron type 10 is much to stiff in bending. Type 13 is a little bit better, but only available for a limited number of material models. Type 13 is favored for large plastic deformation and rubber material.
- The 10 noded quadratic tetrahedron type 16 gives very good bending response also if aspect ratio is poor. Less elements are necessary. But this element type is more sensitive because of its midside nodes and may have problems with large distortions.
- The older 4 noded tetrahedron type 4 (with nodal rotations) gives only acceptable results if aspect ratio is good. This element type may be recommended only if mesh density is always fine, e.q. to allow large local deformation. This element was improved in R4 regarding large strains and large rotation.

Mesh for thin walled structures

Shells or Solids ?

1) if thickness stress is important, then Solids

2) if transversal shear is important, then Solids

3) if only membrane stiffness is important, then Shells

4) for bending: if bending radius R greater than three times thickness
R>3d: then Shells

if bending radius smaller than thickness
R<d: then 5 Solids over thickness

A new alternative are the new shell elements with thickness stretch:

*SECTION_SHELL, elform=25-27

A complete 3d stress state is used in these elements, so thickness stress is considered in the material law. But for bending the cross section always remain rectangular.

Mesh for thin walled structures

Should I use tetrahedrons ?

Advantage:	Disadvantage:
- Tetrahedron meshing is easy and fast.	- Run time is large.

If the same mesh density is desired, 6 tetrahedron elements are needed instead of one hexahedron element. **à** 6 times more effort

One tetrahedron element (type 4 or 16) needs 1.5-5 times more cpu time than one hexahedron (type 1). **à** 9 - 30 times more effort

The 10 noded tetrahedron type 16 needs half the time step size because of mid side nodes **à** 18 - 60 times more effort

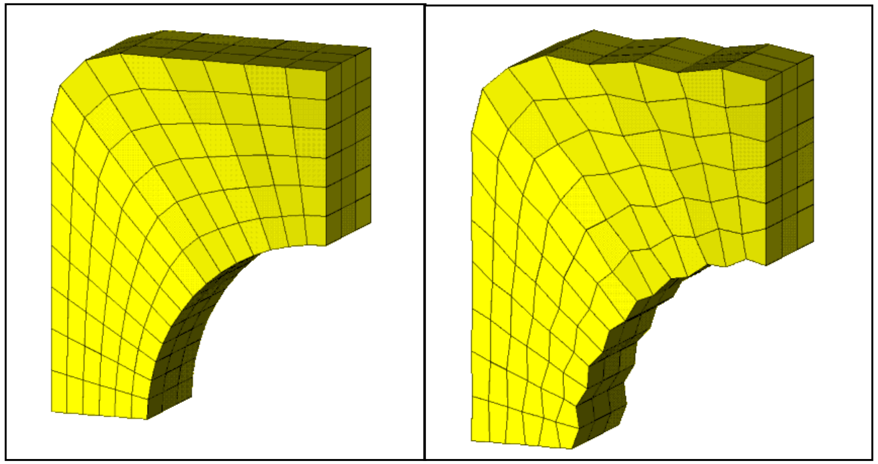
Hourglass control

For hexahedron solid elements Hourglass control is important.
Fully integrated solid elements (*SECTION_SOLID, elform=2) have some disadvantages:

- higher computation time (approc. factor 2.5)
- shear locking if aspect ratio is bad
- may fail if element distortion is to large (negative jacobian)

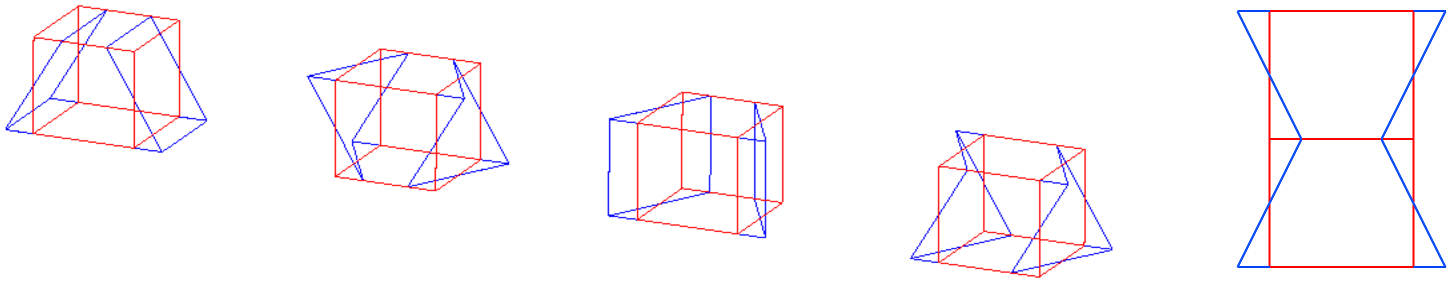
One point integrated element have only two disadvantages:

- only constant stress over element
- zero energy deformation possible (Hourglass modes)



Hourglass control

- Hourglassing is a state of strain, which is free of energy (ZEM: Zero Energy Mode) and can emerge in case of one-point-integrated solid- (hexahedrons) and shell elements



- Hourglass modes are mostly caused by:
  - concentrated loads
  - contact (contact force at several nodes)

- in LS-DYNA there are 2 possibilities to prevent Hourglassing:

à using the automatic stabilisation against this deformation with
- *HOURGLASS (input for each part) or
- *CONTROL_HOURGLASS (global control)

à using a fully integrated element type

disadvantages: - more computation time
- more sensitive with respect to large element deformations

Hourglass control

Recommendation for *HOURGLASS and. *CONTROL_HOURGLASS

for shell elements	**ihq=4**	(stiffness form, default settings)
for solid elements (in general)	**ihq=5**	(stiffness form, default settings)
for solid elements (foam)	**ihq=3**	(viscous form, default settings)
for solid elements (honeycomb)	**ihq=3**	(viscous form, default settings)
for solid elements (elastic)	**ihq=6**,**qm=1.0**	(stiffness form)
for solid elements (plastic)	**ihq=6**,**qm=0.01-0.001**	(stiffness form)
for solid elements (rubber)	**ihq=7**,**qm=1**	(stiffness form)
for solid elements (viscoelast.)	**ihq=7**,**qm=1**	(stiffness form)

Note: ihq=6 and 7 is a special solid element formulation according to Belytschko-Bindemann

Danger: Default **ihq=1** it not orthogonal to rigid body rotation
à do not use, except for Eulerian elements !

Pentahedrons and Tetrahedrons are not influenced by *HOURGLASS.

Hourglass control

For elastic plastic material models the Belytschko-Bindeman assumed strain formulation (*HOURGLASS, IHQ=6) is successfully used.
Unfortunately the Hourglass stiffness is controlled by the elastic material properties only. This tends to that Hourglass control is very accurate for pure elastic deformation but may be much to stiff if material becomes plastic. The user should scale down the Hourglass coefficient manually. A factor of QM=0.1 or QM=0.01 is common.

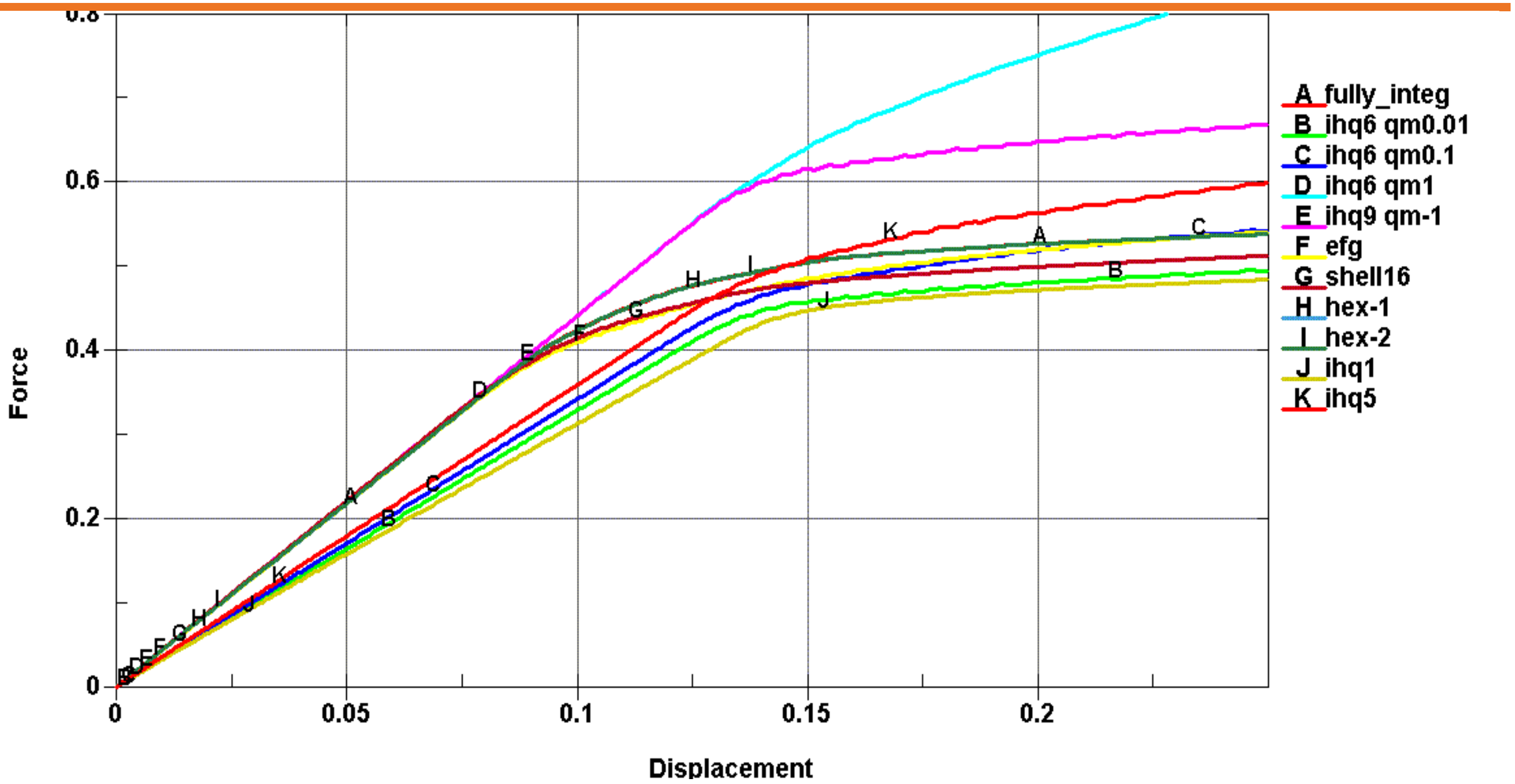
With LS971 the Puso enhanced assumed strain formulation is also possible (*HOURGLASS, IHQ=9). The behaviour is comparable to the latter one, with one exception: For IHQ=9 the Hourglass coefficient QM can be defined as a negative number. Then the absolute value scales the Hourglass stiffness based on the current material properties. If deformation is purely elastic, the youngs modulus it used, in case of plastic deformation the tangent modulus is used. This automatic correction is only available for material models:

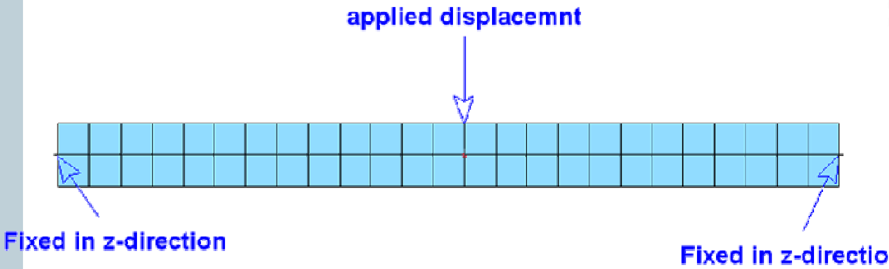
*MAT_PIECEWISE_LINEAR_PLASTICITY (24)
*MAT_PLASTIC_KINEMATIC (3)
*MAT_POWER_LAW_PLASTICITY (18)

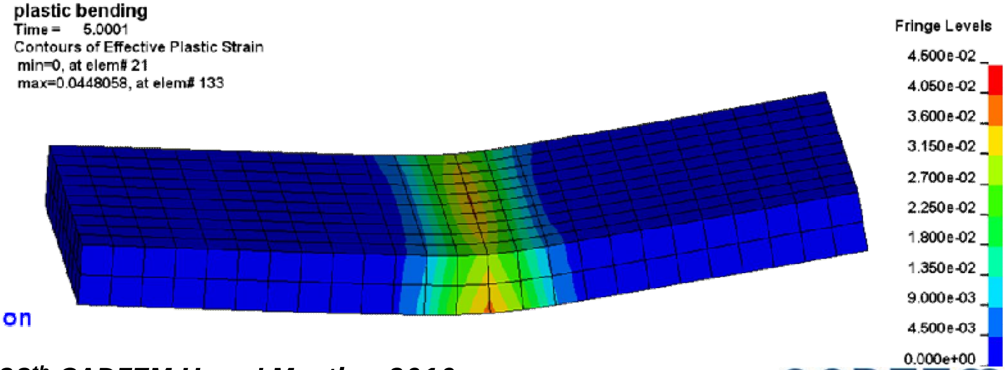
Plastic Bending – HOURGLASS control

Test:

Plastic bending with two elements over thickness



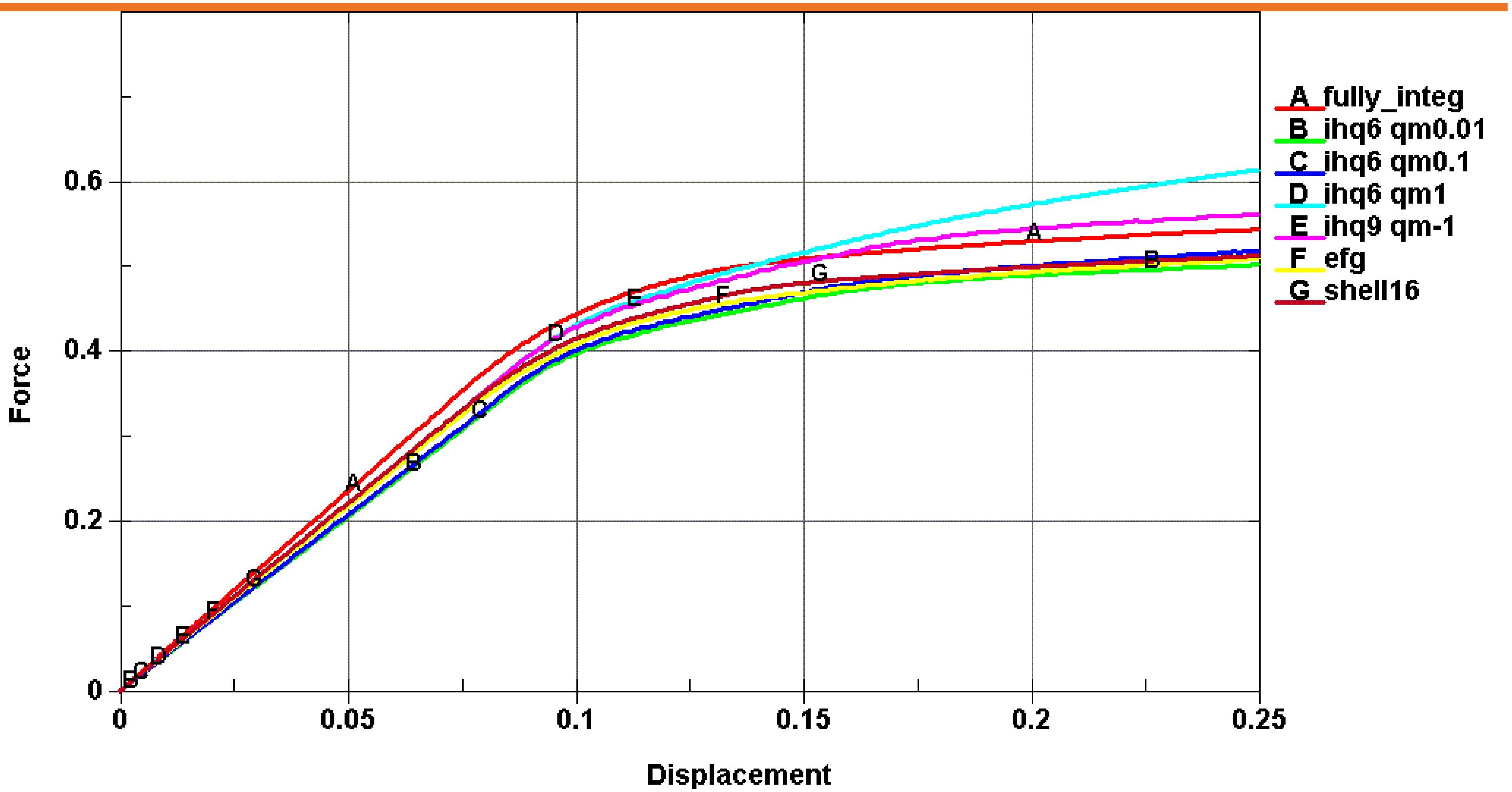


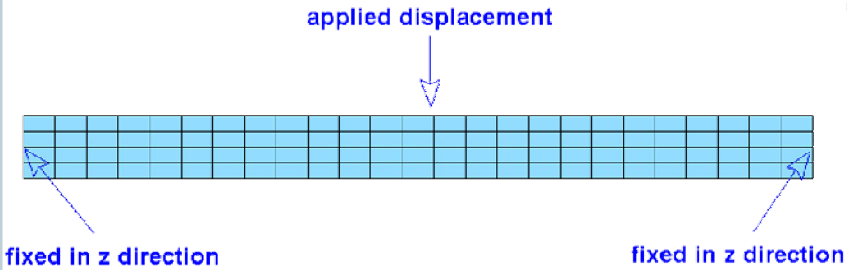


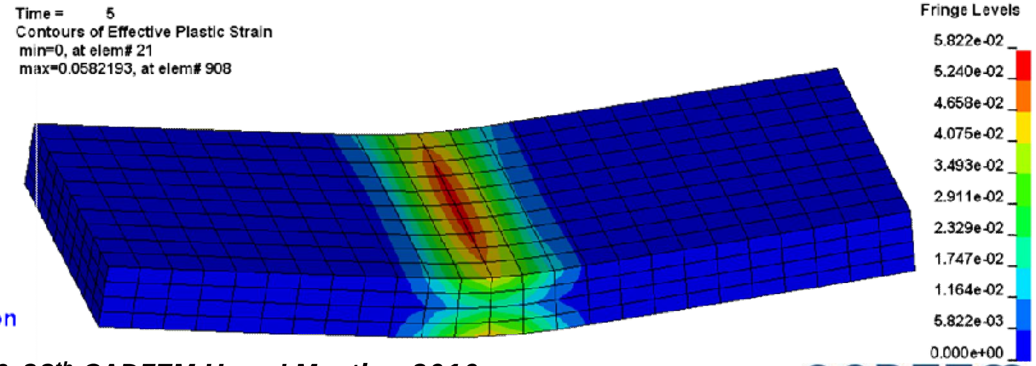
Plastic Bending – HOURGLASS control

Test:

Plastic bending with four elements over thickness







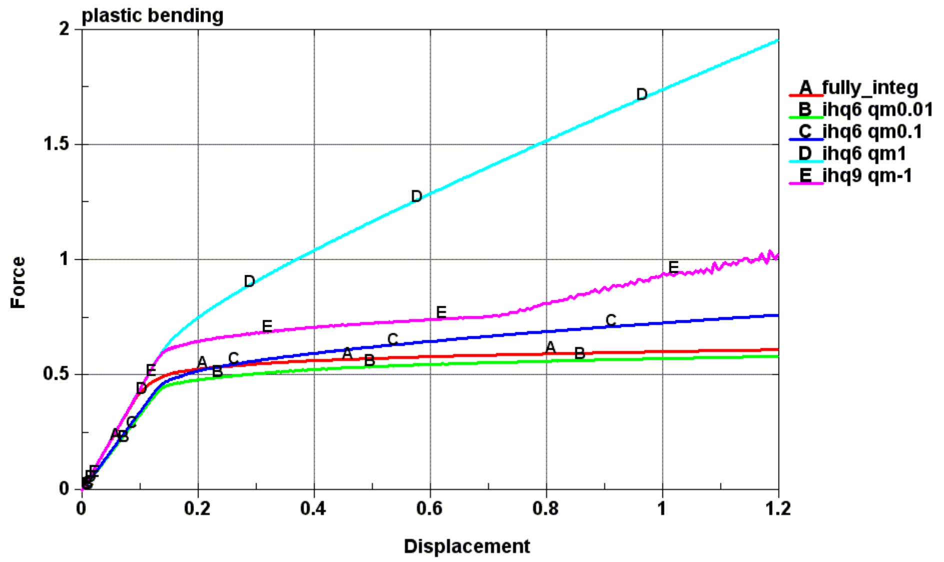
Hourglass control

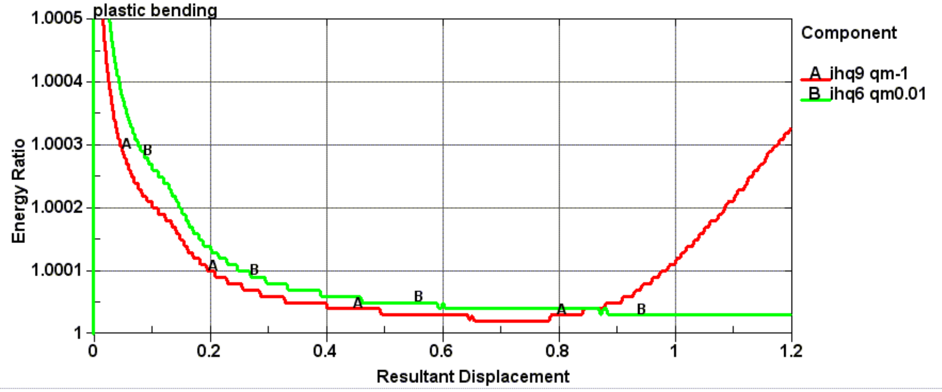
In the current program version, the Puso Hourglass control sometimes tends to become unstable if element deformation becomes to large. This can be identified by an increasing energy ratio.

May be further improvements are necessary.

$$\text{Energy ratio} = \frac{\Sigma E(t)}{\Sigma E(t=0)+W_{ext}(t)}$$

Σ E = kinetic + internal + hourglass + damping + sliding





EFG

EFG
Element Free Gallerkin

EFG

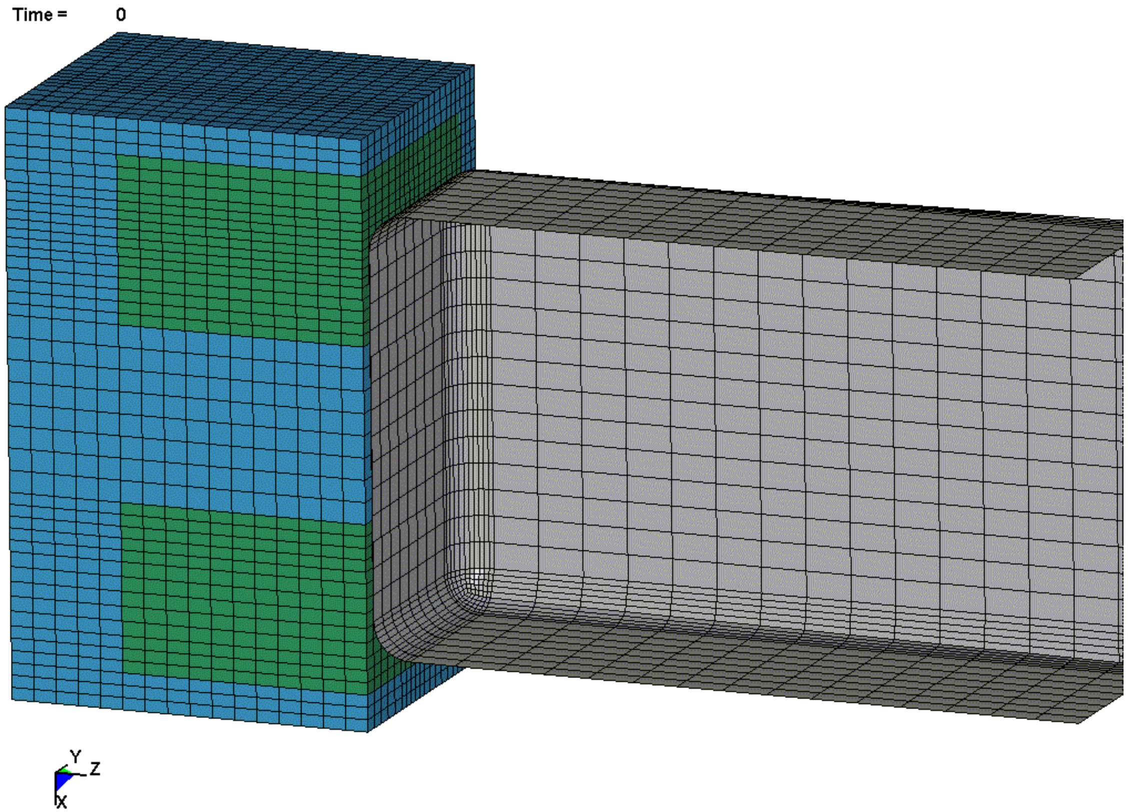
Althoug EFG is a particle method, it can be used in LS-DYNA like standard elements, because EFG works with a background mesh.

Simply use *SECTION_SOLID_EFG and set ELFORM=41 (hex) or 42 (tet)

EFG parts and FEM parts can share nodes.

EFG Background mesh is only needed in initialization phase, for contact and for postprozessing.

The main advantage of EFG over FEM is robustness and accuracy in large distorted elements. EFG has less danger of „negative volume“.

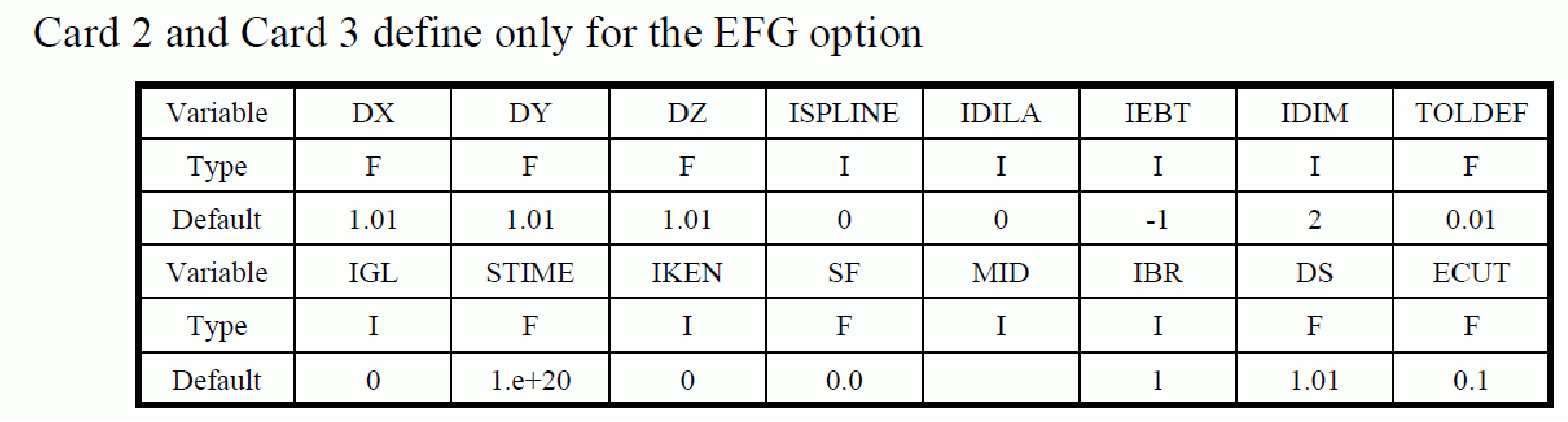


EFG

Disadvantage of EFG over FEM:

- Larger CPU time
- More memory needed
 (automatic memory allocation may fail, define memory by hand
- a lot of settings possible, results are dependent on these settings
 (e.g. DX, DY, DZ influencing CPU time and stiffness)

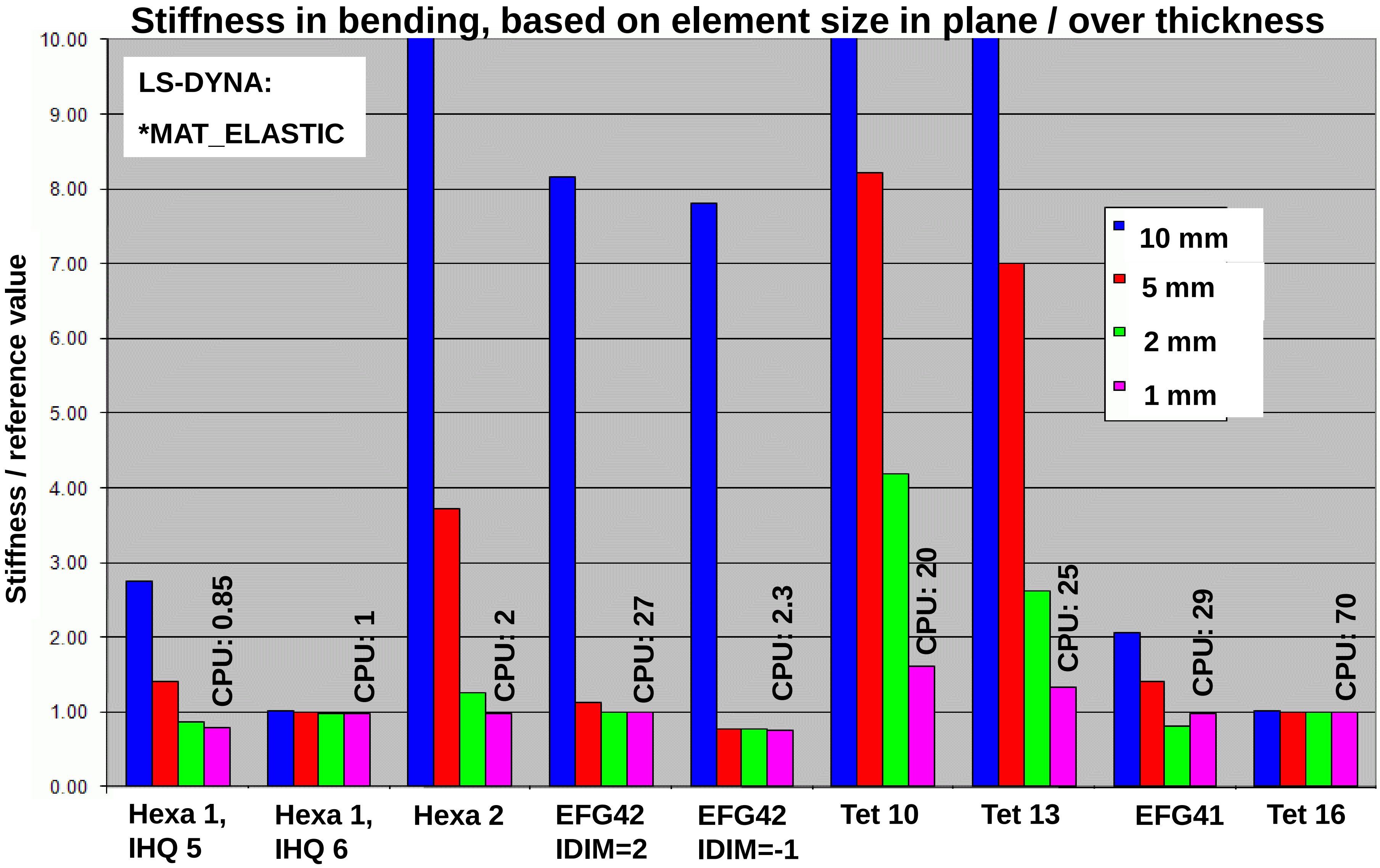
*SECTION_SOLID_EFG



Card 2 and Card 3 define only for the EFG option

Variable	DX	DY	DZ	ISPLINE	IDILA	IEBT	IDIM	TOLDEF
Type	F	F	F	I	I	I	I	F
Default	1.01	1.01	1.01	0	0	-1	2	0.01
Variable	IGL	STIME	IKEN	SF	MID	IBR	DS	ECUT
Type	I	F	I	F	I	I	F	F
Default	0	1.e+20	0	0.0		1	1.01	0.1

EFG for thin walled structures



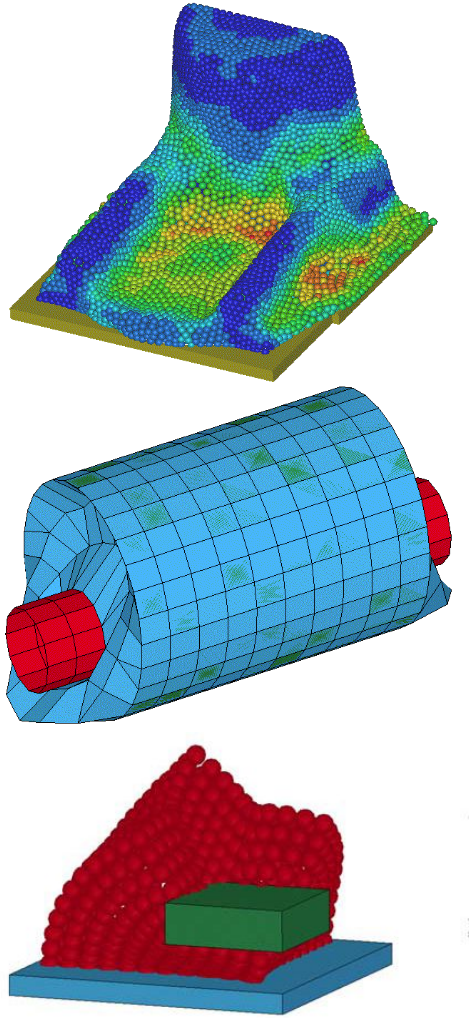
EFG

Recomendations

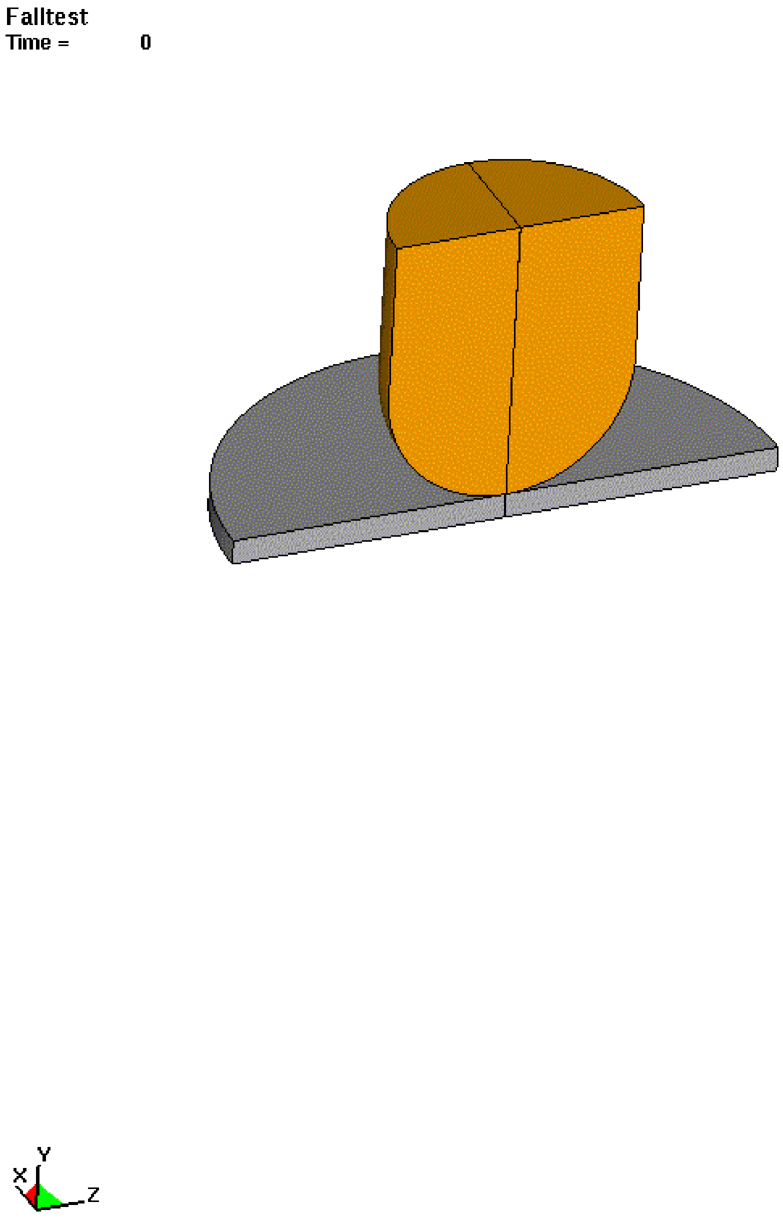
Metal materials in Forging:
- Lagrangian kernel (TOLDEF=0)
- Maximum Entropy approximation (IEBT=7)
- Two point Gauss integration (IDIM=2)

Rubber materials:
- Lagrangian kernel (TOLDEF=0)
- Maximum Entropy approximation(IEBT=7)
- Stabilized domain integration (IDIM=-1)

Foam materials:
- Semi-Lagrangian kernel (TOLDEF>0)
- Fast transformation (IEBT=4)
- Default domain integration (IDIM=1)

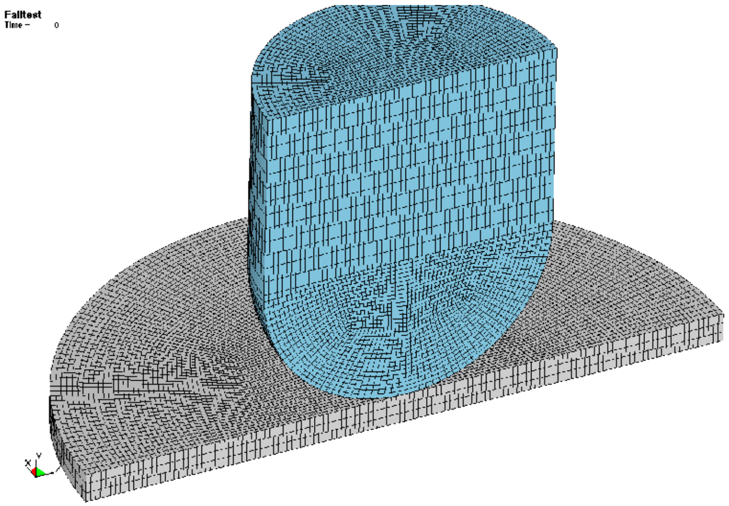


EFG



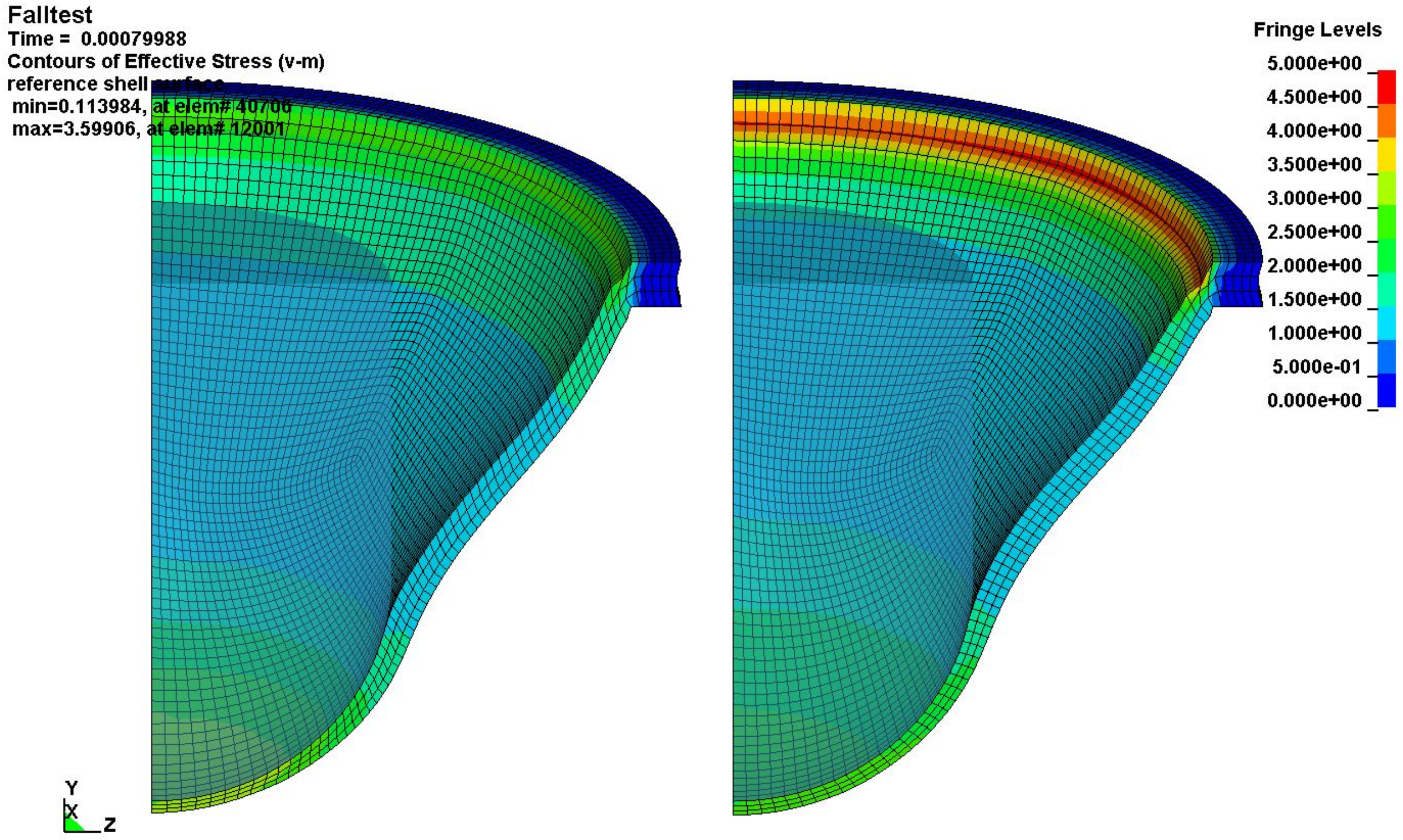
Example:
Rubber Impact with rigid projectile

- -EFG type 41 (hexahedron)
- -Maximum Entropy approximation IEBT=7
- -Stabilized method IDIM=-1 (very fast)
- -*MAT_SIMPLIFIED_RUBBER



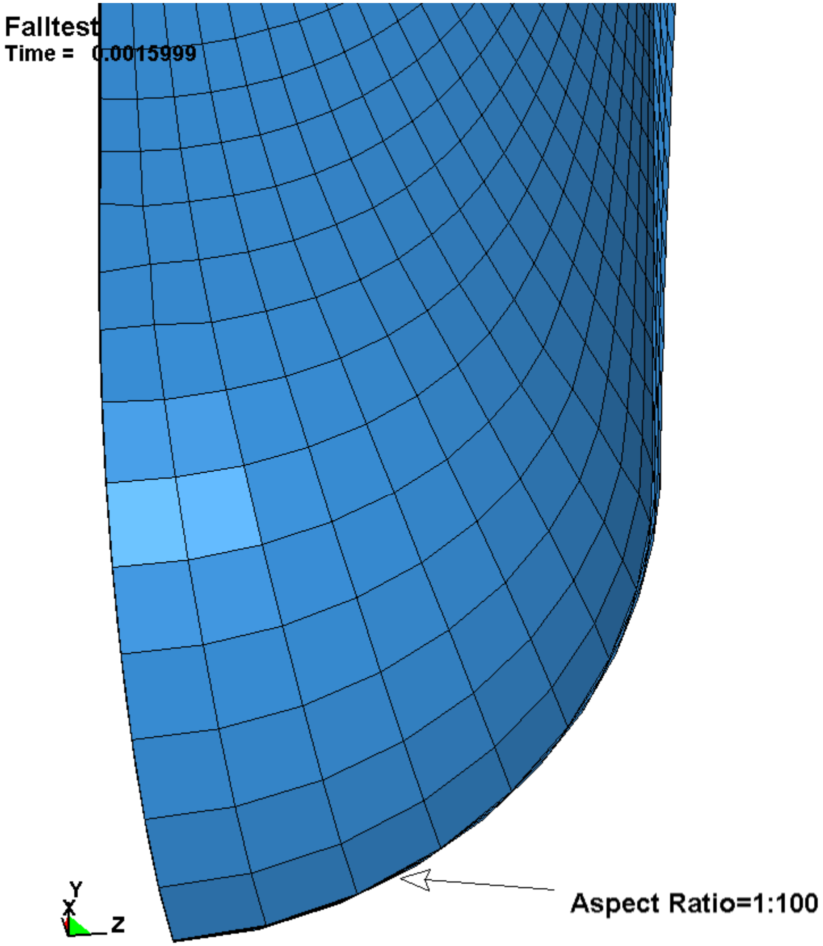
EFG

Comparims: EFG Type 41 FEM Type 1

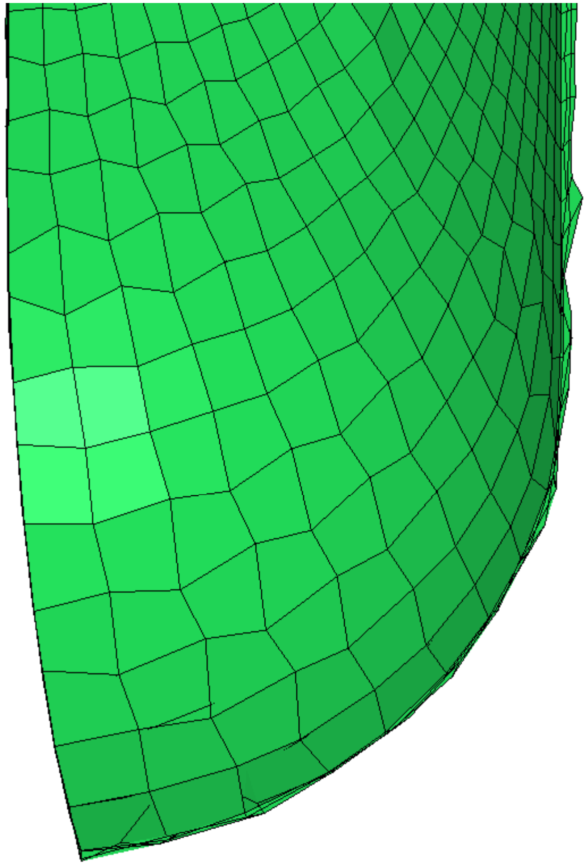


EFG

Comparims: EFG Type 41



FEM Type 1 (last state)



ALE

ALE
Arbitrary Lagrangian Eulerian

ALE

Lagrangian- formulation:

- is used in structure mechanics
- material and elements are bonded together
- large deformation induces element distortion

Eulerian- formulation:

- is used in fluid mechanics
- mesh of elements is fixed in space
- material ‘flows’ through the elements
- variable boundary conditions are complicated

ALE: Arbitrary- Lagrangian- Eulerian:

- both formulations in combination: the mesh can move and deform, the material can flow trough the mesh
- two possible kinds of applications:

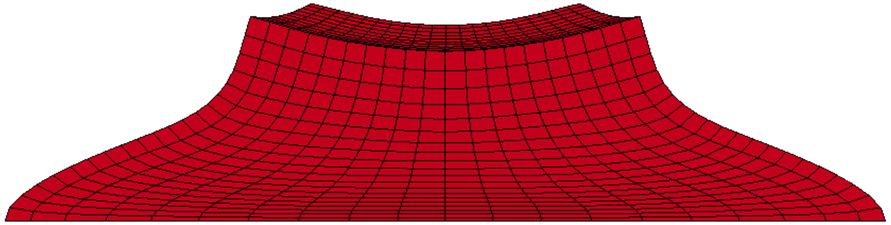
  à REZONING: large deformation in structure mechanics;
  mesh must be corrected

  à FLUID-STRUCTURE-INTERACTION: Airbag inflation, Tank sloshing

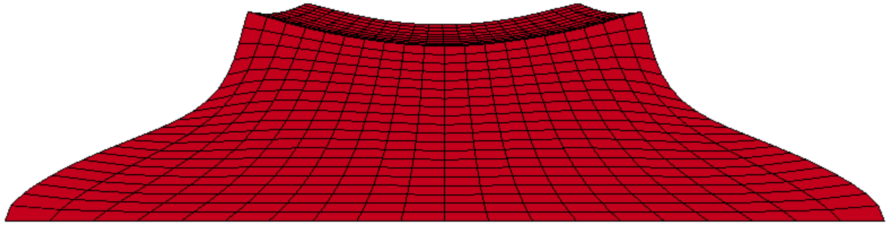
ALE

ALE :=Material(s) can *flow* through the mesh which itself can *move*

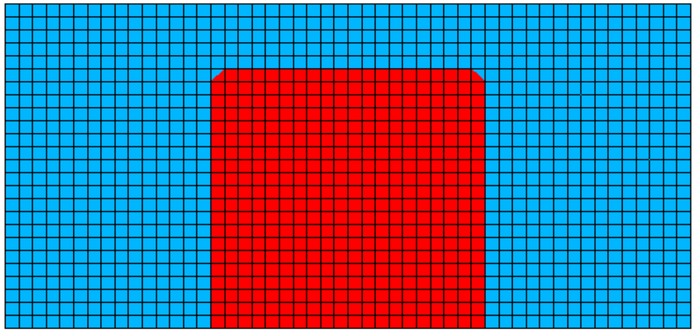
Lagrangian formulation



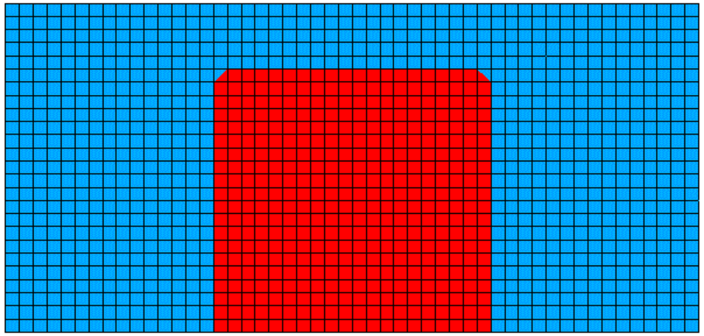
Single material ALE



Multi-material Eulerian



Multi-material ALE formulation



ALE *ELEMENT_SOLID

ALE hexahedron (type 5)

- 8 noded hexahedron element with trilinear shape function with reduced integration
- single material Arbitrary Eulerian Lagrangian element

à coupling of lagrangian and eulerian formulation: materials flow through elements

- useful for simulations with large element distortion but remeshing not necessary
- approx. 2 times slower than a lagrangian element elform=1

Single material and void ALE - (type 12)

- Eulerian element which can be filled by one material or void
- use in combination with void-definition (12)

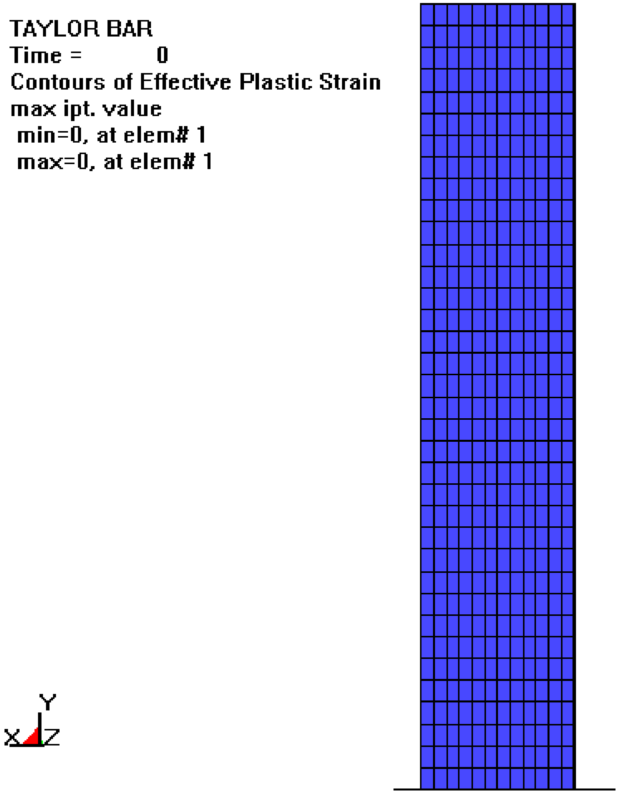
Multi material ALE - (type 11)

- Eulerian element which can be filled by an arbitrary number of materials
- use in combination with *ALE_MULTI-MATERIAL_GROUP (11)
- approx. 2.5 times slower than a lagrangian element elform=1

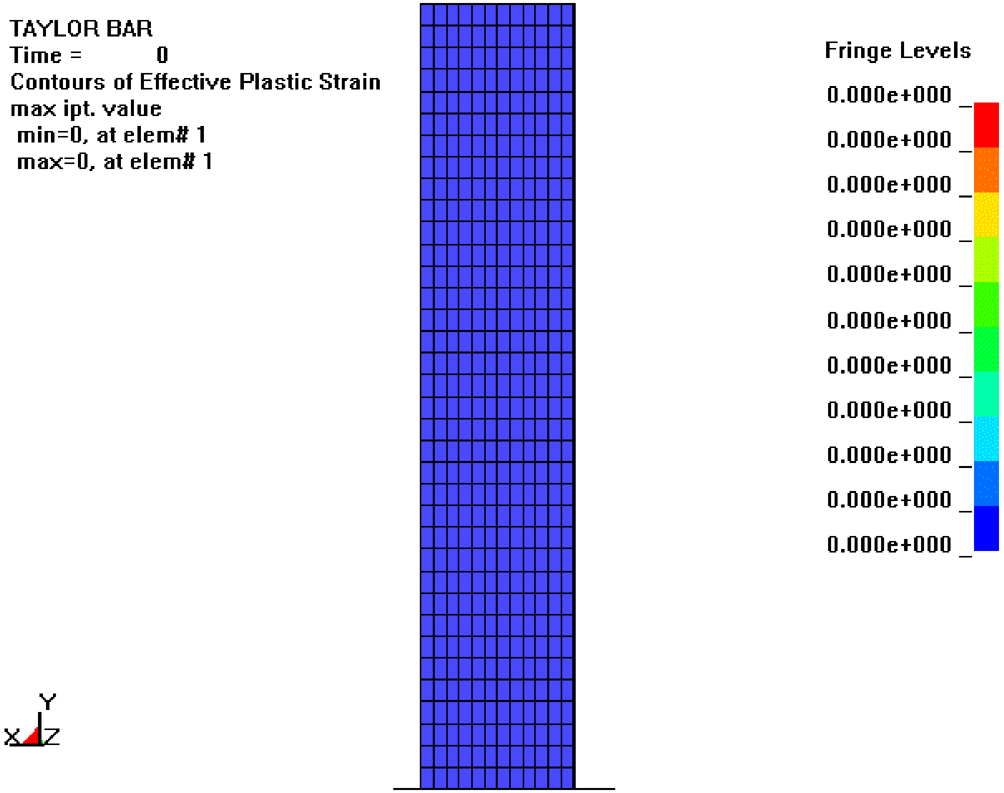
ALE Example

Testcase: Taylor beam

Hexahedron elform=1



Single material euler element elform=5

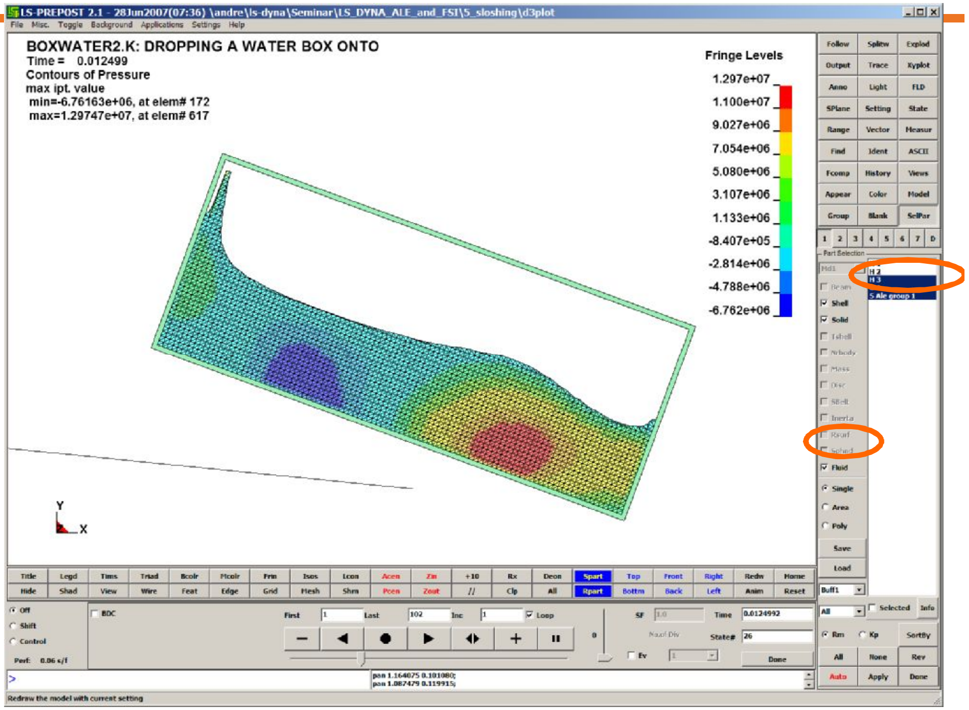


ALE Postprocessing

Display of interface reconstruction for multi material euler elements

Part selection: Fluid
 à additional ALE parts

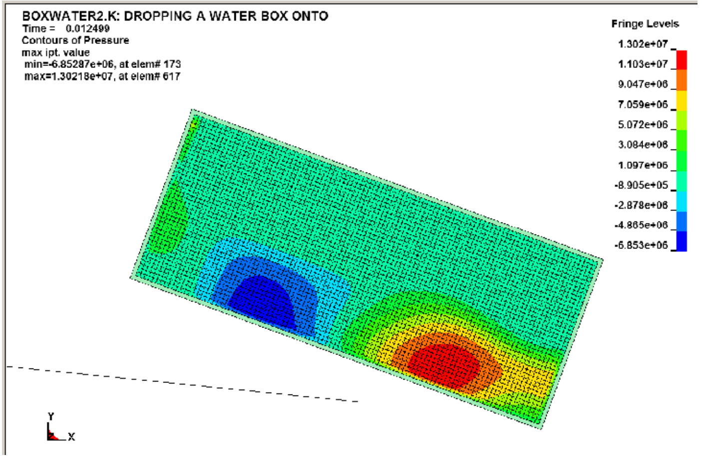
-reconstruction of geometry, mesh with tetrahedrons
-results on ALE material outer surface

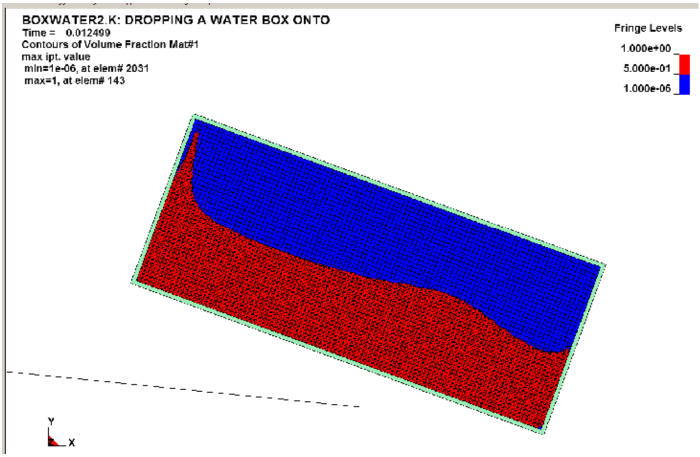

LS-PREPOST 2.1 - 28Jun2007(07:36) \andre\ls-dyna\Seminar\LS_DYNA_ALE_and_FSI\5_sloshing\d3plot

BOXWATER2.K: DROPPING A WATER BOX ONTO
Time = 0.012499
Contours of Pressure
max ipt. value
min=-6.76163e+06, at elem# 172
max=1.29747e+07, at elem# 617

pan 1.164075 0.101080;
pan 1.087479 0.119915;

Redraw the model with current setting

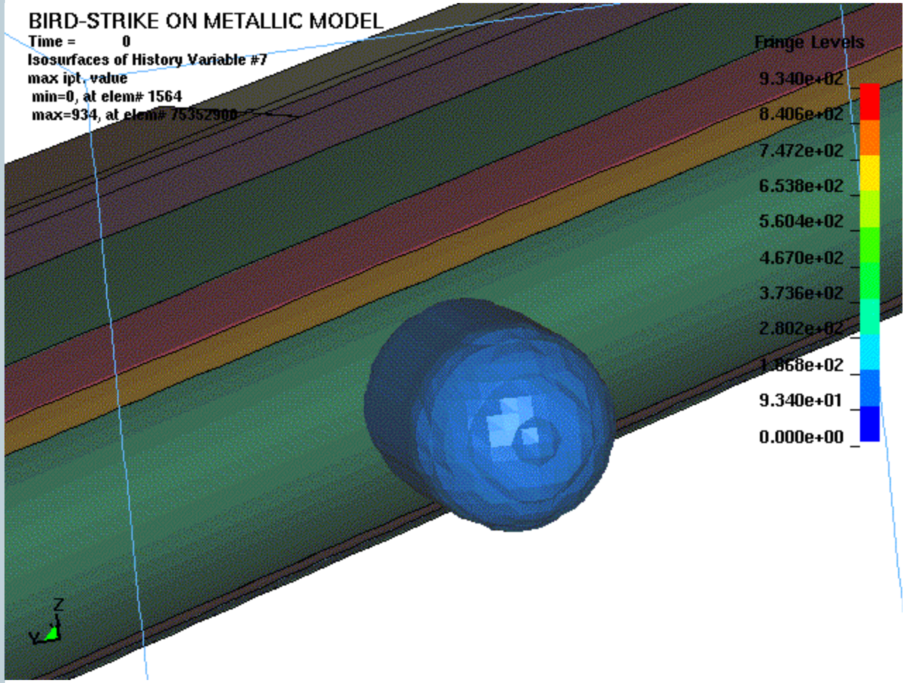

BOXWATER2.K: DROPPING A WATER BOX ONTO
Time = 0.012499
Contours of Pressure
max ipt. value
min=-6.85287e+06, at elem# 173
max=1.30218e+07, at elem# 617


BOXWATER2.K: DROPPING A WATER BOX ONTO
Time = 0.012499
Contours of Volume Fraction Mat#1
max ipt. value
min=1e-06, at elem# 2031
max=1, at elem# 143

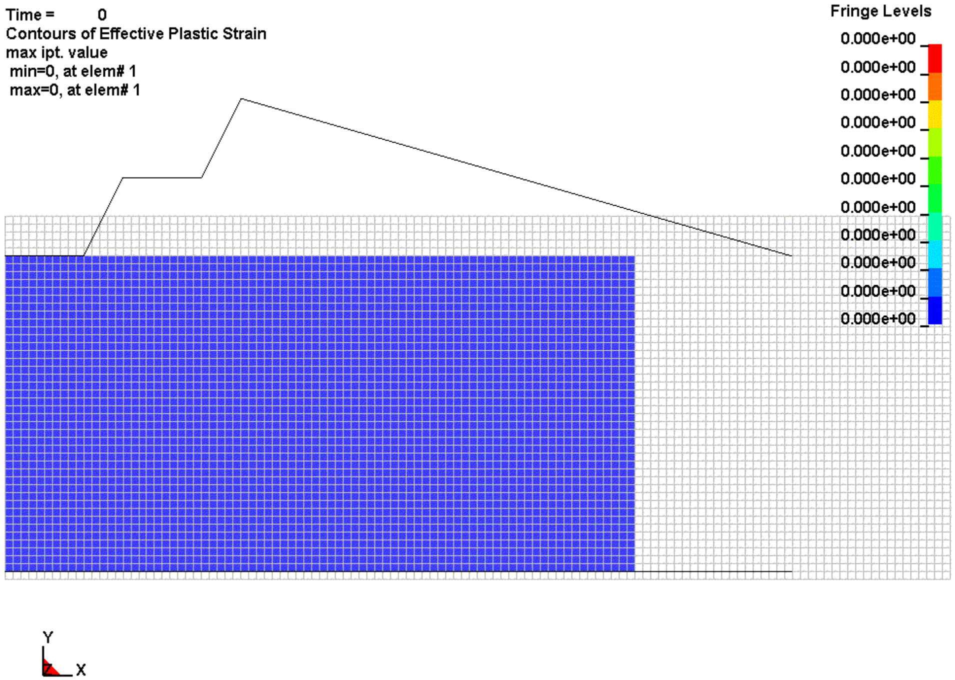
ALE Example

Examples for multi material euler:

Bird strike



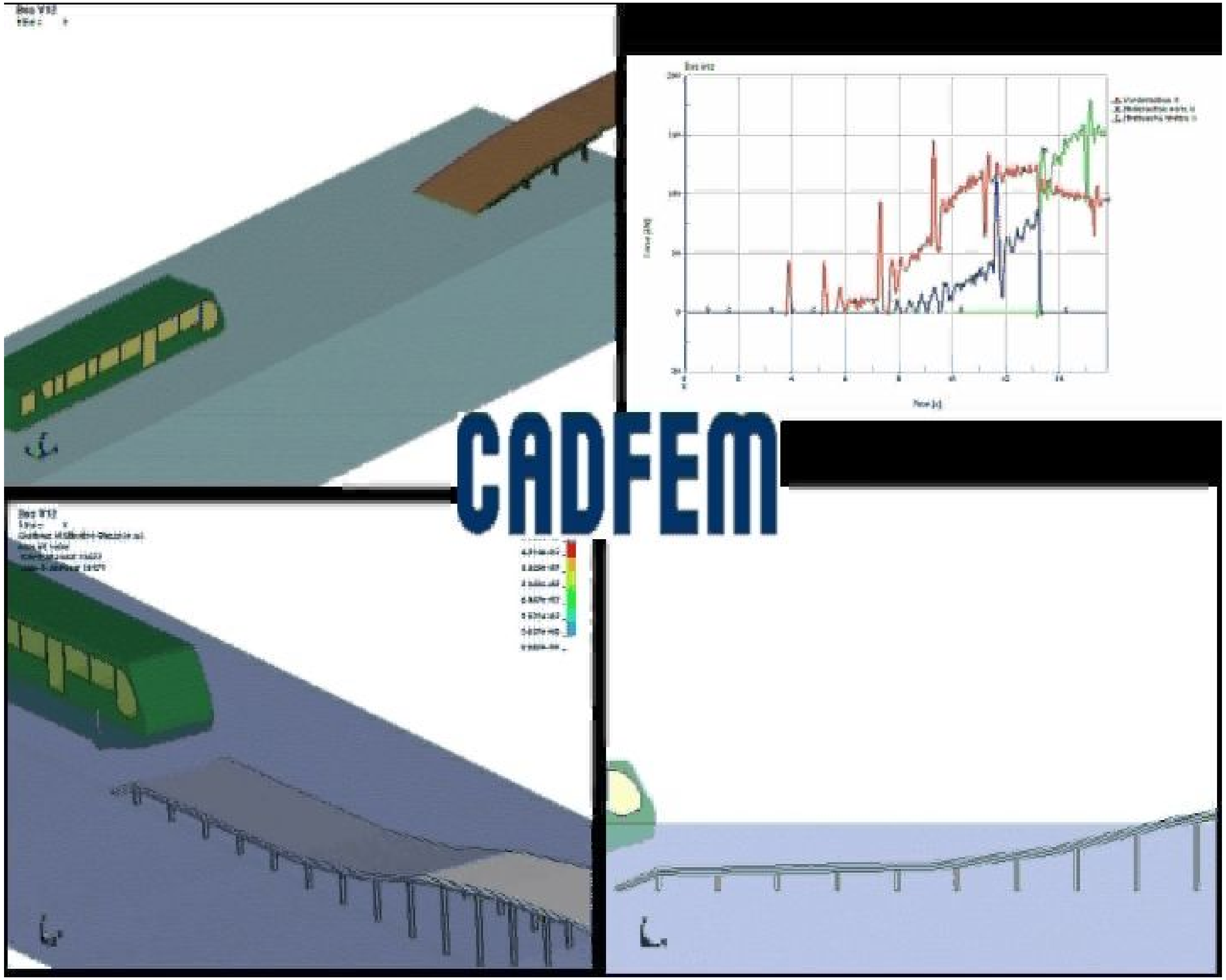
2d forging



ALE

Example

Fluid structure interaction. Eulerian elements for water.



SPH

Not really Elements, only particles

- but defined with *ELEMENT_SPH

SPH

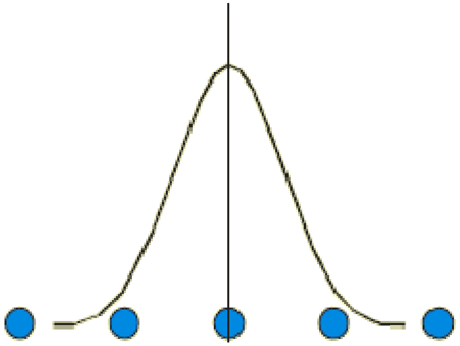
Smooth Particle Hydrodynamics

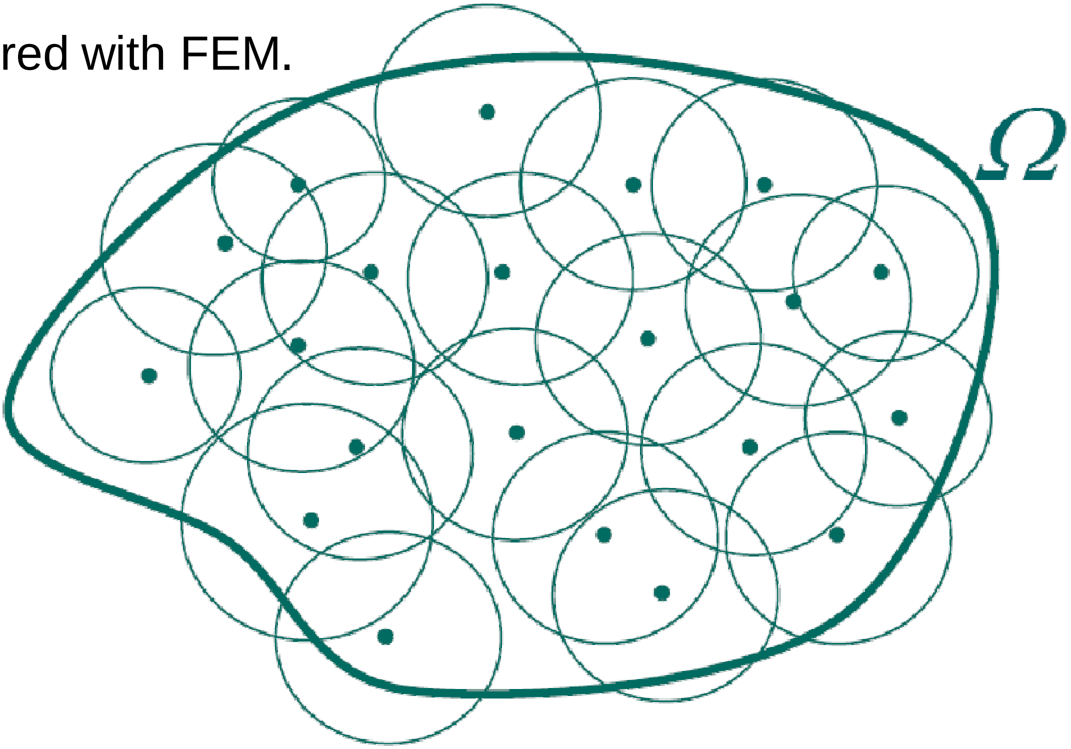
SPH

-A collection of nodes (particles) discretize a continuum.
-These are mass points that describe the motion of the continuum.

-SPH is a lagrangian collocative method, i.e. the nodes and the integration points are the same. Shape functions are centered on the particles.

-Accuracy is not high compared with FEM.



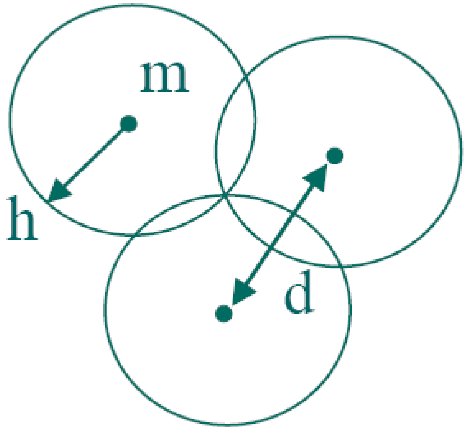


SPH

Particle has mass m – has to be defined independently from density

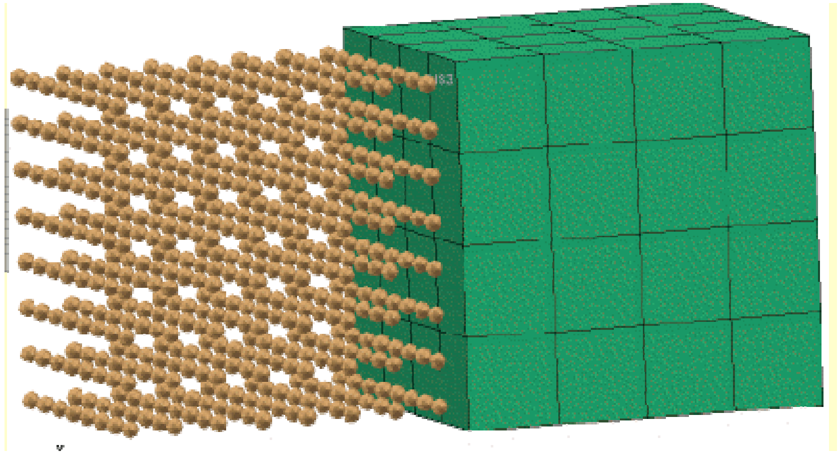
2 parameters of discretization:
d is distance between particles
h is smoothing length

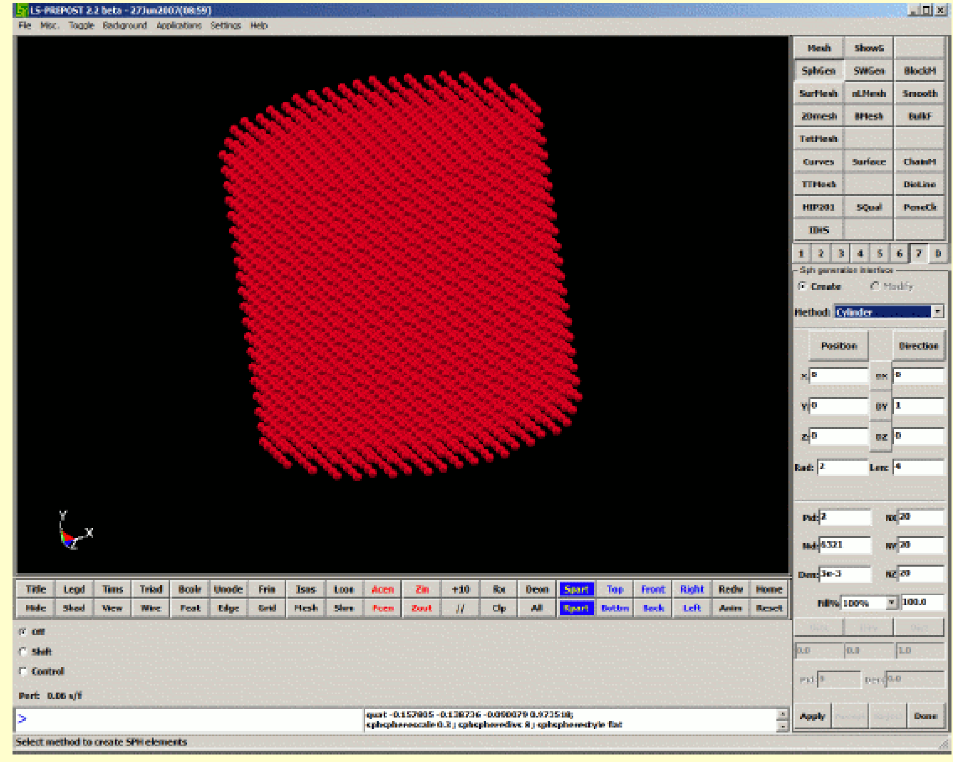
– different than classical methode



SPH Meshing

- Minimum of four particles per one solid element face
- SPH mesh should always be regular





Method: Cylinder

Select method to create SPH elements

SPH meshing in LS-PREPOST:
Page 7 à SphGen

Computational cost:
-Time step size is comparable to solidelements
- CPU time approx. 2.5-3 times slower that Hexahedron type 1

SPH Connection with FE Model

Regular contact with:

-*CONTACT_AUTOMATIC_NODES_TO_SURFACE

- SPH nodes on the slave side

Tied contact:

-*CONTACT_TIED_NODES_TO_SURFACE

-SPH nodes on the slave side

-Note: SPH nodes do not have any thickness in contact!

Contact between SPH parts:

-see *CONTROL_SPH, parameter CONT

SPH Recommendations

Other recommendations:

-reduce time step scale factor (*CONTROL_TIMESTEP, tssfac<0.8

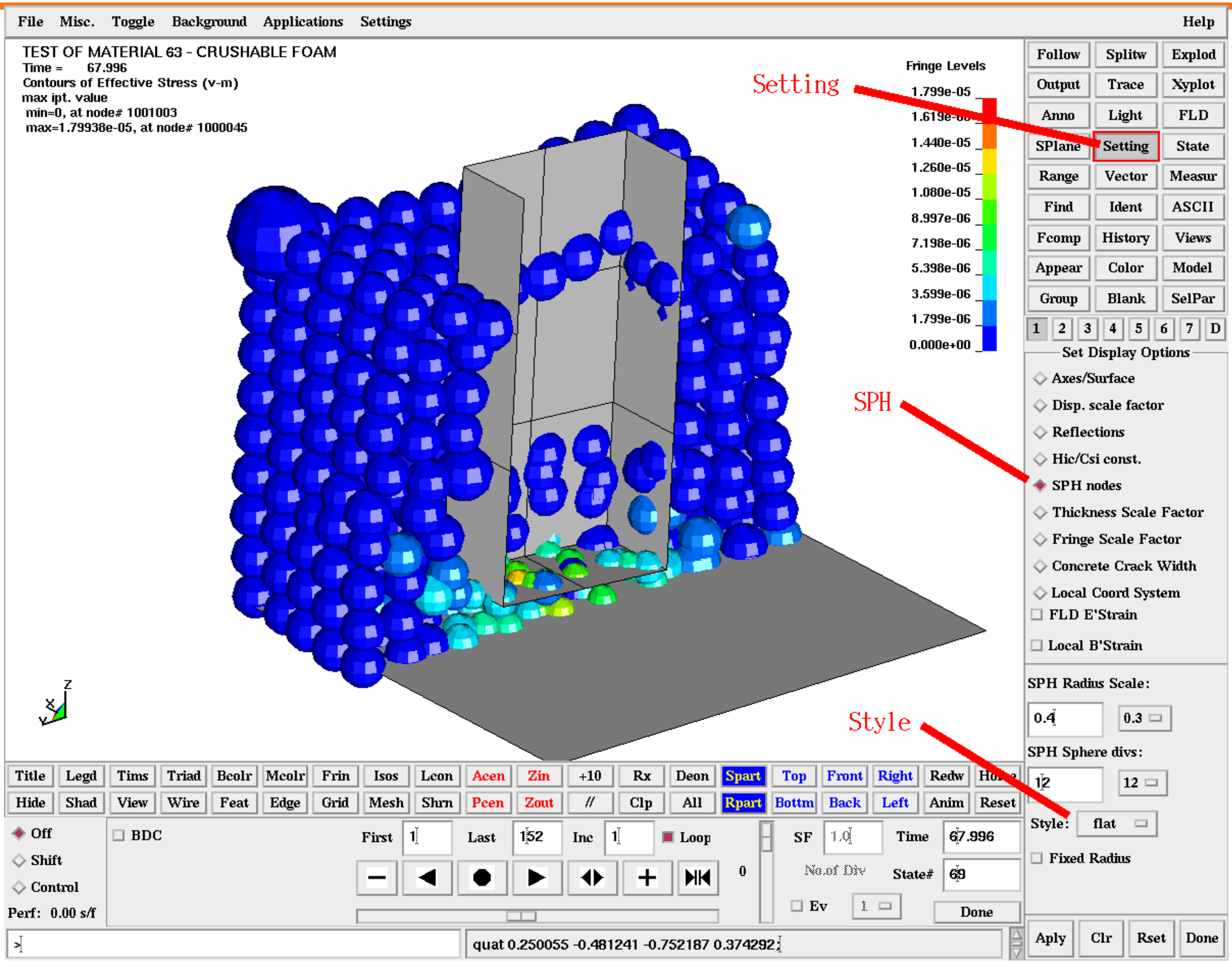
-do not use mass scaling

-use a SPH box (*CONTROL_SPH, boxid), particles flowing outside of this box not longer computed

-use negative MEMORY value (*CONTROL_SPH) is often more stable

-look in the material section of Keyword users manual for valid material model

SPH Postprocessing



TEST OF MATERIAL 63 - CRUSHABLE FOAM
Time = 67.996
Contours of Effective Stress (v-m)
max ipt. value
min=0, at node# 1001003
max=1.79938e-05, at node# 1000045

SPH Radius Scale: 0.4 0.3
SPH Sphere divs: 12 12
Style: flat

First 1 Last 152 Inc 1
SF 1.0 Time 67.996
State# 69
Perf: 0.00 s/f

quat 0.250055 -0.481241 -0.752187 0.374292

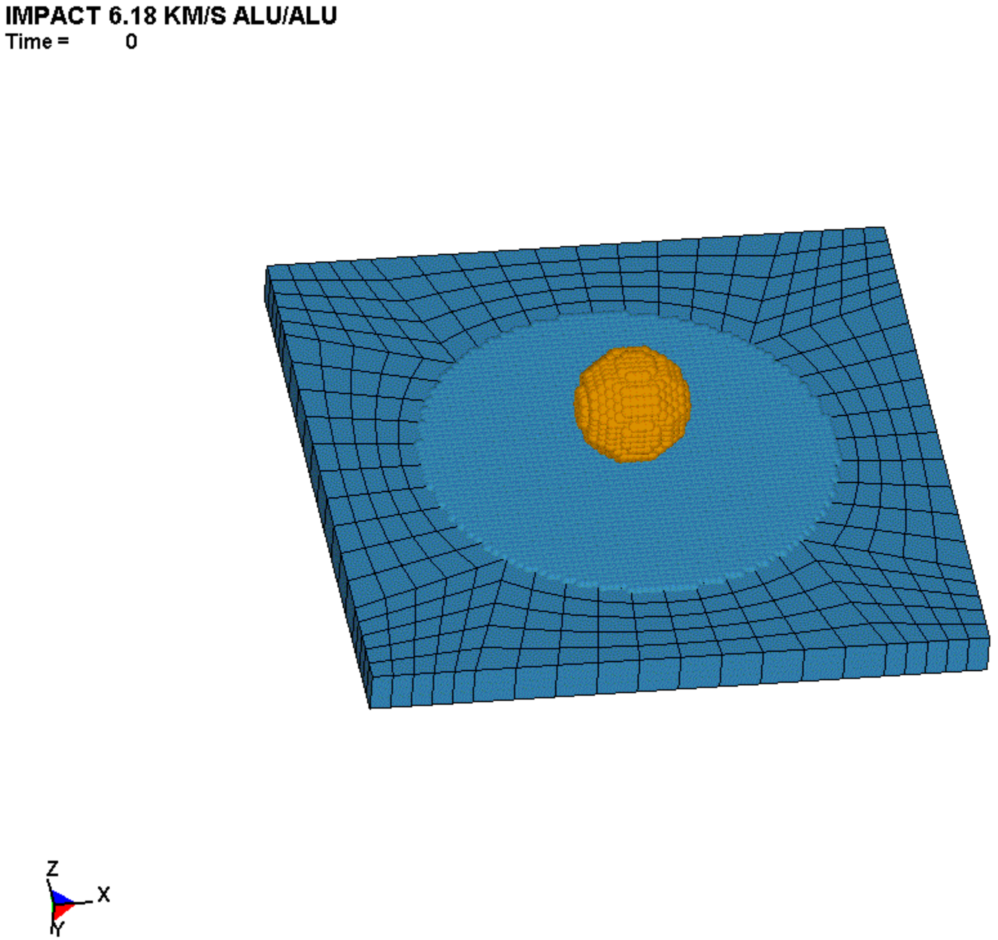
SPH Examples

Testmodel:

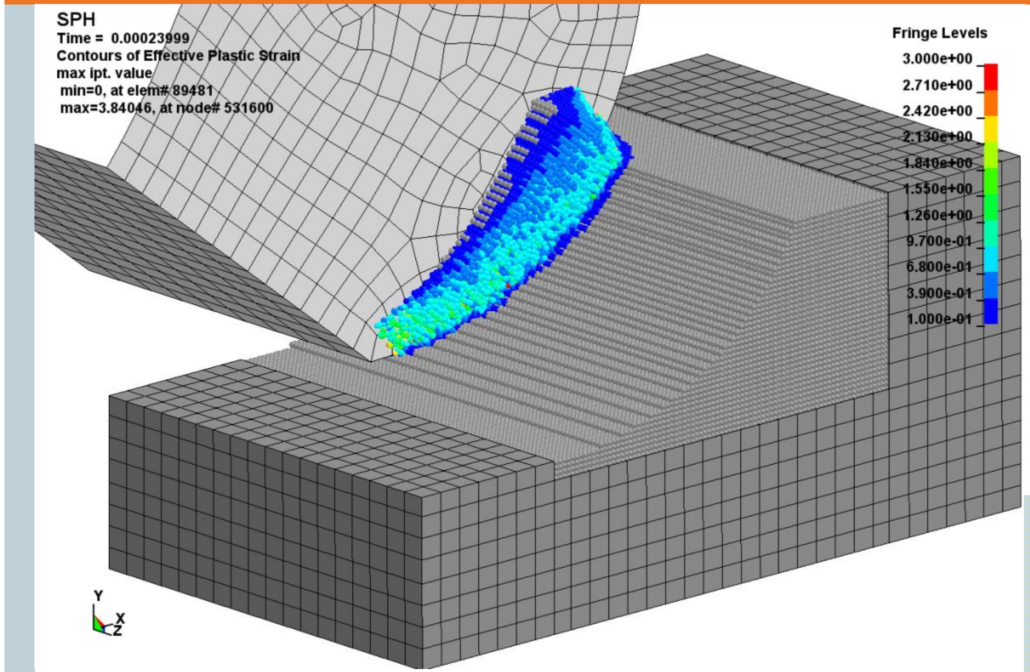
High velocity impact:
v=6.18 km/s

19.000 SPH elements
300 Solid Elements
600 Cycles

2min on this laptop



SPH Examples



Testmodel:
130.000 SPH elements
5.000 Solid Elements
60.000 Cycles

4h 30min on a quad core CPU

CPM

Looks like SPH but is completely different:

-

CPM

Corpuscular Particle Method

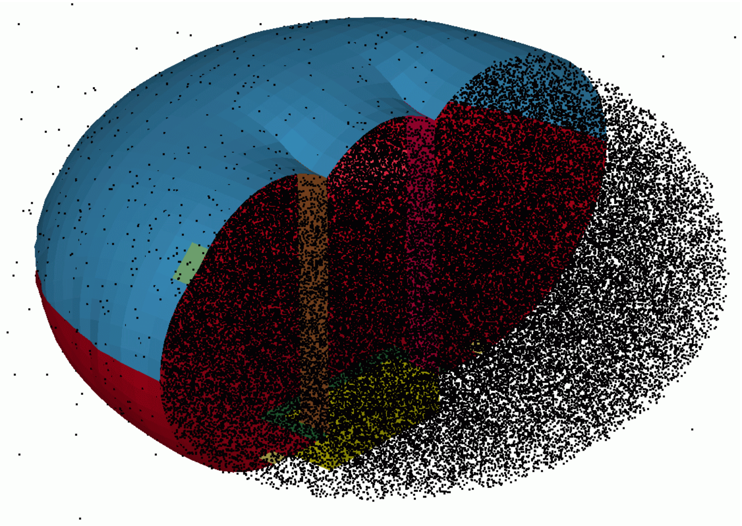
for gases only

CPM

Basic idea: Model the gas as a set of rigid particles in random motion

Advantages:

- no field equations – easy and numerically robust
- contact with solid parts simple with *contact_nodes_to_surface



The theory describes the interaction between gas molecules on a microscopic level.

Original application: Airbag deployment

Problem: A deployed airbag is filled up by roughly 10^{24} molecules.
Solution: Reduce system from many molecules to a “few” particles
Now a airbag is filled by 500.000 particles

CPM *airbag_particle

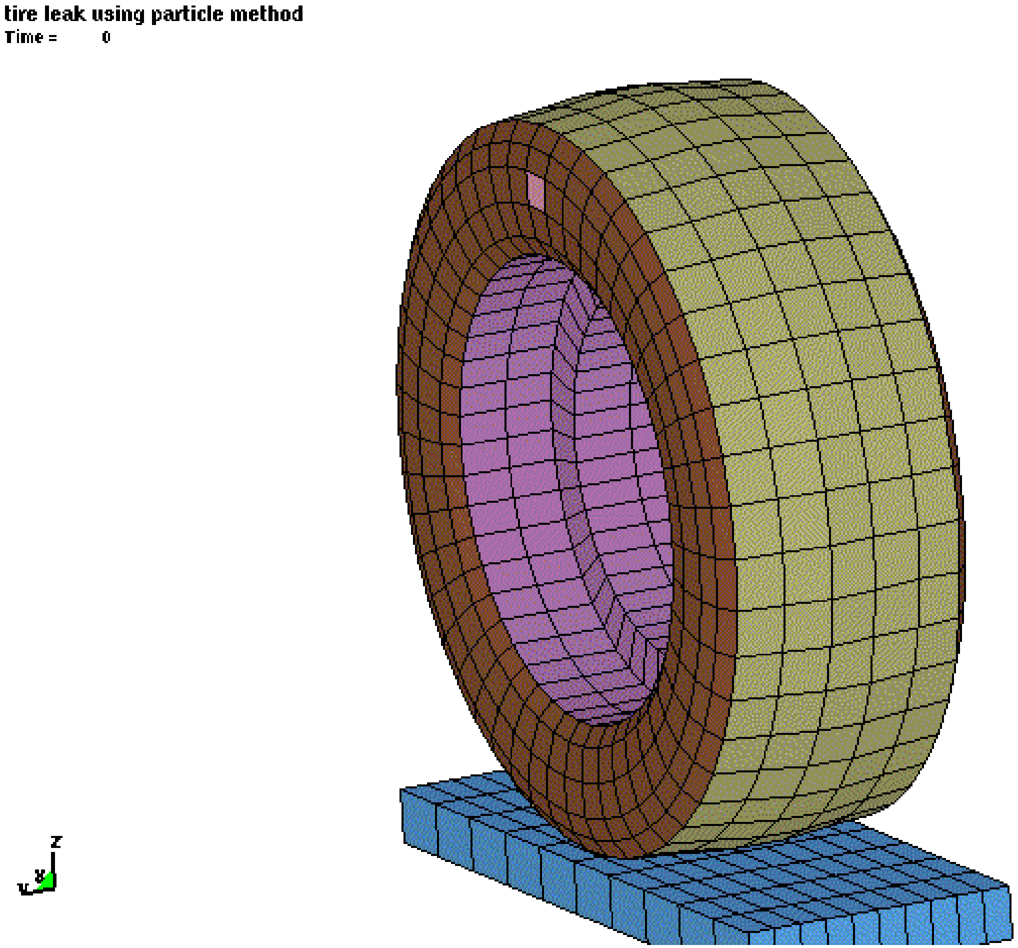
IAIR=2: Initial gas in Bag

Application:

Initialize pressure in a closed volume

- Airbags
- Door cavity
- Tires

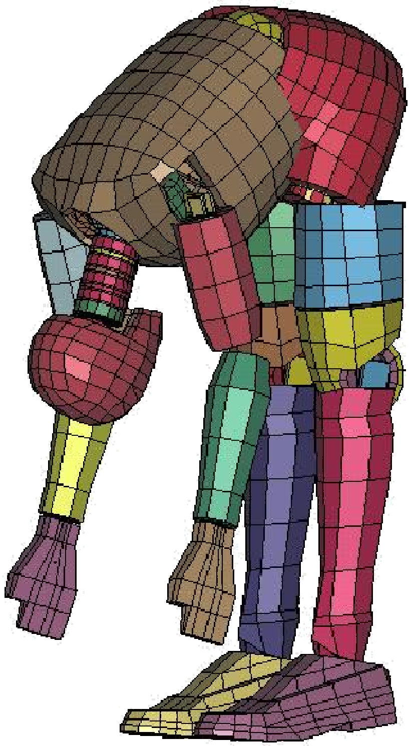
This makes the method open for much more than airbags.



Kontakt

CAD-FEM GmbH
Marktplatz 2
DE-85567 Grafing b. München

Telefon +49 (0)8092-7005-0
Telefax +49 (0)8092-7005-77
E-Mail info@cadfem.de
www.cadfem.de



Vielen Dank für Ihre Aufmerksamkeit!