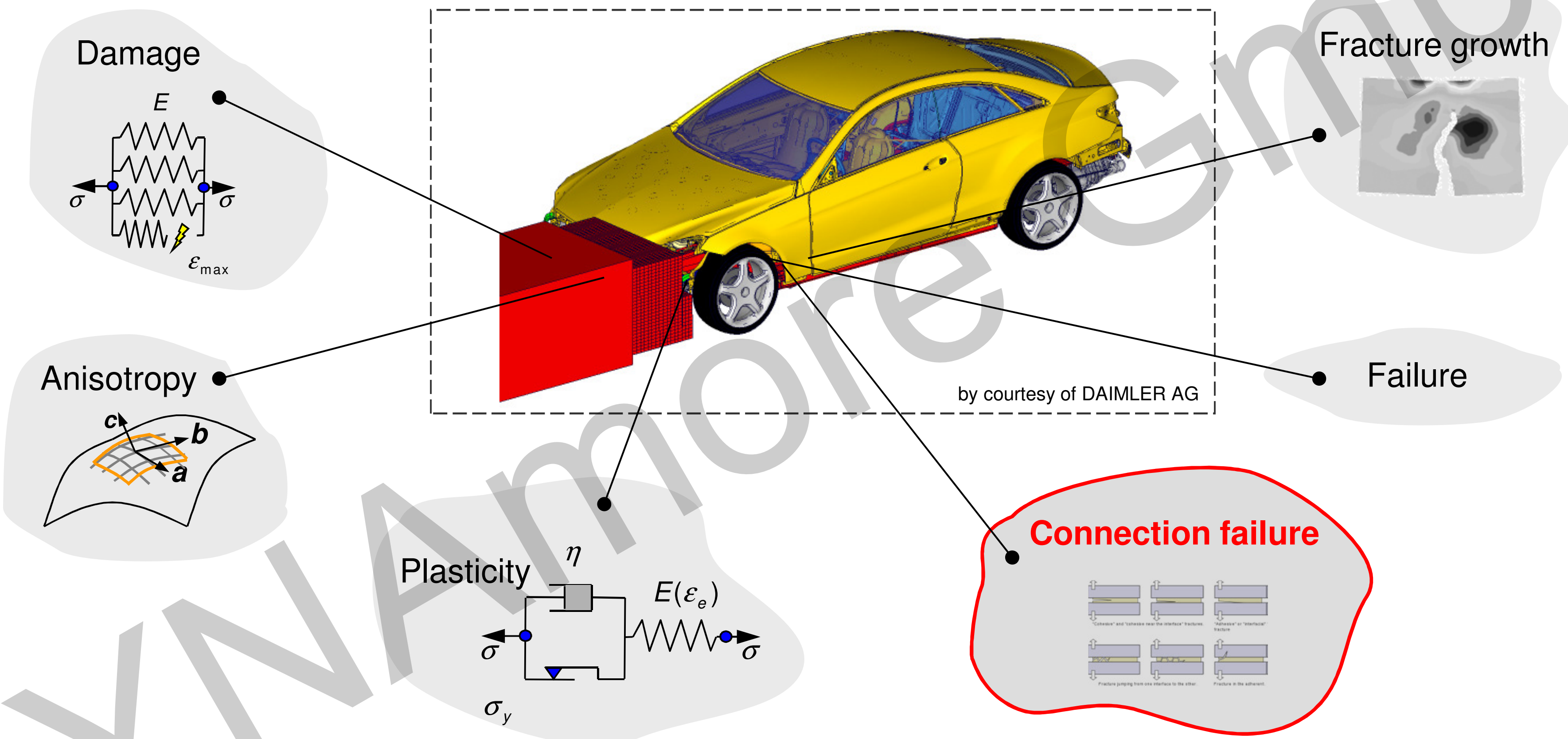
Workshop on connection modeling with LS-DYNA

- General introduction
- Spotweld modeling – modeling of punctiform con.
- Adhesives modeling – modeling of line and area shaped con.
- Bolted connections

Tobias Graf
{tg}@dynamore.de

- **General introduction**
  - **Preliminary remarks**
    - current challenges in a full car crash simulation



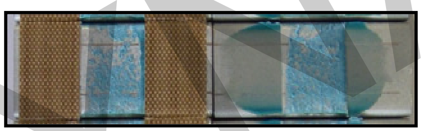
→ **due to a more often usage of high strength steels, the connections become more and more the weak points in crash**

- failure forces
  - main focus in connection modeling lies on the correct prediction of the respective failure forces
  - one has to distinguish between two totally different connection categories:

Failure forces depend on the connection material

- e.g., adhesive bonding
- identify material parameters using a certain material combination
- **idea:** The material model can be applied to any other material combination

→ not so much experiments are necessary

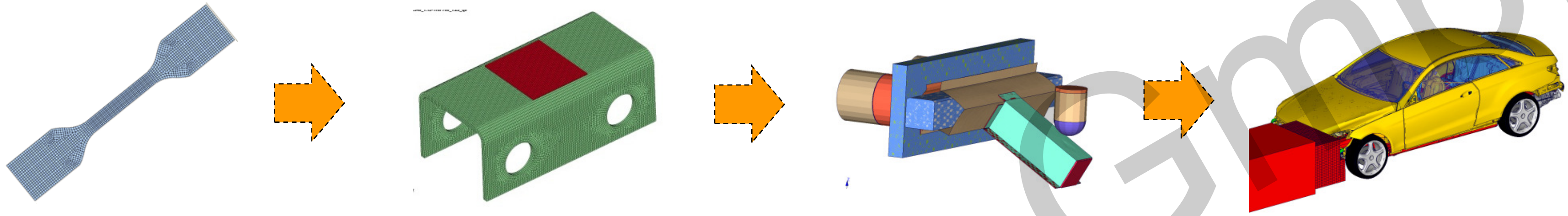


Failure forces depend on the connected materials

- e.g., spot welding
- material parameters have to be identified for every individual material combination
- **idea:** Find a relation to predict failure forces based on the material parameters of the sheet materials and geometry information

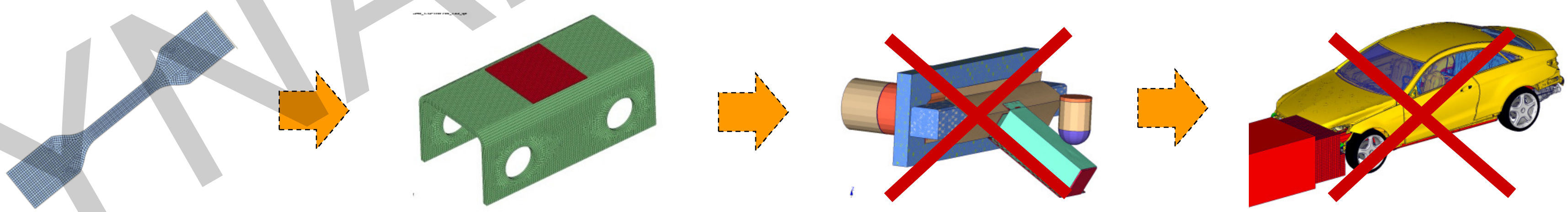
→ a lot of experiments are necessary

- verification and validation process
  - **problem I:** element size and explicit time integration
    - » ideal procedure: A detailed model with physical material parameters can be used on every scale of interest

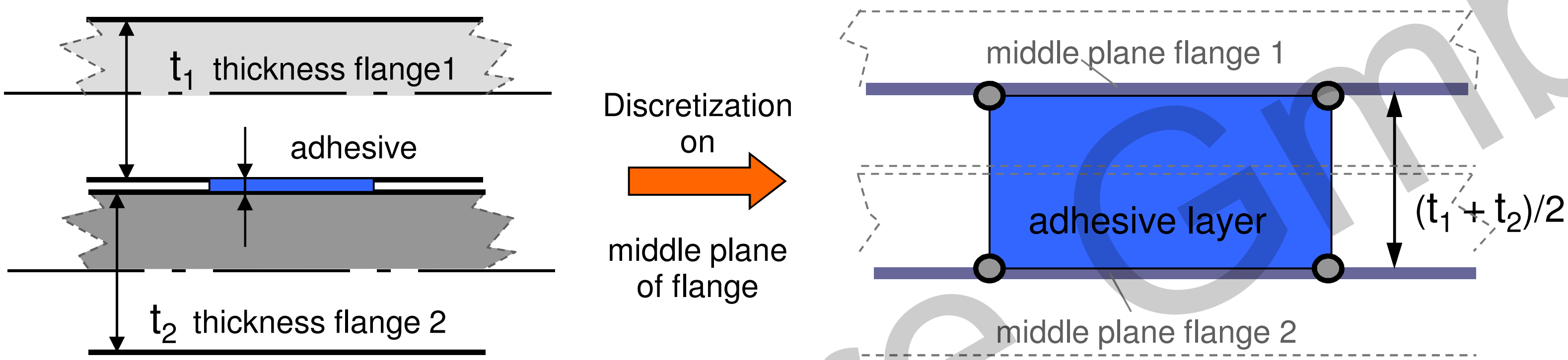


→ verification is done on the smallest scale
→ validation can be done on KS-II and component scale

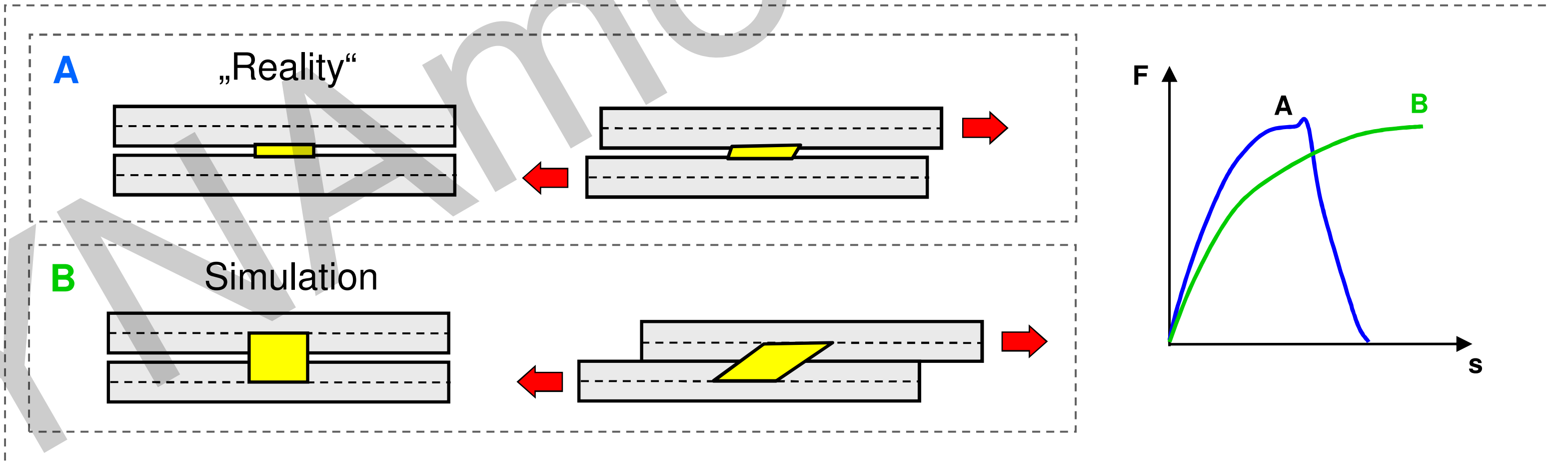
  - the spatial discretization of the connection has to be very fine compared to the element size usually used on the component and full car scale
  - explicit time integration → decreasing of time step or increasing of additional mass
  - because of limited CPU-power, the highest scale for the usage of a detailed model is currently the scale of the KS-II specimen



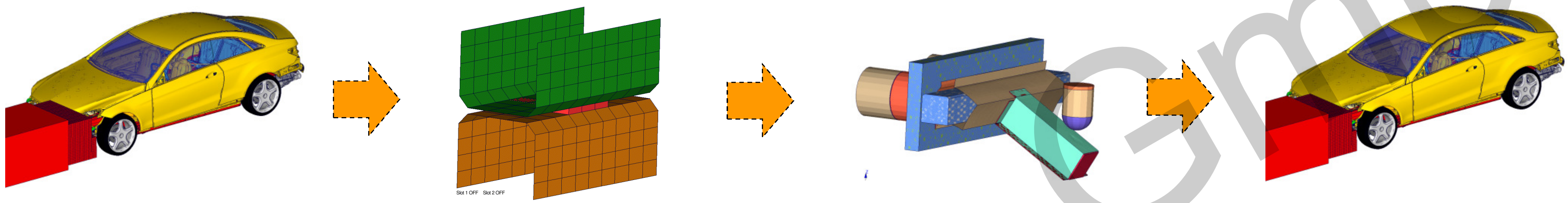
- verification and validation process
  - **problem II:** discretization issue
    - » flange materials are currently discretized with shell elements



- » consequence: Material behavior is too flexible using physical material parameters



– **requirement:** The spatial discretization and the respective material model of the connection has to be chosen in such a way that the performance and the validity of the full car simulation is not negatively affected.

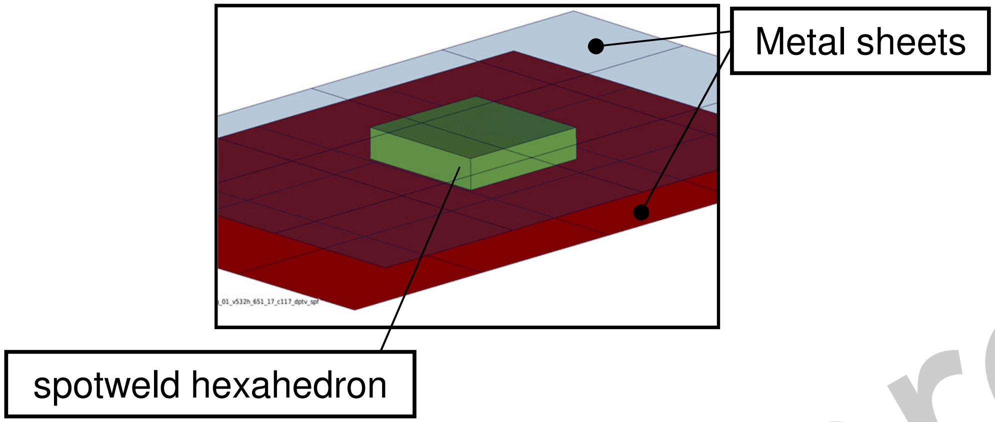


→ **The only applicable procedure for connection modeling is the usage of so-called substitute models with artificial material parameters**

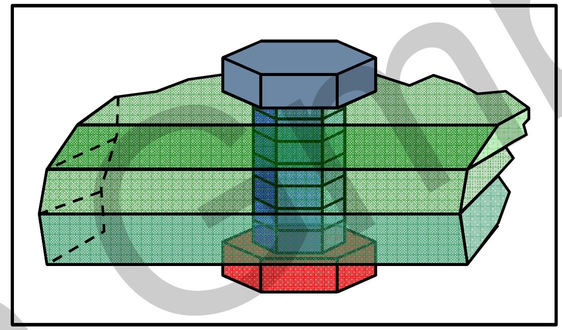
– currently used robust element types and corresponding material models
  - » hexahedron elements in combination with 3-d material models, e.g.,
    - » *MAT_SPOTWELD (*MAT_100),
    - » *MAT_SPOTWELD_DAIMLERCHRYSLER,
    - » *MAT_ARUP_ADHESIVE (*MAT_169),
    - » *MAT_FU_CHANG_FOAM (*MAT_083), …

- different joining techniques and the corresponding material models
  - punctiform
    - » spot welding, RIVTAC, …
    - » currently used robust material models:
      *MAT_SPOTWELD_{DAMAGE_FAILURE},
      *MAT_SPOTWELD_DAIMLERCHRYSLER +
      *DEFINE_CONNECTION_PROPERTIES, …
  - line-shaped
    - » MIG welding, MIG soldering, …
    - » currently used robust material model:
      *MAT_ARUP_ADHESIVE, …
  - area-shaped
    - » adhesive bonding: Structural adhesive, hood adhesive, PU windshield, …
    - » currently used robust material models:
      *MAT_ARUP_ADHESIVE,
      *MAT_FU_CHANG_FOAM, …

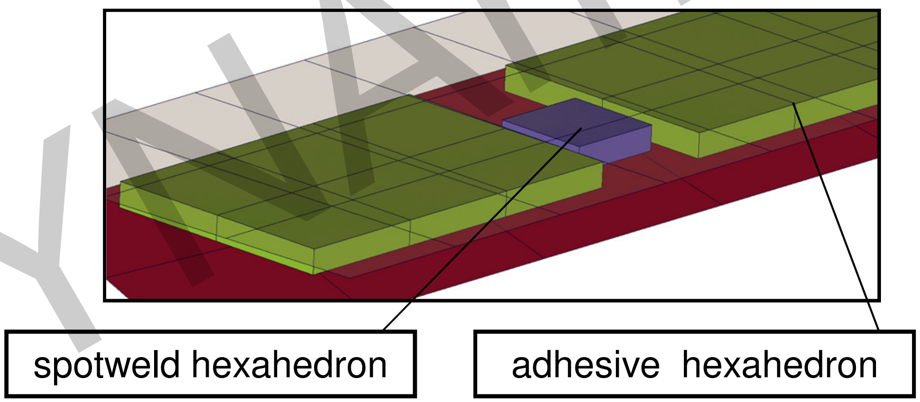
- several spatial discretization methods for different joining techniques
  - spot welding



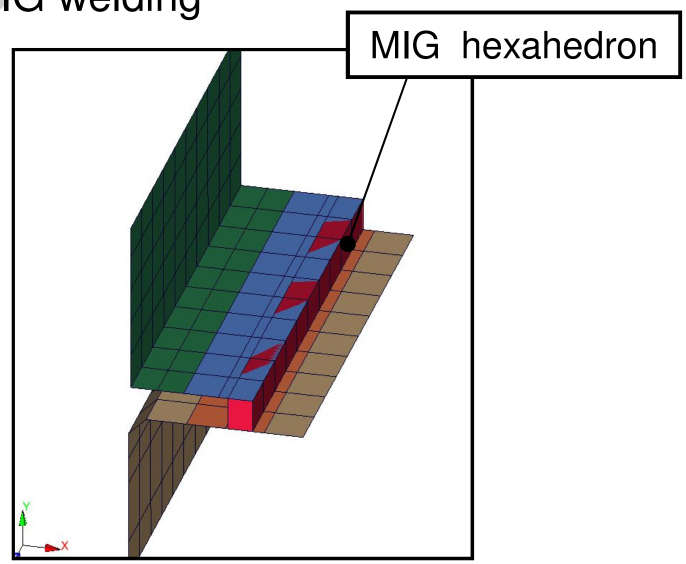
  - bolted connection



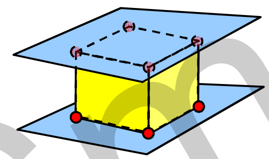
  - combination of spot welding and adhesive bonding



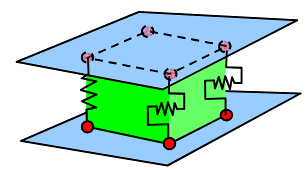
  - MIG welding



- **Recent trends in LS-DYNA**
  - element types and corresponding material models
    - volume elements and 3-d material models, e.g., *MAT_SPOTWELD (*MAT_100), *MAT_ARUP_ADHESIVE (*MAT_169)

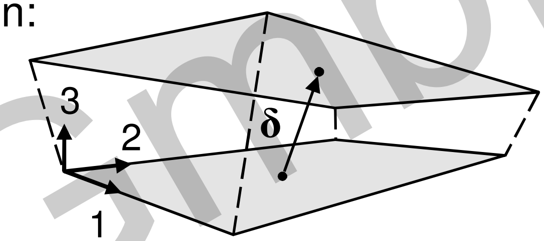


      - **material law:** stress vs. strain
        → critical time step depends on thickness
      - **disadvantage:** If element height tends to zero, e.g., switching from a shell disc. of the flanges to a discretization with solids, the critical time step tends to zero as well
        → impossible to use standard element formulations and corresponding material models
    - cohesive elements and corresponding material models, e.g., *MAT_COHESIVE_...



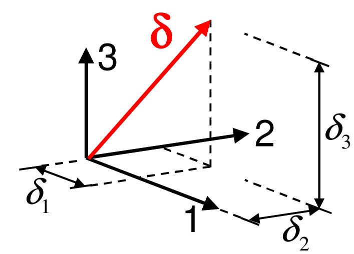
      - **material law:** stress vs. displacement
        → critical time step is independent of thickness
      - **advantage:** elements with zero height can be used without running into troubles regarding the critical time step

- cohesive elements and material modeling
  - material behavior can be defined individually for the normal and shear direction
    → correct definition of the thickness direction is extremely important
  - cohesive material laws are displacement (not strain) driven:
    → local relative displacements at integration points
    → local (interface) stresses



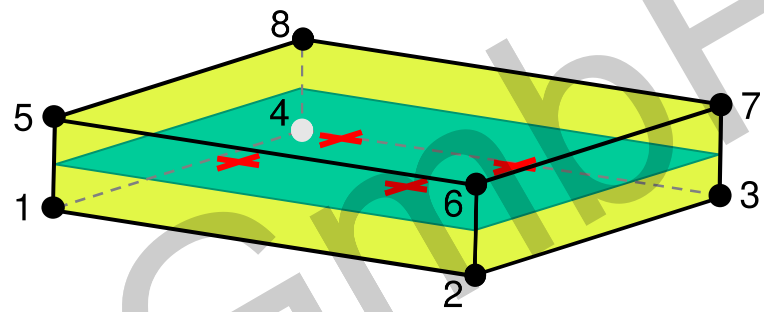
$$\begin{bmatrix} t_1 \\ t_2 \\ t_3 \end{bmatrix} = \begin{bmatrix} E_T & 0 & 0 \\ 0 & E_T & 0 \\ 0 & 0 & E_N \end{bmatrix} \begin{bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \end{bmatrix} \quad \left[N/mm^2\right] = \left[N/mm^3\right] \cdot [mm]$$

Interface stiffness is not the same
as *classical* stiffness



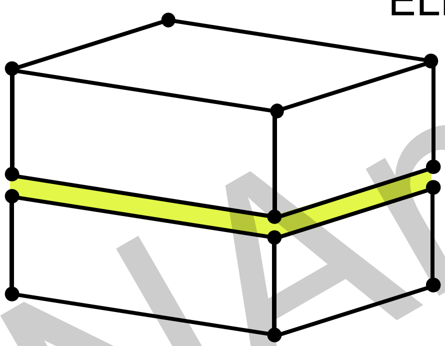
  - density can be specified per unit volume or per unit area
    → handling of elements with an initial volume of zero
  - LS-DYNA provides “special" volume elements:
    Account for orientation (element numbering),
    special treatment of thickness (critical time step)

- cohesive elements
  - » attached via coincident nodes or tied contact
  - » element numbering defines thickness direction
  - » in plane integration: 2x2 *Gauss*
  - » INTFAIL: Number of integration points required for the element to be deleted (*MAT_COHESIVE_...)



tie volume elements

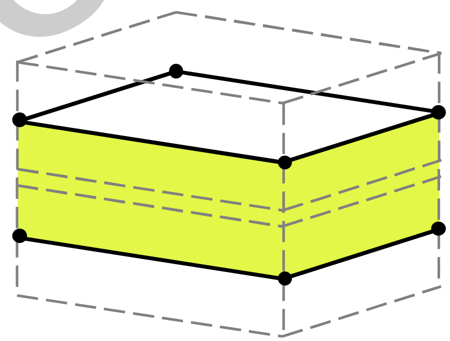
ELFORM=19



- moments have not to be transferred
- "null" thickness (→ ROFLG)
  → calculation of mass based on area

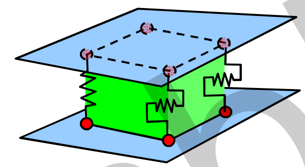
tie shell elements

ELFORM=20



- moments are transferred
- cohesive element with „offset"
- moments=forces · offset

– available cohesive material models in LS-DYNA (overview)



» *MAT_184: (*MAT_COHESIVE_ELASTIC)
→ simple elastic cohesive model

» *MAT_185: (*MAT_COHESIVE_TH)
→ cohesive model by *Tvergaard* and *Hutchinson*
→ tri-linear traction-separation law
→ same loading and unloading path; completely reversible

» *MAT_186: (*MAT_COHESIVE_GENERAL)
→ three irreversible mixed-mode formulations (TES=0,1,2)
→ arbitrary normalized traction-separation law → load curve (TSLC)

» *MAT_138: (*MAT_COHESIVE_MIXED_MODE)
→ simplification of *MAT_186 → restricted to linear softening
→ bi-linear traction-separation law
→ quadratic mixed mode delamination criterion

» *MAT_240: (*MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE)
→ rate-dependent, elastic-ideal plastic
→ tri-linear traction-separation law
→ quadratic yield and damage criterion in mixed-mode loading
→ damage evolution is governed by a power-law

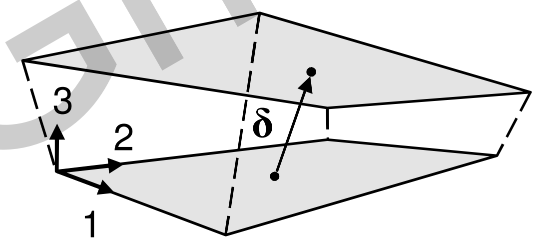
» **special case:** *MAT_169: (*MAT_ARUP_ADHESIVE)
→ rate-dependent, elastic-ideal plastic with damage
→ conti.-mech. material model, but several comp. of the stress tensor are neglected

R7.1.1

– 3-d models in conjunction with cohesive elements – *MAT_ADD_COHESIVE

» using this keyword, it is possible to combine currently the following material models with cohesive elements (ELFORM=19, 20):
*MAT_{1, 3, 4, 6, 15, 24, 41-50, 81, 82, 89, 96, 98, 103-107, 115, 120,
123, 124, 141, 168, 173, 187, 188, 193, 224, 225, 252 and 255}
assumption: No lateral expansion and no in-plane shearing

$$\begin{bmatrix}\delta_1\\ \delta_2\\ \delta_3\end{bmatrix} \rightarrow \begin{bmatrix}\dot{\varepsilon}_{xx}\\ \dot{\varepsilon}_{yy}\\ \dot{\varepsilon}_{zz}\\ \dot{\varepsilon}_{xy}\\ \dot{\varepsilon}_{yz}\\ \dot{\varepsilon}_{zx}\end{bmatrix} = \begin{bmatrix}0\\ 0\\ \dot{\delta}_3/(t+\delta_3)\\ 0\\ \dot{\delta}_2/(t+\delta_3)\\ \dot{\delta}_1/(t+\delta_3)\end{bmatrix} \qquad \begin{bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \sigma_{zz}\\ \sigma_{xy}\\ \sigma_{yz}\\ \sigma_{zx}\end{bmatrix} \rightarrow \begin{bmatrix}t_1\\ t_2\\ t_3\end{bmatrix} = \begin{bmatrix}\sigma_{zx}\\ \sigma_{yz}\\ \sigma_{zz}\end{bmatrix}$$



» keyword definition

```
*MAT_ADD_COHESIVE
$       PID     ROFLG   INTFAIL     THICK
$         I         F         F         F
```

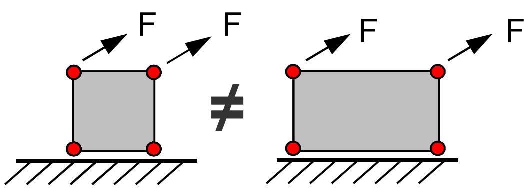
» density can be specified per unit volume or per unit area
→ handling of elements with an initial volume of zero

» number of integration points required for the element to be deleted can be specified

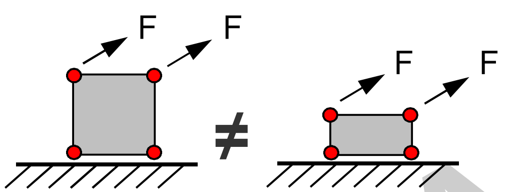
– differences in material behavior

Continuum formulation

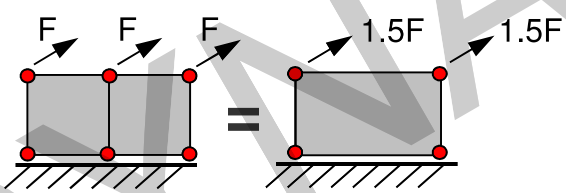
Constitutive relation is based on **stresses and strains**



A change in area of an element does matter



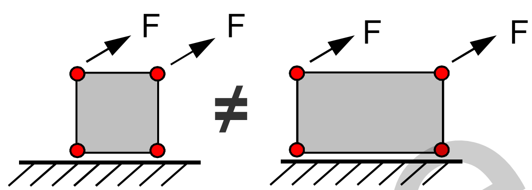
A change of height of an element does matter



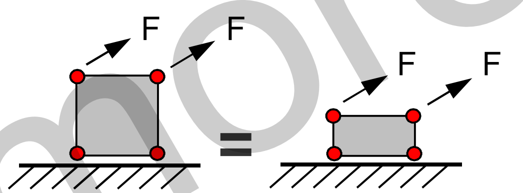
The increase of elements (same area) does not matter

Cohesive formulation in LS-DYNA

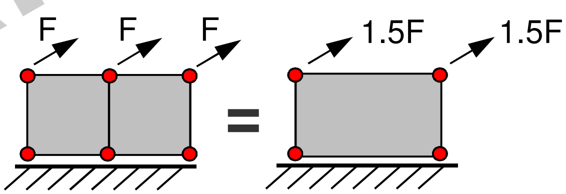
Constitutive relation is based on **stresses and relative displacements**



A change in area of an element does matter



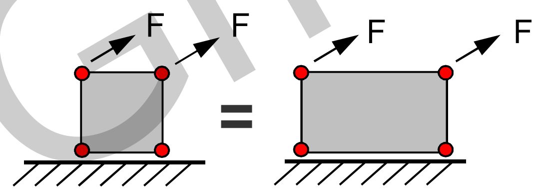
A change of height of an element does not matter



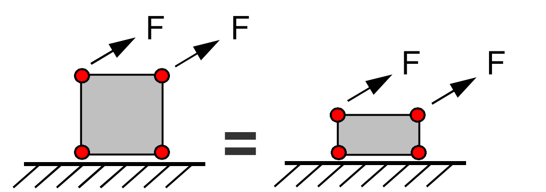
The increase of elements (same area) does not matter

True cohesive formulation/ discrete elements

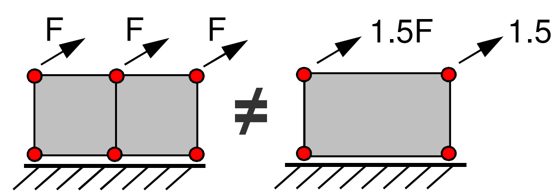
Constitutive relation is based on **forces and relative displacements**



A change in area of an element does not matter

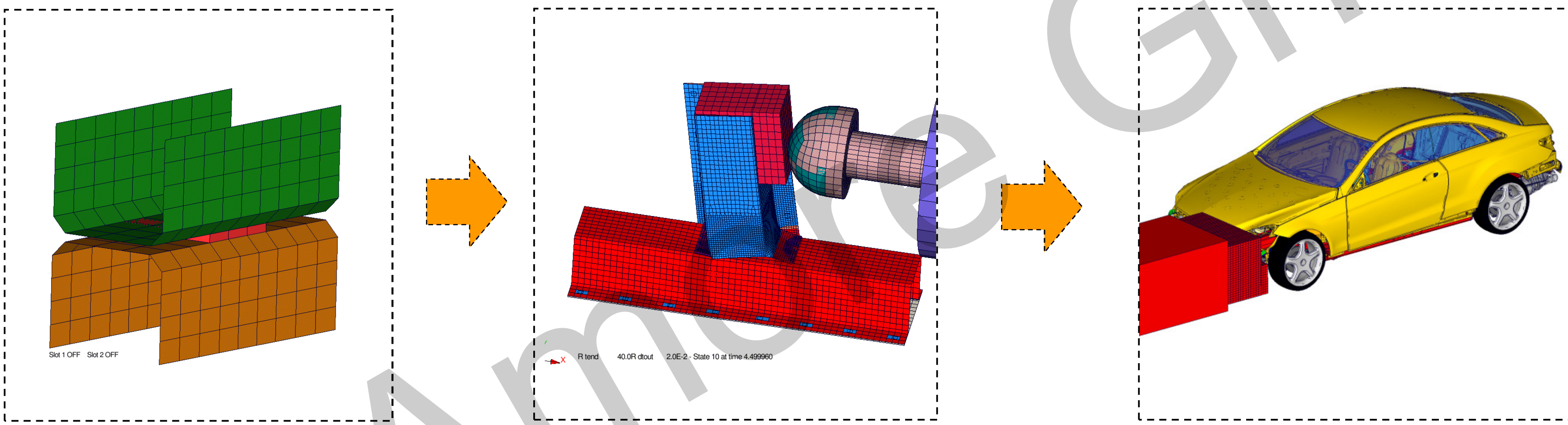


A change of height of an element does not matter

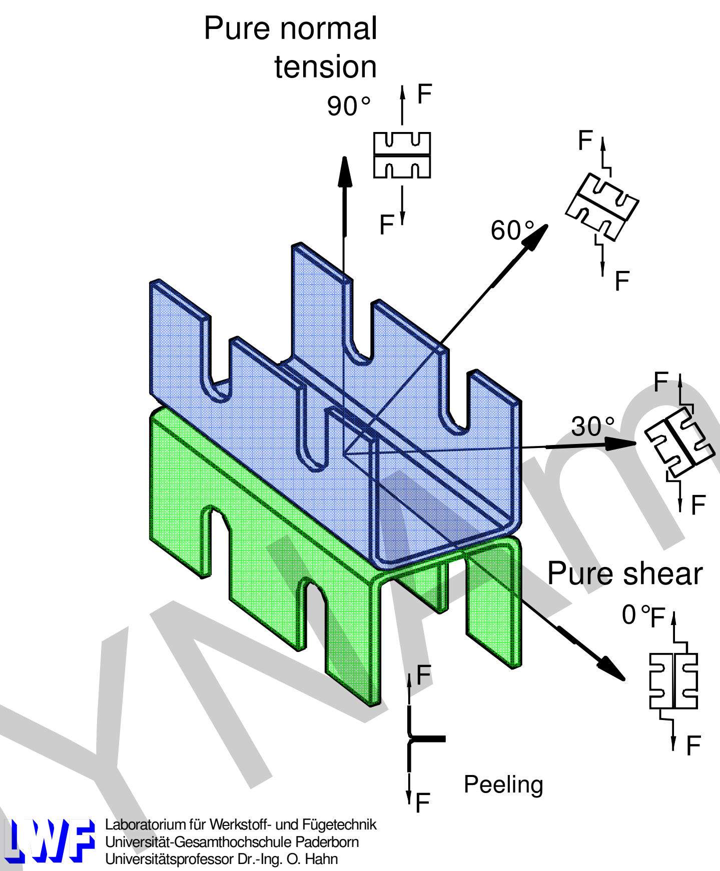


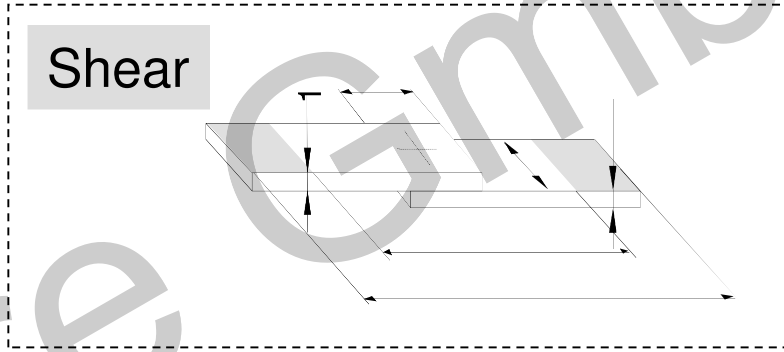
The increase of elements (same area) does matter

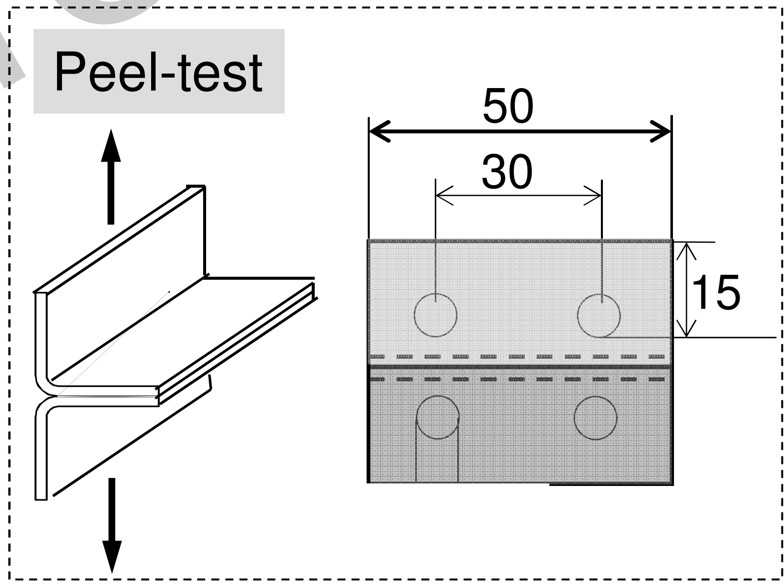
- **Verification and validation process**
  - **important:** The spatial discretization as well as every other definition in the LS-DYNA input file should be the same for all applications



- experimental investigations for validation
  - generally, fasteners are loaded by a combination of tension, compression, shear, bending and torsion
  - experimental validation is based on test specimens (e.g., LWF-KS2)
  - quasi-static and dynamic loading conditions



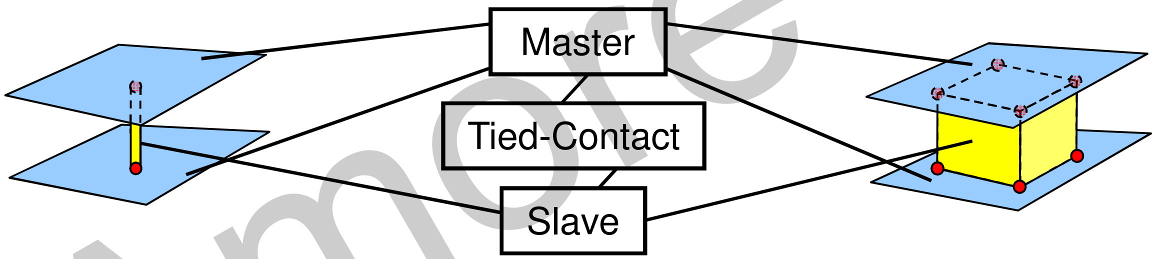




Spotweld modeling –
Modeling of punctiform connections

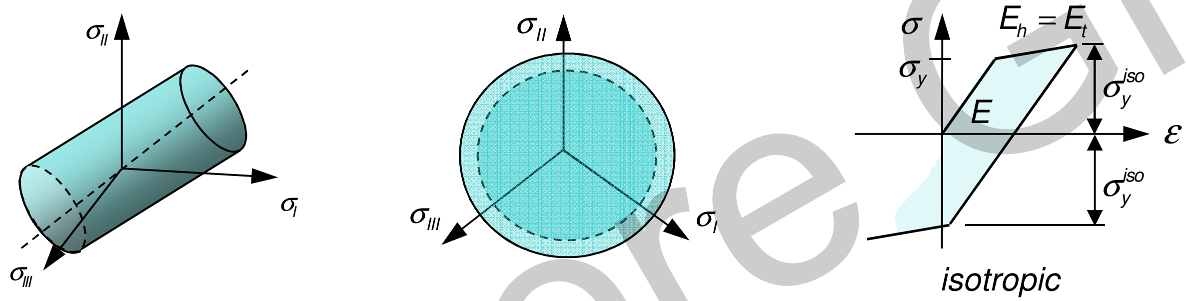
- **Spotweld modeling**
  - ***MAT_SPOTWELD_{DAMAGE-FAILURE}**
    - discretization techniques

definition with beams	definition with solids
– ELFORM=9 – 3x3 *Lobatto* integration (QR=4) – CST=1 (tubular section)	– ELFORM=1 – hourglass stabilization IHQ=6



- flange connection by *CONTACT_TIED_SHELL_EDGE_TO_SURFACE (equiv. to *CONTACT_SPOTWELD)
  SLAVE: Spotweld hexahedron, spotweld beam
  MASTER: Flange elements
  Recommendation: Put all elements in one single tied contact
- post-processing with SWFORC

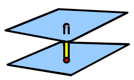
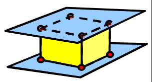
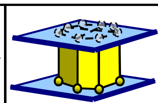
- material model
  - – bilinear elastoplastic *von Mises* material law
  - – isotropic hardening
  - – always try to use real material parameters to circumvent instabilities!!! Adjust the elasticity modulus to get an acceptable time step.



- *MAT_SPOTWELD – failure criterion

$$\left(\frac{\max(N_{rr},0)}{N_{rr_F}}\right)^2+\left(\frac{N_{rs}}{N_{rs_F}}\right)^2+\left(\frac{N_{rt}}{N_{rt_F}}\right)^2+\left(\frac{M_{rr}}{M_{rr_F}}\right)^2+\left(\frac{M_{ss}}{M_{ss_F}}\right)^2+\left(\frac{M_{tt}}{M_{tt_F}}\right)^2-1>0$$

- *MAT_SPOTWELD_DAMAGE-FAILURE
  - several failure criteria are available and can be specified via variable OPT (CARD 3)

OPT				failure criterion (FC)
0	✓	✓	✓	resultant based, forces & moments of failure is written
-1	✓	✓	✓	like OPT=0, FC is comp. and written, element is not deleted
-2	✓	✓	✓	like OPT=-1, but peak value of FC and exact time is written
1	✓	✓	✓	stress resultant based (TOYOTA), no individual bending term
2	✓	✗	✓	user defined spotweld failure
3	✓	✗	✓	notch stress based failure
4	✓	✗	✓	structural stress intensity factor based failure
5	✓	✗	✓	maximum structural stress based failure
6	✓	✗	✓	like OPT=1, LS-DYNA looks for failure parameter for each beam node, FC is evaluated for each beam node independently, failure parameters have to be defined for each connected part, strain rate effects can be defined (TOYOTA)
7	✗	✓	✗	stress resultant based, LS-DYNA looks for fail. param. dep. on flange partners, failure parameters have to be defined for every flange connection combination, strain rate effects can be defined, definition of default values possible
9	✓	✗	✗	like OPT=6 but failure is split into nugget pull out and fracture failure (TOYOTA)
-9	✓	✗	✗	like OPT=9, FC is comp. and written, element is not deleted
11	✓	✗	✗	resultant based failure, load curves or table can be defined as result. fail. forces vs. load. direction or result. fail. forces vs. load. direction vs. strain rate

R7.1.1

- *MAT_SPOTWELD_DAMAGE-FAILURE
  - new failure model OPT=11 for beam elements, where failure depends on loading direction via curves

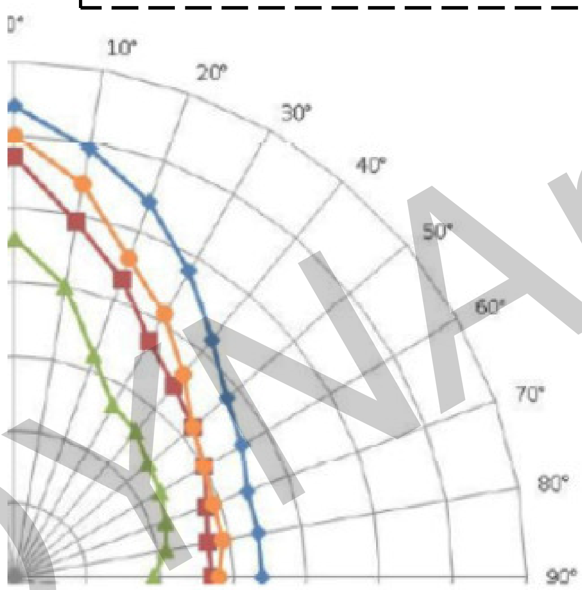
R7.1.1

OPT = 11 invokes a resultant force based failure criterion for beams. With corresponding load curves or tables LCT and LCC, resultant force at failure F_{fail} can be defined as function of loading direction γ (curve) or loading direction γ and effective strain rate $\dot{\varepsilon}$ (table):

$$F_{\mathrm{fail}} = f(\gamma) \quad \text{or} \quad F_{\mathrm{fail}} = f(\gamma, \dot{\varepsilon})$$

with the following definitions for loading direction (in degree) and effective strain rate:

$$\gamma = \tan^{-1}\left(\left|\frac{F_{\mathrm{shear}}}{F_{\mathrm{axial}}}\right|\right), \quad \dot{\varepsilon} = \left[\frac{2}{3}\left(\dot{\varepsilon}_{\mathrm{axial}}^2 + \dot{\varepsilon}_{\mathrm{shear}}^2\right)\right]^{1/2}$$

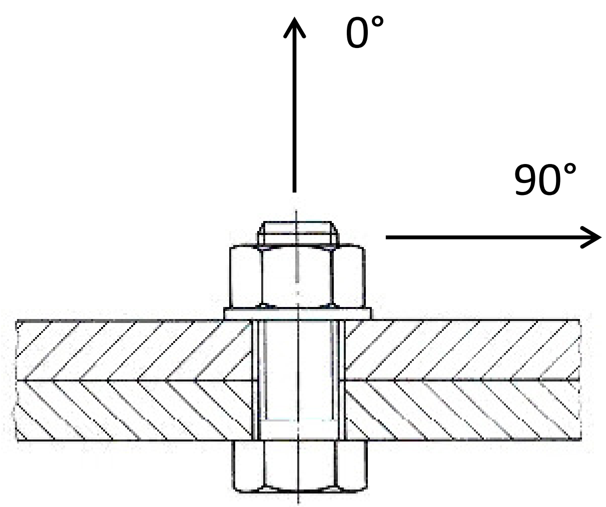


ISO thread

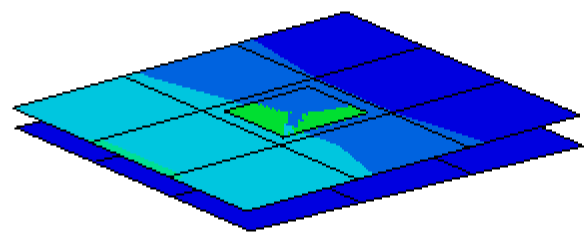
round thread

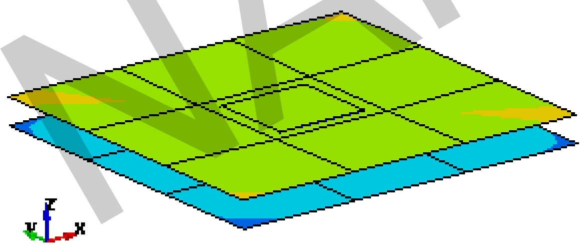
trapezoidal thread

buttress thread

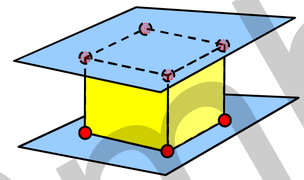


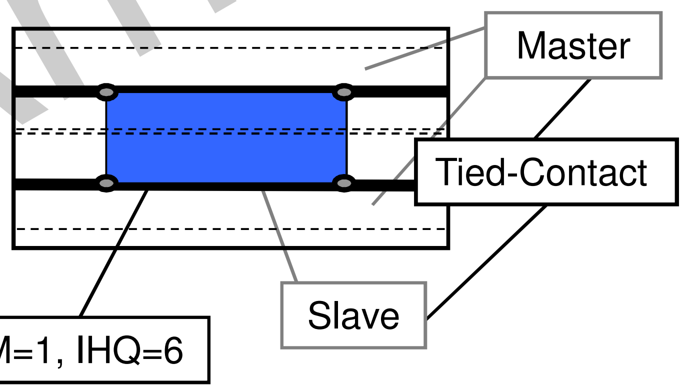
- numerical example – *MAT_SPOTWELD
  – run spotweld_user.k and take a look on the simulation results
  – define *CONTACT_SPOTWELD to tie the nodes of the hexahedron elements on the shell surfaces
  – define *MAT_SPOTWELD with the following properties:
    RHO=7.85E-6, E=5.0, PR=0.3, SIGY=0.25, ET=0.6
  – define NRR=5.0, NRS=NRT=3.0 and check failure in the message-file
  – define plastic failure strain via EFAIL=0.15
  – define an additional output for the SWFORC-file with a time interval of DT=1.0E-2

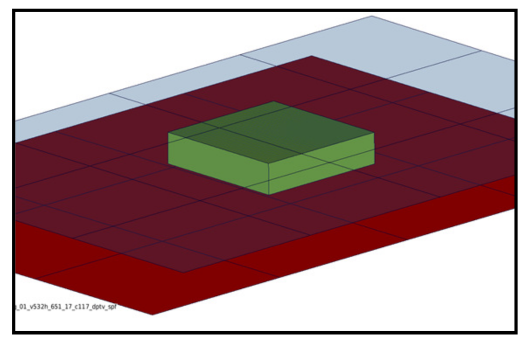




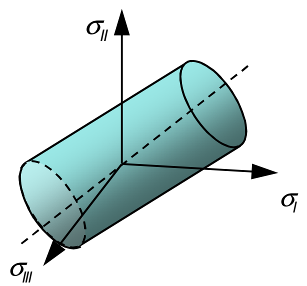
- ***MAT_SPOTWELD_DAIMLER**
  - can only be used with underintegrated solid elements (ELFORM=1, constant stress) combined with hourglass stabilization IHQ=6.
    Remark: If beam elements are defined, use the cluster option to generate solid elements automatically in LS-DYNA
  - flange connection by *CONTACT_TIED_SHELL_EDGE_TO_SURFACE (equiv. *CONTACT_SPOTWELD)
    SLAVE: Spotweld hexahedron,
    MASTER: Flange elements
    Recommendation: Put all elements in one single tied contact
  - post-processing with SWFORC- and DCFAIL-file
  - techniques

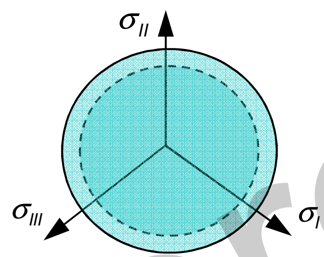


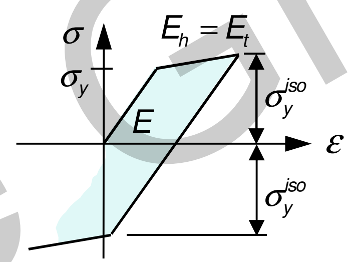




- material model
  - bilinear elastoplastic *von Mises* material law
  - isotropic hardening
  - always try to use real material parameters to circumvent instabilities!!! Adjust the elasticity modulus to get an acceptable time step.





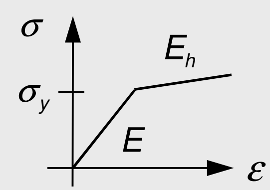


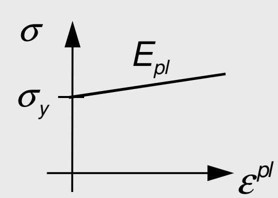
isotropic

» tangent modulus E_h
→ relation between stress and total strain

» Plastic modulus E_{pl}
→ relation between stress and plastic strain

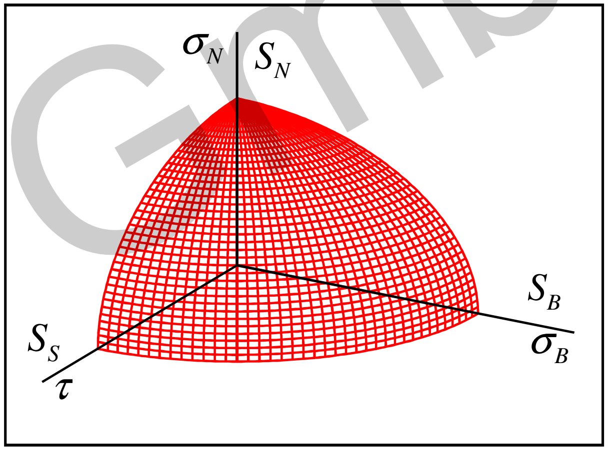
$$E_{pl} = \frac{E\,E_h}{E - E_h}$$





- failure model
  - the failure model consists of three terms that take normal stresses, shear stresses and stresses due to bending into account:

$$\sigma_n = \frac{N_{rr}}{A} \quad \sigma_b = \frac{\sqrt{M_{rs}^2 + M_{rt}^2}}{Z} \quad \tau = \frac{M_{rr}}{2Z} + \frac{\sqrt{N_{rs}^2 + N_{rt}^2}}{A}$$



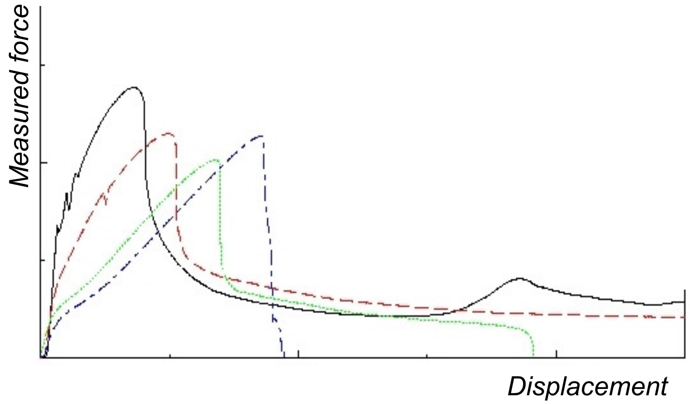
  - the failure criterion reads as follows:

$$f = \left(\frac{\sigma_n}{s_n^F}\right)^{m_n} + \left(\frac{\sigma_b}{s_b^F}\right)^{m_b} + \left(\frac{\tau}{s_s^F}\right)^{m_\tau} - 1$$

where the inner moment lever is defined as

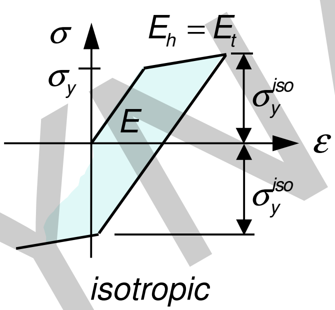
$$Z = \pi \frac{d^3}{32}$$

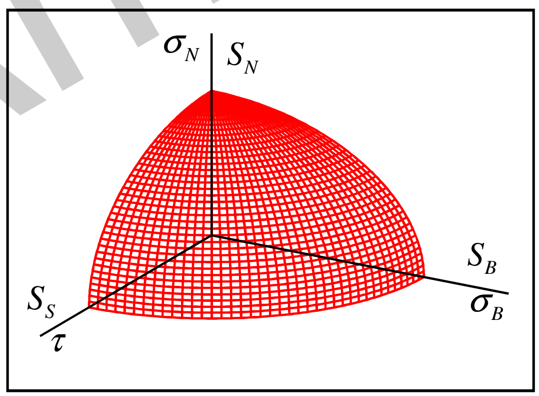
Here *d* is the equivalent diameter of the solid spot weld element assuming a circular cross section.



- keyword definition

```
*MAT_SPOTWELD_DAIMLER
$      MID        RO         E        PR                                DT     TFAIL
        10   7.85E-6      10.0       0.3
$    EFAIL                                                                        NF
                                                                                30.0
$       RS      ASFF               TRUE_T    CON_ID
       1.0                        1.0e-10         1
$-------------------------------------------------------------------------------
*DEFINE_CONNECTION_PROPERTIES
$   CON_ID  PROPRULE    AREAEQ                        DG_TYP    MOARFL
         1         1         1                             1
$             D_SIGY    D_ETAN   D_DG_PR    D_RANK      D_SN      D_SB      D_SS
              0.3124     1.400     0.001       0.1 1.000E+20 1.000E+20 1.000E+20
$   D_EXSN    D_EXSB     D_EXS    D_LCSN    D_LCSB    D_LCSS    D_GFAD  D_SCLMRR
    2.E+00    2.E+00    2.E+00      1002      1002      1002
```



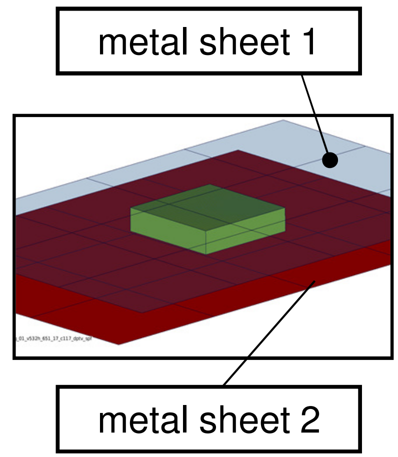


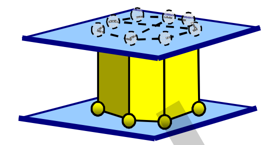
$$f = \left(\frac{\sigma_n}{s_n^F}\right)^{m_n} + \left(\frac{\sigma_b}{s_b^F}\right)^{m_b} + \left(\frac{\tau}{s_s^F}\right)^{m_\tau} - 1$$

– splitting of the keyword using the ADD-option

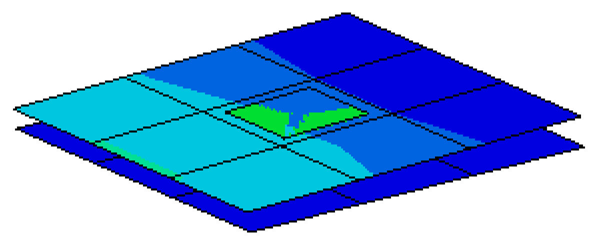
```
*DEFINE_CONNECTION_PROPERTIES
$   CON_ID  PROPRULE    AREAEQ              DG_TYP    MOARFL
         1         1         I                   I         I
$             D_SIGY    D_ETAN   D_DG_PR    D_RANK      D_SN      D_SB      D_SS
                   F         F         F       0.3         F         F         F
$   D_EXSN    D_EXSB     D_EXS    D_LCSN    D_LCSB    D_LCSS    D_GFAD  D_SCLMRR
         F         F         F         I         I         I         F         F
*MAT_ELASITC
$      MID
      1000
*DEFINE_CONNECTION_PROPERTIES_ADD
$   CON_ID
         1
$      MID      SIGY      ETAN     DG_PR      RANK        SN        SB        SS
      1000         F         F         F       0.5         F         F         F
$     EXSN      EXSB       EXS    D_LCSN    D_LCSB    D_LCSS    D_GFAD  D_SCLMRR
         F         F         F         I         I         I         F         F
*MAT_ELASITC
$      MID
      2000
*DEFINE_CONNECTION_PROPERTIES_ADD
$   CON_ID
         1
$      MID      SIGY      ETAN     DG_PR      RANK        SN        SB        SS
      2000         F         F         F       0.2         F         F         F
$     EXSN      EXSB       EXS    D_LCSN    D_LCSB    D_LCSS    D_GFAD  D_SCLMRR
         F         F         F         I         I         I         F         F
```

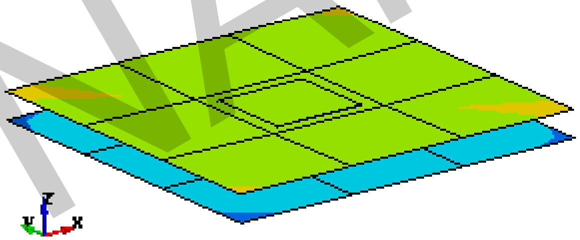
default values





- numerical example – *MAT_SPOTWELD_DAIMLER
  - run spotweld_daimler_user.k and take a look on the simulation results
  - define *CONTACT_SPOTWELD to tie the nodes of the hexahedron elements on the shell surfaces
  - define *MAT_SPOTWELD_DAIMLER with the following properties: RHO=7.85E-6, E=5.0, PR=0.3, SIGY=0.25, ET=0.6
  - activate the failure criterion with the following properties: DSN=DSB=0.2, DSS=0.3, DEXSN=DEXSB=DEXSS=1.0
  - define an additional damage behavior using DGTYP=4 and DGFAD=0.08
  - define an additional output for the DCFAIL-file with a time interval of DT=1.0E-2



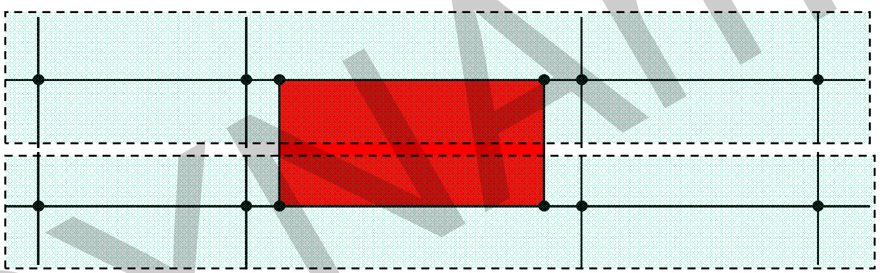


• SPOTHIN – Contact thickness scale factor

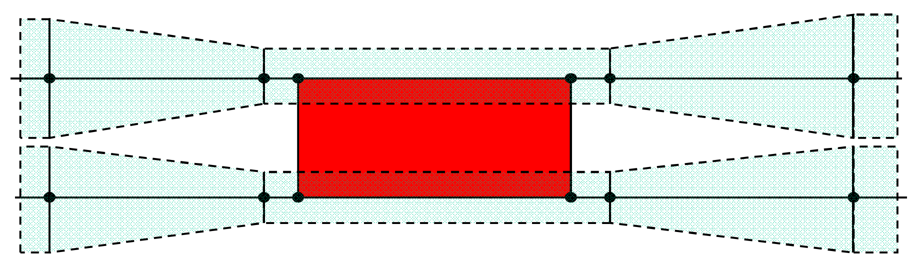
- SPOTHIN is only applied to *CONTACT_AUTOMATIC_SINGLE_SURFACE
- only shell elements with a connected slave node of solid spotweld will be scaled by the contact thickness scale factor SPOTHIN.

```
*CONTROL_CONTACT
$   SLSFAC    RWPNAL    ISLCHK    SHLTHK    PENOPT    THKCHG     ORIEN    ENMASS
       0.1       0.0         2         0         1         0         2         0
$   USRSTR    USRFRC     NSBCS    INTERM     XPENE     SSTHK      ECDT   TIEDPRJ
         0         0         0         0       4.0         0         0         0
$    SFRIC     DFRIC       EDC       VFC        TH     TH_SF    PEN_SF
       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$   IGNORE    FRCENG   SKIPRWG    OUTSEG   SPOTSTP   SPOTDEL   SPOTHIN
         2         0         0         0         0         1       0.5
```

without contact thickness scale factor



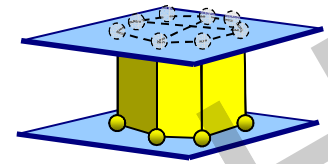
with contact thickness scale factor

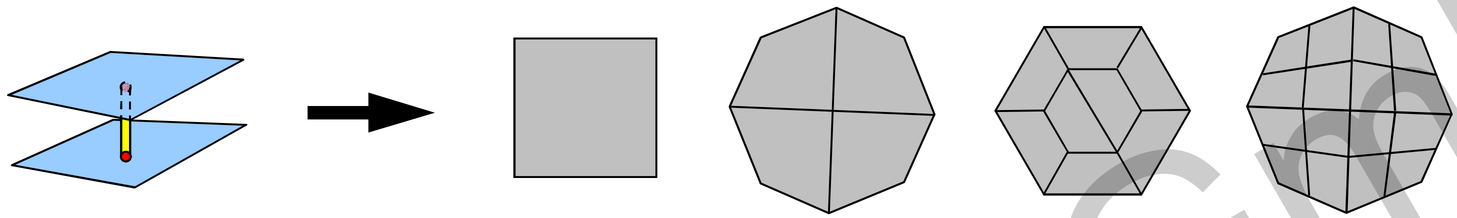


→ Artificially created parasitic contact forces will not be generated and thus, failure of the spotweld at a too early stage is circumvented!

▪ Spotweld cluster

- automatic generation of spotweld clusters from beam elements



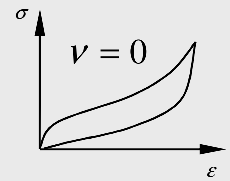
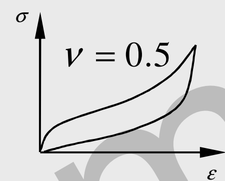
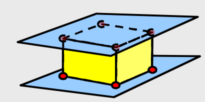
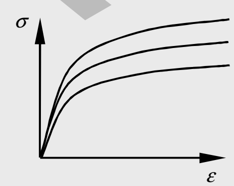



```
*CONTROL_SPOTWELD_BEAM
$       LCT       LCS     T_ORT    PRTFLG     T_ORS     RPBHX     BMSID    ID_OFF
        0.1       0.0         0         2         0         8
```

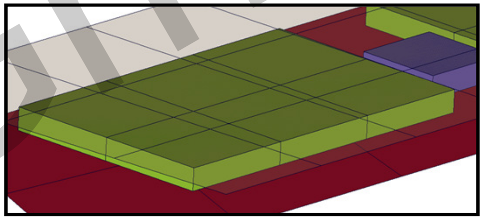
- RPBHX: Replace each spot weld beam element with a cluster of RPBHX solid elements. RPBHX may be set to 1, 4, 8
- BMSID: Optional beam set ID defining the beam element ID's that are to be converted to hex assemblies
- ID_OFF: Part ID of generated hex assemblies equals original part ID plus ID_OFF
- if RPBHX=4/8, the keyword *DEFINE_SPOTWELD_ASSEMBLY is created automatically, which allows the definition of output data to the SWFORC-file. The ID of the parent beam is used subsequently for the solid cluster.
- the beam element is automatically deleted from the model

Adhesives modeling –
modeling of line and area shaped con.

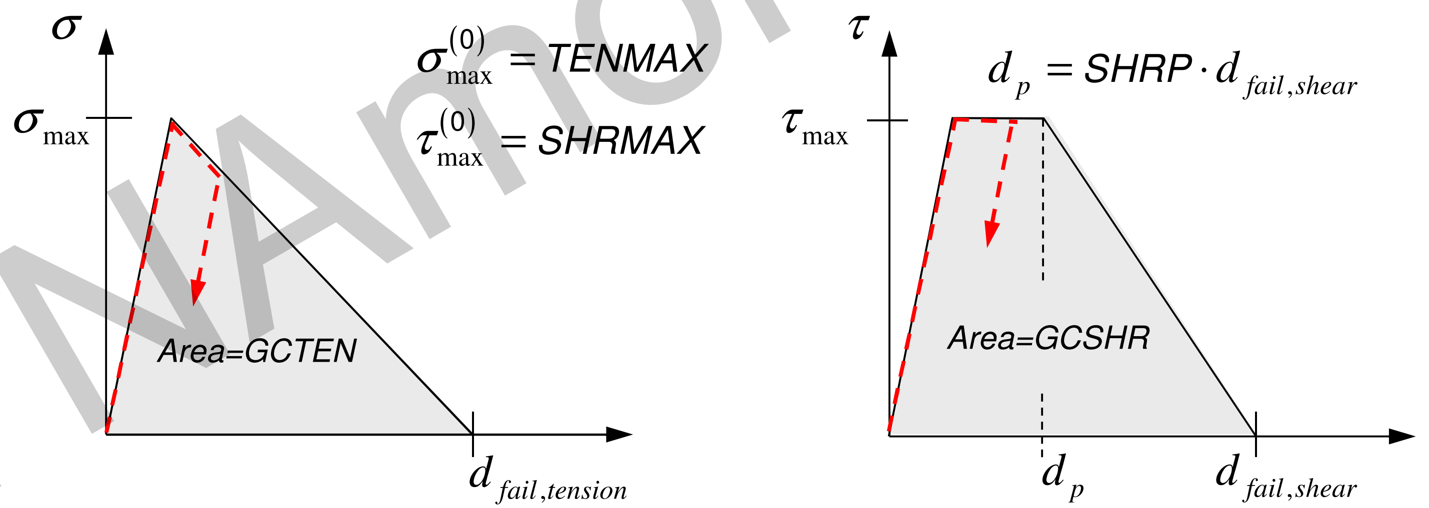
- **Adhesives modeling**
  - **Mechanical behavior of bonding materials**

Material behavior		Choice of material model
▪ hood adhesive → foam-like, → viscoelastic		▪ hyperelastic and foam-like, strain rate dependent → ***MAT_FU_CHANG_FOAM**
▪ PU windshield → rubber-like, → viscoelastic		▪ hyperelastic and rubber-like, strain rate dependent → *MAT_SIMPLIFIED_RUBBER_{}, ν=0.499 (*OGDEN*) → using substitute model with one element attached with tied contact results in ν=0 for the element → extremely high in plane stress values → use ***MAT_FU_CHANG_FOAM** 
▪ structural adhesive → elasto-viscoplastic		▪ elastoplastic, strain rate dependent, failure → ***MAT_ARUP_ADHESIVE**

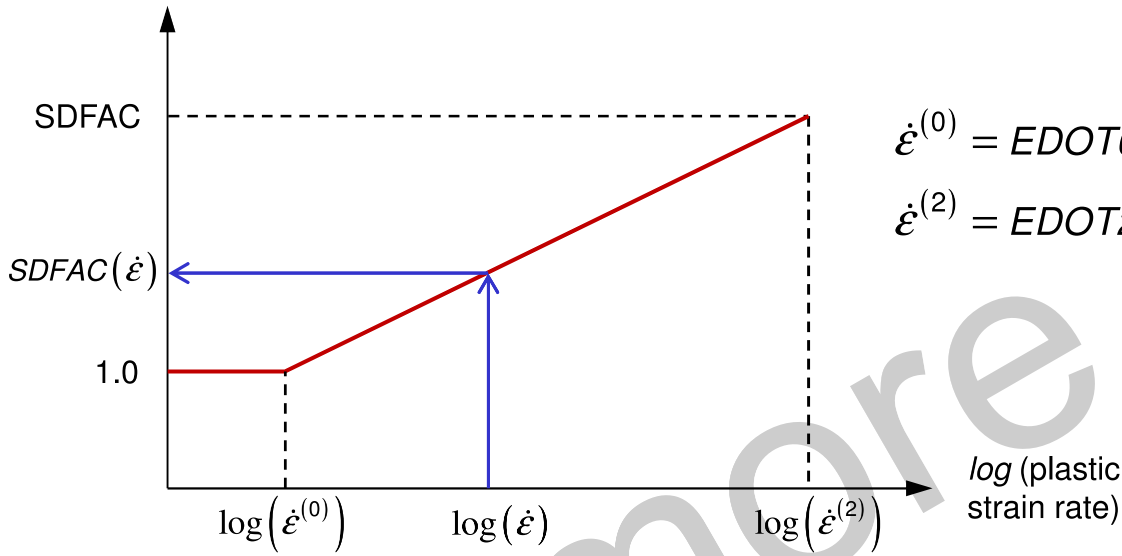
- ***MAT_ARUP_ADHESIVE (*MAT_169)**
  - works for solid element types 1, 2 and 15
  - material behavior can be defined individually for the normal and shear direction
    → correct definition of the thickness direction is extremely important
  - thickness direction can be defined via smallest element side or node numbering
  - bond thickness can be defined individually
    → material behavior independent of element height
  - elastoplastic material model with the following yield surface:



$$\left(\frac{\sigma(\dot{\varepsilon})}{\sigma_{max}(\dot{\varepsilon})}\right)^{PWRT}+\left(\frac{\tau(\dot{\varepsilon})}{\tau_{max}(\dot{\varepsilon})-SHL\_SL\cdot\sigma(\dot{\varepsilon})}\right)^{PWRS}-1.0=0$$



- possibility to account for strain rate effects:
  → strengths and fracture energies are scaled linearly in the log-scale of the plastic strain rate



$\dot{\varepsilon}^{(0)} = EDOT0$

$\dot{\varepsilon}^{(2)} = EDOT2$

$$\sigma_{max}(\dot{\varepsilon}) = SDFAC(\dot{\varepsilon}) \cdot \sigma_{max}^{(0)}$$

$$\tau_{max}(\dot{\varepsilon}) = SDFAC(\dot{\varepsilon}) \cdot \tau_{max}^{(0)}$$

$$GCTEN(\dot{\varepsilon}) = SGFAC(\dot{\varepsilon}) \cdot GCTEN$$

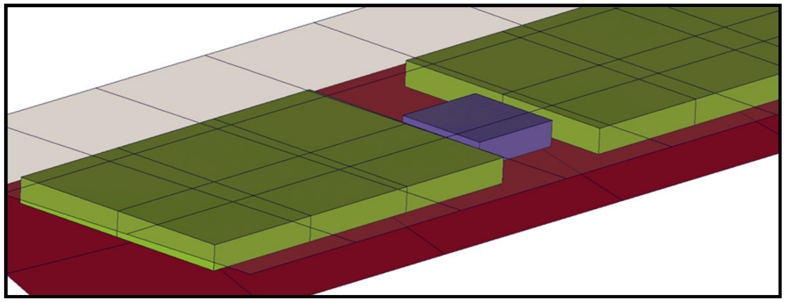
$$GCSHR(\dot{\varepsilon}) = SGFAC(\dot{\varepsilon}) \cdot GCSHR$$

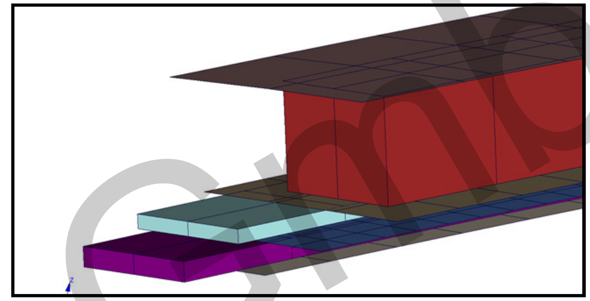
- additionally output can be activated via OUTFAIL

```
1808 t 9.9979E-01 dt 5.53E-04 write d3plot file             04/11/14 13:56:15
    mat_arup element 13 (IP= 1) had damage initiated at time 1.0468E+00
      axial term of failure function ... 0.00000
      shear term of failure function ... 1.00044
      resultant axial force ............ 2.9562E-04
      resultant shear force ............ 2.8676E+00
```

□ thickness direction can be defined via

smallest element side (default) or node numbering (THKDIR)





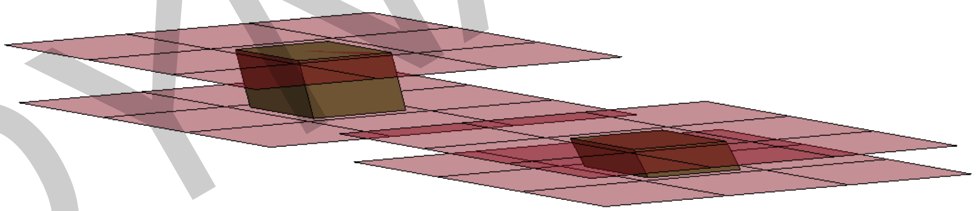
future work: Thickness direction is calculated based on tied contact information

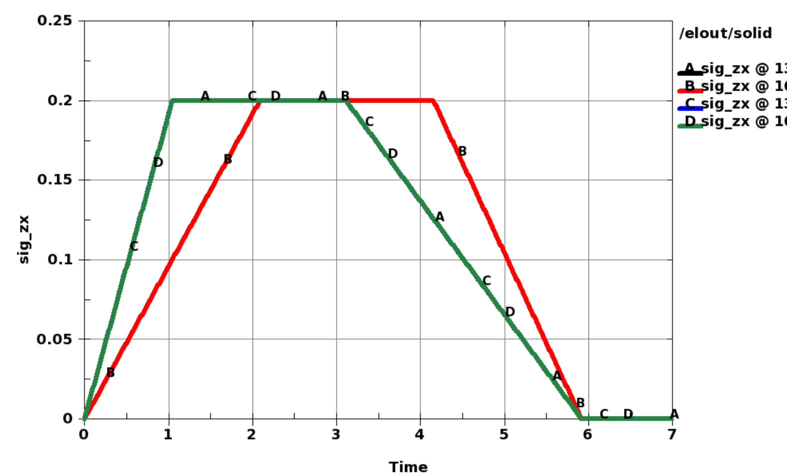
□ bond thickness can be defined individually (BTHK)

→ material behavior independent of element height

→ reduce errors due to an incorrect spatial discretization in the full car crash model

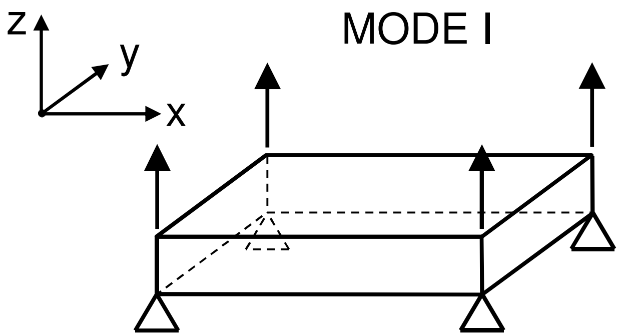
→ negative value: BTHK is bond thickness
but critical time step is not affected
(can affect stability!!!)





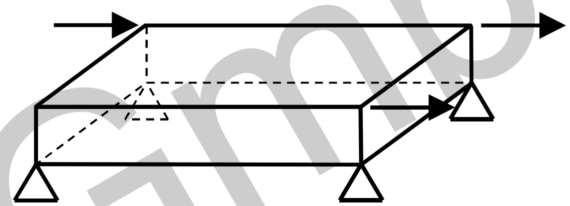
- 1-Element-Test (‘SOLID’, ELFORM=1)

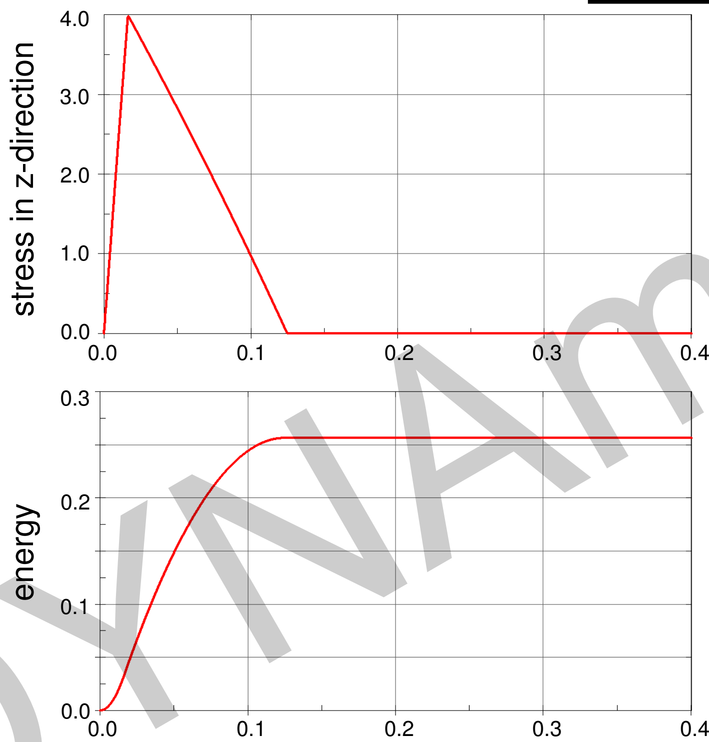
MODE I

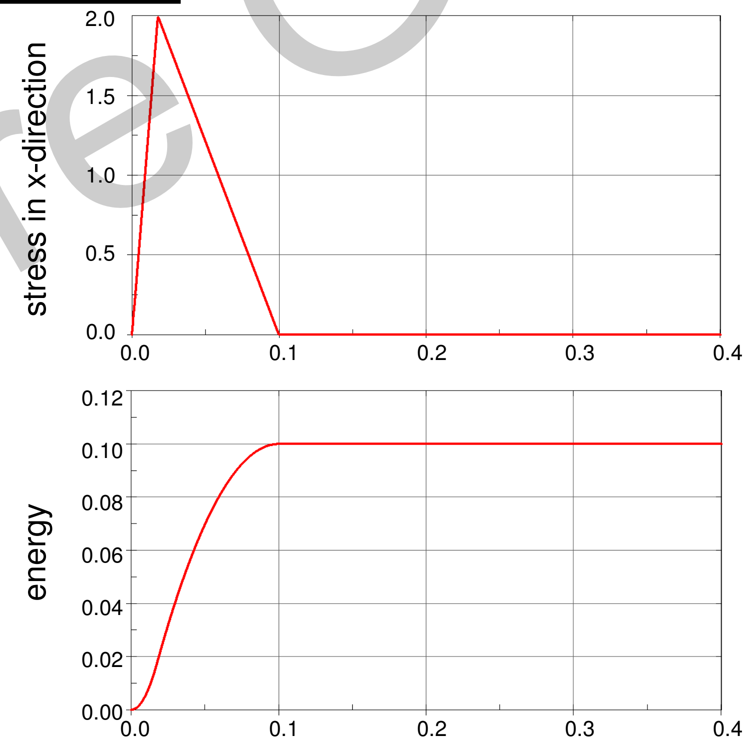


Material Parameter:
E= 50.0 [kN/mm²]
GCTEN = 0.25 [kNmm]
GCSHR = 0.10 [kNmm]
TENMAX = 4.0 [kN/mm²]
SHRMAX = 2.0 [kN/mm²]

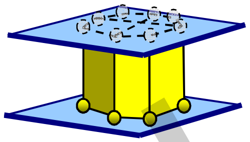
MODE II

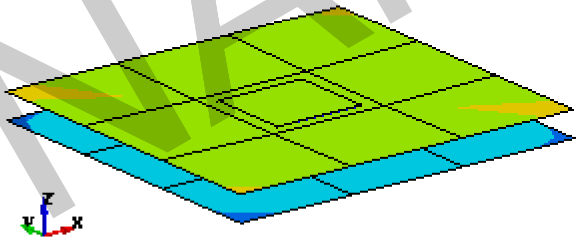


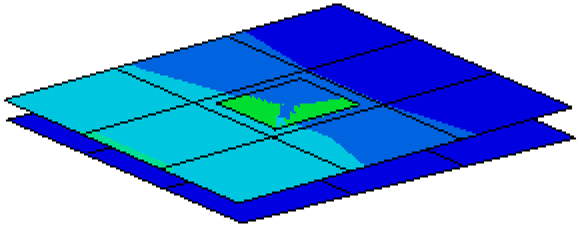




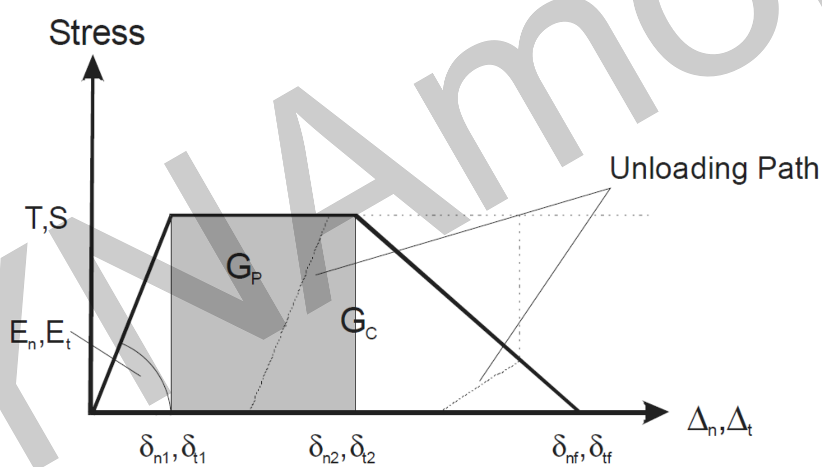
- numerical example – *MAT_ARUP_ADHESIVE
  - run arup_user.k and take a look on the simulation results
  - define *CONTACT_SPOTWELD to tie the nodes of the hexahedron elements on the shell surfaces
  - define *MAT_ARUP_ADHESIVE with the following properties: RHO=7.85E-6, E=5.0, PR=0.3, TENMAX=SHRMAX=0.2, GCTEN=GCSHR=0.08, PWRT=PWRS=2, SHRP=0.02
  - run the simulation and take a look on the results

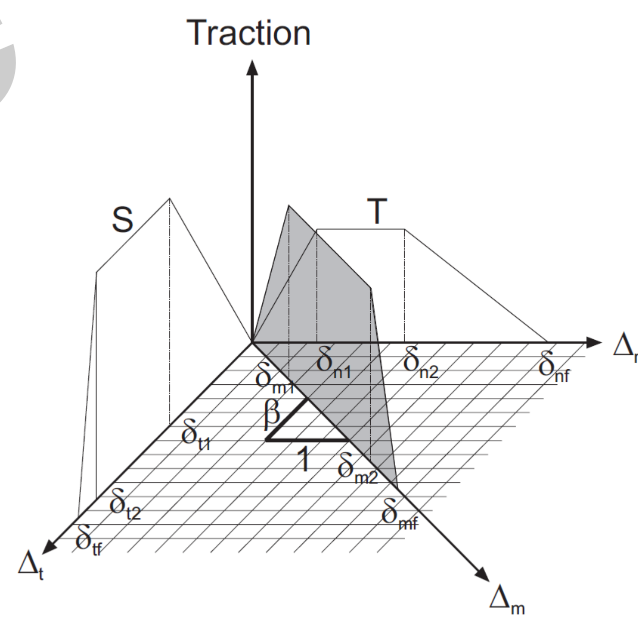




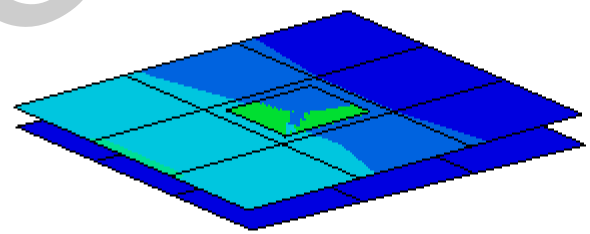


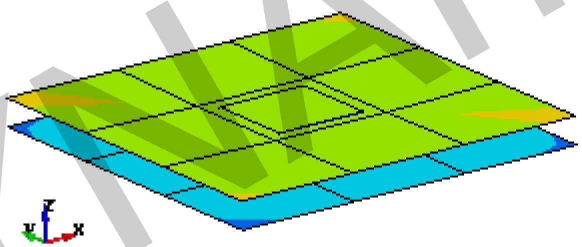
- ***MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE (*MAT_240)**
  - elastic-ideally plastic cohesive zone model with damage
  - rate-dependent
  - tri-linear traction-separation law
  - alternative for *MAT_ARUP_ADHESIVE
  - quadratic yield and damage criterion in mixed-mode loading
  - number of integration points required for the element to be deleted can be specified (INTFAIL)
  - maximum stress, total energy and a further factor describing the traction-separation law can be specified in tension as well as shear





- numerical example – *MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE
  - run mat_cohesive_mixed_mode_elastoplastic_rate.k and take a look on the simulation results
  - define *CONTACT_SPOTWELD to tie the nodes of the hexahedron elements on the shell surfaces
  - define *MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE with the following properties:
    RO=7.85E-6, EMOD=GMOD=1.0, THICK=1.0, T0=S0=0.2, G1C_0=G2C_0=0.08,
  - run the simulation and take a look on the results

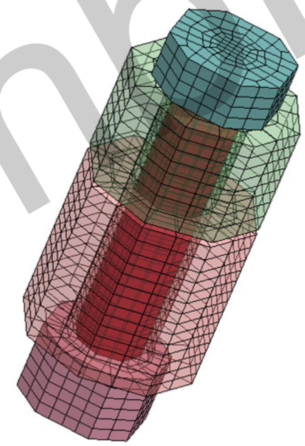


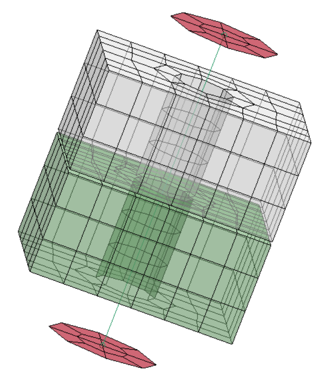


BOLTED CONNECTIONS

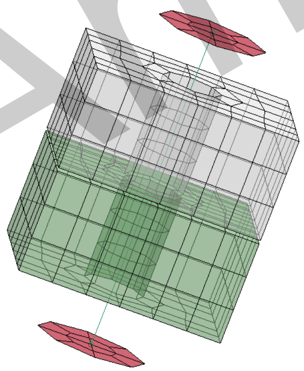
- **Bolted connections**
  - Preliminary remarks
    - □ bolt shaft can be modeled with beam or solid elements
    - □ bolt head and nut can be modeled using shell or solid elements

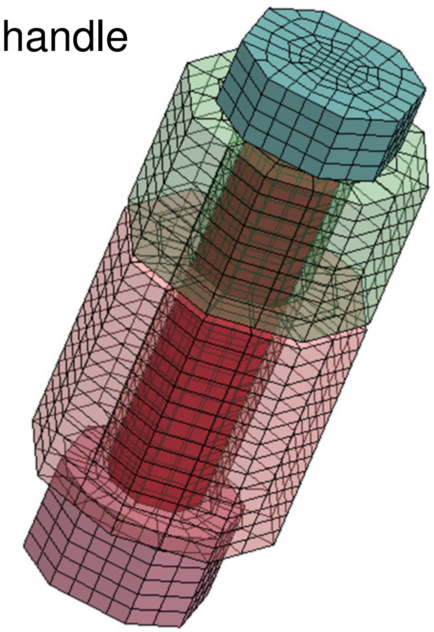
      remark: Keep in mind that shell elements have no rotational d.o.f. around the shell normal, if the shaft is discretized with beams and the nut as well as the bolt head with shell elements
      → no torsional stiffness of the bolt
    - □ pre-stressing can be taken into account
      - – beam element: *INITIAL_AXIAL_FORCE_BEAM
      - – solid element: *INITIAL_STRESS_SECTION
      - – material model via temperature
    - □ connected parts can be modeled using shell or solid elements
      - – **remark:** Standard shell elements assume plane stress state. No stress can arise in normal direction.
    - □ failure can be activated in the respective material model as well as using *MAT_ADD_EROSION
      - – available for 3-d volume and shell elements
      - – only erosion is currently supported for beam elements 1 and 11





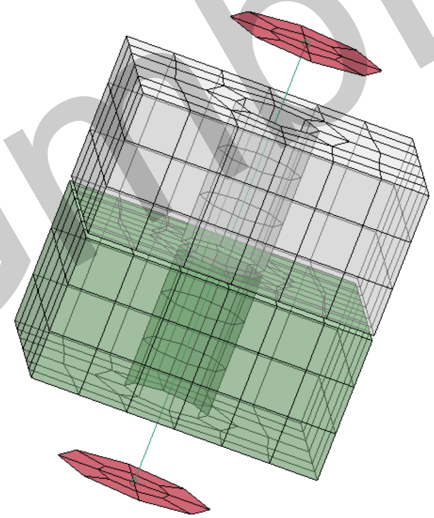
- increase of additional mass can be handed using selective mass scaling (*CONTROL_TIMESTEP, IMSCL)
- use *CONTACT_AUTOMATIC_SINGLE_SURFACE for contact between connected parts, nut and bolt head
- shaft discretized with beam elements:
  - use *CONTACT_AUTOMATIC_GENERAL and null beams (*MAT_SPOTWELD beams are not considered by default) to handle contact between bolt shaft and connected parts
- shaft discretized with solid elements:
  - use *CONTACT_AUTOMATIC_SINGLE_SURFACE to handle contact between bolt shaft and connected parts
- *Coulomb* friction can be defined in the contact keyword





- Pre-stressing for beam elements – *INITIAL_AXIAL_FORCE_BEAM
  - Purpose: Initialize the axial force resultants in beam elements that are used to model bolts.
  - This option works with *MAT_SPOTWELD with beam type 9

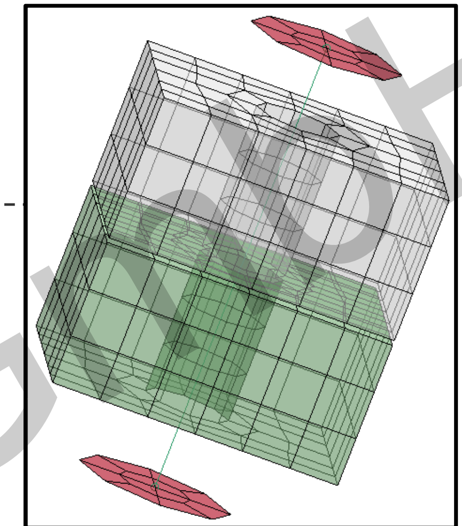
```
*INITIAL_AXIAL_FORCE_BEAM
$         I         I         F
$      BSID      LCID     SCALE
          1         1         2
```

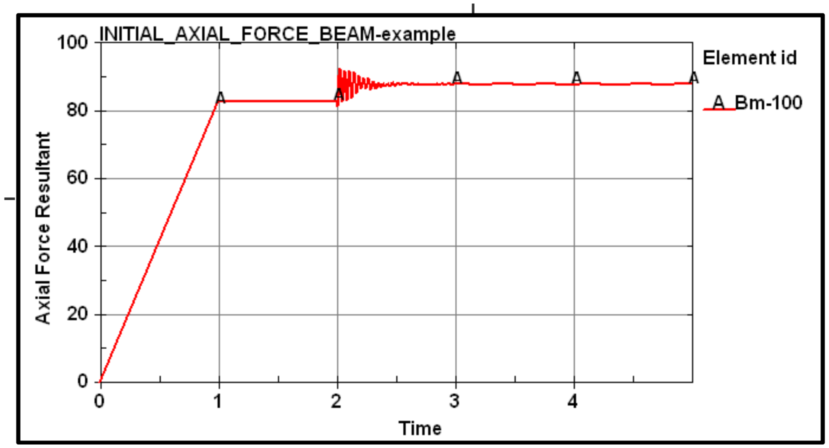


  - BSID: Beam set ID
  - LCID: Load curve ID defining preload force versus time. When the load curve ends or goes to zero, the initialization is assumed to be completed.
  - SCALE: Scale factor on load curve
  - remarks:
    - use contact damping with VDC={0.1-0.2} and/or *DAMPING_PART_STIFFNESS with COEF=0.1
    - always use a smooth ramp in the load curve starting at the origin to apply the load
    - when the end of the load curve is reached or the value of the load decreases form the maximum value, the initialization stops.

- numerical example
  - file: initial_axial_force_beam.key
  - keyword definition

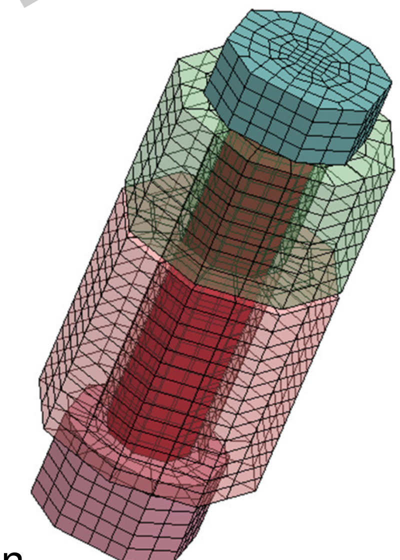
```
*INITIAL_AXIAL_FORCE_BEAM
$     BSID      LCID     SCALE
        99        98       0.0
*SET_BEAM
$      SID
        99
$      ID1
       100
*DEFINE_CURVE
$    LABEL      SIDR       SFA       SFO      OFFA      OFFO    DATTYP
        98         0       0.1       1.0       0.0       0.0         0
$                XVALS               YVALS
                   0.0                 0.0
             10.000000           83.000000
             20.000000           83.000000
```





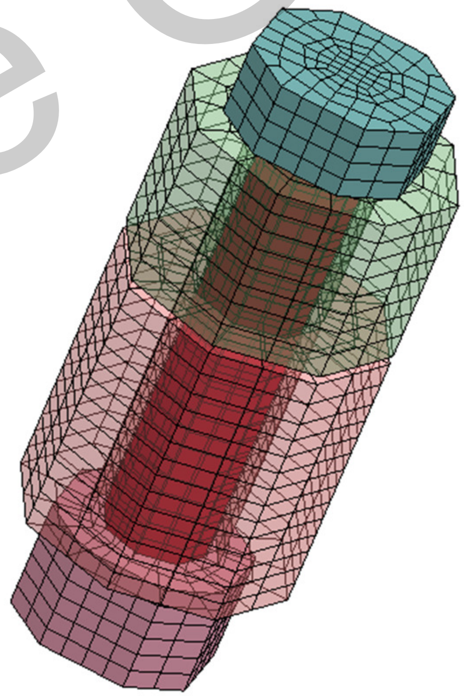
- Pre-stressing for solid elements – *INITIAL_STRESS_SECTION
  - initialize the stress in solid elements that are part of a section definition to create a preload.
  - stress component in the direction normal to the cross-section plane is initialized.
  - option works with a subset of materials that are incrementally updated.
  - rubbers, foams, and materials that are combined with equations-of-state cannot be initialized by this approach.
  - NEW: Hyperelastic materials # 57, 73 and 83 can be initialized with this approach.

```
*INITIAL_STRESS_SECTION
$         I         A8          I          I          I
$     ISSID       CSID       LCID       PSID        VID
          1          1          2
```



- ISSID: Section stress initialization ID
- CSID: Cross-section ID
- LCID: Load curve defining preload stress versus time
- PSID: Part set ID
- VID: Vector ID defining normal direction of the cross section.
  Must be defined if *DATABASE_CROSS_SECTION_SET is used

- remarks:
  - use contact damping with VDC={0.1-0.2} and/or *DAMPING_PART_STIFFNESS with COEF=0.1
  - always use a smooth ramp in the load curve starting at the origin to apply the load
  - when the end of the load curve is reached or the value of the load decreases form the maximum value, the initialization stops.
  - solid elements types 1, 2, 3, 4, 9, 10, 13, 15, 16, 17 and 18 are supported
  - ALE elements are not supported



- numerical example
  - file: initial_stress_section.key
  - keyword definition

```
*INITIAL_STRESS_SECTION
$     ISSID       CSID       LCID       PSID        VID
          1         99         98         97          0
*SET_PART
$       PID
         97
$       ID1
          1
*DATABASE_CROSS_SECTION_PLANE_ID
$     label       title
         99
$      PSID        XCT        YCT        ZCT        XCH        YCH        ZCH
         97       17.0       17.0       44.5       17.0       17.0       45.0
$      XHEV       YHEV       ZHEV       LENL       LENM         ID      ITYPE
      -17.0       17.0       44.5        0.0        0.0          0          0
*DEFINE_CURVE
$     LABEL       SIDR        SFA        SFO       OFFA       OFFO     DATTYP
         98          0        0.1        1.0        0.0        0.0          0
$                XVALS                 YVALS
                   0.0                   0.0
             10.000000             83.000000
             20.000000             83.000000
```