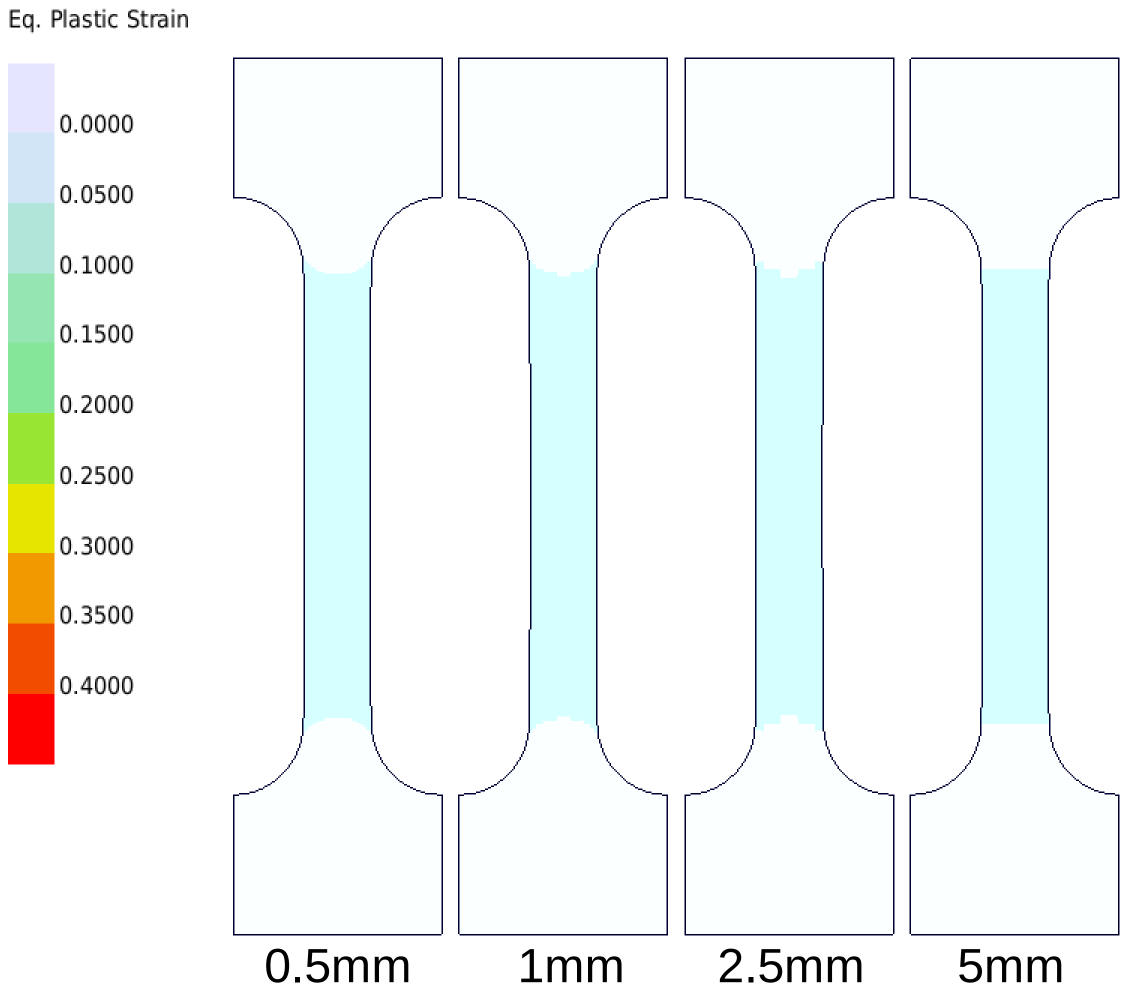
16th LS-DYNA Forum 2022, Bamberg, Germany

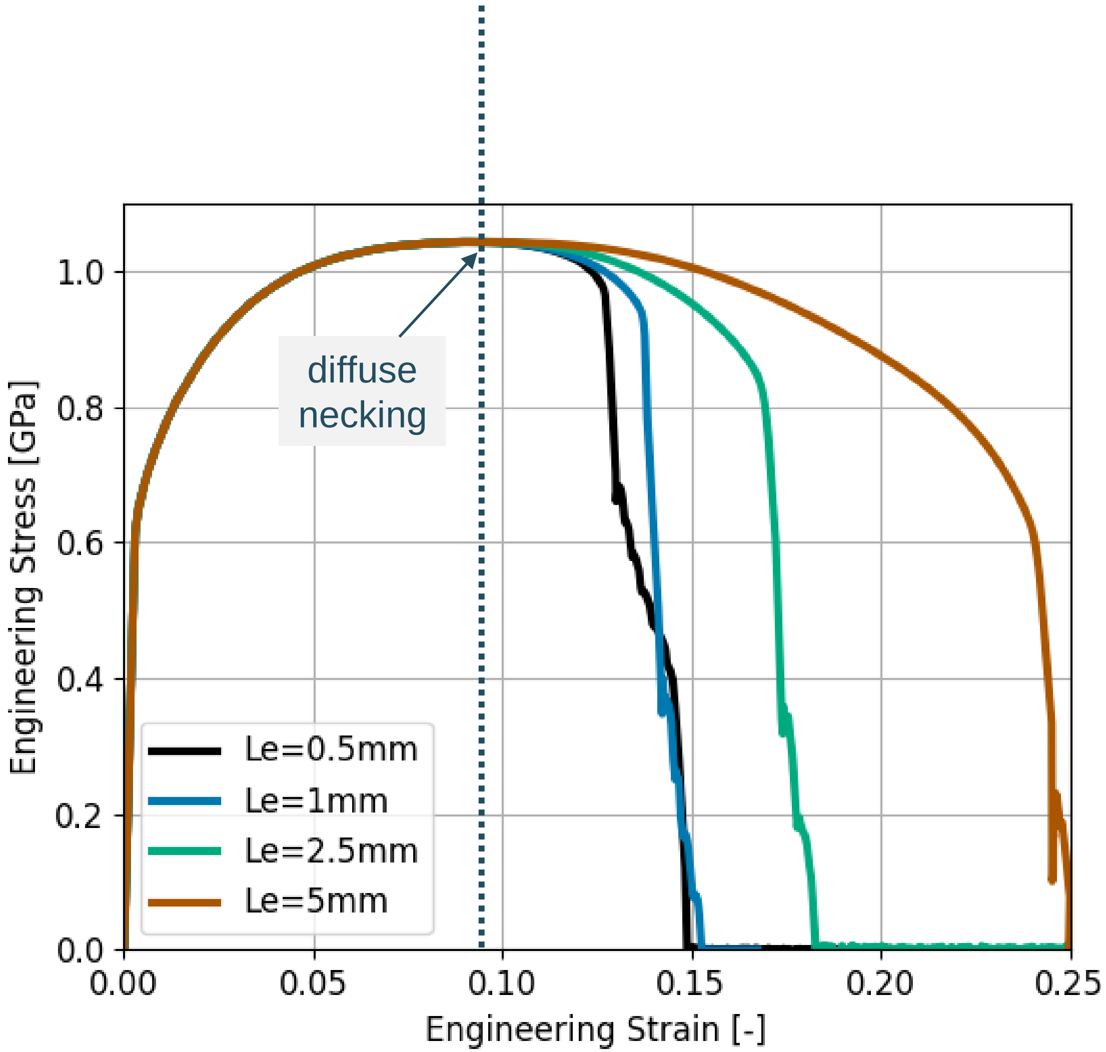
Instability and Mesh Dependence
Part II – Numerical simulation

Filipe Andrade, DYNAmore GmbH
Paul DuBois, Consulting Engineer
Markus Feucht, Mercedes-Benz AG
Tobias Graf, DYNAmore GmbH
Sergio Conde, DYNAmore GmbH
André Haufe, DYNAmore GmbH

Onset of spurious mesh dependence

Reference: Tensile test with different element sizes
Triaxiality 1/3 up to necking point

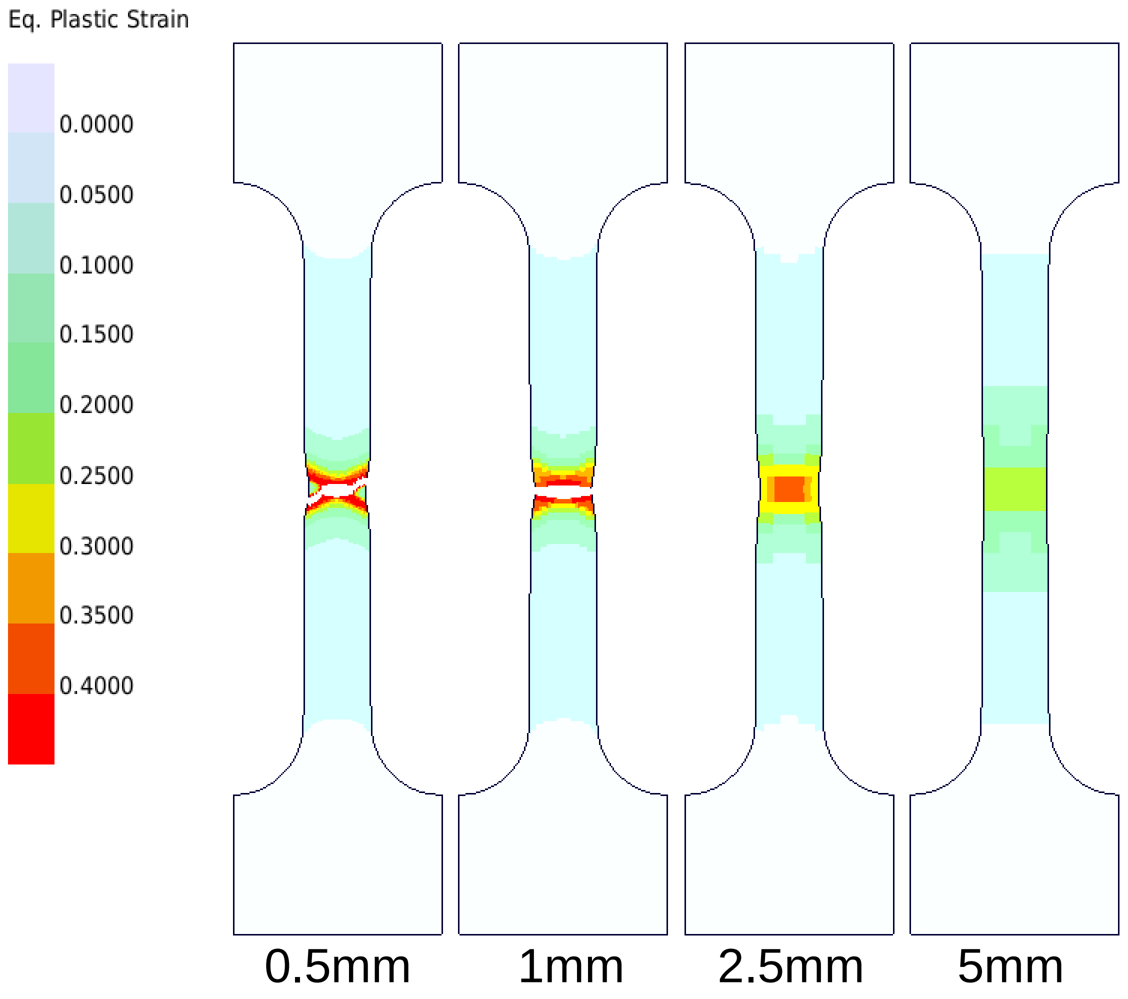


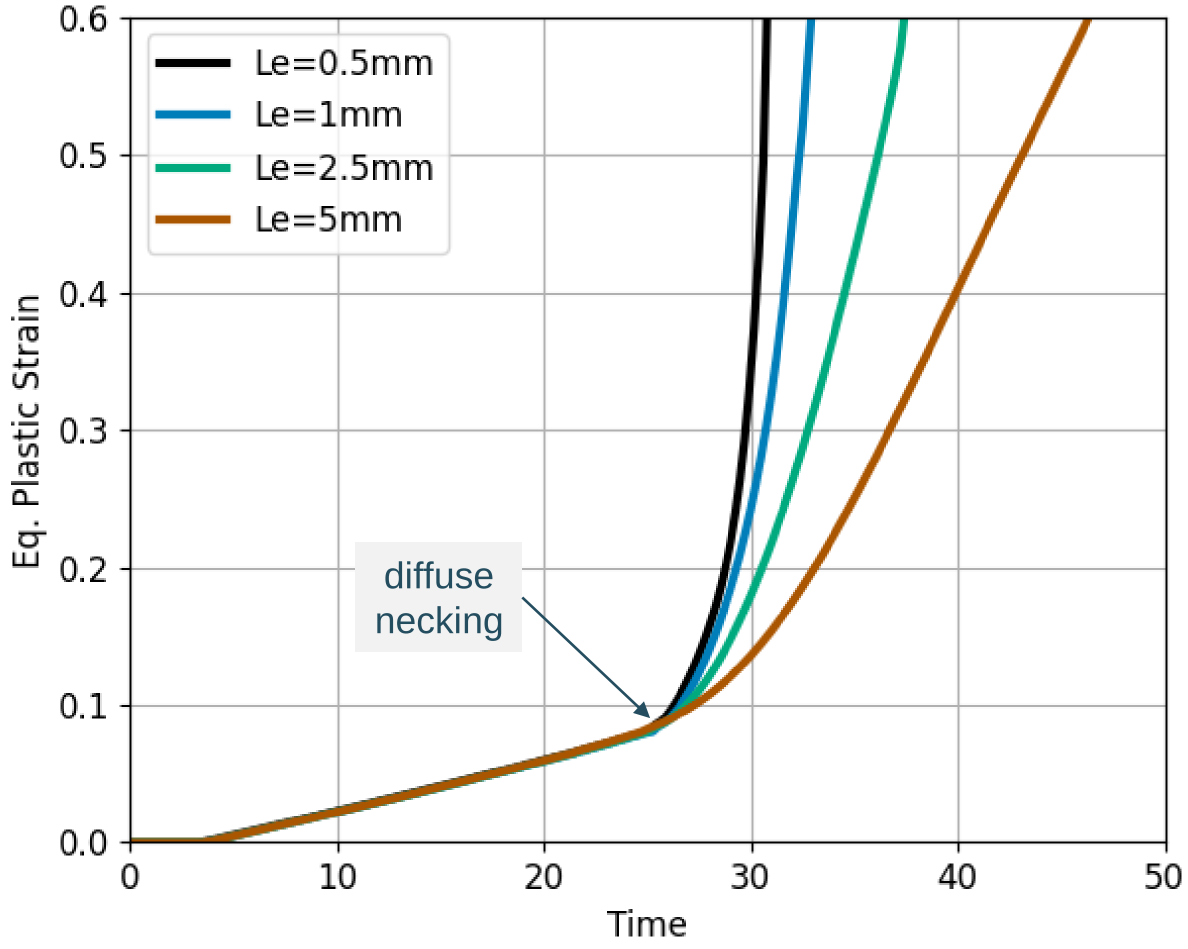


***MAT_024 + GISSMO (without regularization), monotonic hardening curve**

Onset of spurious mesh dependence

Reference: Tensile test with different element sizes
Triaxiality 1/3 up to necking point





***MAT_024 + GISSMO (without regularization), monotonic hardening curve**

Mesh dependence

Different types

The expression “mesh dependence” is somewhat vague and can as such have different interpretations. Therefore, it is important to highlight the main differences between the typical interpretations of this term.

Geometrical mesh dependence

- A consequence of discretization using finite elements
- May affect solution under any loading (purely elastic, plastic, etc.)
- Generally converging when mesh is fine enough → **can be solved by refining or higher order elements**
- Shells and solids affected in a similar way

“Spurious” mesh dependence

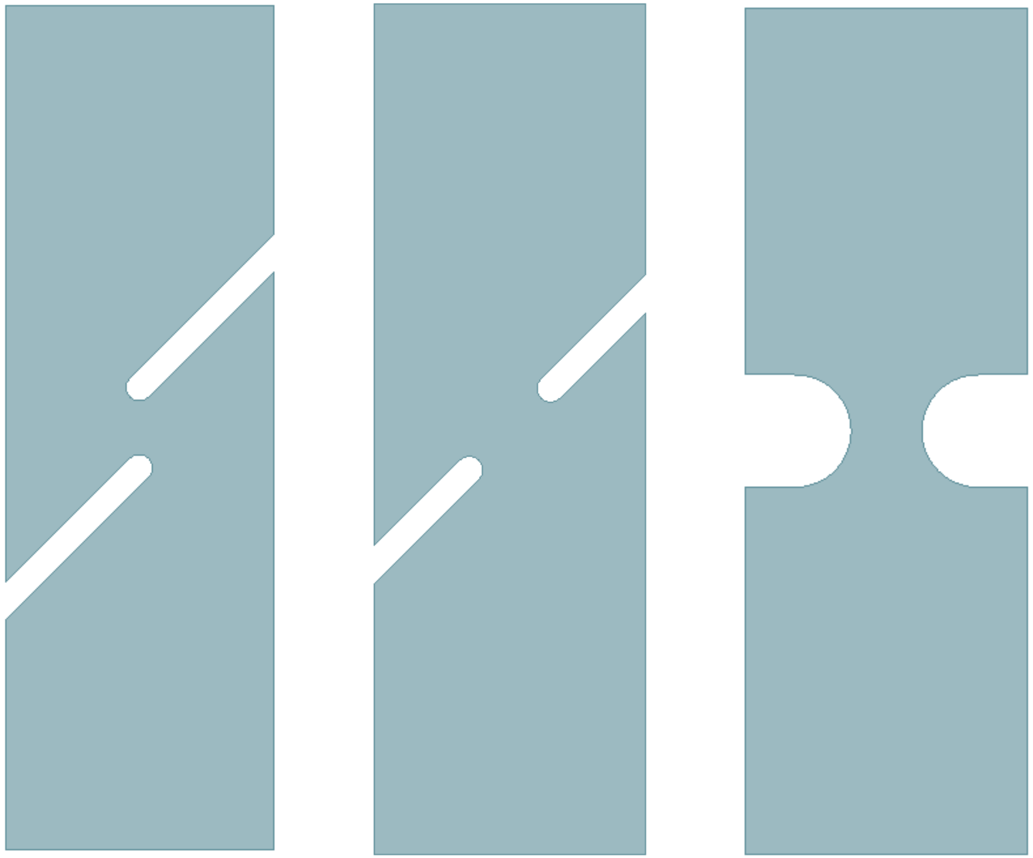
- A consequence of local continuum mechanics
- Only affects solution under certain conditions (e.g., after the necking point under a uniaxial stress state)
- Generally non-converging regardless how fine the mesh is → **cannot be solved by refining**
- Shells generally exhibit more spurious mesh dependence than solids

Ideally, only geometrically converged models should be regularized

Regularization strategies are intended to tackle the **spurious kind of mesh dependence**

Onset of spurious mesh dependence

Triaxialities other than 1/3



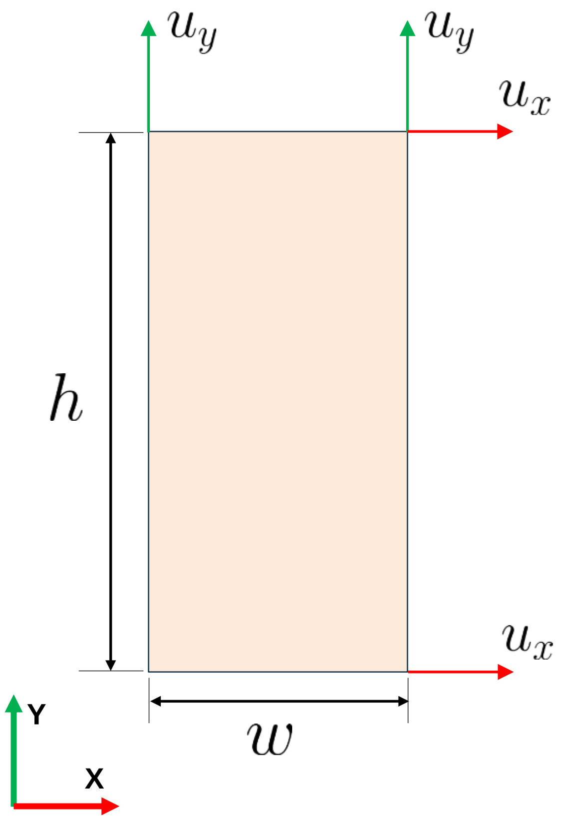
- Idea: Use other specimen geometries,
  discretize with varying element size

- Disadvantages:
  - Generally non-homogeneous deformation right from the beginning of deformation
  - Bad geometric description for large element sizes (geometric mesh dependence)

- Goal: Homogeneous deformation up to necking
  for any arbitrary stress state

Plane stress (shell elements)

How to keep the triaxiality constant throughout deformation



Strain ratio

Stress ratio as a function of the triaxiality

$$\begin{cases} u_x = (\exp \varepsilon_1 - 1)\, w \\ u_y = (\exp(b\varepsilon_1) - 1)\, h \end{cases}$$

$$b = \frac{\varepsilon_2}{\varepsilon_1} = \frac{1-2a}{a-2}$$

$$a = f(\eta)$$

Triaxiality as a function of the stress ratio

$$\eta = \frac{a+1}{3\sqrt{a^2 - a + 1}} \text{sign}\sigma_1$$

Stress ratio

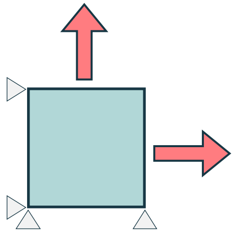
$$a = \frac{\sigma_2}{\sigma_1}$$

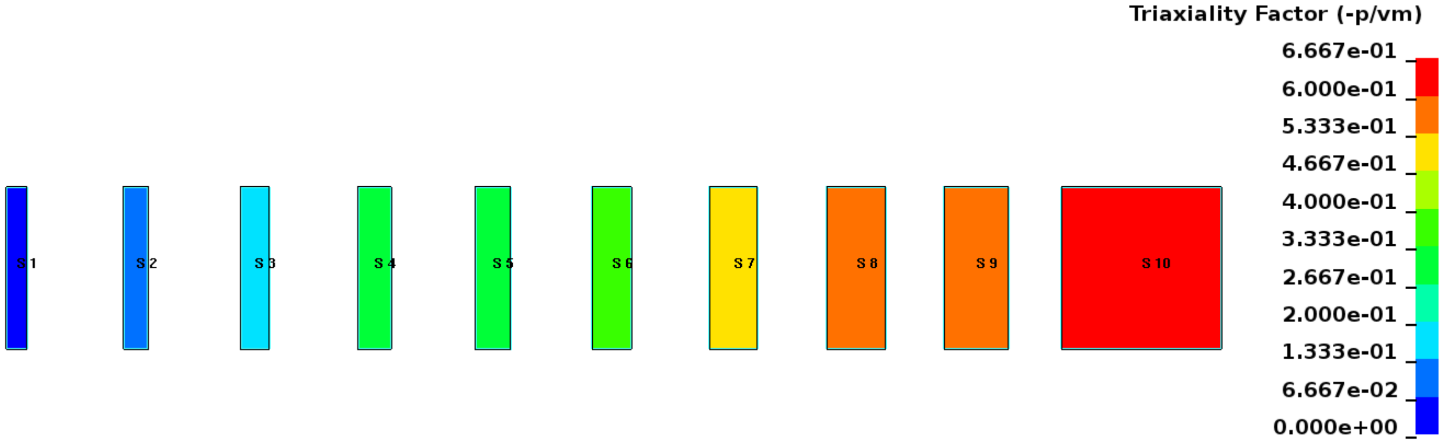
Assumptions/conditions:

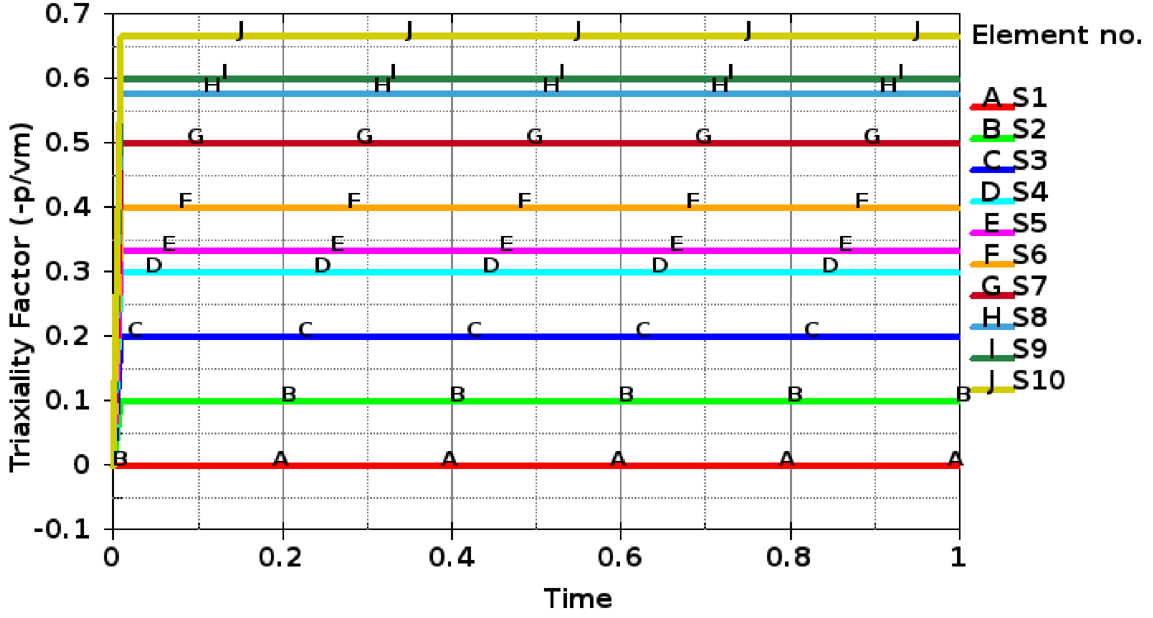
- Plane stress
- Negligible elastic strains
- J2 elastoplasticity (von Mises)
- Proportional loading (within the increment)
- Works well for triax > – 0.5

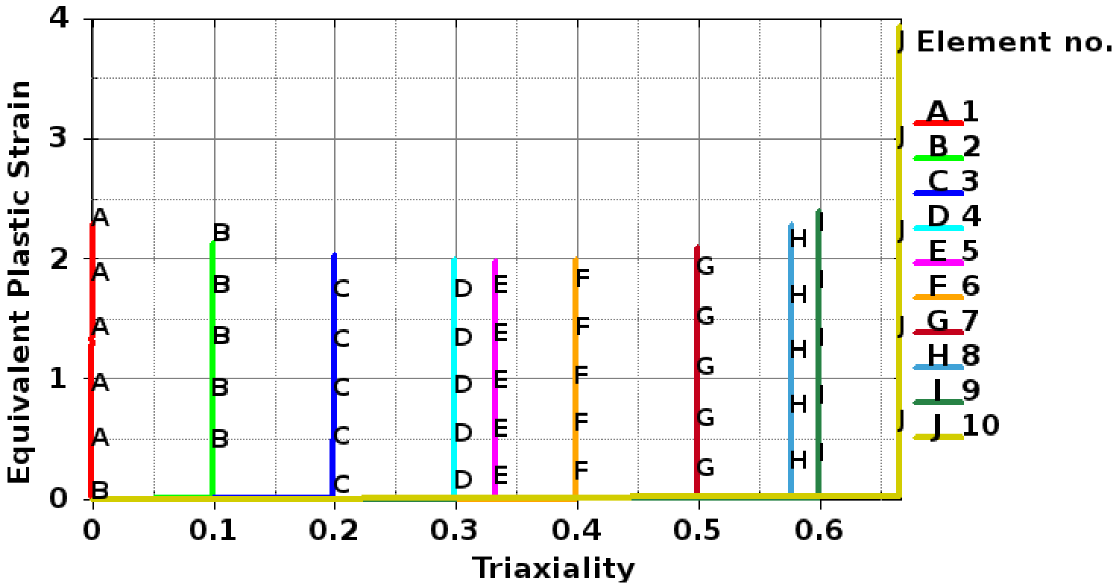
Plane stress (shell elements)

How to keep the triaxiality constant throughout deformation
Single element simulation under different triaxialities









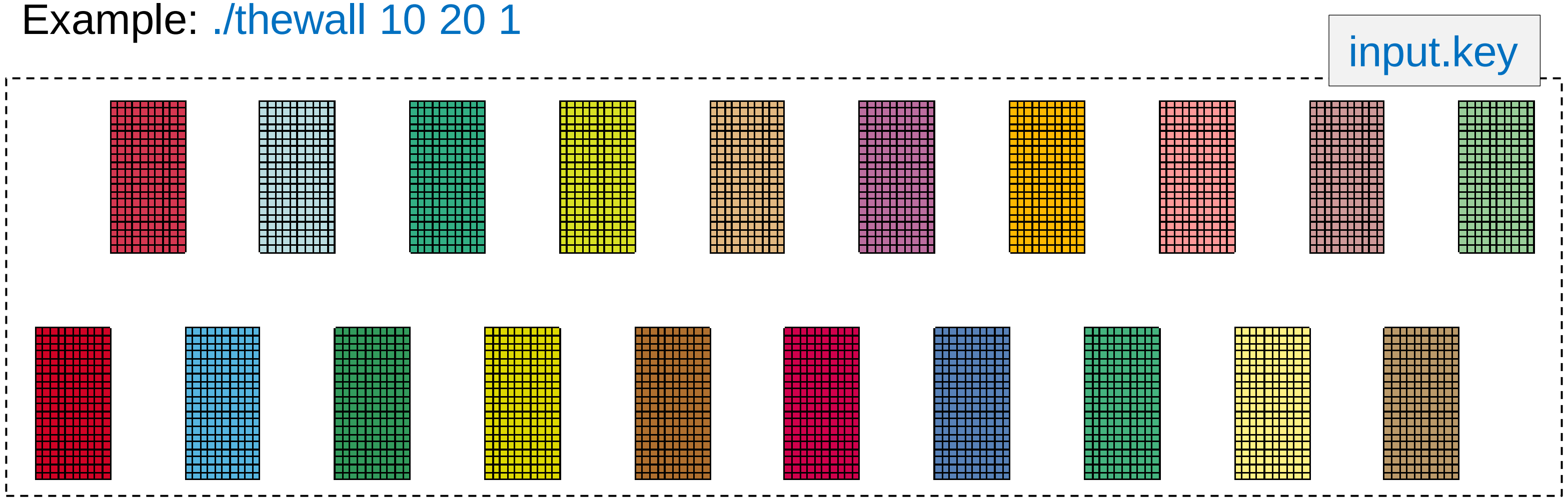
Plane stress (shell elements)

Automatic generation of “element blocks” through an external program

```
$ ./thewall

  ***************************************************
                    T H E  W A L L
                    --------------

                Build a wall of elements.

    An LS-DYNA input file for different triaxialities
    is generated. The aspect ratio and the element
    size can be specified.

                                        F. Andrade
                                          Jan 2019
  ***************************************************

 Usage:
 ------
 thewall <width> <height> <el size>


 Default values:
 ---------------
        width           = 1.0
        height          = 1.0
        element size    = 1.0
```

width height element size

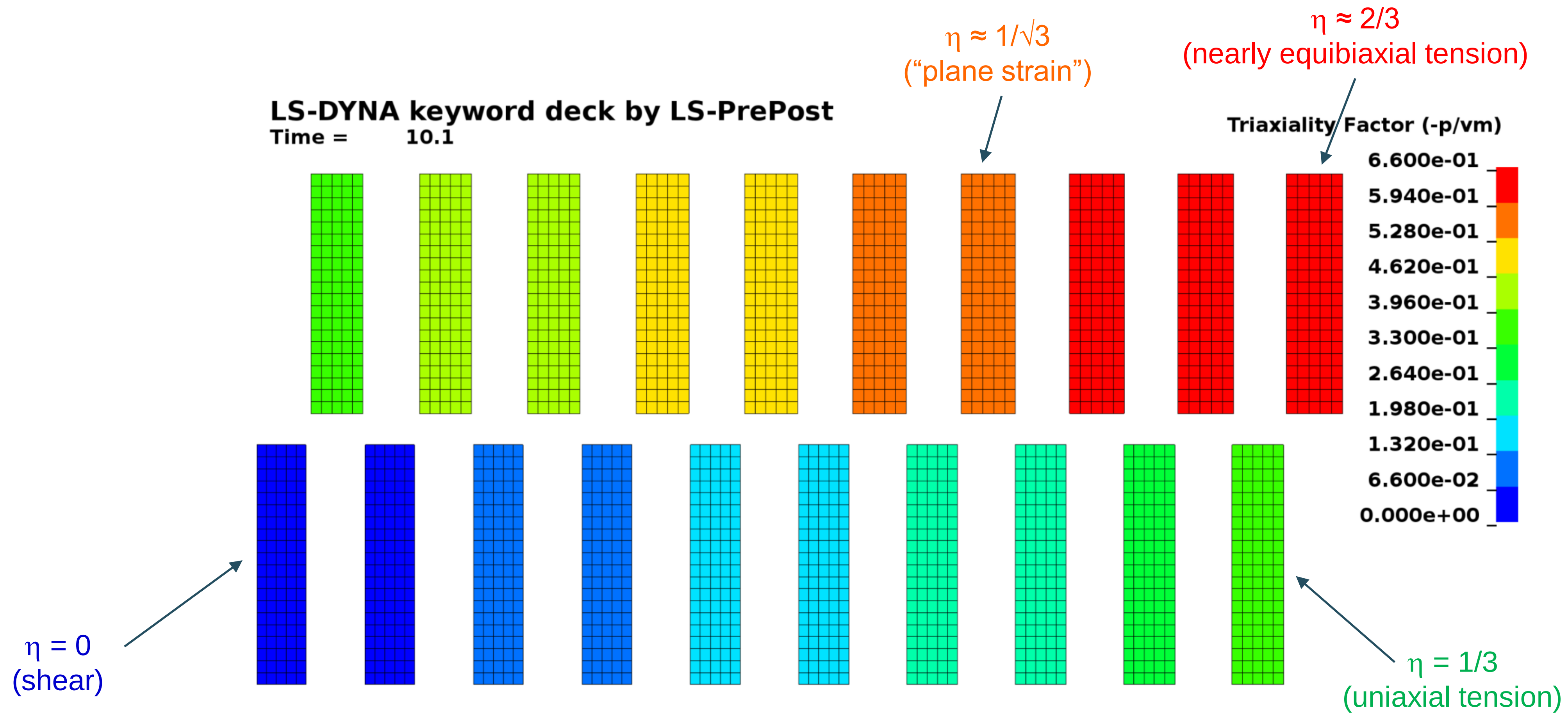
Example: ./thewall 10 20 1



A different triaxiality is assigned to each element block

Plane stress (shell elements)

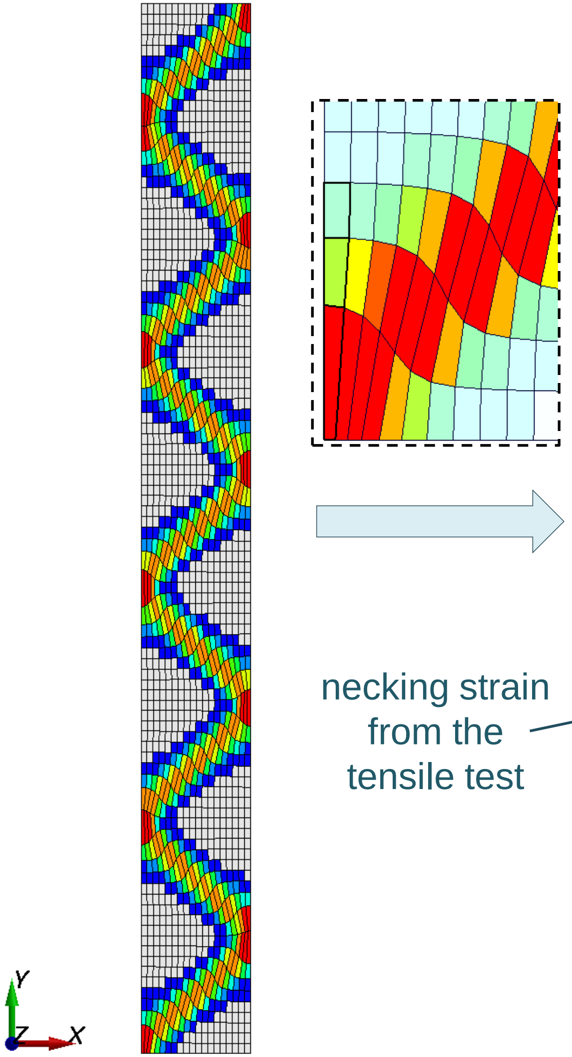
Simulation of the element blocks (width=5mm, height=20mm, element size=1.0mm)



Simulation with *MAT_024, monotonic hardening curve, aluminum properties, failure strain = 2.0 for all triaxialities

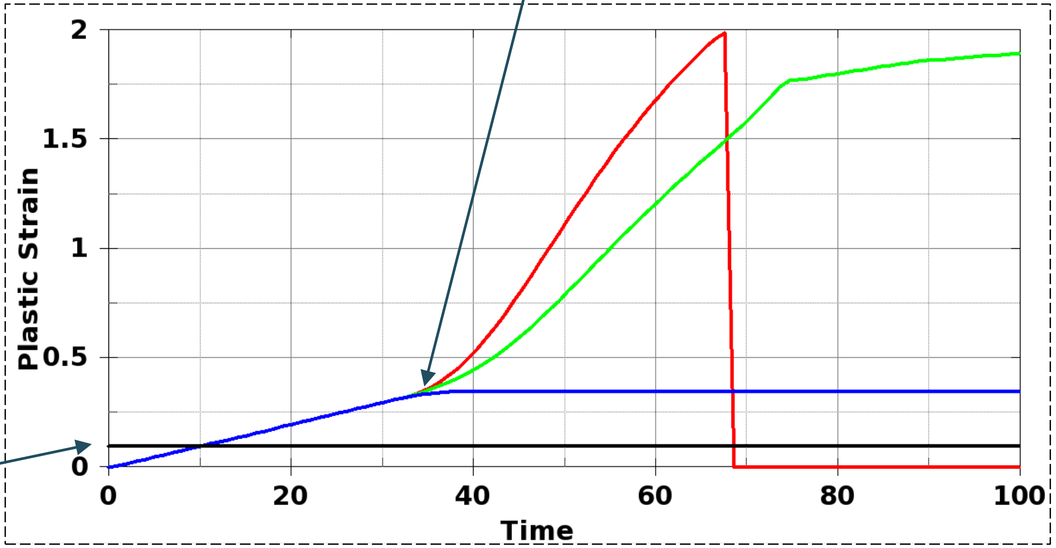
Plane stress (shell elements)

Behavior for triaxiality 1/3 (width=5mm, height=20mm, element size=0.25mm)

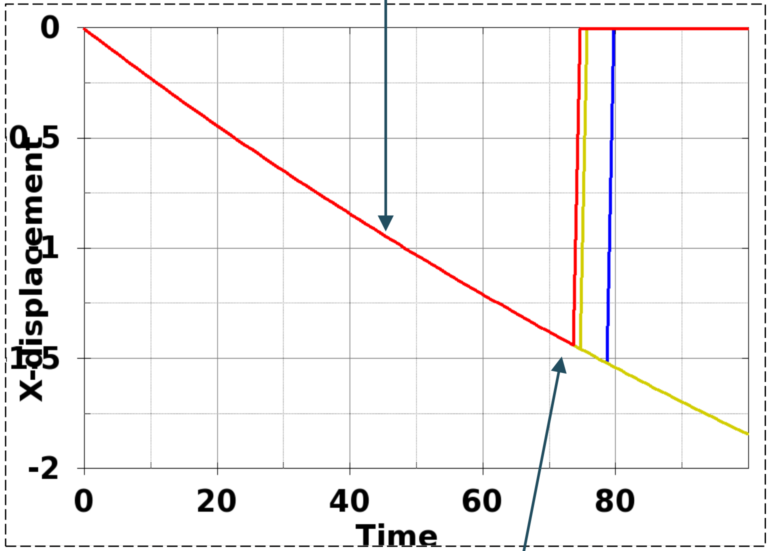


Localization takes place much latter than necking strain from the tensile test

necking strain from the tensile test



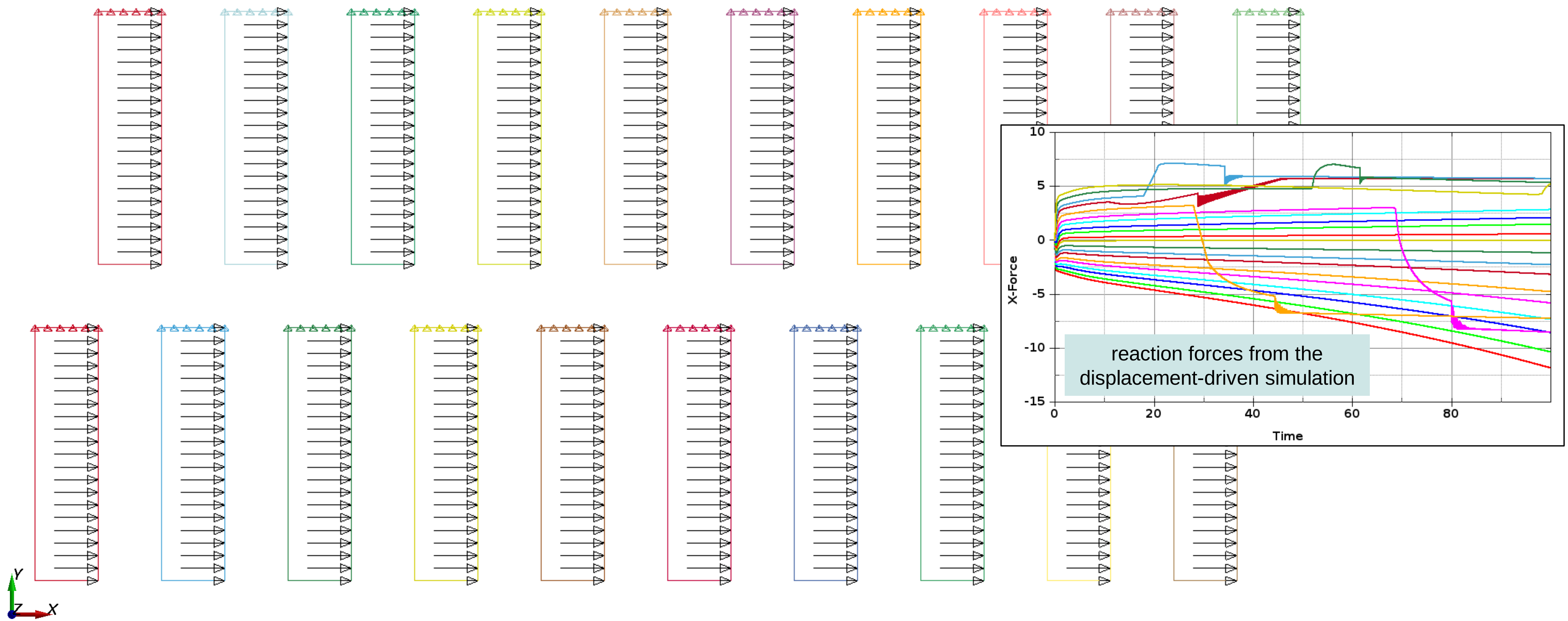
Lateral displacement identical for all nodes until failure at epsp = 2.0



Element failure when epsp = 2.0

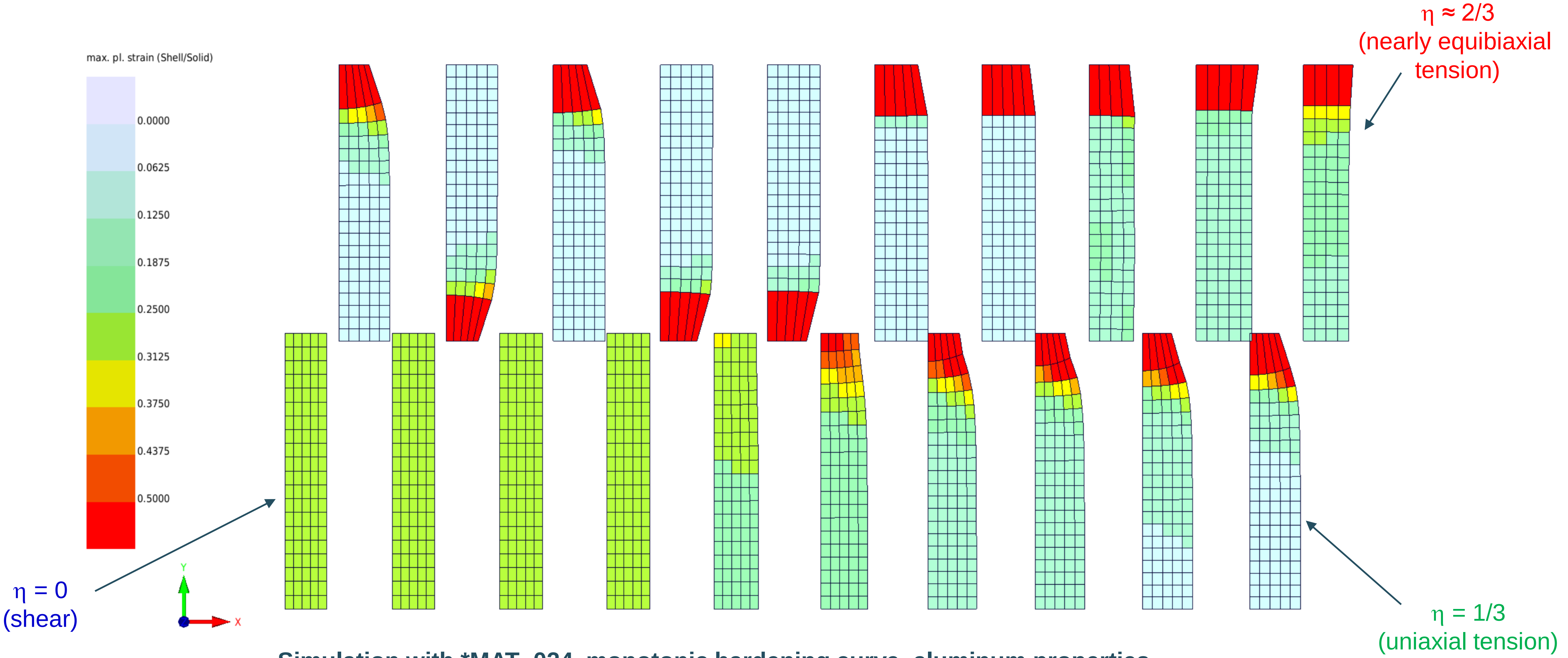
Plane stress (shell elements)

Imposing lateral forces instead of displacements



Plane stress (shell elements)

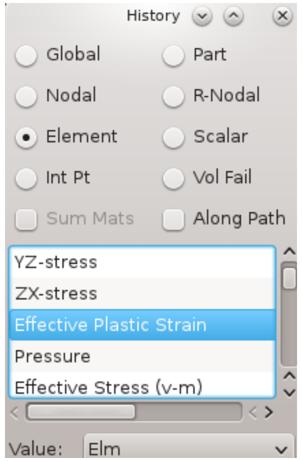
Simulation of the element blocks (width=5mm, height=20mm, element size=1.0mm)

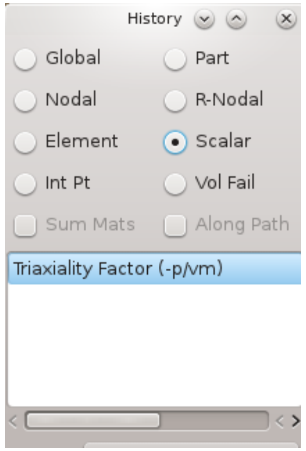


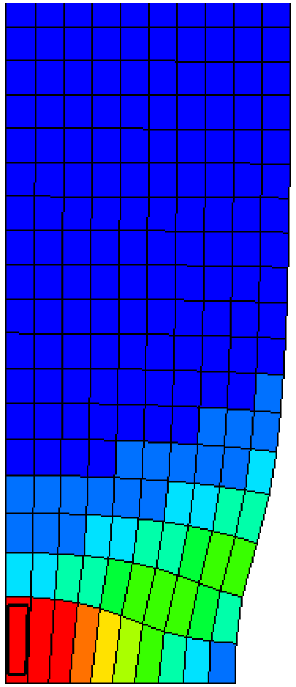
Simulation with *MAT_024, monotonic hardening curve, aluminum properties, failure strain = 2.0 for all triaxialities

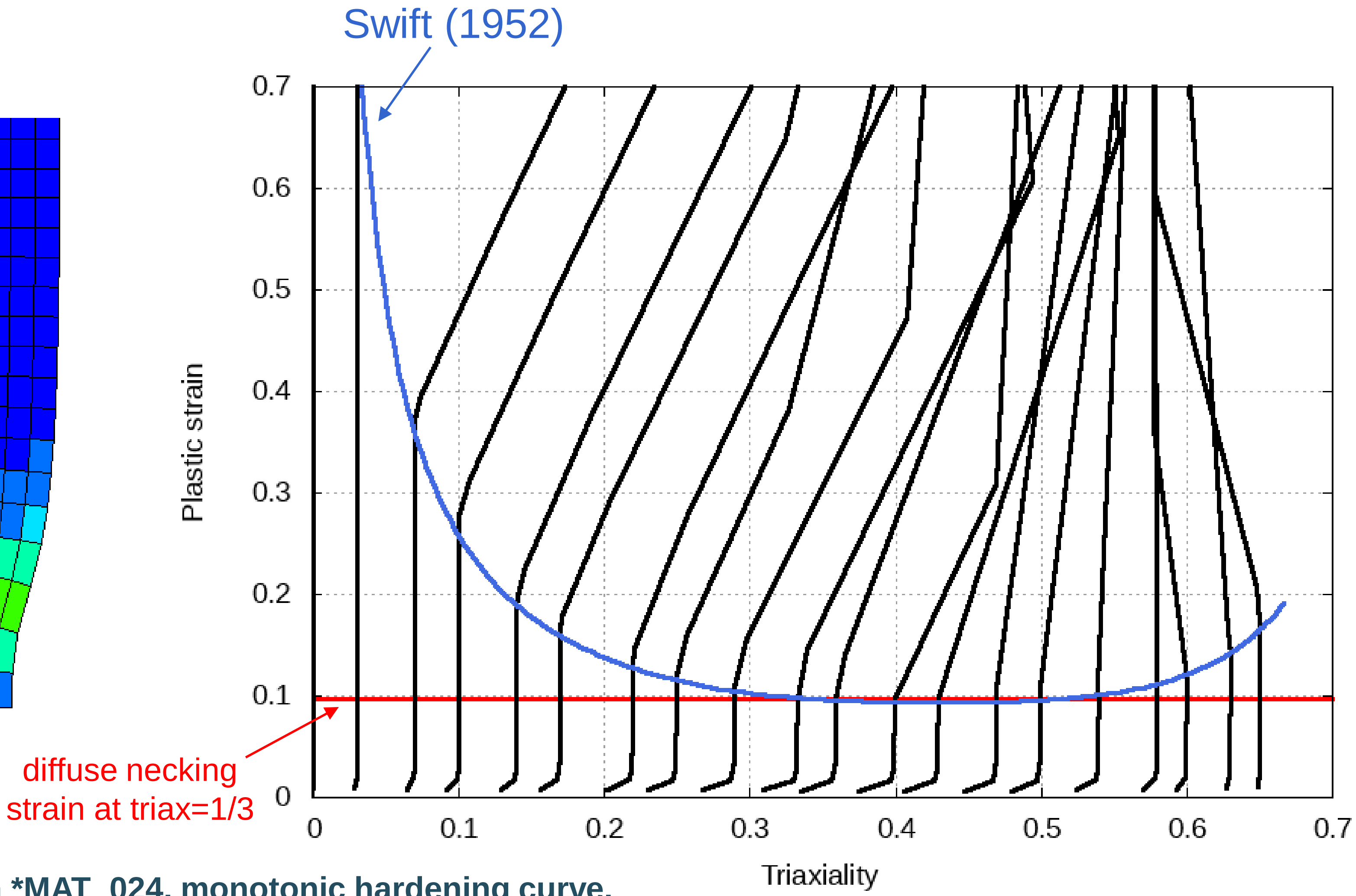
Plane stress (shell elements)

Strain-triaxiality paths (width=5mm, height=40mm, element size=0.5mm)





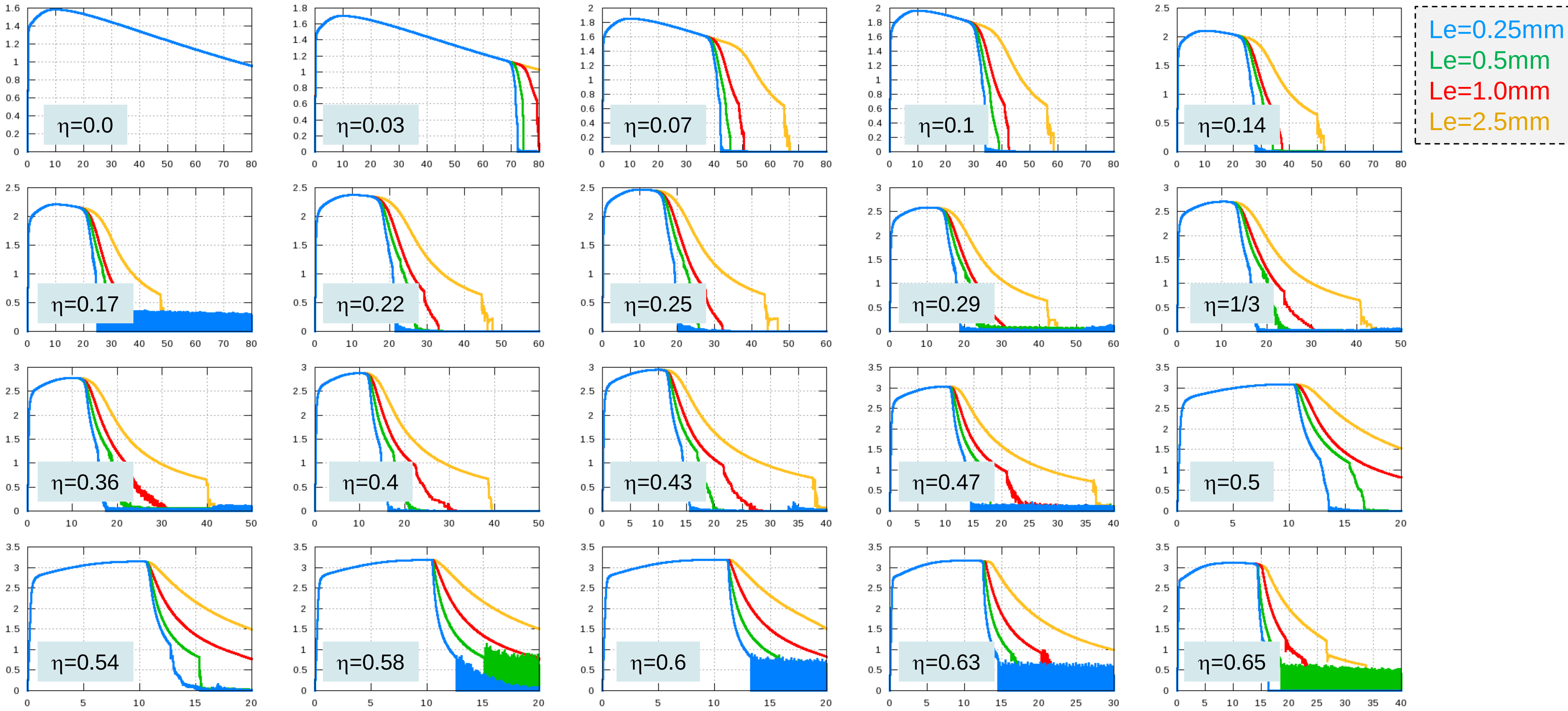




Simulation with *MAT_024, monotonic hardening curve, aluminum properties, failure strain = 2.0 for all triaxialities

Plane stress (shell elements)

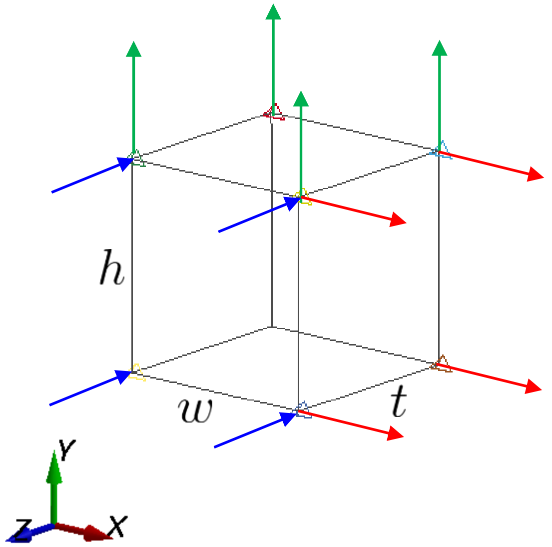
Vertical reaction force vs time (width=5mm, height=40mm, el. size=0.25mm – 2.5mm)



Simulation with *MAT_024, monotonic hardening curve, aluminum properties, failure strain = 2.0 for all triaxialities

Three-dimensional case (volume elements)

How to keep triaxiality and Lode parameter constant throughout deformation



Strain components

$$\begin{cases} u_x = (\exp \varepsilon_1 - 1)\, w \\ u_y = (\exp \varepsilon_2 - 1)\, h \\ u_z = (\exp \varepsilon_3 - 1)\, t \end{cases}$$

$$\varepsilon_1 = f(k, m, E, \nu, \sigma_y)$$
$$\varepsilon_2 = f(k, m, E, \nu, \sigma_y)$$
$$\varepsilon_3 = f(k, m, E, \nu, \sigma_y)$$

Stress ratio values

$$k = \frac{\sigma_2}{\sigma_1} = f(\eta, \xi)$$
$$m = \frac{\sigma_3}{\sigma_1} = f(\eta, \xi)$$

Assumptions/conditions for solids:

- Linear elasticity
- J2 elastoplasticity (von Mises)
- Proportional loading (within the increment)

Triaxiality as a function of the stress ratio values

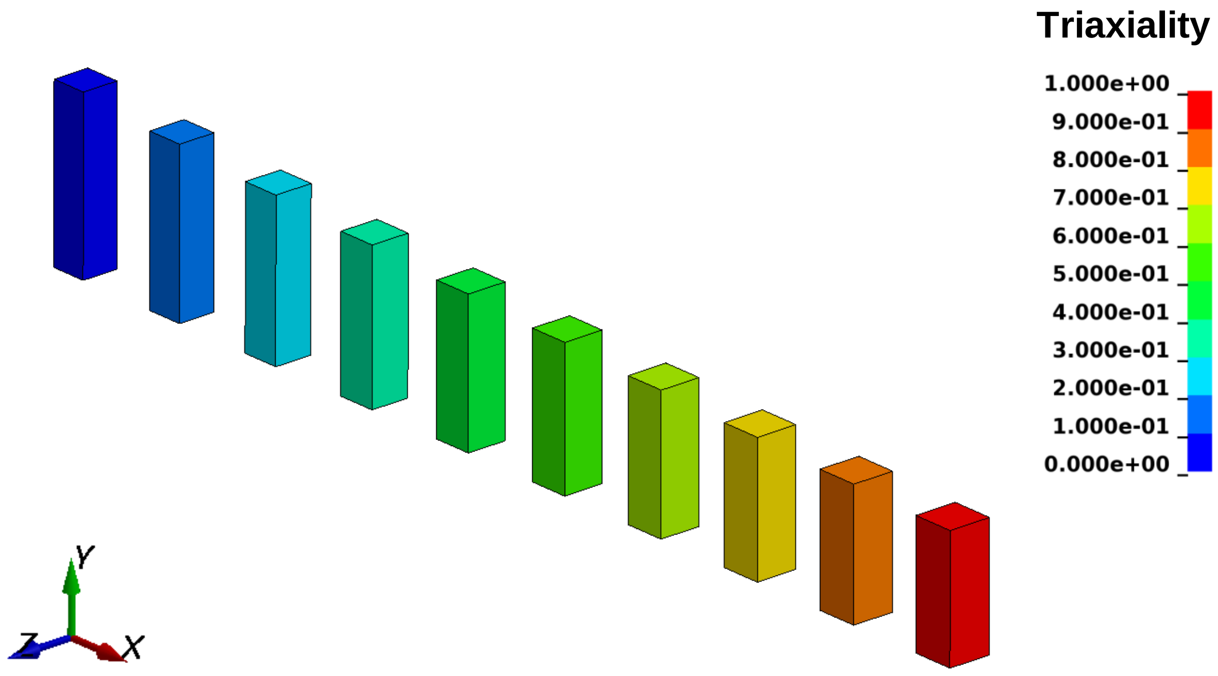
$$\eta = -\frac{k+m+1}{3\sqrt{\frac{1}{2}\left[(1-k)^2 + (k-m)^2 + (m-1)^2\right]}}\,\mathrm{sign}(\sigma_1)$$

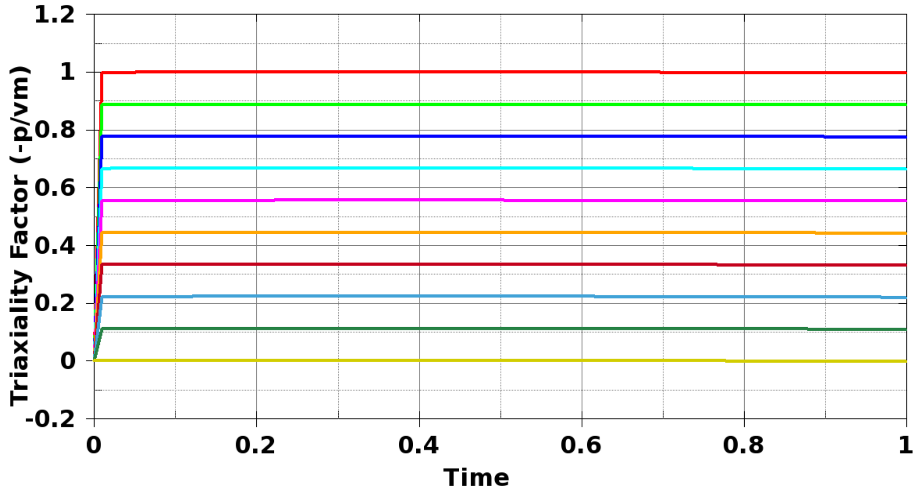
Lode parameter as a function of the stress ratio values

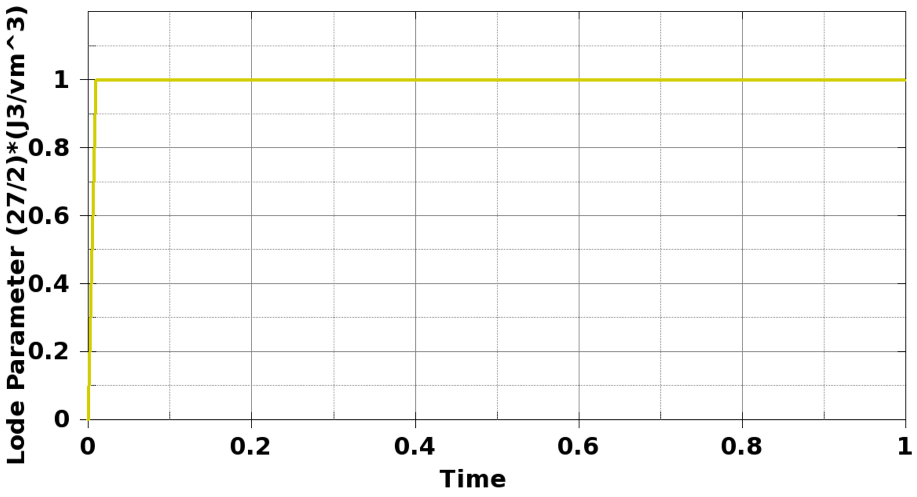
$$\xi = \frac{27}{2}\frac{s_1 s_2 s_3}{\sigma_{eq}^3} = f(k, m)\ \mathrm{sign}\,(\sigma_1)$$

Three-dimensional case (volume elements)

Constant triaxiality and Lode parameter throughout deformation





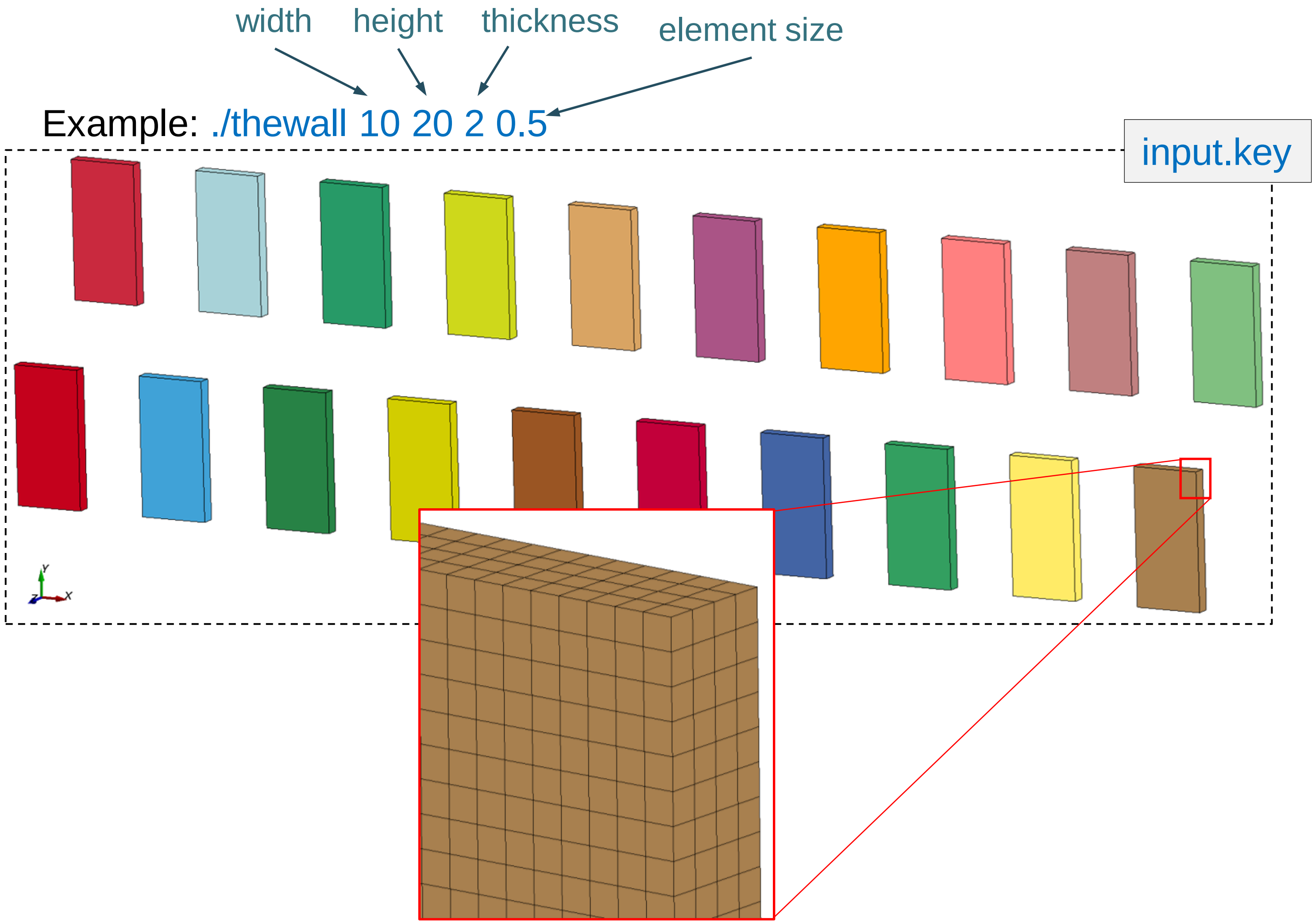


Simulation with *MAT_024, monotonic hardening curve, no failure

Three-dimensional case (volume elements)

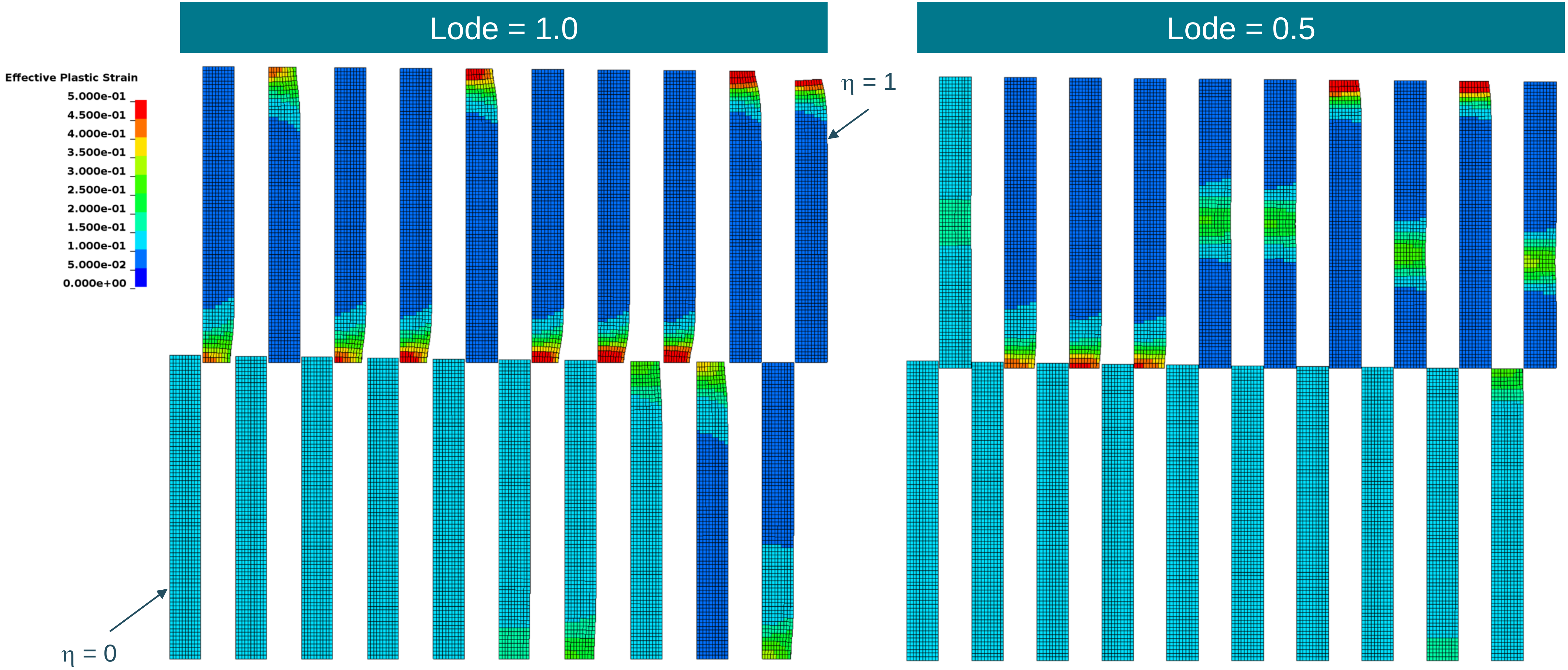
Automatic generation of “element blocks” through an external program

```
$ ./thewall3d

 **************************************************
                 T H E   W A L L - 3 D
                 --------------------

              Build a 3D wall of elements.

    An LS-DYNA input file for different triaxialities
    is generated. The aspect ratio and the element
    size can be specified.

                                         F. Andrade
                                           Mar 2019
  **************************************************


 Usage:
 ------
 thewall3d <width> <height> <thickness> <el size>


 Default values:
 ---------------
        width           = 1.0
        height          = 1.0
        thickness       = 1.0
        element size    = 1.0
```



Three-dimensional case (volume elements)

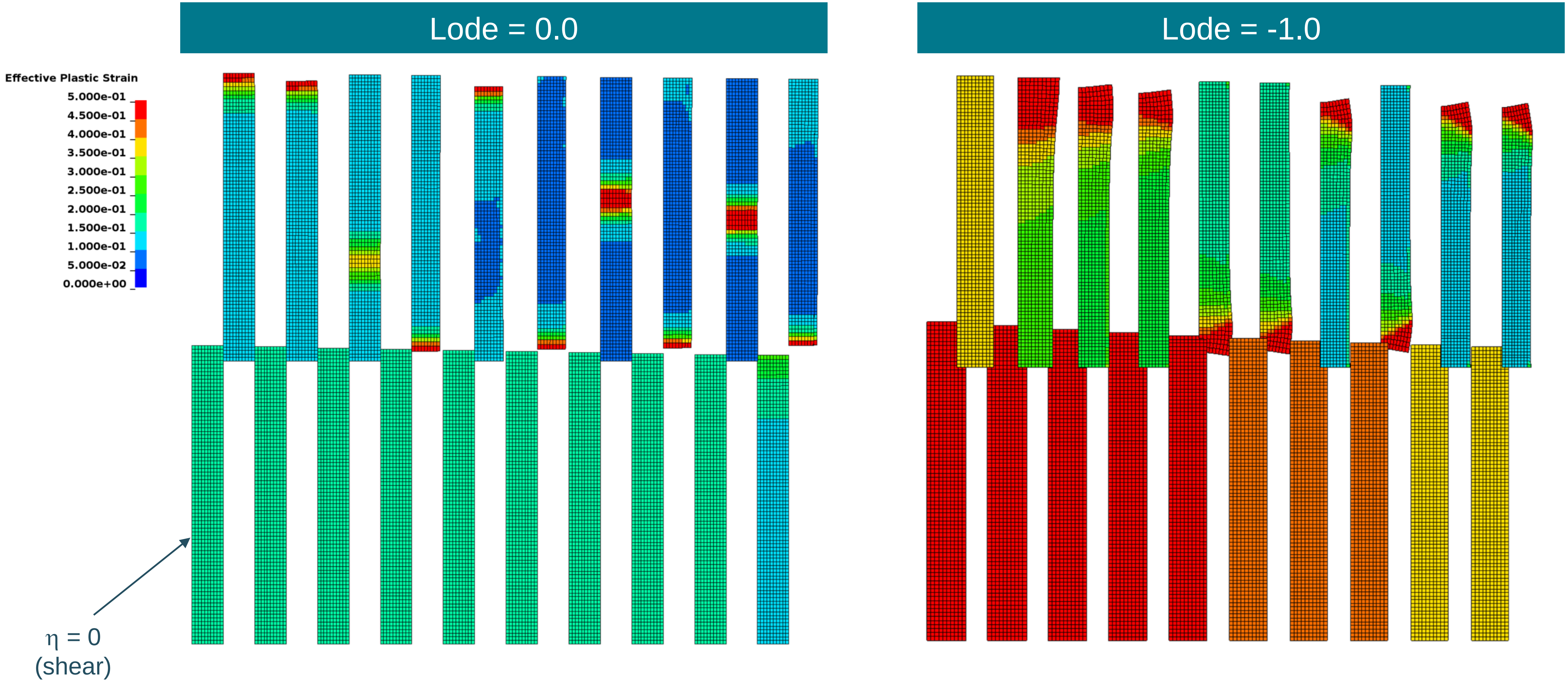
Simulation of the element blocks (w=5mm, h=40mm, t=2mm, el. size=0.5mm)



Simulation with *MAT_024, monotonic hardening curve, aluminum properties, failure strain = 2.0 for all triaxialities

Three-dimensional case (volume elements)

Simulation of the element blocks (w=5mm, h=40mm, t=2mm, el. size=0.5mm)

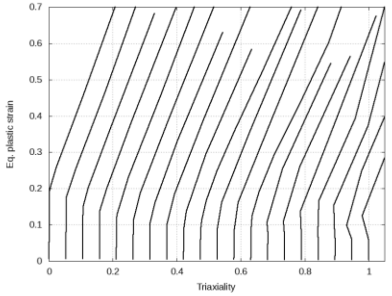


Simulation with *MAT_024, monotonic hardening curve, aluminum properties, failure strain = 2.0 for all triaxialities

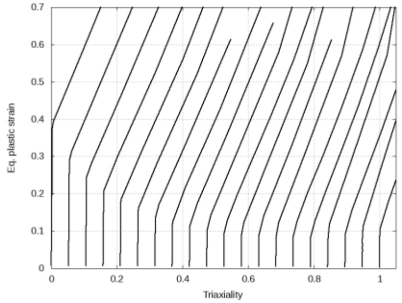
Three-dimensional case (volume elements)

Evaluation of the element blocks (w=5mm, h=40mm, t=2mm, el. size=0.5mm)

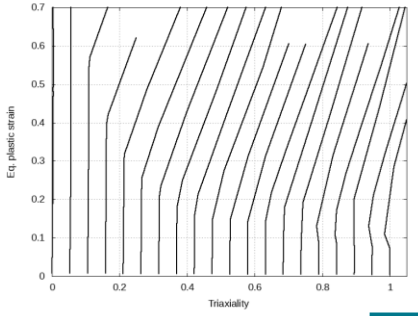
Lode = 1.0



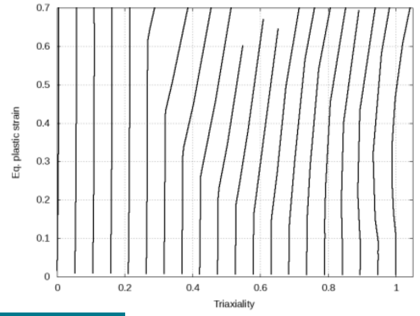
Lode = 0.5



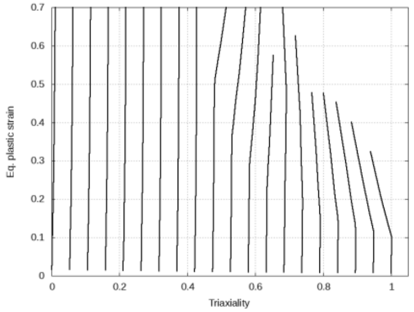
Lode = 0.0



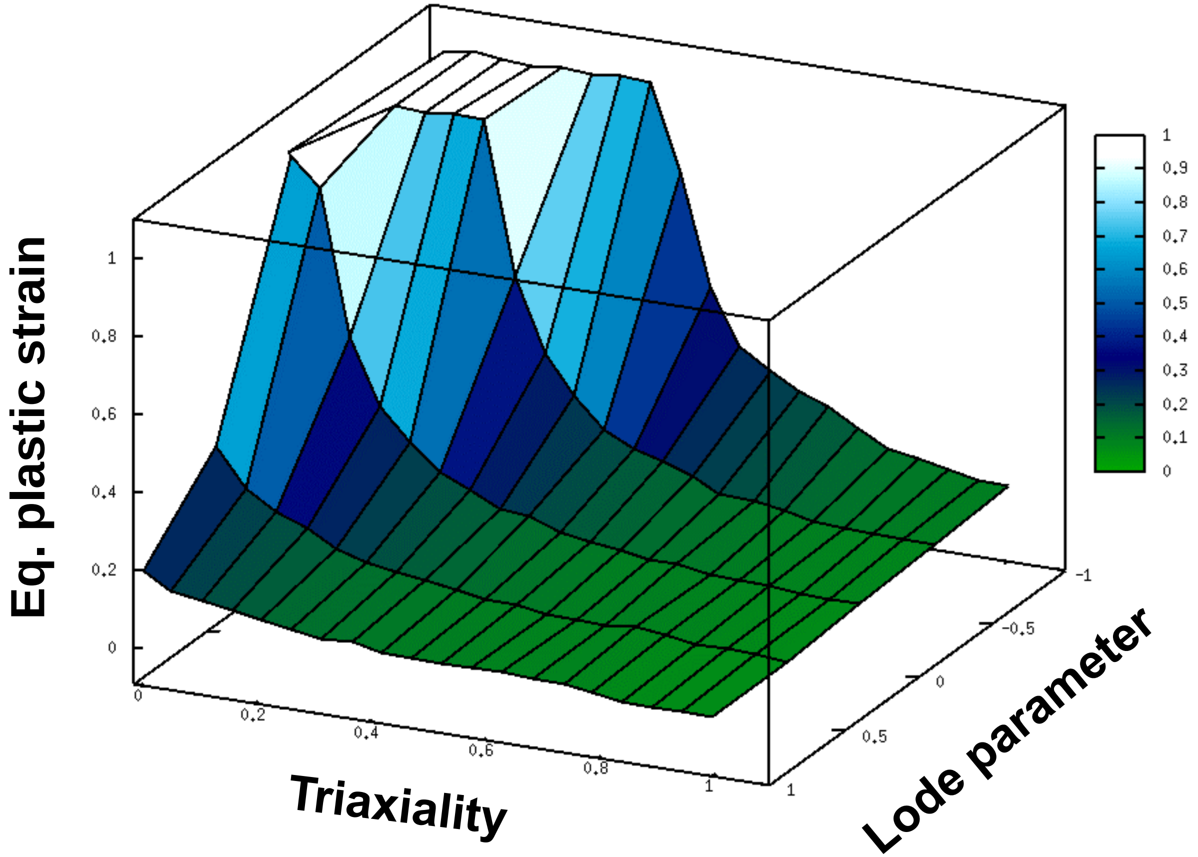
Lode = -0.5



Lode = -1.0

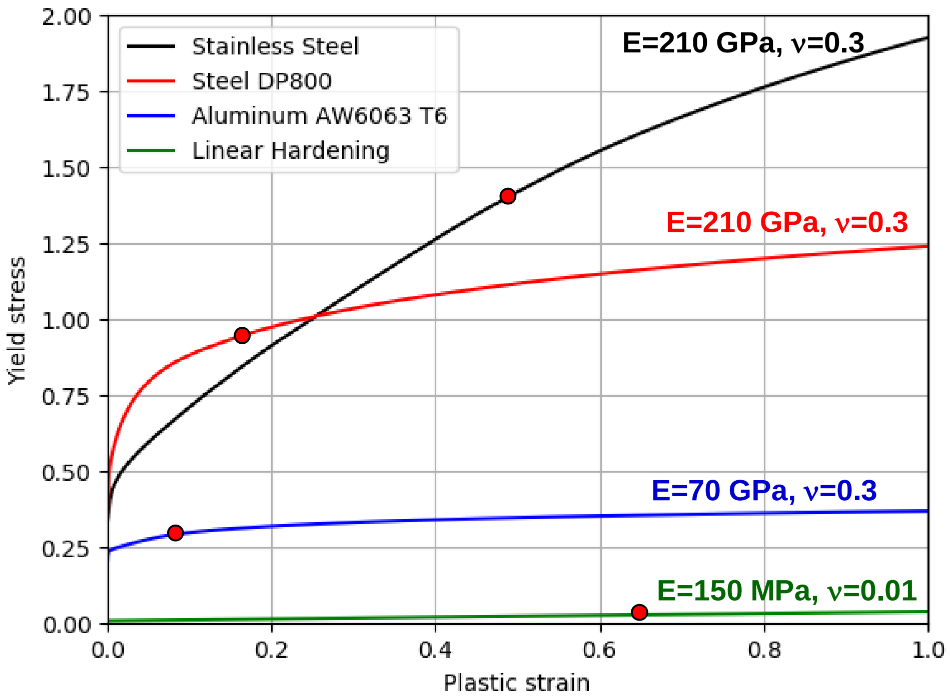


May 9, 2019: First visualization of the instability surface

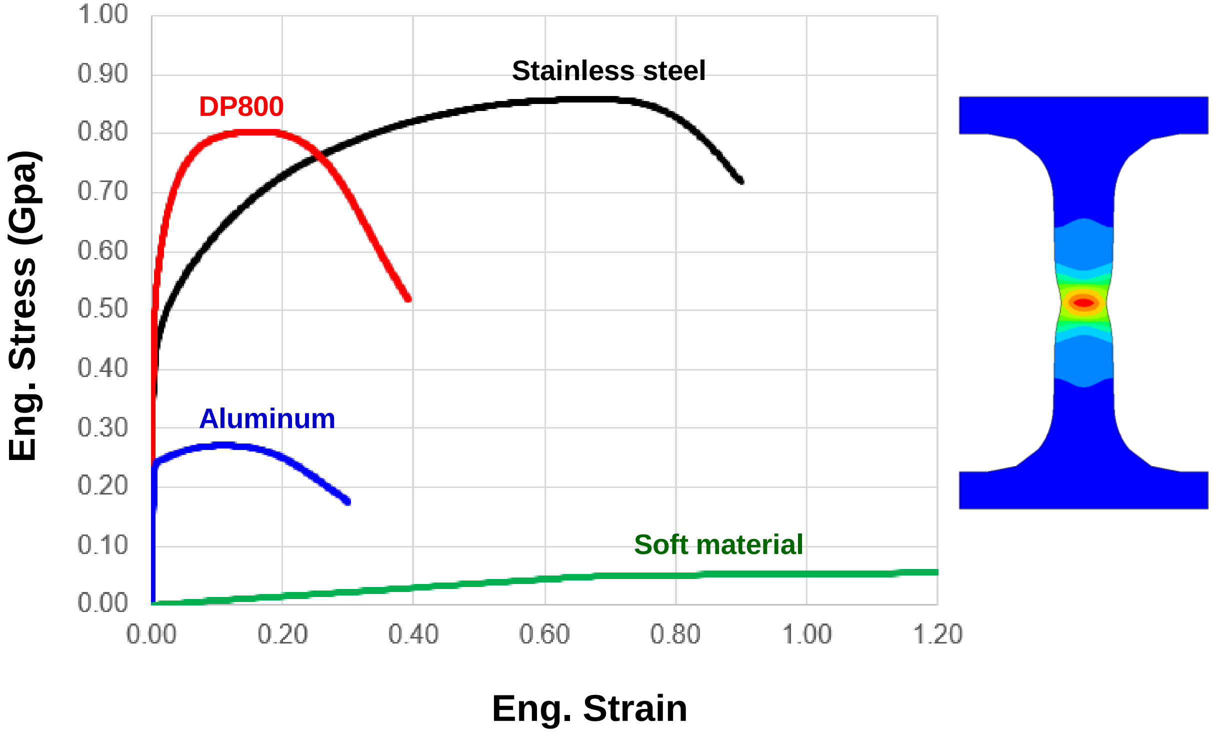


Four different materials

Stainless steel, dual-phase steel, aluminum extrusion, soft material

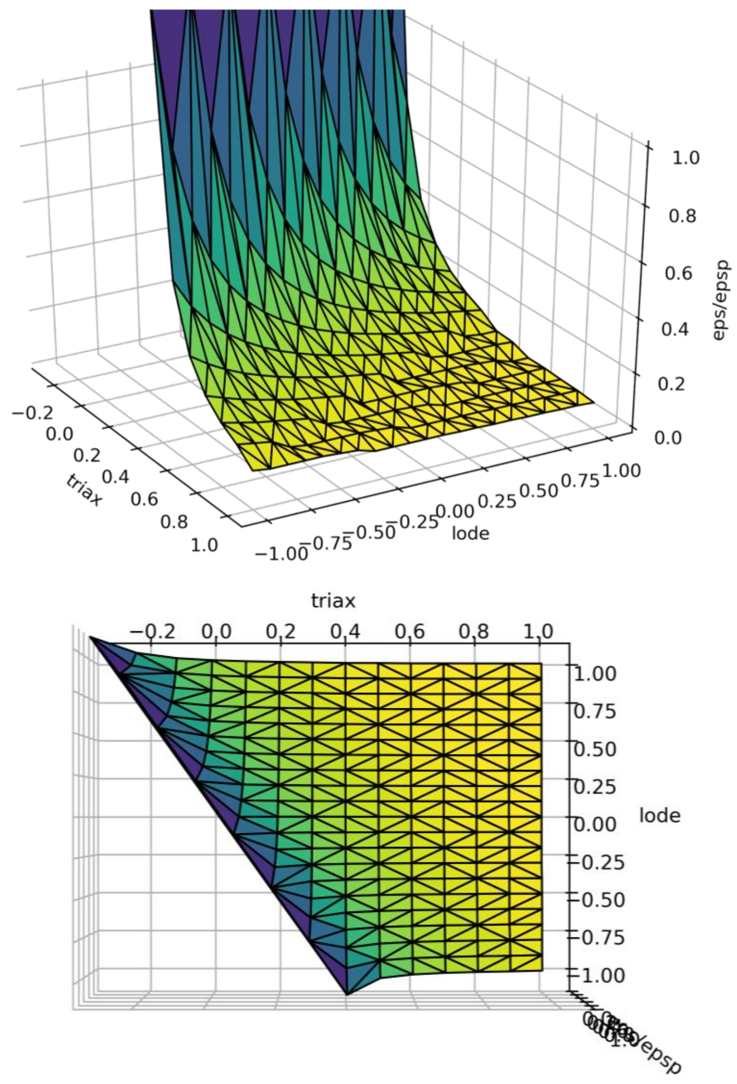


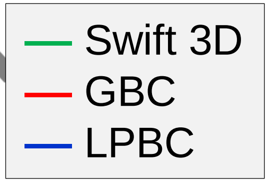
● Necking strain from tensile test

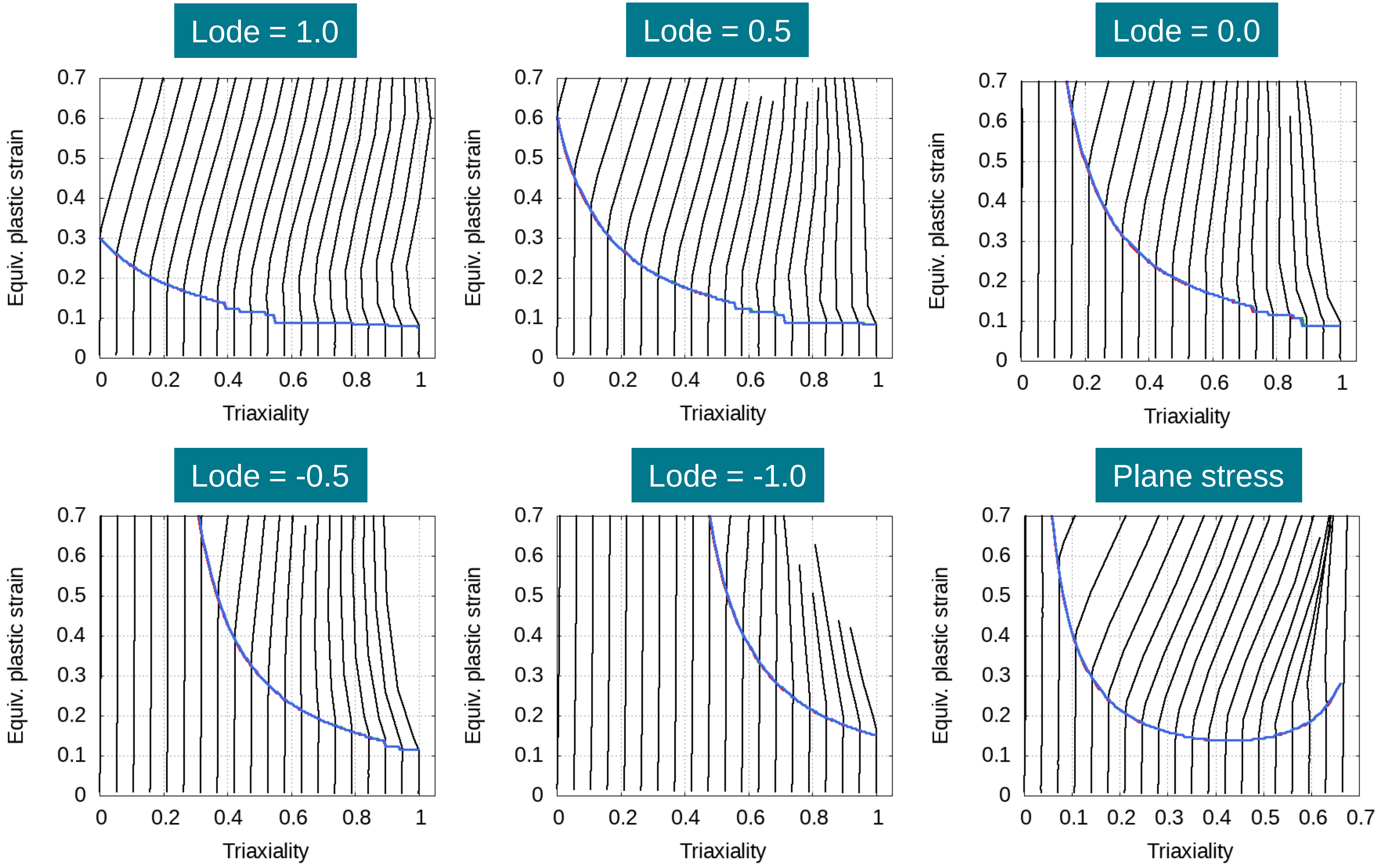


Dualphase steel (DP800)

Comparison between simulation and analytical prediction

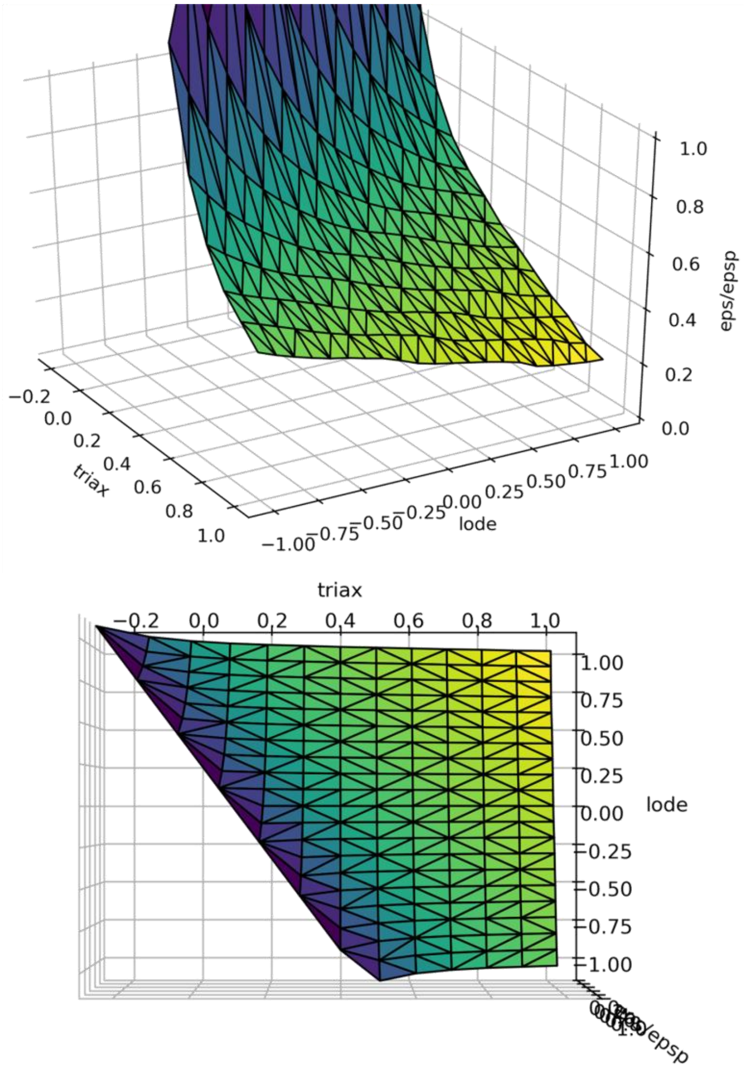


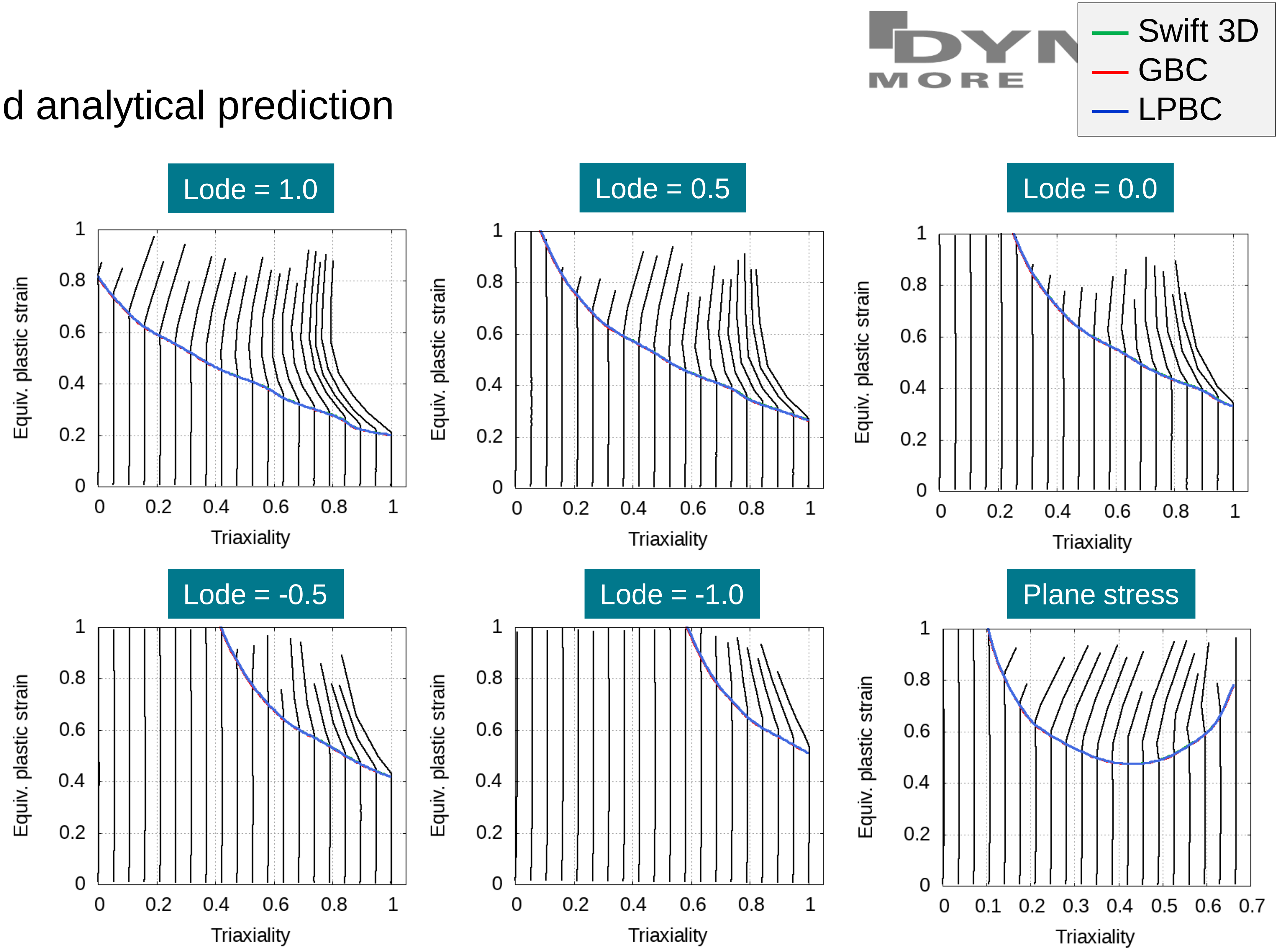




Stainless steel

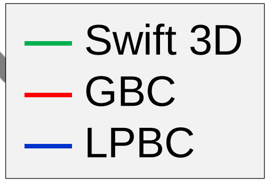
Comparison between simulation and analytical prediction

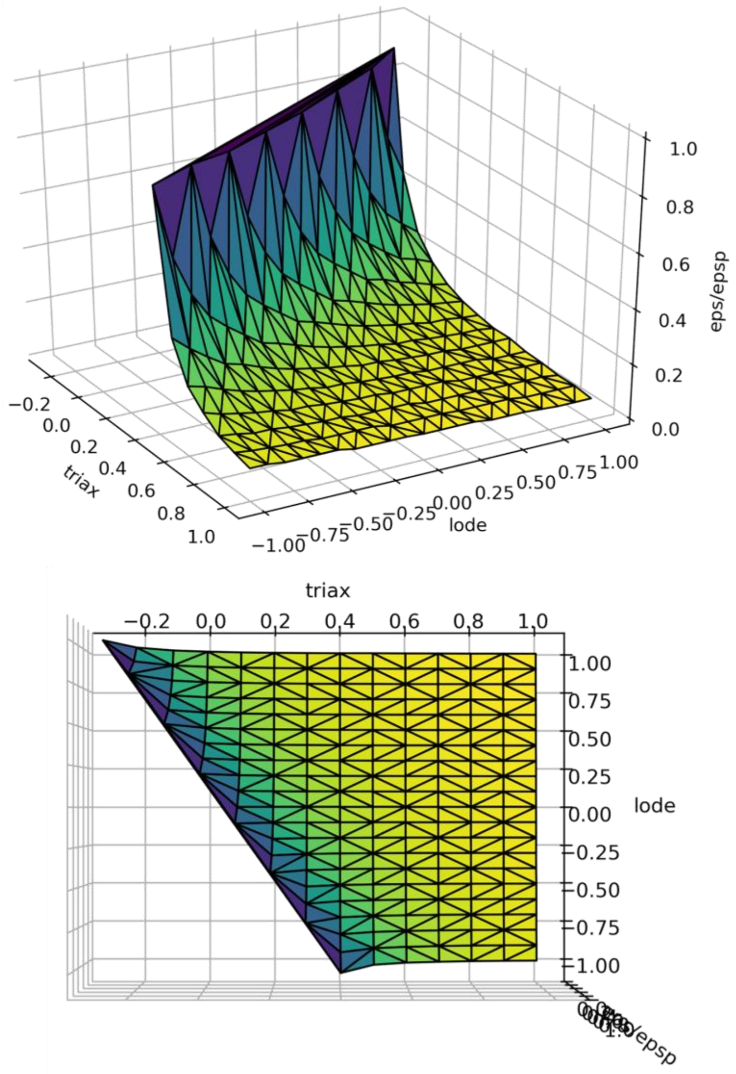


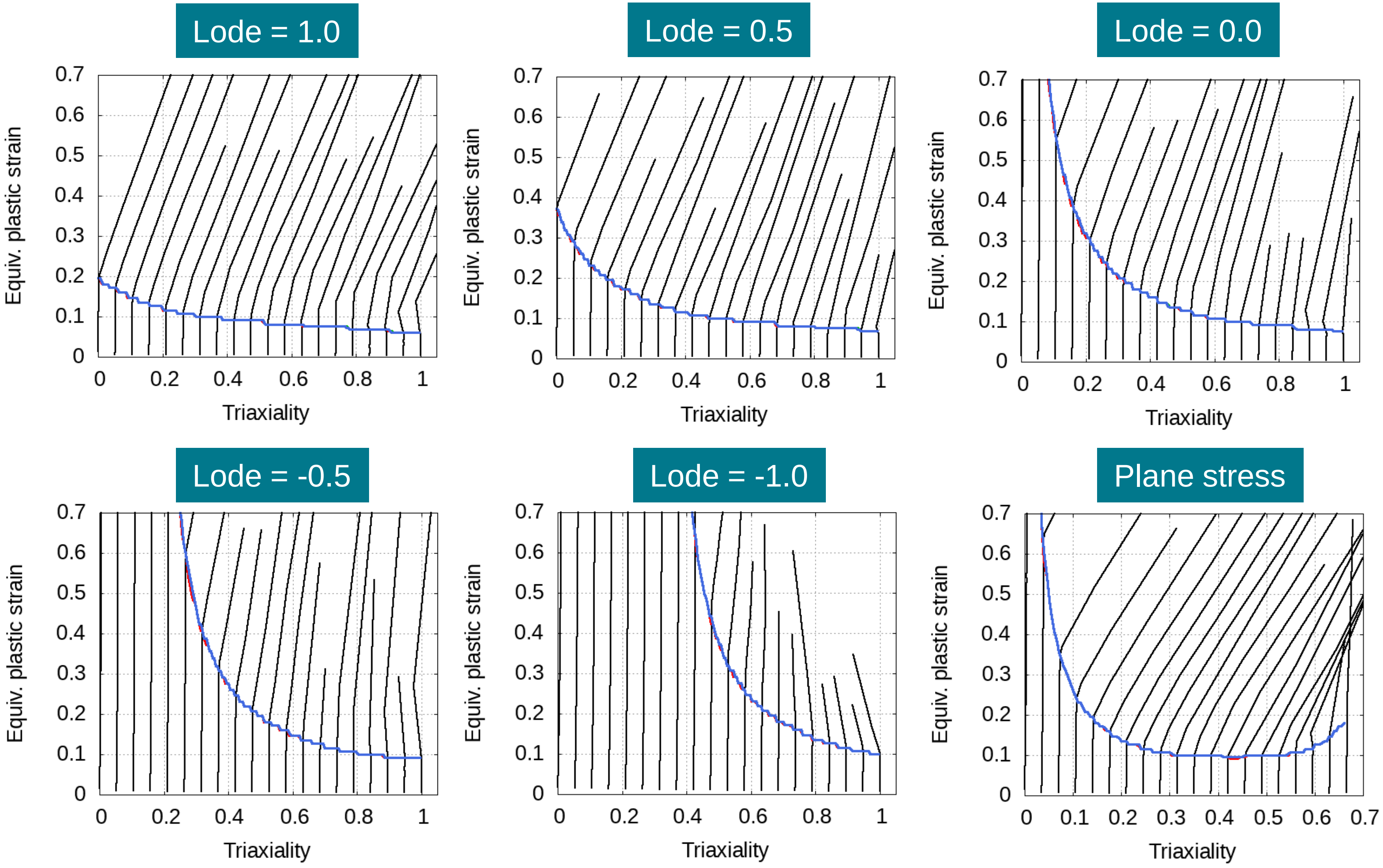


Aluminum extrusion

Comparison between simulation and analytical prediction

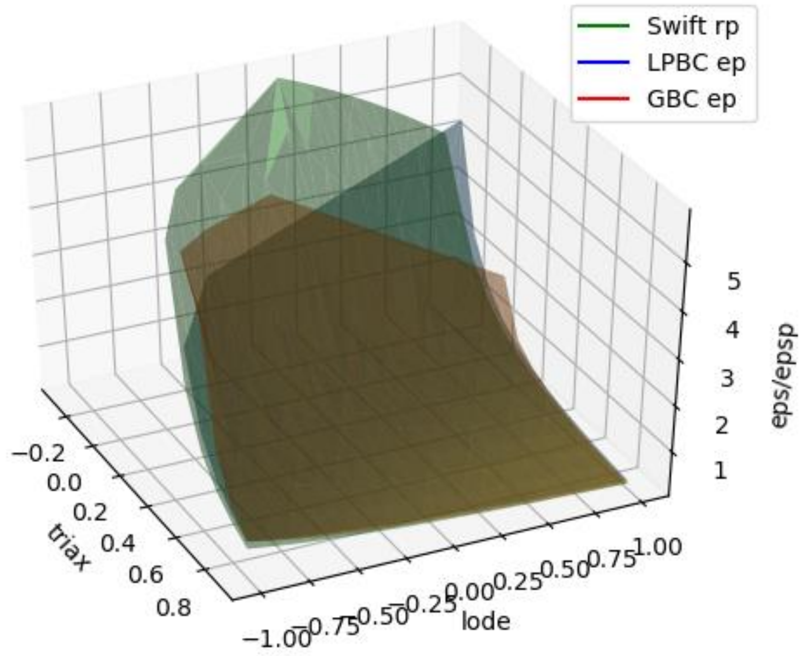


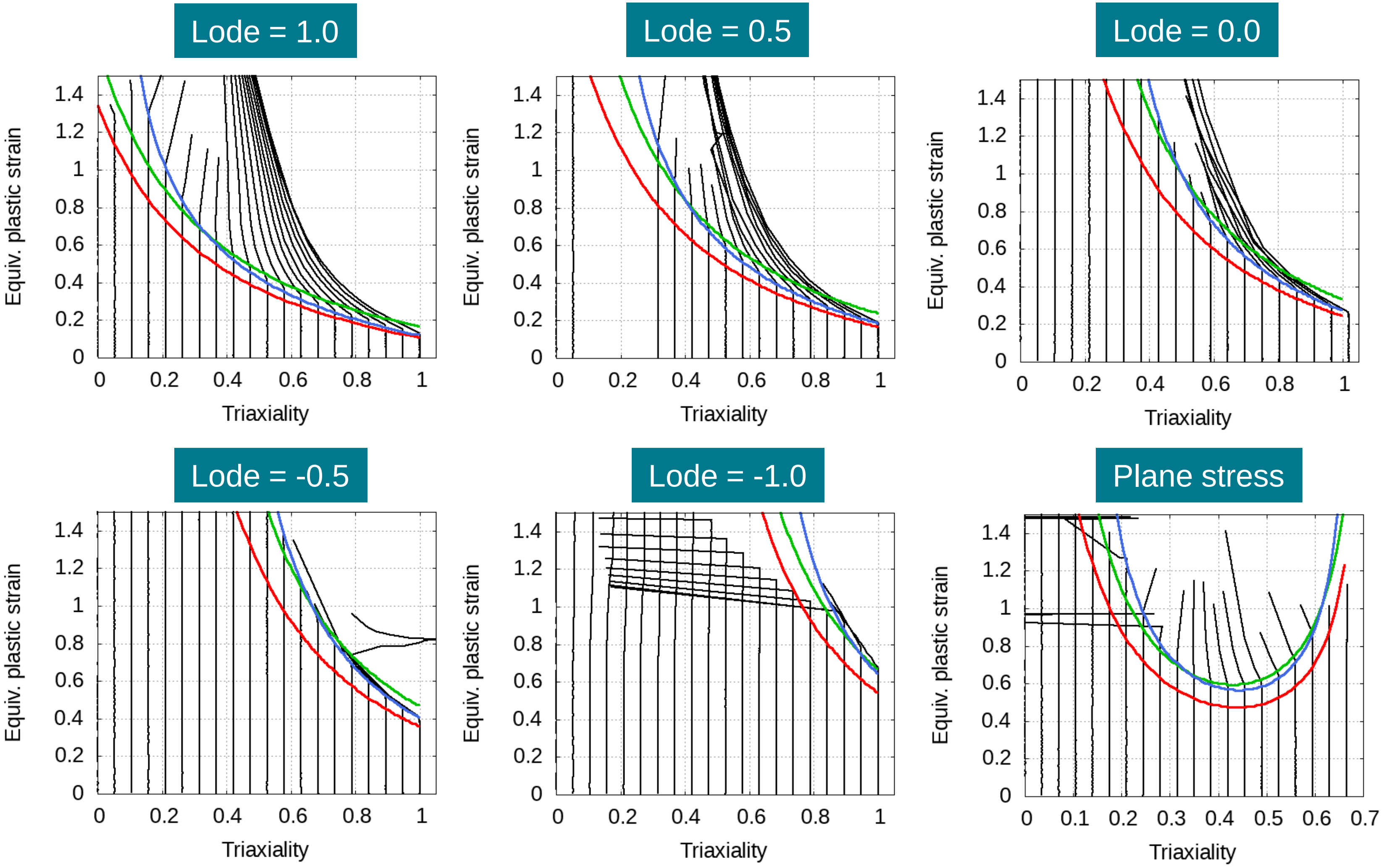




Soft material (E = 150 MPa, ν = 0.01)

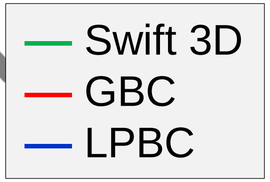
Comparison between simulation and analytical prediction



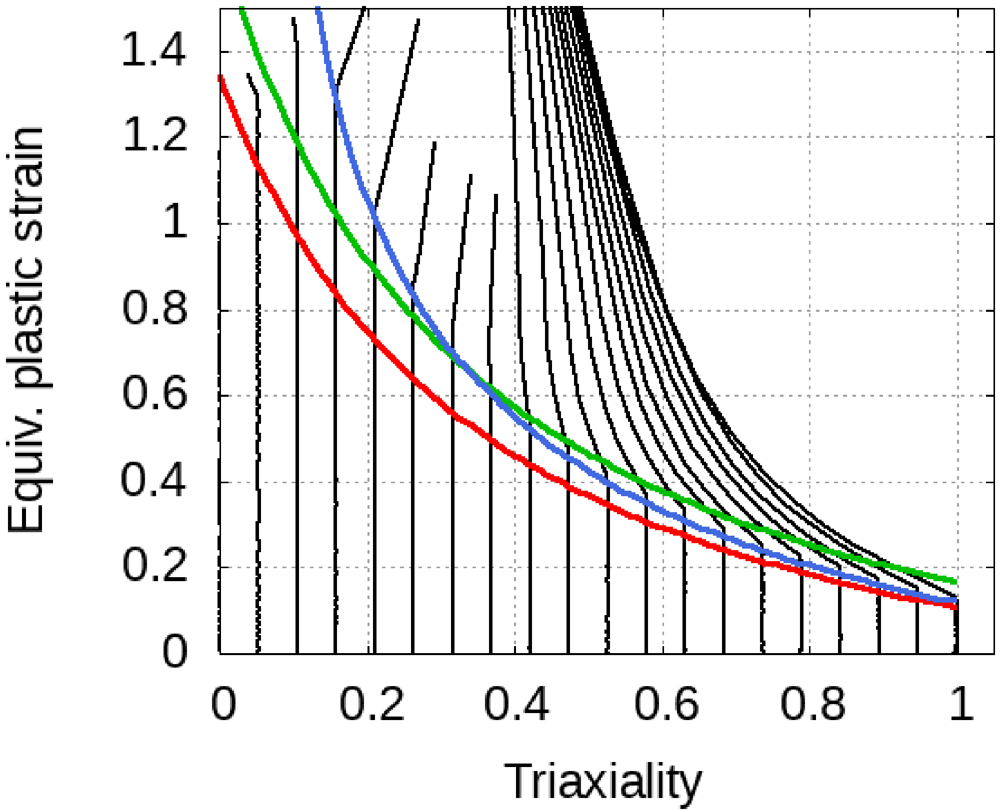


Soft material

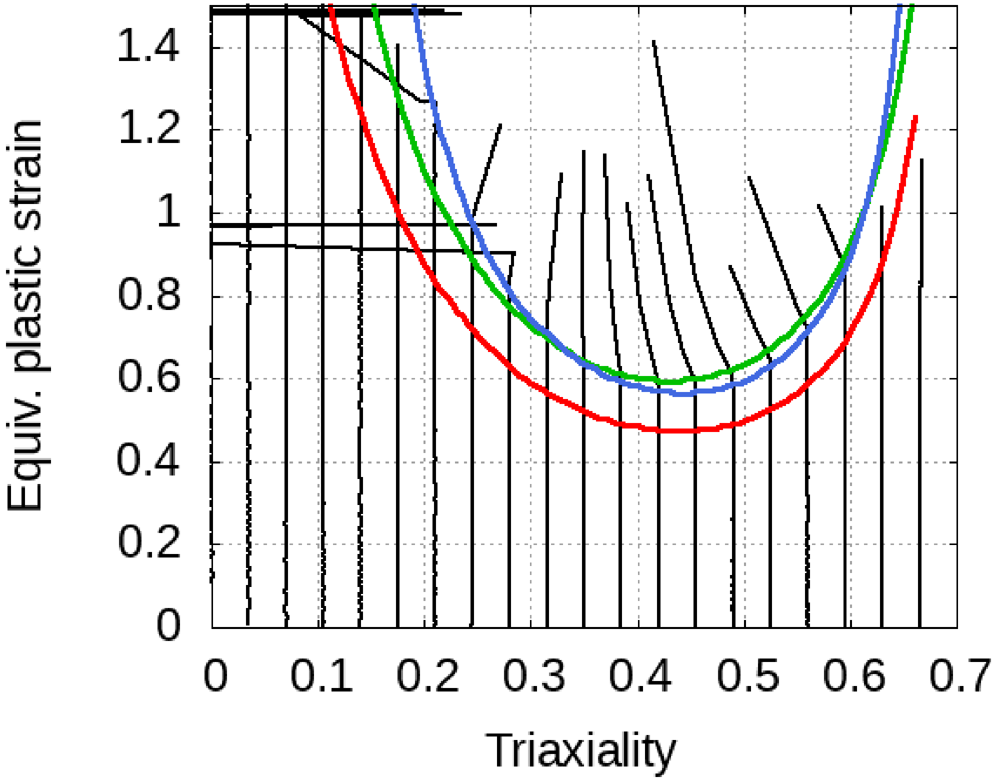
Comparison between simulation and analytical prediction



Lode = 1.0



Plane stress



Conclusions and final remarks

- “What the hell is ECRIT?”

  For the Jaumman stress rate and J2 elastoplasticity (e.g., *MAT_024 in LS-DYNA):
  → It’s LPBC, GBC or Swift if dealing with metallic materials
  → It seems to be LPBC for very soft materials

- The element block simulations can be used as a tool for the regularization as a function of the triaxiality and Lode parameter (SHRF and BIAXF flags often not enough in practical applications)

- Why is all this relevant?
  - Better understanding of mesh dependence, necking
  - Better understanding of unconventional stress states
  - New options in GISSMO (e.g., INSTF)
  - Direct application in practice, for instance, for the correct mapping from forming to crash as well as enhanced regularized failure modeling in crash simulations

Thank You

DYNAmore GmbH
Industriestr. 2
70565 Stuttgart-Vaihingen
Germany

Tel.: +49 - (0)711 - 459 600 0
Fax: +49 - (0)711 - 459 600 29
info@dynamore.de

www.dynamore.de
www.dynaexamples.com
www.dynasupport.com
www.dynalook.com

Find us on

DYNAmore worldwide
Germany ▪ France ▪ Italy ▪ Sweden ▪ Switzerland ▪ USA