OSTBAYERISCHE
TECHNISCHE HOCHSCHULE
REGENSBURG

MASCHINENBAU

Implicit simulation of highly loaded areas of a forming tool for large presses using LS-DYNA

Speaker: Philipp Thumann[1]
F. Koch[1], M. Wagner[1], B. Suck[2], J. Meinhardt[2], S. Marburg[3]

[1] Ostbayerische Technische Hochschule Regensburg, Germany

[2] BMW Group, Munich, Germany

[3] Technical University of Munich

Structure

1. Introduction
    a) Automotive forming tools
    b) Present research projects
2. Simulation
    a) Simulation of the complete forming tool
    b) Reduced simulation model (stand-alone)
    c) Substructure modelling
3. Conclusions

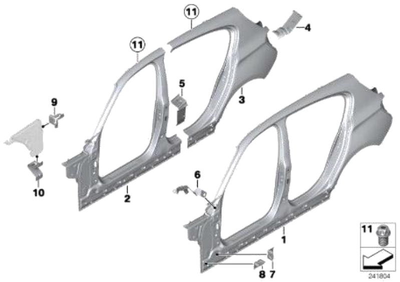
Introduction
Automotive forming tool

Facts about the investigated automotive forming tool

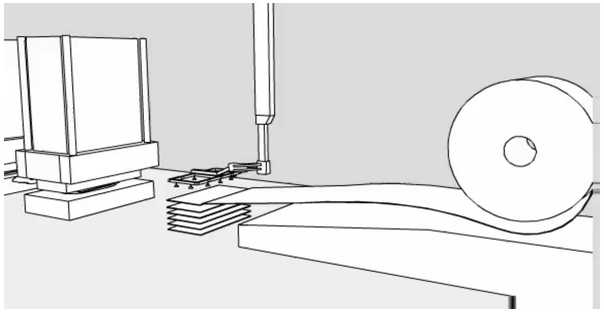
- Dimensions: 4580 x 2370 x 1380 [mm]
- Weight of complete tool: ca. 90 tons

  Weight of moved parts: up to 25 tons
- Output: sidewall framework

Source: http://www.bmw-arnold.com/miniaturen/



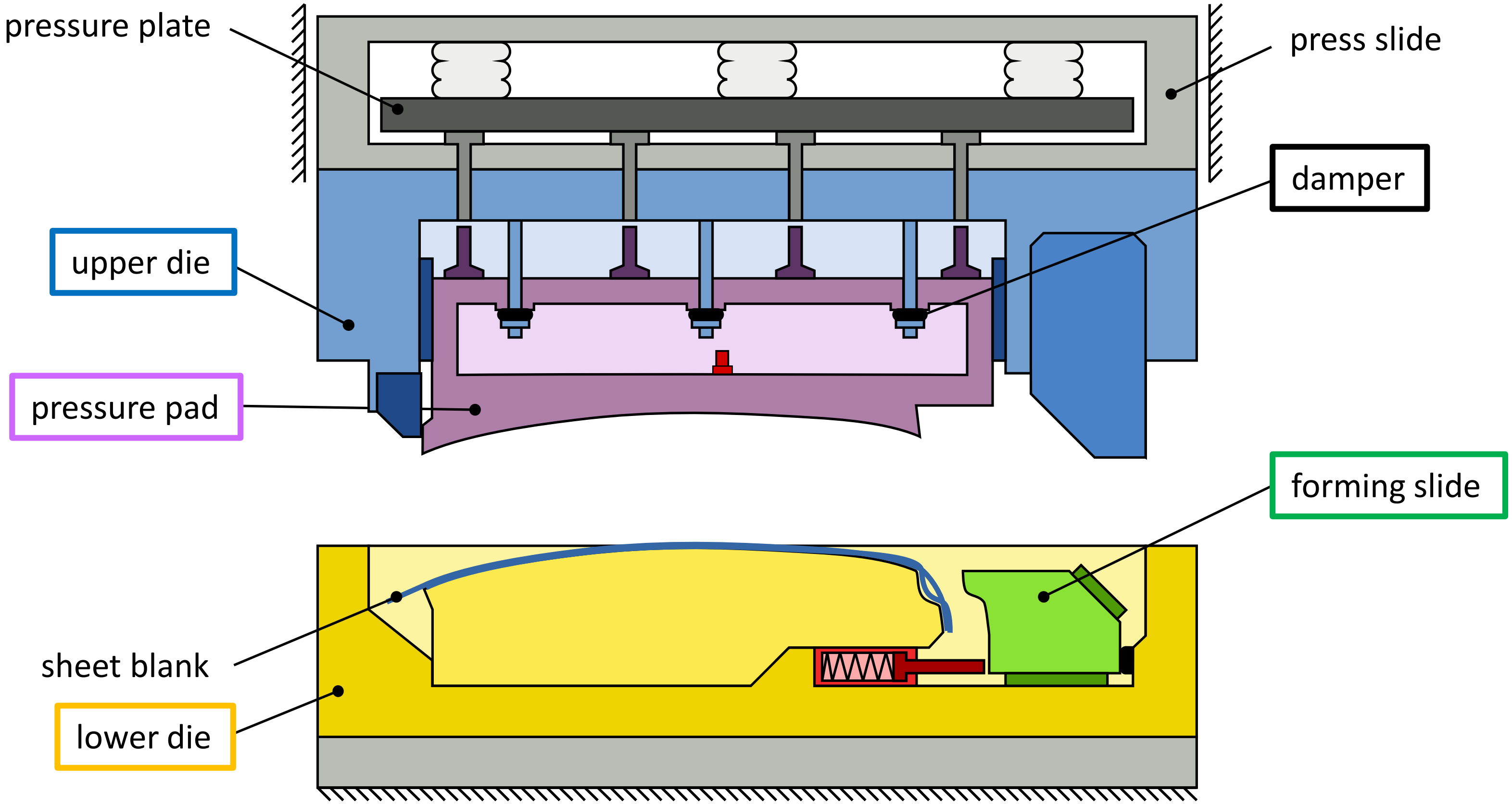
Source: http://www.leebmann24.com/



Source: http://www.bmwgroup.com/

Introduction
Automotive forming tool

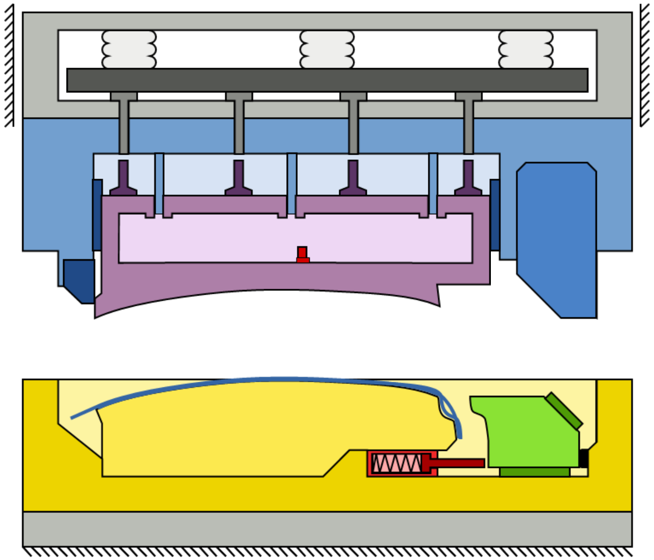
General function of automotive forming tool



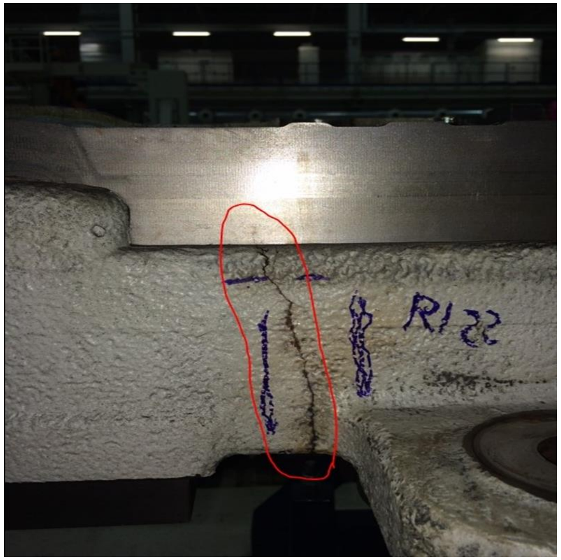
Source: Presentation, K. Swidergal, LS-DYNA Forum 2014

Every tool is unique

Damages at particular tool components



Source: Presentation, K. Swidergal, LS-DYNA Forum 2014



Source: Presentation, B. Suck, BMW Group, München, 21.09.2015

→ e.g. pressure pad

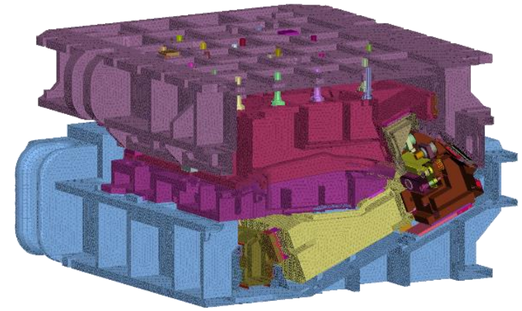
Durability analyses are necessary!
→ input data from implicit FE-analysis

OSTBAYERISCHE
TECHNISCHE HOCHSCHULE
REGENSBURG

MASCHINENBAU

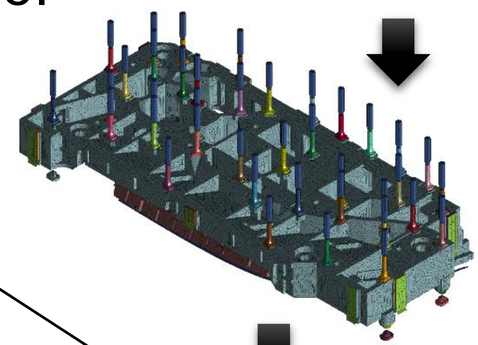
Introduction
Present research projects

Modelling and simulation of a complete tool



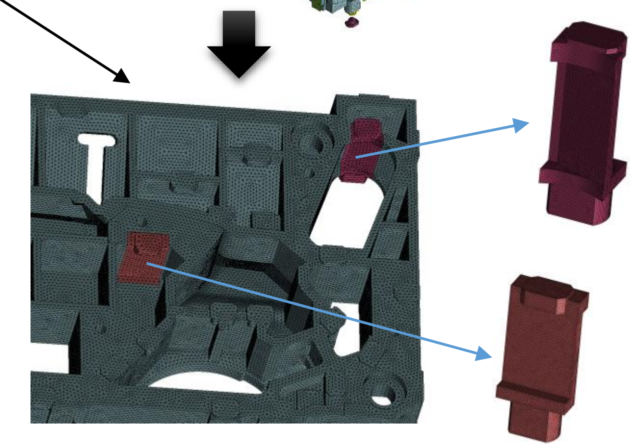
Source: Presentation, K. Swidergal, München, 21.09.2015

Modelling and simulation of a complete tool
+
Simulation of a stand alone model
+
Integration of substructure modelling



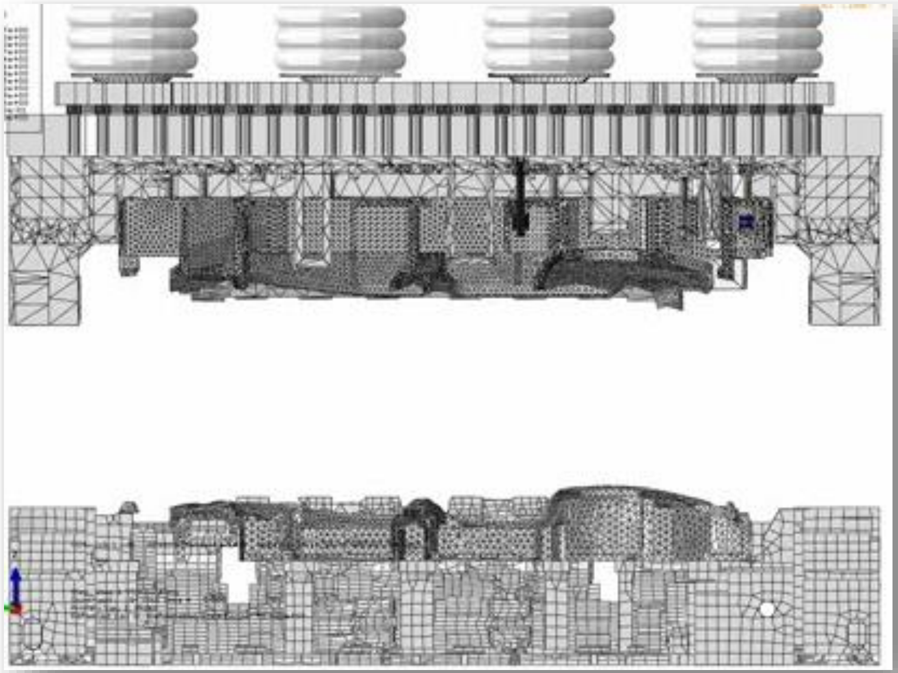
Source: Presentation, F. Koch, Regensburg, 01.07.2016

Using knowledge from previous projects in modelling of another forming tool
+
Special durability tests and analyses (impulsive loads)



Dynamic behaviour of an automotive forming tool for large presses

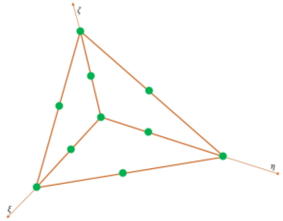
- Modelling of a complete forming tool for the LS-DYNA solver, Literature
  - Swidergal, K., Structural analysis of an automotive forming tool for large presses using LS-DYNA, 10th European LS-DYNA Conference, Augsburg, 2015
  - Swidergal, K., Modeling and simulation of carbon black filled elastomer damper using LS-DYNA, LS-DYNA Forum, Bamberg, 2014



Source: Presentation, K. Swidergal, LS-DYNA Forum 2014

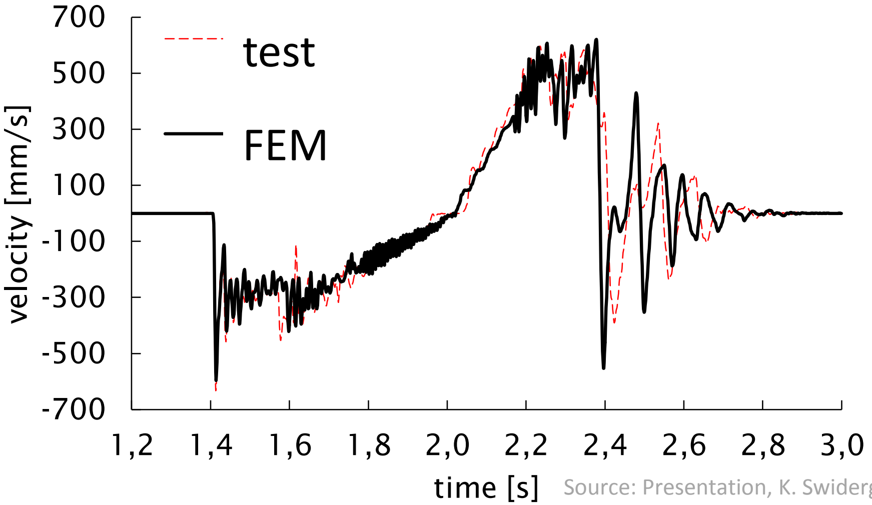
Dynamic behaviour of an automotive forming tool for large presses

- Explicit time integration → explicit solver
  Advantage: For high dynamic investigations
  Disadvantage: Poor in calculations with higher-order elements



Source: Koch, F., Model setup and FE-simulation of a forming tool component, project report, Regensburg, 2016

- Investigation of dynamic behaviour
  → Validation of simulation with measurements
- Output: contact forces and node displacements



Source: Presentation, K. Swidergal, LS-DYNA 2014

Simulation

Reduced simulation model (stand-alone)

Investigated component

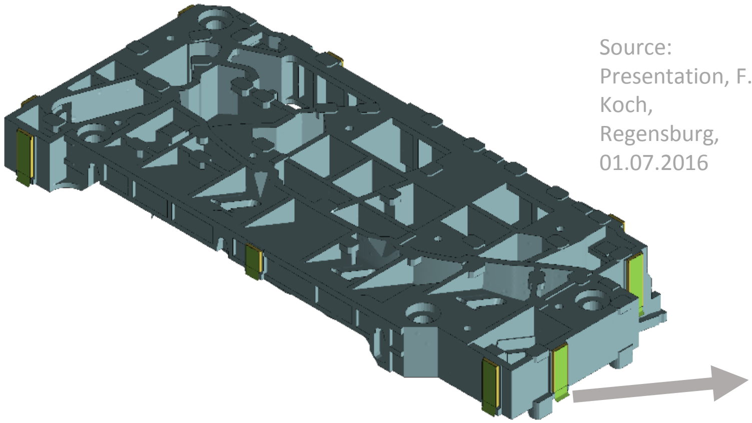
→ Pressure pad

Additional parts

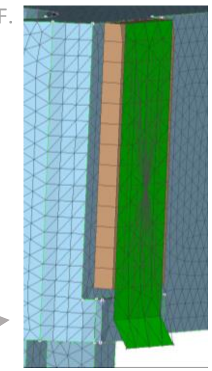
→ Sliding guides and delimiter

→ xy-area

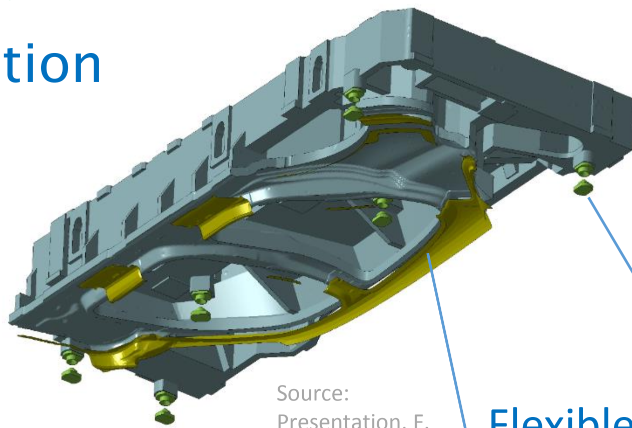
→ z-direction



Source: Presentation, F. Koch, Regensburg, 01.07.2016



Rigid body 2D elements for boundary in xy-area
→ contact pair with sliding guides



Source: Presentation, F. Koch, Regensburg, 01.07.2016

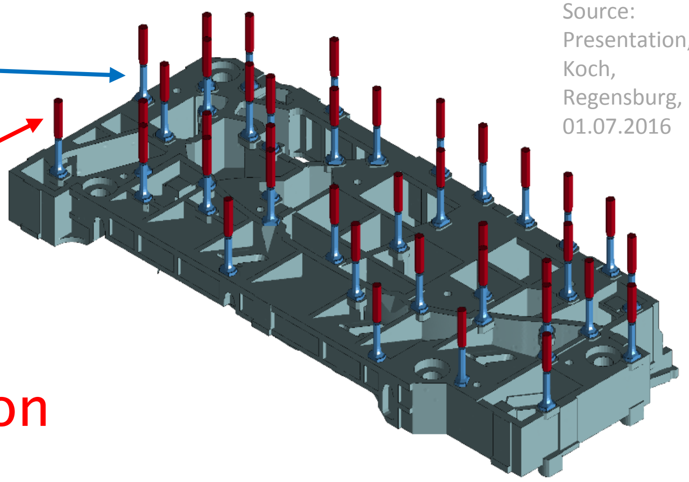
Flexible distance bushs for boundary in z-direction
→ contact pair with flexible counterpart

Rigid body 2D die surface for limitation in z-direction
→ contact pair with flexible surface of pressure pad

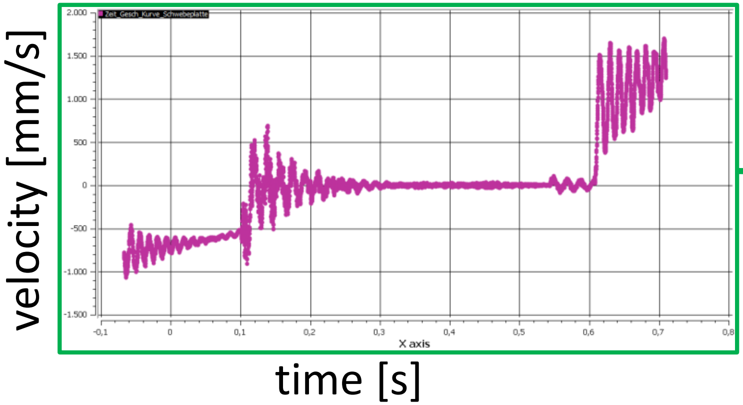
Further additional components in the stand-alone model

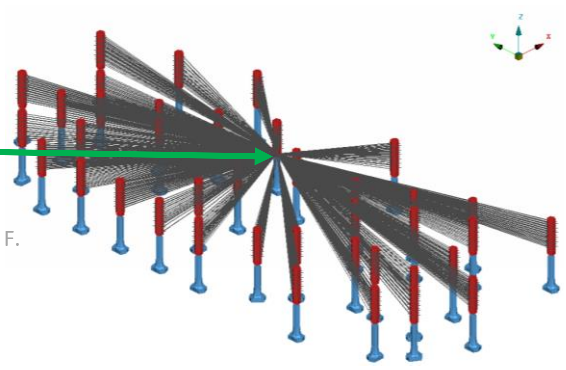
- Pressure pins (tool side)
  → fixed to the pressure pad

- Pressure pins (press side)
  → loaded with the a boundary condition



Source: Presentation, F. Koch, Regensburg, 01.07.2016





Source: Presentation, F. Koch, Regensburg, 01.07.2016

Simulation

Reduced simulation model (stand-alone)

Step 1 – export of the parts from the global model

- explicit solver
- element size according to the global model (ca. 25 mm)
- Location adaptation of parts regarding to the z-position

Step 2 – iterative modelling and settings

- *DAMPING_PART_MASS
- varying the time step size, mass scaling
- various contact adjustments
- mesh size 15 mm
- higher order elements

Final version

- linear elements
- mass scaling DT2MS=-7.2E-7
- CPU: 12 Cores, 3.4 GHz RAM: 128 GB
- calculation time: 22 h

Why explicit, not implicit?

1. RAM memory is not enough!!!
2. Negative volume of solid elements → **Error Termination**

Counteraction

Guidelines for implicit analyses from March 2016

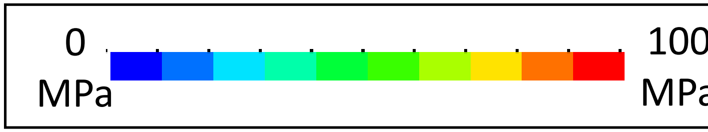
→ solver remains in this state:

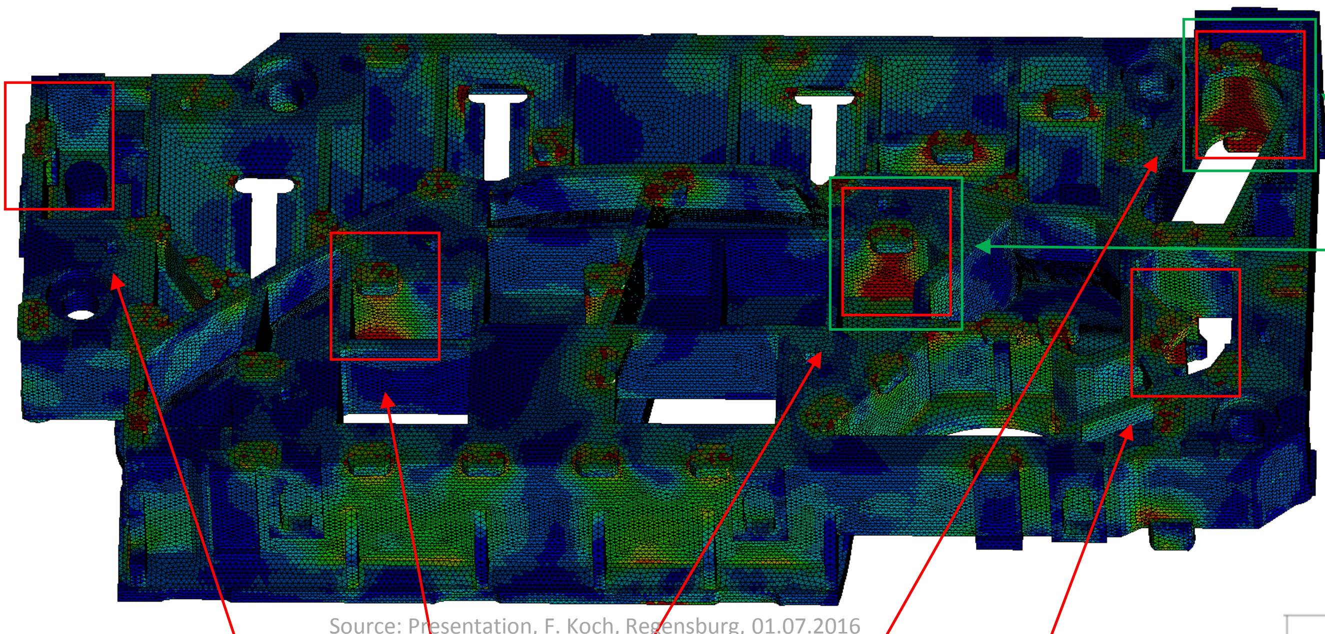
```
Memory required for implicit mtx strg  :   1011330174

        stiffness matrix data
        ---------------------------------------------------------------
                         number of equations =   5281128
                      stiffness coefficients =     205.3 Mw

        Memory Requirements:                     incore
                    TOTAL for linear algebra =    447.6   Mw
                    TOTAL for entire job     =   1458.9   Mw
                    TOTAL available          =  60000.0   Mw
  minimum memory for in-core stiffness factorization =   1458.9 Mw
                          Initialization CPU =  0.000E+00 seconds
```

Simulation

Reduced simulation model (stand-alone)

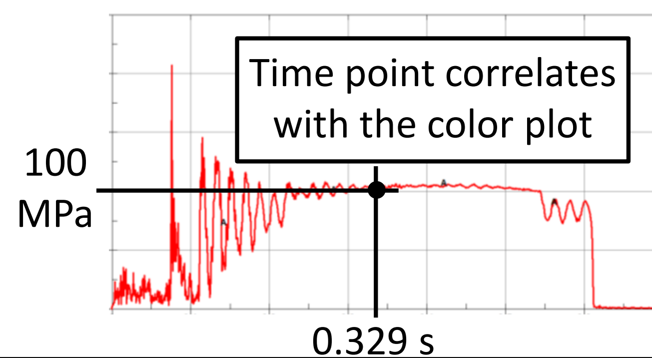
Simulation results (von Mises stress)





Maximum loaded areas

Substructure modelling



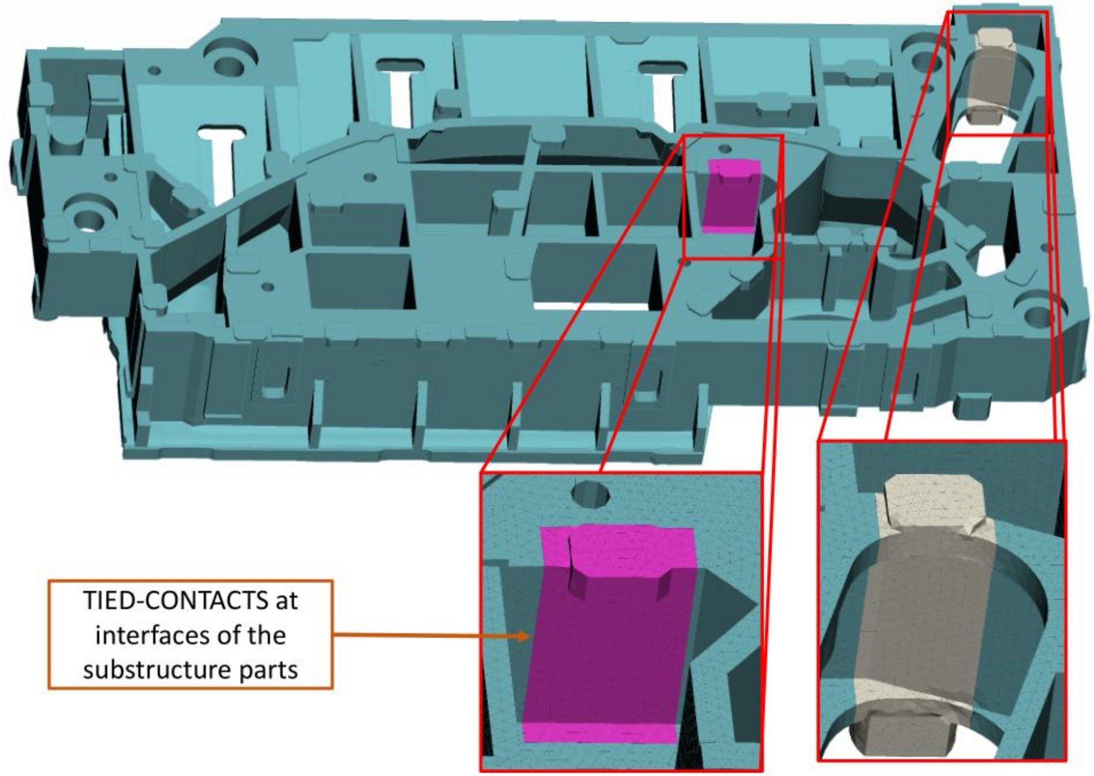
Knowledge:
Maximum stresses occur in the area with contact conditions from above and below, i.e. pressure pins and distance bushs

How to handle substructure modelling?

1. Define the area of interest (maximum load)
2. Cut out geometry in this area
3. Replace it with an extra part
4. Define a tied contact
5. Output surface nodal displacements from the stand-alone model
6. Input surface nodal displacements into the substructure model as boundary conditions

Application to the pressure pad

Explicit Simulation of the stand-alone model with two sub-parts



Source: Koch, F., Model setup and FE-simulation of a forming tool component, project report, Regensburg, 2016

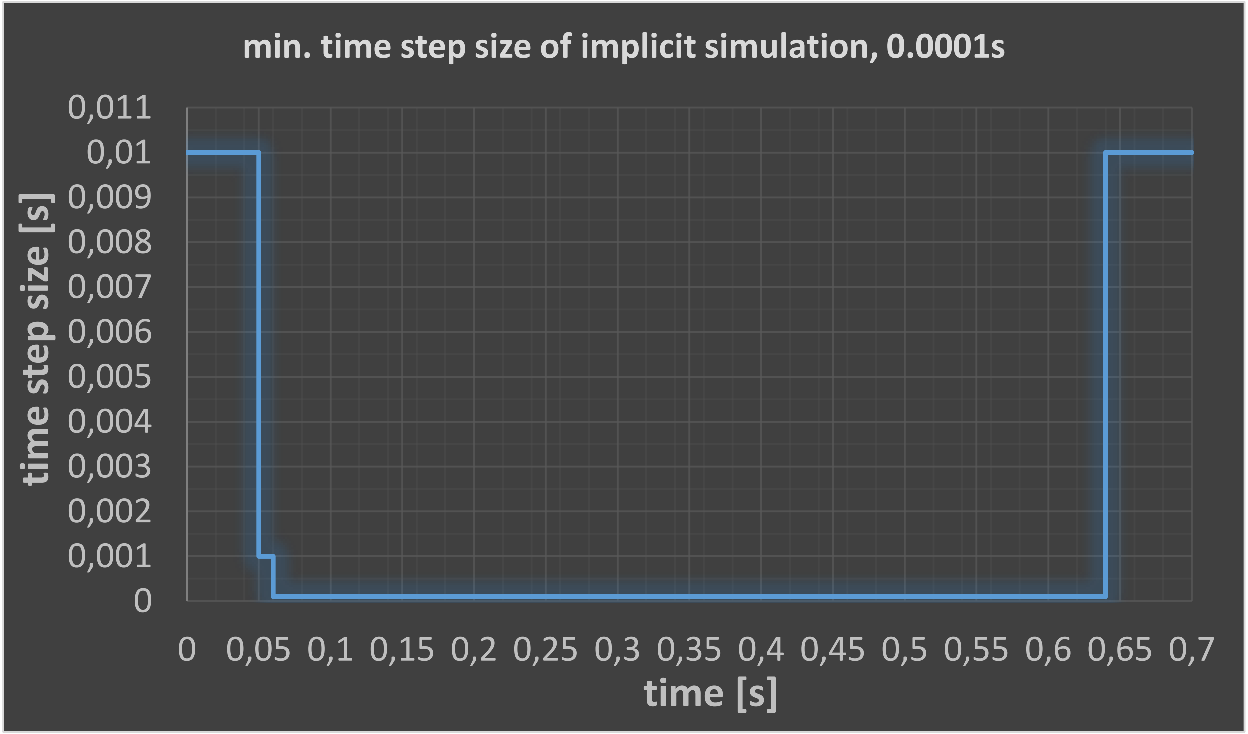
Tetrahedron elements, ca. 4mm quadratic element formulation, ELFORM=16

Determination of the maximum stresses

For 2 sub-parts → 1 calculation with manual time step control

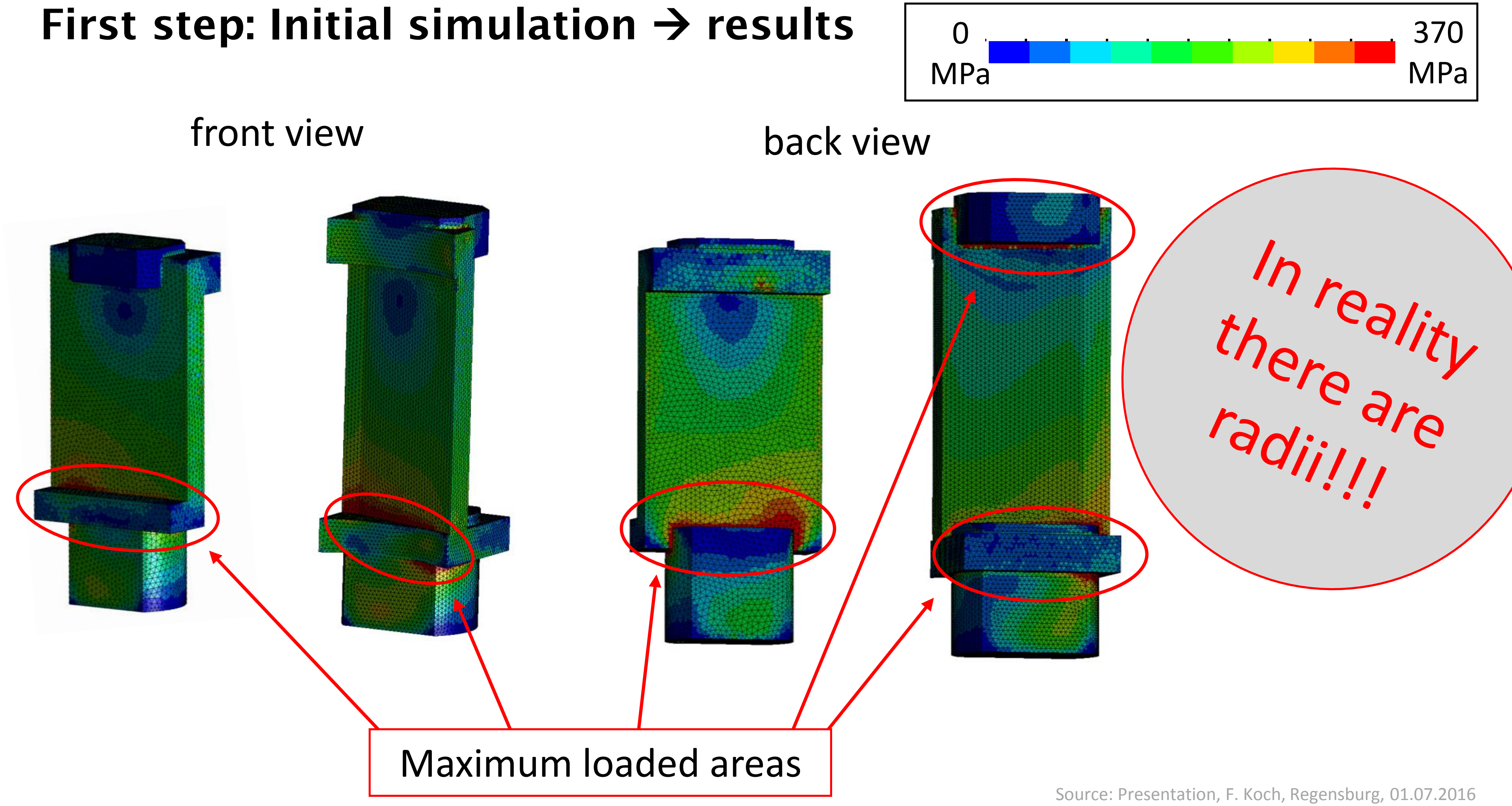
Implicit time integration;
Tetrahedron elements, ca. 4 mm;
Quadratic element formulation ELFORM=16

CPU: 12 cores;
RAM: 128 GB;
calculation time: 49 h



Source: Presentation, F. Koch, Regensburg, 01.07.2016

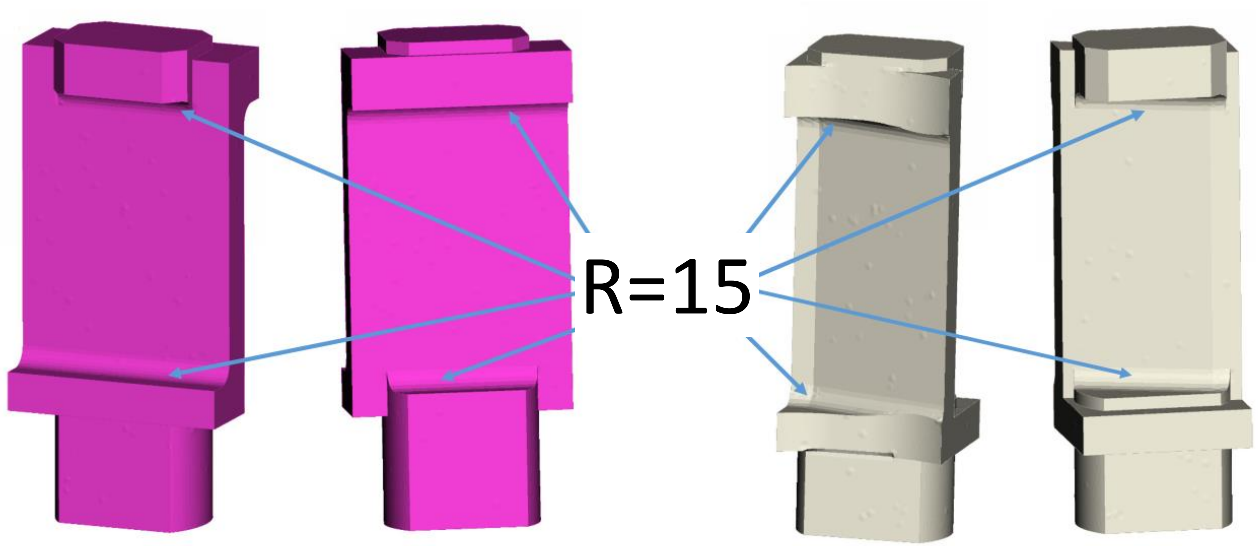
First step: Initial simulation → results



Source: Presentation, F. Koch, Regensburg, 01.07.2016

Second step

1. Add radii at the highly loaded notches

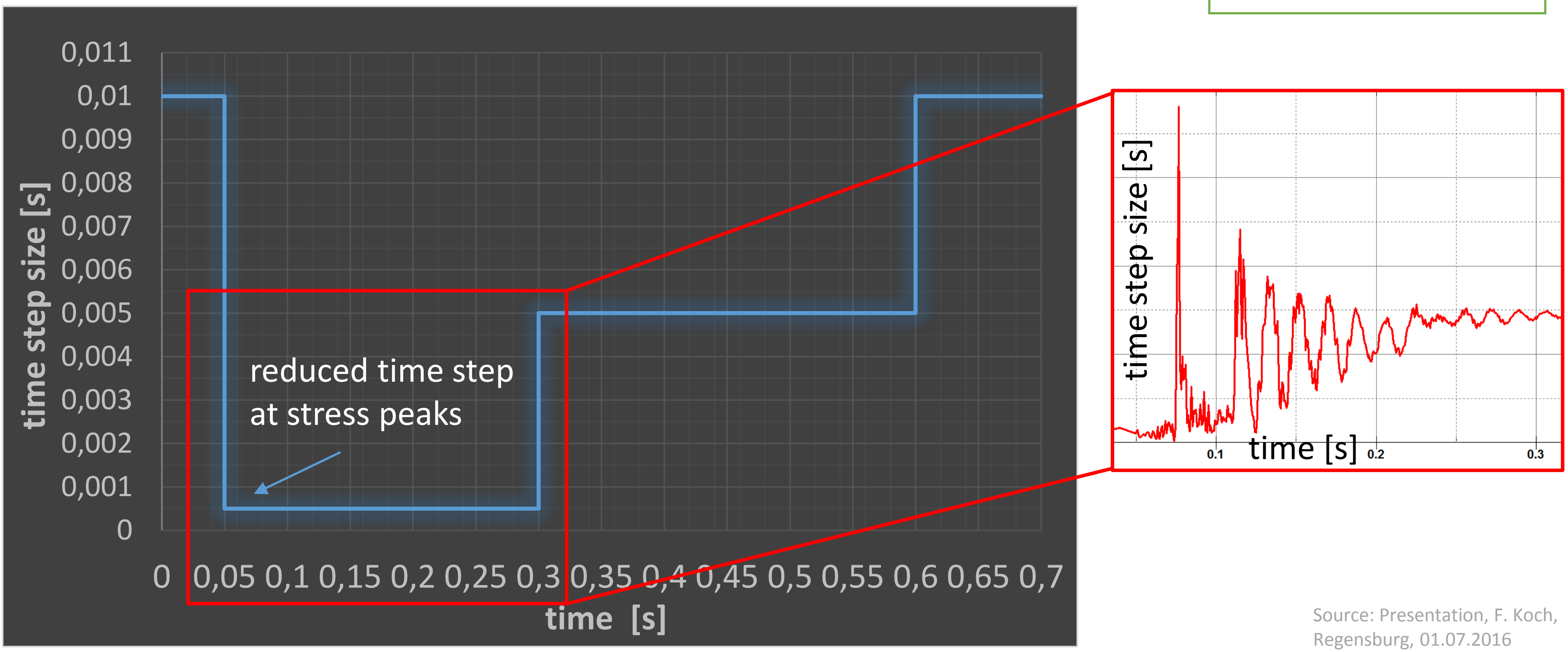


Source: Koch, F., Model setup and FE-simulation of a forming tool component, project report, Regensburg, 2016

Second step

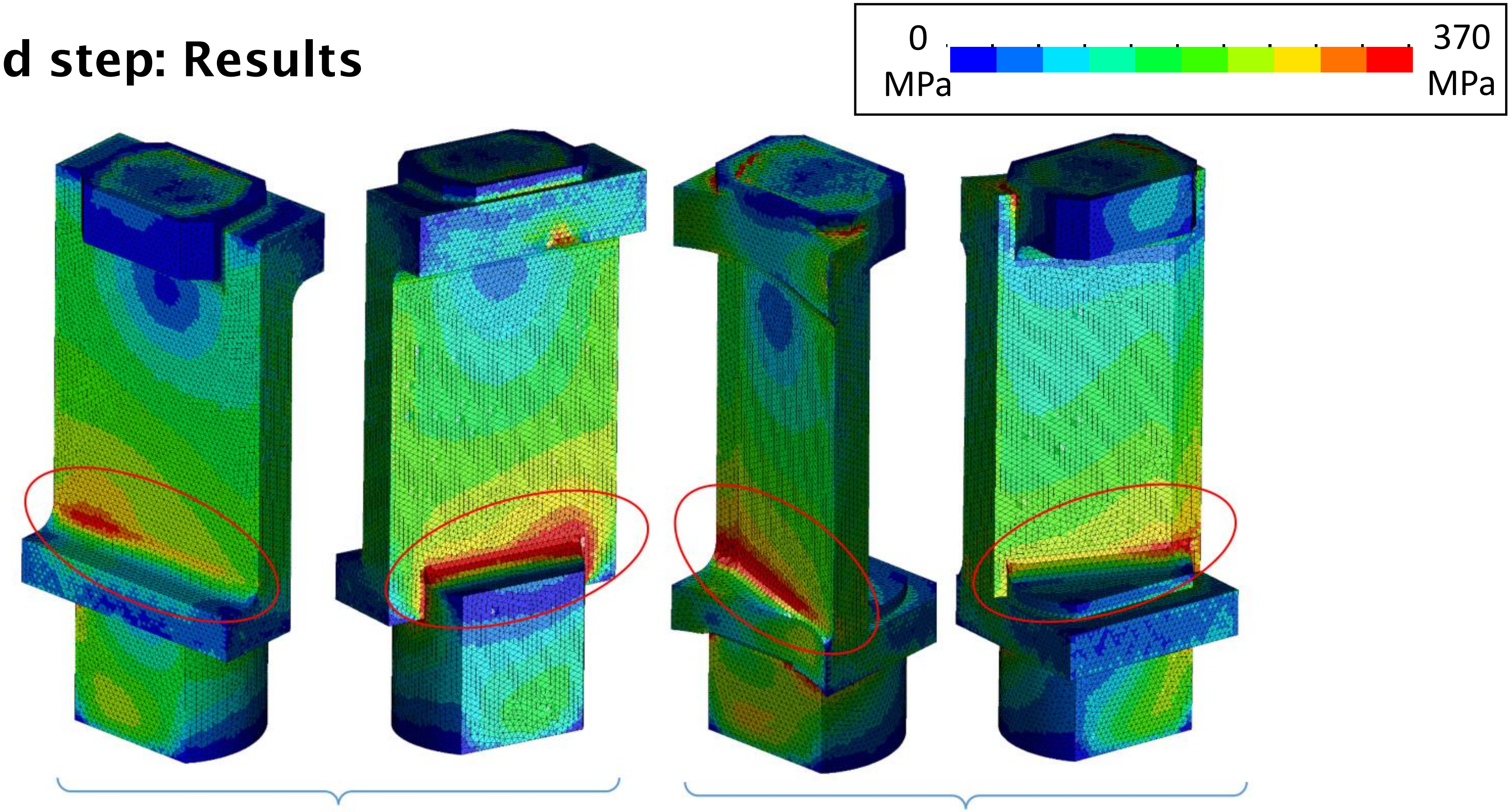
2. Increase the minimum time step size; only reduction during the maximum load

CPU: 12 cores;
RAM: 128 GB;
calculation time: 32 h



Source: Presentation, F. Koch, Regensburg, 01.07.2016

Second step: Results

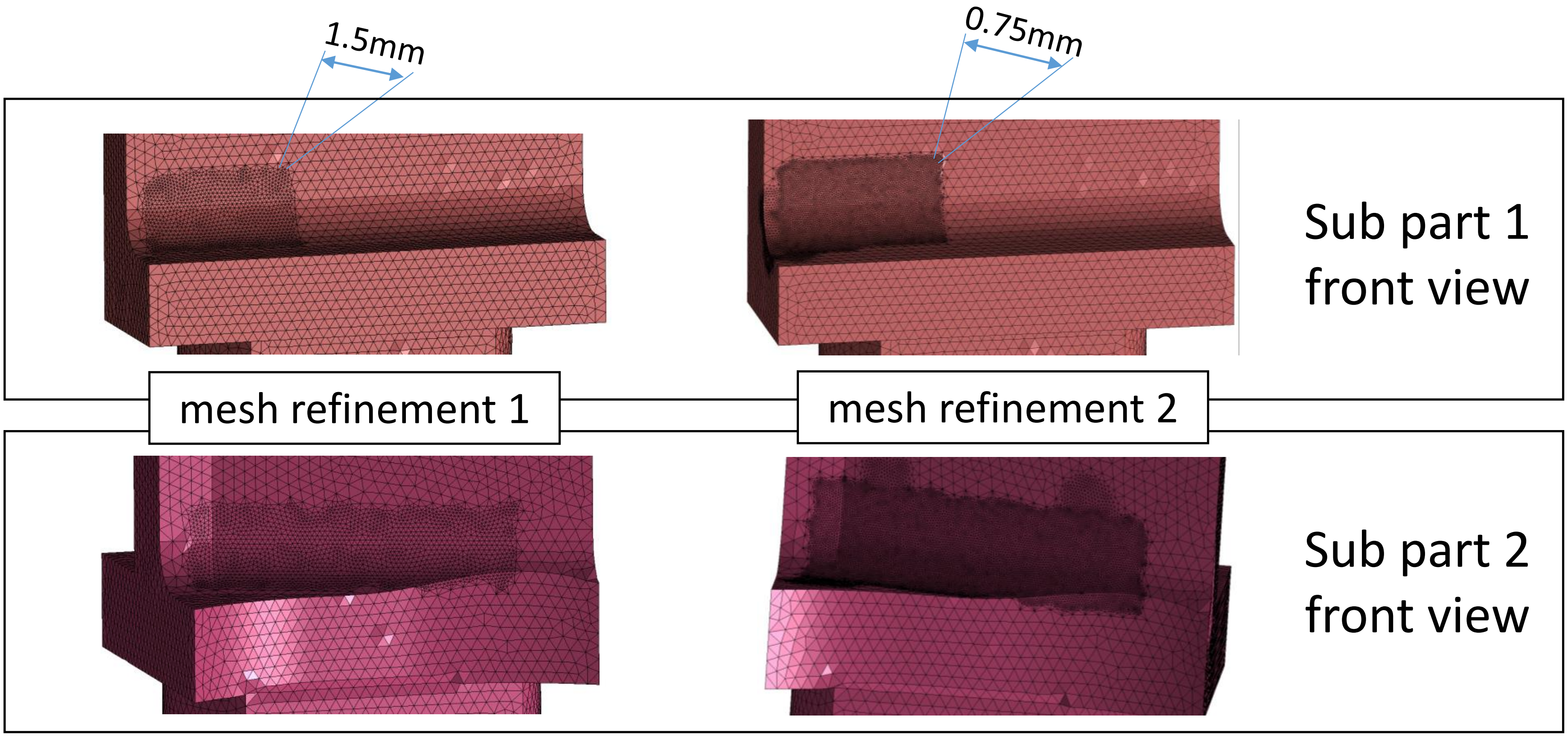


sub part **1** with radii in front and back view

sub part **2** with radii in front and back view

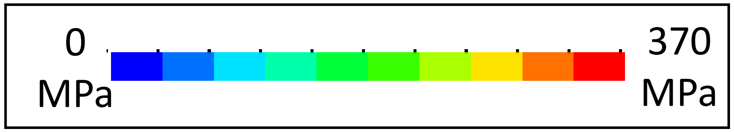
Source: Presentation, F. Koch, Regensburg, 01.07.2016

Third step: Mesh refinements

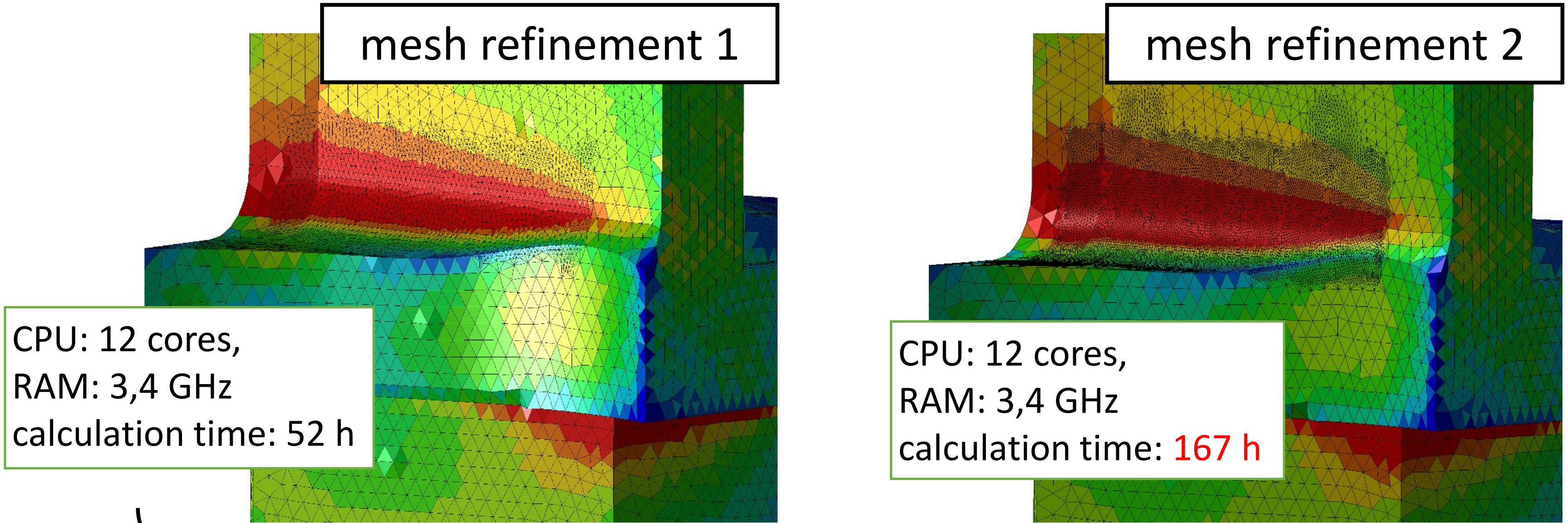


Source: Presentation, F. Koch, Regensburg, 01.07.2016

Third step: Mesh refinements → results



E.g. sub part 2, front side

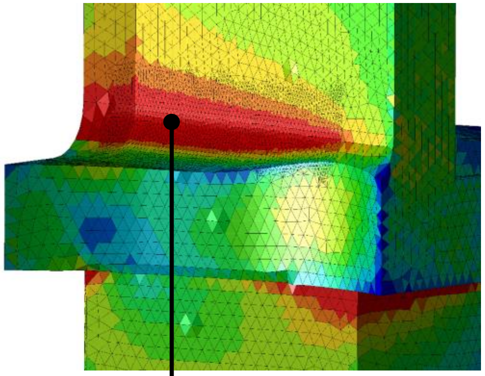


Source: Koch, F., Model setup and FE-simulation of a forming tool component, project report, Regensburg, 2016

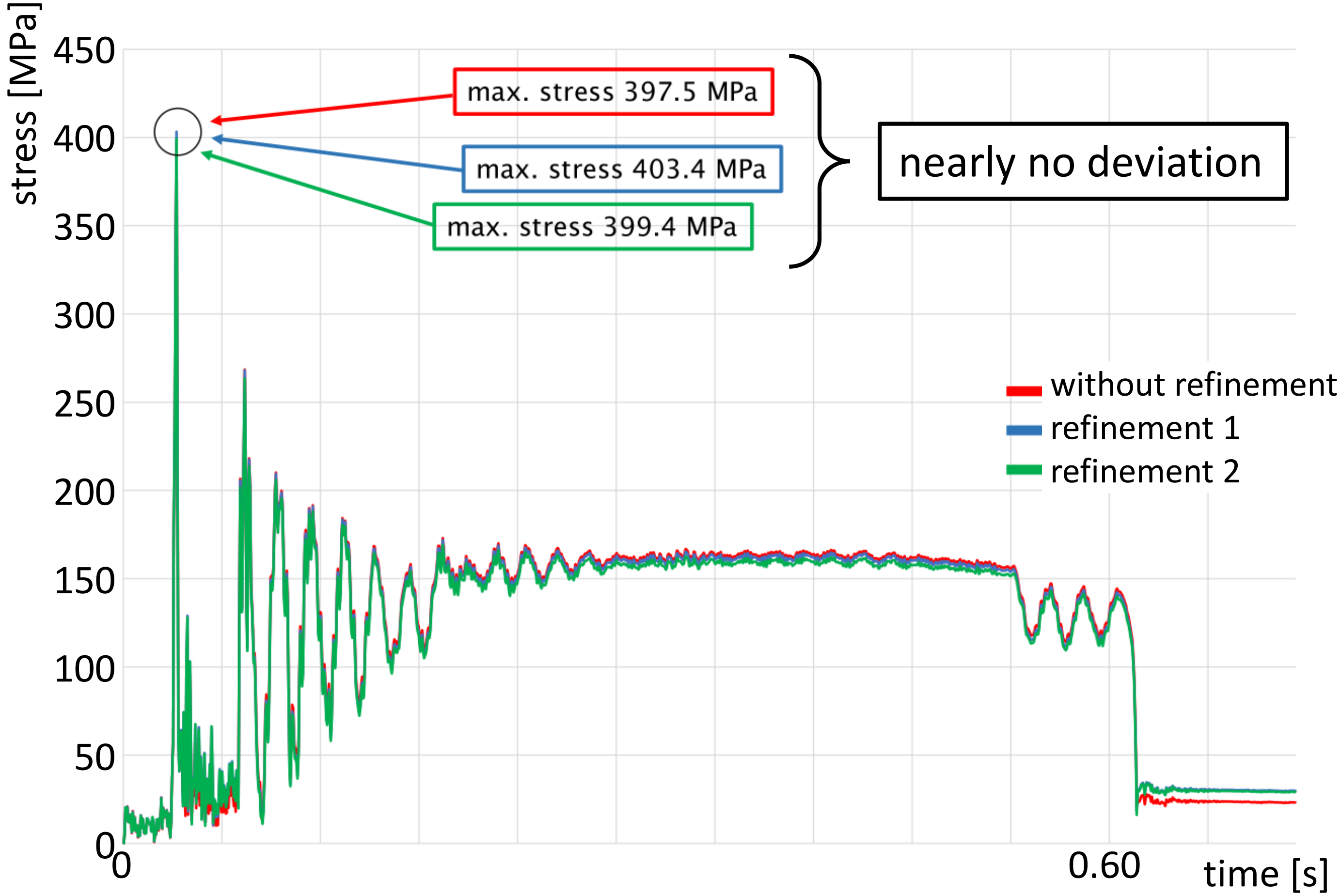
No visual change in the stress distribution

Third step: Mesh refinements → results

Sub part 2



Location of elements with maximum load



Source: Koch, F., Model setup and FE-simulation of a forming tool component, project report, Regensburg, 2016

Conclusions

No possibility to solve the stand-alone model with implicit solver.

It is necessary to simulate separately the

- global model
- stand-alone model
- substructure model

to get data for subsequent durability analyses.

→ For this a procedural method was developed.

No need to refine the mesh in substructure modelling.

Thank you for your attention