BAM

# Finite element mesh design of a cylindrical cask under puncture drop test conditions


Uwe Zencker, Mike Weber, Frank Wille

BAM Federal Institute for Materials Research and Testing
12200 Berlin
Germany



Looking to the future PATRAM 2010

**October 3-8, 2010, IMO, London, UK**

## Finite element mesh design of a cylindrical cask under puncture drop test conditions

- Background
- Puncture drop test
- Finite element model
- Mesh refinement
- Numerical results
- Validation
- Summary

# Background

### Safety analysis reports (SAR) based on numerical calculations

- BAM assesses SAR within the approval process of transport and storage casks
- increasingly base completely or partly on **numerical calculations**
  - an approximate solution is not an exact solution
- deviations of the approximate solution from the exact solution arise from
  - the **spatial discretization** of the components
  - simplifications or inaccuracies at the initial or boundary conditions, contact conditions
  - incomplete description of the material behavior of the numerous parts of the finite element model

No generally valid rule for the generation of a finite element (FE) mesh which is suitable for a numerically stable and sufficiently accurate dynamic finite element calculation.

### Mesh refinement study

- numerical approximation should approach the exact solution of the mathematical problem more and more if the mesh gets finer
  - in absence of a discontinuity or mathematical singularity!!!
- supports the convergence studies of the **Task Group on Computational Modeling for Explicit Dynamics** reporting to the ASME BPV Code Working Group on Design Methodology → experience from meshing of real casks

# Puncture drop test

## Example of a well-defined load scenario

- **1 m puncture drop test of a cask with the center of the cask wall onto a steel puncture bar according to the IAEA regulations**
- Recently an extensive drop test series was carried out with a half-scale test cask at the drop test facility of the BAM Test site Technical Safety (TTS) near Berlin.
- Tests were conducted for verification of numerical calculation methods.
- **Highly dynamic load scenario.**

## Test cask

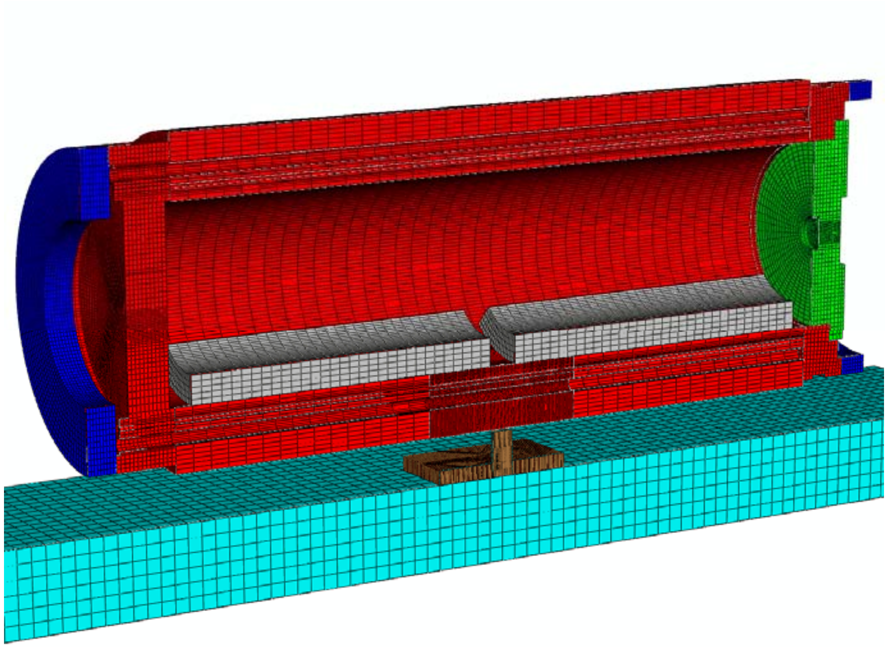
- manufactured by GNS
- cylindrical cask made of ductile cast iron
- cask wall contains two series of boreholes for moderator material *(not filled during the test)* located near the shaft
- cooling fins in the contact area removed
- equipped with strain gauges and accelerometers

Cask before puncture drop test, cooled down to a temperature of -40°C

# Finite element model

## Complete model (cask with lid system)

- cask hits with prescribed impact velocity to the puncture bar which is linked with the IAEA target
- neglected parts considered by increased mass density
- dummy masses as contents
- mesh with **refined sub-model**

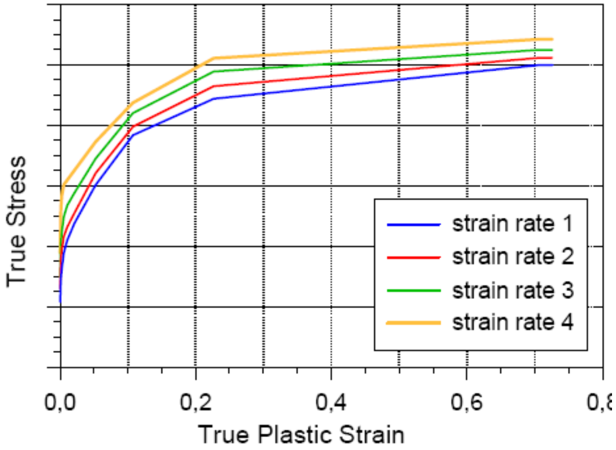


## Material model (DCI, steel)

- elastic-viscoplastic model based on measured dynamic flow curves

## Pre-calculations

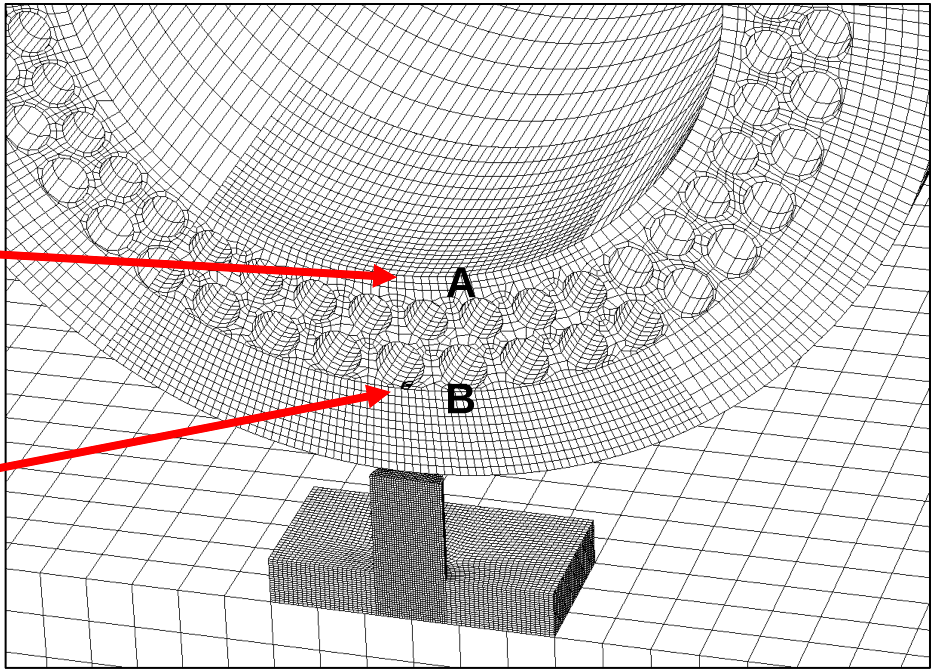
- very simplified FE model (hollow cylinder) used for testing of modeling of puncture bar and contact conditions
- friction coefficient for contact with puncture bar determined by comparing typical barrel shape of puncture bar with test result



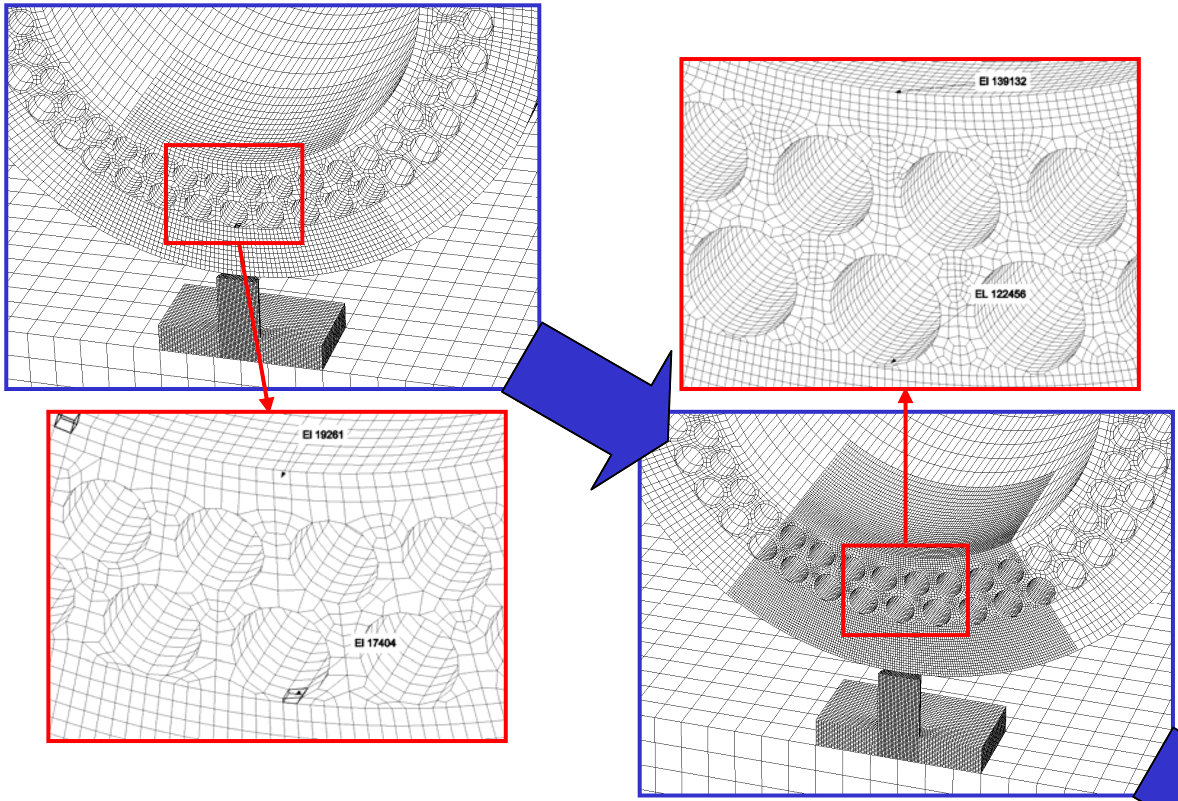
# Mesh refinement

## Pre-calculations for refined sub-model

- Convergence of numerical calculation results was checked
  - by variation of the finite element mesh near the contact area between puncture bar and cask body
  - with **simplified model**.
- **Shaft surface** is of special interest because the strains can be measured there with strain gauges very well.
- High stresses are expected near the **moderator boreholes** due to the geometrical conditions.
- Two positions were inspected:
  - intersection point of the vertical puncture bar axis with the inner surface of the cylindrical cask shaft (**position A**)
  - intersection point of the vertical puncture bar axis with the outmost element of the outer series of the moderator boreholes (**position B**)

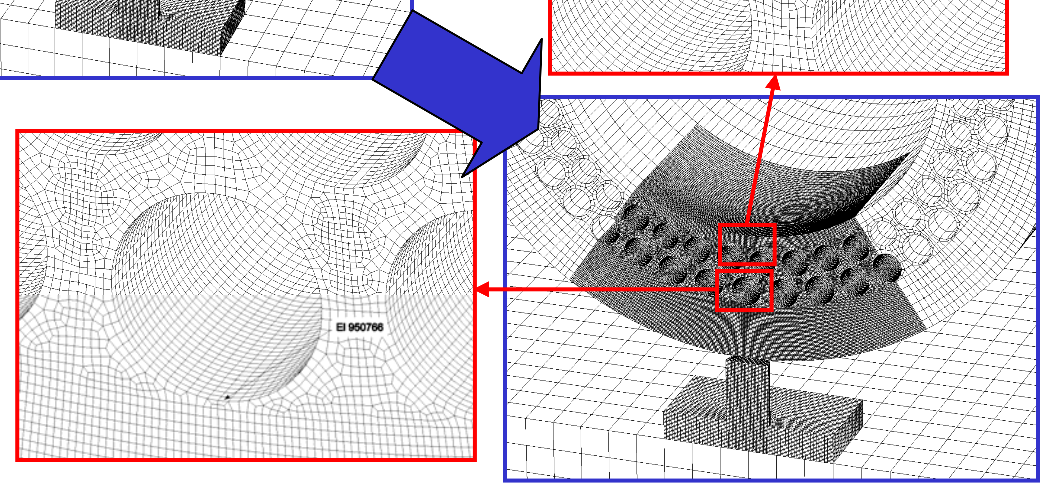


# Mesh refinement (cont'd)




## Mesh refinement strategy

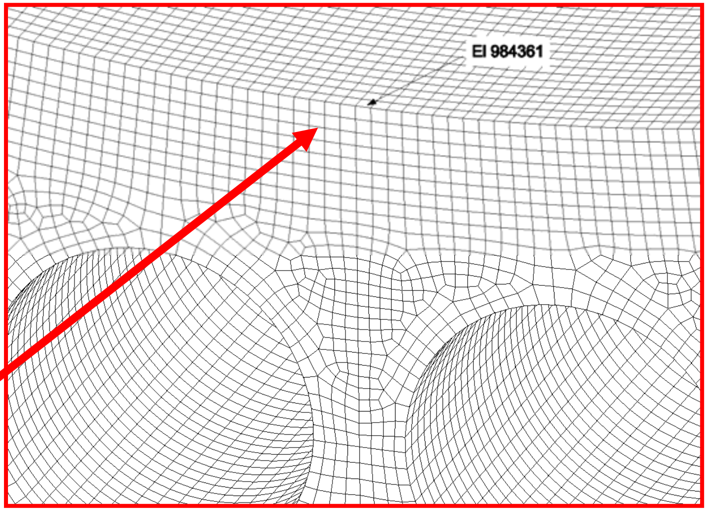
- a coarse mesh was created
- mesh was refined in two steps
- size of elements was bisected in each step.




- element closest to the discussed intersection point was investigated to find a local value on the body surface
- evaluated at integration point

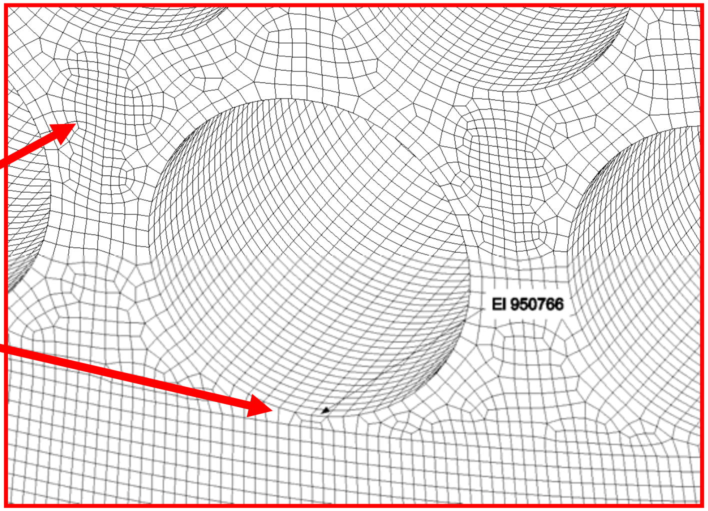
# Mesh refinement (cont'd)

- first-order (linear) interpolation elements with hourglass control used
  - stresses and strains are constant within the finite element
  - spatial course of stresses and strains in the structure is described more exactly by a higher number of smaller elements
- code: ABAQUS/Explicit™ version 6.8




- elements are almost well-shaped cubes in the region of the inner shaft.

- mesh consists not only of perfectly cubic elements in most parts of a real cask.

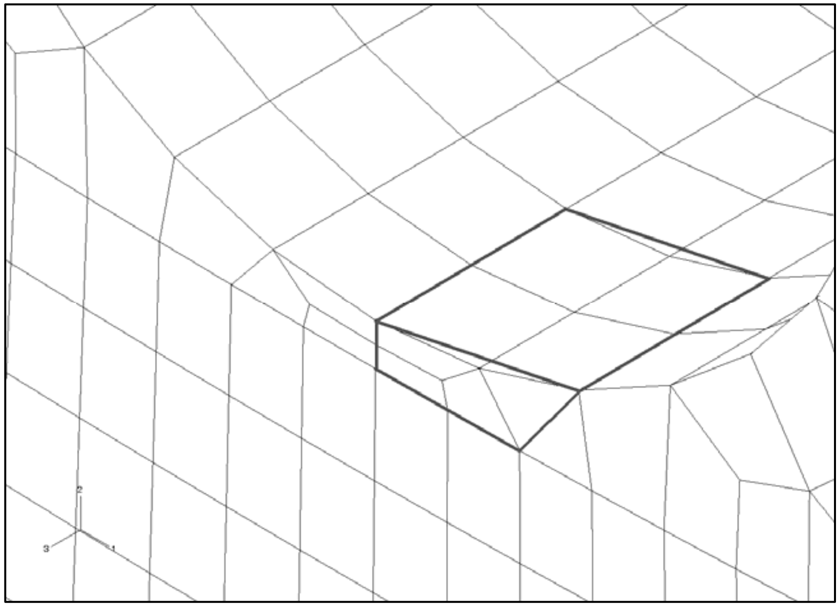



$2^{nd}$ refinement

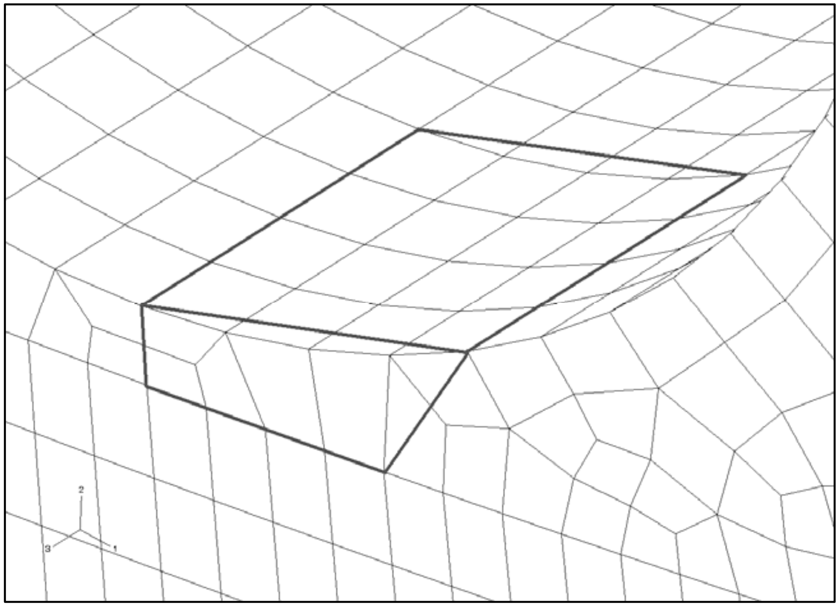
# Mesh refinement (cont'd)

## Mesh refinement strategy for local mesh nearby the moderator boreholes

- Starting element of the corresponding coarse mesh is marked.
- Bisecting the edges of the elements, the complicated geometrical shapes also must be described well.
- Element edges were bisected at the second refinement step only in two directions to reduce the total number of finite elements of the model.



1st refinement

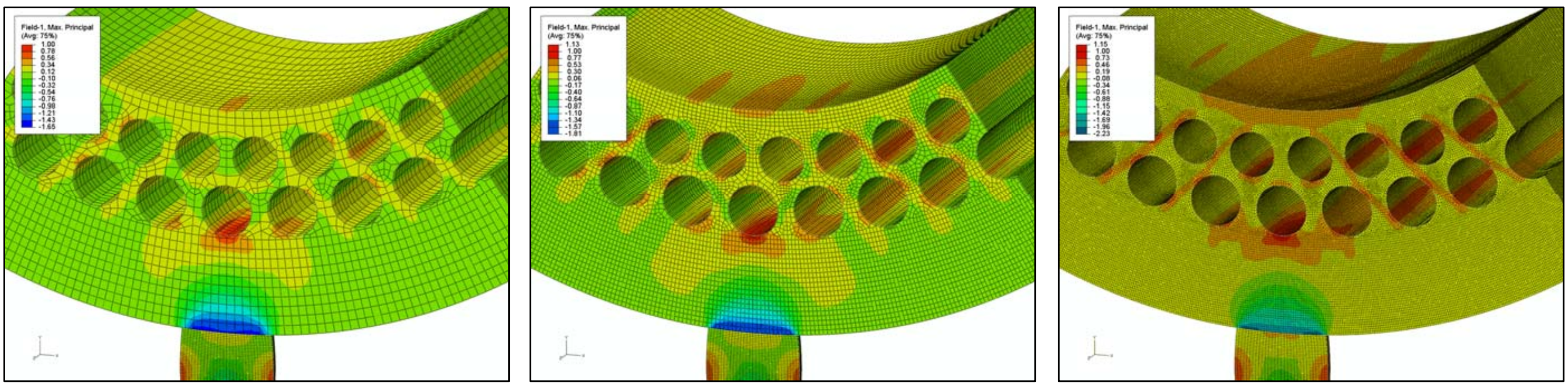


2nd refinement

### Normalized first principal stress at time of maximum pressure

- For the mesh refinement study, the cask model consisted only of the cask body to reduce computational effort.
- From fringe plots, all three FE meshes seem to be suitable.
- However, the fringe plots alone do not suffice for the assessment of the quality of the FE mesh.

→ **Investigation of convergence of the numerical solution necessary**



coarse mesh | 1st refinement | 2nd refinement

- Similar results for von Mises stress.
- Near the shaft, the behavior of the cask material is elastic.
- Localized plastic deformation occurs only in the vicinity of boreholes.

# Numerical results (cont'd)

## History of normalized first principal stress

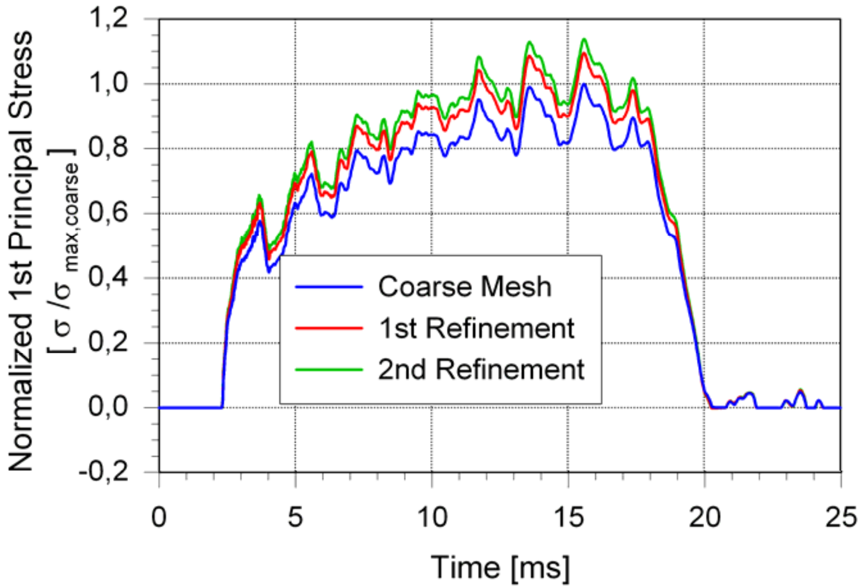
- stress normalized on the maximum stress from the calculation with coarse mesh
- impact duration is about 18 ms (smaller compared to the complete cask model)
- it can be seen that the solution is **numerically stable** and approaches a limit curve

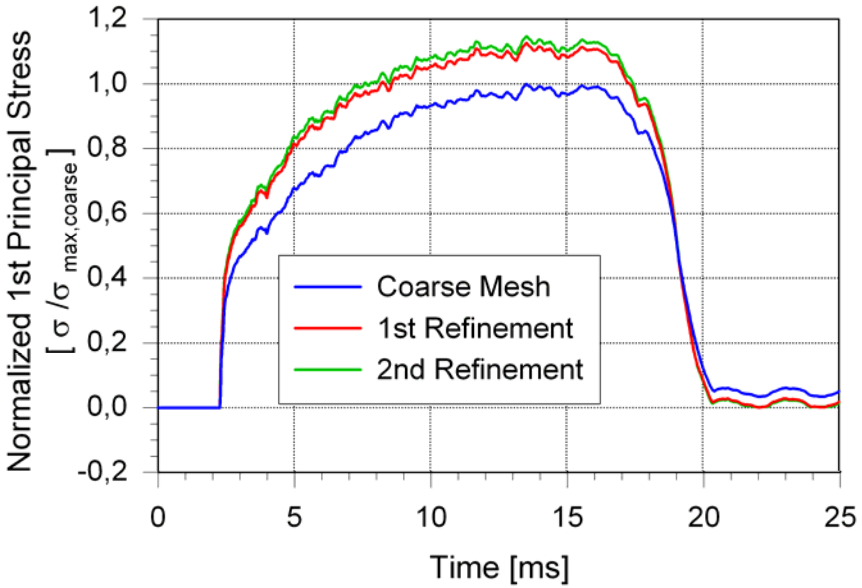
**Shaft (position A):**

- first refinement step leads to a noticeable stress increase
- further stress increase at the second refinement step is smaller

**Borehole (position B):**

- first refinement step leads to a distinct improvement in the numerical solution
- hardly improved at the second refinement step



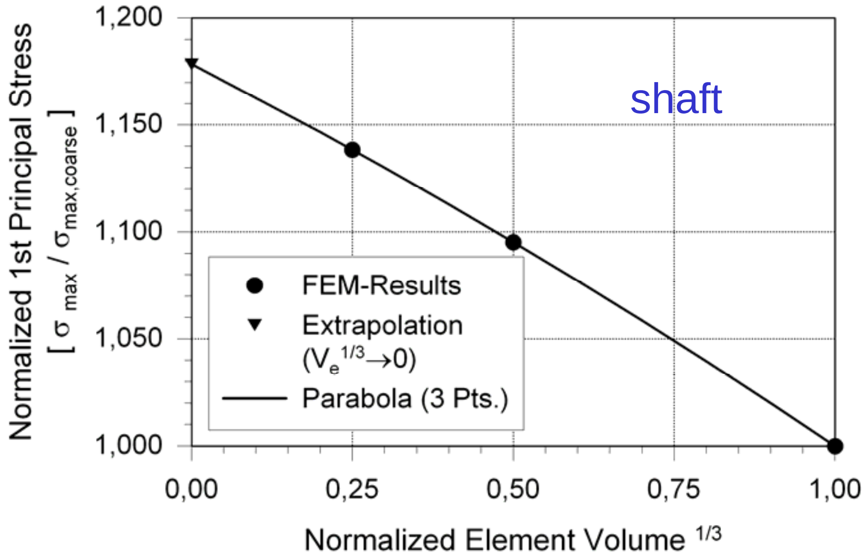
# Numerical results (cont'd)

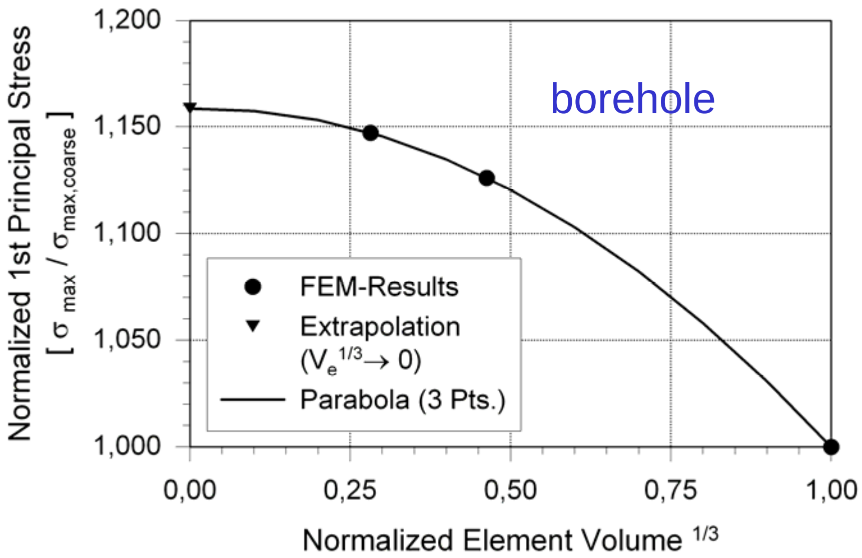
## Convergence of normalized first principal stress

- mean element size used for characterization also of non-cubic element shapes
  - corresponds exactly to the real element size only for a cube-shaped element
- merely three supporting points are available
  - parabola is one possibility among others for interpolation to illustrate the trend
- extrapolation to a vanishing element size corresponds to the infinite fine mesh or the continuous body respectively

**Shaft (position A):**
- almost linear curve
- limit stress is 18 % higher (for parabola)

**Borehole (position B):**
- seems to converge to constant value
- limit stress is 16 % higher (for parabola)



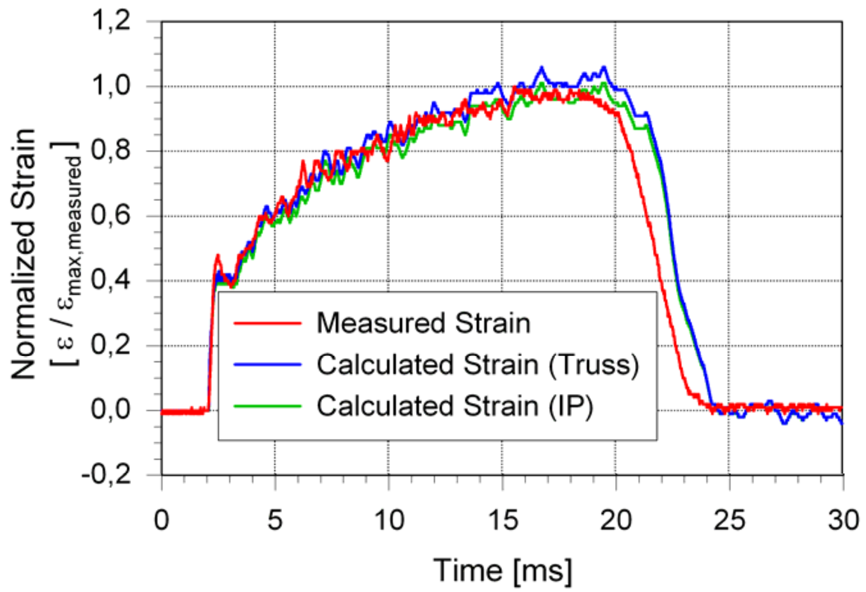



## Comparison with experimental data

- special attention to regions of high stress
  - measuring points nearby the lid system and the cask bottom also included in a complete assessment of the total model
- mesh density corresponds to the FE mesh of the first refinement step
- stresses and strains were evaluated at the integration points *(IP)*
- one-dimensional truss elements *(Truss)* were linked with surface nodes at the position of selected strain gauges
  - direct measure of local surface strains
- cf. **Position A**; assessment of other positions on shaft surface leads to similar results

**Measured and calculated strains at position A on the surface of the shaft**

- maximum values and the qualitative behavior agree well
- impact duration is slightly longer compared to simplified cask model
- deviation between “IP” and “Truss” acceptable



## Summary

- **The convergence behavior of the finite element solution was examined with a step-by-step refinement of the mesh.**
  - It could be shown for the considered mesh that the stress converges to a mesh-independent value for the continuous body which is **less than 20 %** above the stress calculated in the coarse mesh.
  - **Safety factor** of 1.2 may not be sufficient in other numerical simulations.
  - **Convergence and sensitivity studies** required.
  - Cask model could be **validated** by experimental data from a puncture drop test.
- **Even simple load scenarios possibly require very fine meshes with many elements.**
  - Complex load scenarios can be calculated with a **sub-model technique** if computer resources are limited.
  - There are **no general rules** about the number of needed elements over a given wall thickness of a cask or the element size.
- **Here a manually made finite element mesh was investigated to clarify the human factor.**
  - The **practical experience of the engineer** is of decisive importance at manual meshing.

*We thank cask manufacturer GNS for kind permission to use experimental data of the test cask in this study.*