Finite element method

evropský sociální fond v ČR
EVROPSKÁ UNIE
MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY
OP Vzdělávání pro konkurenceschopnost
INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

Among the up-to-date methods of stress state analysis, finite element method (abbreviated as FEM below, or often as FEA for analyses as well) dominates clearly nowadays; it is used also in other fields of engineering analyses (heat transfer, fluid flow, electric and magnetic fields, etc.).

In mechanics, the FEM enables us to solve the following types of problems:

- **stress-strain analysis** under static, cyclic or dynamic (including impact) loading, incl. various non-linear problems;
- natural as well as forced **vibrations** (eigenvalues of frequencies), with or without damping;
- **contact** problems (contact pressure distribution);
- **stability** problems (buckling of structures);
- stationary or non-stationary heat transfer and evaluation of consequential **temperature stresses** (incl. **residual stresses**), including those induced by phase transformation.

Functional

Fundamentals of FEM differ on principle from the analytical methods of stress-strain analysis. While analytical methods of stress-strain analysis are based on the differential and integral calculus, FEM is based on the **calculus of variations** which is generally not well known; it is based on seeking for minimum of a **functional**.

Explanation of the basis of the concept – analogy with functions:

- **Function** - is a mapping between two sets of numbers, it is mathematical term for a rule which enables us to assign unambiguously some numerical value (from the image of mapping) to an input numerical value (from the domain of mapping).
- **Functional** - is a mapping from a set of functions to a set of numbers. It is a rule which enables us to assign unambiguously some numerical value to a function (on the domain of the function or on its part). Definite integral is example of such a functional.

Principle of minimum of the quadratic functional

Among all the allowable displacements (i.e. those which meet geometric and physical equations of the problem and its boundary conditions), only those displacements can come into existence between two close loading states (change of displacement by variation δu) which **minimize the quadratic functional** Π_L. This functional (called **Lagrange potential**) represents the total potential energy of the body, and the corresponding displacements, stresses and strains minimizing its value represent the **elasticity functions** we are seeking for.

This principle is called **Lagrange variation principle**.

Lagrange potential Π_L can be written as follows:

$$\Pi_L = W - P$$

where W - total strain energy of the body

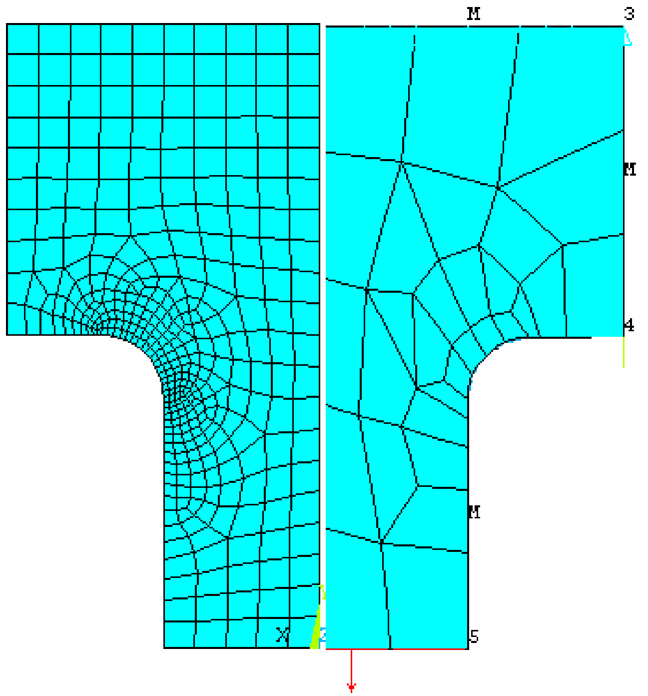
P – total potential energy of external loads

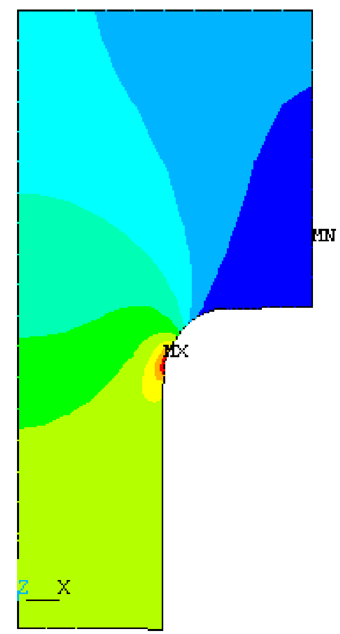
Basic terms of FEM

- **Finite element** – a subregion of the solved body with a simple geometry.
- **Node** – a point in which the numerical values of the unknown deformation parameters are calculated.
- **Base function** – a function describing the distribution of degrees of freedom (DOF), i.e. deformation parameters (typically displacements) throughout the element (between its nodes).
- **Shape function** – a function describing the distribution of strains throughout the element, it represents a derivative of the base function.
- **Discretization** – transformation of a continuous problem to a solution of a finite number of discontinuous (discrete) numerical values.
- **Mesh density** – density of elements (inverse to their size) which influences the accuracy of the solution and its computer time consumption.
- **Matrixes** (they are created by summarization of contributions of the individual elements)
  - of displacements
  - of stiffness
  - of base functions
- **Convergence** – the basic property of the method, meaning the solution tends to the real (continuous) solution when the mesh (discretization) density increases (element size decreases).
- **Percentual energy error** – assessment of total inaccuracy of the solution, it represents a difference between the calculated stress values and their values smoothed by postprocessing tools used for their graphical representation, when transformed into difference in strain energies.
- **Isoparametric element** – element with the same order of the polynoms used in description of both geometry and base functions.

FEA of the stress concentration in a notch

(bar under tension, nominal stress in the shoulder is 1 MPa)





Type of element	mesh density	calculated maximum stress [MPa]
linear - four nodes	rough	1,28
linear - four nodes	fine	1,67
quadratic - eight nodes	rough	1,59
quadratic - eight nodes	fine	1,67

Stress distribution

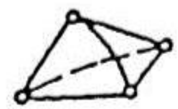
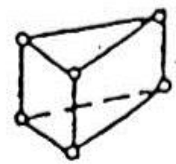
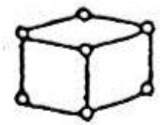
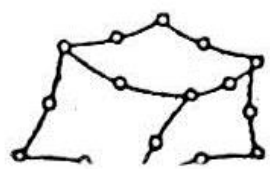
in the dangerous cross section of the notch

Overview of basic types of finite elements

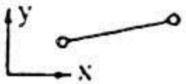
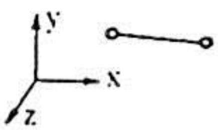
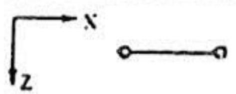
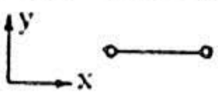
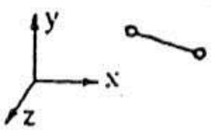
They can be distinguished from the point of view of the assumptions the element is based on (bar assumptions, axisymmetry, Kirchhoff plates, membrane shells, etc.), or for what family of problems the element is formulated.

- **Three-dimensional** elements (volume elements - bricks)
- **Two-dimensional** elements (plane stress, plain strain, axisymmetry)
- **Bar**-like (truss) elements (either for tension-compression only, or for flexion and torsion as well)
- **Shell** elements (with combination of in-plane and out-of-plane) loads (wall and plate or shell)
- **Special** elements (contact elements, crack elements, cohesive elements, etc.)

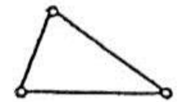
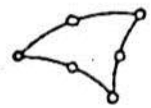
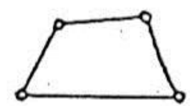
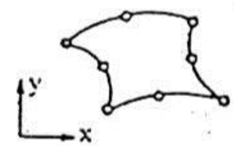
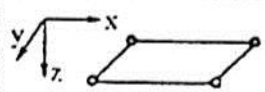
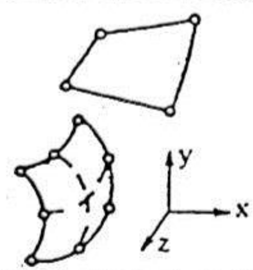
Types of 3D elements

Type of element	Representation	Deformation parameters (DOFs)
Tetrahedron		u, v, w
Pentahedron		
Hexahedron with 8 nodes		
Hexahedron with 20 nodes		

Types of 1D elements

Type of element		Representation	Deformation parameters (DOFs)
Link (1D) element (strut, bar – tension-compression only)	in 2D		u, v
	in 3D		u, v, w
Beam element (2D flexion only)			w, φ_y
Frame element (flexion + tension)	in 2D		u, v, φ_z
	in 3D – with torsion included		u, v, w $\varphi_x, \varphi_y, \varphi_z$

Types of 2D elements

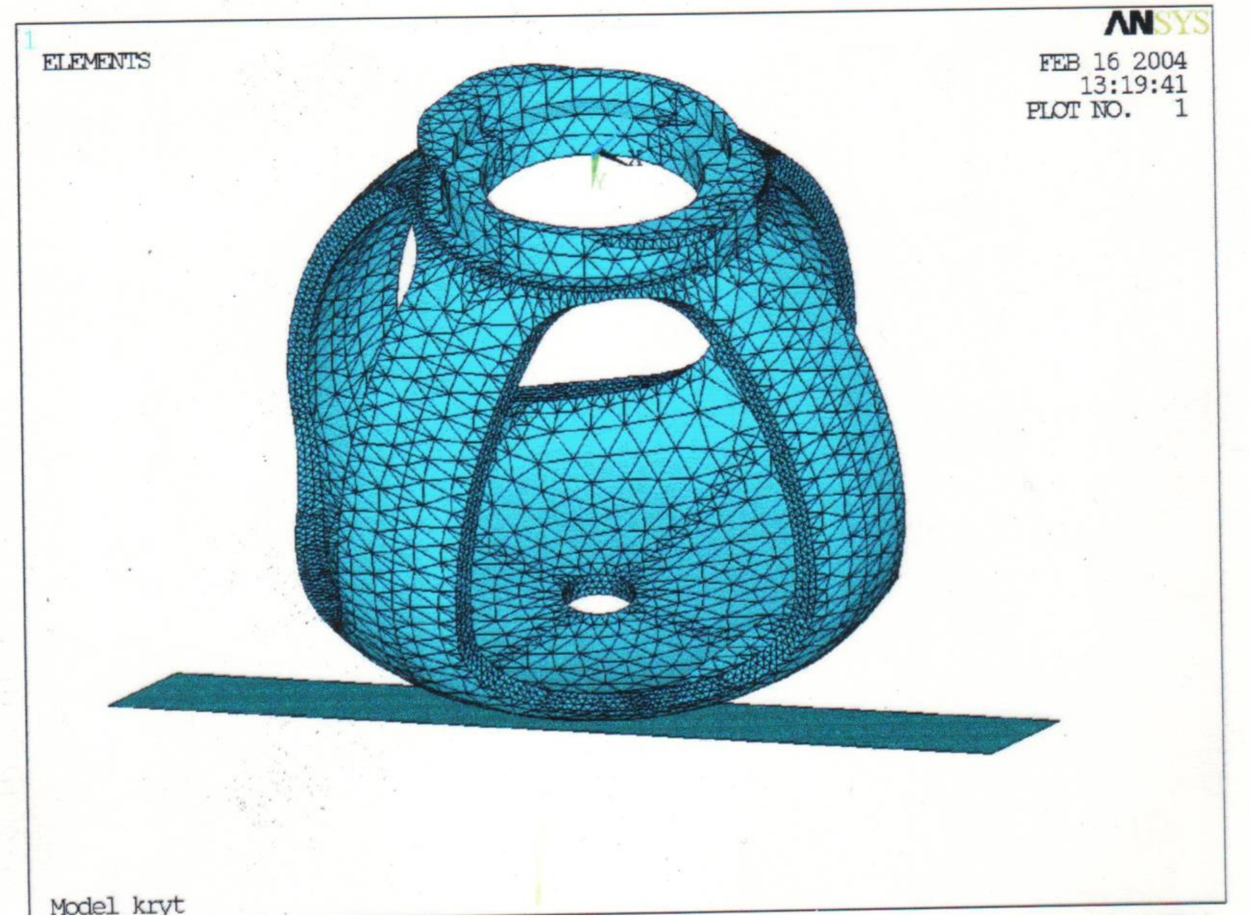
Type of element	Representation	Deformation parameters
Membrane shell elements		u, v
Linear triangular shell element		
Quadratic triangular shell element		
Linear quadrilateral shell element		
Quadratic isoparametric quadrilateral shell element		
Plate element		w, φ_x, φ_y
General (momentum) shell elements (linear - quadratic isoparametric)		u, v, w $\varphi_x, \varphi_y, \varphi_z$

Basic types of constitutive relations soluble in FEA

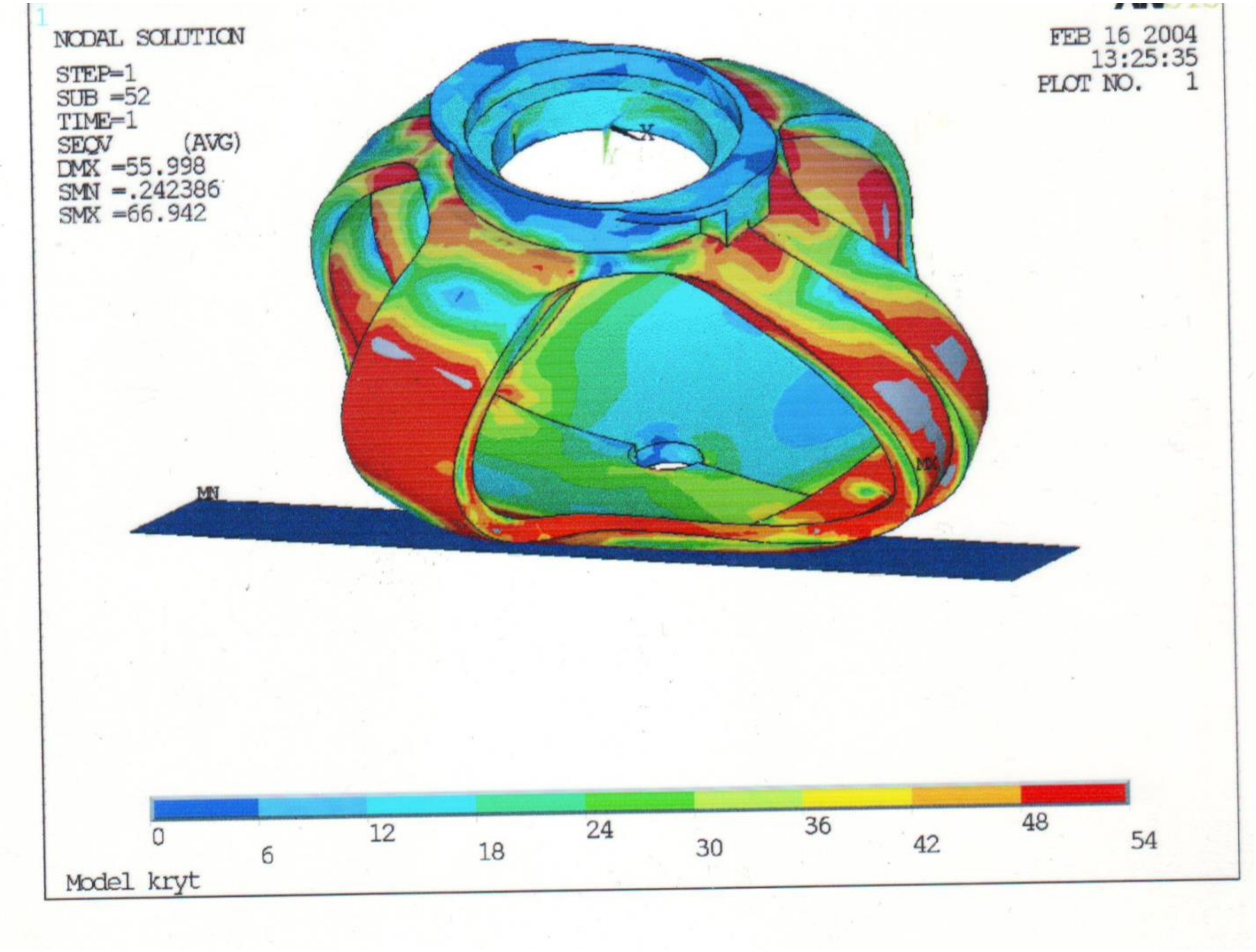
- **linear elastic anisotropic ─** elastic parameters are direction dependent (monocrystals, wood, fibre composites or multilayer materials),
- **elastic-plastic** (steel above the yield stress) with different types of behaviour above the yield stress (perfect elastic-plastic materials, various types of **hardening**),
- **non-linear elastic ─** deformations are reversible, but non-linearly related to the stress (rubber and other elastomers, soft biological tissues),
  - **hyperelastic** (showing elastic strain on the order of 10^1 – 10^2percents and thus being always non-linear),
- **viscoelastic ─** deformation is recoverable but time-dependent, i.e. rather than instantaneously occuring with some transition time when the material is out of equilibrium; the material shows creep, stress relaxation and hysteresis (plastics, pure alluminium, steel above 400°C) ,
- **viscoplastic ─** plastic (permanent) deformation is time-dependent (plasticity under high temperatures)

etc.

Examples of non-linear problems:
a FE mesh in a plastic guard of a pressure bottle valve

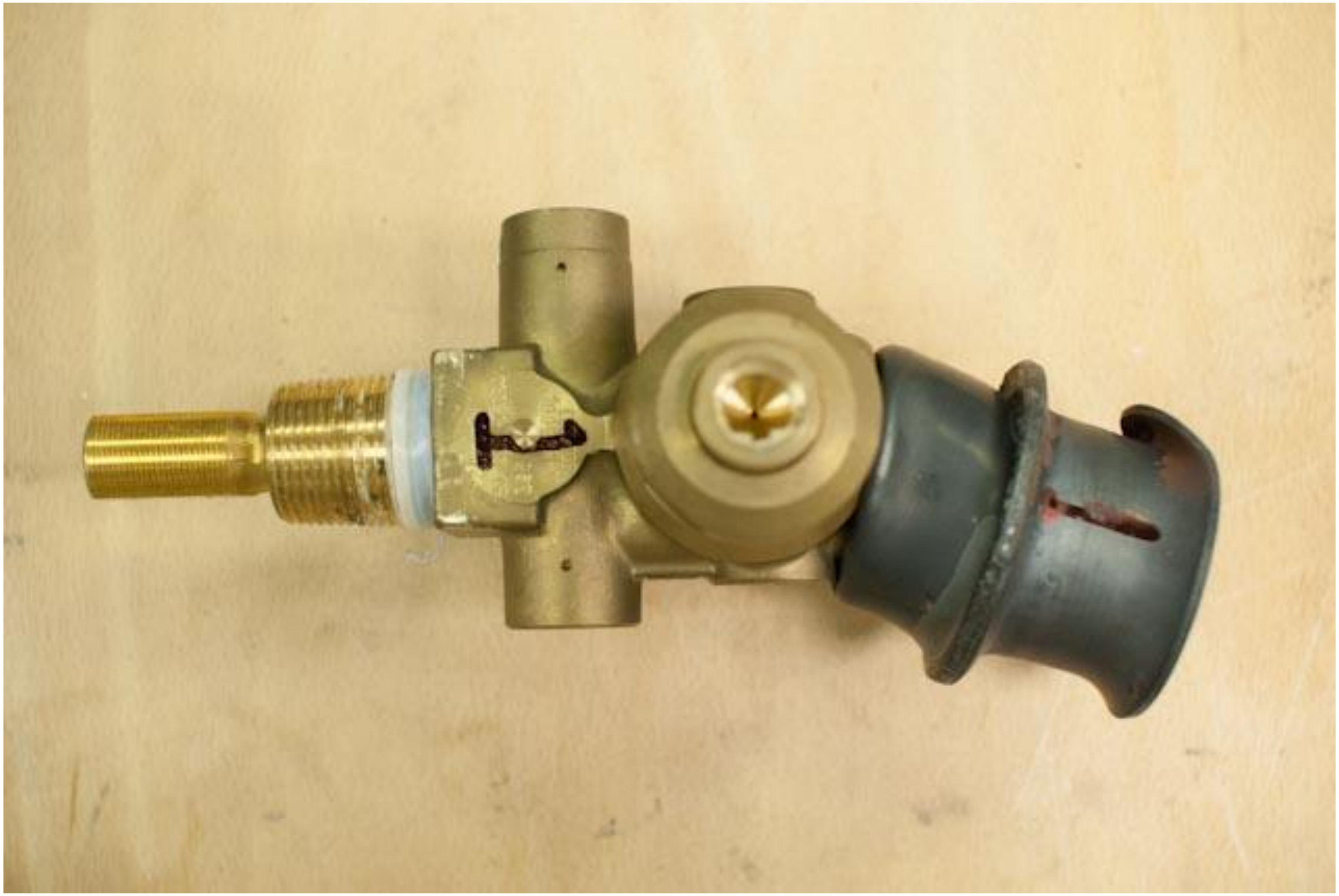


Distribution of Mises stresses in a plastic safety guard after impact, grey colour shows regions where yield stress R_e=54MPa was exceeded.)



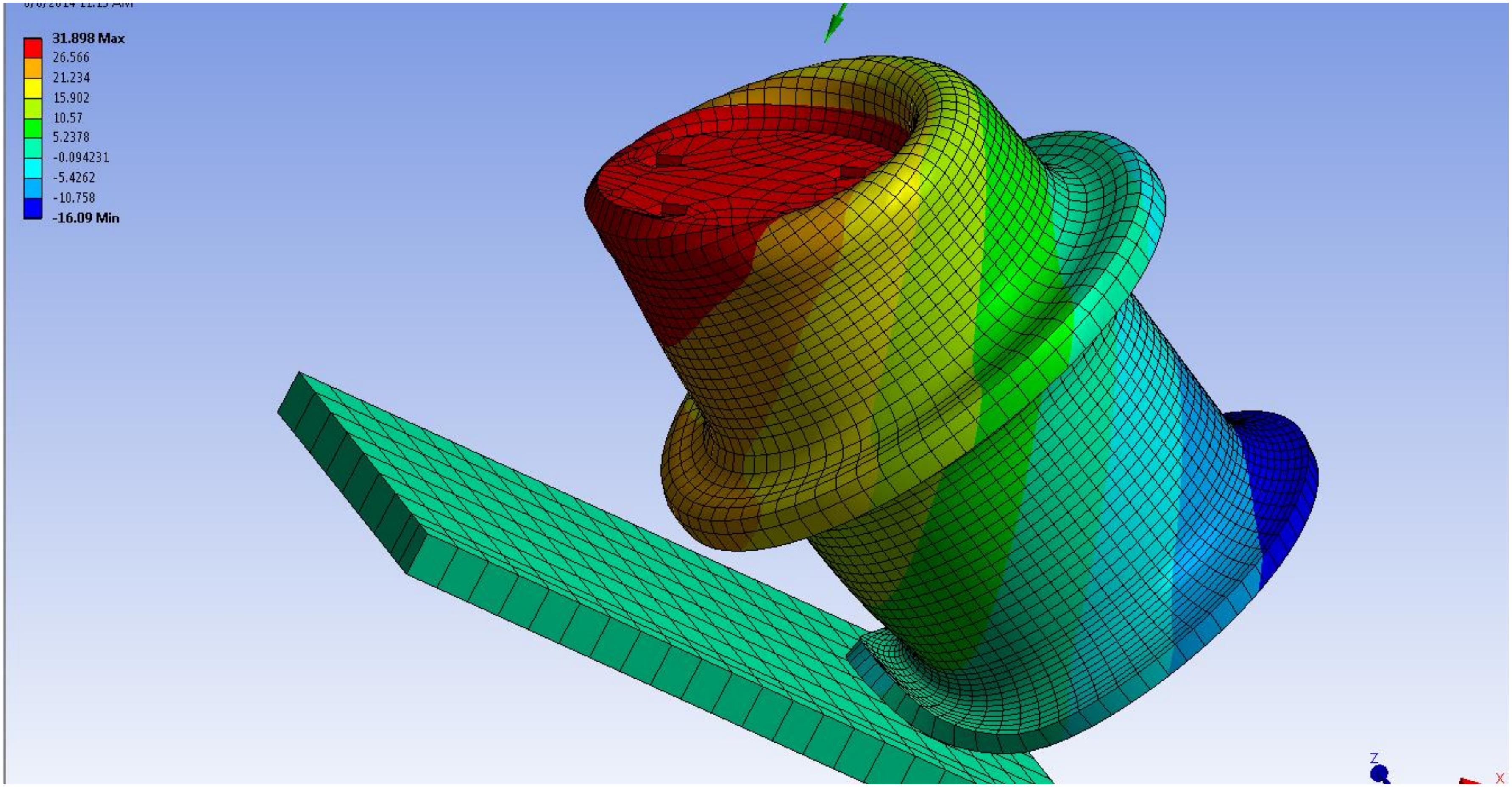
Assembly with a steel shock absorber after impact test

(total assembly with bottle and plastic guard in the video)

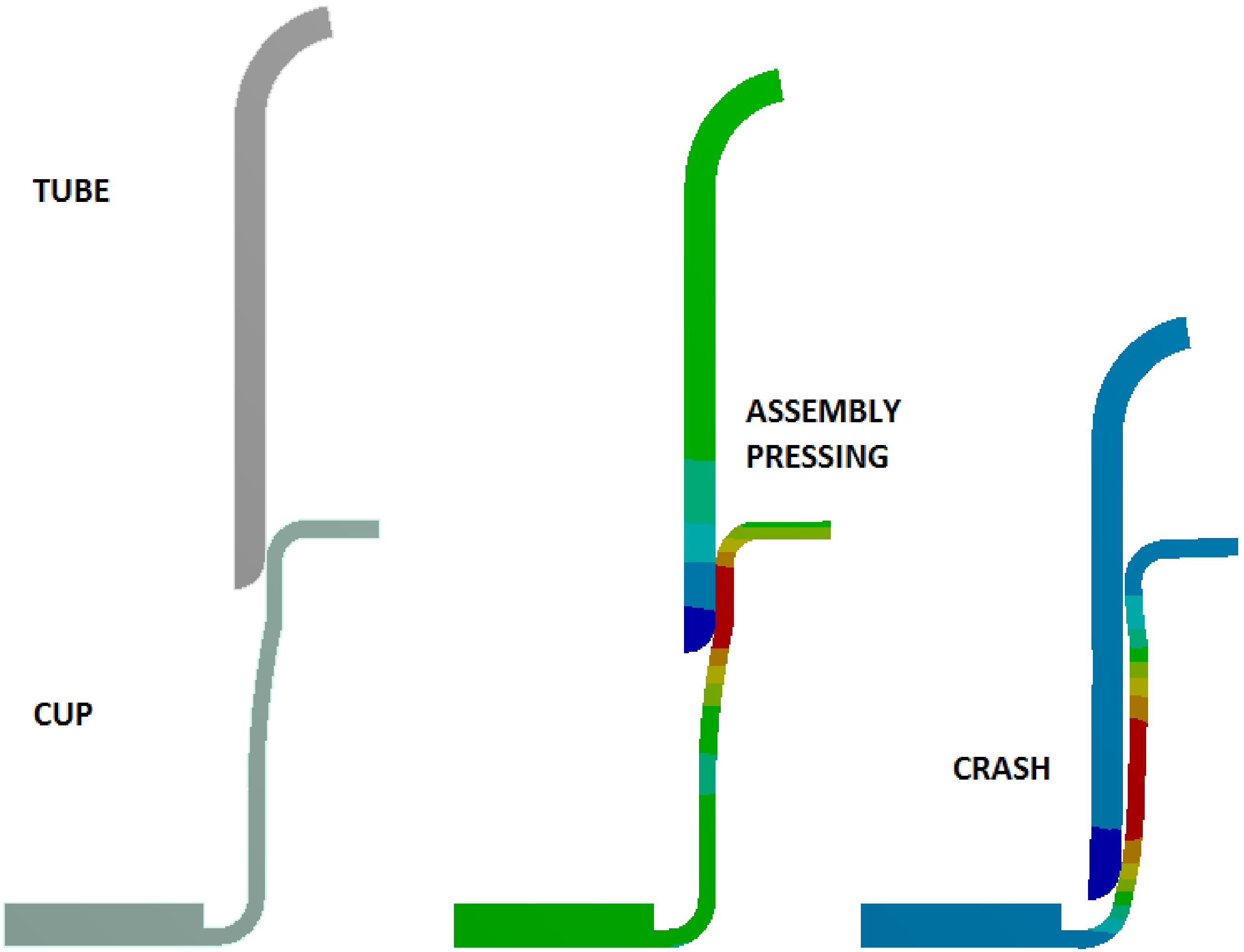


Steel shock absorber

original welded design under simulated impact test



Steel shock absorber
new design with controlled plastic deformation
(details in pdf file)



Comparison of different solutions of steel shock absorbers

The area below the curve corresponds to the absorbed deformation energy.