

Využitie COMSOL Multiphysics na modelovanie a simuláciu technologických procesov obrábania a tvárnenia

Ing. Martin Necpal, PhD.

martin.necpal@stuba.sk



SLOVAK UNIVERSITY OF
TECHNOLOGY IN BRATISLAVA
FACULTY OF MATERIALS SCIENCE
AND TECHNOLOGY IN TRNAVA

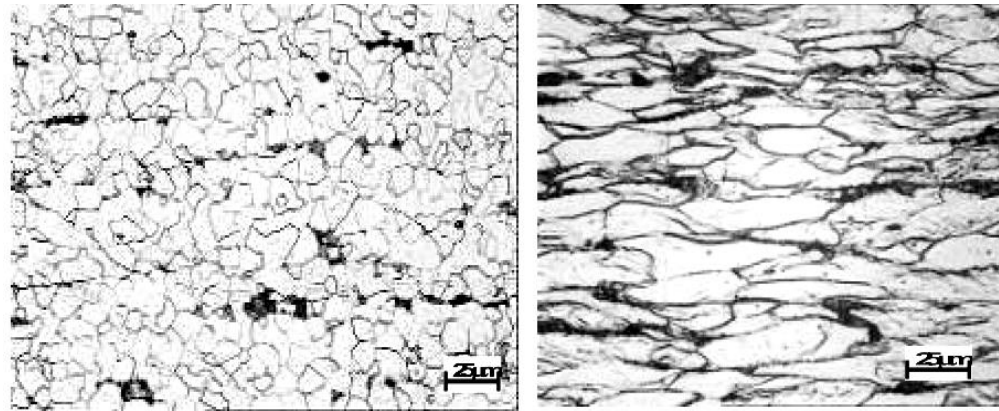
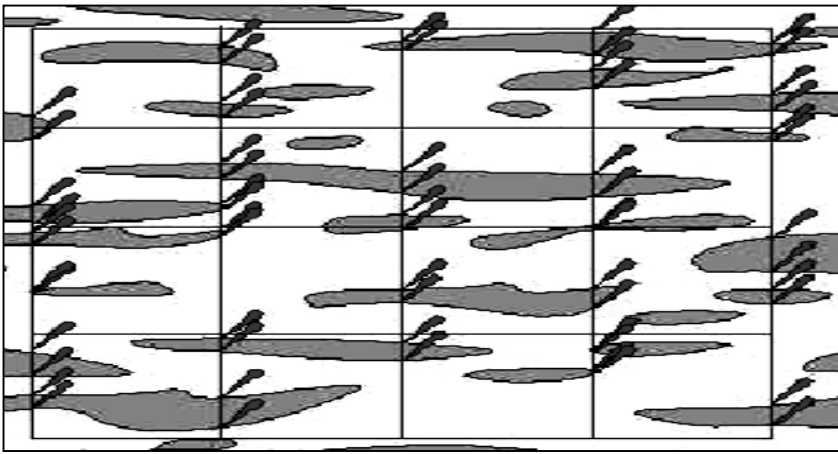


- Ústav materiálov
- **Ústav výrobných technológií**
- Ústav priemyselného inžinierstva a manažmentu
- Ústav integrovanej bezpečnosti
- Ústav aplikovanej informatiky
- Ústav výskumu progresívnych technológií

- **Katedra obrábania a tvárnenia**
- Katedra zvarovania a zlievarenstva
- Katedra výrobných systémov, metrológie a montáže







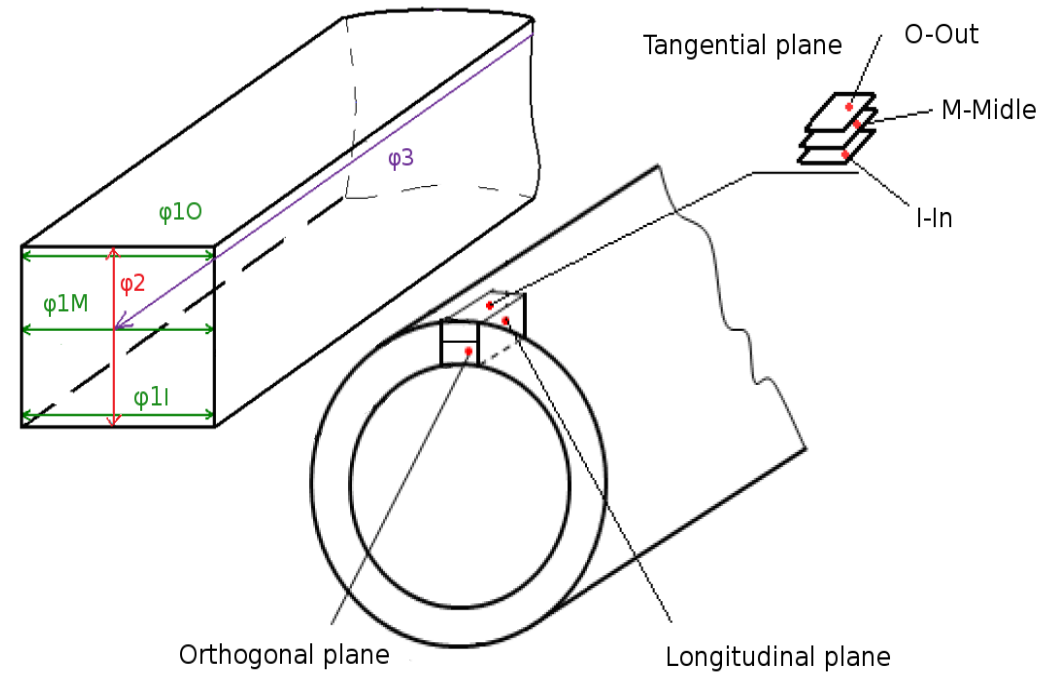
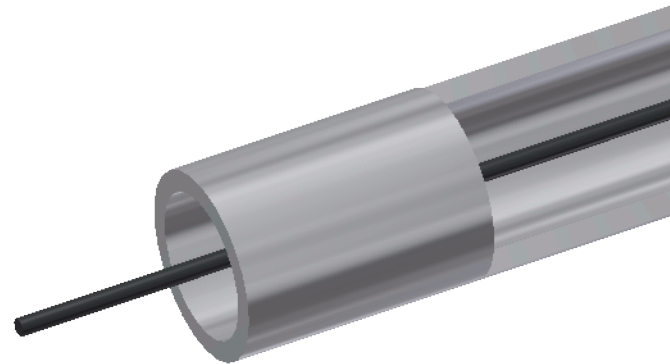
Saltykova stereologická metóda orientovaných
skúšobných čiar

$$(S_V)_{TOT} = (P_L)_O + (P_L)_P$$

$$(S_V)_{OR} = (P_L)_O - (P_L)_P$$

$$O = \frac{(S_V)_{OR}}{(S_V)_{TOT}}$$

$$\varphi = \ln \left(\frac{1 + O\sqrt{2 - O^2}}{1 - O^2} \right)^{\frac{2}{3}}$$



Slovak Research and Development Agency under the contract No. APVV-15-0319

Simulácia COMSOL

GEOMETRIA

- Dxf file import
- 2 domeny (prievlak a rúra)
- Form Assembly

The screenshot displays the COMSOL Model Builder interface. On the left, the tree view shows the project structure: 6st-penalty_ML.mph (root) > Global Definitions > Parameters > Draw (draw1) > Materials > Component 1 (comp1) > Definitions > Geometry 1 > Import 1 (imp1). The 'Import 1' node is selected. The central panel shows the 'Import' settings for 'Import 1'. The 'Source' is set to 'DXF file'. The 'Filename' is 'C:\Users\MartinNecpal\Documents\comsol draw\6st.dxf'. The 'Layer selection' is set to 'All'. The 'Import options' are set to 'Form solids'. The 'Repair imported objects' checkbox is checked, and the 'Relative repair tolerance' is set to '1E-5'. The 'Selections of Resulting Entities' section shows 'Contribute to:' set to 'None'. The 'Resulting objects selection' and 'Individual object selections' checkboxes are unchecked, and their 'Show in physics' dropdowns are set to 'Domain selection'. On the right, the 'Graphics' window shows a 2D schematic of two overlapping gray shapes: a vertical rectangle and a trapezoidal shape overlapping its bottom edge.

Simulácia COMSOL

Nastavenie SIETE

The screenshot displays the COMSOL Multiphysics interface. On the left is the **Model Builder** tree, showing the hierarchy from the root model to the specific mesh size settings. The **Mesh 1 > Size 1** node is selected. The middle panel shows the **Settings** for this size, including a list of active geometric entities (1, 3, 12) and the **Element Size** section. The **Calibrate for** dropdown is set to **General physics**, and the **Custom** radio button is selected. The **Element Size Parameters** section has the following values: **Maximum element size** (1 m), **Minimum element size** (0.0365 m), **Maximum element growth factor** (1.3), **Curvature factor** (0.3), and **Resolution of narrow regions** (1). On the right, the **Graphics** window shows a 3D view of a meshed part with a blue border.

Model Builder

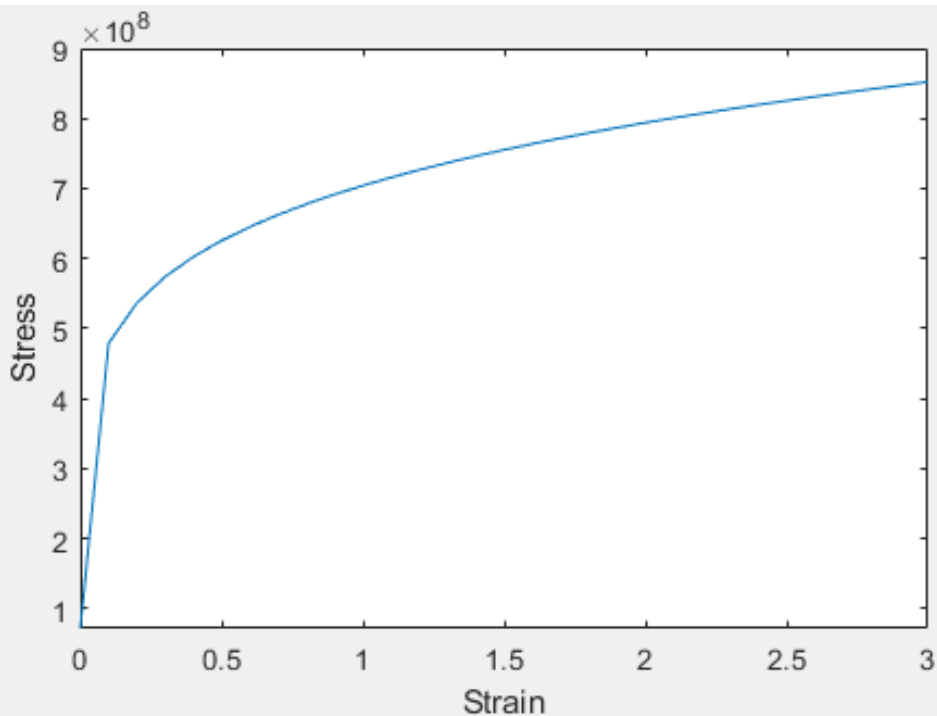
- 6st-penalty_ML.mph (root)
 - Global Definitions
 - Parameters
 - Draw (draw1)
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Materials
 - Solid Mechanics (solid)
 - Linear Elastic Material
 - Axial Symmetry 1
 - Free 1
 - Initial Values 1
 - Fixed Constraint 1
 - Prescribed Displacement 1
 - Equation View
 - Contact 1
 - Fixed Constraint 2
 - Equation View
 - Equation View
 - Mesh 1
 - Size
 - Size 1
 - Mapped 1
 - Distribution 1
 - Study 1
 - Step 1: Stationary
 - Solver Configurations
 - Solution 1 (sol1)
 - Compile Equation
 - Dependent Variables
 - Stationary Solver
 - Job Configurations
 - Results

Simulácia COMSOL

Materiálový model - Ludwik

$$\sigma_{ys} = \sigma_{ys0} + k(\epsilon_{pe})^n$$

- $\sigma_{ys0} = 71,27 \text{ MPa}$
- $k = 632,8 \text{ MPa}$
- $n = 0.1914$



Model Builder

- 6st-penalty_ML.mph (root)
 - Global Definitions
 - Parameters
 - Draw (draw1)
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Import 1 (imp1)
 - Form Assembly (fin)
 - Materials
 - Structural steel (mat1)
 - Material 2 (mat2)
 - Solid Mechanics (solid)
 - Linear Elastic Material 1
 - Plasticity 1
 - Equation View
 - Axial Symmetry 1
 - Free 1
 - Initial Values 1
 - Fixed Constraint 1
 - Prescribed Displacement 1
 - Contact 1
 - Fixed Constraint 2
 - Equation View
 - Mesh 1

Settings Properties

Plasticity

Label: Plasticity 1

Domain Selection

Selection: Manual

Active

Override and Contribution

Equation

Model Input

Plasticity Model

Plasticity model: Large plastic strains

Yield function F: von Mises stress

$F = \sigma_{mises} - \sigma_{ys}$, $Q = F$

Initial yield stress: σ_{ys0} From material

Isotropic hardening model: Ludwik

$\sigma_{ys} = \sigma_{ys0} + k(\epsilon_{pe})^n$

Strength coefficient: k From material

Hardening exponent: n From material

Graphics

7/21

Simulácia COMSOL

Okrajová podmienka trenie

- $\mu_{star} = 0,04117$

The screenshot displays the COMSOL Model Builder interface. The left-hand tree view shows a project named '6st-penalty_ML.mph' with a 'Friction 1' boundary condition selected under 'Solid Mechanics (solid)'. The right-hand 'Properties' pane is configured for 'Friction 1' with the following settings:

- Label: Friction 1
- Boundary Selection: All boundaries
- Active: ON
- Friction model: Static Coulomb friction
- Static frictional coefficient: μ_{stat} (mu) = 1
- Cohesion sliding resistance: T_{cohe} = 0 N/m²
- Maximum tangential traction: $T_{t,max}$ = Inf N/m²
- Tangential Force Method: Augmented Lagrangian
- Penalty Factor: Penalty factor control: Preset
- Tuned for: Stability

The 'Graphics' window on the right shows a 3D model of a vertical rectangular block with a blue line representing the contact interface at the bottom edge.

Model Builder

- 6st-penalty_ML.mph (root)
 - Global Definitions
 - Parameters
 - Draw (draw1)
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Materials
 - Solid Mechanics (solid)
 - Linear Elastic Material 1
 - Axial Symmetry 1
 - Free 1
 - Initial Values 1
 - Fixed Constraint 1
 - Prescribed Displacement 1
 - Equation View
 - Contact 1
 - Fixed Constraint 2
 - Equation View
 - Mesh 1
 - Study 1
 - Step 1: Stationary
 - Solver Configurations
 - Job Configurations
 - Results

Settings | **Properties**

Fixed Constraint

Label: Fixed Constraint 2

Boundary Selection

Selection: Manual

Active: 2

Override and Contribution

Equation

Constraint Settings

Excluded Points

Graphics

Model Builder

- 6st-penalty_ML.mph (root)
 - Global Definitions
 - Parameters
 - Draw (draw1)
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Materials
 - Solid Mechanics (solid)
 - Linear Elastic Material 1
 - Axial Symmetry 1
 - Free 1
 - Initial Values 1
 - Fixed Constraint 1
 - Prescribed Displacement 1
 - Equation View
 - Contact 1
 - Fixed Constraint 2
 - Equation View
 - Mesh 1
 - Study 1
 - Step 1: Stationary
 - Solver Configurations
 - Job Configurations
 - Results

Settings | **Properties**

Prescribed Displacement

Label: Prescribed Displacement 1

Boundary Selection

Selection: Manual

Active: 9, 11, 12

Coordinate system: Global coordinate system

Prescribed Displacement

- Standard notation
- Prescribed in r direction
 - u_{0r} 0 m
- Prescribed in z direction
 - u_{0z} -draw1(para) m
- General notation

Constraint Settings

Excluded Points

Graphics

Simulácia COMSOL

Riešič

Study Extensions

Auxiliary sweep

Sweep type: Specified combinati

Parameter name	Parameter value list
para (Solution para	range(0,1e-2,1.1)

Prescribed Displacement

Standard notation

Prescribed in r direction

u_{0r} 0 m

Prescribed in z direction

u_{0z} -draw1(para) m

General notation

Model Builder

- 6st-penalty_ML.mph (root)
 - Global Definitions
 - Parameters
 - Draw (draw1)
 - Materials
 - Component 1 (comp 1)
 - Definitions
 - Geometry 1
 - Materials
 - Solid Mechanics (solid)
 - Linear Elastic Material
 - Axial Symmetry 1
 - Free 1
 - Initial Values 1
 - Fixed Constraint 1
 - Prescribed Displacement
 - Equation View
 - Contact 1
 - Fixed Constraint 2
 - Equation View
 - Mesh 1
 - Study 1
 - Step 1: Stationary
 - Solver Configurations
 - Solution 1 (sol 1)
 - Compile Equation
 - Dependent Variables
 - Stationary Solver
 - Job Configurations
 - Results

Settings Properties

Interpolation

Plot Create Plot

Label: Draw

Definition

Data source: Local tab

Function name: draw1

t	f(t)
-1e-3	0
0	0
1	-disp
1.5	-disp

Define primitive function
Primitive function name: int

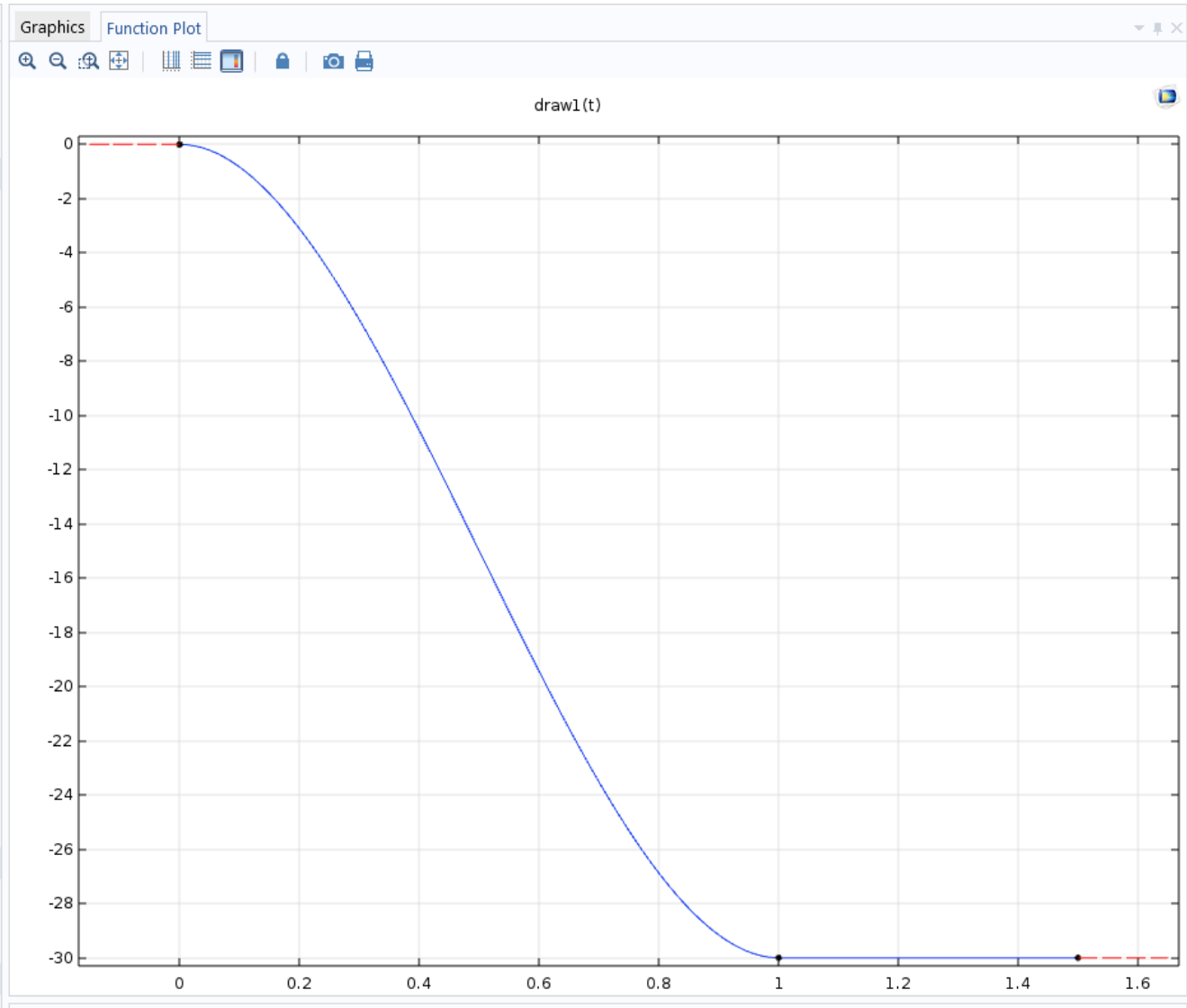
Define inverse function
Inverse function name: int

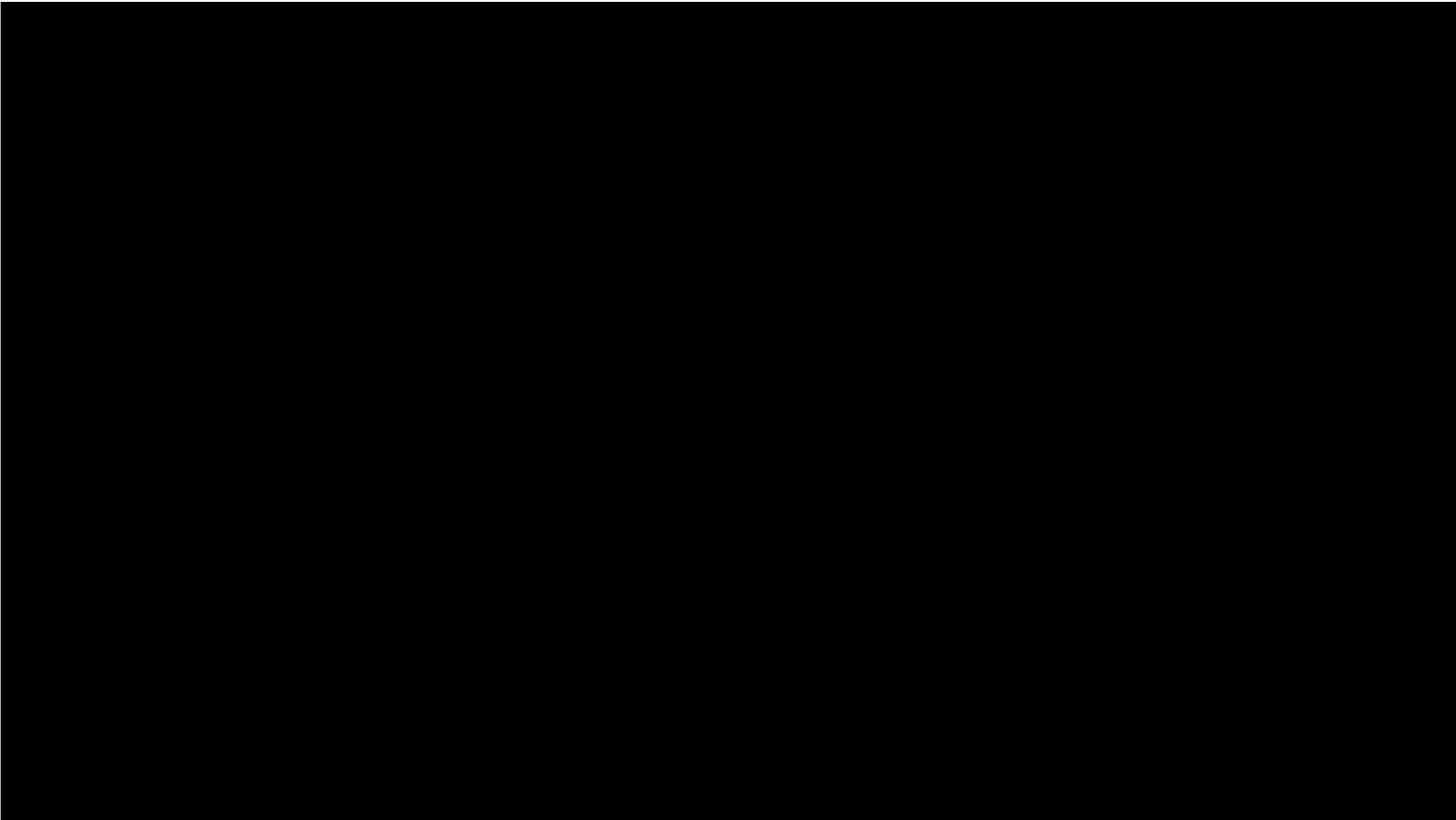
Interpolation and Extrapolation

Interpolation: Piecewise

Extrapolation: Constant

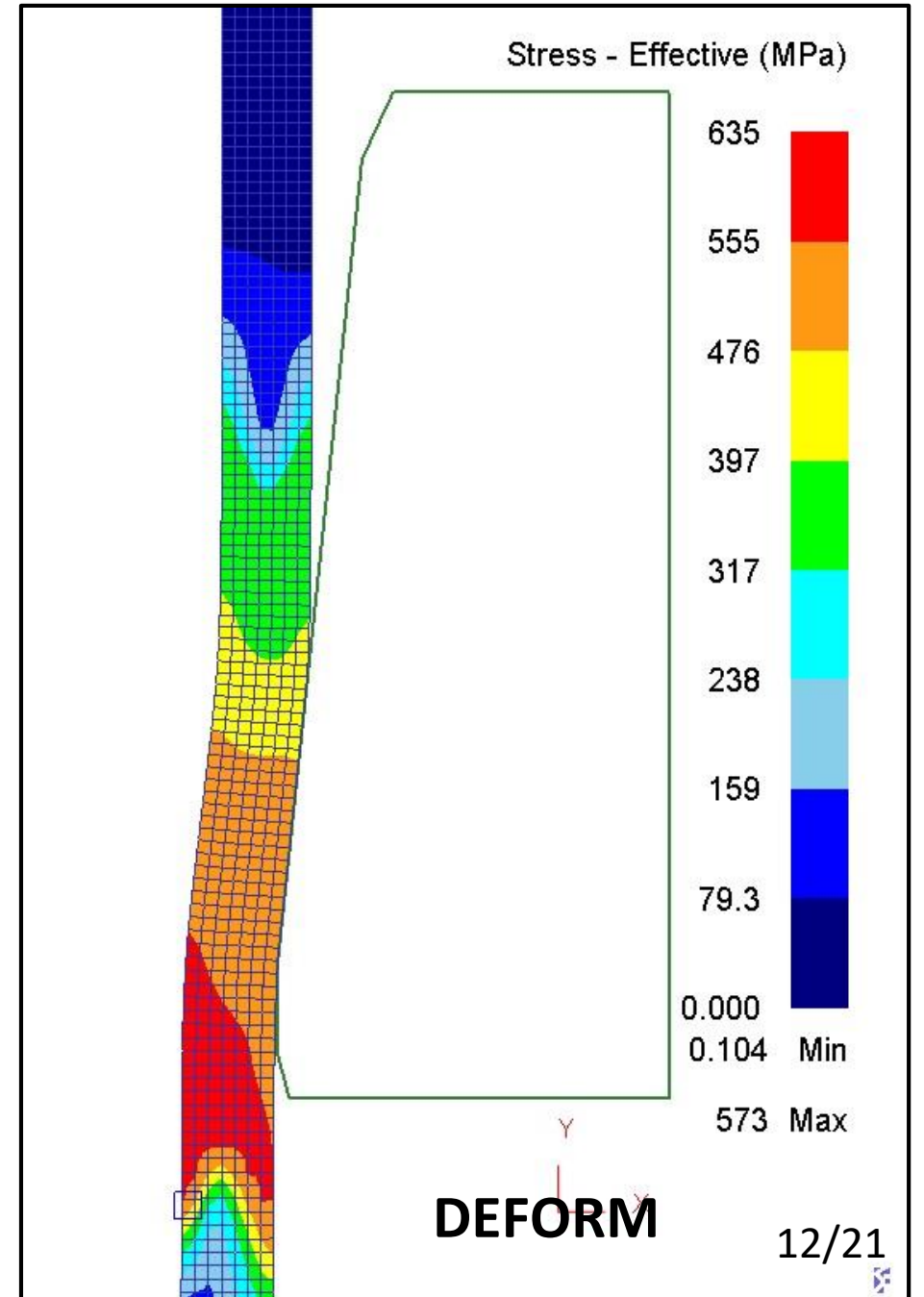
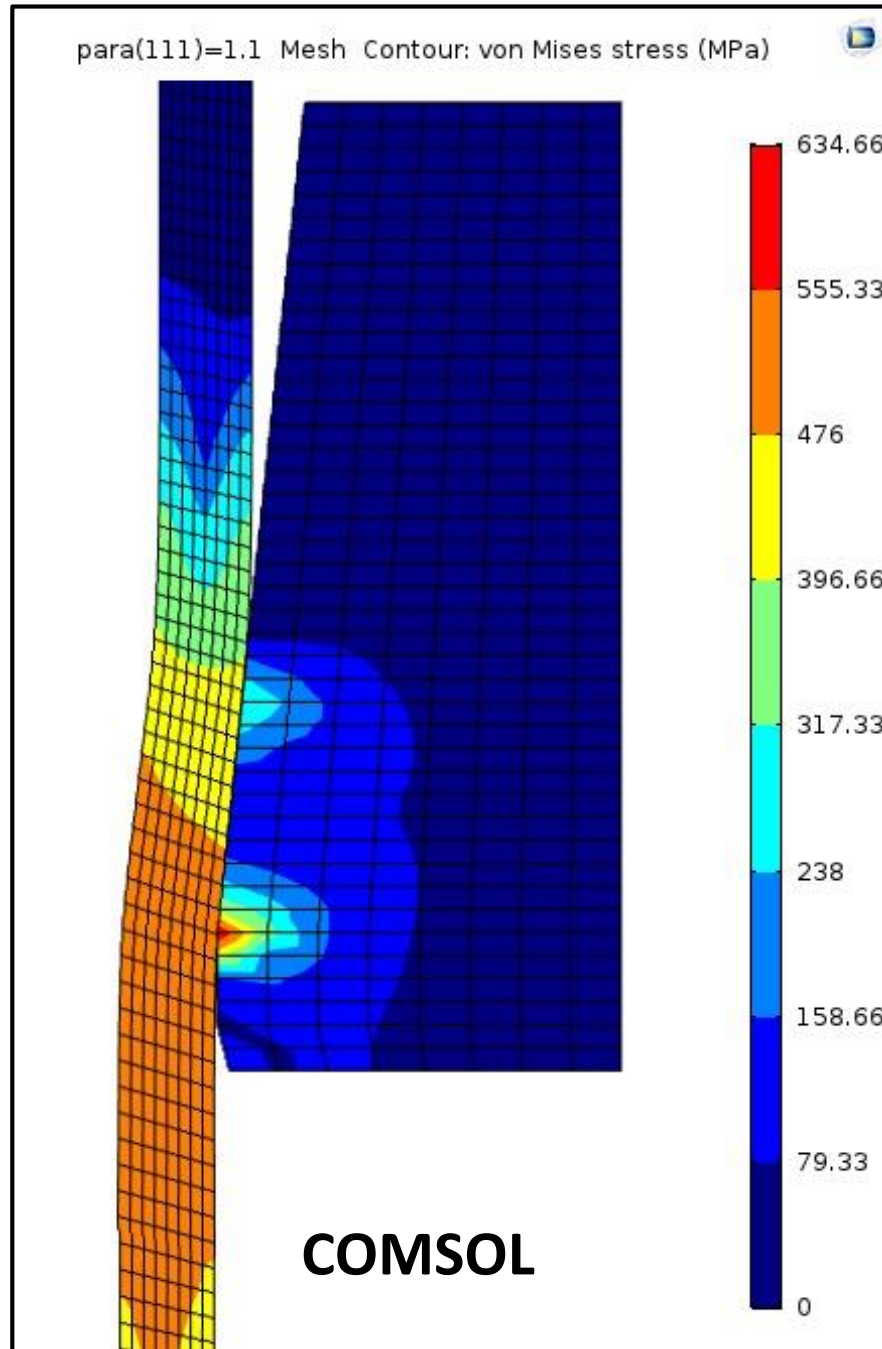
Units





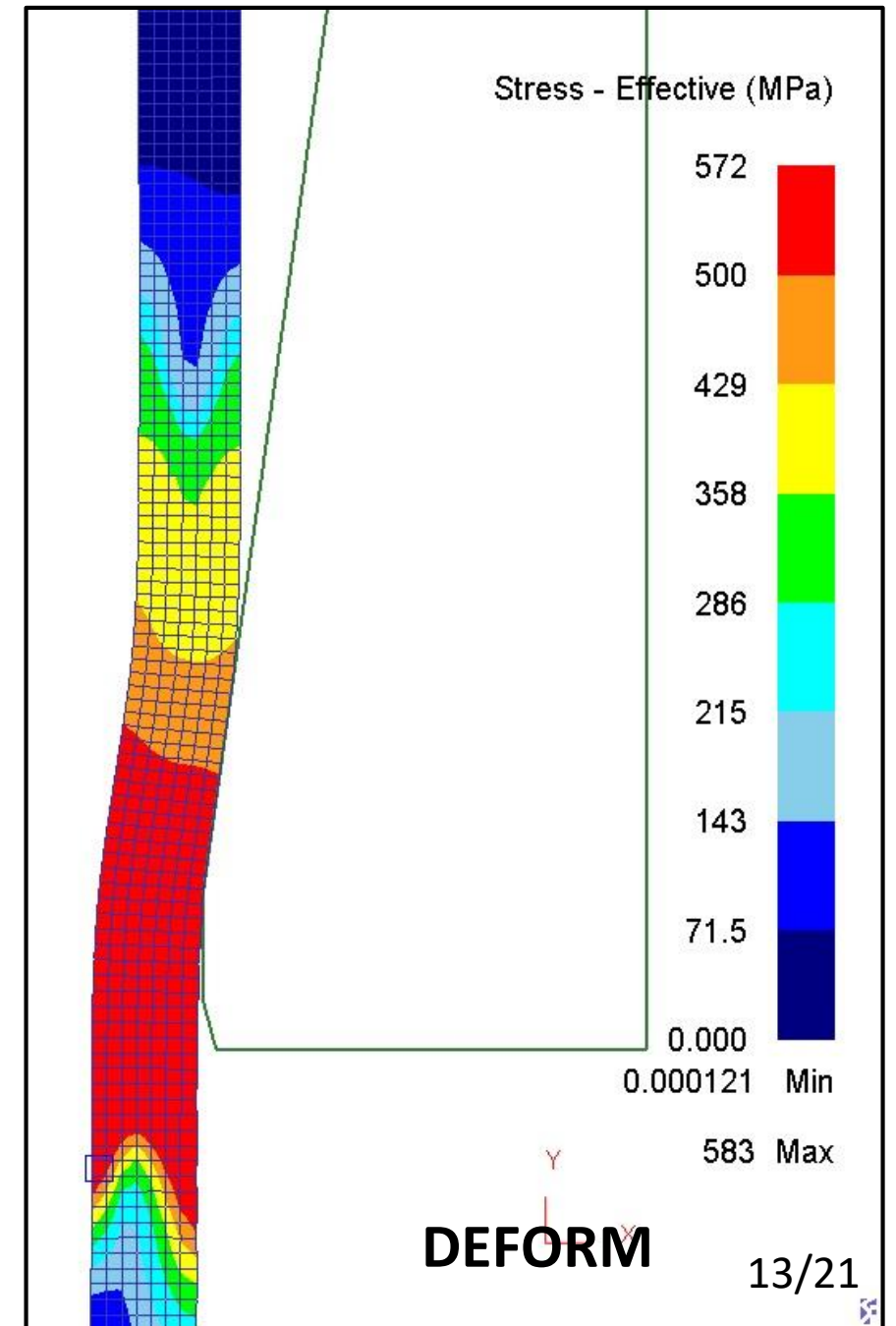
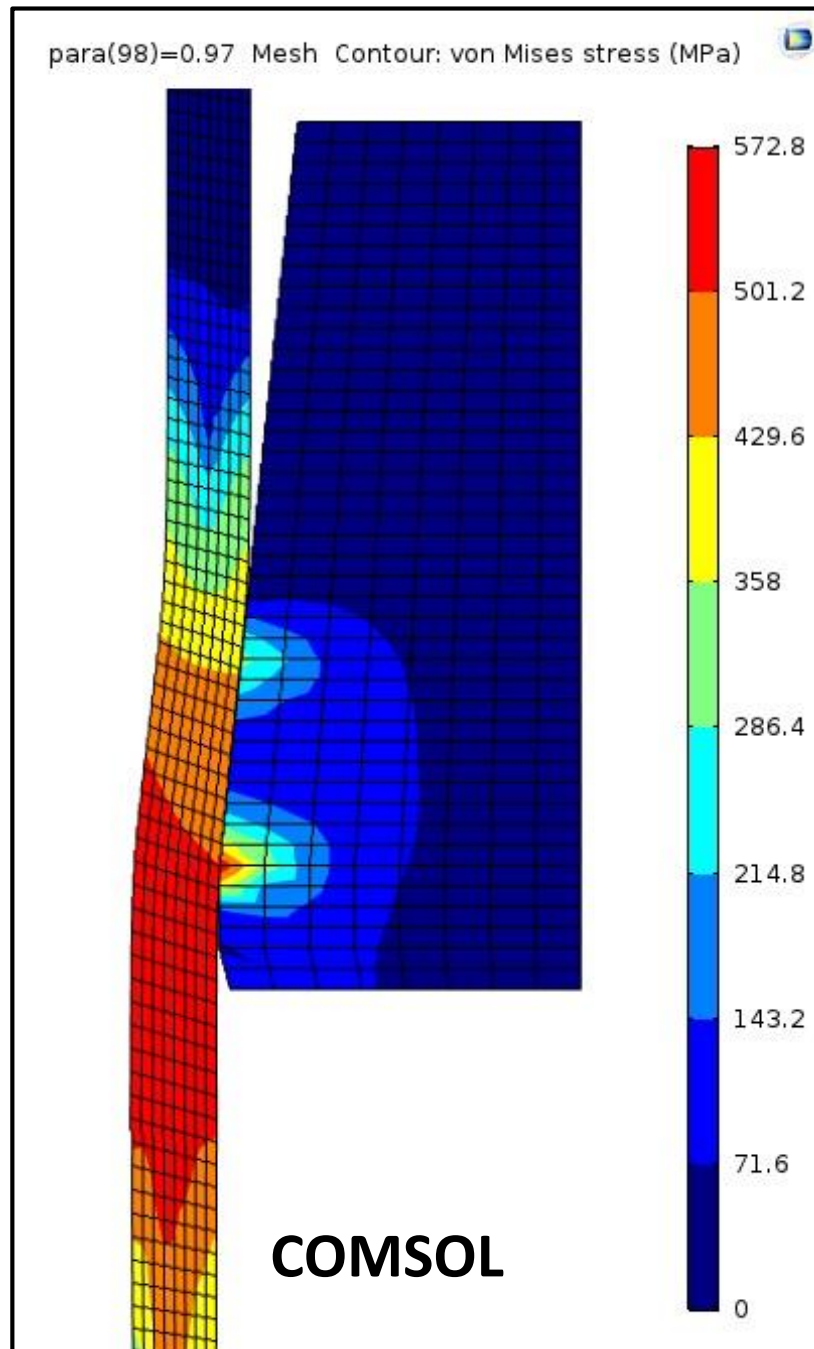
Porovnanie výsledkov simulácie

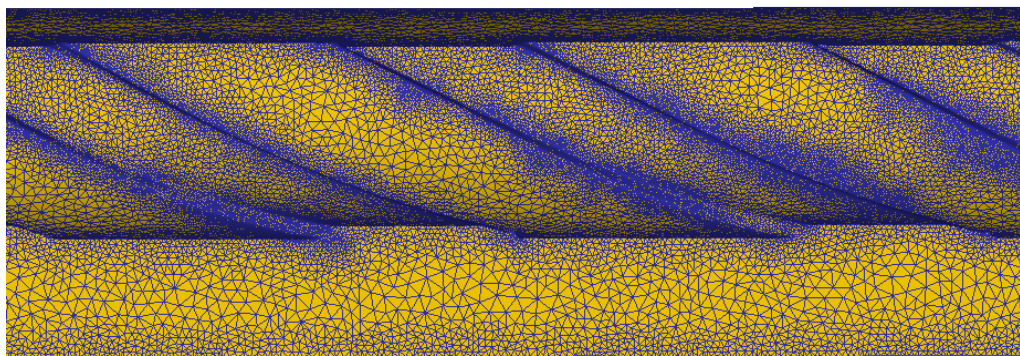
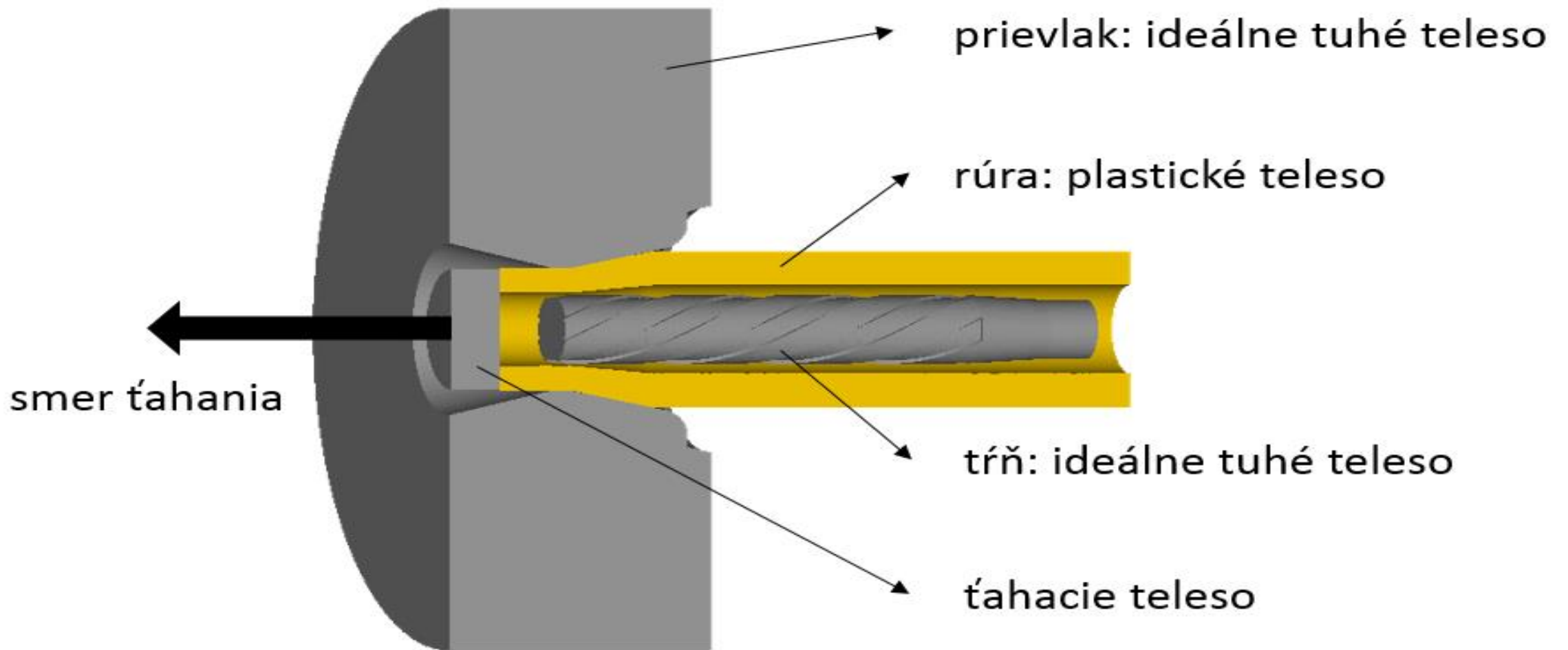
Prievlačný
uhol 6°



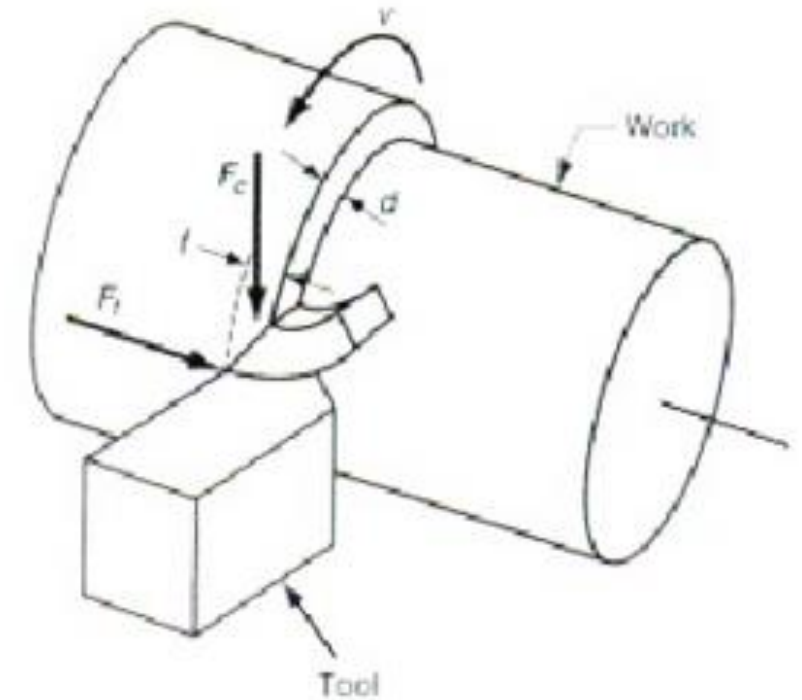
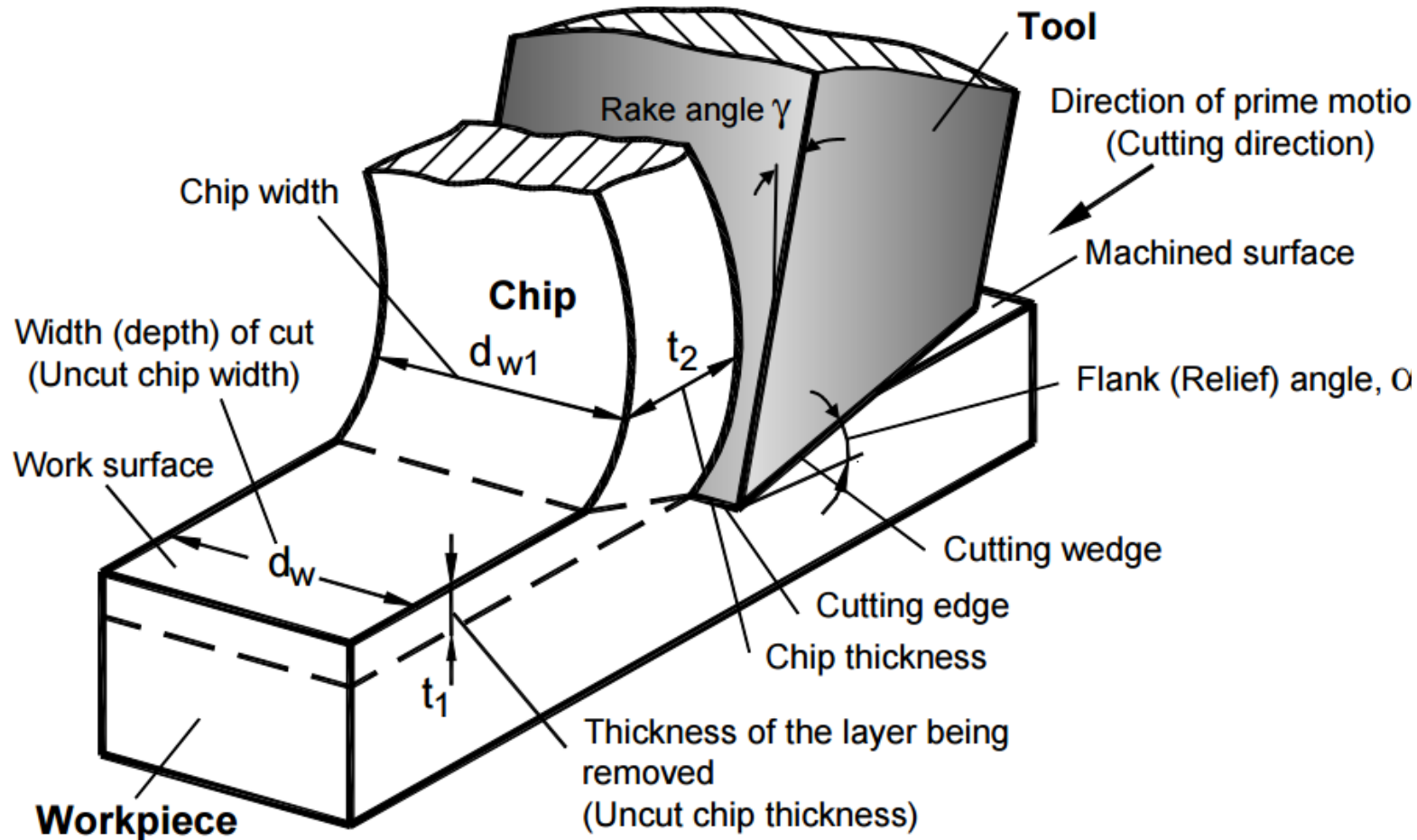
Porovnanie výsledkov simulácie

Prievlačný
uhol 8°

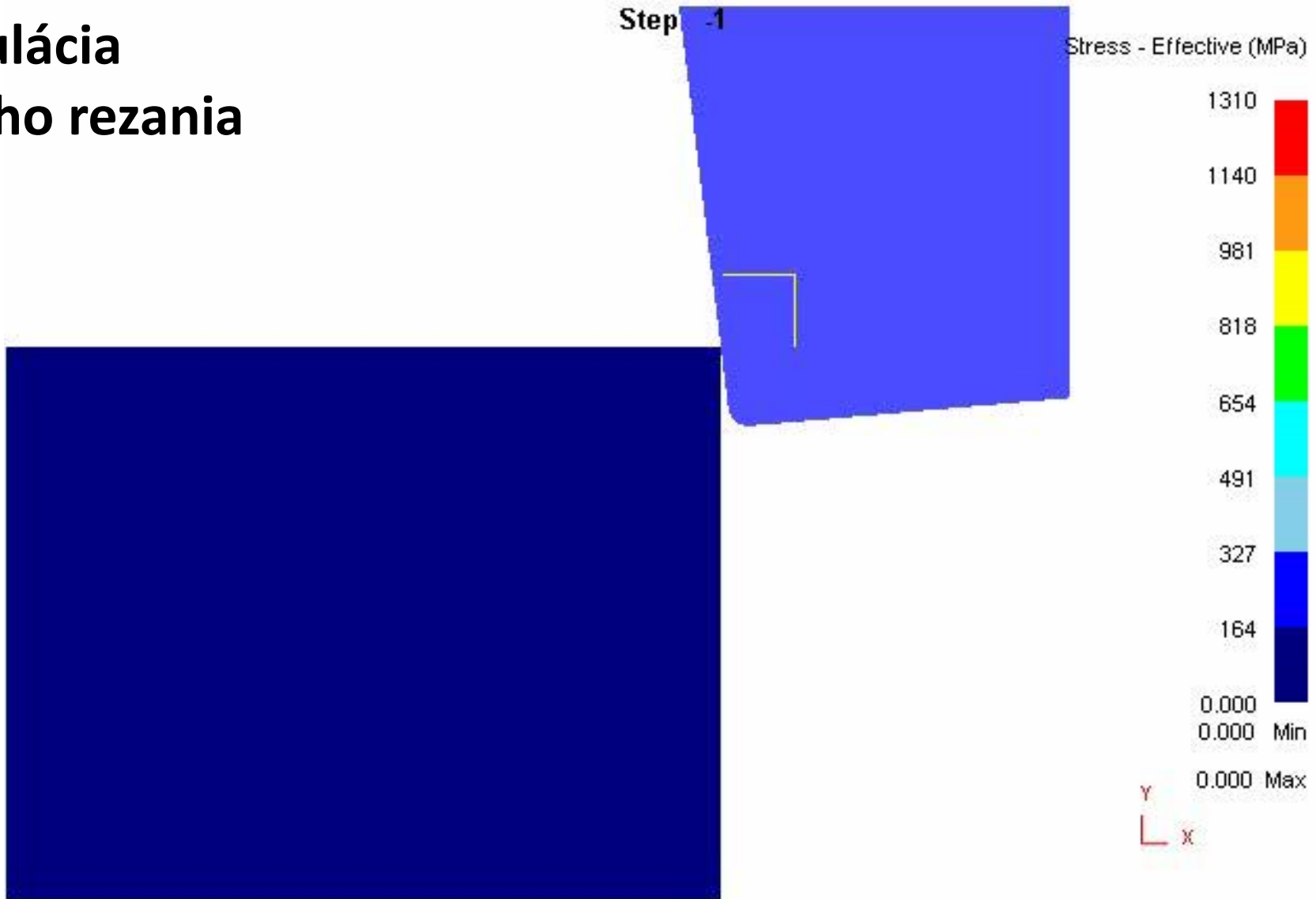


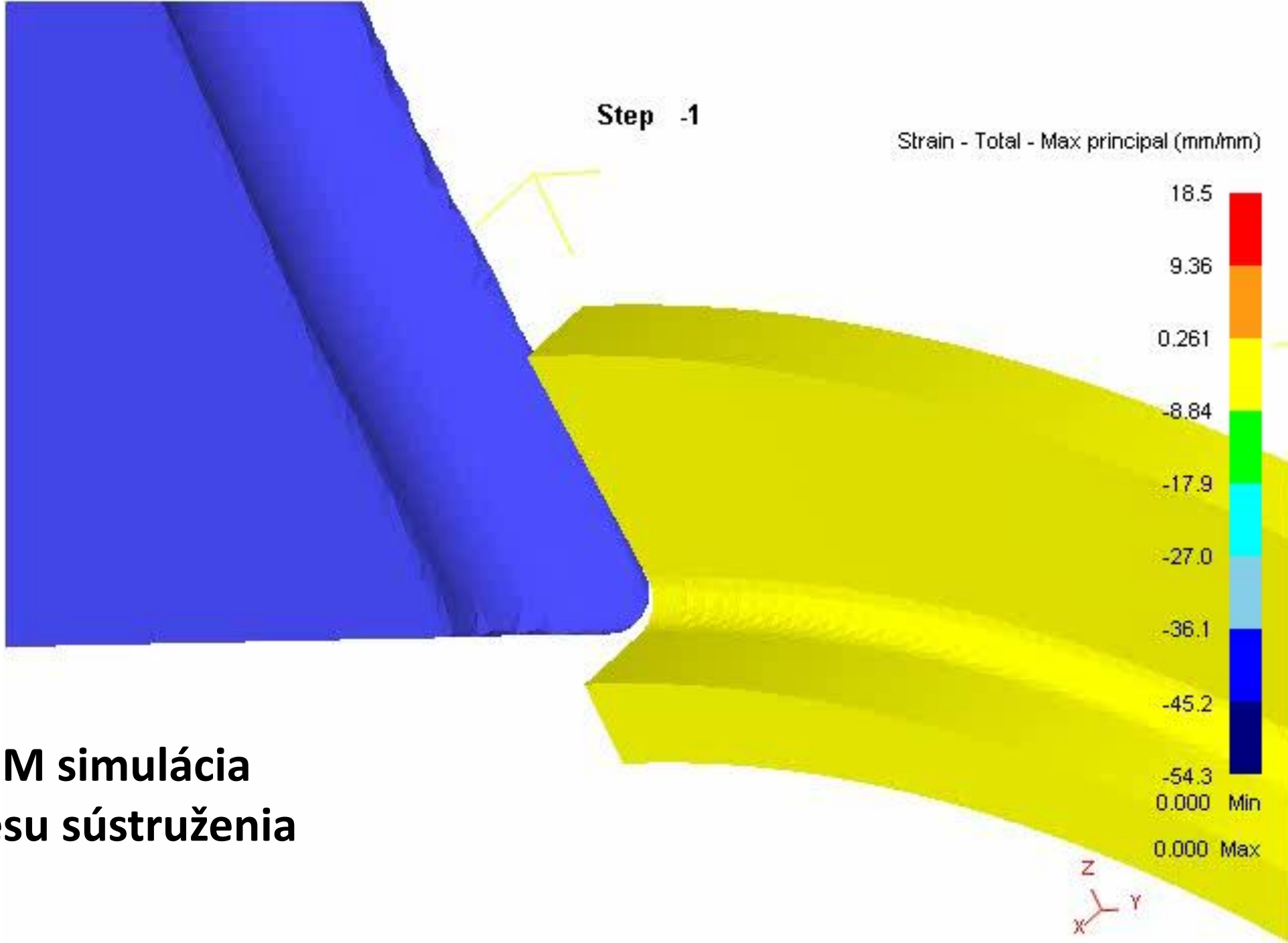


FEM simulácia obrábania



2D FEM simulácia ortogonálneho rezania





**3D FEM simulácia
procesu sústruženia**

LASEROVÁ ABLÁCIA

LASERTEC 80 Shape



Model Builder

- laser_machining.mph (root)
 - Global Definitions
 - Parameters
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Ramp 1 (Ramped_Coefficient)
 - Boundary System 1 (sys1)
 - View 1
 - Axis
 - Geometry 1
 - Materials
 - Heat Transfer in Solids (ht)
 - Deformed Geometry (dg)
 - Fixed Mesh 1
 - Prescribed Mesh Displacement 1
 - Free Deformation 1
 - Prescribed Mesh Displacement 2
 - Prescribed Normal Mesh Velocity 1
 - Equation View
 - Mesh 1
 - Size
 - Size 1
 - Free Triangular 1
 - Study 1
 - Results

Settings Properties

Prescribed Normal Mesh Velocity

Label: Prescribed Normal Mesh Velocity

Boundary Selection

Selection: Manual

Active

3

Override and Contribution

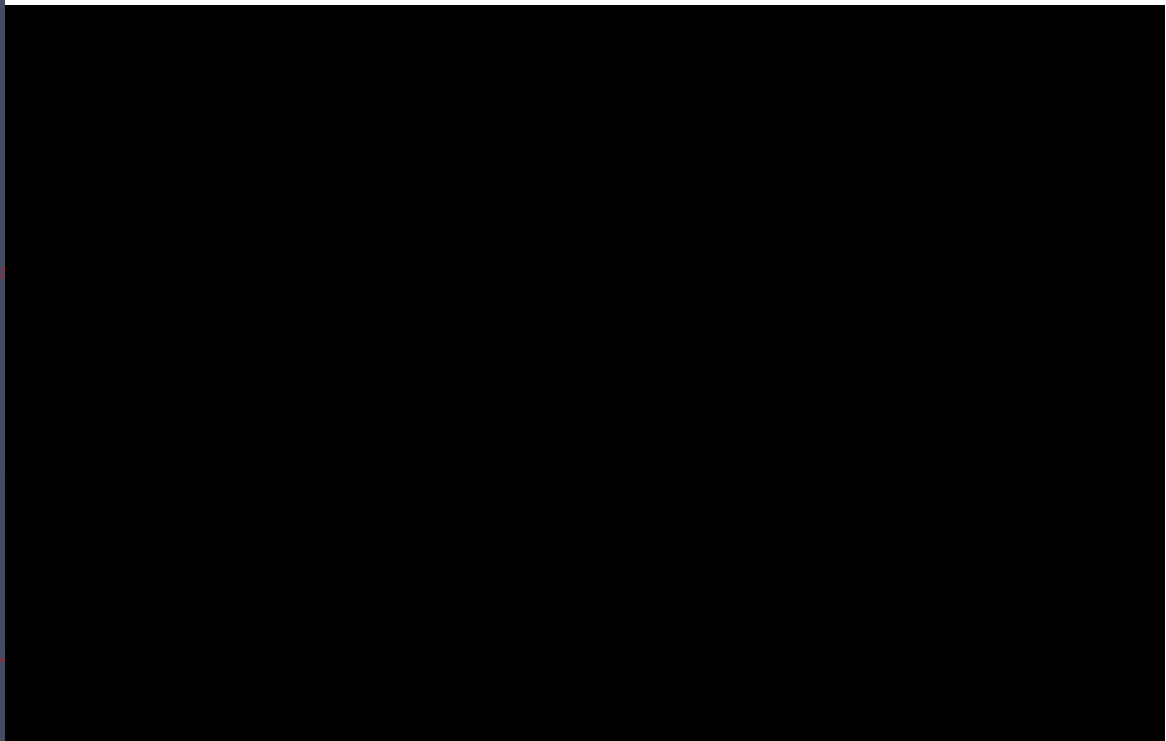
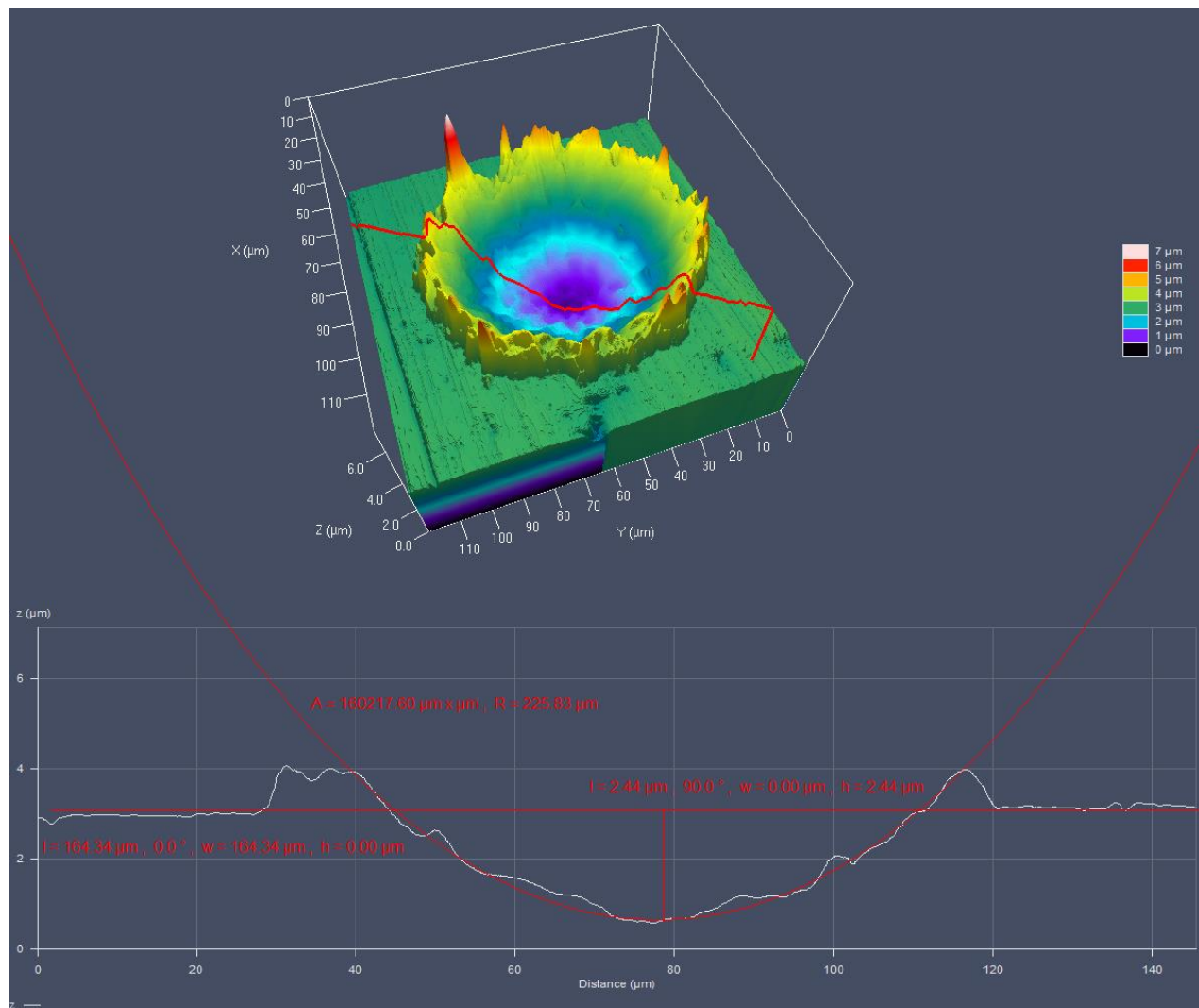
Equation

Normal Mesh Velocity

Normal mesh velocity:

$V_n = \text{ht.hf2.q0}/(1000[\text{kg}/\text{m}^3]*500[\text{kJ}/\text{kg}] \text{ m/s}$

Moving Boundary Smoothing



Ďakujem za pozornosť

Ing. Martin Necpal, PhD.

martin.necpal@stuba.sk