

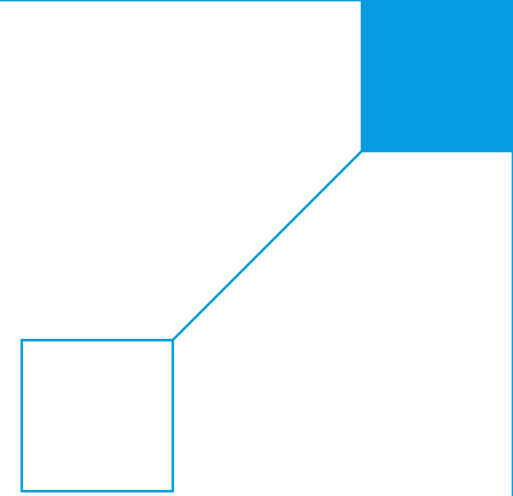
Studsvik RTT Analyzer

A COMSOL[®] App to Evaluate Mechanical Properties for the Nuclear Industry

Daniel Ericsson¹, Joakim Karlsson², Martin König²

¹ Deflexional AB, Täby, Sweden

² Studsvik Nuclear AB, Nyköping, Sweden



Contents

- Introduction
- Using Studsvik RTT Analyzer
- Implementation in COMSOL[®]
- Conclusions

Introduction

Collaboration and background information



Collaboration

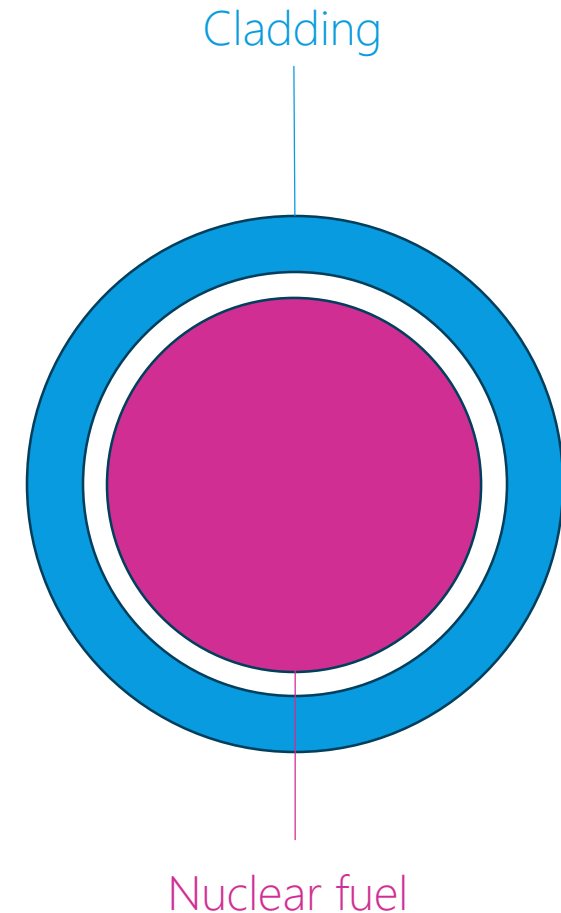
- Studsvik Nuclear AB is a company in the field of nuclear technology and services
 - With a history dating back over seven decades, Studsvik has established itself as a global leader in providing advanced solutions for the nuclear industry
- Deflexional is a COMSOL Certified Consultant with 25 years of experience in simulations with COMSOL Multiphysics®

Studsvik



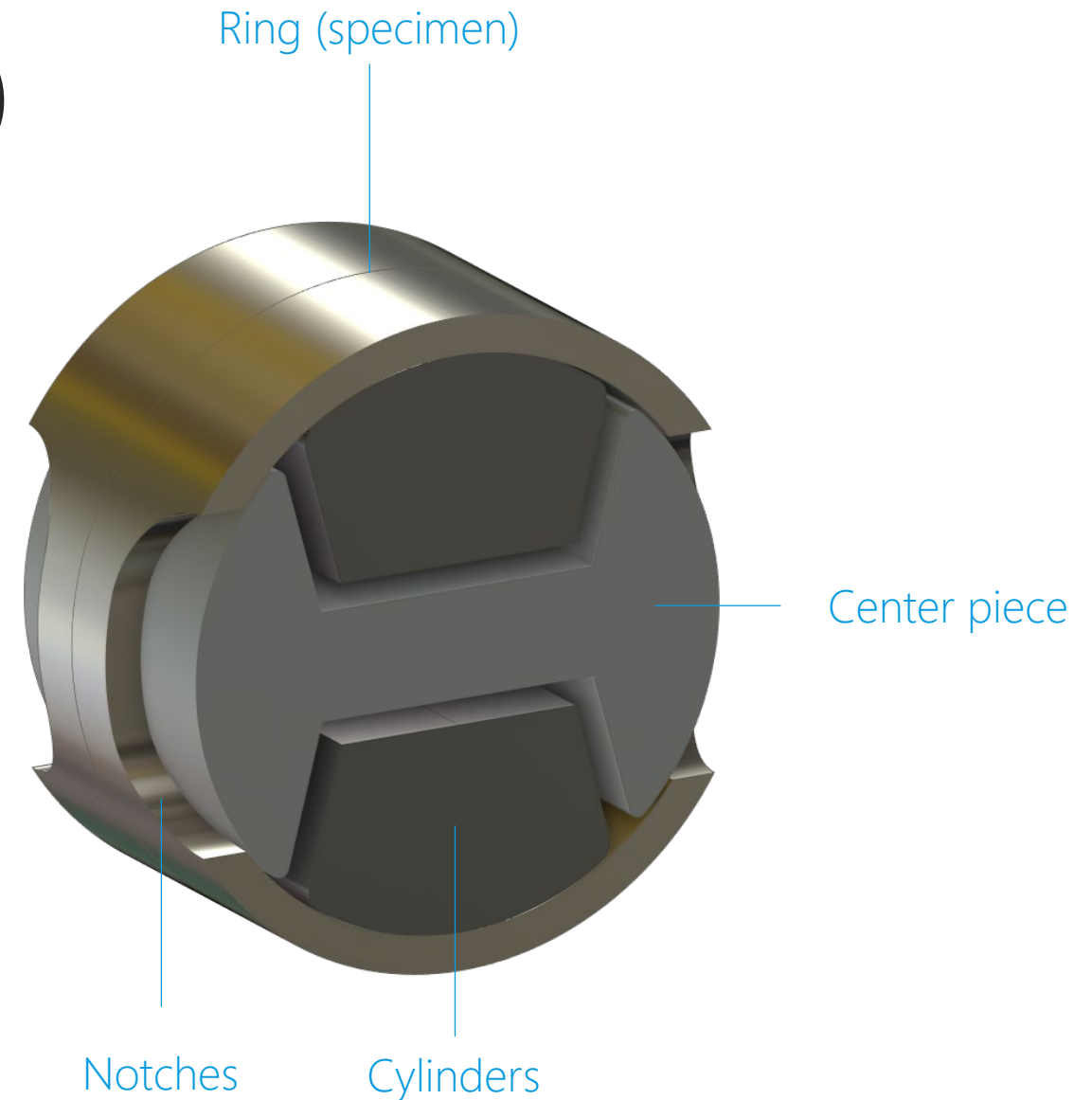
Background

- Cladding is the thin-walled metal tube that forms the outer jacket of a nuclear fuel rod
- The cladding is made of Zircaloy which is an anisotropic material
- Acquiring precise material properties for the hoop direction is imperative due to the predominant internal pressure loading



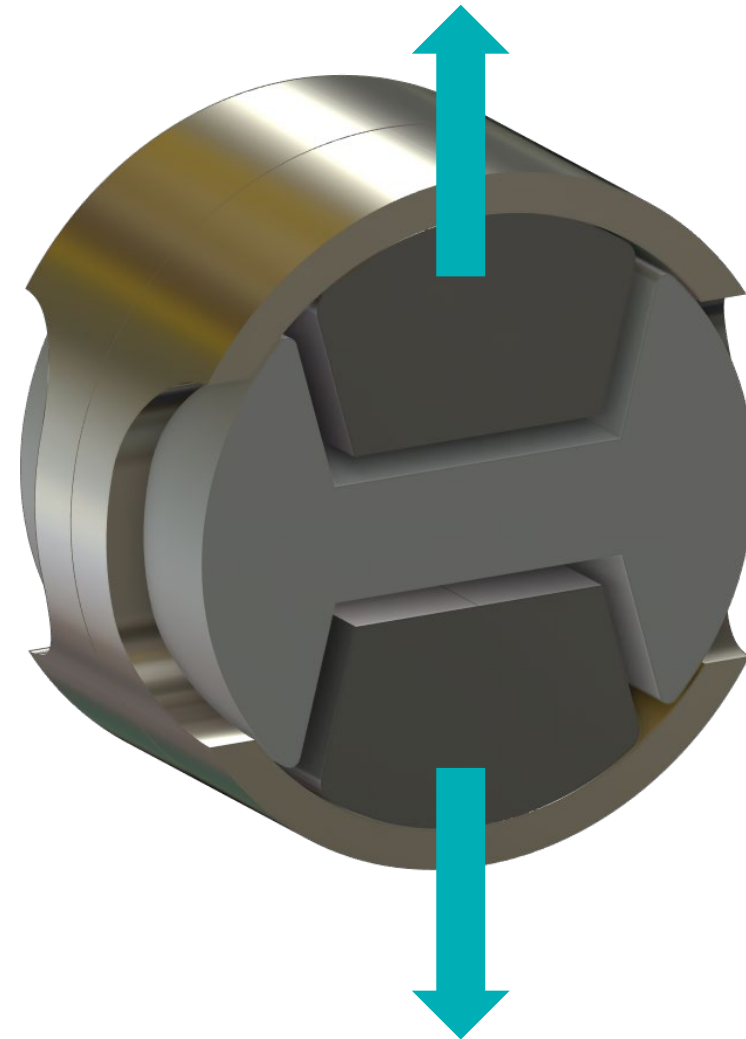
Ring Tensile Tests (RTT)

- Studsvik extensively employs RTT for the evaluation of mechanical properties for cladding
- The specimen is a ring-shaped section cut from the chosen material
- It features symmetrically placed notches for stress concentration, accompanied by a dog bone-shaped center to reduce bending (Ref 1)



Ring Tensile Tests (RTT)

- The specimen is subjected to an increasing applied load
- Force and displacement are recorded
- Previously, an algorithm relied on Excel[®] to calculate yield strength, ultimate strength, and elongation, **but only for specific dimensions**
- **This limitation required a new method** and COMSOL Multiphysics[®] provides a solution through a dedicated COMSOL[®] App



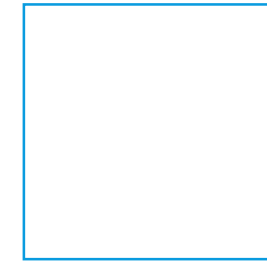
The COMSOL[®] App

- To enhance accessibility, the application is transformed into a standalone compiled app using COMSOL Compiler[™]



Using Studsvik RTT Analyzer

Extract vital material data from measurements

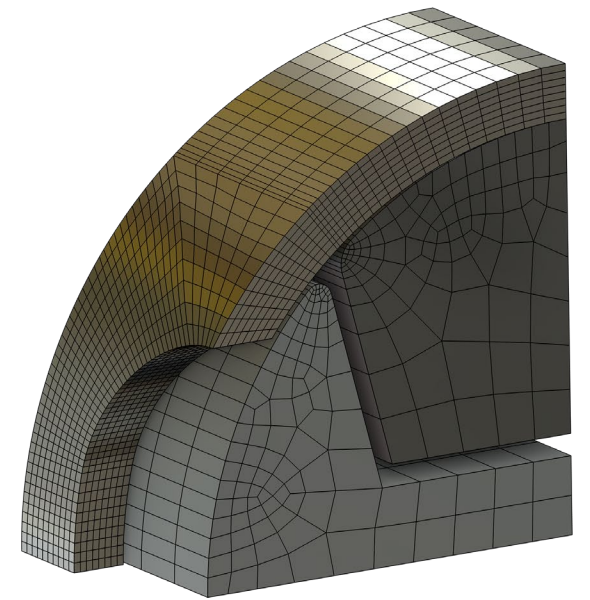
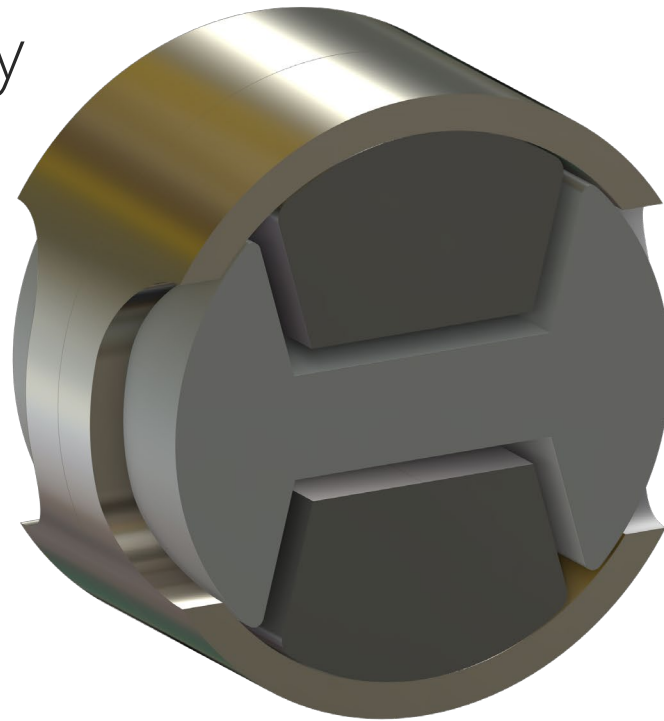


Workflow

- Set dimensions and material data
- Import measured data for force and displacement
- Use a linear elastic study to quickly gain insights to how measurements and simulations correlates
- Perform a plasticity study
- Optionally, use optimization to obtain hardening parameters
- Extract stress-strain graphs to obtain yield strength, ultimate strength, and elongation

Modeling

- 1/8 symmetry is used to save computational time
- The mesh is automatically generated, along with element discretization options

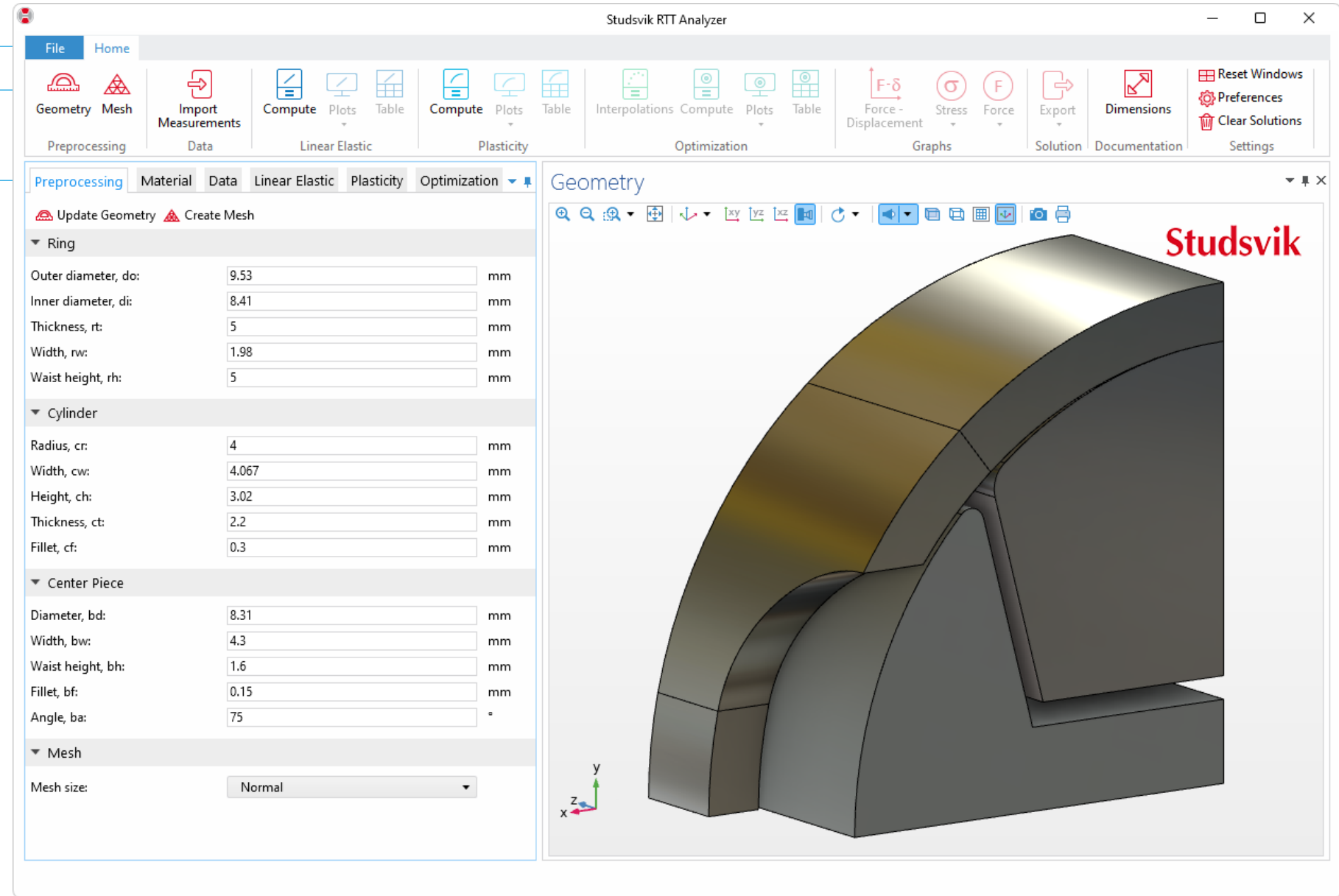


File menu

Ribbon

Tabs

- The different components in the graphical user interface



Settings

Graphics

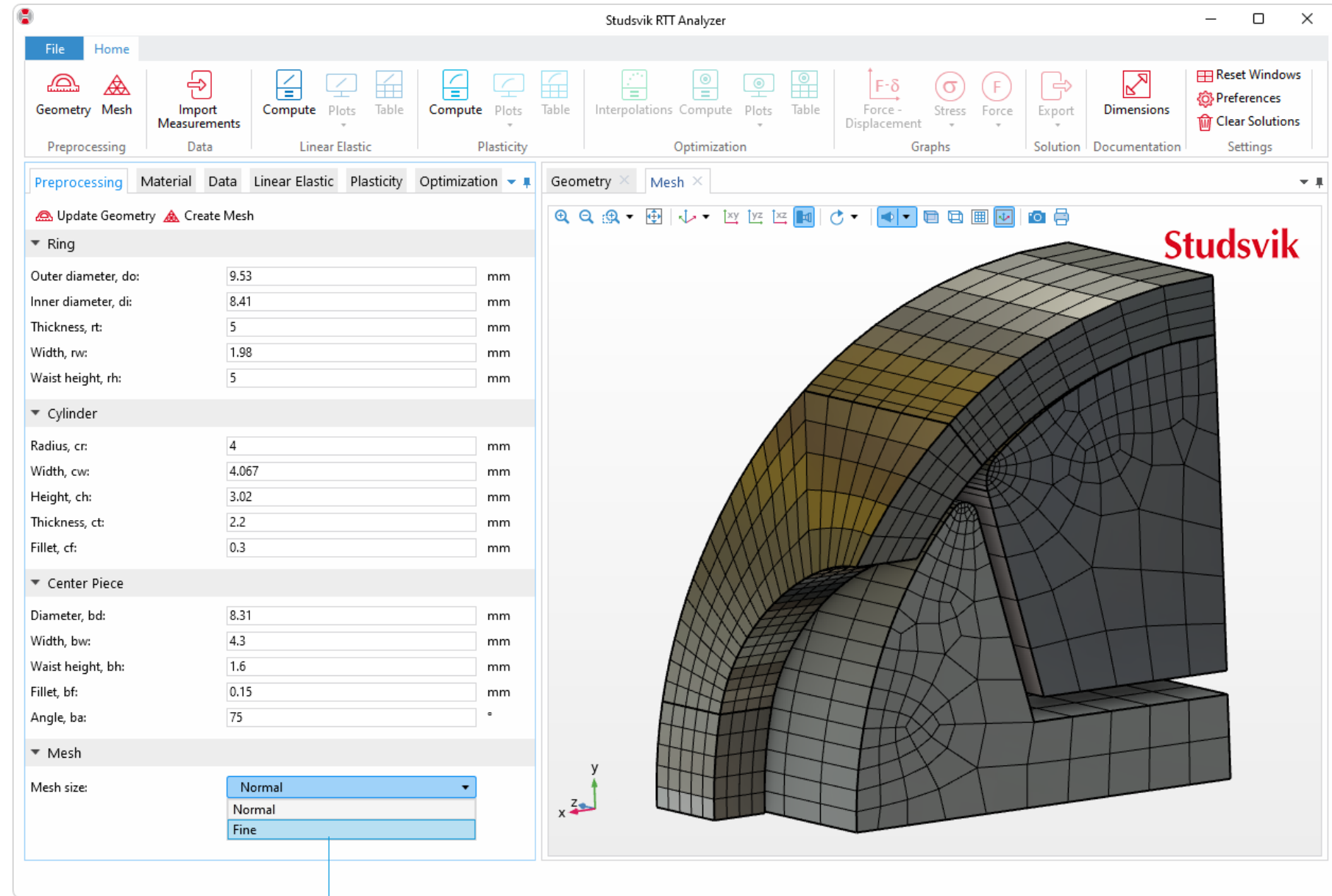
Mesh

- Create the mesh with the *Mesh* button

The screenshot shows the Studsvik RTT Analyzer software interface. The ribbon menu at the top has the 'Mesh' button highlighted. The left panel contains a table of parameters for the meshed part, categorized into Ring, Cylinder, Center Piece, and Mesh. The 3D view on the right shows a detailed mesh of a mechanical part, with a coordinate system (x, y, z) visible at the bottom left.

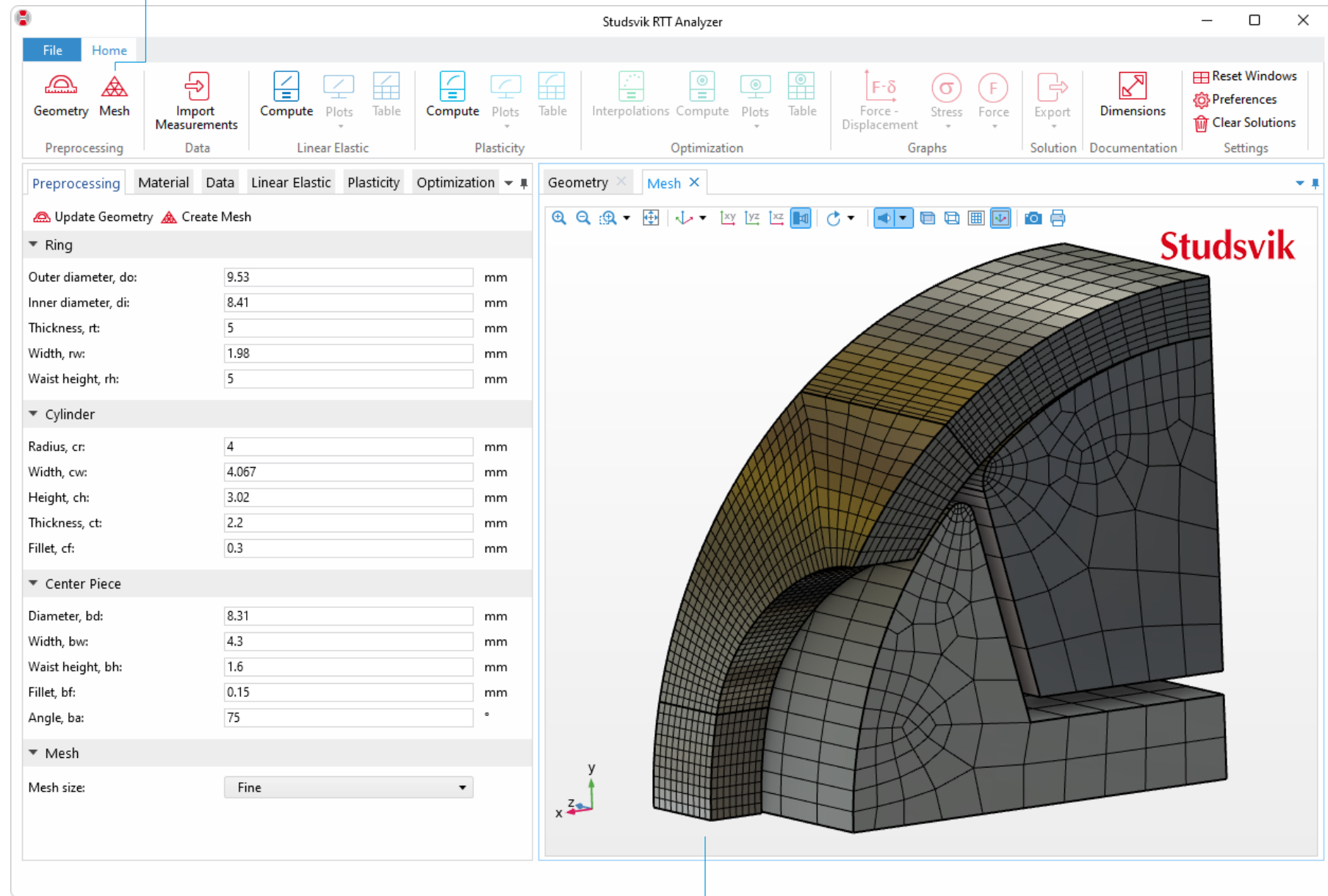
Category	Parameter	Value	Unit
Ring	Outer diameter, do:	9.53	mm
	Inner diameter, di:	8.41	mm
	Thickness, rt:	5	mm
	Width, rw:	1.98	mm
	Waist height, rh:	5	mm
Cylinder	Radius, cr:	4	mm
	Width, cw:	4.067	mm
	Height, ch:	3.02	mm
	Thickness, ct:	2.2	mm
	Fillet, cf:	0.3	mm
Center Piece	Diameter, bd:	8.31	mm
	Width, bw:	4.3	mm
	Waist height, bh:	1.6	mm
	Fillet, bf:	0.15	mm
	Angle, ba:	75	°
Mesh	Mesh size:	Normal	

- Change the mesh size from *Normal* to *Fine*



Mesh size

Mesh



- Note that the mesh is only refined in the ring

Mesh

- Switch back to *Normal* mesh for faster calculations

The screenshot displays the Studsvik RTT Analyzer software interface. The main window shows a 3D model of a mechanical part with a mesh applied. The mesh is currently set to 'Normal' in the 'Mesh' section of the 'Preprocessing' tab. The 'Mesh size' is set to 'Normal'. The 'Preprocessing' tab is active, and the 'Mesh' section is expanded. The 'Ring' section contains the following parameters:

Parameter	Value	Unit
Outer diameter, do:	9.53	mm
Inner diameter, di:	8.41	mm
Thickness, rt:	5	mm
Width, rw:	1.98	mm
Waist height, rh:	5	mm

The 'Cylinder' section contains the following parameters:

Parameter	Value	Unit
Radius, cr:	4	mm
Width, cw:	4.067	mm
Height, ch:	3.02	mm
Thickness, ct:	2.2	mm
Fillet, cf:	0.3	mm

The 'Center Piece' section contains the following parameters:

Parameter	Value	Unit
Diameter, bd:	8.31	mm
Width, bw:	4.3	mm
Waist height, bh:	1.6	mm
Fillet, bf:	0.15	mm
Angle, ba:	75	°

The 'Mesh' section contains the following parameter:

Parameter	Value	Unit
Mesh size:	Normal	

The 3D model in the main window shows a complex mechanical part with a mesh applied. The mesh is currently set to 'Normal' in the 'Mesh' section of the 'Preprocessing' tab. The 'Mesh size' is set to 'Normal'. The 'Preprocessing' tab is active, and the 'Mesh' section is expanded. The 'Ring' section contains the following parameters:

Parameter	Value	Unit
Outer diameter, do:	9.53	mm
Inner diameter, di:	8.41	mm
Thickness, rt:	5	mm
Width, rw:	1.98	mm
Waist height, rh:	5	mm

The 'Cylinder' section contains the following parameters:

Parameter	Value	Unit
Radius, cr:	4	mm
Width, cw:	4.067	mm
Height, ch:	3.02	mm
Thickness, ct:	2.2	mm
Fillet, cf:	0.3	mm

The 'Center Piece' section contains the following parameters:

Parameter	Value	Unit
Diameter, bd:	8.31	mm
Width, bw:	4.3	mm
Waist height, bh:	1.6	mm
Fillet, bf:	0.15	mm
Angle, ba:	75	°

The 'Mesh' section contains the following parameter:

Parameter	Value	Unit
Mesh size:	Normal	

The 3D model in the main window shows a complex mechanical part with a mesh applied. The mesh is currently set to 'Normal' in the 'Mesh' section of the 'Preprocessing' tab. The 'Mesh size' is set to 'Normal'. The 'Preprocessing' tab is active, and the 'Mesh' section is expanded. The 'Ring' section contains the following parameters:

Parameter	Value	Unit
Outer diameter, do:	9.53	mm
Inner diameter, di:	8.41	mm
Thickness, rt:	5	mm
Width, rw:	1.98	mm
Waist height, rh:	5	mm

The 'Cylinder' section contains the following parameters:

Parameter	Value	Unit
Radius, cr:	4	mm
Width, cw:	4.067	mm
Height, ch:	3.02	mm
Thickness, ct:	2.2	mm
Fillet, cf:	0.3	mm

The 'Center Piece' section contains the following parameters:

Parameter	Value	Unit
Diameter, bd:	8.31	mm
Width, bw:	4.3	mm
Waist height, bh:	1.6	mm
Fillet, bf:	0.15	mm
Angle, ba:	75	°

The 'Mesh' section contains the following parameter:

Parameter	Value	Unit
Mesh size:	Normal	

Benchmark Test of an Aluminum Alloy

- Tests are normally performed for Zircalloys
 - The test temperature is high
- In this benchmark an aluminum alloy is tested in room temperature

Material settings

- Open the *Material* tab
- Enter material data
- Thermal expansion parameters are of huge importance when the test is performed in high temperatures
- Friction can be included

The screenshot displays the Studsvik RTT Analyzer interface. The 'Material' tab is active, showing settings for three components: Ring, Cylinder, and Center Piece. The 'Friction' section is checked, with a coefficient of 0.125. The 3D model on the right shows a complex, curved structure with a mesh overlay. The Studsvik logo is visible in the top right corner of the model view.

Property	Value	Unit
Ring		
Poisson's ratio:	0.33	
Density:	2810	kg/m ³
Thermal expansion coefficient:	6e-6	1/K
Cylinder		
Young's modulus:	205	GPa
Poisson's ratio:	0.33	
Density:	8117	kg/m ³
Thermal expansion coefficient:	1.8e-5	1/K
Center Piece		
Young's modulus:	205	GPa
Poisson's ratio:	0.33	
Density:	8117	kg/m ³
Thermal expansion coefficient:	1.8e-5	1/K
Thermal Properties		
Reference temperature:	25	°C
Test temperature:	25	°C
Friction		
<input checked="" type="checkbox"/> Include friction		
Friction coefficient:	0.125	

Import measured force and displacement

- Press *Import Measurements* to import force and displacement data

Studsvik RTT Analyzer

File Home

Geometry Mesh **Import Measurements** Compute Plots Table Compute Plots Table Interpolations Compute Plots Table Force - Displacement Stress Force Export Dimensions Reset Windows Preferences Clear Solutions

Preprocessing Data Imports and plots measured data. Plasticity Optimization Graphs Solution Documentation Settings

Preprocessing **Material** Data Linear Elastic Plasticity Optimization

▼ Ring

Poisson's ratio: 0.33

Density: 2810 kg/m³

Thermal expansion coefficient: 6e-6 1/K

▼ Cylinder

Young's modulus: 205 GPa

Poisson's ratio: 0.33

Density: 8117 kg/m³

Thermal expansion coefficient: 1.8e-5 1/K

▼ Center Piece

Young's modulus: 205 GPa

Poisson's ratio: 0.33

Density: 8117 kg/m³

Thermal expansion coefficient: 1.8e-5 1/K

▼ Thermal Properties

Reference temperature: 25 °C

Test temperature: 25 °C

▼ Friction

Include friction

Friction coefficient: 0.125

Studsvik

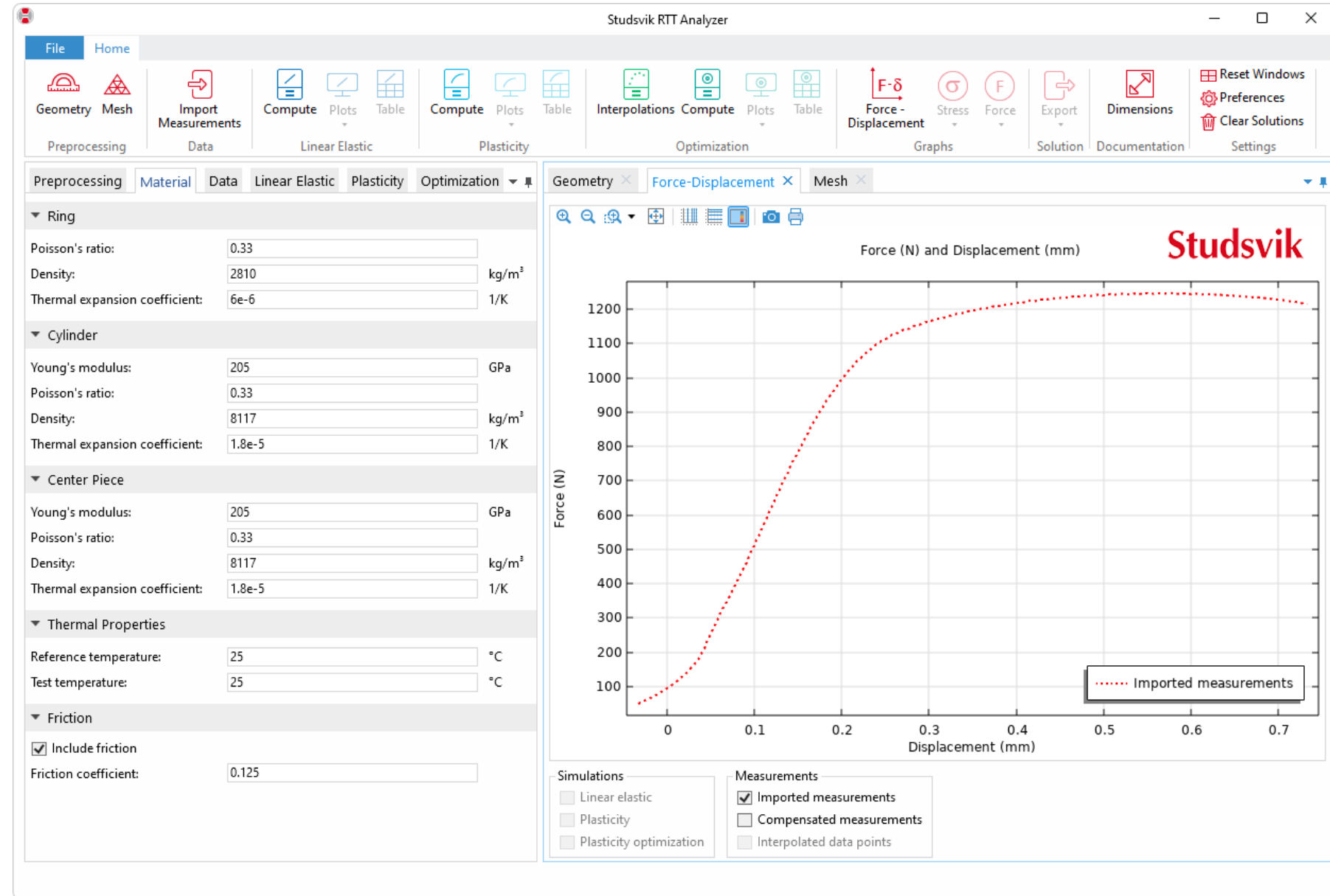
x y z

- Paste data from Excel[®] or import files

The screenshot displays the Studsvik RTT Analyzer software interface. The main window shows a 3D model of a mechanical part with a mesh overlay. The software is divided into several sections: Preprocessing, Data, Imports and plots measured data, Plasticity, Optimization, Graphs, Solution, Documentation, and Settings. The 'Material' section is active, showing properties for three components: Ring, Cylinder, and Center Piece. The 'Import Measurements' dialog box is open, displaying a table of data.

Displacement (mm)	Force (N)
0.033	171.12
0.036	180.37
0.038	189.47
0.040	200.48
0.043	211.96
0.044	222.13
0.046	234.99
0.049	248.42
0.051	259.55
0.053	271.82
0.056	285.16
0.058	296.13
0.059	306.06
0.062	318.27
0.064	330.09
0.067	341.06
0.069	352.31
0.072	364.87
0.074	375.83
0.076	387.21
0.079	400.79
0.081	412.32
0.083	423.10
0.086	436.53
0.088	449.21
0.091	460.41

- A force and displacement graph is automatically plotted

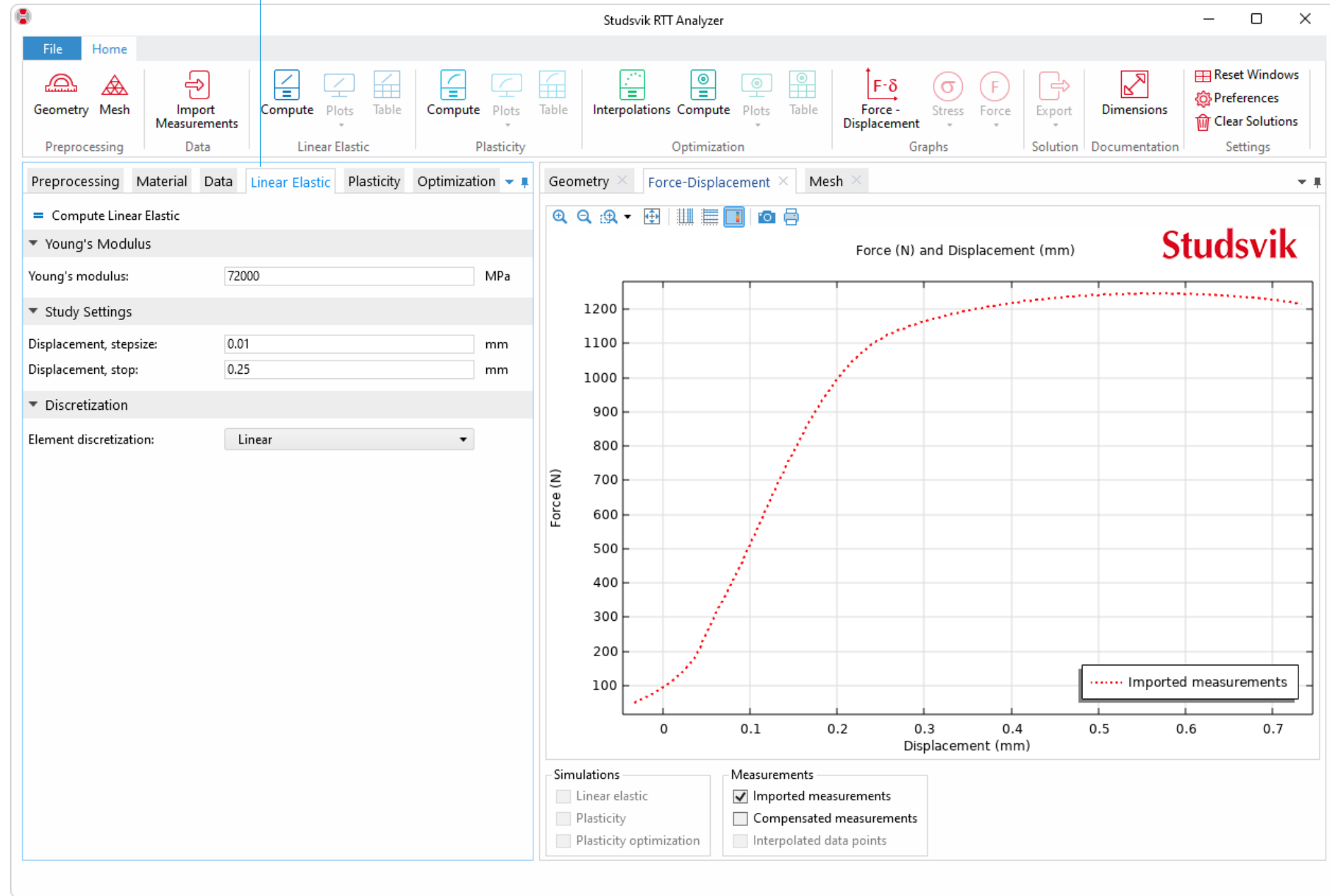


Linear Elastic Analysis

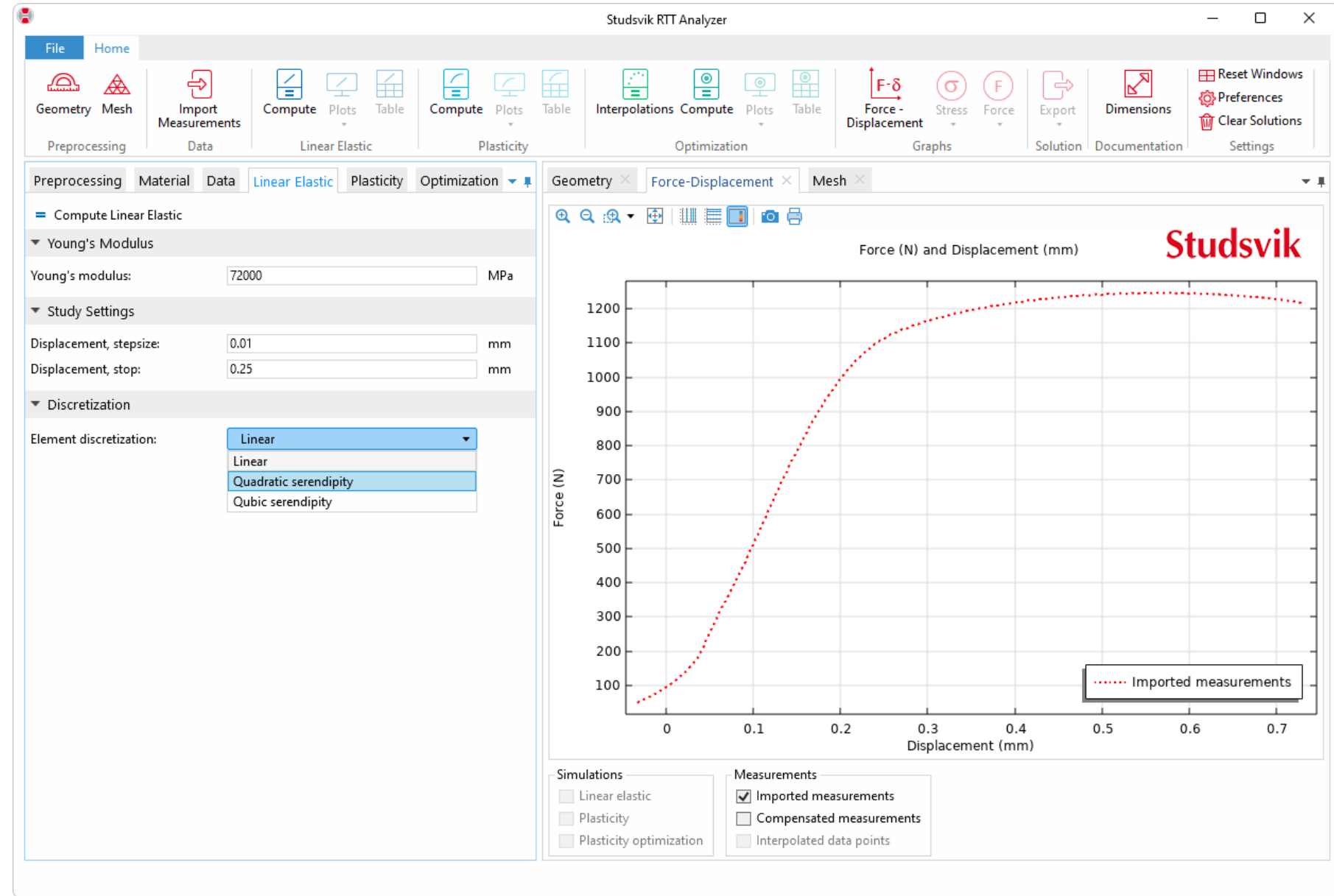
- Young's Modulus is known
- Begin with a linear elastic study to obtain results fast
- Compare the results with the measured data

Linear Elastic settings

- Click on the *Linear Elastic* tab



- Switch to Quadratic elements

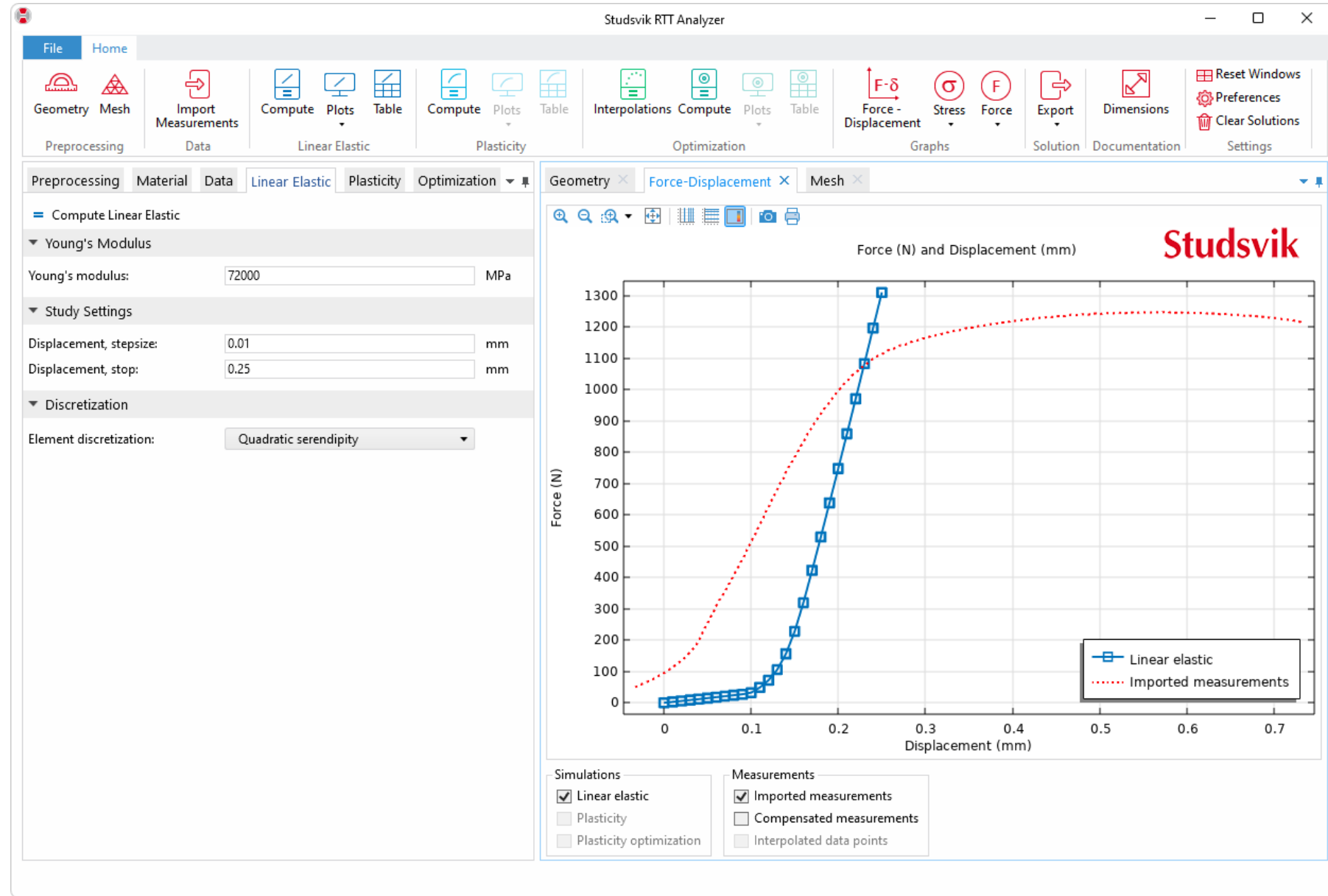


- Enter Young's modulus which is a well-known parameter for most materials
- In the *Study Settings* section, enter the displacement step size and stop
- Press any of the *Compute* buttons to start the simulation

Compute

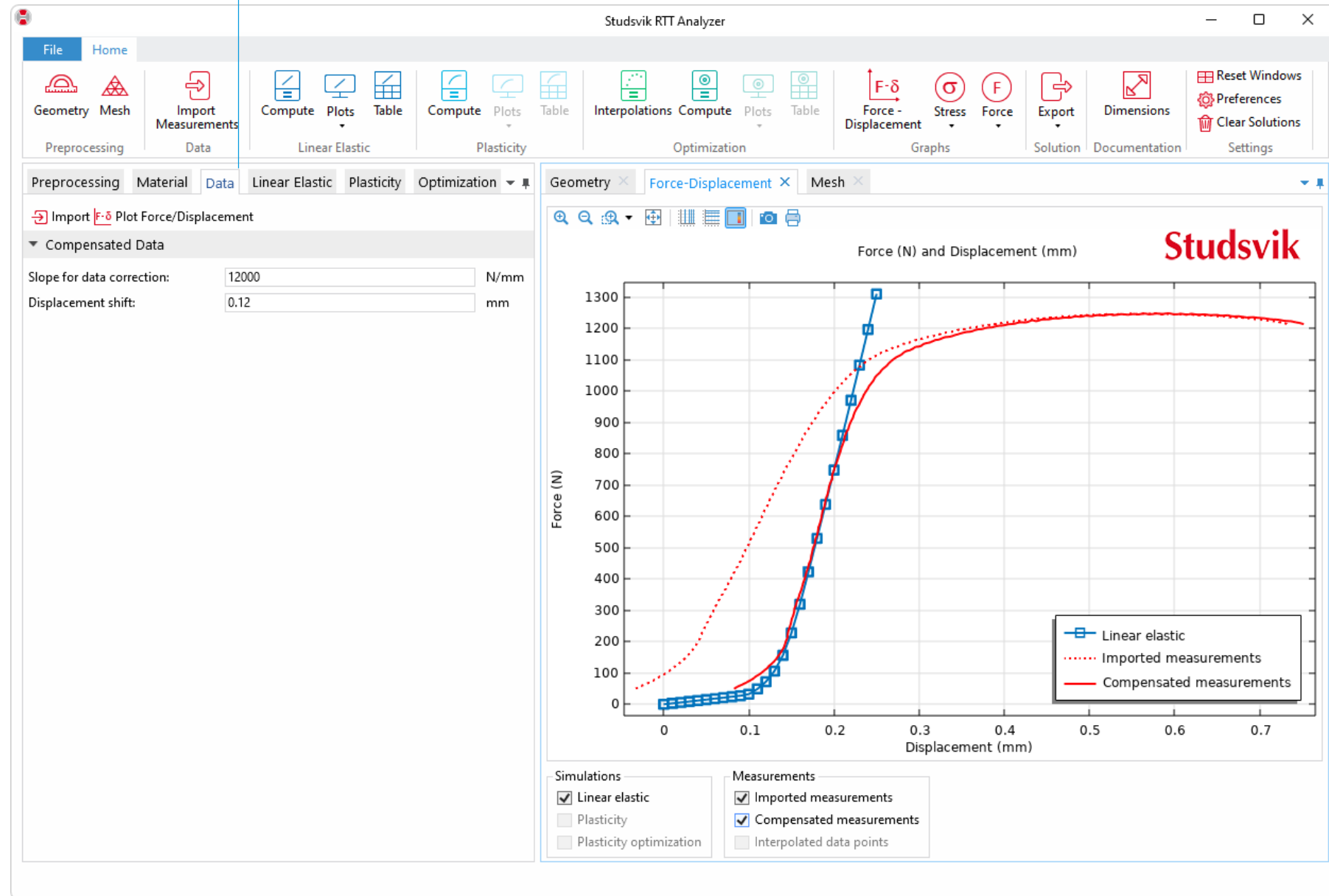
The screenshot shows the Studsvik RTT Analyzer software interface. The main window is titled 'Studsvik RTT Analyzer' and features a ribbon menu with various tabs: File, Home, Geometry, Mesh, Import Measurements, Compute, Plots, Table, Interpolations, Compute, Plots, Table, Force-Displacement, Stress, Force, Export, Dimensions, and Settings. The 'Compute' button is highlighted with a blue arrow and the word 'Compute' above it. Below the ribbon, the 'Compute Linear Elastic' settings are displayed, including Young's Modulus (72000 MPa), Study Settings (Displacement step size: 0.01 mm, Displacement stop: 0.25 mm), and Discretization (Element discretization: Quadratic serendipity). On the right, a plot titled 'Force (N) and Displacement (mm)' shows a red dotted line representing 'Imported measurements'. The plot shows Force (N) on the y-axis (ranging from 0 to 1200) and Displacement (mm) on the x-axis (ranging from 0 to 0.7). The curve starts at (0,0) and rises to a peak of approximately 1200 N at 0.5 mm displacement, then slightly decreases. A legend in the bottom right of the plot area identifies the red dotted line as 'Imported measurements'. Below the plot, there are checkboxes for 'Simulations' (Linear elastic, Plasticity, Plasticity optimization) and 'Measurements' (Imported measurements, Compensated measurements, Interpolated data points). The 'Imported measurements' checkbox is checked.

- The simulation and the imported measurements don't match due to test rig deformations



Data settings

- Click on the *Data* tab
- In the *Compensated Data* section, enter a slope correction and a shift to fit the compensated measurements with the linear elastic study



Plasticity

- Depending on the tested material, the use of different hardening models are required
- Studsvik RTT Analyzer supports several hardening models and it's possible to define user-defined functions
- Since the test performed in this presentation is an aluminum alloy, the *Ludwik* hardening model is used

Preprocessing Material Data Linear Elastic Plasticity Optimization

Compute Plasticity

Young's Modulus

Plasticity

Plasticity model: Large plastic strains

Isotropic hardening model: Ludwik

Equation: Perfectly plastic
Linear
Ludwik MPa
Interpolation MPa
Function
Bilinear softening
Exponential softening
Power Law softening
Lognormal softening

Plot

Minimum plastic strains : 1

Maximum plastic strains : 1

Plot Hardening

Study Settings

Discretization

- Click the *Plasticity* tab
- Set Young's Modulus or use the same as in the linear elastic study
- Select isotropic hardening model and the hardening parameters
- Define the steps for the simulation in the *Study Settings* section

Plasticity settings

The screenshot displays the Studsvik RTT Analyzer software interface. The **Plasticity** tab is selected in the top navigation bar. The settings panel on the left is configured as follows:

- Compute Plasticity**
 - Young's Modulus: From linear elastic study (72000 MPa), User-defined
 - Plasticity model: Large plastic strains
 - Isotropic hardening model: Ludwik
 - Equation: $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$
 - σ_{ys0} : 387 MPa
 - k : 425 MPa
 - n : 0.17
 - Plot: Plot Hardening
- Study Settings**
 - Step type: Start/Step/Stop
 - Displacement, start: 0 mm
 - Displacement, stepsize: 0.02 mm
 - Displacement, stop: 0.74 mm
- Discretization**
 - Element discretization: Linear

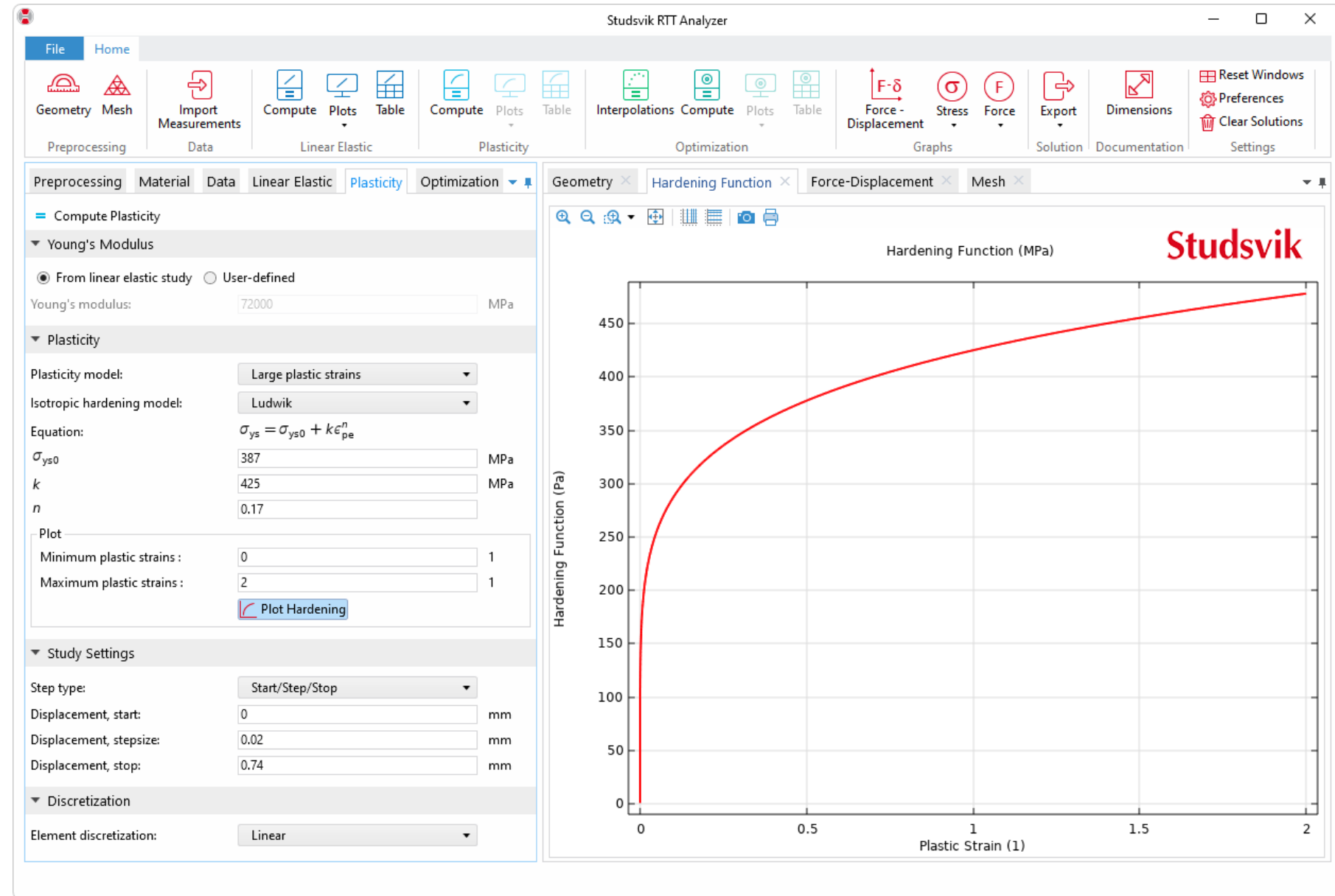
The main window displays a **Force-Displacement** graph titled "Force (N) and Displacement (mm)". The graph shows three data series:

- Linear elastic**: Represented by a blue line with square markers, showing a linear relationship up to approximately 250 N.
- Imported measurements**: Represented by a red dotted line, showing the full force-displacement curve up to 1300 N.
- Compensated measurements**: Represented by a solid red line, which follows the linear elastic region and then smoothly transitions to the imported measurements.

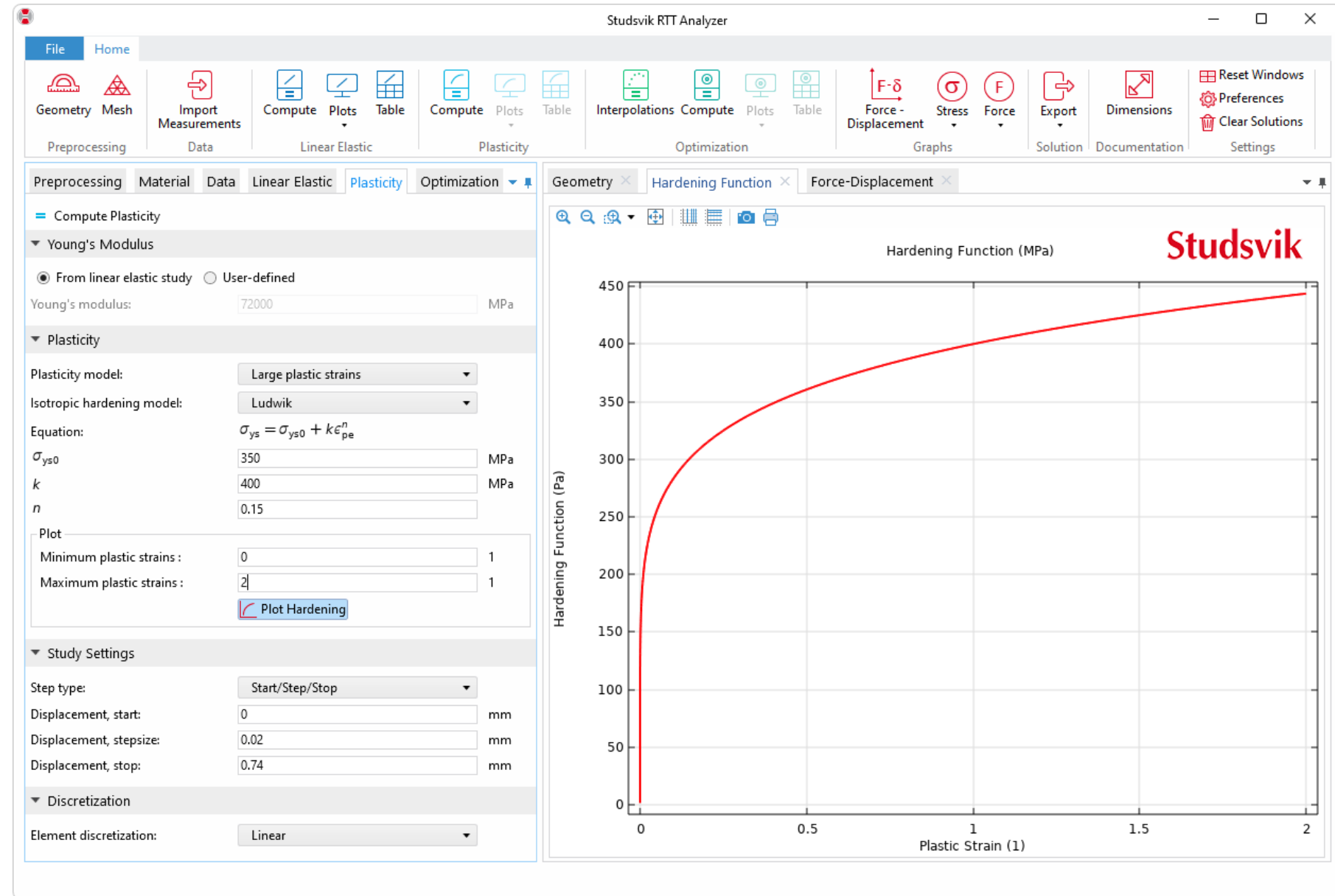
At the bottom of the interface, there are two panels:

- Simulations**: Linear elastic, Plasticity, Plasticity optimization
- Measurements**: Imported measurements, Compensated measurements, Interpolated data points

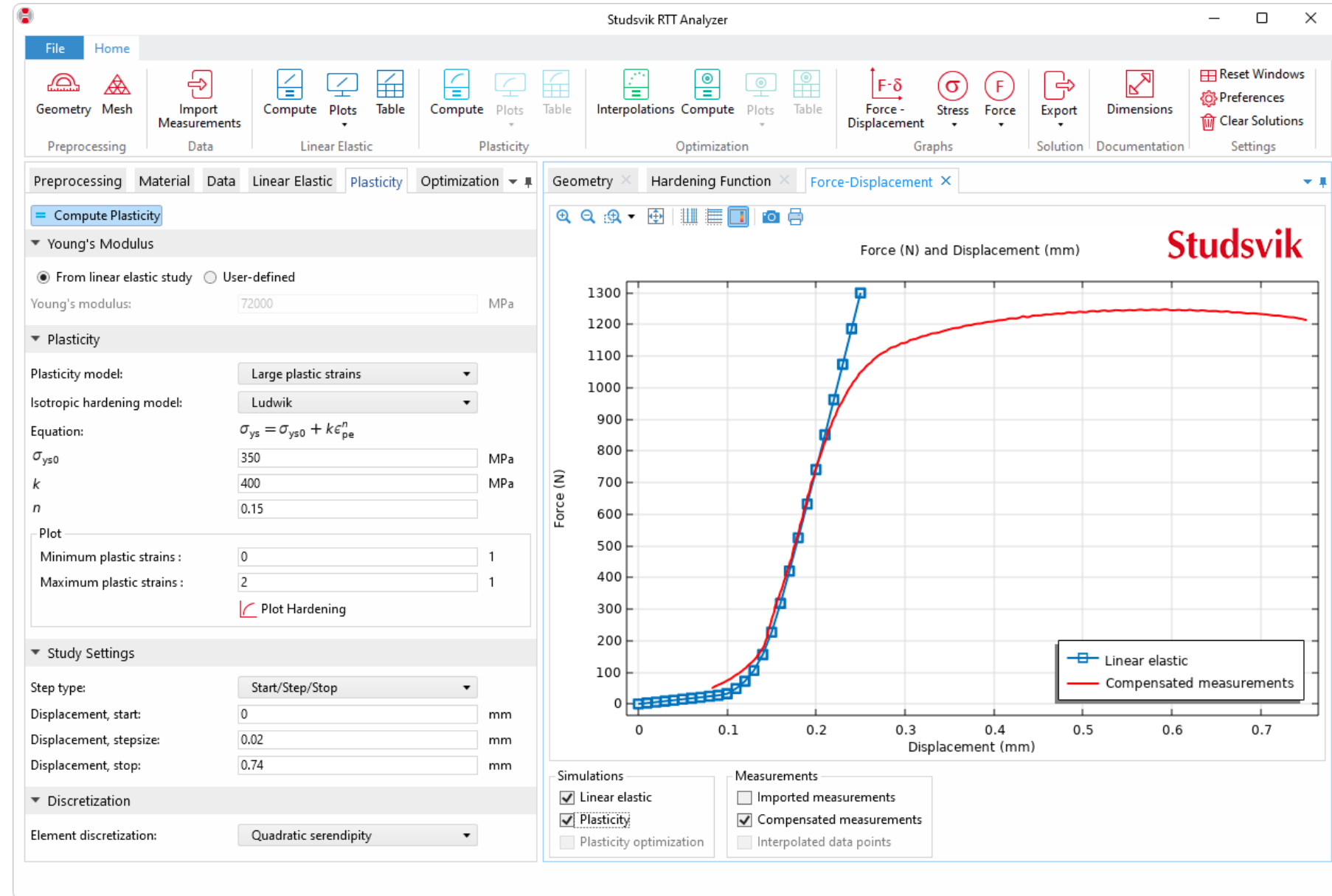
- Press the *Plot Hardening* button to visualize the selected hardening function



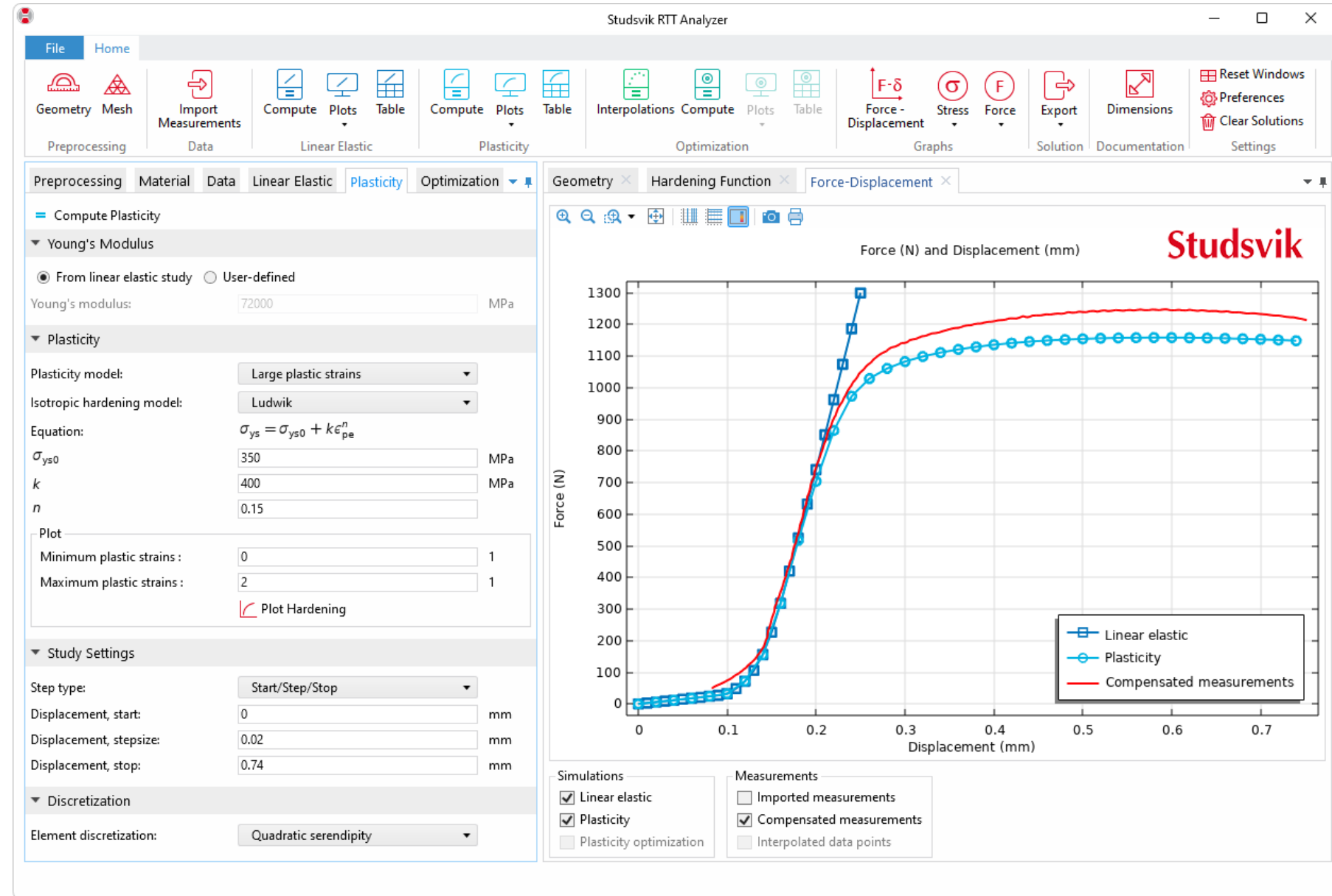
- Enter new hardening parameters and plot the hardening function



- Change the *Element discretization* to *Quadratic serendipity* for better accuracy
- Press *Compute Plasticity*



- The result is not satisfactory, and the user can try with new parameters to fit the plasticity results to the compensated measurements



Optimization

- The app provides optimization routines utilizing the Optimization Module
- Using the BOBYQA method, the hardening parameters are automatically fitted to the measured data
- The user can easily select which parameters that should be optimized

▼ Plasticity

Plasticity model:

Isotropic hardening model:

Equation: $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$

▼ Plasticity Parameters

Optimize σ_{ys0}

Initial value (MPa)	Scale (MPa)	Lower bound (MPa)	Upper bound (MPa)
387	387	300	450

Optimize k

Initial value (MPa)	Scale (MPa)	Lower bound (MPa)	Upper bound (MPa)
400	400	300	500

Optimize n

Initial value (MPa)	Scale (MPa)	Lower bound (MPa)	Upper bound (MPa)
0.1	0.1	0.05	0.25

Plot Hardening Initial Values

▼ Optimization Settings

Optimality tolerance:

Maximum number of evaluations:

Optimization settings

- Click the *Optimization* tab

The screenshot displays the Studsvik RTT Analyzer interface. The **Optimization** tab is selected in the top ribbon and the left-hand settings pane. The settings pane includes sections for Interpolation Data Points, Young's Modulus, Plasticity, and Plasticity Parameters.

Interpolation Data Points:

- Displacement start: 0.14 mm
- Displacement step size: 0.01 mm
- Displacement end: 0.3 mm

Young's Modulus:

- From linear elastic study (selected)
- Young's modulus: 72000 MPa

Plasticity:

- Plasticity model: Large plastic strains
- Isotropic hardening model: Ludwik
- Equation: $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$

Plasticity Parameters:

Parameter	Optimize	Initial value (MPa)	Scale (MPa)	Lower bound (MPa)	Upper bound (MPa)
σ_{ys0}	<input checked="" type="checkbox"/>	387	387	300	450
k	<input checked="" type="checkbox"/>	400	400	300	500
n	<input checked="" type="checkbox"/>	0.1	0.1	0.05	0.25

The right-hand pane shows a plot titled "Force (N) and Displacement (mm)". The plot compares "Plasticity" (blue circles) and "Compensated measurements" (red line). The force increases from 0 N at 0 mm displacement to approximately 1150 N at 0.7 mm displacement. The "Compensated measurements" line closely follows the "Plasticity" data points.

Simulations:

- Linear elastic
- Plasticity
- Plasticity optimization

Measurements:

- Imported measurements
- Compensated measurements
- Interpolated data points

- In the *Interpolation Data Points* section, enter start, step size and stop
- This data will create points following the *Compensated measured* graph
- The optimization will use these interpolation points as “targets”
- Press *Create Interpolation Data Points*

Studsвик RTT Analyzer

File Home

Geometry Mesh Import Measurements Compute Plots Table Compute Plots Table Interpolations Compute Plots Table Force - Displacement Stress Force Export Dimensions Reset Windows Preferences Clear Solutions

Preprocessing Data Linear Elastic Plasticity Optimization Graphs Solution Documentation Settings

Preprocessing Material Data Linear Elastic Plasticity Optimization

Create Interpolation Data Points = Compute Optimization

Interpolation Data Points

Displacement start: 0.14 mm
 Displacement step size: 0.02 mm
 Displacement end: 0.74 mm

Young's Modulus

From linear elastic study User-defined
 Young's modulus: 72000 MPa

Plasticity

Plasticity model: Large plastic strains
 Isotropic hardening model: Ludwik
 Equation: $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$

Plasticity Parameters

Optimize σ_{ys0}

Initial value (MPa)	Scale (MPa)	Lower bound (MPa)	Upper bound (MPa)
387	387	300	450

Optimize k

Initial value (MPa)	Scale (MPa)	Lower bound (MPa)	Upper bound (MPa)
400	400	300	500

Optimize n

Initial value (MPa)	Scale (MPa)	Lower bound (MPa)	Upper bound (MPa)
0.1	0.1	0.05	0.25

Geometry x Hardening Function x Force-Displacement x Mesh x

Force (N) and Displacement (mm)

Studsвик

Force (N)

Displacement (mm)

Plasticity
Compensated measurements

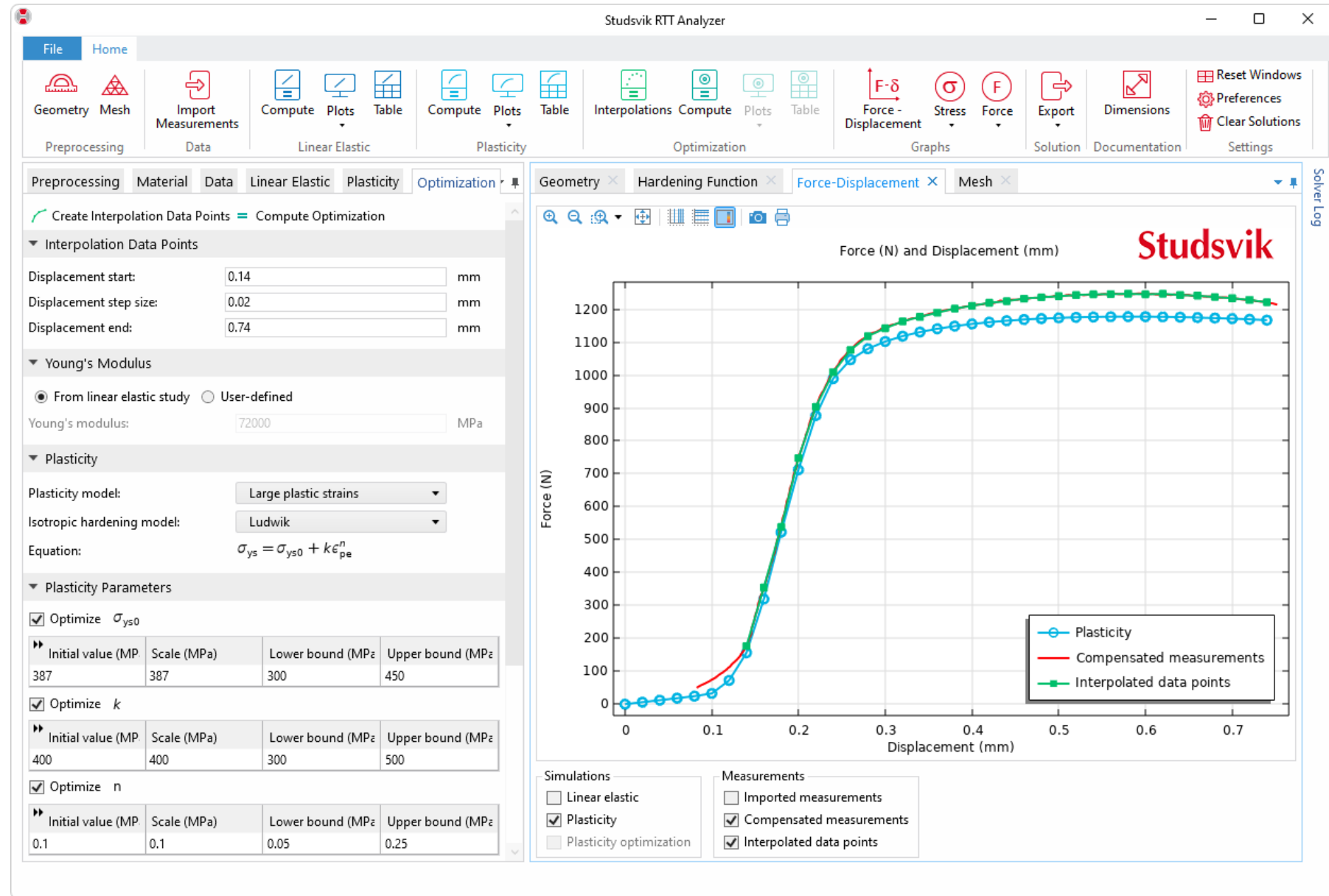
Simulations

Linear elastic
 Plasticity
 Plasticity optimization

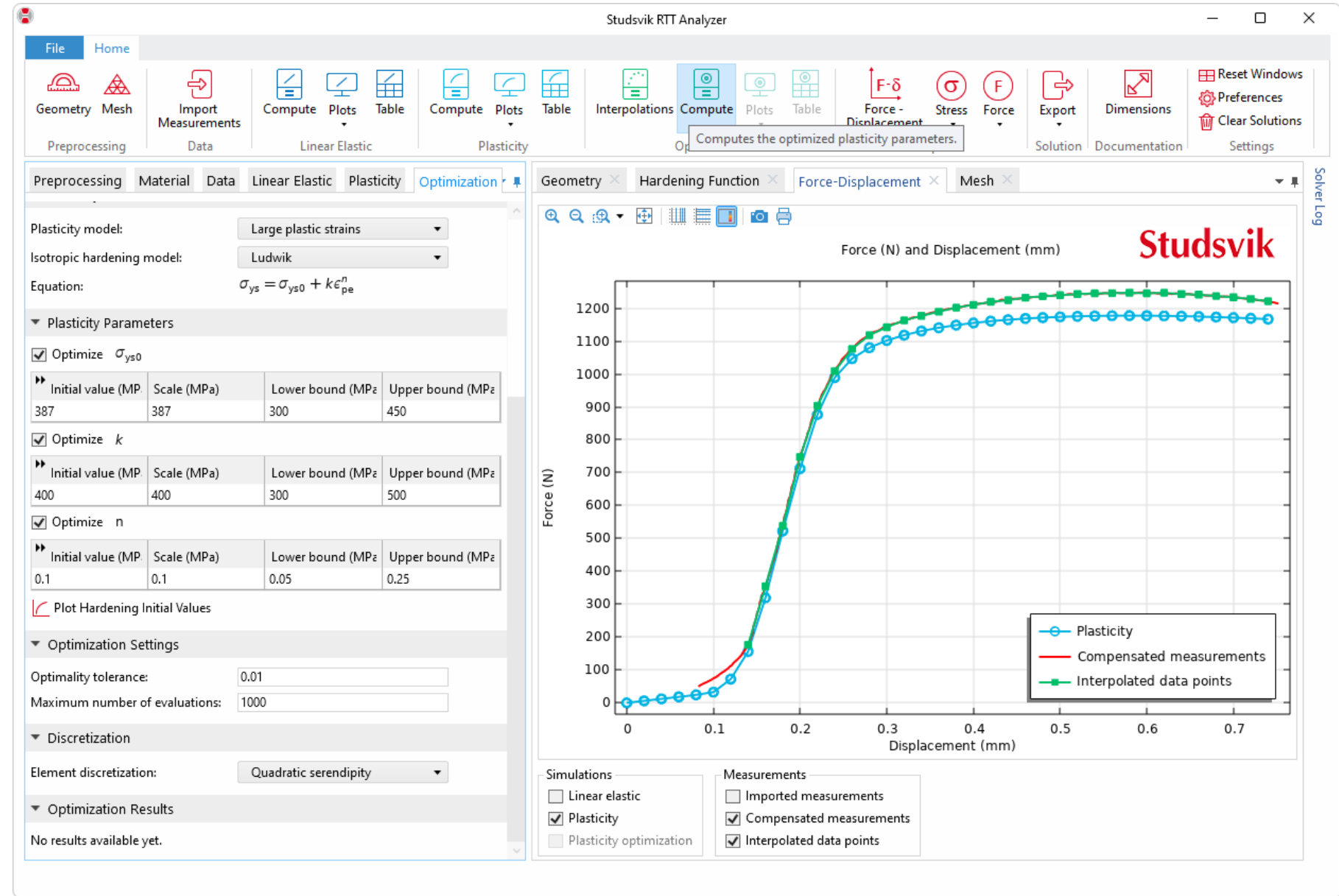
Measurements

Imported measurements
 Compensated measurements
 Interpolated data points

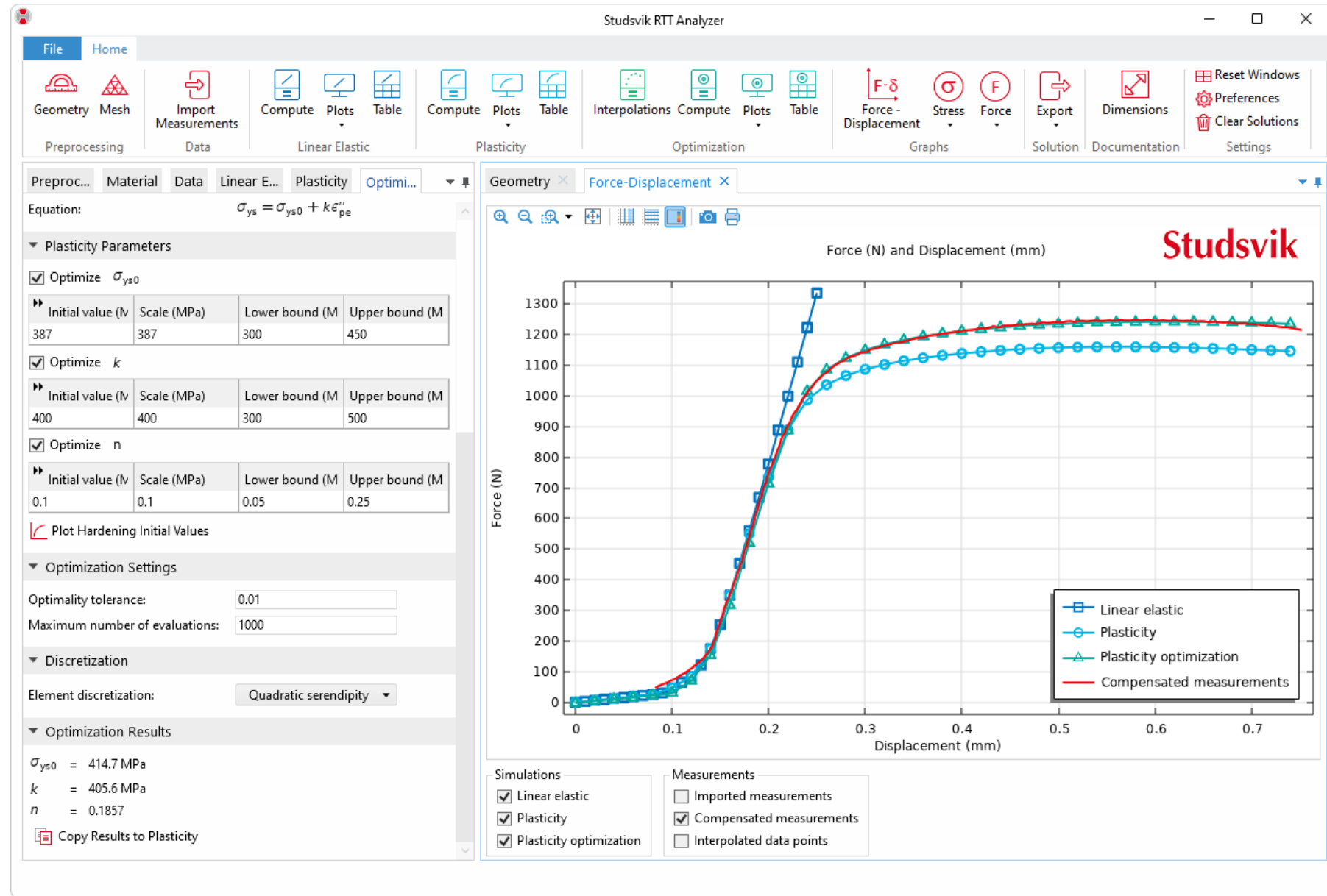
- The interpolation data points (green) follow the compensated measurements with the step size set in the *Interpolation Data* section
- Note that the first point is located at a displacement of 0.14 mm since this is the first to fit the measurements and the linear elastic simulation



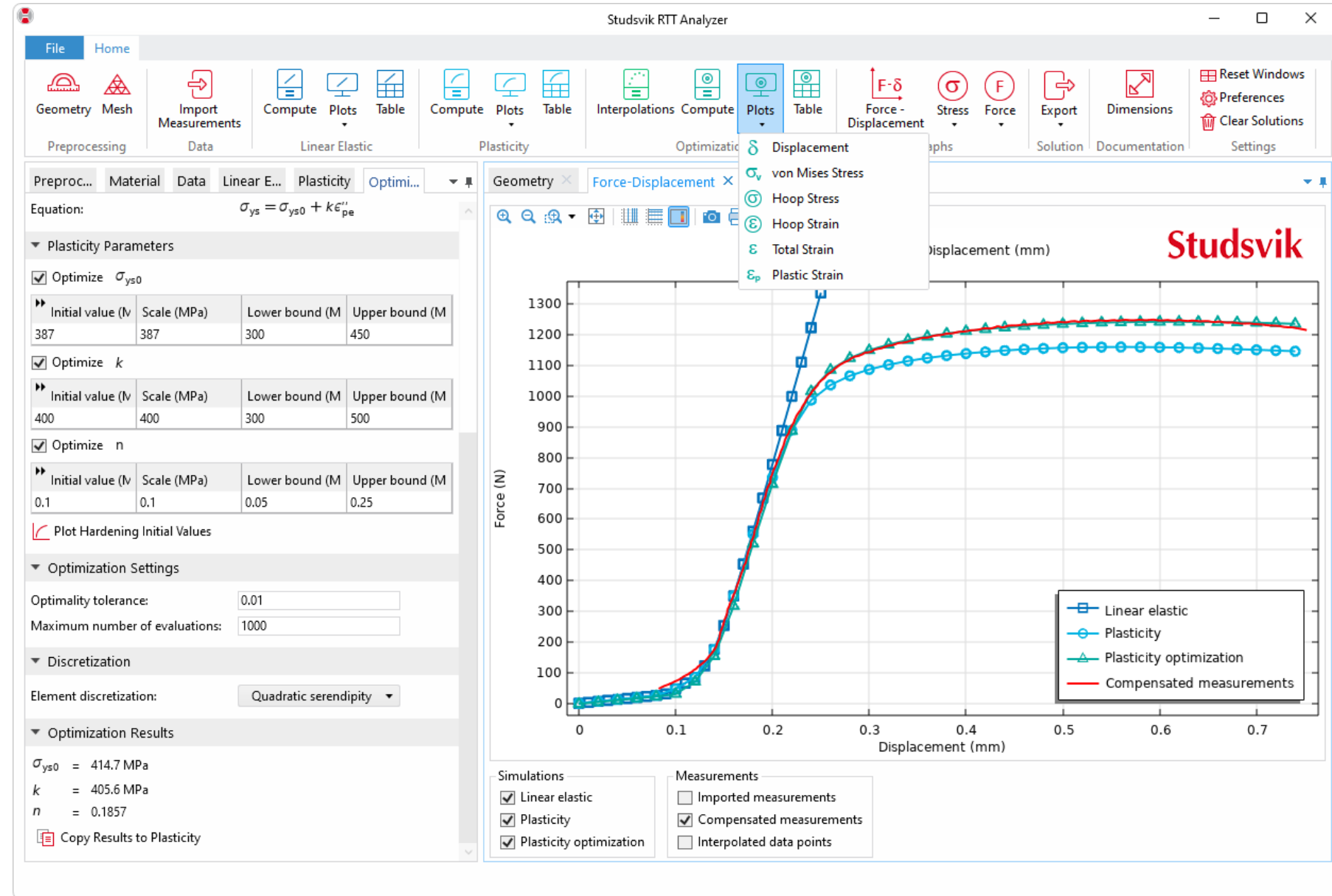
- In the *Plasticity Parameters* section, check the hardening parameters that should be optimized
- Provide initial values, scales, and lower and upper bounds
- In the *Optimization Settings* section, a tolerance and the maximum number of evaluations can be set
- Press the *Compute* button in the *Optimization* section of the ribbon



- The *Plasticity optimization* curve follows the measurement
- The calculated hardening parameters are presented in the *Optimization Results* section
- The optimized results can be copied to the plasticity study for further analysis



- 3D solution plots are available from the *Plots* menu in the ribbon



- The initial displacement is zero

Studsvik RTT Analyzer

File Home

Geometry Mesh Import Measurements Compute Plots Table Compute Plots Table Interpolations Compute Plots Table Force-Displacement Stress Force Export Dimensions Reset Windows Preferences Clear Solutions

Preprocessing Data Linear Elastic Plasticity Optimization Graphs Solution Documentation Settings

Preproc... Material Data Linear E... Plasticity Optim...

Equation: $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$

Plasticity Parameters

Optimize σ_{ys0}

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
387	387	300	450

Optimize k

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
400	400	300	500

Optimize n

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
0.1	0.1	0.05	0.25

Plot Hardening Initial Values

Optimization Settings

Optimality tolerance: 0.01

Maximum number of evaluations: 1000

Discretization

Element discretization: Quadratic serendipity

Optimization Results

$\sigma_{ys0} = 414.7$ MPa

$k = 405.6$ MPa

$n = 0.1857$

Copy Results to Plasticity

Geometry x Displacement - Optimization x Force-Displacement x

disp(1)=0 Displacement (mm)

Studsvik

Displacement

0

- Use the combo-box below the graphics to change the solution step

The screenshot shows the Studsvik RTT Analyzer software interface. The main window is titled "Studsvik RTT Analyzer" and features a ribbon menu with tabs for File, Home, Preprocessing, Data, Linear Elastic, Plasticity, Optimization, Graphs, Solution, Documentation, and Settings. The "Optimization" tab is active, displaying the equation $\sigma_{ys} = \sigma_{ys0} + k\epsilon''_{pe}$ and various optimization parameters.

Optimization Parameters:

- Optimize σ_{ys0}

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
387	387	300	450
- Optimize k

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
400	400	300	500
- Optimize n

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
0.1	0.1	0.05	0.25

Optimization Settings:

- Optimality tolerance: 0.01
- Maximum number of evaluations: 1000

Discretization: Quadratic serendipity

Optimization Results:

- $\sigma_{ys0} = 414.7$ MPa
- $k = 405.6$ MPa
- $n = 0.1857$

3D Plot: The main window displays a 3D model of a mechanical part with a color scale for displacement (mm) ranging from 0 to 0.25. The plot is titled "Displacement (mm)" and shows a value of $disp(33)=5E-4$. A color bar on the right indicates the displacement scale. Below the plot, a "Displacement" control panel shows a dropdown menu set to "5E-4".

Change the solution step

- Visualize the last step

Studsvik RTT Analyzer

File Home

Geometry Mesh Import Measurements Compute Plots Table Compute Plots Table Interpolations Compute Plots Table Force-Displacement Stress Force Export Dimensions Reset Windows Preferences Clear Solutions

Preprocessing Data Linear Elastic Plasticity Optimization Graphs Solution Documentation Settings

Preproc... Material Data Linear E... Plasticity Optimi...

Equation: $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$

Plasticity Parameters

Optimize σ_{ys0}

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
387	387	300	450

Optimize k

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
400	400	300	500

Optimize n

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
0.1	0.1	0.05	0.25

Plot Hardening Initial Values

Optimization Settings

Optimality tolerance: 0.01

Maximum number of evaluations: 1000

Discretization

Element discretization: Quadratic serendipity

Optimization Results

$\sigma_{ys0} = 414.7$ MPa
 $k = 405.6$ MPa
 $n = 0.1857$

Copy Results to Plasticity

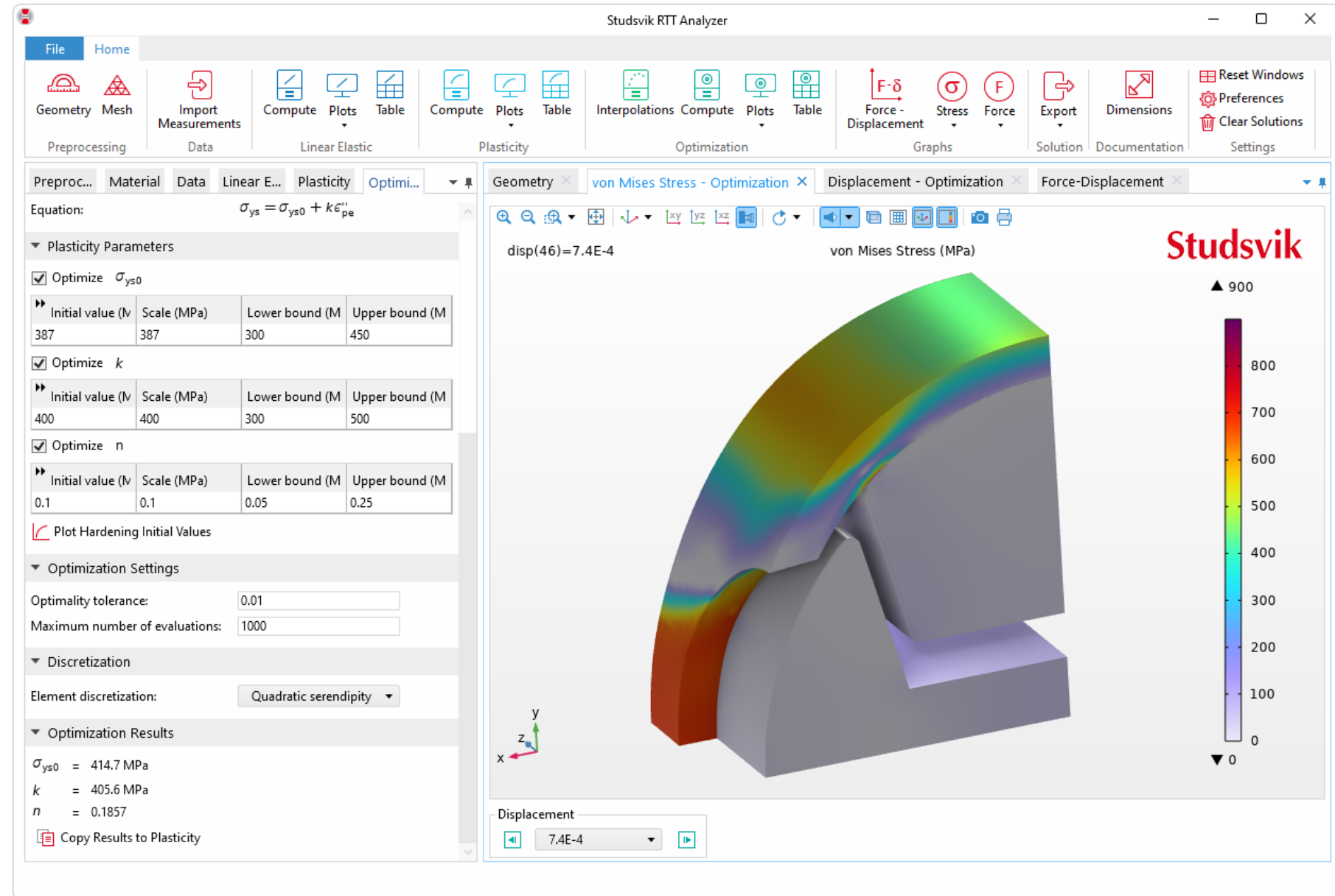
Geometry x Displacement - Optimization x Force-Displacement x

disp(46)=7.4E-4 Displacement (mm)

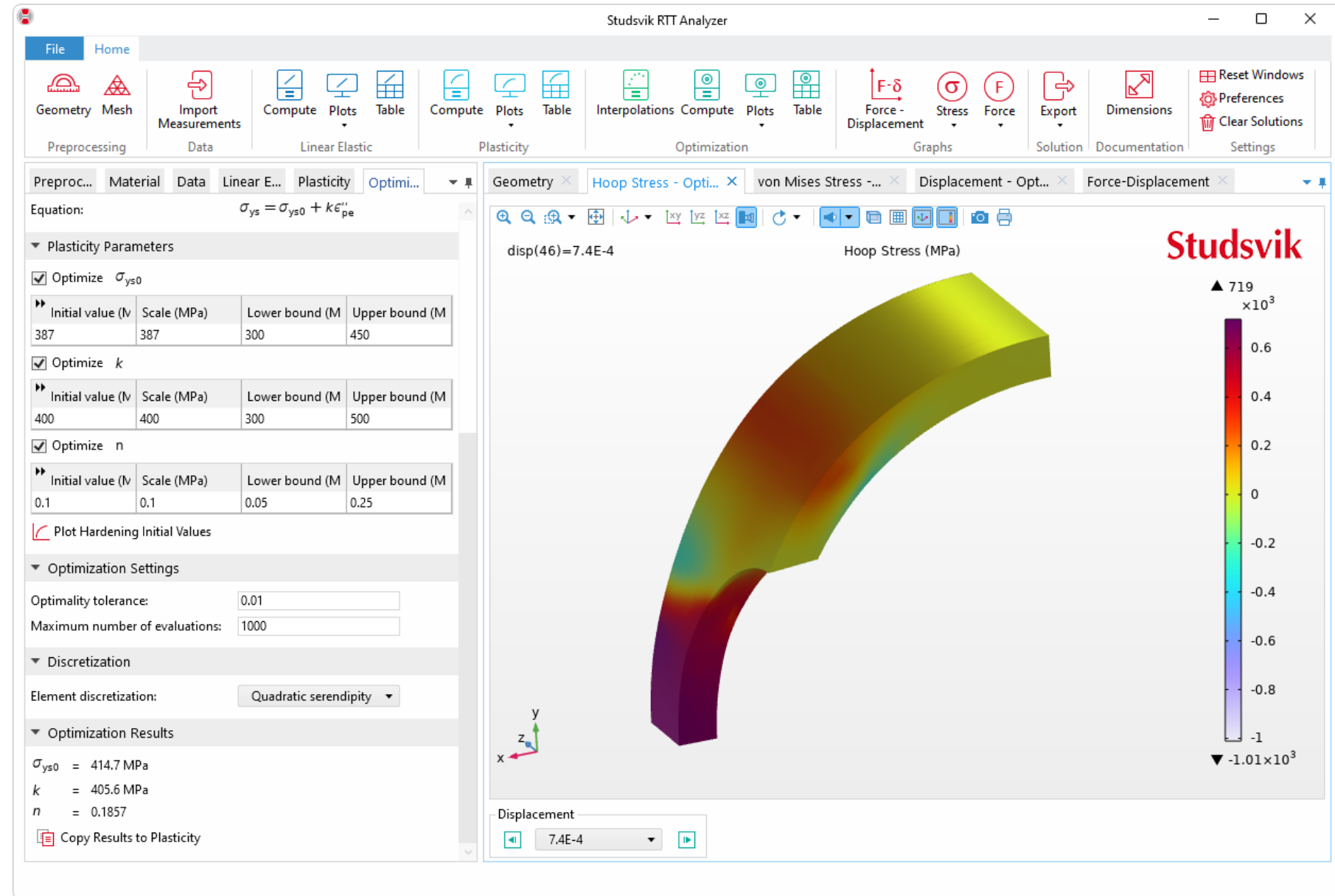
Studsvik

Displacement: 7.4E-4

- The *von Mises Stress* at the last step



- The *Hoop Stress* at the last step



- The *Hoop Strain* at the last step

Studsvik RTT Analyzer

File Home

Geometry Mesh Import Measurements Compute Plots Table Compute Plots Table Interpolations Compute Plots Table Force - Displacement Stress Force Export Dimensions Reset Windows Preferences Clear Solutions

Preprocessing Data Linear Elastic Plasticity Optimization Graphs Solution Documentation Settings

Preproc... Material Data Linear E... Plasticity Optimi...

Equation: $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$

Plasticity Parameters

Optimize σ_{ys0}

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
387	387	300	450

Optimize k

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
400	400	300	500

Optimize n

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
0.1	0.1	0.05	0.25

Plot Hardening Initial Values

Optimization Settings

Optimality tolerance: 0.01

Maximum number of evaluations: 1000

Discretization

Element discretization: Quadratic serendipity

Optimization Results

$\sigma_{ys0} = 414.7$ MPa
 $k = 405.6$ MPa
 $n = 0.1857$

Copy Results to Plasticity

Geometry x Hoop Strain -... x Hoop Stress -... x von Mises Stre... x Displacement -... x Force-Displace... x

disp(46)=7.4E-4

Hoop Strain (1)

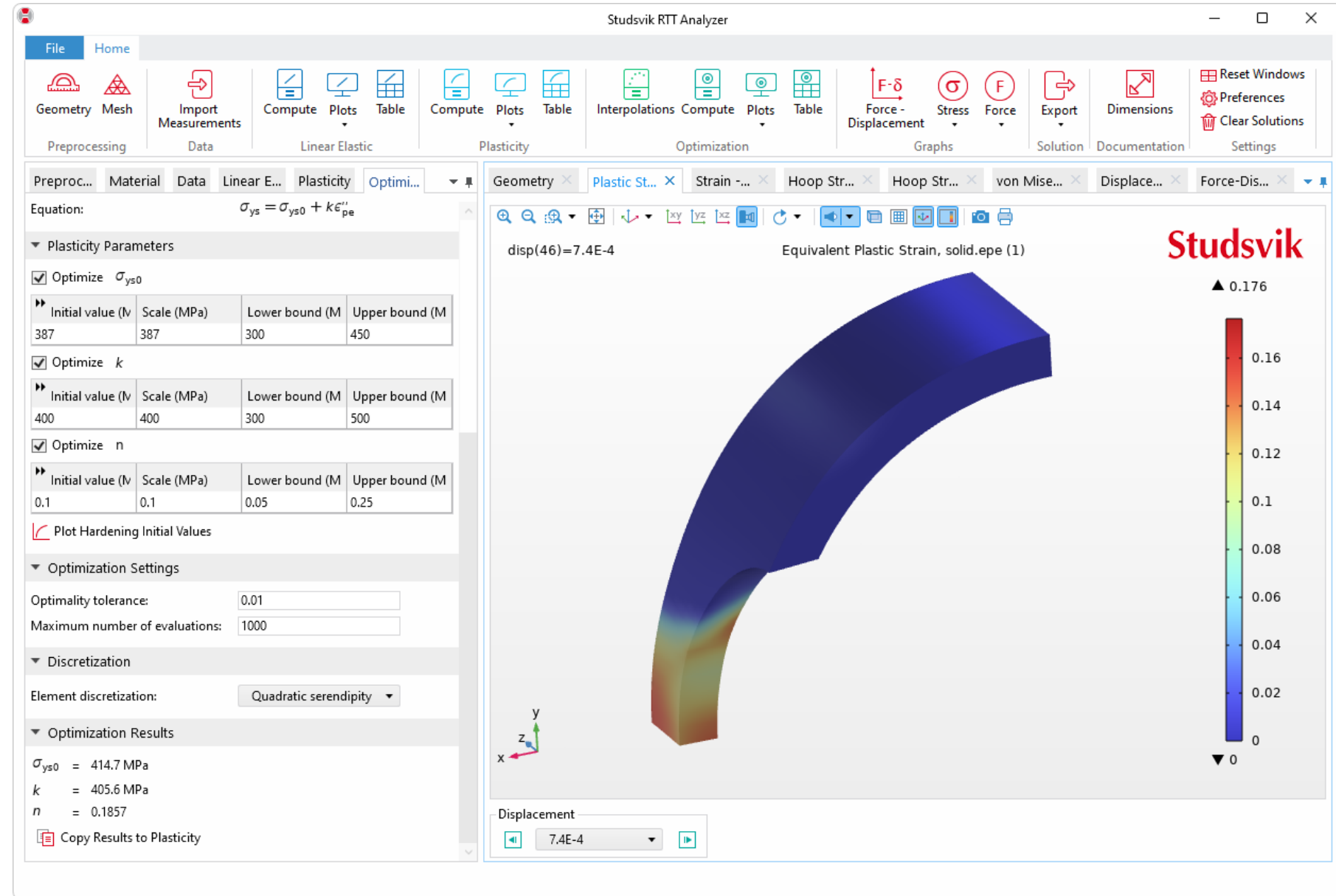
▲ 0.223

0.2
0.15
0.1
0.05
0
▼ -0.0148

Displacement

7.4E-4

- The *Plastic Strain* at the last step



- Numerical values are available by clicking the *Table* button in the ribbon

Studsvik RTT Analyzer

File Home

Geometry Mesh Import Measurements Compute Plots Table Compute Plots Table Interpolations Compute Plots Table Force-δ Stress Force Export Dimensions Reset Windows Preferences Clear Solutions Settings

Preprocessing Data Linear Elastic Plasticity Optimization

Preproc... Material Data Linear E... Plasticity Optimi...

Equation: $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$

Plasticity Parameters

Optimize σ_{ys0}

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
387	387	300	450

Optimize k

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
400	400	300	500

Optimize n

Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
0.1	0.1	0.05	0.25

Plot Hardening Initial Values

Optimization Settings

Optimality tolerance: 0.01

Maximum number of evaluations: 1000

Discretization

Element discretization: Quadratic serendipity

Optimization Results

$\sigma_{ys0} = 414.7$ MPa

$k = 405.6$ MPa

$n = 0.1857$

Copy Results to Plasticity

Geometry x Plastic St... x Strain -... x Hoop Str... x Hoop Str... x von Mise... x Displace... x Force-Dis... x

disp(46)=7.4E-4

Equivalent Plastic Strain, solid.epε (1)

Studsvik

▲ 0.176

0.16

0.14

0.12

0.1

0.08

0.06

0.04

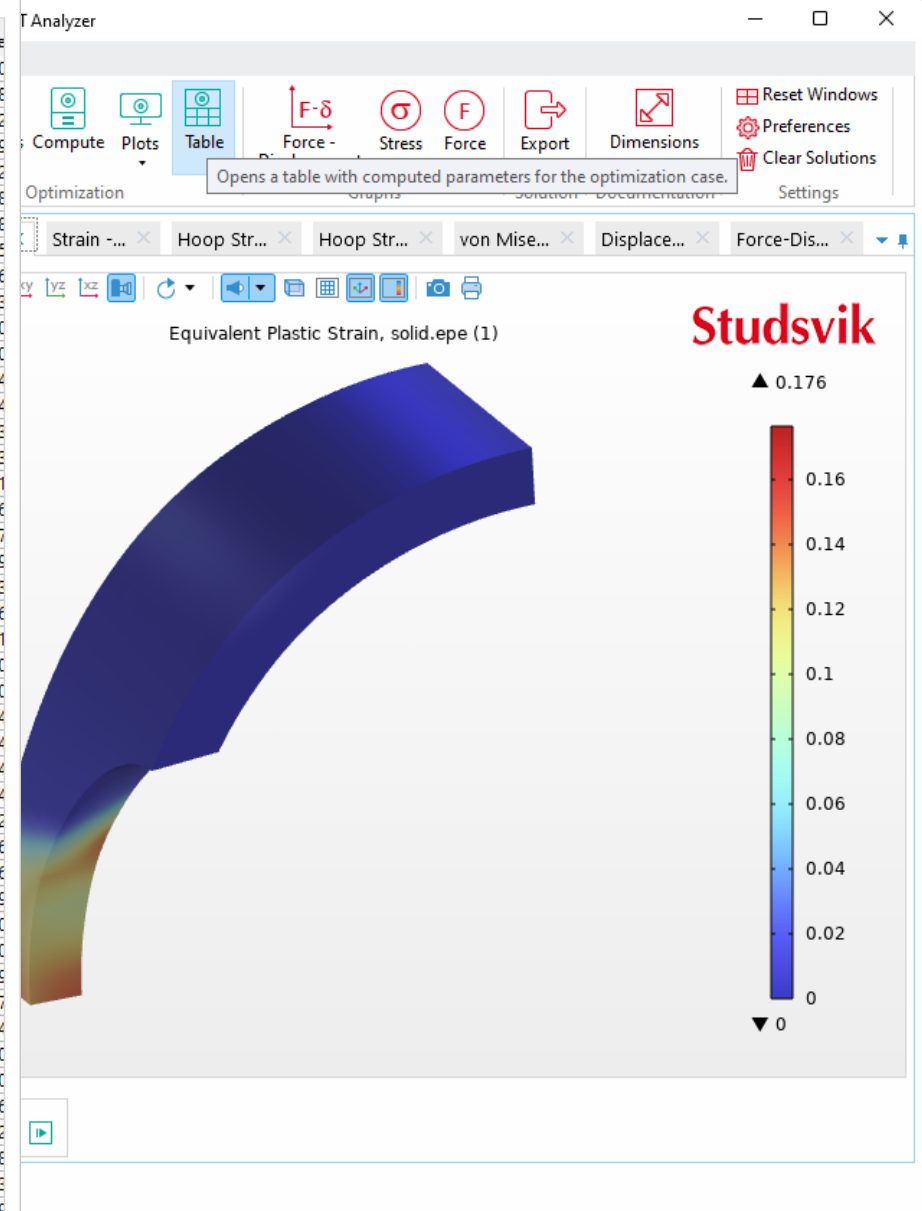
0

▼ 0

Displacement

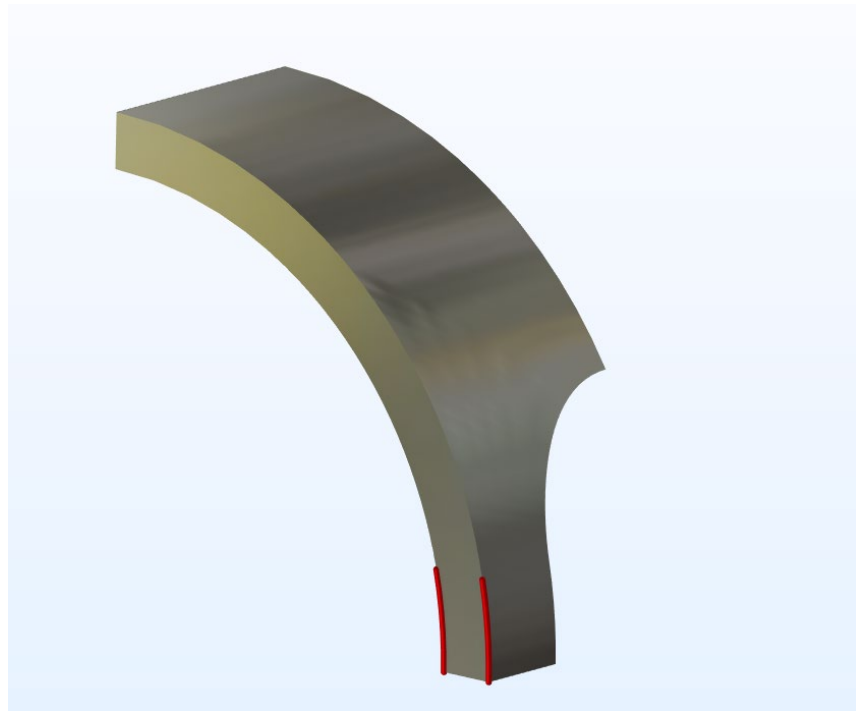
7.4E-4

disp	disp (mm)	Initial yield stress, Ludwik hardening (MPa)	k, Ludwik hardening (MPa)	n, Ludwik hardening	Force (N)	Engineering stress (MPa)	Engineering stress without friction (MPa)	True
0.0000	0.0000	414.68	405.63	0.18567	0.0000	0.0000	0.0000	0.000
2.0000E-5	0.020000	414.68	405.63	0.18567	5.9617	2.6883	3.0724	2.688
4.0000E-5	0.040000	414.68	405.63	0.18567	11.981	5.4025	6.1743	5.402
6.0000E-5	0.060000	414.68	405.63	0.18567	18.049	8.1389	9.3016	8.139
8.0000E-5	0.080000	414.68	405.63	0.18567	24.216	10.920	12.480	10.92
1.0000E-4	0.10000	414.68	405.63	0.18567	32.342	14.584	16.668	14.58
1.2000E-4	0.12000	414.68	405.63	0.18567	71.998	32.467	37.105	32.46
1.4000E-4	0.14000	414.68	405.63	0.18567	155.66	70.195	80.222	70.22
1.6000E-4	0.16000	414.68	405.63	0.18567	318.11	143.45	163.94	143.6
1.8000E-4	0.18000	414.68	405.63	0.18567	520.73	234.82	268.36	235.3
2.0000E-4	0.20000	414.68	405.63	0.18567	714.14	322.03	368.04	323.0
2.0000E-4	0.20000	414.68	405.63	0.18567	714.14	322.03	368.04	323.0
2.2000E-4	0.22000	414.68	405.63	0.18567	886.53	399.77	456.88	401.4
2.2000E-4	0.22000	414.68	405.63	0.18567	886.53	399.77	456.88	401.4
2.4000E-4	0.24000	414.68	405.63	0.18567	1015.5	457.94	523.36	460.3
2.4000E-4	0.24000	414.68	405.63	0.18567	1015.5	457.94	523.36	460.3
2.6000E-4	0.26000	414.68	405.63	0.18567	1084.8	489.16	559.04	493.1
2.8000E-4	0.28000	414.68	405.63	0.18567	1123.1	506.46	578.82	512.6
3.0000E-4	0.30000	414.68	405.63	0.18567	1148.6	517.94	591.93	526.7
3.2000E-4	0.32000	414.68	405.63	0.18567	1167.2	526.35	601.54	537.9
3.4000E-4	0.34000	414.68	405.63	0.18567	1181.7	532.87	608.99	547.3
3.6000E-4	0.36000	414.68	405.63	0.18567	1193.3	538.09	614.96	555.6
3.8000E-4	0.38000	414.68	405.63	0.18567	1202.8	542.37	619.85	563.1
4.0000E-4	0.40000	414.68	405.63	0.18567	1210.6	545.91	623.90	570.0
4.0000E-4	0.40000	414.68	405.63	0.18567	1210.6	545.91	623.90	570.0
4.2000E-4	0.42000	414.68	405.63	0.18567	1217.1	548.86	627.27	576.4
4.2000E-4	0.42000	414.68	405.63	0.18567	1217.1	548.86	627.27	576.4
4.4000E-4	0.44000	414.68	405.63	0.18567	1222.6	551.32	630.08	582.4
4.4000E-4	0.44000	414.68	405.63	0.18567	1222.6	551.32	630.08	582.4
4.6000E-4	0.46000	414.68	405.63	0.18567	1227.1	553.36	632.41	588.2
4.8000E-4	0.48000	414.68	405.63	0.18567	1230.9	555.05	634.34	593.6
4.8000E-4	0.48000	414.68	405.63	0.18567	1230.9	555.05	634.34	593.6
5.0000E-4	0.50000	414.68	405.63	0.18567	1233.9	556.42	635.91	598.9
5.2000E-4	0.52000	414.68	405.63	0.18567	1236.4	557.52	637.17	604.0
5.4000E-4	0.54000	414.68	405.63	0.18567	1238.2	558.37	638.14	609.0
5.6000E-4	0.56000	414.68	405.63	0.18567	1239.6	558.99	638.84	613.9
5.8000E-4	0.58000	414.68	405.63	0.18567	1240.5	559.40	639.31	618.7
6.0000E-4	0.60000	414.68	405.63	0.18567	1241.0	559.62	639.57	623.4
6.2000E-4	0.62000	414.68	405.63	0.18567	1241.1	559.66	639.62	628.0
6.2000E-4	0.62000	414.68	405.63	0.18567	1241.1	559.66	639.62	628.0
6.4000E-4	0.64000	414.68	405.63	0.18567	1240.8	559.54	639.47	632.6
6.6000E-4	0.66000	414.68	405.63	0.18567	1240.2	559.25	639.14	637.2
6.8000E-4	0.68000	414.68	405.63	0.18567	1239.2	558.80	638.63	641.8
7.0000E-4	0.70000	414.68	405.63	0.18567	1237.9	558.21	637.95	646.3
7.2000E-4	0.72000	414.68	405.63	0.18567	1236.3	557.47	637.11	650.9
7.4000E-4	0.74000	414.68	405.63	0.18567	1234.3	556.60	636.12	655.5

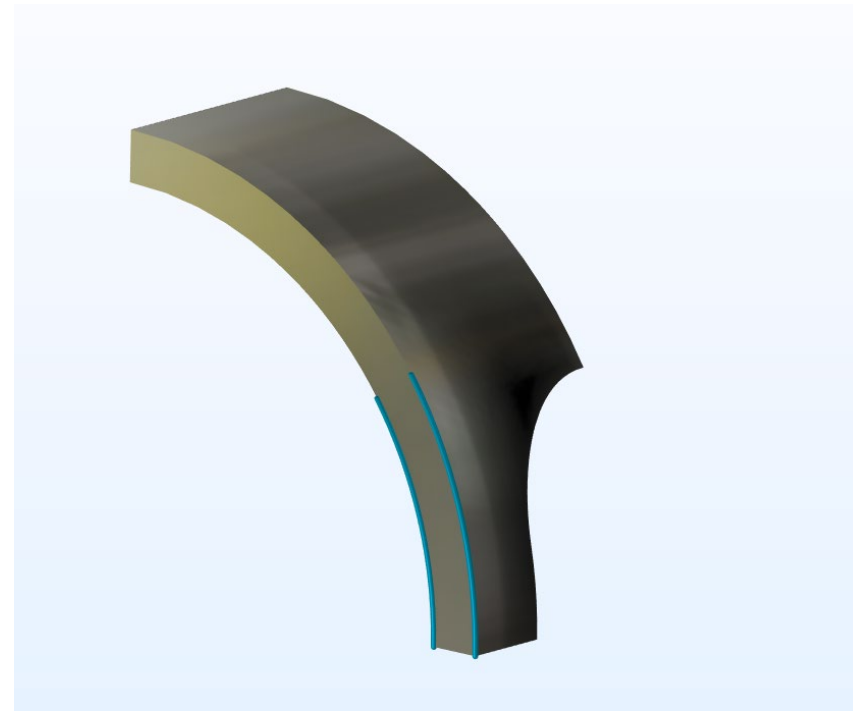


Engineering Strain in Gauges

- For historical reasons, the engineering strain is calculated in two gauges with different length



Gauge #1



Gauge #2

- To visualize the stress and strain graphs, click the *Stress* menu in the *Graphs* section in the ribbon

The screenshot displays the Studsvik RTT Analyzer software interface. The ribbon at the top includes sections for Preprocessing, Data, Linear Elastic, Plasticity, Optimization, and Graphs. The **Stress** menu in the Graphs section is highlighted, showing options for Engineering Strain in Gauge #1, Engineering Strain in Gauge #2, and Displacement.

The **Plasticity** section is active, showing the equation $\sigma_{ys} = \sigma_{ys0} + k\epsilon_{pe}^n$. The Plasticity Parameters are as follows:

Parameter	Optimize	Initial value (N)	Scale (MPa)	Lower bound (M)	Upper bound (M)
σ_{ys0}	<input checked="" type="checkbox"/>	387	387	300	450
k	<input checked="" type="checkbox"/>	400	400	300	500
n	<input checked="" type="checkbox"/>	0.1	0.1	0.05	0.25

Optimization Settings:

- Optimality tolerance: 0.01
- Maximum number of evaluations: 1000

Discretization:

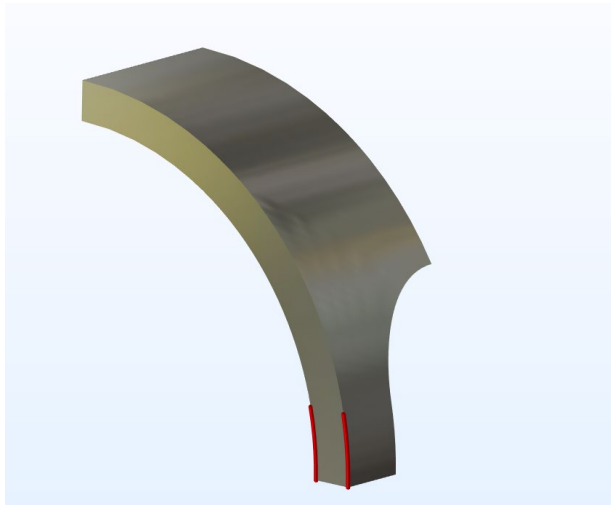
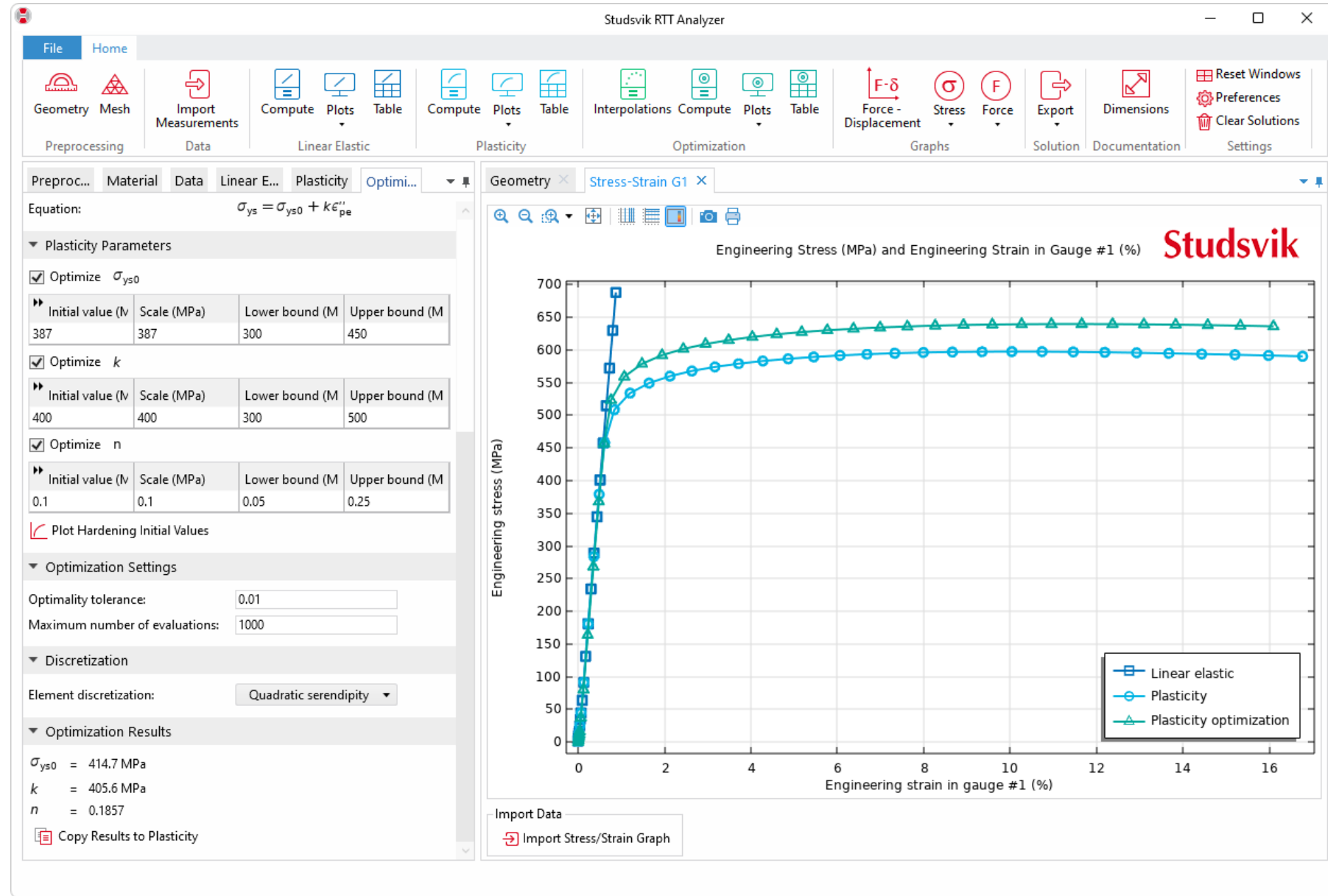
- Element discretization: Quadratic serendipity

Optimization Results:

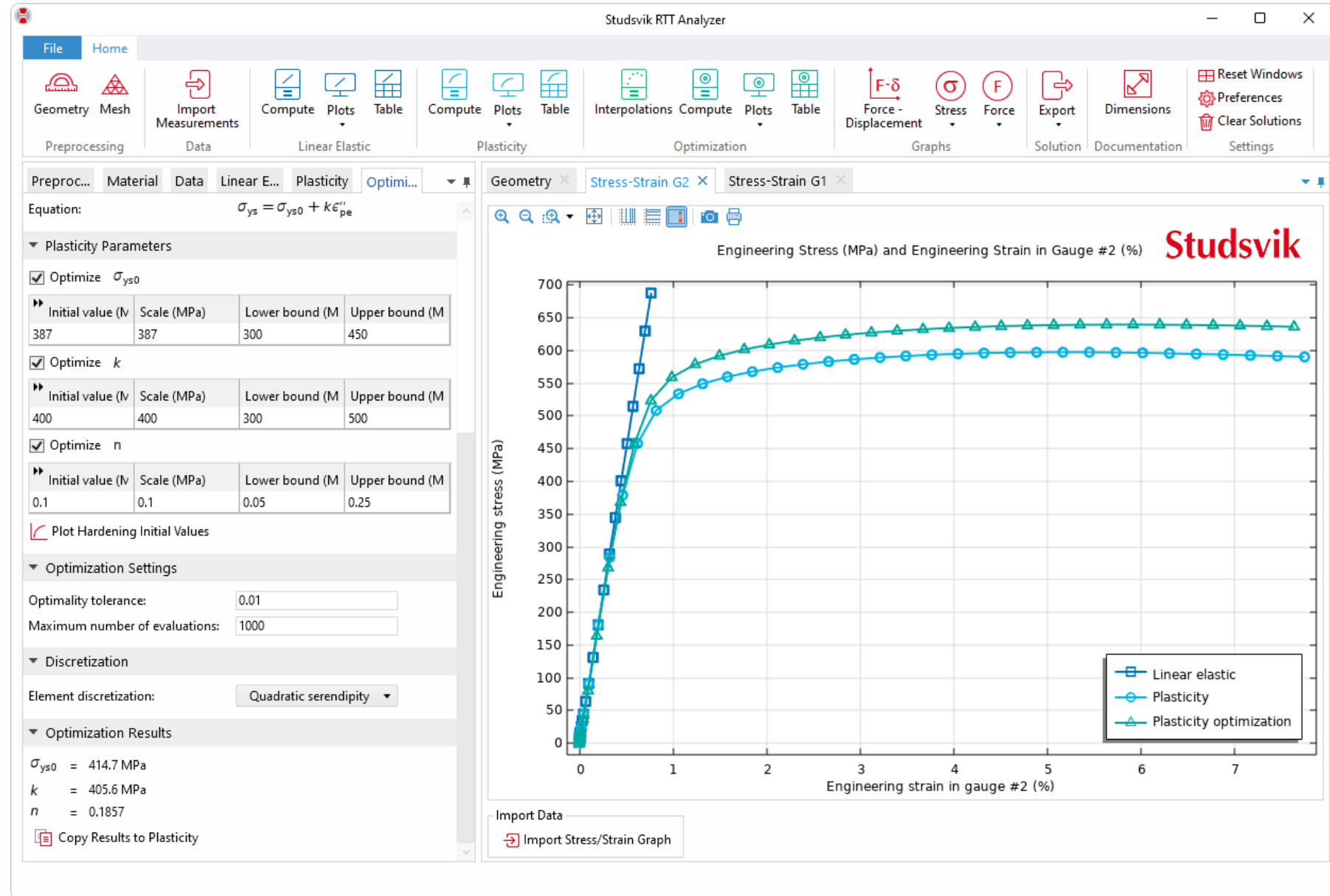
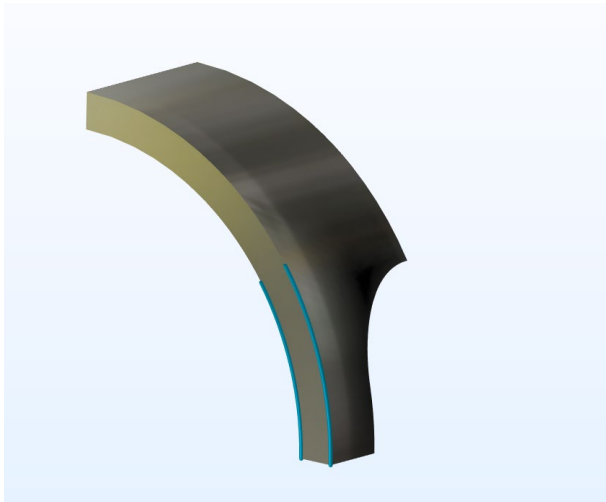
- $\sigma_{ys0} = 414.7$ MPa
- $k = 405.6$ MPa
- $n = 0.1857$

A 3D model of a mechanical part is shown in the main view, with a coordinate system (x, y, z) visible at the bottom left. The Studsvik logo is in the top right corner of the main view.

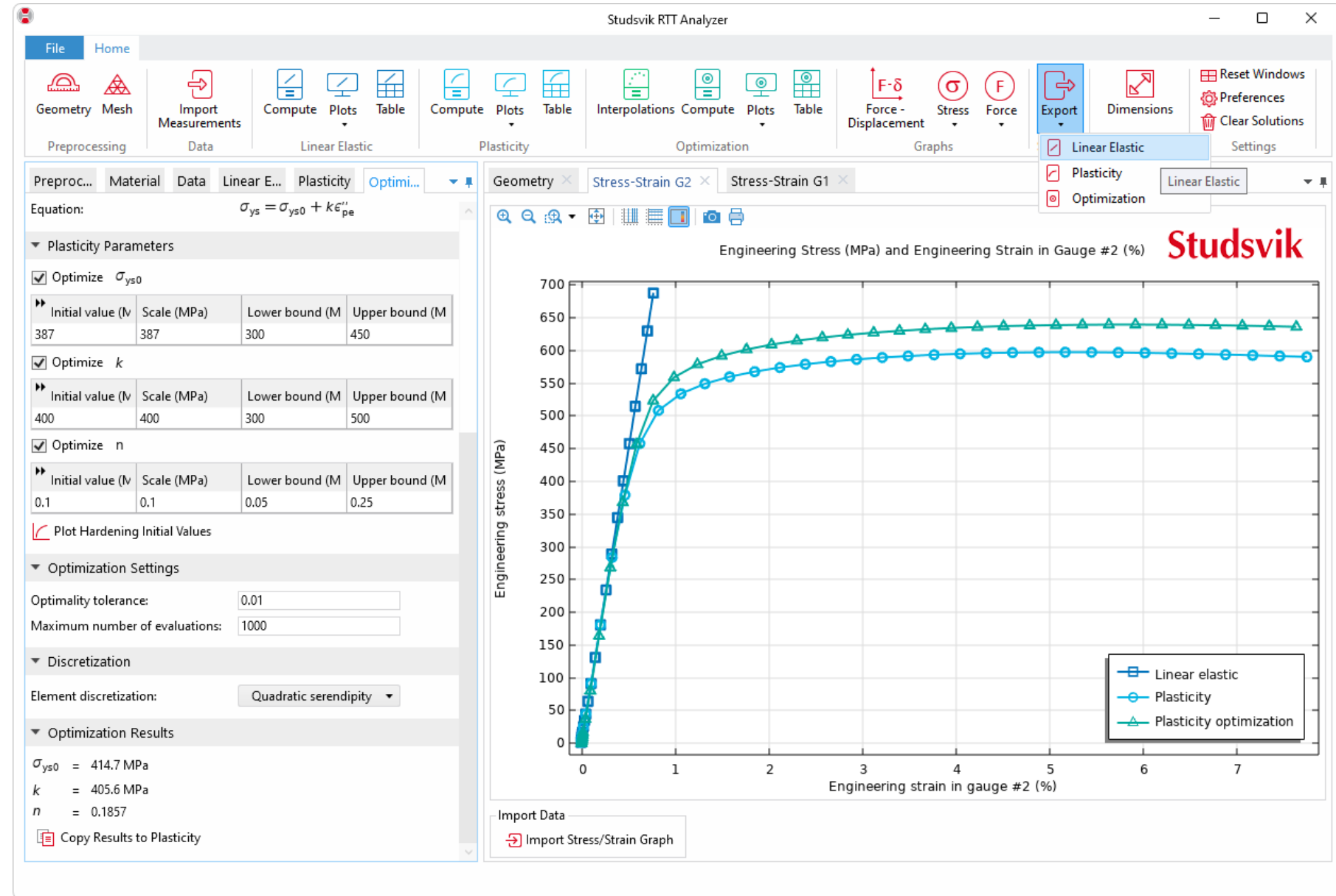
- The *Engineering Stress versus the Engineering Strain in Gauge #1* is visualized



- The *Engineering Stress* versus the *Engineering Strain in Gauge #2* is visualized



- Export results data directly to Excel[®] for further analysis



Results to Customers

- The data can now be extracted, and the customer will receive the requested material data
 - yield strength
 - ultimate strength
 - elongation

Implementation in COMSOL®

The used products to create the app



Used Products in the Model

- COMSOL Multiphysics[®] and the following add-on products
 - Structural Mechanics Module
 - Non-linear Structural Materials Module or Geomechanics Module
 - Optimization Module
 - Design Module or CAD Import Module

Used Products to Create the Application

- The app was built with Application Builder
- By utilizing Java[®] programming code, many model setup tasks are simplified, demonstrating how a complex simulation can be made accessible to a wider audience
- To enhance accessibility, the application is transformed into a standalone compiled app using COMSOL Compiler[™]

Conclusion

Ring tensile tests enter a new dimension
with new dimensions



Summary

- Ring tensile tests are employed to evaluate the mechanical properties of fuel cladding materials
- New technology allows for the import of measured data into a COMSOL[®] App to extract vital material information
- Simulated results are compared with the measured force-displacement diagram, and upon reaching satisfactory agreement, the stress-strain curve is obtained
- This new Finite Element Method-based approach enables testing with varying dimensions, offering the potential to reduce result uncertainties

References

1. S. Arsene, and J. Bai, A New Approach to Measuring Transverse Properties of Structural Tubing by a Ring Test, Journal of Testing and Evaluation, 1996.

deflexional.com

