Studsvik RTT Analyzer
A COMSOL® App to Evaluate Mechanical Properties for the Nuclear Industry

Daniel Ericsson[1], Joakim Karlsson[2], Martin König[2]

[1] Deflexional AB, Täby, Sweden
[2] Studsvik Nuclear AB, Nyköping, Sweden

deflexional

Contents

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

Introduction

Collaboration and background information

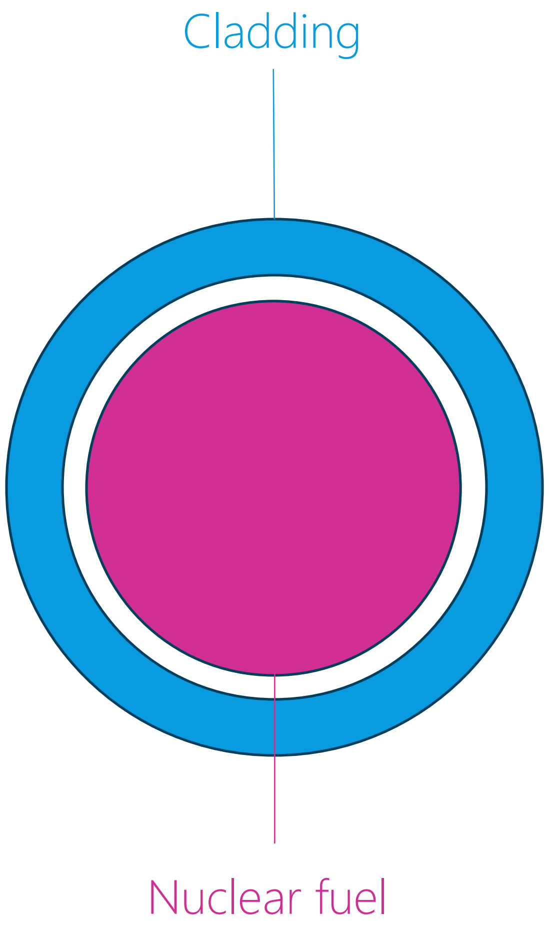
Collaboration

- 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®

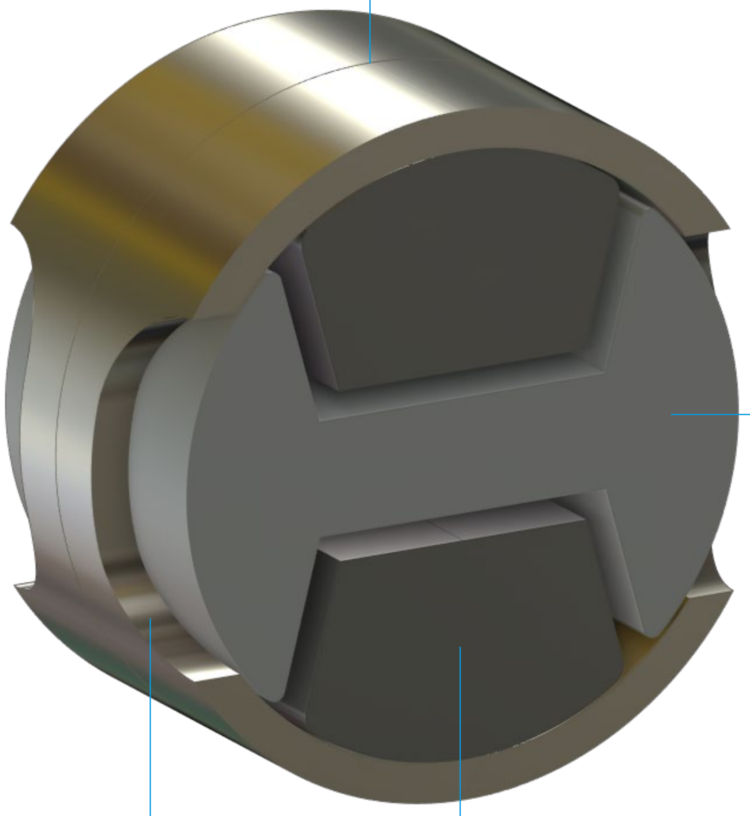
Background

- 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



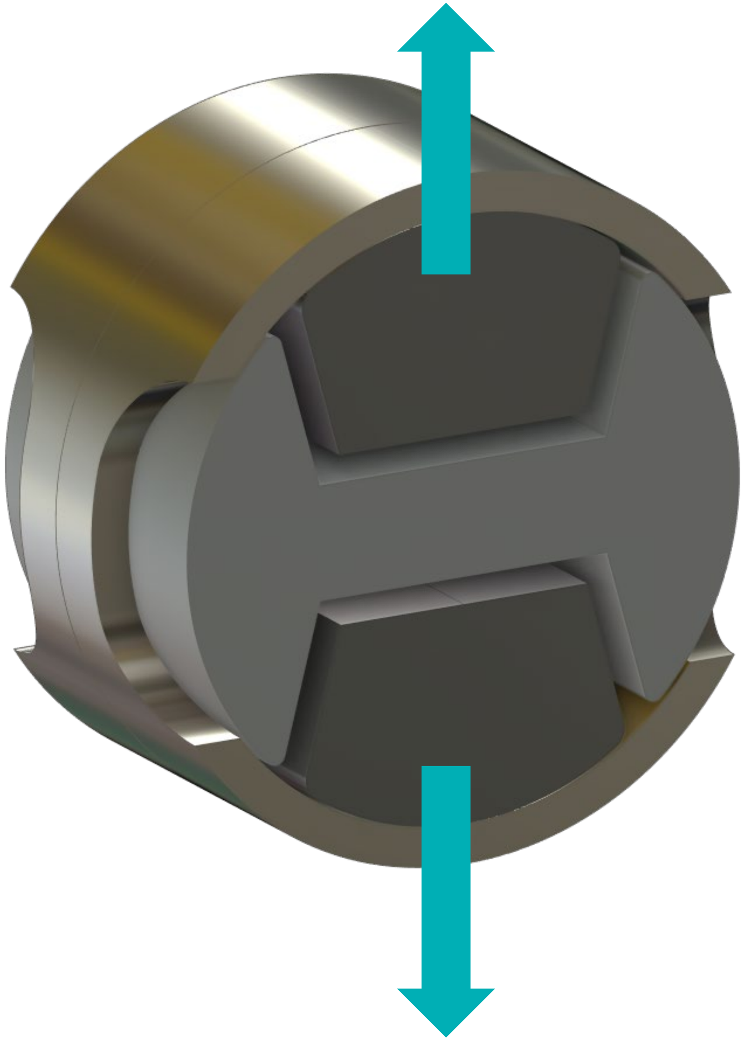
Ring Tensile Tests (RTT)

- 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

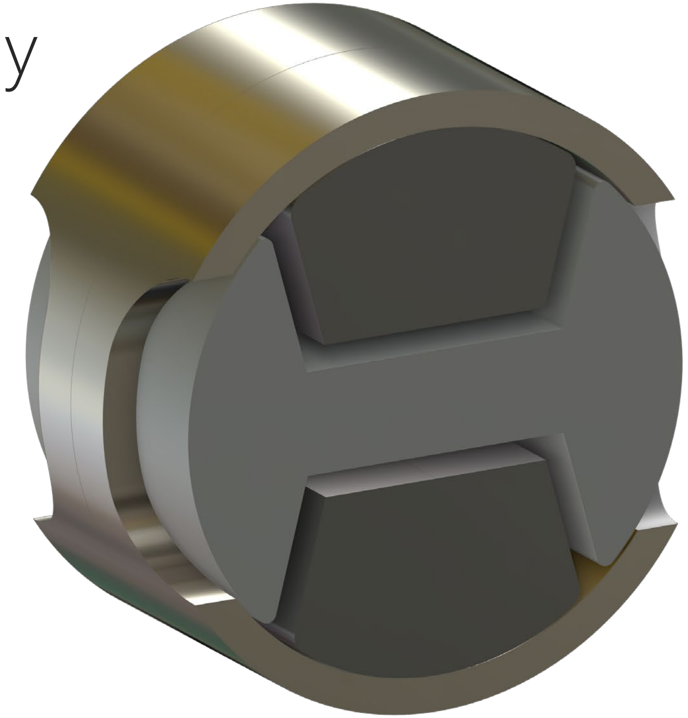
deflexional

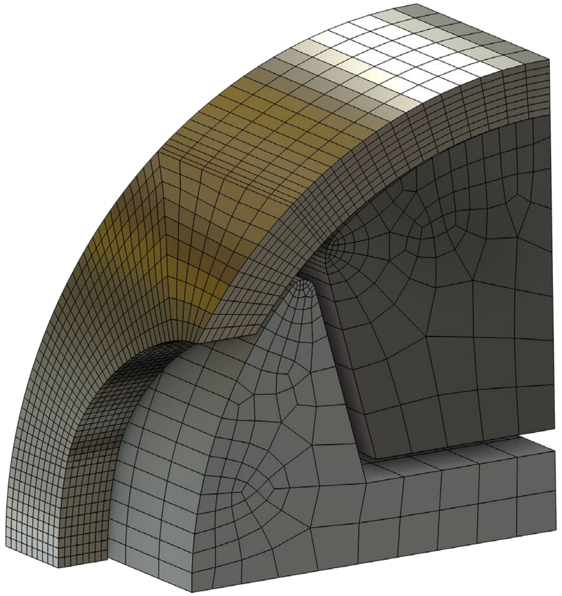
Workflow

- 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

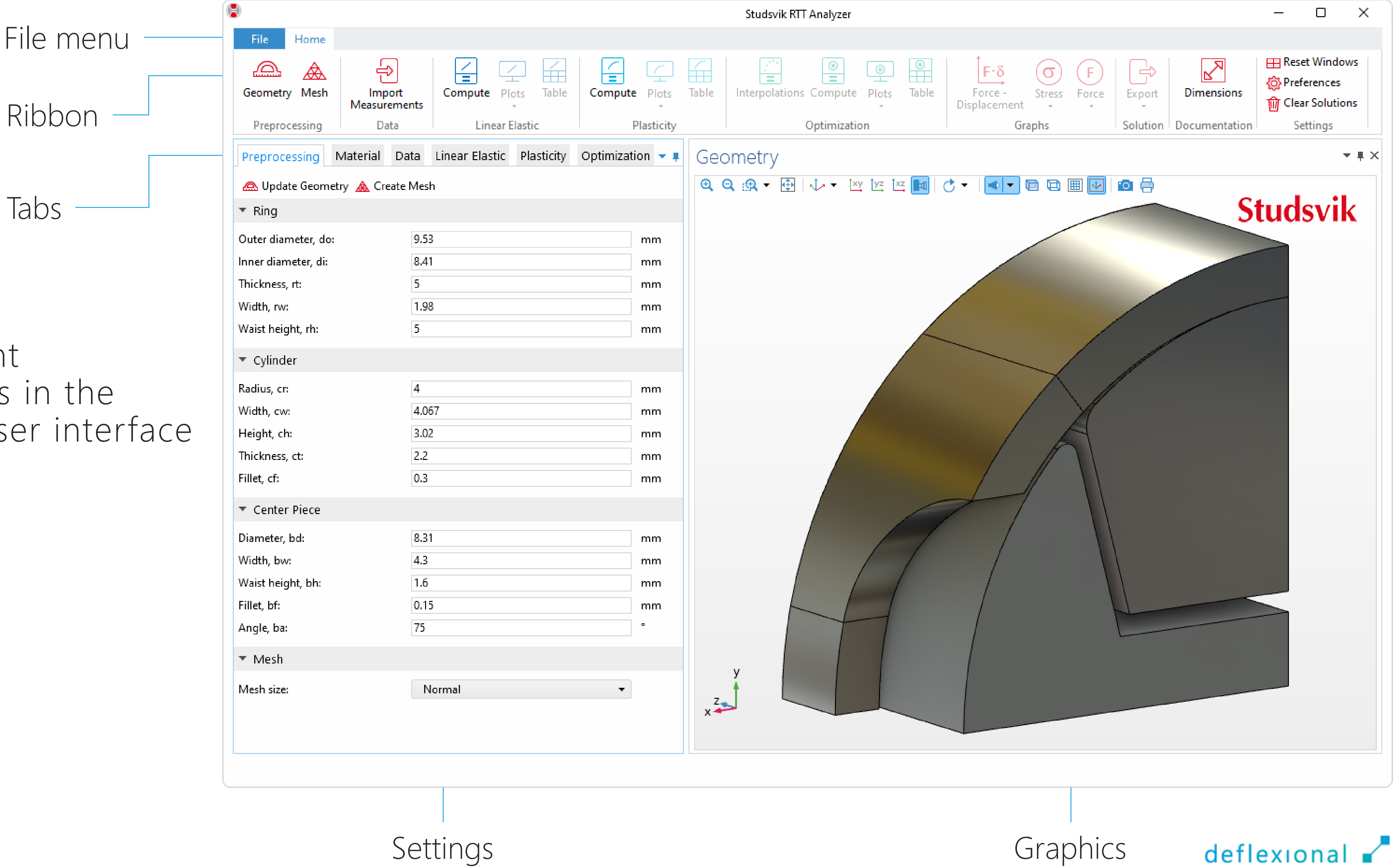
Modeling

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





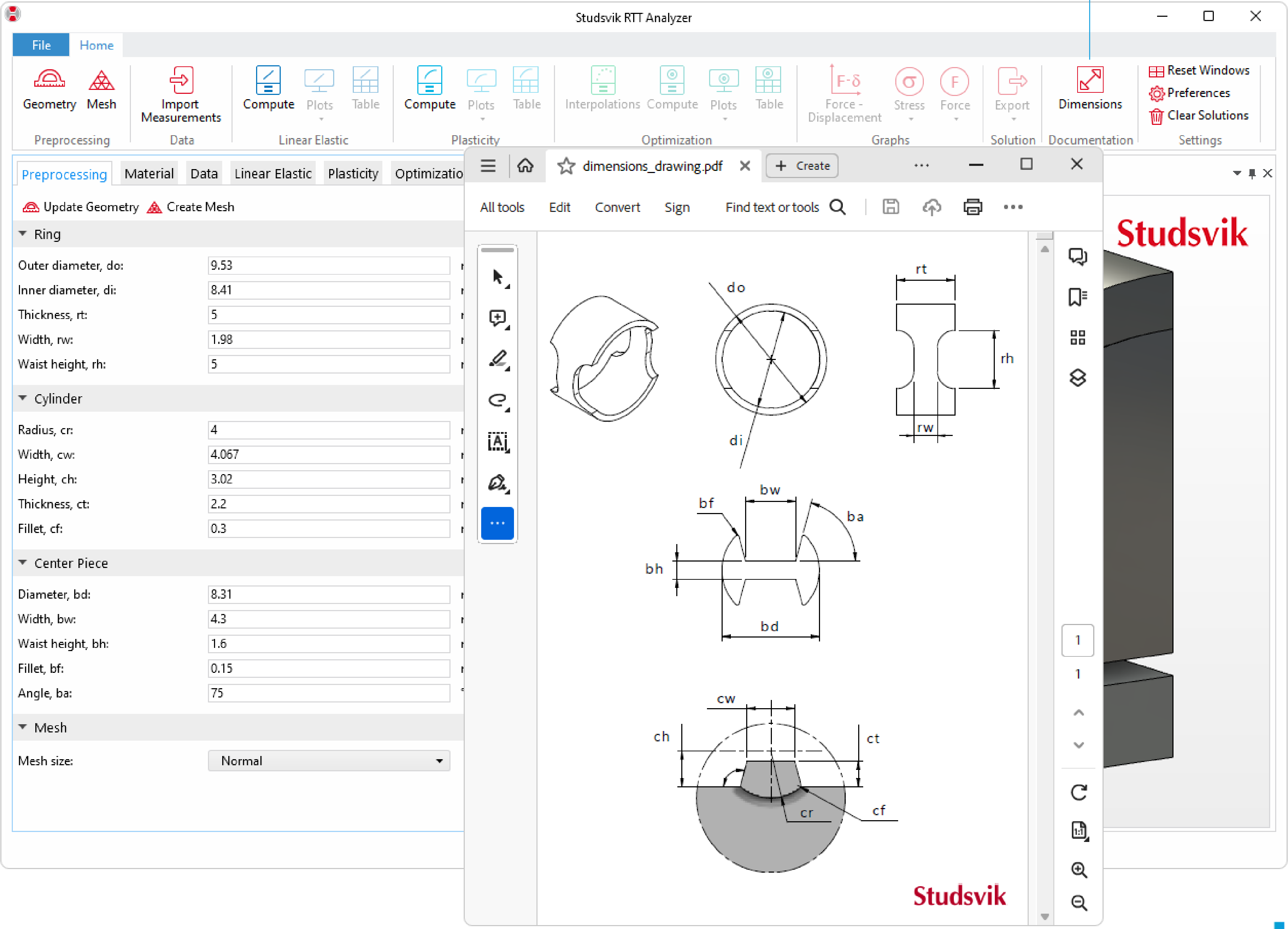
- The different components in the graphical user interface

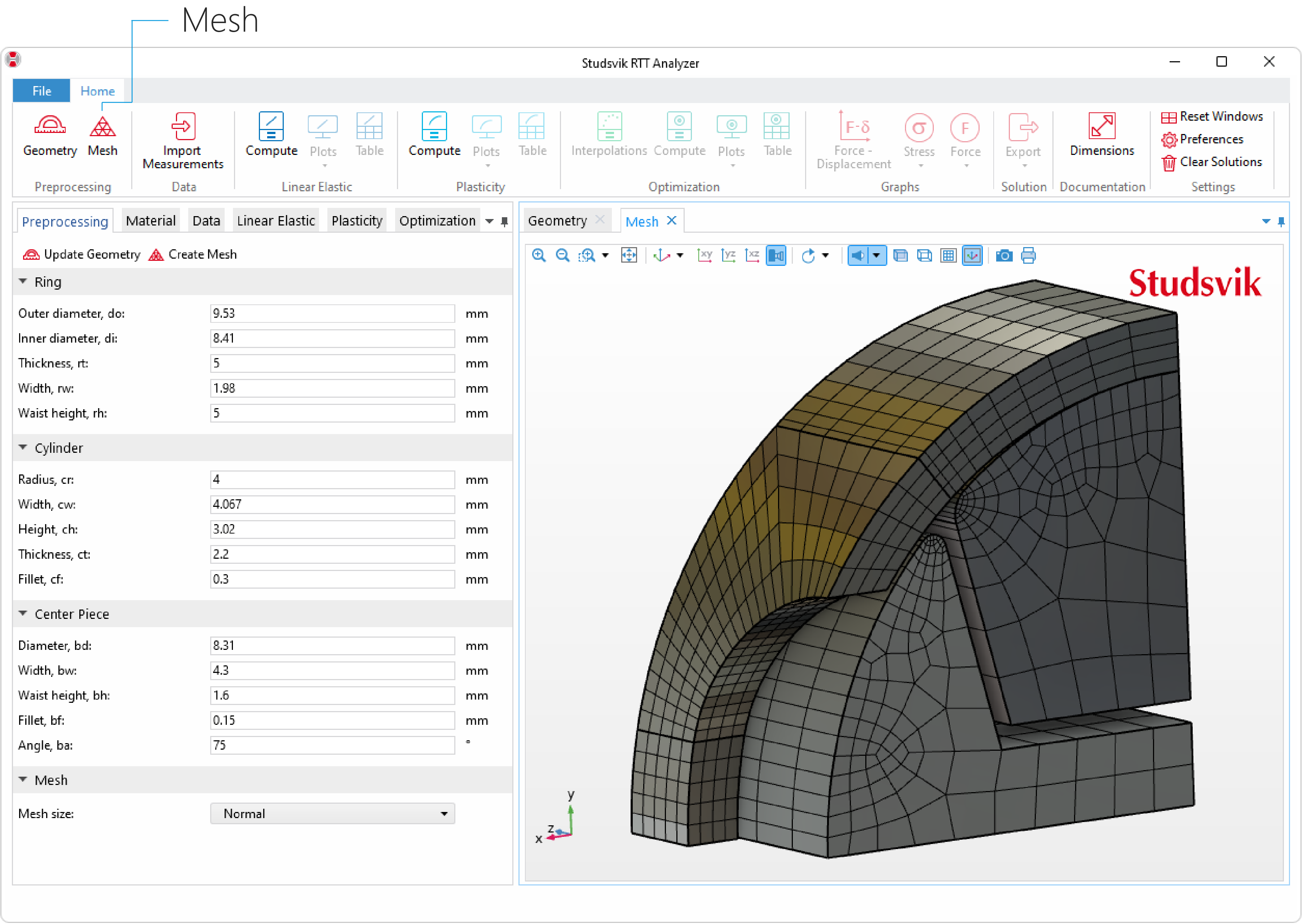


deflexional

Dimensions

- Open the PDF file with a description of the geometry parameters by clicking the *Dimensions* button in the ribbon
- Press the *Geometry* button to update the graphics

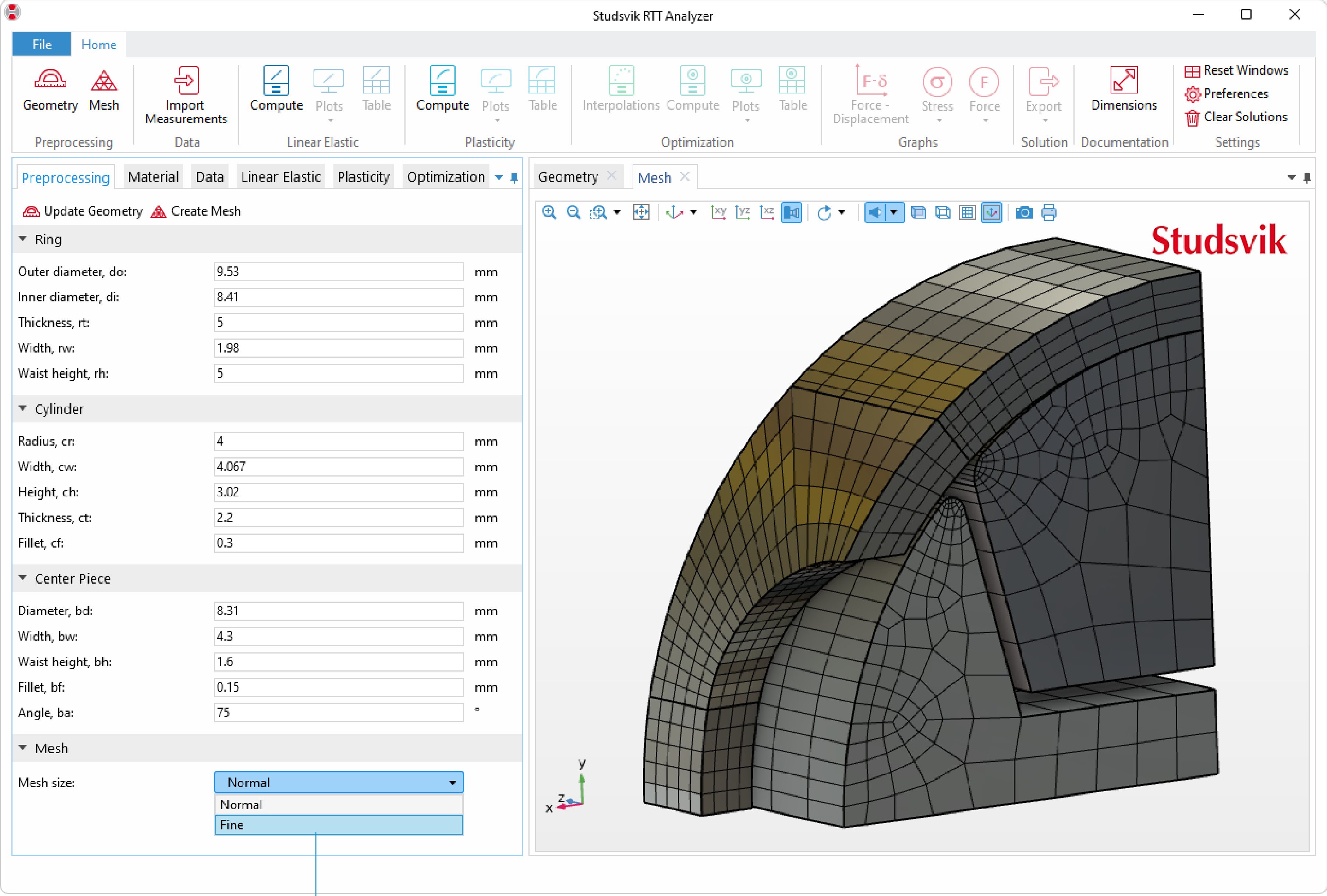




- Create the mesh with the *Mesh* button

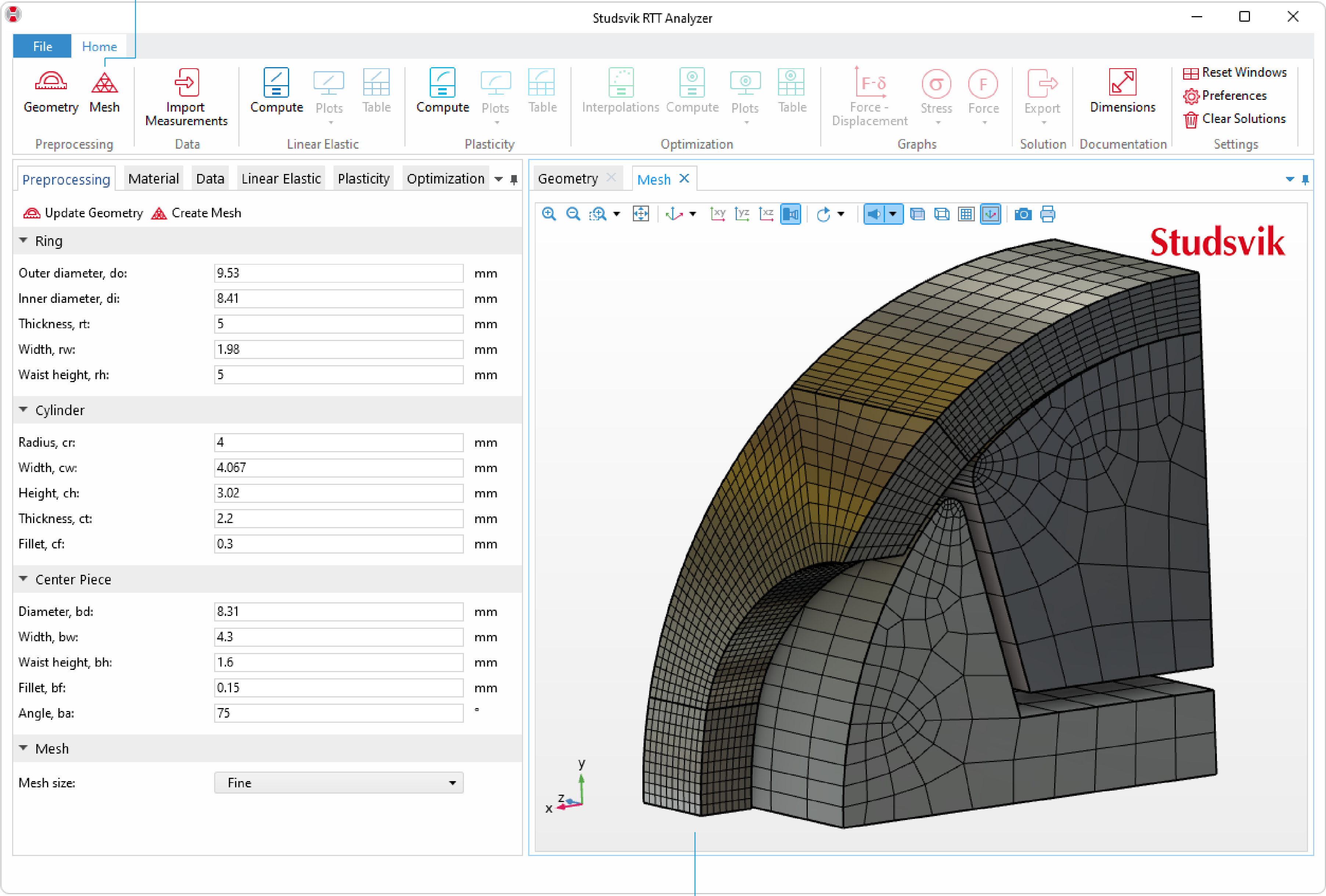
deflexional

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



Mesh size

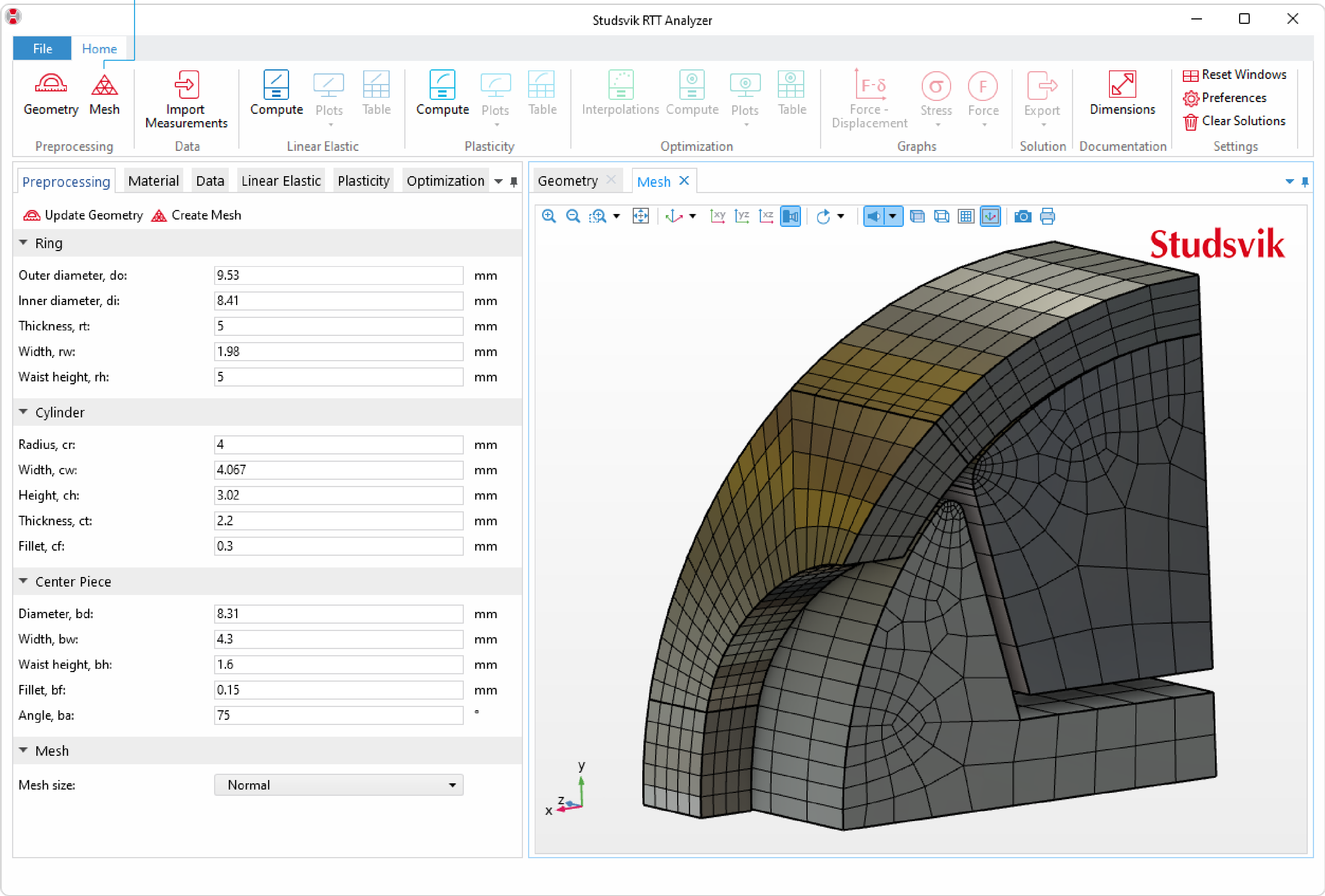
Mesh



- Note that the mesh is only refined in the ring

Mesh refinement

Mesh

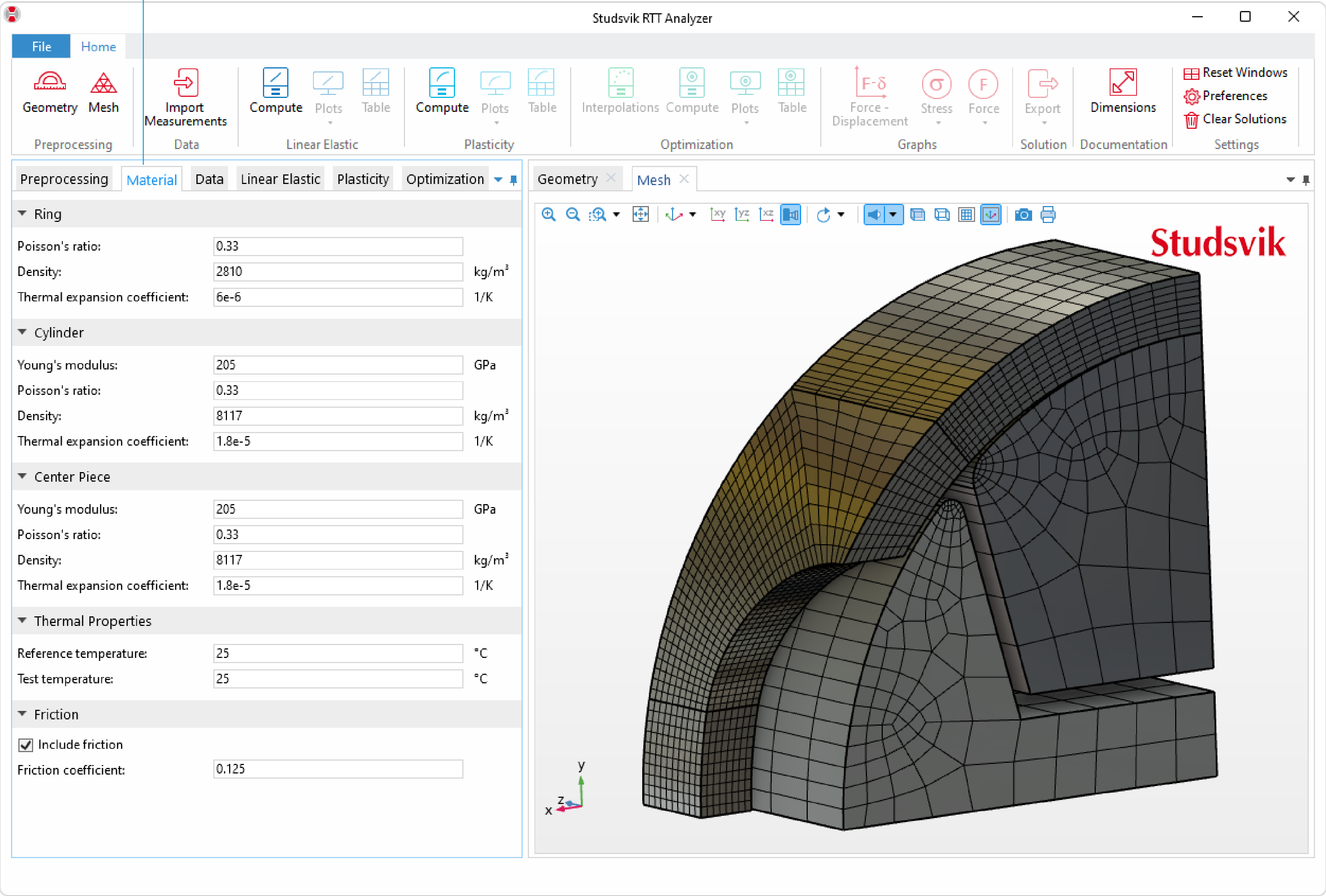


- Switch back to *Normal* mesh for faster calculations

Benchmark Test of an Aluminum Alloy

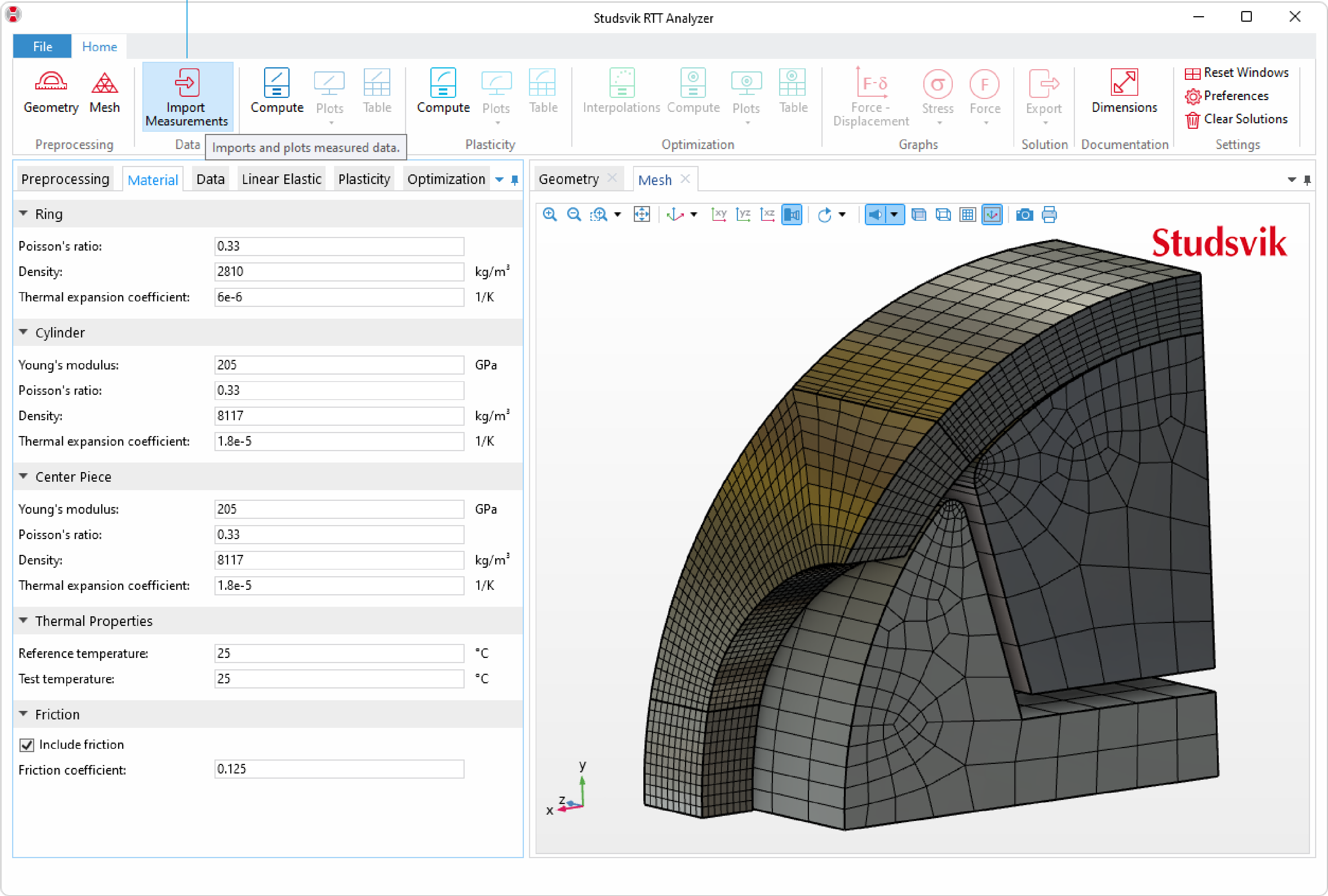
- Tests are normally performed for Zircalloys
  - The test temperature is high
- In this benchmark an aluminum alloy is tested in room temperature

Material settings



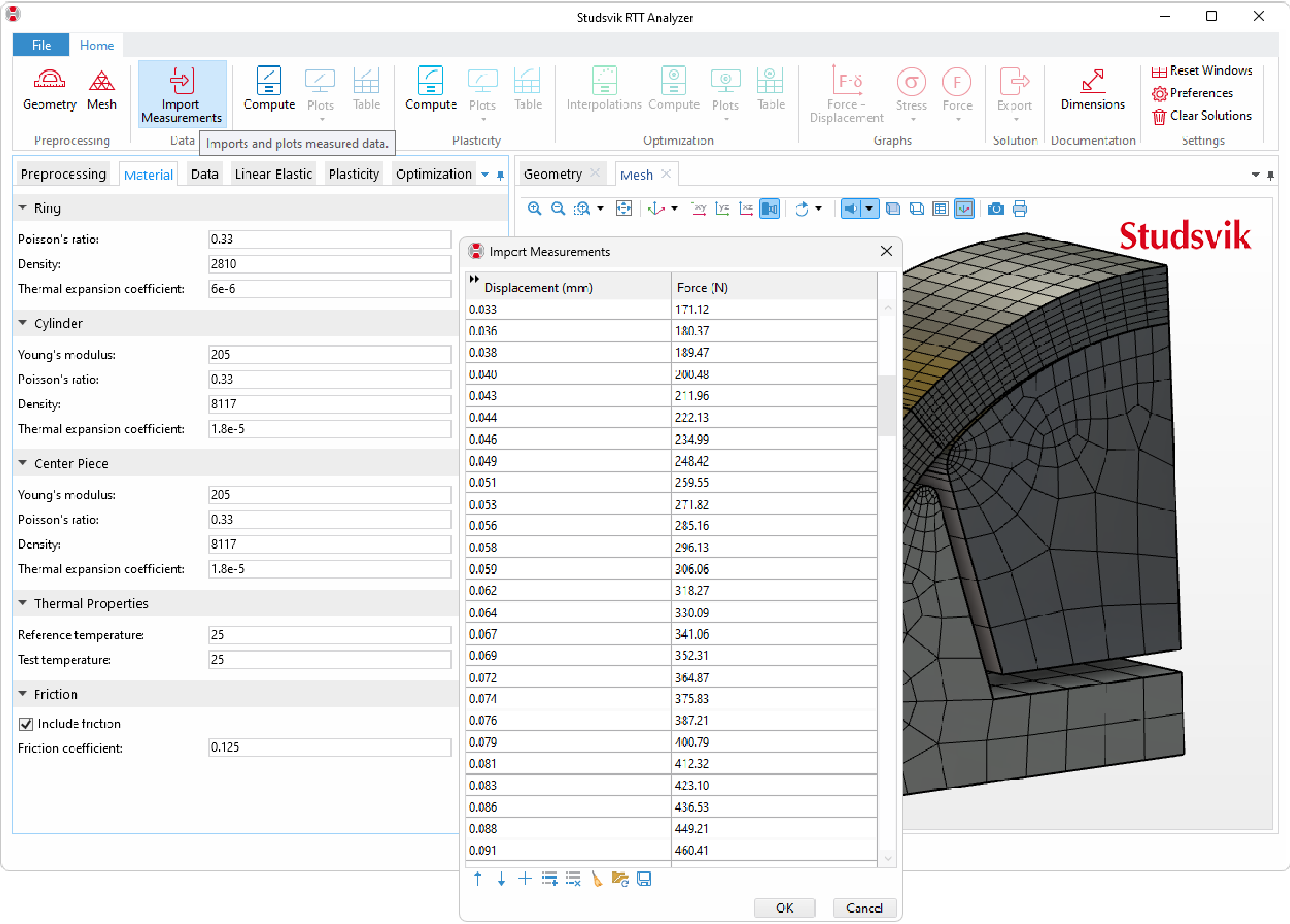
- Open the *Material* tab
- Enter material data
- Thermal expansion parameters are of huge importance when the test is performed in high temperatures
- Friction can be included

Import measured force and displacement

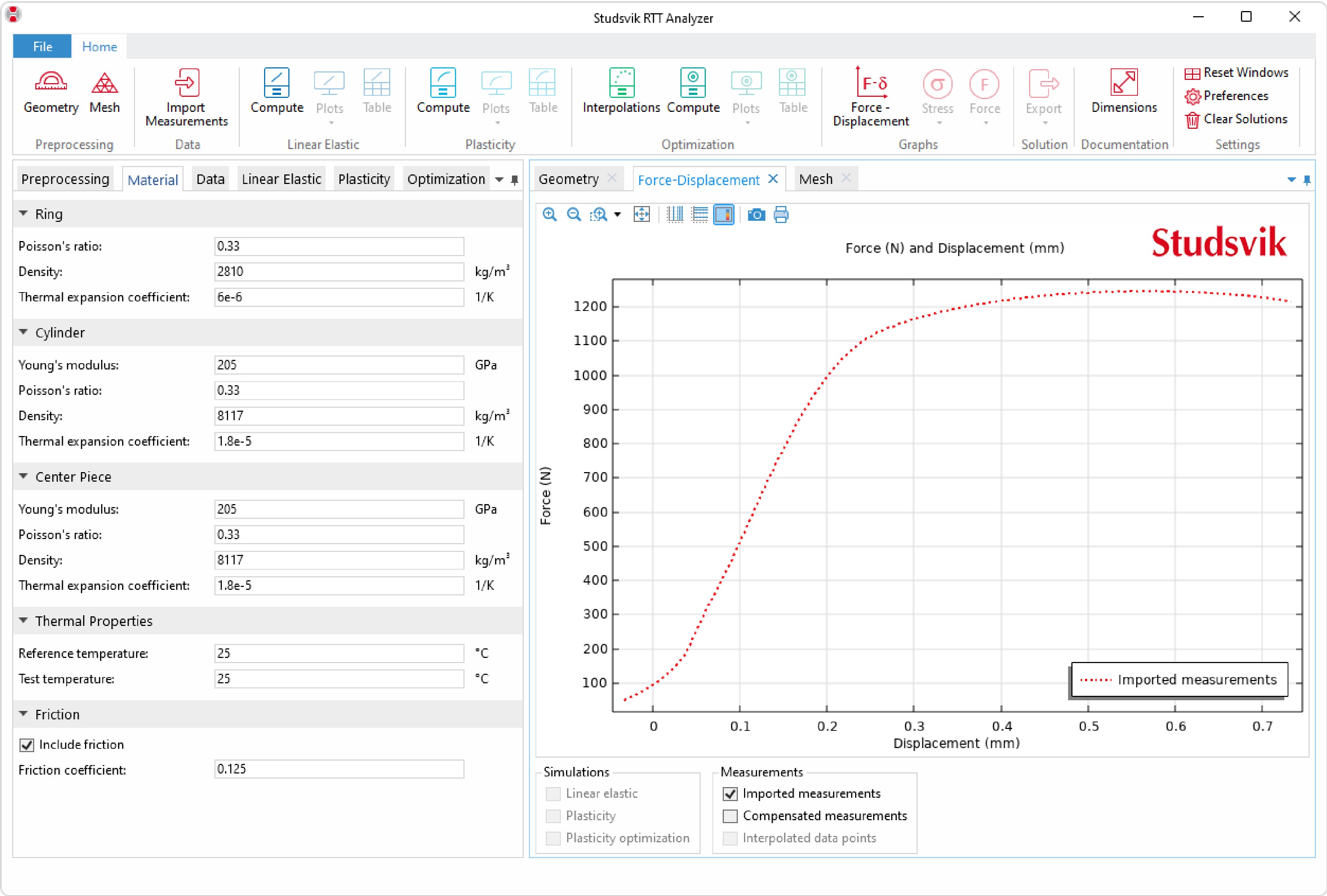


- Press *Import Measurements* to import force and displacement data

- Paste data from Excel® or import files



- A force and displacement graph is automatically plotted

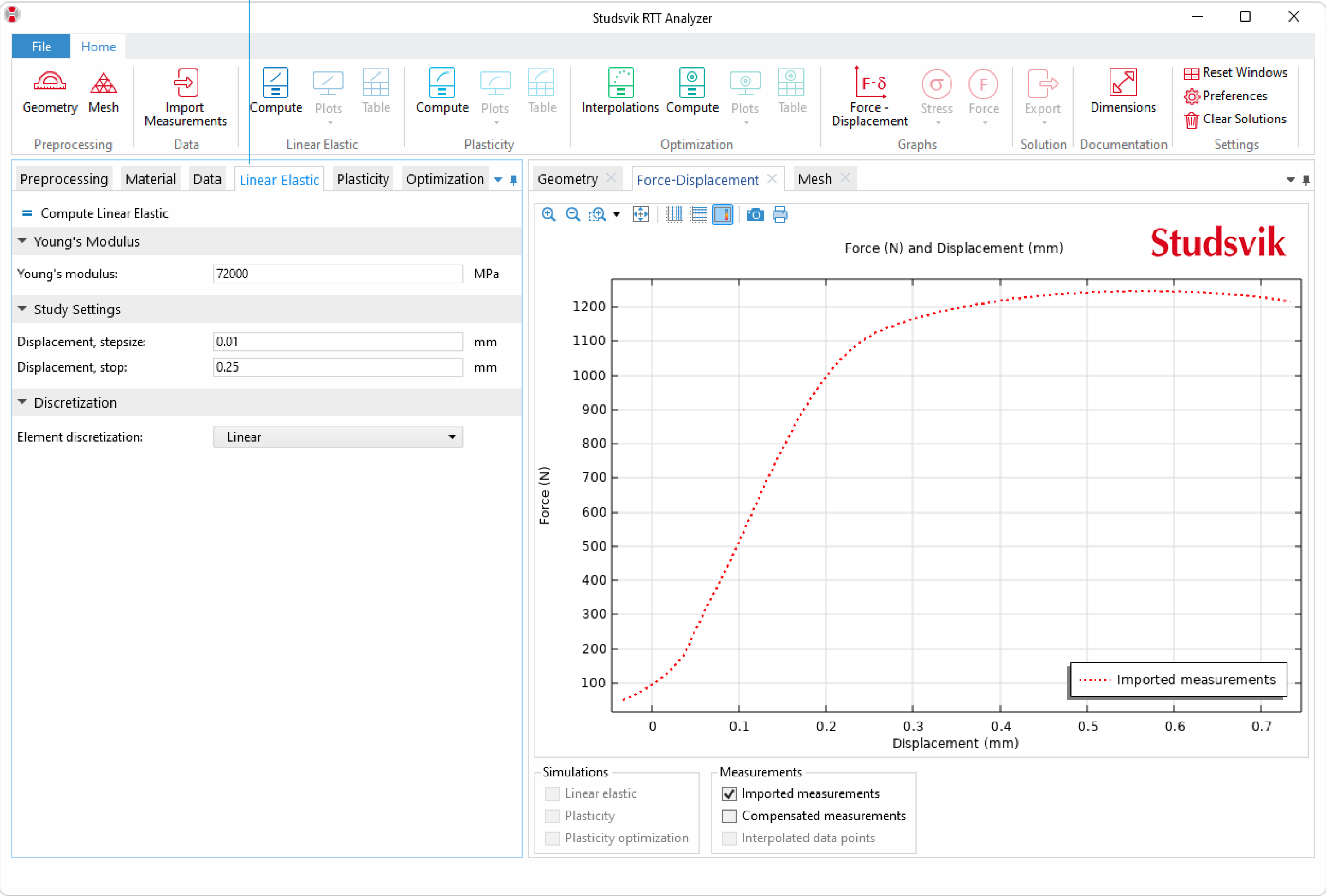


Linear Elastic Analysis

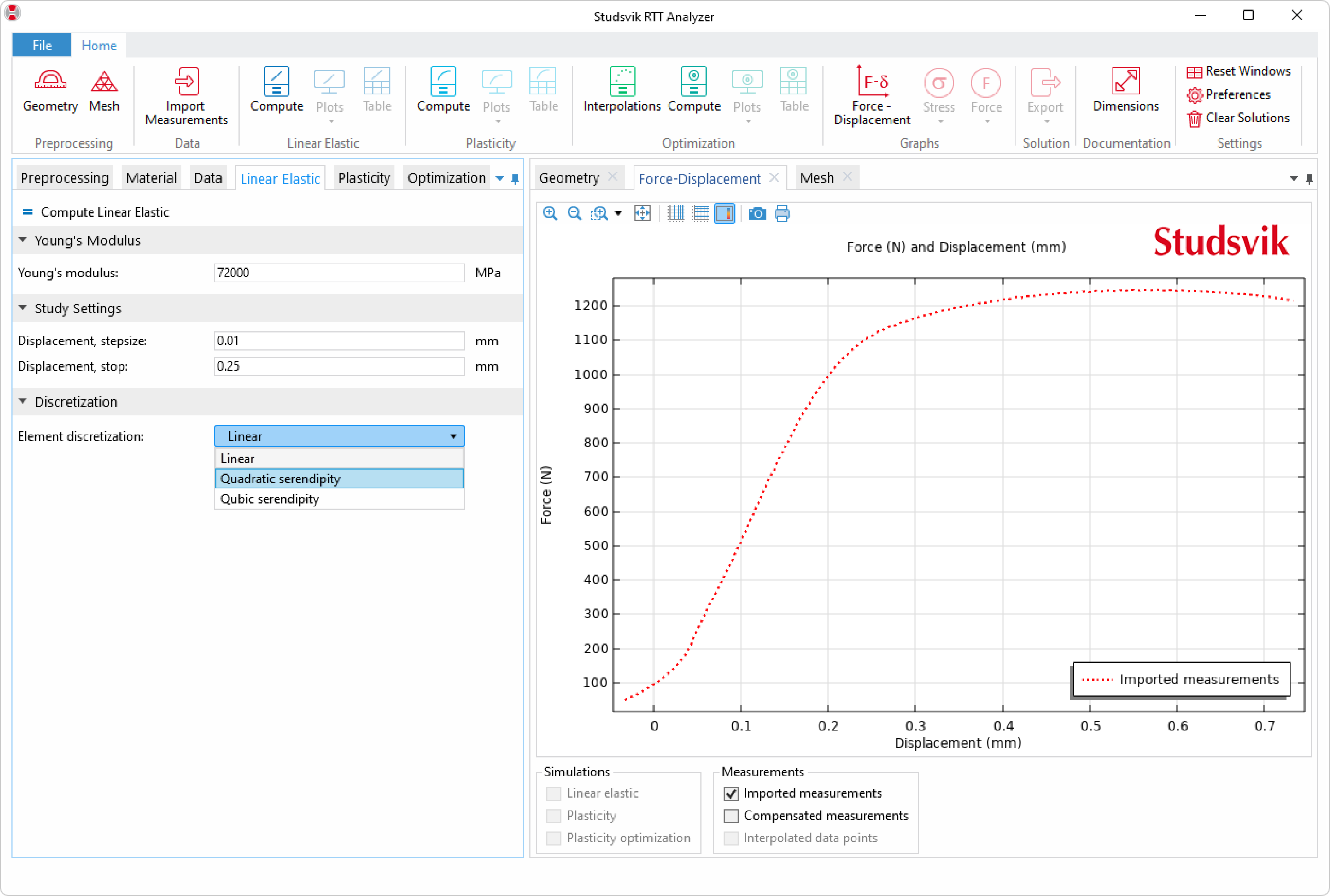
- Young’s Modulus is known
- Begin with a linear elastic study to obtain results fast
- Compare the results with the measured data

Linear Elastic settings

- Click on the *Linear Elastic* tab



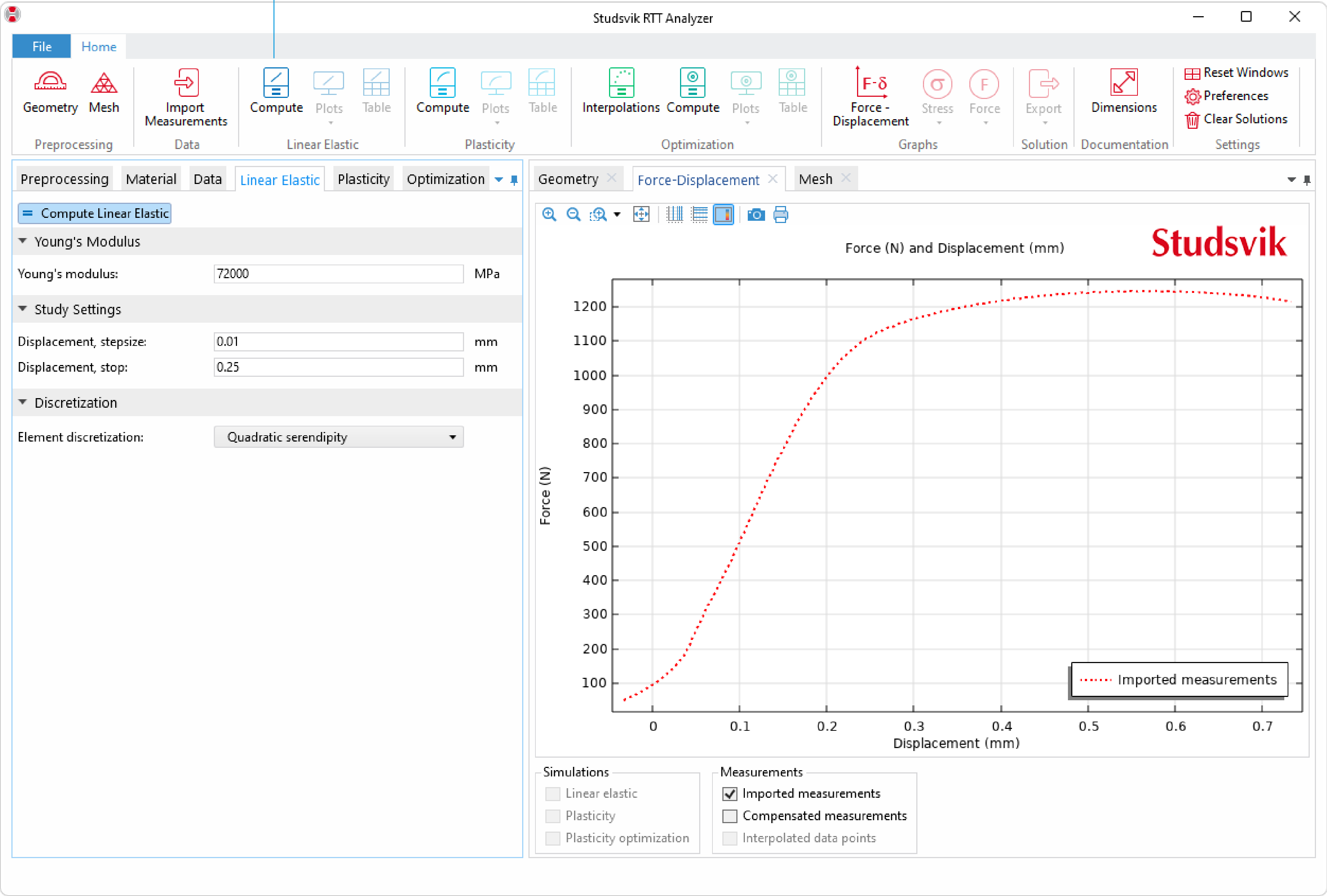
- Switch to Quadratic elements



deflexional

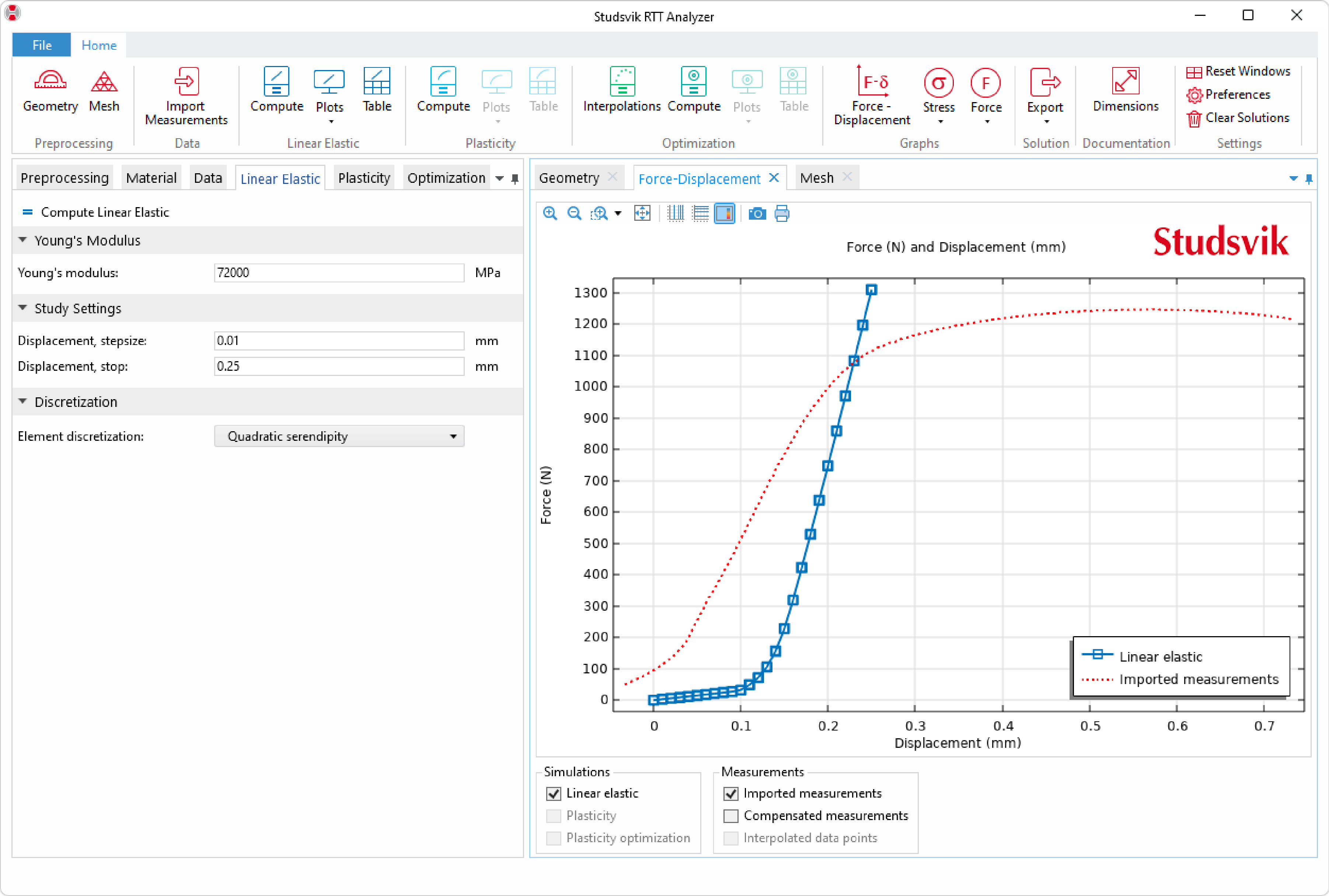
Compute

Compute



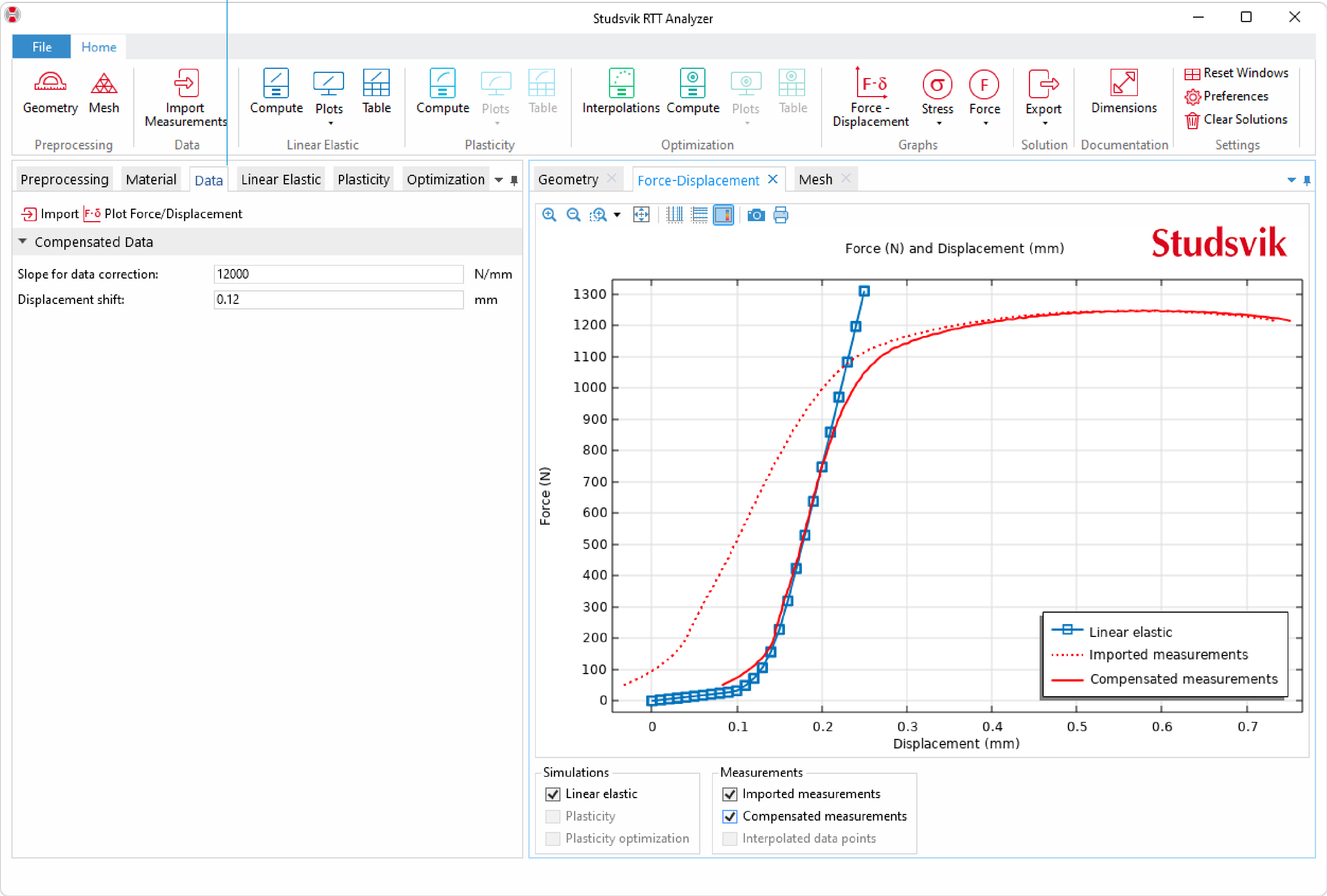
- Enter Young's modulus which is a well-known parameter for most materials
- In the *Study Settings* section, enter the displacement step size and stop
- Press any of the *Compute* buttons to start the simulation

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



deflexional

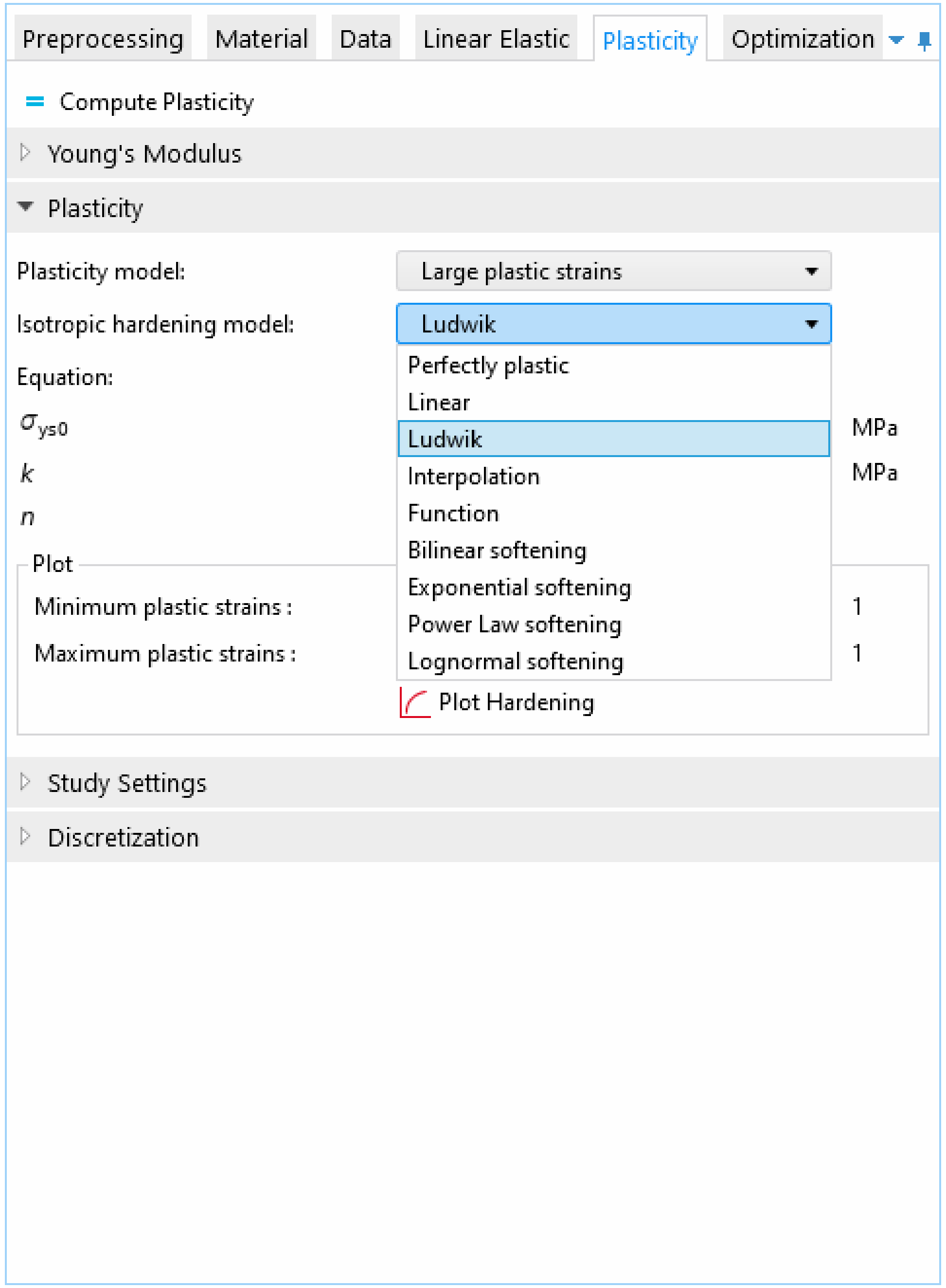
Data settings



- 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

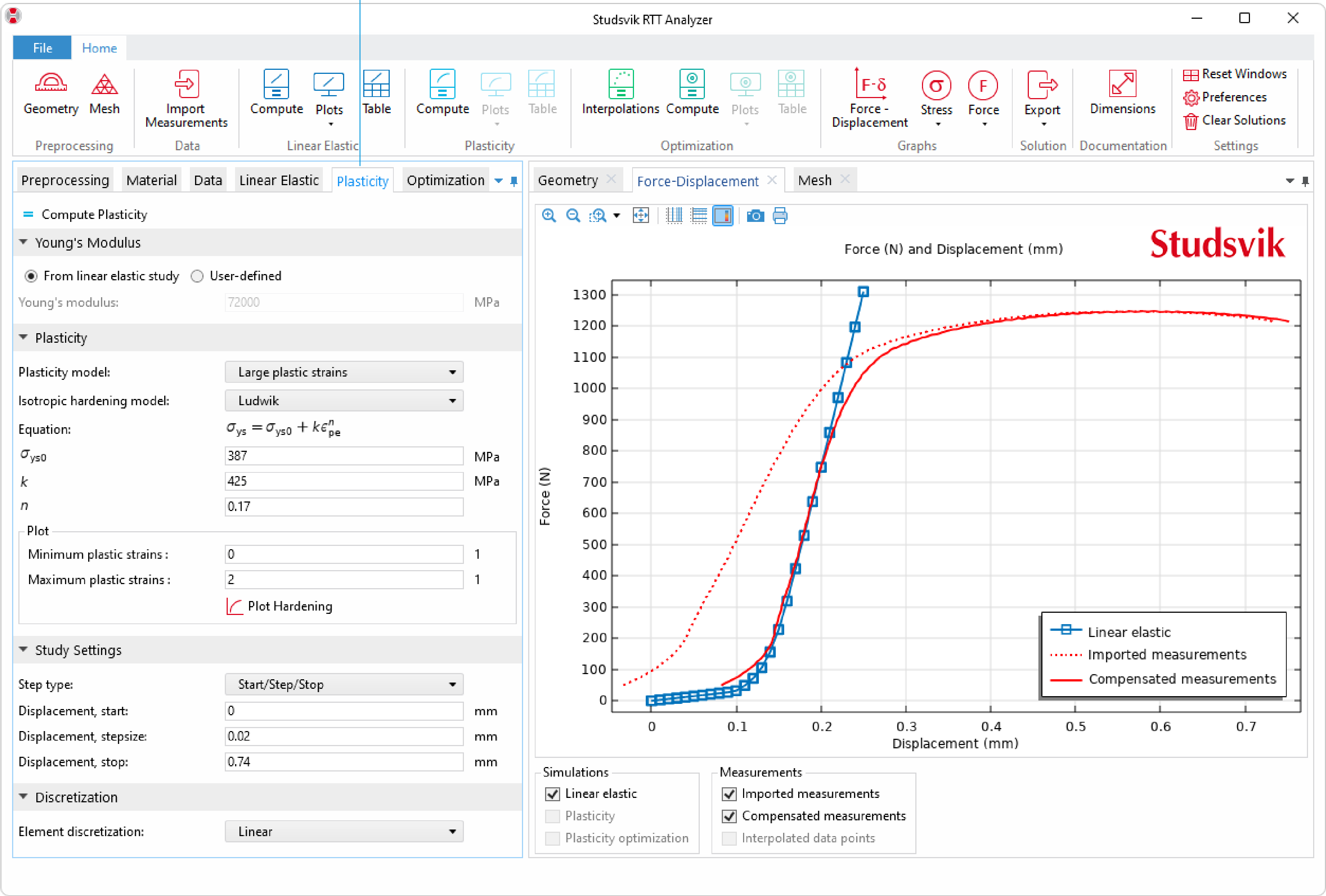
Plasticity

- 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

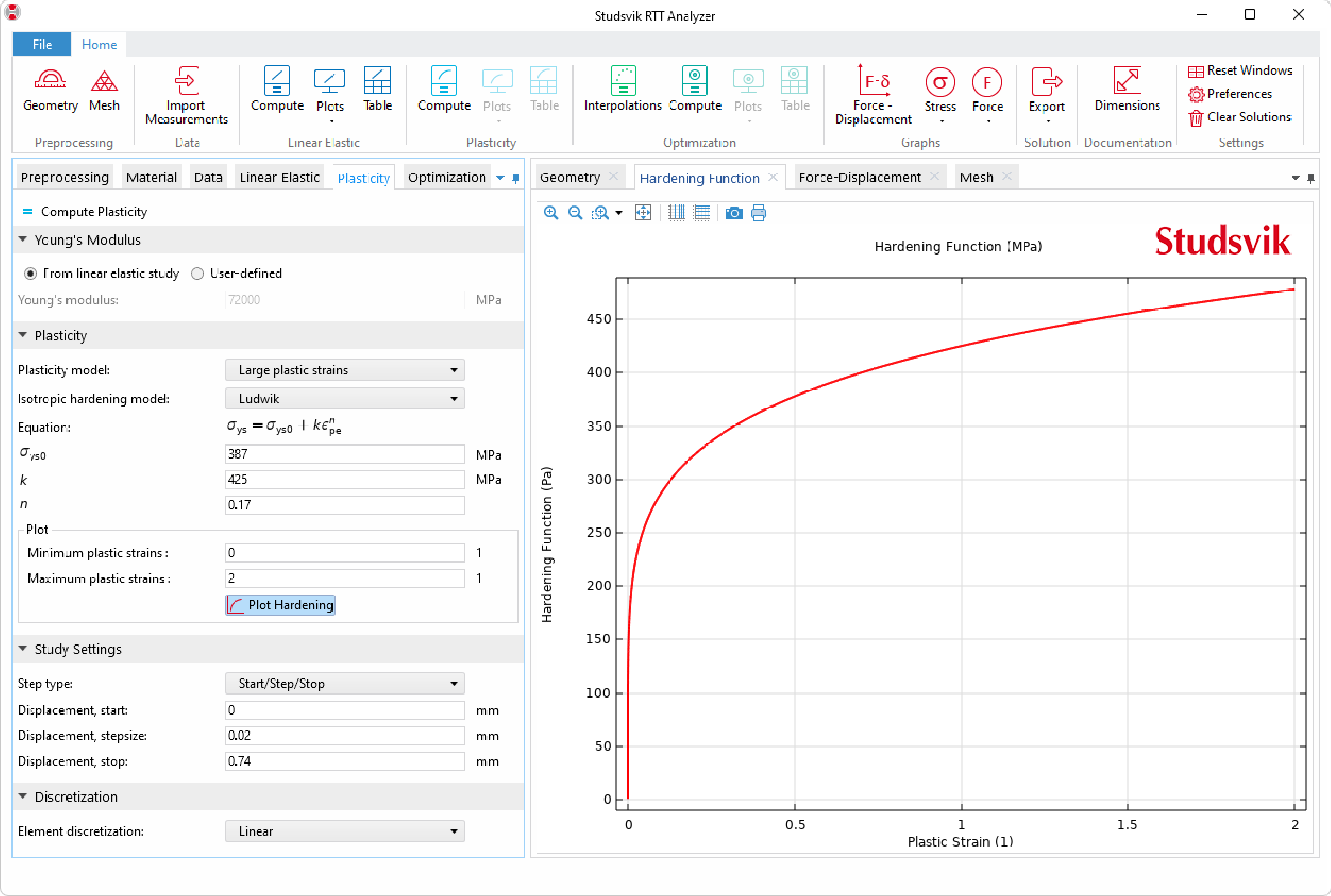


Plasticity settings

- 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

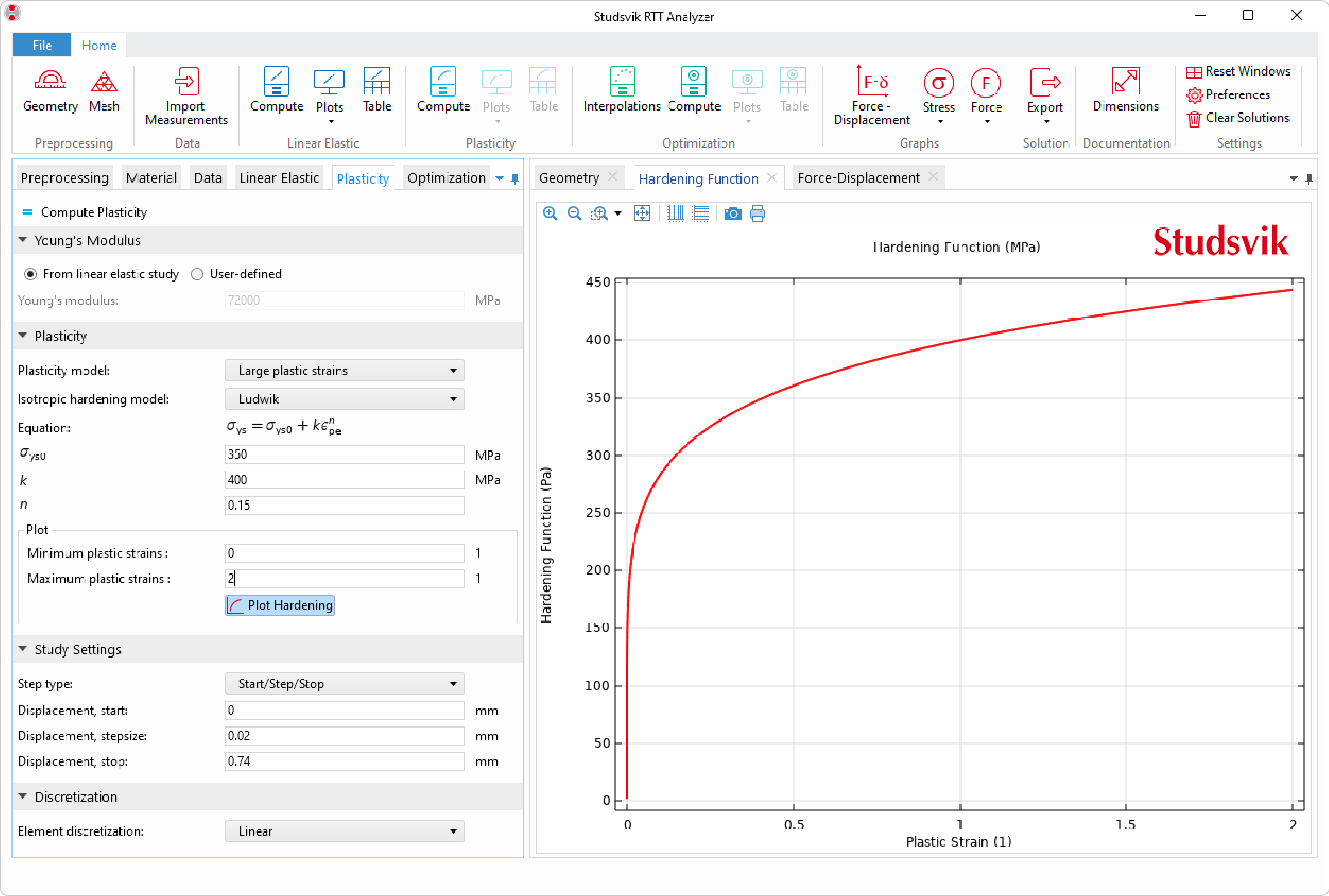


- Press the *Plot Hardening* button to visualize the selected hardening function

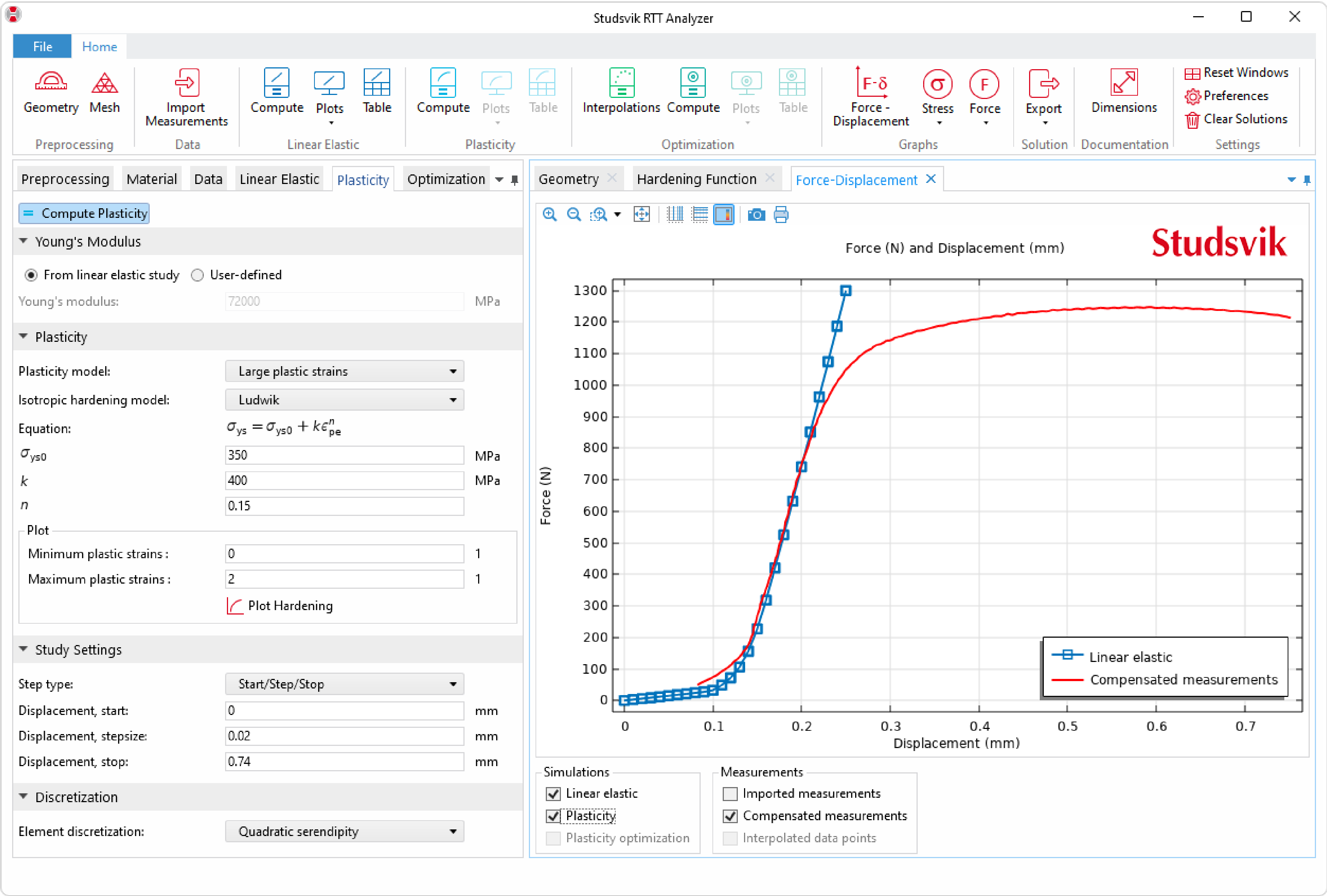


deflexional

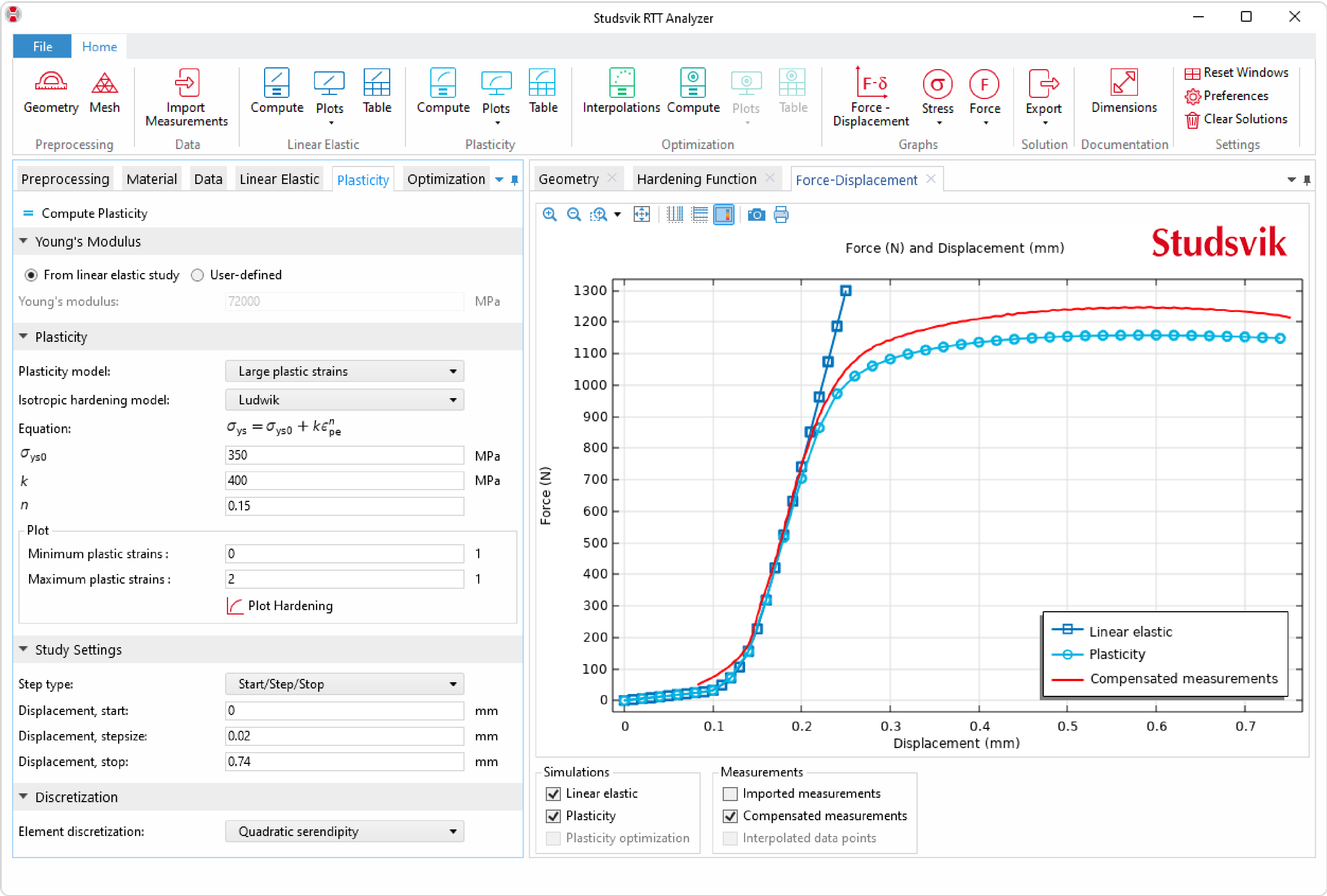
- Enter new hardening parameters and plot the hardening function



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

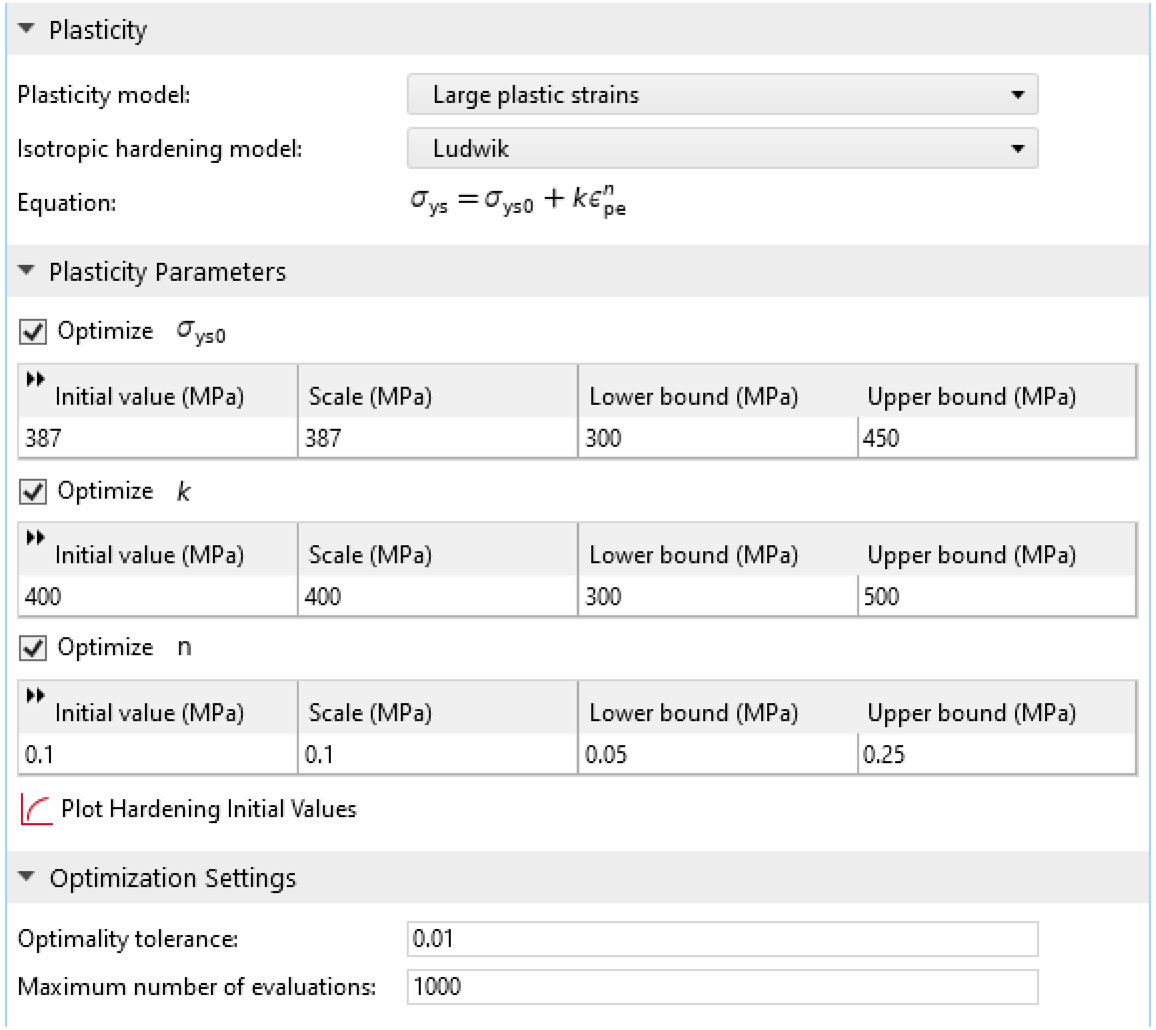


- The result is not satisfactory, and the user can try with new parameters to fit the plasticity results to the compensated measurements



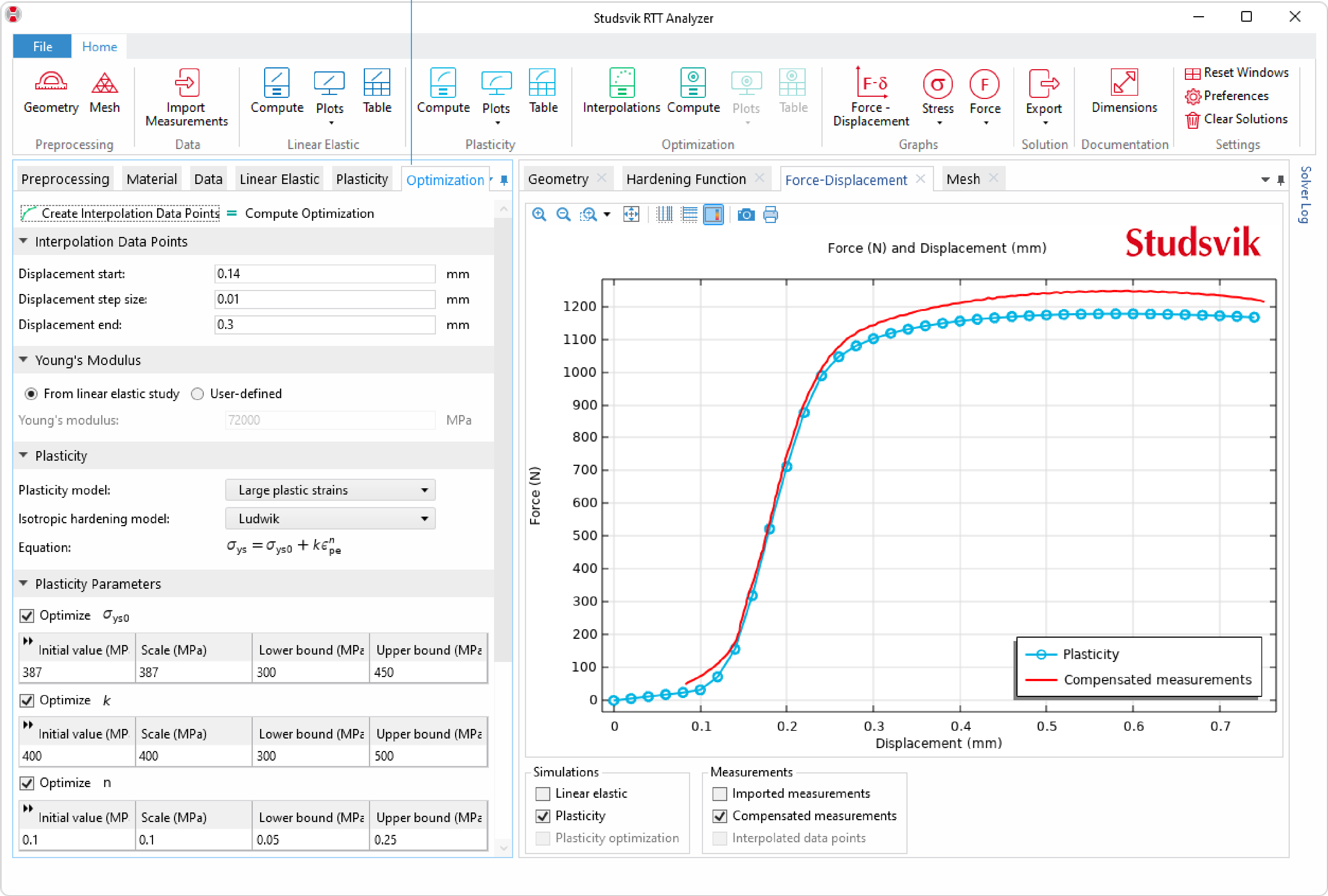
Optimization

- 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

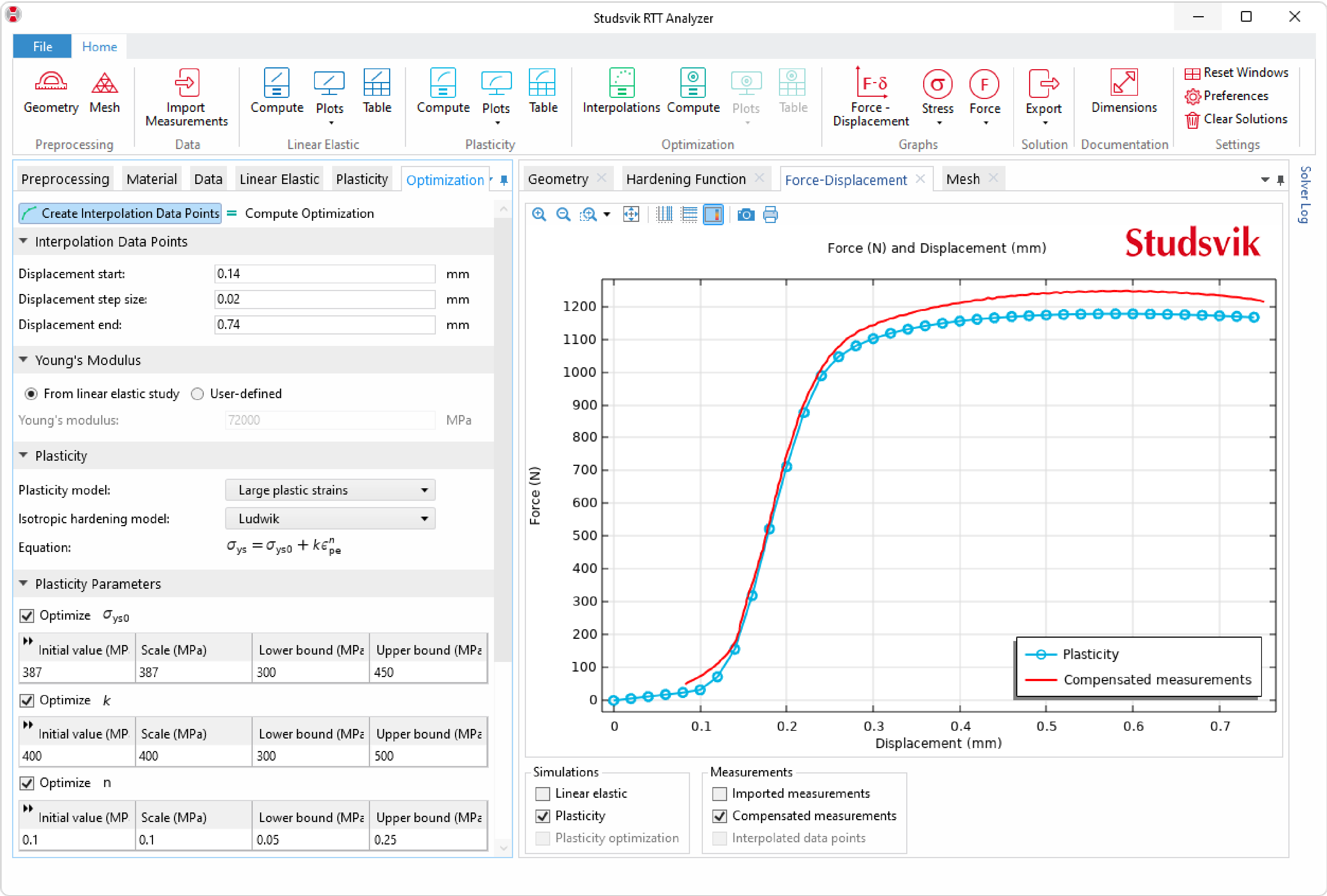


Optimization settings

- Click the *Optimization* tab

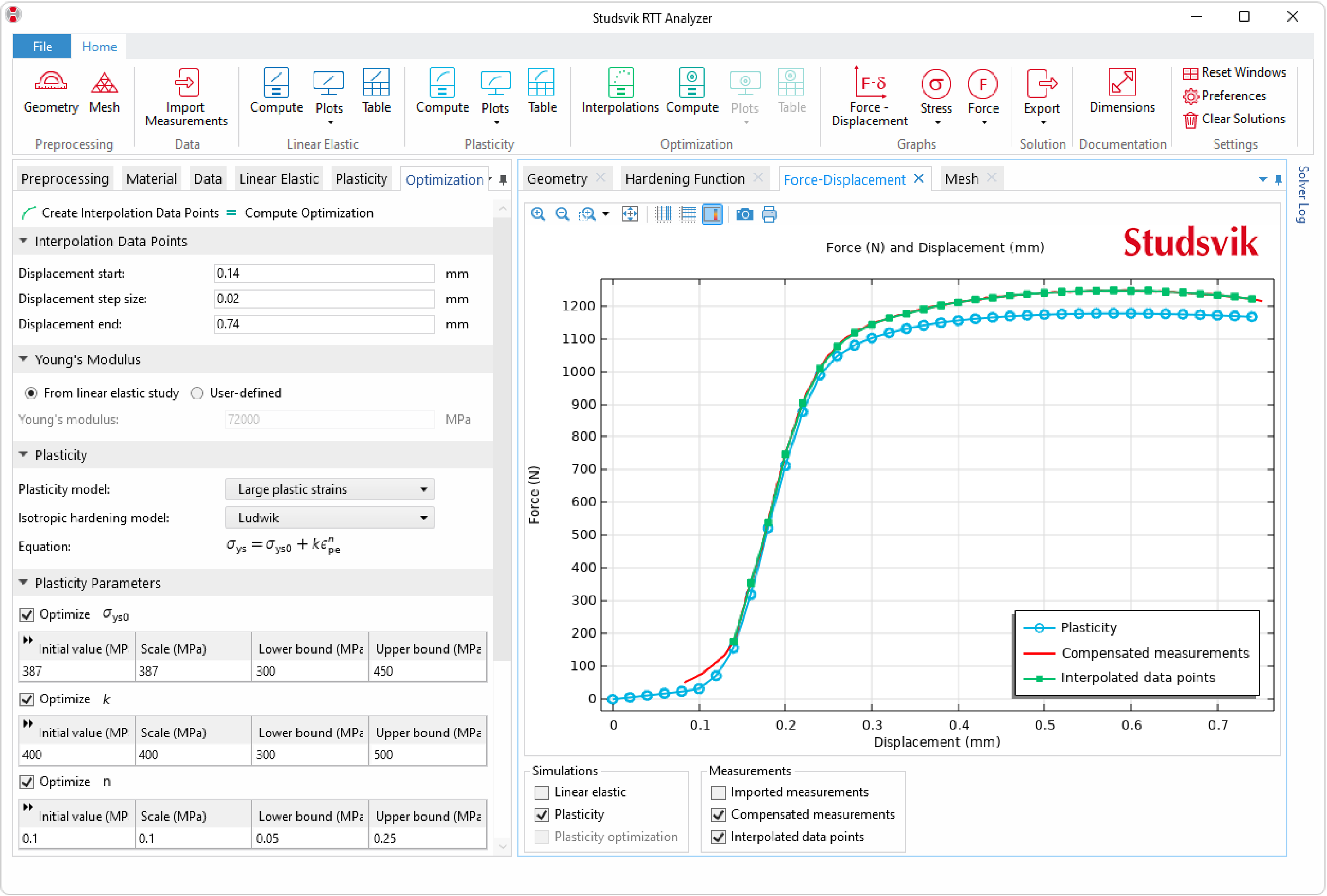


- In the *Interpolation Data Points* section, enter start, step size and stop
- This data will create points following the *Compensated measured* graph
- The optimization will use these interpolation points as “targets”
- Press *Create Interpolation Data Points*

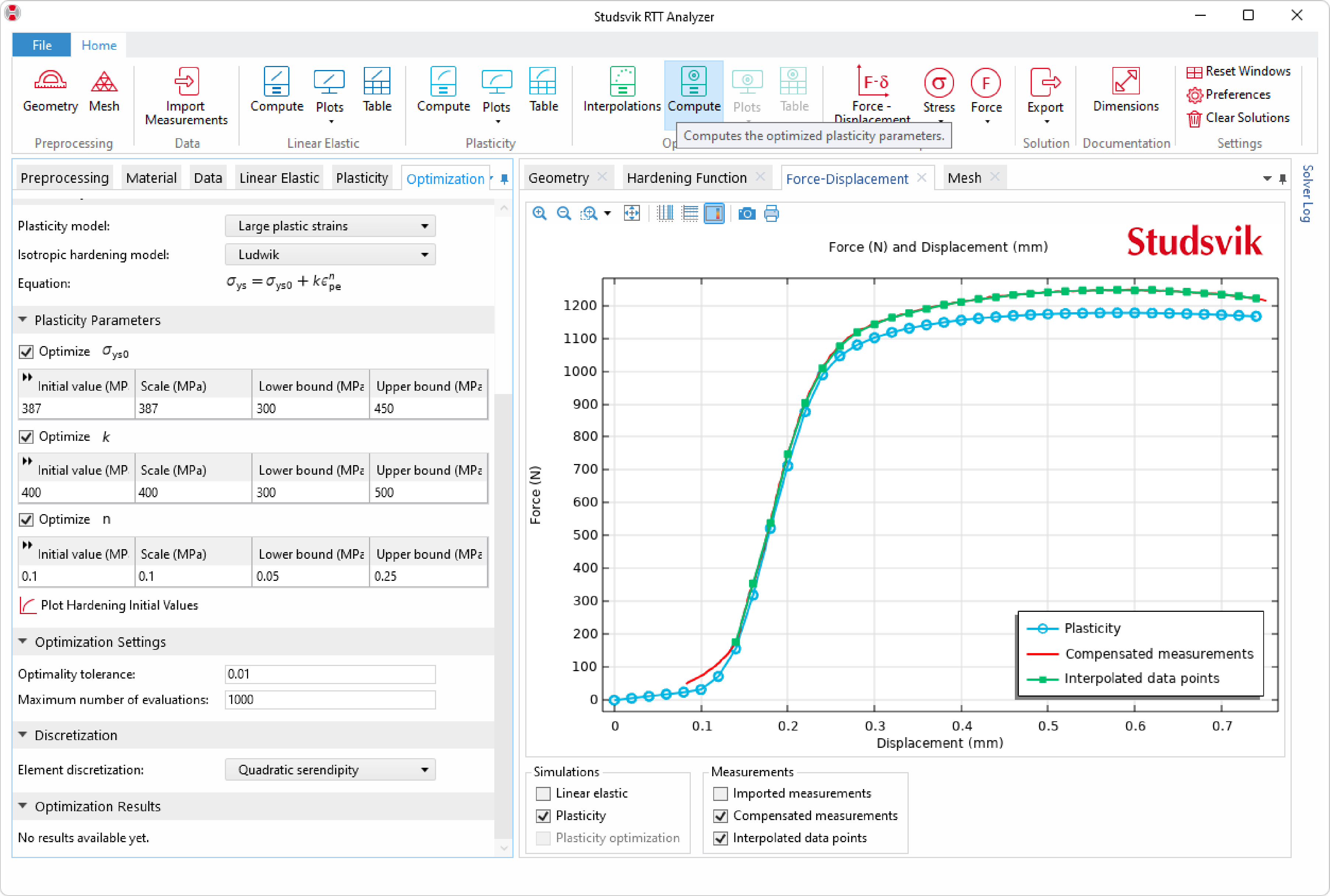


deflexional

- 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

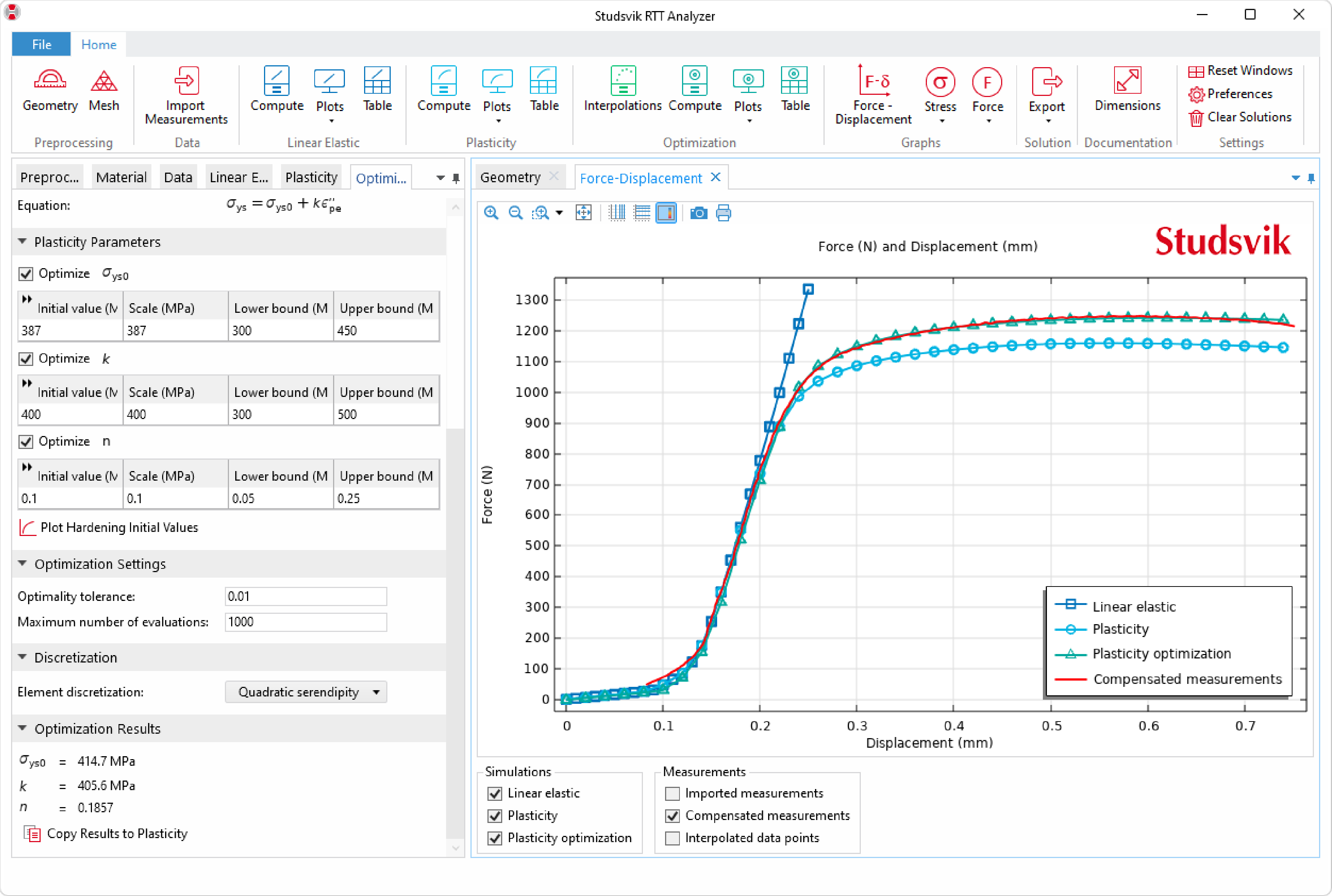


- In the *Plasticity Parameters* section, check the hardening parameters that should be optimized
- Provide initial values, scales, and lower and upper bounds
- In the *Optimization Settings* section, a tolerance and the maximum number of evaluations can be set
- Press the *Compute* button in the *Optimization*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

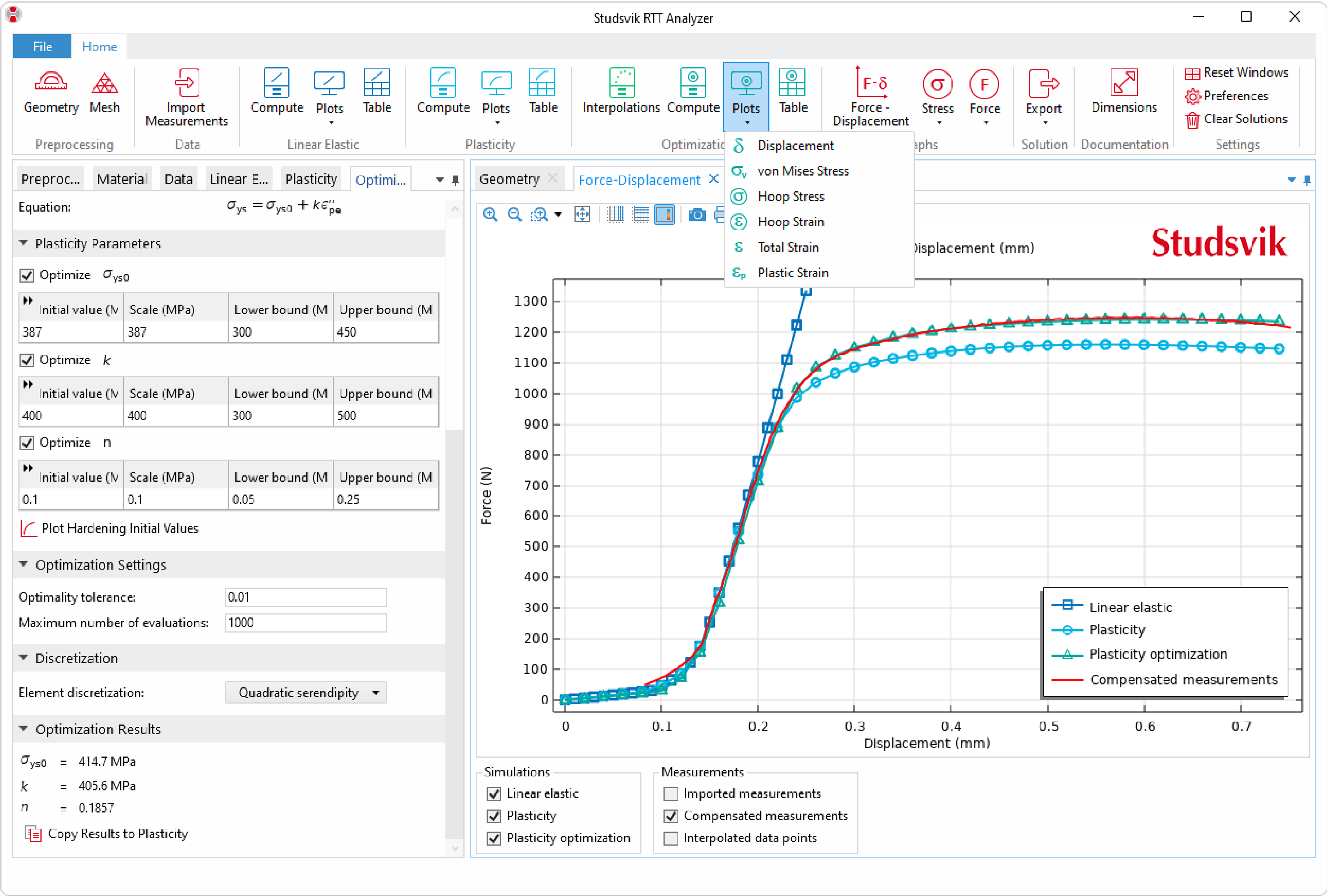


deflexional

- 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

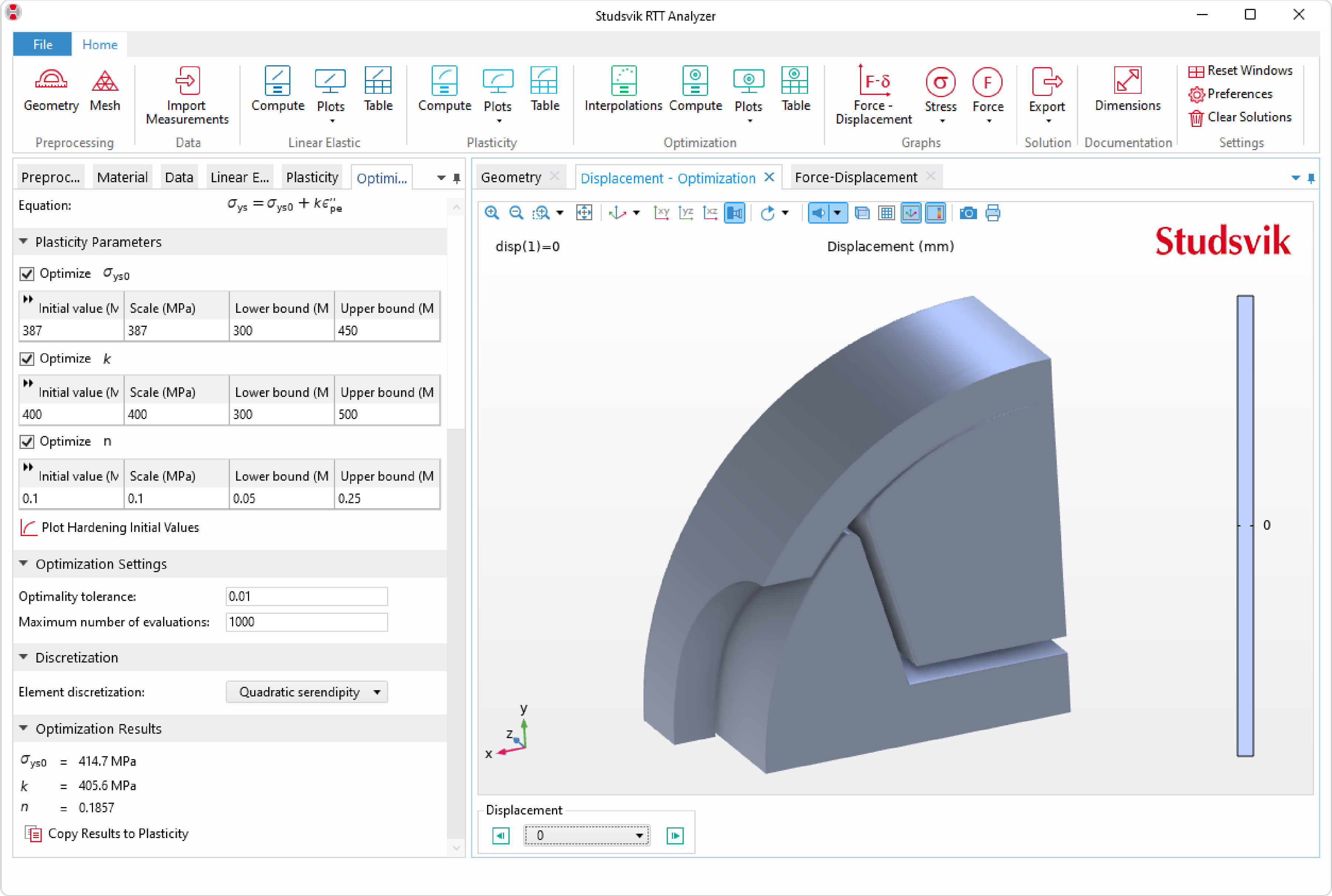


- 3D solution plots are available from the *Plots* menu in the ribbon



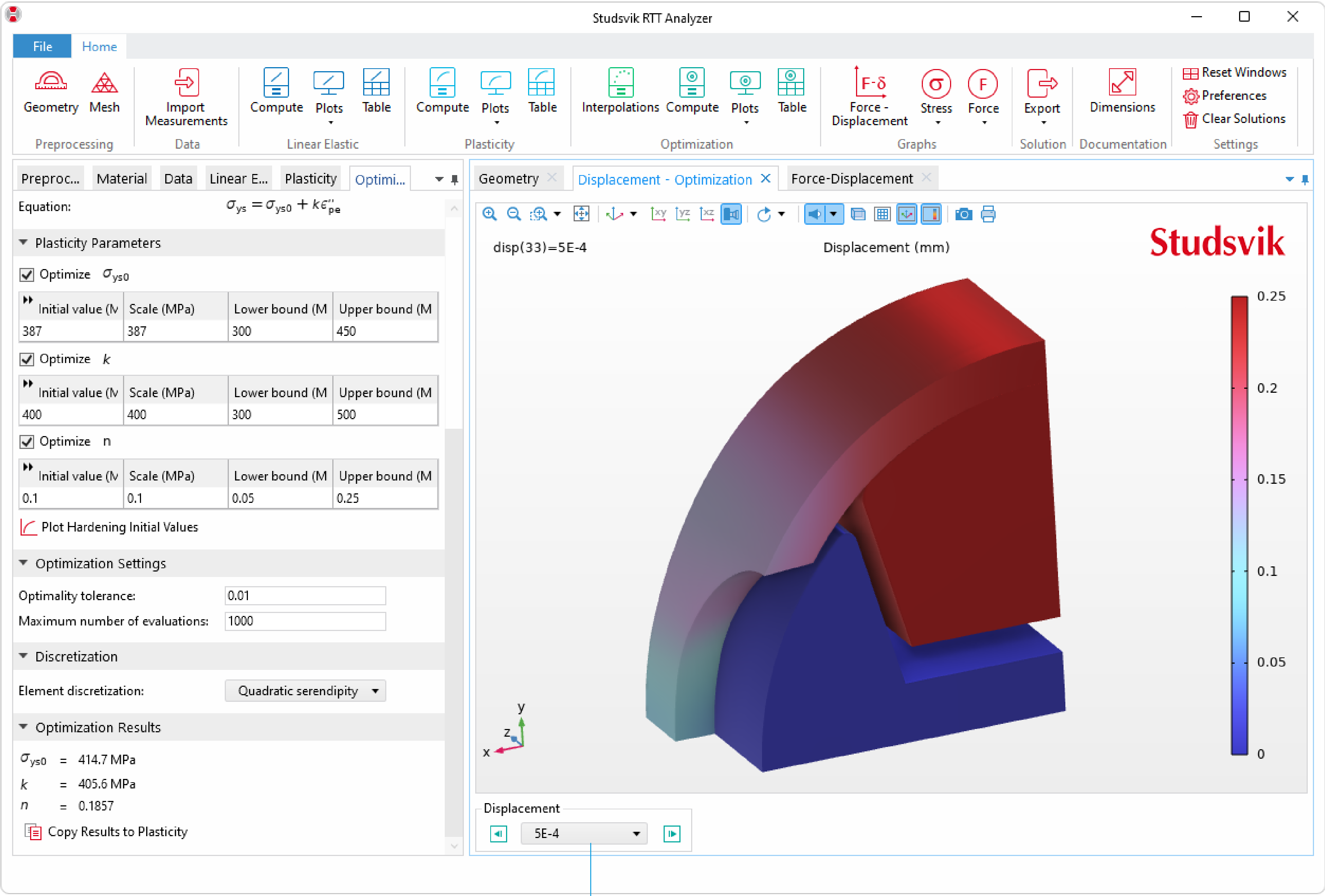
deflexional

- The initial displacement is zero



deflexional

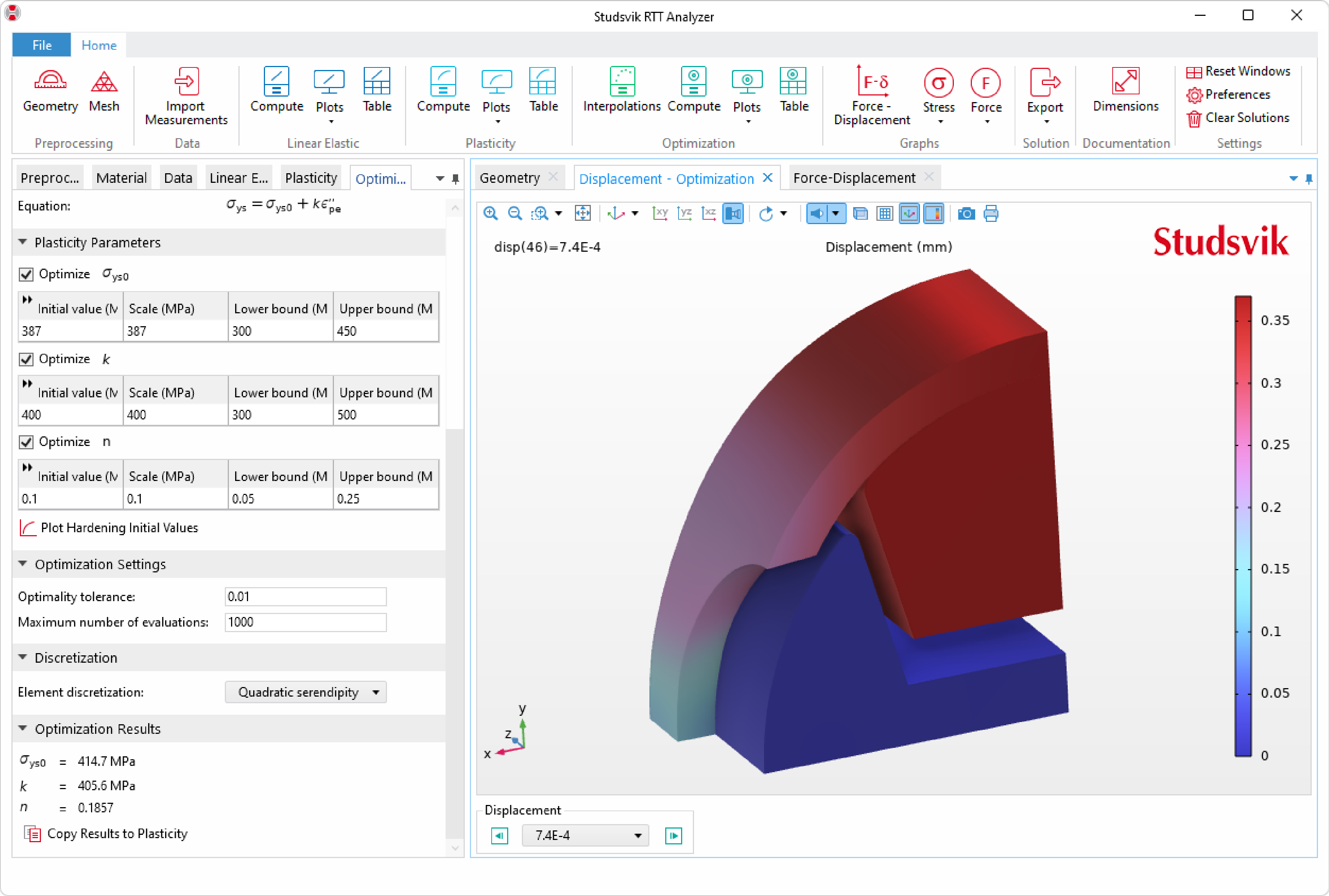
- Use the combo-box below the graphics to change the solution step



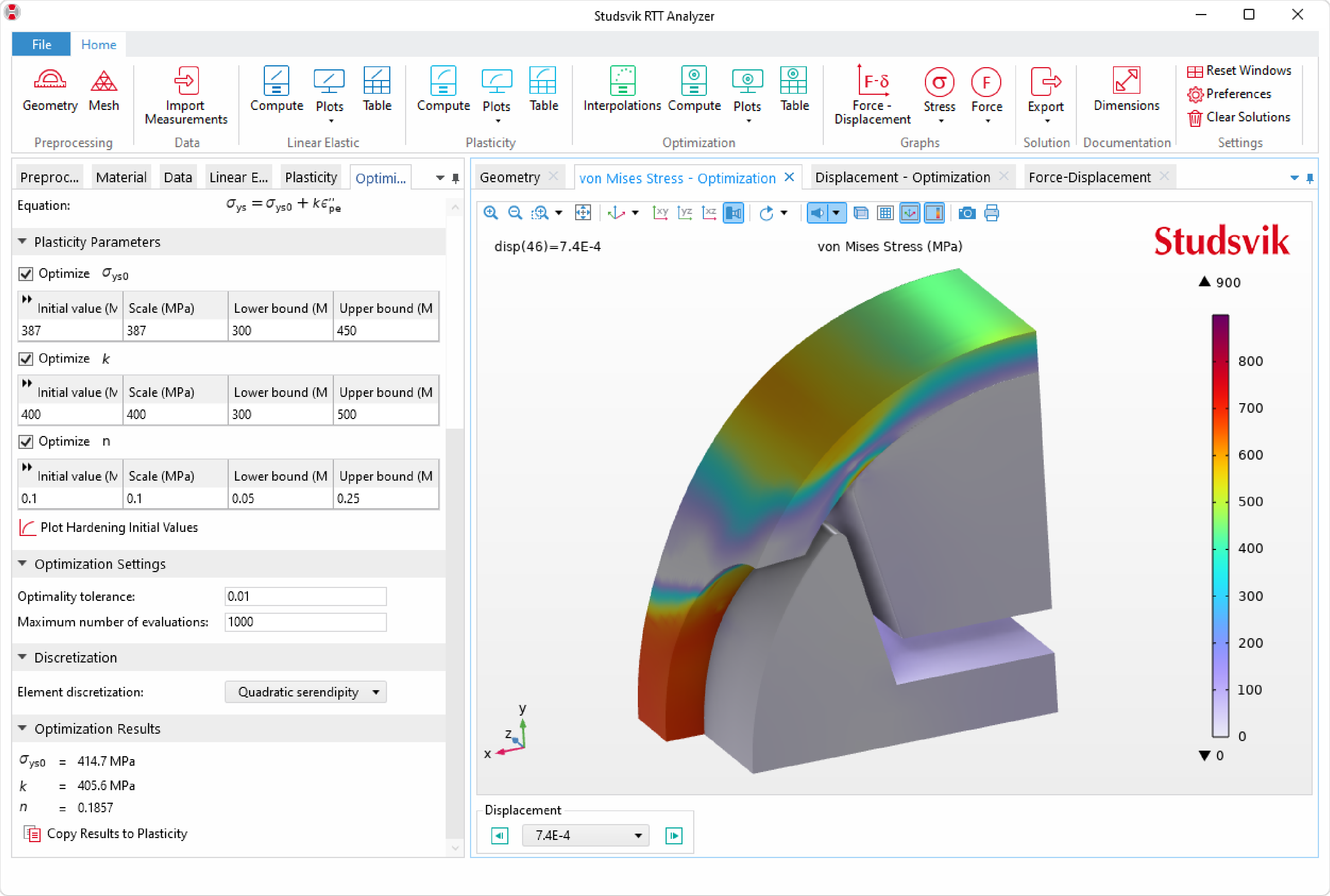
Change the solution step

deflexional

- Visualize the last step

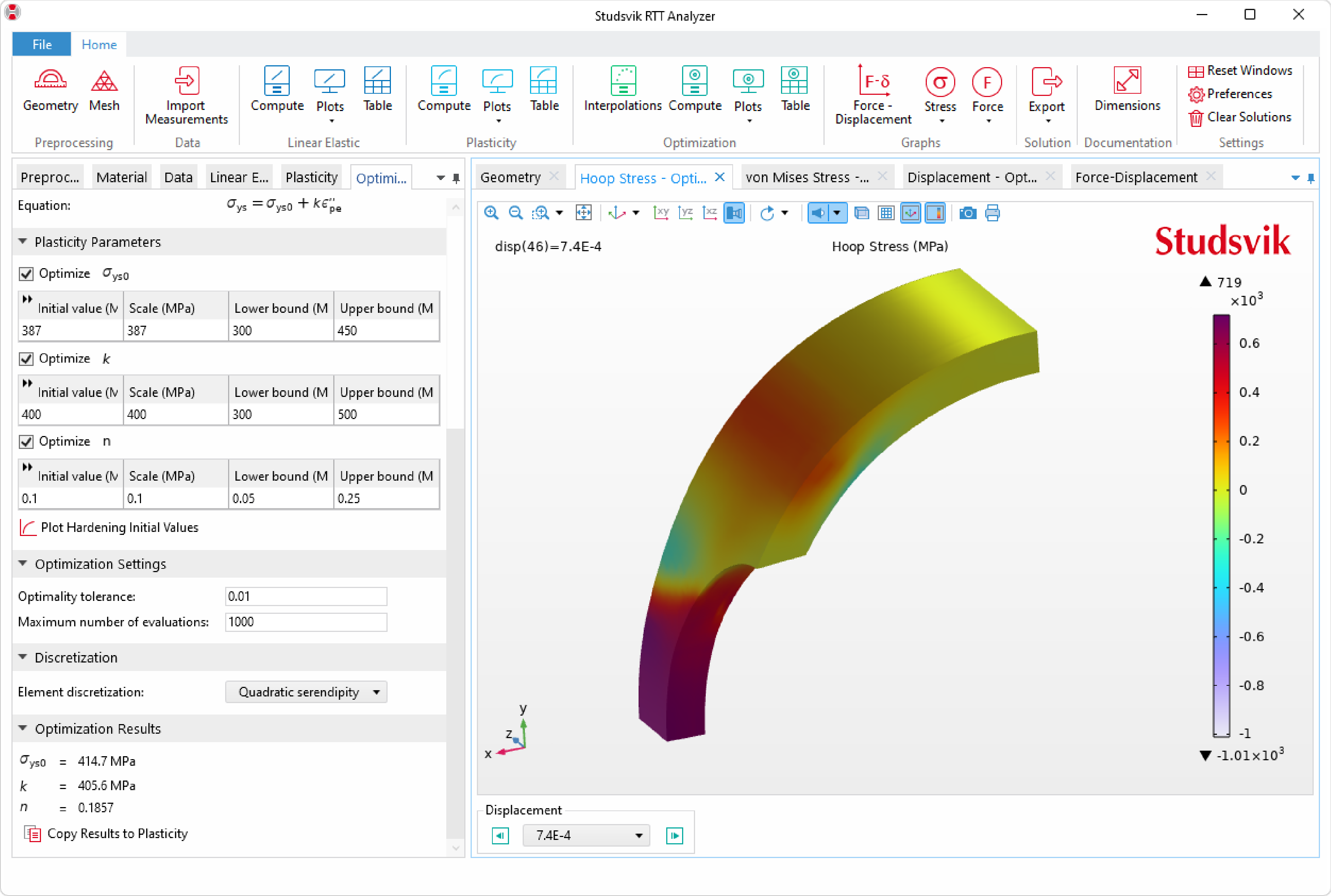


- The *von Mises Stress* at the last step



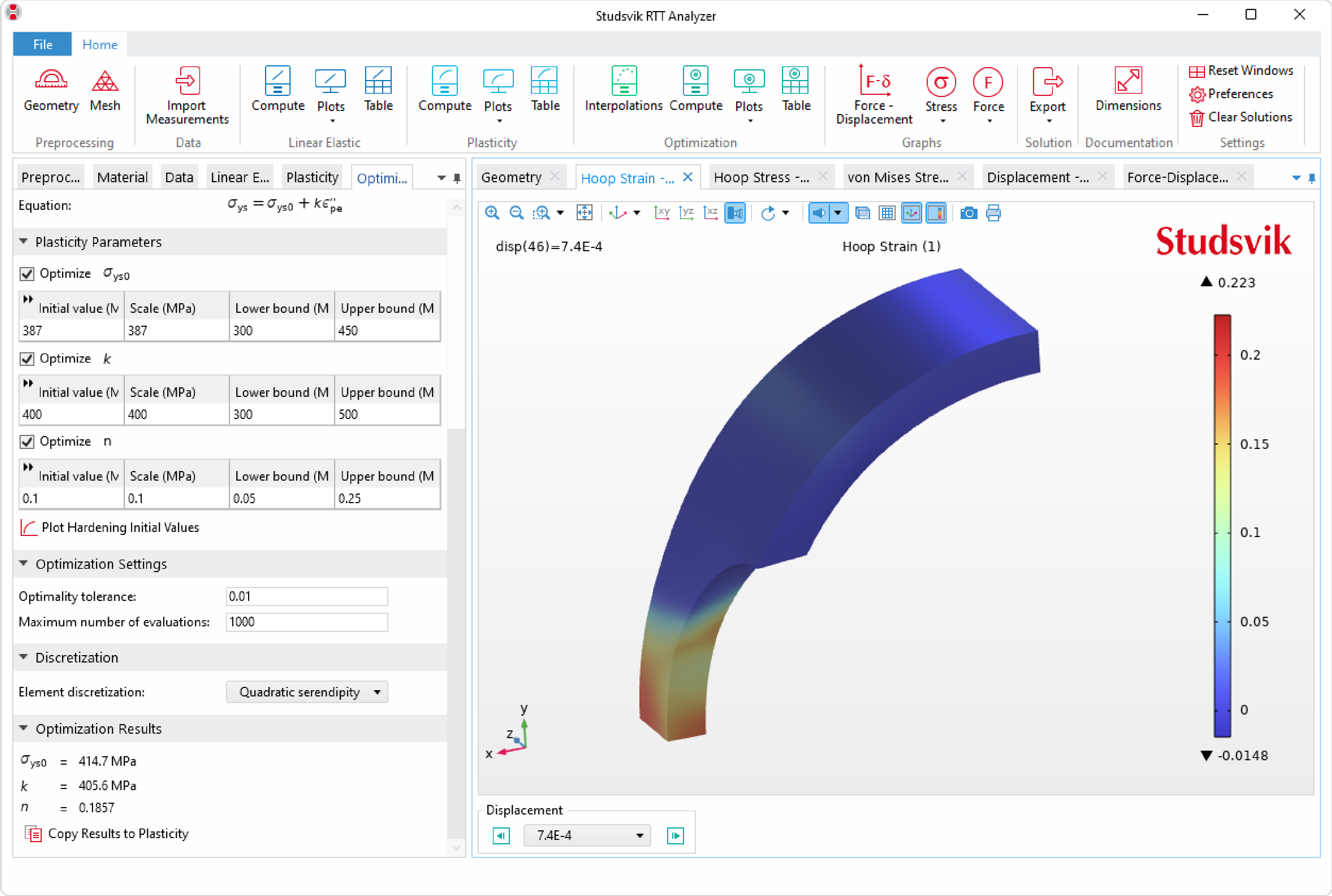
deflexional

- The *Hoop Stress* at the last step

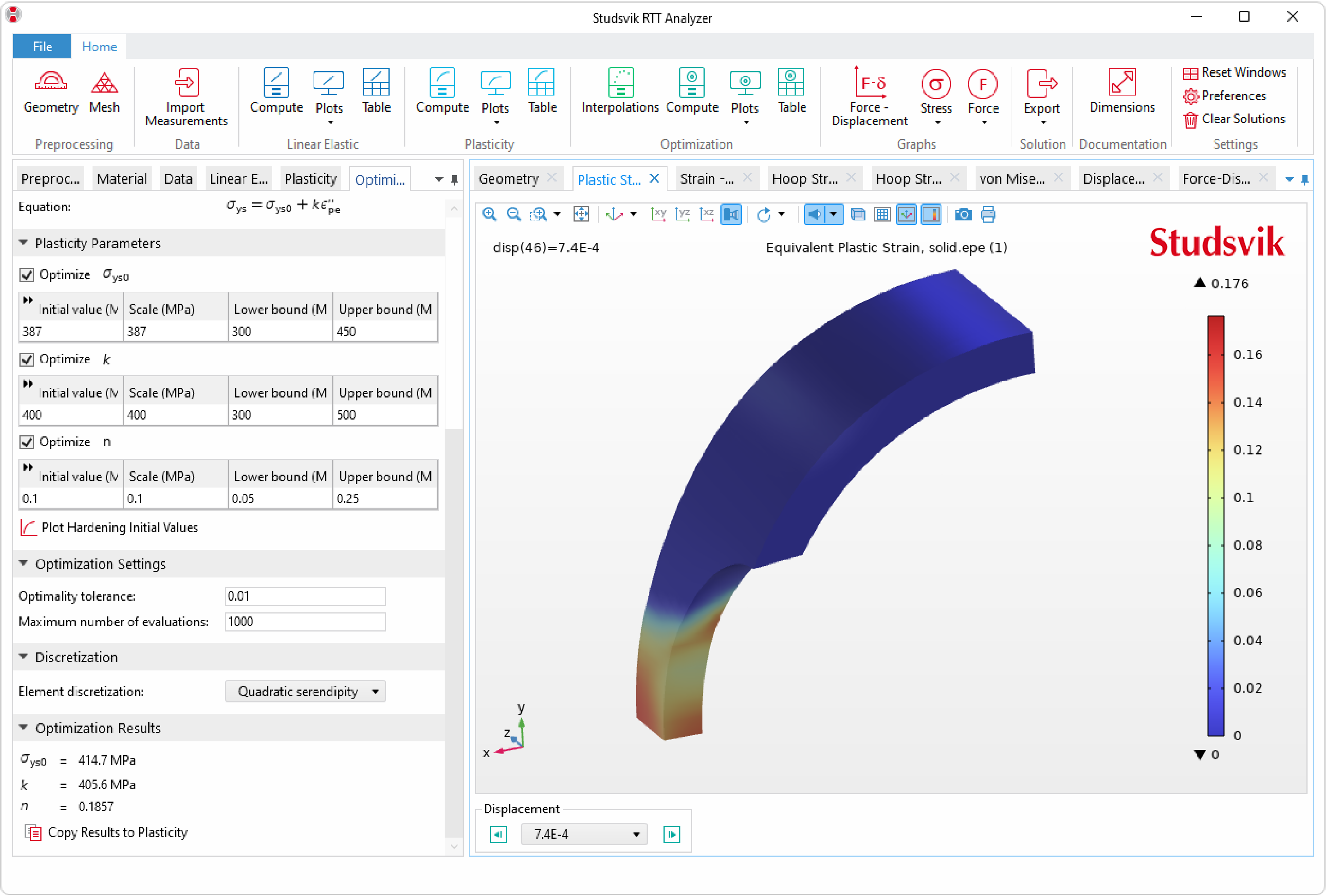


deflexional

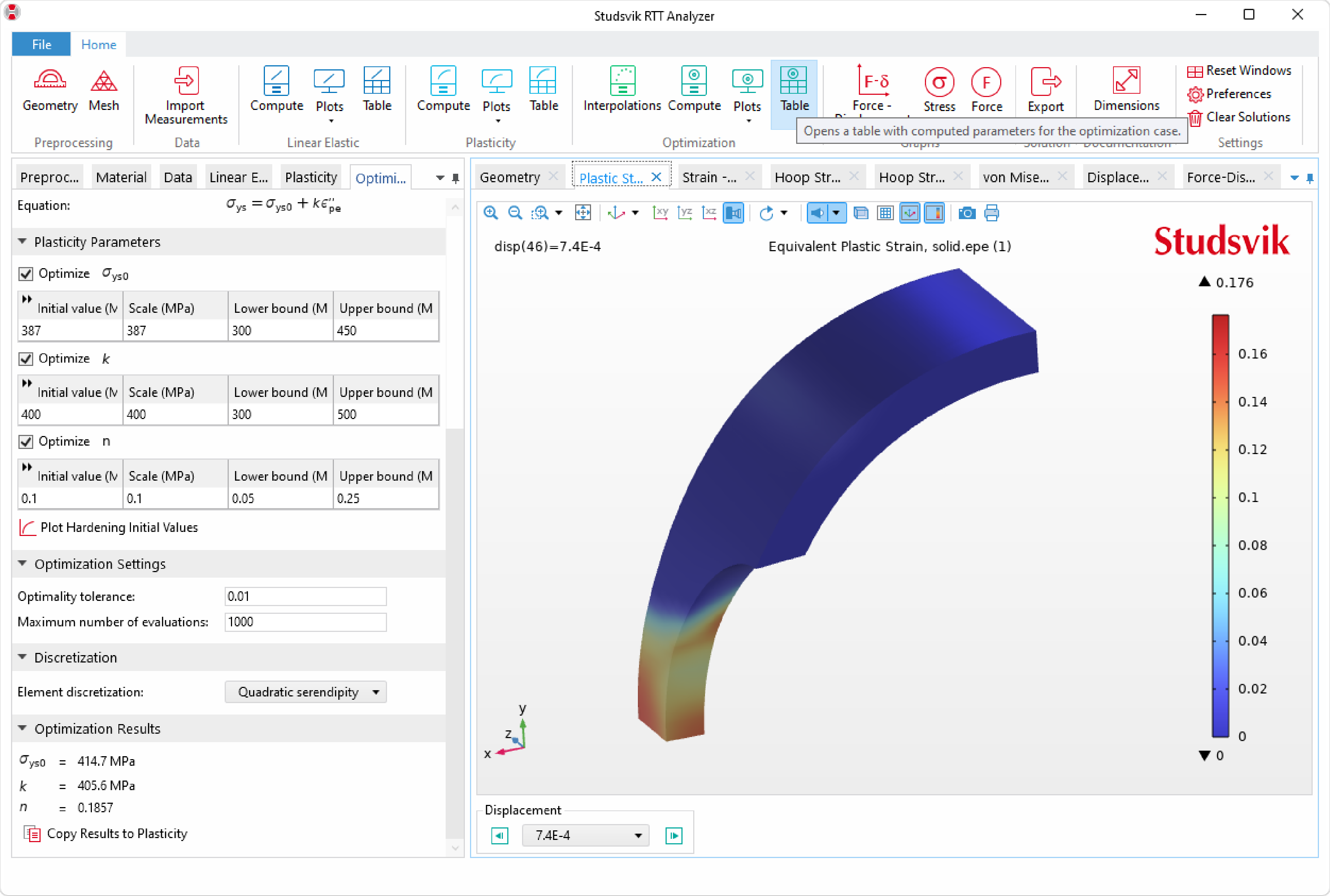
- The *Hoop Strain* at the last step



- The *Plastic Strain* at the last step



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

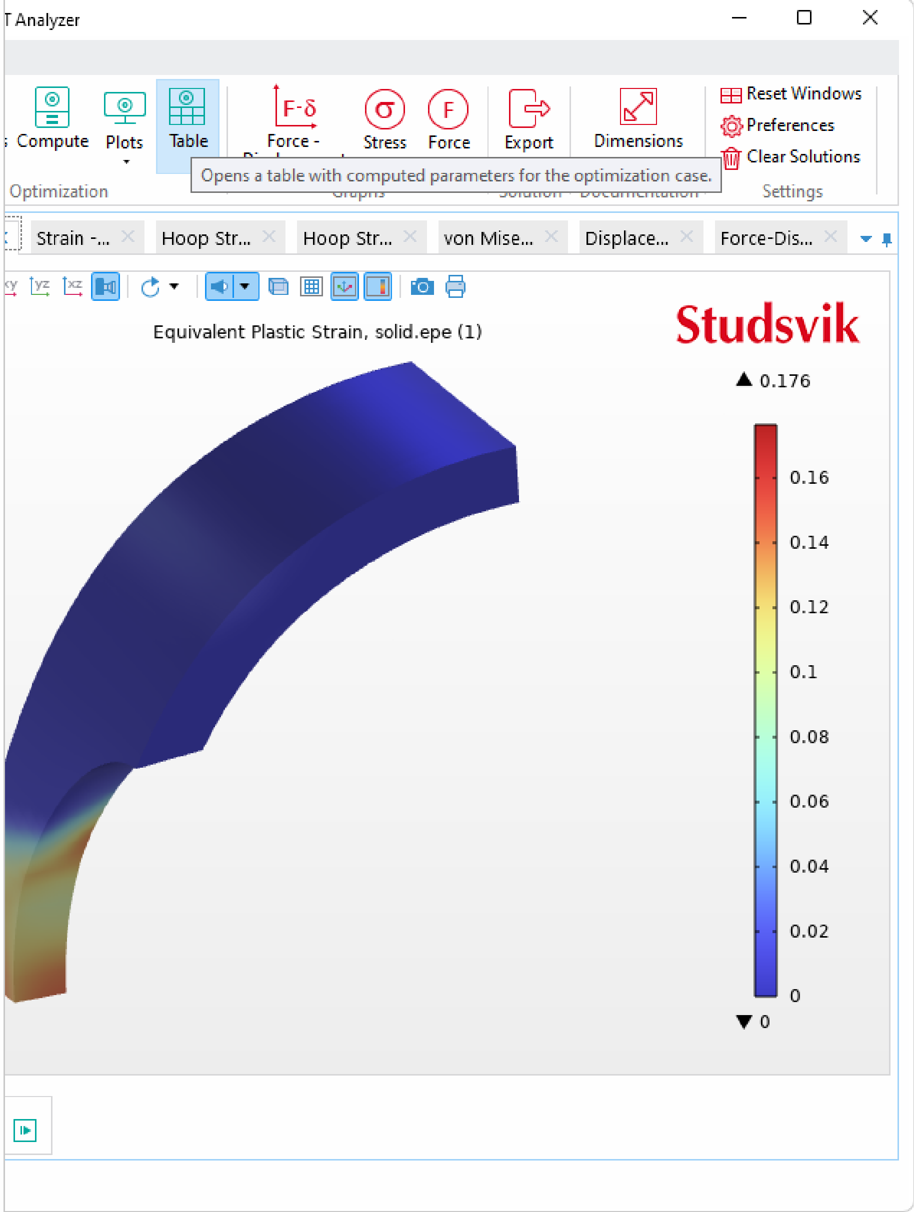


deflexional

Results Table - Optimization

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	2.68
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	8.13
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	14.5
1.2000E-4	0.12000	414.68	405.63	0.18567	71.998	32.467	37.105	32.4
1.4000E-4	0.14000	414.68	405.63	0.18567	155.66	70.195	80.222	70.2
1.6000E-4	0.16000	414.68	405.63	0.18567	318.11	143.45	163.94	143.
1.8000E-4	0.18000	414.68	405.63	0.18567	520.73	234.82	268.36	235.
2.0000E-4	0.20000	414.68	405.63	0.18567	714.14	322.03	368.04	323.
2.0000E-4	0.20000	414.68	405.63	0.18567	714.14	322.03	368.04	323.
2.2000E-4	0.22000	414.68	405.63	0.18567	886.53	399.77	456.88	401.
2.2000E-4	0.22000	414.68	405.63	0.18567	886.53	399.77	456.88	401.
2.4000E-4	0.24000	414.68	405.63	0.18567	1015.5	457.94	523.36	460.
2.4000E-4	0.24000	414.68	405.63	0.18567	1015.5	457.94	523.36	460.
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.
3.0000E-4	0.30000	414.68	405.63	0.18567	1148.6	517.94	591.93	526.
3.2000E-4	0.32000	414.68	405.63	0.18567	1167.2	526.35	601.54	537.
3.4000E-4	0.34000	414.68	405.63	0.18567	1181.7	532.87	608.99	547.
3.6000E-4	0.36000	414.68	405.63	0.18567	1193.3	538.09	614.96	555.
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.
4.0000E-4	0.40000	414.68	405.63	0.18567	1210.6	545.91	623.90	570.
4.2000E-4	0.42000	414.68	405.63	0.18567	1217.1	548.86	627.27	576.
4.2000E-4	0.42000	414.68	405.63	0.18567	1217.1	548.86	627.27	576.
4.4000E-4	0.44000	414.68	405.63	0.18567	1222.6	551.32	630.08	582.
4.4000E-4	0.44000	414.68	405.63	0.18567	1222.6	551.32	630.08	582.
4.6000E-4	0.46000	414.68	405.63	0.18567	1227.1	553.36	632.41	588.
4.8000E-4	0.48000	414.68	405.63	0.18567	1230.9	555.05	634.34	593.
4.8000E-4	0.48000	414.68	405.63	0.18567	1230.9	555.05	634.34	593.
5.0000E-4	0.50000	414.68	405.63	0.18567	1233.9	556.42	635.91	598.
5.2000E-4	0.52000	414.68	405.63	0.18567	1236.4	557.52	637.17	604.
5.4000E-4	0.54000	414.68	405.63	0.18567	1238.2	558.37	638.14	609.
5.6000E-4	0.56000	414.68	405.63	0.18567	1239.6	558.99	638.84	613.
5.8000E-4	0.58000	414.68	405.63	0.18567	1240.5	559.40	639.31	618.
6.0000E-4	0.60000	414.68	405.63	0.18567	1241.0	559.62	639.57	623.
6.2000E-4	0.62000	414.68	405.63	0.18567	1241.1	559.66	639.62	628.
6.2000E-4	0.62000	414.68	405.63	0.18567	1241.1	559.66	639.62	628.
6.4000E-4	0.64000	414.68	405.63	0.18567	1240.8	559.54	639.47	632.
6.6000E-4	0.66000	414.68	405.63	0.18567	1240.2	559.25	639.14	637.
6.8000E-4	0.68000	414.68	405.63	0.18567	1239.2	558.80	638.63	641.
7.0000E-4	0.70000	414.68	405.63	0.18567	1237.9	558.21	637.95	646.
7.2000E-4	0.72000	414.68	405.63	0.18567	1236.3	557.47	637.11	650.
7.4000E-4	0.74000	414.68	405.63	0.18567	1234.3	556.60	636.12	655.

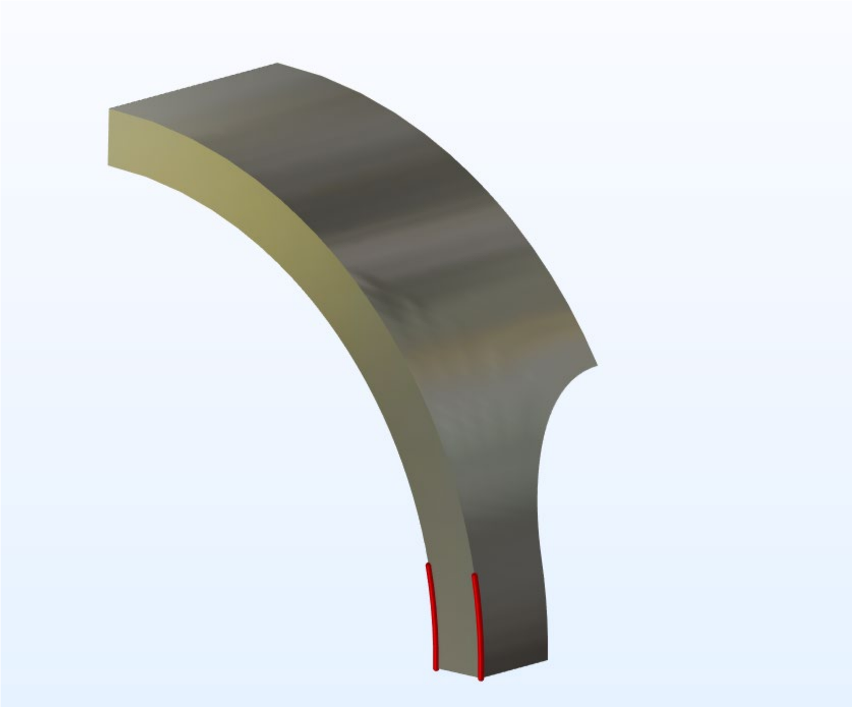
Close



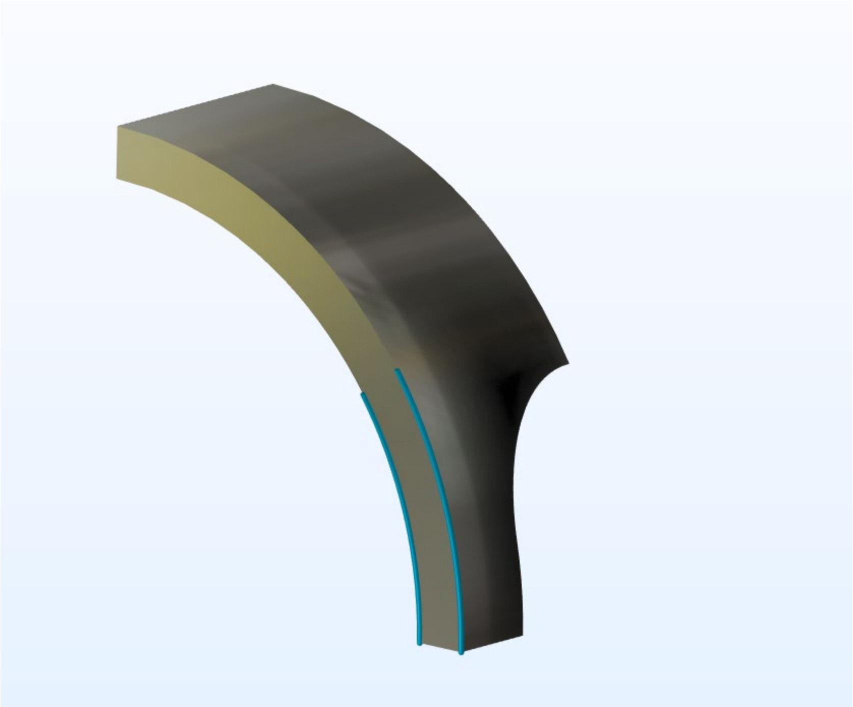
deflexional

Engineering Strain in Gauges

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

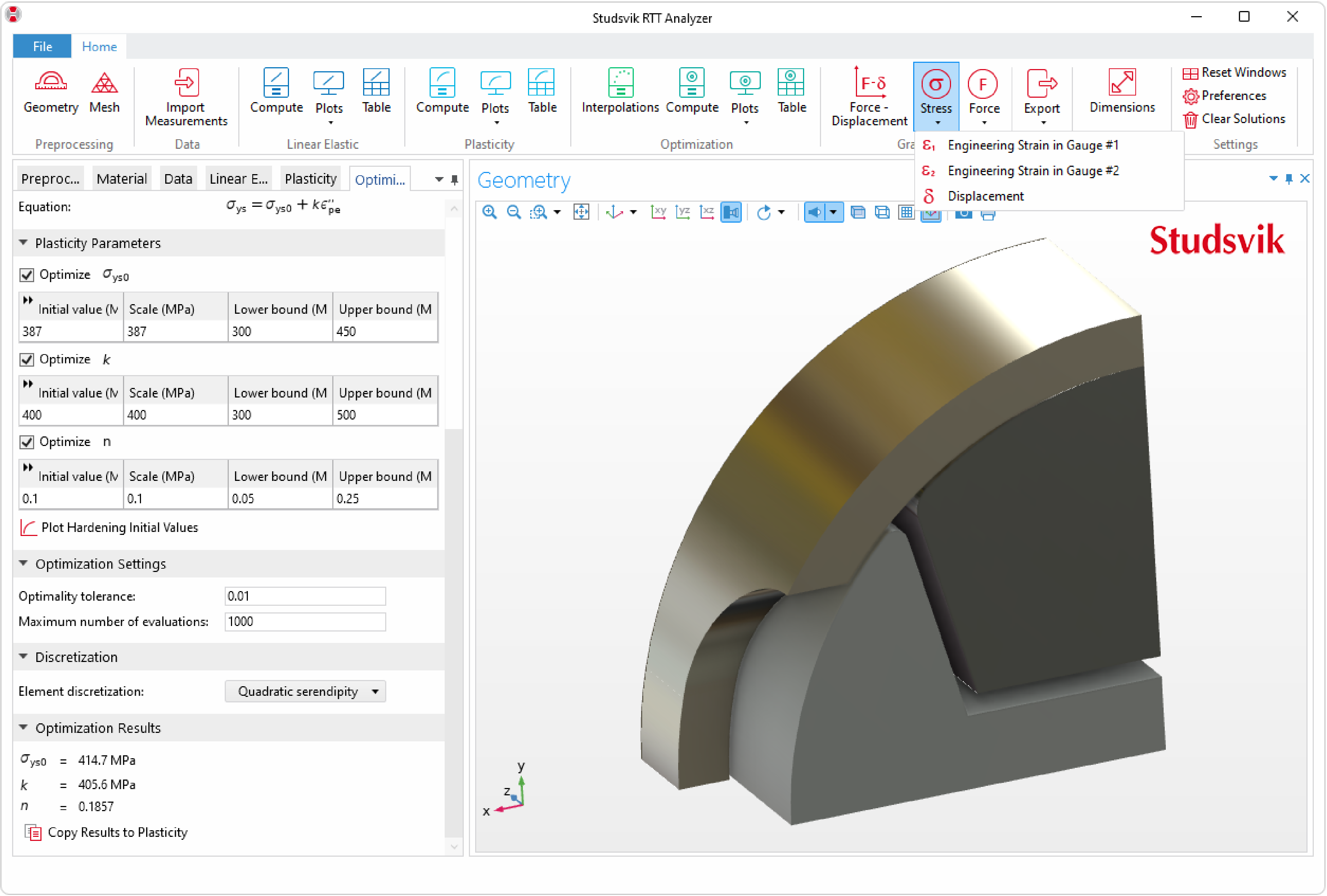


Gauge #1

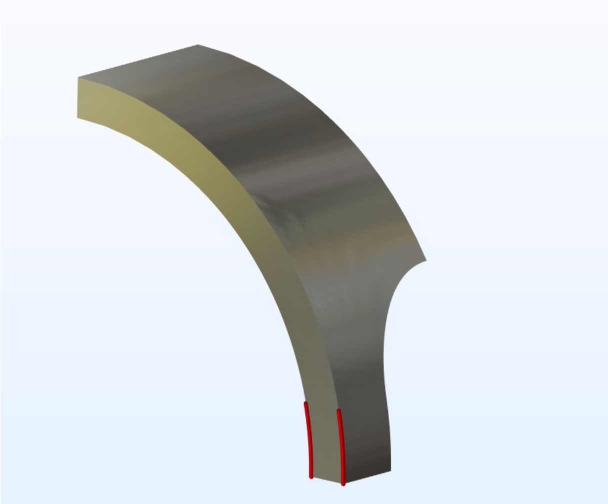


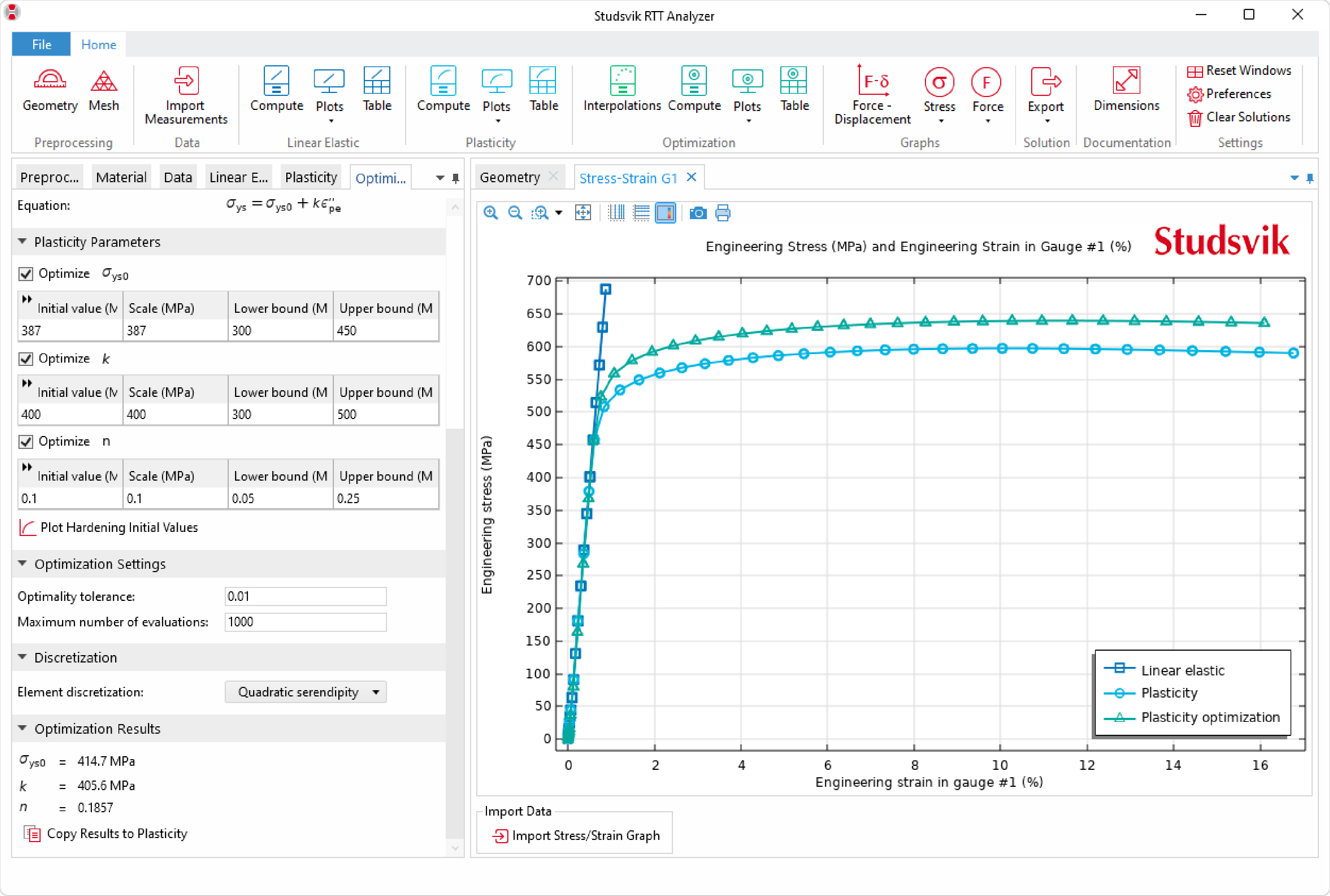
Gauge #2

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



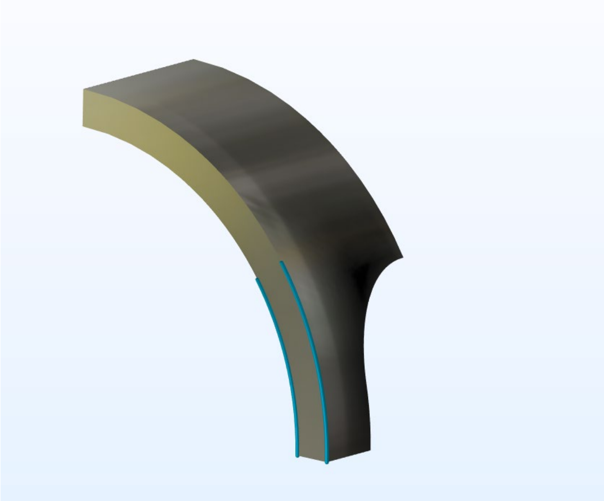
- The *Engineering Stress* versus the *Engineering Strain in Gauge #1* is visualized

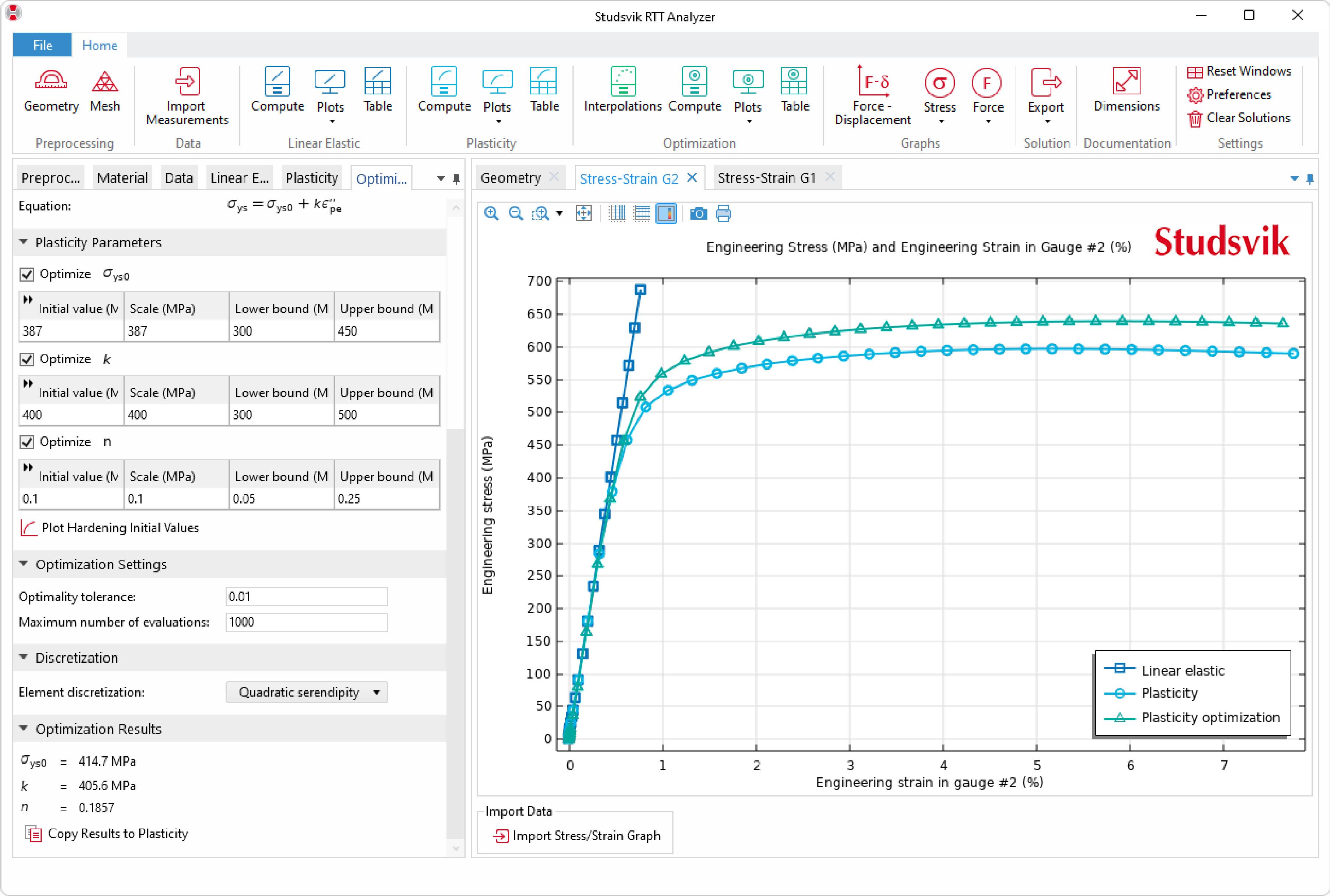




deflexional

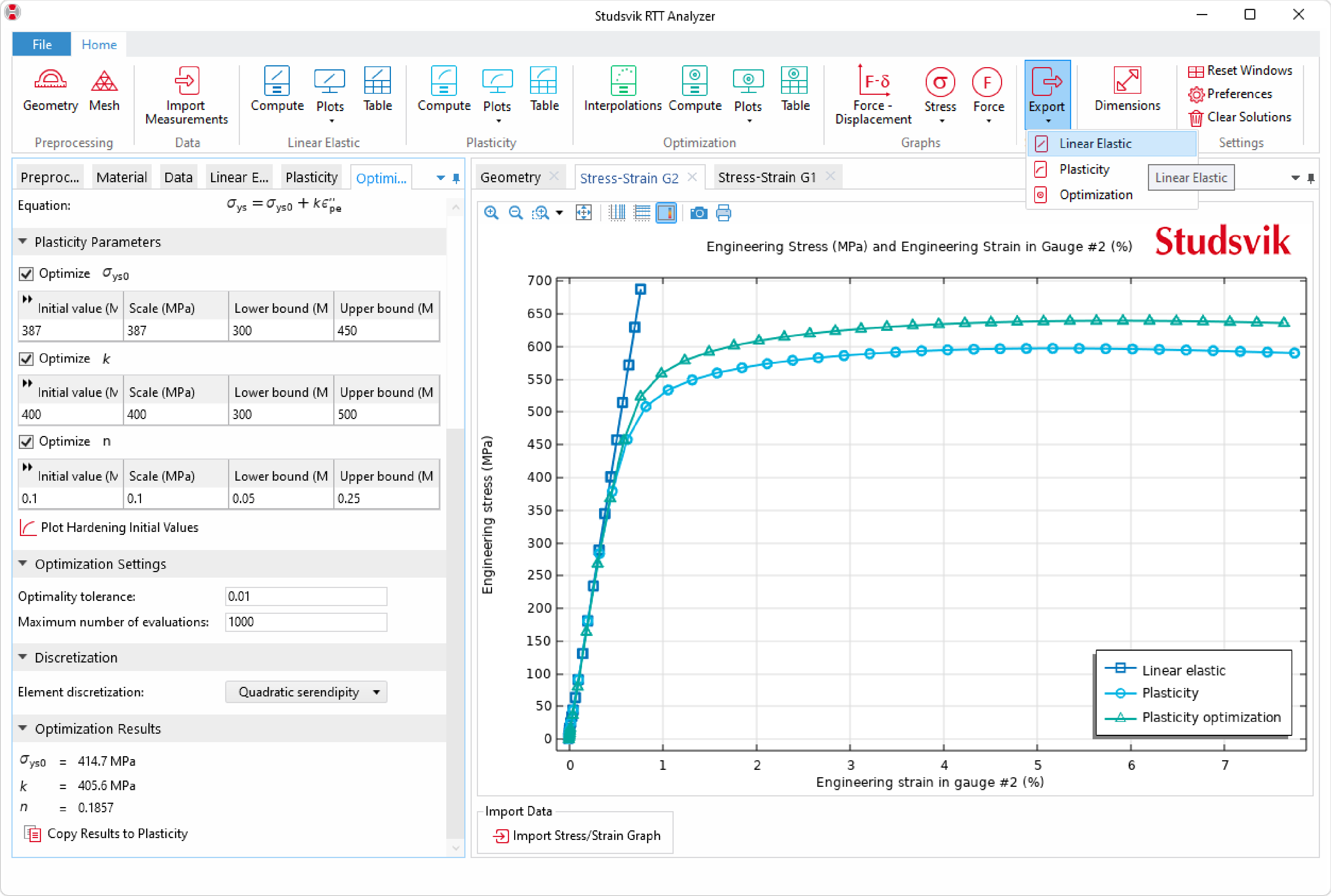
- The *Engineering Stress* versus the *Engineering Strain in Gauge #2* is visualized





deflexional

- Export results data directly to Excel® for further analysis



deflexional

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. 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