Tensile Test Specimen
Outline

1 Description

2 Finite Element Model
   2.1 Geometry
   2.2 Properties
   2.3 Boundary Constraints
   2.4 Loads
   2.5 Mesh

3 Linear Analysis
   3.1 Commands
   3.2 Results

Appendix A Additional Information


1 Description

The purpose of this example is to create and analyze a finite element model of a tensile test specimen made of steel [Fig. 1].

Figure 1: 3D test specimen model

The test specimen is loaded with 100 kN at one end. The other end is fully restrained, i.e., translations in \( X \), \( Y \), and \( Z \) directions are suppressed. The dimensions of the test specimen and the global \( XYZ \) axes that we use for this model are given in Figure 2.

Figure 2: Dimensions of the model [mm]
2 Finite Element Model

For the modeling session we start a new project. We will use quadratic elements.

![New project dialog](https://dianafea.com)

Figure 3: New project dialog
We choose millimeter for the unit length and newton for force.

Geometry browser  ➔ Reference system  ➔ Units  [Fig. 4]
Property Panel  [Fig. 5]

Figure 4: Geometry browser
Figure 5: Property panel - units
2.1 Geometry

First we create a cylinder shape with a radius of 15 mm, the base for the tensile test specimen. Then we create cylinders and blocks which we subtract from the base cylinder to get the geometry of the test specimen.

![Figure 6: Geometry - add cylinder shape (base test specimen)](https://dianafea.com)

![Figure 7: Geometry - add cylinder shape](https://dianafea.com)
We fit all the geometry in workspace.

Figure 8: Geometry view
We copy and translate the cylinder using the array copy functionality.

Figure 9: Geometry - array copy

Figure 10: Geometry view
We create a block shape between the two cylinders.

**Main menu ➔ Geometry ➔ Create ➔ Add block**  [Fig. 11]

**Figure 11: Geometry - Add block**

**Figure 12: Geometry**

We create a block shape between the two cylinders.
We array copy the two cylinders and block to create the shapes that will be subtracted at the bottom of the *Test specimen model.*

**Figure 13:** Geometry - array copy  
**Figure 14:** Geometry view
We subtract all shapes from the *Test Specimen* shape to get the final shape.

**Main menu** → Geometry → Modify → Subtract shapes  
**Main menu** → Viewer → Viewpoints → Isometric view 1  

**Figure 15:** Geometry - Subtract  

**Figure 16:** Geometry
2.2 Properties

We assign material properties to the test specimen. We create a linear elastic material model with steel properties: $E = 207000$ N/mm$^2$ and $\nu = 0.29$. Because this is a 3D model, we do not need to specify geometry properties.

---

**Main menu** ➔ Geometry ➔ Assign ➔ Shape Properties [Fig. 17]

**Property assignments** ➔ Add new material [Fig. 18] [Fig. 19]

---

![Figure 17: Assign properties floor](https://dianafea.com)

![Figure 18: Add new material](https://dianafea.com)

![Figure 19: Linear material properties plate](https://dianafea.com)
2.3 Boundary Constraints

We suppress the translations in the three global directions in the left end face.

Figure 20: Attach supports: left end face

Figure 21: Geometry - supports
2.4 Loads

We apply a tensile force of 140 N/mm² in the X direction.
2.5 Mesh

We define an element size of 2.5 mm to create the mesh.
We generate the mesh of the test specimen.

Figure 26: Finite element mesh
3 Linear Analysis

3.1 Commands

We perform a linear analysis.

Main menu ➔ Analysis ➔ Add analysis ➔ Analysis browser [Fig. 27]
Analysis browser ➔ Analysis1 ➔ Add command ➔ Structural linear static [Fig. 29]
Main menu ➔ Analysis ➔ Run all analyses 📂

Figure 27: Analysis browser
Figure 28: Command menu
Figure 29: Analysis browser
3.2 Results

We check the displacements in the axial direction (X direction) [Fig. 31].

Results browser ➔ Output linear static analysis ➔ Nodal results ➔ Displacement ➔ DtX [Fig. 30]
We check the axial stresses $S_{XX}$ [Fig. 33]. We only view the feature edges of the mesh.

**Results browser** → Output linear static analysis → Element results → Cauchy Total Stresses → $S_{XX}$ [Fig. 32]

**Main menu** → Results → Undeformed mesh feature edges

![Figure 32: Result tree](https://dianafea.com)

![Figure 33: Axial stresses](https://dianafea.com)
We virtually cut away half of the model by adding a clipping plane. The normal of the plane is in the global Y direction (0 1 0). This shows the stress contours in the inside of the model [Fig. 35].

**View setting → show view setting**

**Property panel → Result → Contour plot settings → Clip settings → Add plane**  

**Figure 34: Properties clipping plane**

**Figure 35: Axial stresses in the inside of the model**
A clipping plane view gives a fair impression of the stresses inside of the model. However, the view is a bit disturbed by the contours on the surface of the remaining part of the model. A clearer display is to plot the stresses in a cross-section, in the clipping plane [Fig. 37]. The maximum axial stress in this cross-section is equal to 372 N/mm² (see legend).

**Property panel ➔ Result ➔ Contour plot settings ➔ Clip settings ➔ Slice < ON >**  

**Figure 36: Properties clipping plane**

**Figure 37: Axial stresses in clipping plane**
We add another plane at $Y = 10\,\text{mm}$ and hide the first clipping plane [Fig. 39]. The maximum axial stress is in this cross section is equal to $357\,\text{N/mm}^2$ (see legend).

If we compare it with Figure 37 we can conclude that the stress distribution in the narrow shaft of the specimen is approximately homogeneous.
Appendix A  Additional Information

Folder: Tutorials/TensileSpecimen

Number of elements $\approx 5900$

Keywords:
- ANALYS: linear static.
- CONSTR: suppor.
- ELEMEN: chx60 cpy39 cte30 ctp45 solid.
- LOAD: elemen face force.
- MATERI: elasti isotro.
- OPTION: direct units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy displa extern force green reacti strain stress total.
Disclaimer: The aim of this technical tutorial is to illustrate various tools, modelling techniques and analysis workflows in DIANA. DIANA FEA BV does not accept any responsibility regarding the presented cases, used parameters, and presented results.