Linear Static Analysis of a Stayed Steel Chimney
1 Description

This example illustrates the use of rods and beams in three-dimensional space. Therefore we have chosen the problem of a steel chimney (simple cylinder) stayed by four cables on two thirds of the height as shown in Figure 1. It is assumed that the cables are prestressed in such a way that they will not become slack during any loading.

![Figure 1: Model of the stayed steel chimney](https://dianafea.com)
2 Finite Element Model

The three dimensional finite element model consists of beam elements for the chimney and truss elements for the stays.

Figure 2: New project dialog
We choose millimeter for the unit length, ton for mass and newton for force.

---

**Geometry browser** ➔ Reference system ➔ Units  [Fig. 3]

**Property Panel**  [Fig. 4]
2.1 Geometry

Six lines are required to build the model: one for the bottom part of the chimney, one for the top part and one line for each of the support cables.
Figure 11: View of the model
2.2 Properties

We assign the element class and the material and geometry properties to the vertical line segments. Class-I beam elements are applied and the material is linear elastic.

**Main menu** → **Geometry** → **Assign** → **Shape Properties** [Fig. 12]

**Shape Properties** → **Material** → **Add material** [Fig. 13] → **Edit material** [Fig. 14]
The geometry of the cross-section is specified in Figure 1.

Figure 15: Element geometry properties
We now define the sloped line segments. Regular truss elements are applied and the material is linear elastic. The cross-section of the stays is defined by the area.

**Main menu** → Geometry → Assign → Shape Properties [Fig. 16]
Shape Properties → Material → Add material [Fig. 17] → Edit material [Fig. 18]
Shape Properties → Geometry → Add new geometry [Fig. 19]
2.3 Boundary Conditions

We clamp the base of the chimney by fixing the translations and rotations at the bottom end of the vertical line segment [Fig. 20].

Figure 20: Attach supports - clamping the base

Figure 21: View of model - clamp
Now the cables are pinned to the ground by fixating the translations at the bottom end of the diagonal line segments [Fig. 22].

Figure 22: Attaching supports - fix the cables to the ground

Figure 23: View of model - supports
2.4 Loads

Three load cases are defined: a prestress of 0.1322 N/mm² is applied to the stays, a uniformly distributed wind load of 10 kN/m is applied along the chimney in the positive X direction and a uniformly distributed wind load of 10 kN/m is applied along the chimney in the positive Y direction.

< Repeat for each load case >

Figure 24: Prestress in the stays
Figure 25: Wind load in the X direction
Figure 26: Wind load in the Y direction
2.4.1 Load combinations

Two load combinations are defined: one that is equivalent to load case 3 (wind in the X direction) and another that is a combination of the prestress in the stays and a wind load in the 45° XY direction.

Main menu ➔ Geometry ➔ Loads ➔ Open geometry load combinations table

Figure 28: Geometry browser - load combinations
Figure 29: Geometry load combinations table
2.5 Mesh

The mesh is defined by dividing the line segments in two parts for the chimney top and four parts for the chimney bottom. The truss elements are not divided; each stay segment is meshed with one truss element.

Main menu ➔ Geometry ➔ Assign ➔ Mesh properties

[Fig. 30] [Fig. 31]

Main menu ➔ Geometry ➔ Generate mesh

[Fig. 32]

Figure 30: Mesh properties - Top vertical line segment

Figure 31: Mesh properties - Bottom vertical line segment

Figure 32: Finite element mesh
To check how the profile of the beam and truss elements compare to the structural dimensions, the 3D graphical view of beams and trusses is generated¹.

¹Note: you can also see the cross-section of reduced elements in the geometry view.
3 Analysis

3.1 Commands

We perform a structural linear static analysis.
3.2 Results

The displacements, axial forces and bending moments are presented for both load combinations.

3.2.1 Displacements

![Results browser](image1)

![Results browser](image2)

Figure 37: Results browser

Figure 38: Load combination 1 - displacements Dtx
Results browser ➔ Case ➔ Load combination 2  [Fig. 39]

Results browser ➔ LinSta ➔ Output linear static analysis ➔ Nodal results ➔ Displacements ➔ DtX ➔ Show contours  [Fig. 39] [Fig. 40]

Figure 39: Results browser

Figure 40: Load combination 2 - displacements DtX
3.2.2 Axial forces

Results browser ➔ Case ➔ Load combination 1  [Fig. 41]
Results browser ➔ LinSta ➔ Output linear static analysis ➔ Element results ➔ Cross-section Forces ➔ Nx ➔ Show contours  [Fig. 41]  [Fig. 42]

Figure 41: Results browser

Figure 42: Load combination 1 - axial forces Nx
Linear Static Analysis of a Stayed Steel Chimney | https://dianafea.com

**Figure 43:** Results browser

**Figure 44:** Load combination 2 - axial forces Nx
3.2.3 Bending moments

Results browser ➔ Case ➔ Load combination 1  [Fig. 45]
Results browser ➔ LinSta ➔ Output linear static analysis ➔ Element results ➔ Cross-section Moments ➔ My ➔ Show contours  [Fig. 45]  [Fig. 46]

Figure 45: Results browser

Figure 46: Load combination 1 - bending moment My
Linear Static Analysis of a Stayed Steel Chimney | https://dianafea.com
Appendix A  Additional Information

Folder:  Tutorials/StayedSteelChimney

Number of elements $\approx 10$

Keywords:
  ANALYS:  linear static.
  CONSTR:  suppor.
  ELEMEN:  beam class1 l12be l2tru pipe truss.
  LOAD:  elemen force line prestr.
  MATERI:  elasti isotro.
  OPTION:  direct units.
  POST:  binary ndiana.
  PRE:  dianai.
  RESULT:  cauchy displa extern force green moment 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.