Bending of an Hexagonal Plate
# Outline

1. **Description**  
2. **Finite Element model**  
   - 2.1 Geometry  
   - 2.2 Properties  
   - 2.3 Boundary Conditions  
   - 2.4 Loads  
   - 2.5 Mesh  
3. **Linear Analysis**  
   - 3.1 Commands  
   - 3.2 Results  
     - 3.2.1 Displacements  
     - 3.2.2 Bending Moments  
     - 3.2.3 Reaction Forces  
Appendix A **Additional Information**

## Finite Element model

1. **Geometry**
2. **Properties**
3. **Boundary Conditions**
4. **Loads**
5. **Mesh**

## Linear Analysis

1. **Commands**
2. **Results**
   - 2.1 Displacements
   - 2.2 Bending Moments
   - 2.3 Reaction Forces
1 Description

This example demonstrates the modelling and linear elastic analysis of a plate as shown in Figure 1.

The outer edge of the plate is a regular hexagon with corners on a circle with radius $r_o$ of 10 m. Concentric with the outer edge, the plate has a circular hole with a radius $r_i$ of 4 m. The plate is vertically supported ($u_Z = 0$) at each of the corners along the outer edge.

We assume that the plate is made of concrete with a Young’s modulus $E$ of 22000 MPa, a Poisson’s ratio $\nu$ of 0.2 and a mass density $\rho$ of 2400 kg/m$^3$. The thickness of the plate $t$ is 0.30 m. The loading consists of the deadweight and of a vertical load $q_Z = -20$ kN/m uniformly distributed along the edge of the circular hole.

![Figure 1: Characteristics of the model](https://dianafea.com)
2 Finite Element model

For the modeling session we start a new project for three-dimensional structural analysis [Fig. 2]. The dimensions of the domain are set equal to 100 m. Quadratic quadrilateral finite elements are predominantly used in the analysis.

Figure 2: New project dialog
We use meter, newton and kilogram for length, force and mass, respectively. We chose to use degree for the angles.

Geometry browser ➔ Reference system ➔ Units  [Fig. 3]
Property Panel  [Fig. 4]
2.1 Geometry

Due to symmetry of the geometry, the supports and the loading, it is sufficient to model and analyze only one quarter of the plate. To model the geometry of the plate in DianaIE, we start by creating an arc line [Fig. 5] and a polygonal line with the X, Y, Z coordinates (4, 0, 0), (10, 0, 0), (5, 8.66, 0), (0, 8.66, 0), (0, 4, 0) Figure 6.

---

**Main menu ➔ Geometry ➔ Create ➔ Add arc**  [Fig. 5]

**Main menu ➔ Geometry ➔ Create ➔ Add polyline**  [Fig. 6]

---

Figure 5: Add arc line – *Inner*

Figure 6: Add polyline – *Outer*

Figure 7: Top view of the hexagonal plate perimeter
We create a sheet from the wires. We can do this by right-clicking on the selected wire shapes [Fig. 8 to 11].

**Main menu ➔ Geometry ➔ Modify ➔ Create sheet from wires [Fig. 8] - [Fig. 11]**

< (Or right-click on selected items and select the command) >
2.2 Properties

We assign the material and geometry properties to the plate [Fig. 12]. Therefore we create a new linear elastic material [Fig. 13]. The linear material parameters are $E = 2.2 \times 10^{10}$ N/m$^2$, $\nu = 0.2$ and $\rho = 2400$ kg/m$^3$ [Fig. 14].
We create a new element geometry property. The thickness of the plate is 0.3 m [Fig. 15].

Figure 15: Element geometry properties – Plate
2.3 Boundary Conditions

The plate is vertically supported at each of the corners of the outer edge [Fig. 17].

Figure 16: Attach supports

Figure 17: Top view – points supported in the $Z$ direction
Due to the symmetry conditions, we constrain the rotation along the $X$ axis [Fig. 19] and the rotation along the $Y$ axis [Fig. 21].

Figure 18: Attach support

Figure 19: Supported edge along $X$ axis
Figure 20: Attach support

Figure 21: Supported edge along Y axis
We show the isometric view of the hexagonal plate including the boundary constraints [Fig. 22].

Figure 22: Isometric view of the hexagonal plate
2.4 Loads

We consider two load cases: a gravity load and a distributed load acting downward along the inner edge. We create a load case for the deadweight [Fig. 23].

**Main menu ➔ Geometry ➔ Assign ➔ Add global loads 📈 [Fig. 23]**

![Edit global loads](image)

Figure 23: Global load - Deadweight
We create a distributed force load along the inner edge [Fig. 24 to 25].
2.5 Mesh

We assign the mesh properties to the plate by defining an element size of 0.5 m [Fig. 26].

Figure 26: Set mesh properties
Figure 27: Mesh properties preview
Now, we can generate the finite element mesh [Fig. 28].

**Figure 28:** Finite element mesh

Bending of an Hexagonal Plate | https://dianafea.com
3 Linear Analysis

3.1 Commands

We start by adding a new analysis (that we call Linear) [Fig. 29]. We add to it a Structural linear static command to perform a linear static analysis [Fig. 30 to 31]. We run the analysis.
3.2 Results

3.2.1 Displacements

To start with the assessment of the analysis results we present the contour plot of the vertical displacements $D_{tZ}$ due to the deadweight [Fig. 33].
We present the contour plot of the vertical displacements $D_tZ$ due to the distributed force along the inner edge [Fig. 35].

**Results browser** → Inner edge load → Output linear static analysis → Nodal results → Displacements → $D_tZ$  

**Figure 34:** Results browser  

**Figure 35:** Vertical displacements
3.2.2 Bending Moments

We present the contour plot of the bending moments $M_{xx}$ due to the distributed force along the inner edge [Fig. 37].

Results browser ➔ Inner edge load ➔ Output linear static analysis ➔ Element results ➔ Distributed Moments ➔ $M_{xx}$  [Fig. 36]  [Fig. 37]
We present the contour plot of the bending moments $M_{yy}$ due to the distributed force along the inner edge [Fig. 39].
3.2.3 Reaction Forces

We present the vector plot of the vertical reaction forces FBZ for the inner edge load case [Fig. 41].
Appendix A  Additional Information

Folder: Tutorials/HexagonalPlate

Number of elements ≈ 206

Keywords:
- ANALYS: linear static.
- CONSTR: support.
- ELEMEN: cq24p plate.
- LOAD: edge element force weight.
- MATERI: elasti isotro.
- OPTION: direct units.
- POST: binary ndiana.
- PRE: diana.
- 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.