1 Description

In this example, the buckling behavior of a thin square plate is analyzed. Figure 1 shows that the plate is subjected to a uniform compression load $f$.

![Thin plate under uniform compression loading](image)

Figure 1: Thin plate under uniform compression loading

Beyond a certain load value $f_{crit}$ (i.e., the critical load) the plate becomes unstable and buckles. According to Timoshenko and Gere\(^1\), the critical buckling load for this example is:

$$f_{crit} = \frac{4\pi^2 D}{b^2}$$

with

$$D = \frac{Et^3}{12(1-\nu^2)},$$

where $b$ is the length of the plate, $t$ the thickness, $E$ the Young’s modulus and $\nu$ the Poisson’s ratio. With reference to Figure 1, for the present example we have $f_{crit} = 180.76$ N/mm.

\(^1\)Timoshenko and Gere, *Theory of Elastic Stability*, 1963
2 Modeling Approach

As both membrane stresses and bending stresses are of importance, shell elements are used. No use is made of symmetry conditions and thus the entire domain is subdivided in finite elements. A rather coarse mesh of $4 \times 4$ elements is used and quadratic square elements will be employed. The plate is supported in $Z$ direction along its edges. In-plane displacements in the mid-points of the edges are supported as shown in Figure 2.

Figure 2: Model used in the finite element analysis (the supports in the $X$, $Y$ and $Z$ directions are colored in red, green and blue, respectively)

Note that in spite of the fact that the geometry of the structure is two-dimensional, we should perform a three-dimensional structural analysis. That is because the model shows a phenomenon in the third dimension (i.e., the out-of-plane displacement).
3 Finite Element Model

For the modeling session we start a new project for structural analysis [Fig. 3]. The dimensions of the domain for the three-dimensional model are set equal to 10 m. We will use quadratic quadrilateral finite elements.

Figure 3: New project dialog
We choose millimeter for length, ton for mass, and newton for force.

**Geometry browser**  ➔  Reference system ➔  Units  [Fig. 4]

**Property Panel** [Fig. 5]
3.1 Geometry

We create the *PlateZ* sheet. The coordinates used for this geometry are in Table 1.

---

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

**Main menu ➔ Viewer ➔ Viewpoints ➔ Top view**

---

![Add polygon sheet](image)

**Table 1: Plate coordinates**

<table>
<thead>
<tr>
<th>PlateZ</th>
<th>0</th>
<th>0</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1000</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2000</td>
<td>2000</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1000</td>
<td>2000</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>2000</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

![Figure 6: Add plate sheet](image)

![Figure 7: View of the plate geometry](image)
3.2 Properties

We assign the element class, the material, and geometrical properties to the PlateZ. We use curved shell elements with a thickness of 10 mm [Fig. 1]. The material is assumed isotropic linear elastic and the material properties presented in Section 1 are used.

**Main menu** ➔ Geometry ➔ Assign ➔ Shape Properties [Fig. 8]

Shape Properties ➔ Material ➔ Add material [Fig. 9] ➔ Edit material [Fig. 10]

---

Figure 8: Property assignments to plate

Figure 9: Add new material steel

Figure 10: Edit material steel
Figure 11: Edit geometry *thickness*
3.3 Boundary Conditions

We now apply the boundary conditions shown in Figure 2. We first constrain the displacement in the $X$ direction at the top and bottom mid-nodes, then the displacement in the $Y$ direction at the left and right mid-nodes and, finally, the displacement of the edges in the $Z$ direction.

Figure 12: Constraints in the $X$ direction

Figure 13: Constraints in the $Y$ direction

Figure 14: Constraints in the $Z$ direction
Figure 15: Geometry of the model with constraints
3.4 Loading

We apply a vertical pressure to the PlateZ by assigning a vertical distributed load at the top edge (-100 N/mm) and bottom edges (100 N/mm).

**Main menu → Geometry → Assign → Add loads**  
[Fig. 16] – [Fig. 18]

---

**Figure 16:** Vertical load at top edge

**Figure 17:** Vertical load at bottom edge

**Figure 18:** Top view of the model with the vertical loads
Then, we need to create a load combination that considers the two vertical loads. This is needed during the set-up of the buckling analysis as it will be considered as the buckling load.

**Figure 19:** Load combination table
3.5 Mesh

For the mesh properties we specify an element size of 500 mm. Then, we generate the mesh.

---

**Main menu ➔ Geometry ➔ Assign ➔ Mesh properties [Fig. 20]**

**Main menu ➔ Geometry ➔ Generate mesh [Fig. 21]**

---

**Figure 20: Mesh properties**

**Figure 21: Finite element mesh**
4 Structural Linear Static Analysis

4.1 Commands

We first perform a linear elastic analysis for a quick assessment of the model.

Main menu → Analysis → Add analysis
Analysis browser → Analysis1 → Rename → Linear [Fig. 22]
Main menu → Analysis → Run all analyses

Figure 22: Analysis browser

Figure 23: Analysis browser
4.2 Results

Here we compare the vertical displacement of the top of the PlateZ obtained numerically with DianaIE and analytically from the equation:

\[ u_y = \frac{f_b}{2EI} = 0.05 \text{ mm}. \]

Figure 25 shows that the numerical result obtained with DianaIE is the same as the analytical solution.
5 Structural Stability Analysis

5.1 Commands

Now, we perform a structural stability analysis to derive the buckling load for the *PlateZ*.

[Fig. 26] Analysis browser → Linear → Add command → Structural stability

[Fig. 27] Analysis browser → Linear → Structural stability → Eigenvalue analysis → Define stability analysis → Edit properties

---

**Analysis browser**

- Linear
- Add command
- Structural stability

**Properties**

- Linearized buckling eigenvalue analysis
- Load set: Geometry load combination 1
- Incorporate initial displacements
- Incorporate pre-buckling nonlinearity

---

**Figure 26: Analysis browser**

**Figure 27: Buckling load**
5.2 Results

We create a contour plot of the total displacement of the buckled PlateZ.

Figure 29 shows that the buckling load obtained from DianaIE (1.7989 · $f = 179.89$ N/mm) is in good agreement with the theoretical value of 180.76 N/mm calculated in Section 1.
Appendix A  Additional Information

Folder: Tutorials/BucklingPlate

Number of elements \( \approx 16 \)

Keywords:
- ANALYS: euler linear stabil static.
- CONSTR: suppor.
- ELEMEN: cq40s curved shell.
- LOAD: edge elemen force.
- MATERI: elasti isstro.
- OPTION: direct units.
- POST: binary ndiana.
- PRE: diana1.
- RESULT: buckli cauchy displa extern force green modes moment reacti strain stress total values.

References:

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.