Structural Nonlinear Analysis of a Shear Wall Panel
1 Description

This example presents the modeling of the shear wall panel in Figure 1. The shear wall has a central panel, two columns on the side and a loading plate on the top where a vertical pressure \( q \) and an horizontal load \( u \) are applied. The structure is modeled using plane stress elements and adopting the total strain based rotating crack model as constitutive behavior for the concrete central panel and side columns. A linear elastic material model is used for the modeling of the top loading plate.

First we check the model using a linear elastic analysis. Then, we perform a nonlinear analysis with a monotonic increase of the horizontal load \( u \). In addition to the visualization of the contour plot of displacement, stress and strain fields, we display in DianaFE the load-displacement curve associated to this problem.

![Figure 1: Shear wall panel with grid reinforcement](https://dianafea.com)
The geometry and the layout of the reinforcements are shown in Figure 2. The steel bars are modeled using grid reinforcement layers. The concrete and steel material properties are in listed in Table 1.

![Figure 2: Geometry and reinforcement layout of the shear wall panel (dimensions in mm)](image)

<table>
<thead>
<tr>
<th>Linear concrete</th>
<th>Nonlinear concrete</th>
<th>Reinforcement steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>30000 N/mm²</td>
<td>30E3 N/mm²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.15</td>
<td>0.15</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>2.2 N/mm²</td>
<td></td>
</tr>
<tr>
<td>Ultimate strain</td>
<td>0.0028</td>
<td></td>
</tr>
<tr>
<td>Compressive strength</td>
<td>27.5 N/mm²</td>
<td></td>
</tr>
<tr>
<td>Yielding stress</td>
<td>574 N/mm²</td>
<td></td>
</tr>
<tr>
<td>Ultimate strength</td>
<td>8574 N/mm²</td>
<td></td>
</tr>
<tr>
<td>Ultimate strain $\varepsilon_{su}$</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

Table 1: Concrete and steel properties
2 Finite Element Model

For the modeling session we start a new project for structural analysis [Fig. 3]. The dimensions of the domain for the 2D model are set to 10 m. We use quadratic quadrilateral finite elements.

Main menu → File → New  [Fig. 3]

Figure 3: New project dialog
We choose millimeter for the unit Length, ton for mass and newton for Force.
2.1 Geometry

We create a sheet for the panel, a second one for the left column and a third one for the loading plate.
We create the *right column* by copying and translating the *left column* shape.
The reinforcements are modelled as 2D layers with the same geometry of the embedding shapes. Therefore, we can generate the reinforcement layers by duplicating the panel, left column and right column sheets.

**Main menu ➔ Geometry ➔ Shapes ➔ panel ➔ Duplicate**

< Repeat for left column and right column >

**Geometry browser ➔ Geometry ➔ Shapes ➔ panel 1 ➔ Rename ➔ grid reinforcement panel**

< Repeat for left column 1 and right column 1 >

**Figure 14: Duplicate shapes**

**Figure 15: Geometry browser**

**Figure 16: Geometry browser**
2.2 Properties

We assign the element class and the material and geometrical properties to the *panel*. We use regular plane stress finite elements with a thickness of 100 mm as shown in Figure 2. The material is nonlinear as described in Section 1.

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

Shape Properties ➔ ➔ Material ➔ Add material ➔ [Fig. 18] ➔ Edit material ➔ [Fig. 19] – [Fig. 21]

Shape Properties ➔ ➔ Geometry ➔ Add new geometry ➔ [Fig. 22]

---

**Figure 17:** Property assignments to *panel*

**Figure 18:** Add new material *nonlinear concrete*

**Figure 19:** Edit material *nonlinear concrete*
Figure 20: Edit material *nonlinear concrete*

Figure 21: Edit material *nonlinear concrete*

Figure 22: Edit *panel geometry*
We assign the properties to the left column and the right column. As shown in Figure 2, the thickness is 400 mm. Here, we do not need to add a new material, since we assign to the left column and the right column the same material used for the panel.

Figure 23: Property assignments to left column and right column

Figure 24: Edit left column and right column geometry
We assign the properties to the *loading plate*. We create a new material for the linear concrete as described in Section 1. As shown in Figure 2, the thickness is 700 mm.

---

**Main menu** → Geometry → Assign → Shape Properties [Fig. 25]

**Shape Properties** → Material → Add material [Fig. 26] → Edit material [Fig. 27]

**Shape Properties** → Geometry → Add new geometry [Fig. 28]
We assign the properties to the steel reinforcements. We start with the grid reinforcement panel. We need to specify the strain-stress diagram at yielding based on the material properties as described in Table 1.

**Main menu** ➔ Geometry ➔ Assign ➔ Reinforcement properties  
Reinforcement properties ➔ Material ➔ Add material  
Reinforcement properties ➔ Geometry ➔ Edit geometry

*Select the correspondent reinforcement in the Geometry browser >*

**Figure 29**: Property assignments to grid reinforcement panel

**Figure 30**: Add new material steel

**Figure 31**: Edit material steel
Figure 32: Edit material *steel* (strain-stress diagram at yielding)

Figure 33: Edit *grid reinforcement panel* geometry
We assign the properties to the grid reinforcement left column and the grid reinforcement right column. Comparing to the grid reinforcement panel, only the geometry changes while the material properties are the same.

DianaIE

<Select the correspondent reinforcement in the Geometry browser >

Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties 📊 [Fig. 34] [Fig. 35]

Reinforcement properties ➔ Geometry ➔ Edit geometry 📊 [Fig. 36]

Figure 34: Property assignments to grid reinforcement left column

Figure 35: Property assignments to grid reinforcement right column

Figure 36: Edit grid reinforcement left column and grid reinforcement right column geometry
2.3 Boundary Conditions

We support the bottom edge of the model in the $X$ and $Y$ directions.

Main menu ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 37] [Fig. 38]
2.4 Loads

2.4.1 Vertical Load

We apply a vertical distributed load of -366.95 N/mm to the top edge of the loading plate.

Figure 39: Apply vertical distributed force

Figure 40: Vertical distributed load
2.4.2 Prescribed Deformation

To apply a prescribed deformation in the center node of the loading plate, we need to create a node at this location. We first create a vertex at the center of the loading plate, that we later imprint on the loading plate.

Main menu → Geometry → Create → Add point ' + ' [Fig. 41]
Main menu → Geometry → Modify → Imprint shapes 📘 [Fig. 42] [Fig. 43]

Figure 41: Create point body
Figure 42: Imprint point body on loading plate
Figure 43: Imprinted point on loading plate
To apply a prescribed displacement in a node in DianaIE, we first constrain the displacement of the node and then apply a prescribed deformation.

Main menu ➔ Geometry ➔ Assign ➔ Add supports [Fig. 44]
Main menu ➔ Geometry ➔ Assign ➔ Add loads [Fig. 45] [Fig. 46]

Figure 44: Constrain center node of loading plate
Figure 45: Apply prescribed deformation
Figure 46: View prescribed deformation
2.5 Mesh

We set the mesh properties such that the elements size is 50 mm. We generate the mesh.

**Figure 47: Mesh properties**

**Figure 48: Finite element mesh**
3 Structural Linear Static Analysis

3.1 Commands

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

![Analysis window](image1)

![Add command](image2)

![Analysis tree](image3)
3.2 Results

3.2.1 Total Displacements

First we create a contour plot for the total displacements $\text{Dt}_{\text{XYZ}}$ for the two load cases (top pressure and displacement).
3.2.2 Vertical Stresses

We display the contour plot of the vertical stresses $S_{YY}$. The positive stresses induced by the displacement are higher than the tensile strength of the concrete. Therefore, in the nonlinear analysis we will observe cracks forming in the panel and in the left and right column.
4 Structural Nonlinear Analysis

4.1 Commands

We perform a nonlinear structural analysis in which the horizontal displacement is incrementally increased.

Main menu → Analysis → Add analysis
Analysis browser → Analysis2 → Rename → Nonlinear [Fig. 58]
Analysis browser → Nonlinear → Add command → Structural nonlinear [Fig. 59] [Fig. 60]
The default execute block is used to apply the vertical load on the structure. We rename this execute block as *top pressure*. Then we add a new execute block to the nonlinear analysis to take into account the horizontal displacement. We rename this second execute block as *displacement*. 

**Figure 61:** Rename execute block

**Figure 62:** Add new execute block

**Figure 63:** Rename execute block
The horizontal displacement is applied 30 times. During these steps we set the maximum number of iterations to 50 and the convergence norm to energy with a tolerance equal to 0.001.

**Analysis browser** → Nonlinear → Structural nonlinear → displacement → Load steps → Edit properties [Fig. 64]

**Analysis browser** → Nonlinear → Structural nonlinear → displacement → Equilibrium iteration → Edit properties [Fig. 65] [Fig. 66]

---

**Figure 64:** Load steps

**Figure 65:** Equilibrium iteration

**Figure 66:** Convergence norm
We select the desired output results listed in Table 2 and we run the analysis.

**Analysis browser** → Nonlinear → Structural nonlinear → Output → Edit properties [Fig. 67]

Properties - OUTPUT → Result → User selection → Modify [Fig. 68] [Table 2] [Fig. 69]

Main menu → Analysis → Run selected analysis

---

**Table 2: Required output data**

<table>
<thead>
<tr>
<th>Output Item</th>
<th>Required Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Global displacements</td>
<td>DISPLA TOTAL TRANSL GLOBAL</td>
</tr>
<tr>
<td>Global total stresses</td>
<td>STRESS TOTAL CAUCHY GLOBAL</td>
</tr>
<tr>
<td>Global plastic strain</td>
<td>STRAIN PLASTI GREEN GLOBAL</td>
</tr>
<tr>
<td>Crack width</td>
<td>STRAIN CRKWDT GREEN PRINCI</td>
</tr>
<tr>
<td>Reaction forces</td>
<td>FORCE REACTI TRANSI GLOBAL</td>
</tr>
</tbody>
</table>

**Figure 67: Output properties**

**Figure 68: Results selection**

**Figure 69: Output properties**
4.2 Results

4.2.1 In-Plane Principal Stresses

We create a contour plot for the in-plane principal stresses at load step 2 and 31.

Already at step 2, some of the concrete in the panel already cracks since the tensile exceeds the tensile strength (see Table 1). At step 31, also the compressive stresses reached the tensile strength of the concrete.
4.2.2 Reinforcement Plastic Strains

We display the contour plot for the reinforcement plastic strains $Ep_{YY}$ at load step 2 and 31. We define the limits of the contour plot to -0.06 and 0.04 as minimum and maximum values, respectively.

At step 2 the plastic strains are nearly zero in the entire wall, thus the steel reinforcements are not yielded. However, as shown for step 31, due to the increasing of the horizontal displacement the reinforcements yield.

---

**Figure 73:** Results browser

**Figure 74:** Results properties

**Figure 75:** Reinforcement plastic strains $Ep_{YY}$ - load-step 2

**Figure 76:** Reinforcement plastic strain $Ep_{YY}$ - load-step 31
4.2.3 Crack Widths

We display the contour plot for the crack widths Ecw1 at load step 2 and 31. We set a maximum of crack width to 2.6 mm. As shown in Figure 79, the formation of cracks in the concrete starts from the bottom left corner due to the tension induced by the horizontal load. By increasing the load value, the crack widths increases, especially in the left column where the tension is higher (see Figure 80).

Results browser ➔ Nonlinear ➔ Output ➔ Element results ➔ Crack-widths ➔ Ecw1 [Fig. 77]
Property Panel ➔ Contour plot settings ➔ Specified values ➔ Minimum value ➔ 0 [Fig. 78]
Property Panel ➔ Contour plot settings ➔ Specified values ➔ Maximum value ➔ 2.6 [Fig. 78]
Results browser ➔ Case ➔ Load-step 2, Load-factor 1.0000, displacement [Fig. 79]
Results browser ➔ Case ➔ Load-step 31, Load-factor 30.000, displacement [Fig. 80]

Figure 77: Results browser
Figure 78: Results properties
Figure 79: Crack width Ecw1 - load-step 2
Figure 80: Crack width Ecw1 - load-step 31
4.2.4 Load-Displacement Curve

Finally, we make a load-displacement curve. For this model, the load is actually a prescribed displacement at the center of the loading plate. Therefore, a load-displacement diagram should plot the reaction force versus the displacement of this center node.

Figure 83 shows that the elastic threshold is reached as soon as the horizontal displacement is applied. Then, the stiffness of the wall decreases due to the formation of cracks in the concrete, until a constant slope is reached since the load is undertaken by the hardening steel reinforcements.
Appendix A  Additional Information

Folder: Tutorials/ShearWall

Number of elements ≈ 700

Keywords:
- ANALYS: nonlin physic.
- CONSTR: suppor.
- ELEMEN: cq16m grid pstres reinfo taper.
- LOAD: deform edge elemen force.
- MATERI: consta crack elasti harden isotro linear multil plasti rotati soften strain totstr vonmis.
- OPTION: direct newton regula units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy crkwdt displa force green plasti princ 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.