Shear Failure in Reinforced Concrete Beam
Outline

1 Description 3

2 Finite Element Model 5
   2.1 Geometry 7
   2.2 Properties 18
   2.3 Boundary Conditions 25
   2.4 Loads 27
   2.5 Mesh 29

3 Structural Nonlinear Analysis 30
   3.1 Commands 30
   3.2 Results 34

Appendix A Additional Information 38
1 Description

The publications from Kuchma et al. (1997)\(^1\) and Collins and Kuchma (1999)\(^2\) illustrated that traditional design methods are limited in the assessment of the shear resistance of longitudinal reinforced concrete beams. This tutorial presents the simulation of one of the experiments addressed in these publications (experiment SE50A-45/SE50A-45R). It is a four-point bending test of a reinforced concrete beam failing in shear [Fig. 1].

The beam is 5 m long with a rectangular cross section with 500 mm height and 169 mm width. The beam has two loading points at the top and two support points at the bottom. There are sixteen reinforcement bars in the beam (bars with diameter of 16 mm, which corresponds to a cross section area of 200 mm\(^2\)).

![Diagram of the beam test](https://dianafea.com)

**Figure 1: Characteristics of the beam test**

We model the experiment with plane stress elements for the concrete and the loading and support plates and embedded reinforcements for the longitudinal rebars. We perform a nonlinear analysis with incremental load until failure. For the sake of simplicity and speed of the analysis presented in this tutorial, we use a coarse mesh and less strict iterative settings. In this manner we can illustrate how to model this benchmark in a relatively fast manner\(^3\).

---

\(^1\)Kuchma et al., *The Influence of Concrete Strength, Distribution of Longitudinal Reinforcement, and Member Size, on the Shear Strength of Reinforced Concrete Beams*, 1997

\(^2\)Collins and Kuchma, *How Safe Are Our Large, Lightly Reinforced Concrete Beams, Slabs and Footings?*, 1999

\(^3\)A more refined nonlinear analysis until failure is presented in the Verification Report that is part of the User’s Manual for comparison with the experimental results for verification of DIANA calculations.
According to the reference Kuchma et al. (1997)[1] the concrete properties are \( f_{cm} = 53 \text{ N/mm}^2 \) with an aggregate size of 10 mm. We define the other concrete material parameters - tensile strength \( f_{ct} \), Young modulus \( E_c \) and fracture energy \( G_f \) - using the expressions from CEB-FIP Model Code 1990\(^4\). We use the average values for the tensile strength \( f_{ctm} \) and Young’s modulus \( E_c \) to simulate this experiment.

We assume a rotating total strain crack model for the concrete, with the Hordijk model for the tensile curve and the Thorenfeldt model for the compressive curve. The material properties are listed in Table 1. The crack bandwidth is mesh dependent and is automatically defined in DIANAIE.

For the steel reinforcement and the steel plates, we assume a linear elastic material model with the properties listed in Table 1.

<table>
<thead>
<tr>
<th>Table 1: Material properties</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Concrete</strong></td>
</tr>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>Compressive strength</td>
</tr>
<tr>
<td>Tensile strength</td>
</tr>
<tr>
<td>Fracture energy in tension</td>
</tr>
<tr>
<td><strong>Steel</strong></td>
</tr>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
</tr>
</tbody>
</table>

\(^4\)CEB-FIP, CEB-FIP Model Code 1990, 1993
## 2 Finite Element Model

For the modelling session we start a new project for two-dimensional structural model.

![Figure 2: New project dialog](https://dianafea.com)`

---

**Main menu ➔ File ➔ New** [Fig. 2]
We define the units to be used in the model [Fig. 4]: newton for force and millimeter for length.

Figure 3: Geometry browser

Figure 4: Property panel - units
2.1 Geometry

We now create the geometry. We define the beam, the loading plates and the supporting plates. We start with the beam.

Main menu → Geometry → Create → Polygon sheet → [Fig. 5] [Fig. 6]

Figure 5: Add polygon sheet - Beam

Figure 6: Geometry view - Beam
We now create the left load plate.

**Main menu ➔ Geometry ➔ Create ➔ Polygon sheet**  

![Polygon sheet](https://dianafea.com)

**Figure 7: Add polygon sheet - left load plate**

<table>
<thead>
<tr>
<th>X [mm]</th>
<th>Y [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>150</td>
</tr>
<tr>
<td>2</td>
<td>250</td>
</tr>
<tr>
<td>3</td>
<td>250</td>
</tr>
<tr>
<td>4</td>
<td>200</td>
</tr>
<tr>
<td>5</td>
<td>150</td>
</tr>
</tbody>
</table>

**Figure 8: Geometry view - left load plate**
We now create the right load plate.
We now create the left support plate.

**Main menu** ➔ **Geometry** ➔ **Create** ➔ **Polygon sheet**  📷 [Fig. 11] [Fig. 12]

**Figure 11**: Add polygon sheet - left support plate

**Figure 12**: Geometry view - left support plate
We finally create the right support plate.

**Main menu → Geometry → Create → Polygon sheet  📸 [Fig. 13]  [Fig. 14]**

![Polygon sheet interface](image)

**Figure 13: Add polygon sheet - right support plate**

**Figure 14: Geometry view - right support plate**
We now define the geometry of the reinforcement bars as showed in the following slides.

Main menu ➔ Geometry ➔ Create ➔ Line ➔ [Fig. 15] [Fig. 17]
Shear Failure in Reinforced Concrete Beam

Figure 19: Add line - bar top 1

Figure 20: Add line - bar top 2

Figure 21: Geometry view - bar top 1

Figure 22: Geometry view - bar top 2
Figure 23: Add line - short bar down 1

Figure 24: Add line - short bar down 2

Figure 25: Geometry view - short bar down 1

Figure 26: Geometry view - short bar down 2
We add a reinforcement set and place all the line reinforcements there.

**Main menu ➔ Geometry ➔ Create ➔ Add reinforcement set [Fig. 31]**

<Move all the lines inside the reinforcement set >

**Figure 31: Geometry browser**
And the geometry is complete.

Figure 32: Geometry of the beam test
2.2 Properties

We assign the properties to the beam. We use total strain crack model with the parameters listed in Table 1. We use regular plate stress elements. In the geometry properties we define the thickness as 169 mm.
Shape properties ➔ Geometry ➔ Add new geometry

[Fig. 39]

Figure 39: Add geometry - Beam
We assign the properties to the steel plates. We use linear elastic isotropic material with the parameters listed in Table 1. We use regular plane stress elements. In the geometry properties we define the thickness as 169 mm.
Shape properties ➔ Geometry ➔ Add new geometry [Fig. 43]

Figure 43: Add geometry - Steel plates
We now define the properties of the reinforcements. We use embedded reinforcement with cross section of $2 \times 200 \text{ mm}^2 = 400 \text{ mm}^2$ (see Figure 1) with the Young’s modulus described in Table 1.
Figure 47: Add geometry - Reinforcement
2.3 Boundary Conditions

We fix the horizontal and vertical displacements at the middle point of the left support plate.

![Edit supports dialog box](Fig. 48)

**Figure 48**: Attach support - left plate

![Supports view](Fig. 49)

**Figure 49**: View of the model - supports
We fix the vertical displacements at the middle point of the right support plate.

Figure 50: Attach support - right plate

Figure 51: View of the model - supports
2.4 Loads

We apply a nodal force of -1000 N in the vertical direction at the middle point of the left loading plate.

![Edit loads](image)

**Figure 52:** Attach load - left plate

**Figure 53:** View of the model - loads
We apply a nodal force of -2000 N in the vertical direction at the middle point of the right loading plate.

Figure 54: Attach load - right plate

Figure 55: View of the model - loads
2.5 Mesh

We define the mesh properties. We use an average element size of 50 mm. Finally we generate the mesh.

Figure 56: Mesh properties

Figure 57: Finite element mesh
3 Structural Nonlinear Analysis

3.1 Commands

We set up a nonlinear analysis.

Main menu → Analysis → Add analysis [Fig. 58]
Analysis browser → Analysis1 → Add command → Structural nonlinear [Fig. 59] [Fig. 60]
We apply the load incrementally until failure. For that we define 10 steps with a factor of 5 followed by 105 steps with a factor of 1. We use the arc length control option with respect to the vertical displacements of the two loading points. The arc length control method will allow to capture the post-peak response of the beam after failure.

**Figure 61:** Execute load properties

**Figure 62:** Arc length settings

**Figure 63:** Regular arc length control settings
In the equilibrium iteration we use a line search method and we choose the energy convergence norm. We change the settings of the energy convergence norm from *terminate* to *continue*. We also increase the energy norm to 0.005 and set the maximum number of iterations to 20.5

These settings are less strict than the default ones and we use them to have a fast analysis that is good enough to illustrate the problem presented in this tutorial. A more precise nonlinear analysis is presented in the *Verification Report* that is part of the *User’s Manual* for comparison with the experimental results for verification of DIANA calculations.
In the output we choose the results of displacements, crack strains and crack widths. Finally we run the analysis.

**Analysis browser** → Analysis1 → Structural nonlinear → Output → Edit properties 📜 [Fig. 67] – [Fig. 69]

**Main menu** → Analysis → Run selected analysis 📜

---

[Fig. 67]: Analysis browser

[Fig. 68]: Output properties

[Fig. 69]: Output properties - user selection
3.2 Results

We present the load versus displacement graph for the loading node in the right side.

Figure 70: Chart view - load vs. displacement graph
The experimental failure load reported in the Kuchma et al. (1997)\cite{1} and Collins and Kuchma (1999)\cite{2} is 68.6 kN in the first test and 80.5 kN in the second test. We copy the data from the graph in Figure 70 to excel and add the two experimental loads for comparison in Figure 71. We can observe that the numerical results are in agreement with the observed failure loads.

![Load vs. displacement graph](https://dianafea.com)

Figure 71: Load vs. displacement graph

In the next slides we show the crack strains at three loading points: just after the start of the crack localization (for a load level approximately equal to 61.3kN, which corresponds to the first local decrease of load), at the maximum loading (approximately 78.8 kN) and at the last load step already in post-peak regime.
The crack strains presented in Figure 72 to Figure 74 for the different load steps clearly show the initial bending cracks, resulting in a slight loss of stiffness, followed by the crack localization around load-step 29, resulting in a local snap-back in the force versus displacement curve (see Figure 71). Additional loading can be carried after this initial crack localization with increasing opening of the bending cracks, until bending cracks transform into shear cracks (diagonally oriented), which lead to a definite decrease of the load carrying capacity of the beam.

Figure 72: Crack strains for load step 29 (load ≈ 61.3 kN)
Figure 73: Crack strains for load step 98 (load ≈ 78.8 kN)

Figure 74: Crack strains for load step 120 (last load step)
Appendix A  Additional Information

Folder: Tutorials/ShearConcreteBeam

Number of elements $\approx 1000$

Keywords:
- ANALYS: nonlin physic.
- CONSTR: suppor.
- ELEMEN: bar cq16m pstres reinfo.
- LOAD: force node.
- MATERI: crack elasti harden hordyk isotro rotati soften thoren totstr.
- OPTION: arclen direct newton normal regula select units update.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: crack crkwdt displa green princi strain total.

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.