Bending Test on a Fiber-Reinforced Concrete Notched Beam
Outline

1 Description
   1.1 Experimental Test .................................................. 4
   1.2 Steel Fiber Material Characterization .............................. 5
      1.2.1 Material Parameters ............................................. 6
   1.3 Model Approach .................................................... 7

2 Finite Element Model .................................................. 8
   2.1 Geometry .......................................................... 10
   2.2 Properties ......................................................... 15
   2.3 Boundary Conditions ............................................... 22
   2.4 Loads ............................................................. 25
   2.5 Mesh ............................................................. 26

3 Structural Nonlinear Analysis ........................................ 29
   3.1 Commands .......................................................... 29
   3.2 Results .......................................................... 33
      3.2.1 Displacements ..................................................... 33
      3.2.2 Force vs. Displacement Plot .................................... 34
      3.2.3 Crack Widths ..................................................... 35
      3.2.4 SFRC Tensile Stress vs. CMOD Curve ......................... 36
      3.2.5 Comparison with fib Constitutive Tensile Stress–CMOD Curve .................................................. 38

Appendix A Additional Information ................................. 39
1 Description

This tutorial describes how to model Steel Fiber-Reinforced Concrete (SFRC) elements with DIANA. The example involves the simulation of a three-point bending test of a concrete prism based on the typical setup for the characterization of SFRC materials according to the European Standard EN 14651 (2005). The three-point bending test of a notched beam [Fig. 1] is used for the determination of the material parameters. For a SFRC material, the tensile behavior is described in terms of residual flexural tensile strength determined from the load vs. crack mouth opening displacement curve or load versus deflection curve provided by the bending test.

Figure 1: Notched beam for SFRC bending test

---

1 CEN. *Test method for metallic fibered concrete – Measuring the flexural tensile strength (limit of proportionality (LOP), residual)*. 2005
1.1 Experimental Test

The geometry of the SFRC specimen is shown in Figure 2. The beam has a span of 500 mm, with an extension of 25 mm on both sides of the supports, with a total length of 550 mm. The beam has a cross-section of 150 × 150 mm. A notch is produced in the lower center area of the specimen for a depth of 25 mm.

Figure 2: Beam layout and dimensions (mm)
1.2 Steel Fiber Material Characterization

The mechanical parameters to be used as input of the SFRC material model in DIANA can be determined through the results of laboratory tests. The three-point bending test provides the load-deflection curve, which can be used to obtain the corresponding stress-crack width curve. From the test, the diagram of the applied force (F) versus the deformation can be produced, in which the deformation is expressed in terms of crack mouth opening (CMOD), as shown in Figure 3.

![Typical F-CMOD curve](https://dianafea.com)

The strength parameters are described by the residual flexural tensile strength, \( f_{R,j} \), which is determined from the F-CMOD as:

\[
f_{R,j} = \frac{3F_jl}{2bh^2_{sp}}
\]

where \( f_{R,j} \) and \( F_j \) are, respectively, the residual flexural tensile strength and the load corresponding to a given CMOD value, \( l \) is the span length, \( b \) is the specimen width, \( h_{sp} \) is the distance between the notch tip and the top of the specimen.

The SFRC concrete can be classified based on the values of the residual flexural tensile strength significant for serviceability, \( f_{R1k} \), and ultimate, \( f_{R3k} \), conditions. The SFRC class (for example 3c) is defined by a number representing the strength interval for \( f_{R1k} \), and a letter (a, b, c, d, or e), representing the \( f_{R3k}/f_{R1k} \) ratio. Based on the class, the stress-strain curve (or the stress-CMOD curve), characterizing the uniaxial tensile behavior of the SFRC material, can be defined according to the FIB Model Code 2010\(^2\) for steel fiber reinforced concrete (see Figure 4).

---

\(^2\)fib, fib Model Code for Concrete Structures 2010, 2013

Bending Test on a Fiber-Reinforced Concrete Notched Beam | [https://dianafea.com](https://dianafea.com)
1.2.1 Material Parameters

In this example we assume a SFRC class 3c, with a reference concrete class C30 for plain concrete. The class 3c indicates a $f_{R1k}$ strength interval between 3.0 and 4.0 N/mm$^2$, and a residual strength ratio of $0.9 \leq f_{R3k}/f_{R1k} < 1.1$.

The point B (see Figure 4) corresponds to the peak value of the tensile stress of the SFRC material, before the stress reduction, and it is assumed based on the characteristic value of the tensile stress, $f_{ct}$, of the plain concrete. For the point A, a tensile stress value equal to $0.9 \cdot f_{ct}$ is considered.

The post-peak behavior of the constitutive law is defined based on a linear model, as shown in Figure 5, where $f_{Fts}$ represents the serviceability residual strength and $f_{Ftu}$ represents the ultimate residual strength. These values define the portion of the constitutive diagram between points D and E (Figure 4).

Point C, defining the drop of the tensile stress after the peak value, is determined from the intersection between the softening branch of the plain concrete curve (namely the Hordijk curve defined by the value of tensile fracture energy, $G_f$) and the extrapolated branch of the SFRC model defined by the points D and E.

The values of the characteristic points defining the tensile constitutive law for the SFRC material are summarized in Table 1. The corresponding curve, compared to the plain concrete one, is presented in Figure 6.

<table>
<thead>
<tr>
<th>Point (Figure 4)</th>
<th>Tensile stress (N/mm$^2$)</th>
<th>CMOD (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1.82</td>
<td>0.00082</td>
</tr>
<tr>
<td>B</td>
<td>2.03</td>
<td>0.00225</td>
</tr>
<tr>
<td>C</td>
<td>1.49</td>
<td>0.025</td>
</tr>
<tr>
<td>D</td>
<td>1.35</td>
<td>0.5</td>
</tr>
<tr>
<td>E</td>
<td>0.75</td>
<td>2.5</td>
</tr>
<tr>
<td>F</td>
<td>0</td>
<td>8</td>
</tr>
</tbody>
</table>

Table 1: Tensile stress and CMOD values at characteristic points

Figure 5: Definition of linear post-cracking constitutive relationship

Figure 6: Constitutive law in tension for SFRC material
1.3 Model Approach

A two-dimensional finite element model, representing the middle plane of the concrete beam, is built. The beam’s supporting blocks and the load-transfer block are modeled as surfaces with linear-elastic material properties. Interfaces are placed between these blocks and the concrete beam.

The following aspects will be considered in the model:

- the concrete beam is modeled as two-dimensional
- quadratic mesh order is assumed
- the linear-elastic material properties for concrete, steel and interfaces are reported in Table 2
- for the SFRC material, the fib fiber reinforced concrete model is used for the input of the tensile curve
- a nonlinear static analysis is performed in order to describe the full load–deflection curve
- the analysis is carried out based on displacement control
- displacements and crack-width are shown after the analysis
- the tensile stress vs. CMOD curve provided by DIANA is compared with the FIB Model Code curve

<table>
<thead>
<tr>
<th>Table 2: Material properties</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td>Young’s modulus $E$</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$ (–)</td>
</tr>
<tr>
<td>Mass density $\rho$</td>
</tr>
<tr>
<td>Normal stiffness $k_n$</td>
</tr>
<tr>
<td>Shear stiffness $k_s$</td>
</tr>
</tbody>
</table>
2 Finite Element Model

For the modeling session we start a new project [Fig. 7]. We chose to work on a 2D model and we set the dimensions of the domain equal to 10 m. Quadratic quadrilateral finite elements are predominantly used in the analysis.

Figure 7: New project dialog
We use millimeter and newton for length and force units, respectively.

**Geometry browser** → Reference system → Units  [Fig. 8]

**Property Panel**  [Fig. 9]

---

**Figure 8:** Geometry browser

**Figure 9:** Property panel - units
2.1 Geometry

To model the geometry of the beam in DIANAIE, we start by creating a sheet and we name it *concrete* [Fig. 11]. We specify the coordinates of the points from 1 to 4 as shown in Figure 10. The two-dimensional model is situated in the $XY$ plane, i.e., the $Z$ coordinates are zero.

---

**Main menu** → **Geometry** → **Create** → **Add polygon sheet** 📊 [Fig. 10]

**Main menu** → **Viewer** → **Viewpoints** → **Top view** 📊

**Main menu** → **Viewer** → **Fit all** 📊

---

**Figure 10**: Add sheet – *concrete*

**Figure 11**: View of the model – *concrete*
We add now two sheets in the middle part of the concrete specimen [Fig. 14]. We call the first one crack with the coordinates shown in Figure 12. We use the second one to create the notched area of the specimen with the coordinates shown in Figure 13 and we name it notch.
We imprint now the sheets crack and notch on the sheet concrete. In this operation we do not keep the tools [Fig. 15]. Afterwards, we can remove the surface corresponding to the notch in the concrete specimen [Fig. 16].

**Figure 15:** Subtract shapes  
**Figure 16:** View of the model
We add a sheet for the loading block which is used to transfer the load to the specimen. We name the sheet as *block* and we input the coordinates shown in Figure 17.

**Figure 17: Add sheet – block**

**Figure 18: View of the model – block**
We create two sheets for the left and right support with the coordinates presented in Figure 19 and in Figure 20, respectively. For each sheet, we include an extra point in the center of the lower edge in order to assign the supports. The XYZ coordinates of sheet sup\(L\) are (in mm): (23, -4, 0), (25, -4, 0), (27, -4, 0), (27, 0, 0), (23, 0, 0). The XYZ coordinates of sheet sup\(R\) are (in mm): (523, -4, 0), (525, -4, 0), (527, -4, 0), (527, 0, 0), (523, 0, 0).
2.2 Properties

We assign the material and geometry properties to the concrete beam [Fig. 22]. We use SFRC class 3c and we define the mechanical parameters according to the fib Model Code 2010 for steel fiber reinforced material. The reference concrete class for plain concrete is assumed as C30. We create a total strain based crack material and we name it Concrete_FRC [Fig. 23]. The linear material parameters are $E = 33550.6$ N/mm$^2$ and $\nu = 0.15$ [Fig. 24].

DianaIE

---

**Figure 22:** Property assignments

**Figure 23:** Add new material – Concrete_FRC

**Figure 24:** Edit new material – Concrete_FRC
We choose the rotating option for the total strain crack model [Fig. 25].

For the tensile behavior, we choose the fiber reinforced concrete tensile curve based on the crack mouth opening (CMOD) value. The curve is defined in Section 1.2.1.

For the compressive behavior, we assume a parabolic curve [Fig. 27].

Figure 25: Edit material – Concrete_FRC

Figure 26: Tensile behaviour – Concrete_FRC

Figure 27: Compressive behaviour – Concrete_FRC
We create a new element geometry and we name it *thick*. We explicitly define the orientation of the local \( x \) axis [Fig. 29].

**Figure 28:** Property assignments

**Figure 29:** Add new geometry – *thick*
We also define a linear elastic material for the steel supports and loading blocks [Fig. 31]. The Young’s modulus is $E = 200000 \text{ N/mm}^2$ [Fig. 32]. We assign the same geometry property used for the concrete specimen.

**Main menu** → **Geometry** → **Assign** → **Shape Properties** [Fig. 30]

**Shape Properties** [Fig. 31] → **Material** → **Add material** [Fig. 31] → **Edit material** [Fig. 32]
We create now the interfaces between the support and loading blocks and the concrete beam. We name the connection as \textit{int} [Fig. 33].

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig33.png}
\caption{Connection property assignments}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig34.png}
\caption{View of the model – \textit{int}}
\end{figure}
We define a new linear elastic material for the interfaces as shown in Figure 35 and Figure 36. We create a new element geometry property for the interfaces and we name it *interface*. 

**Figure 35:** Add new material – *interface*

**Figure 36:** Edit new material – *interface*
We create a new element geometry property for the interfaces and we name it *interface* [Fig. 37] [Fig. 38].

---

**Figure 37: Add new geometry – interface**

**Figure 38: Edit geometry – interface**
2.3 Boundary Conditions

The translation of the lower edge’s mid point of the left support is supported in the Y direction [Fig. 39] [Fig. 40].

Figure 39: Attach support – supL
Figure 40: Translational supports
The translation of the lower edge’s mid point of the right support is supported in the $X$ and $Y$ directions [Fig. 41] [Fig. 42].

Figure 41: Attach support – $supR$

Figure 42: Translational supports
The vertical load is applied through a prescribed deformation, therefore we attach a support in the $Y$ direction to the top edge of the loading block [Fig. 43].

**Figure 43: Attach support – sup_block**

**Figure 44: Translational supports**
2.4 Loads

We create a new load case including a prescribed deformation attached to the top edge of the loading block and we name it *Vertical*. The displacement acts in the $Y$ direction [Fig. 45] [Fig. 46].

![Main menu ➔ Geometry ➔ Assign ➔ Add loads](https://dianafea.com)

Figure 45: Attach load – *Vertical*  
Figure 46: Prescribed deformation
2.5 Mesh

We define the mesh properties of the model with a desired element size of 15 mm [Fig. 47]. The selected shapes and the mesh seeding preview can be seen in Figure 49.
We assign an element size of 4 mm to the central part of the concrete specimen [Fig. 50]. The selected edges and the mesh seeding preview can be seen in Figure 52.

Figure 50: Set mesh properties

Figure 51: Selected edges

Figure 52: Mesh properties preview
Now, we can generate the finite element mesh.

Figure 53: Finite element mesh
3 Structural Nonlinear Analysis

3.1 Commands
We start by adding a new analysis (that we call *Bending Test*) [Fig. 54]. We add to it a *Structural nonlinear* command to perform a nonlinear analysis [Fig. 55] [Fig. 56].
We choose the load set and define the step sizes: \([ 0.01(5) \ 0.001(150) \ 0.005(200) \ 0.01(300) ]\) [Fig. 58].

**Analysis browser** ➔ *Bending Test* ➔ *Structural nonlinear* ➔ new execute block ➔ *Load steps* ➔ Edit properties [Fig. 57] [Fig. 58]

Figure 57: Analysis browser

Figure 58: Load steps
We assume a maximum number of 20 iterations and ask to satisfy both the energy and force convergence norms [Fig. 59]. We choose to continue the analysis if convergence is not achieved [Fig. 60] [Fig. 61].
For the *Bending Test* analysis we chose the user selection output [Fig. 62] and we add the results listed in Table 3. Finally, we run the analysis.

<table>
<thead>
<tr>
<th>Analysis browser</th>
<th>Bending Test</th>
<th>Structural nonlinear</th>
<th>Output</th>
<th>Edit properties</th>
<th>[Fig. 62] [Fig. 63]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Main menu</td>
<td>Analysis</td>
<td>Run selected analysis</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Table 3: Required output data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total displacements</td>
<td>DISPLA TOTAL TRANSL GLOBAL –</td>
</tr>
<tr>
<td>Reaction forces</td>
<td>FORCE REACTI TRANS GLOBAL –</td>
</tr>
<tr>
<td>External forces</td>
<td>FORCE EXTERN TRANS GLOBAL –</td>
</tr>
<tr>
<td>Total stresses</td>
<td>STRESS TOTAL CAUCHY GLOBAL Nodes</td>
</tr>
<tr>
<td></td>
<td>STRESS TOTAL CAUCHY PRINCI</td>
</tr>
<tr>
<td></td>
<td>STRESS TOTAL CAUCHY LOCAL</td>
</tr>
<tr>
<td>Total strains</td>
<td>STRAIN TOTAL GREEN GLOBAL Nodes</td>
</tr>
<tr>
<td></td>
<td>STRAIN TOTAL GREEN PRINCI</td>
</tr>
<tr>
<td></td>
<td>STRAIN TOTAL GREEN LOCAL</td>
</tr>
<tr>
<td>Crack widths</td>
<td>STRAIN CRKWDT GREEN PRINCI</td>
</tr>
<tr>
<td>Crack status</td>
<td>STATUS CRACK</td>
</tr>
</tbody>
</table>

### Figure 62: Output properties

### Figure 63: Results selection
3.2 Results

3.2.1 Displacements

We select the last step to assess the results at the end of the nonlinear analysis. We show the contour plot of the vertical displacements $\text{TDtY}$. 

Figure 64: Results browser

Figure 65: Displacements
3.2.2 Force vs. Displacement Plot

We select the top nodes of the loading block [Fig. 66] and we ask to display the vertical reaction force value $F_{BY}$ for each load step.

Figure 66: Node selection

Figure 67: Chart view - reaction forces vs. displacements
3.2.3 Crack Widths

We show the contour plot of the crack widths Ecw1 for the last step of the analysis.
3.2.4 SFRC Tensile Stress vs. CMOD Curve

In order to plot the curve of the tensile stress vs. the CMOD representative of the SFRC specimen, we first select a node and an element in the notched area [Fig. 70] and we ask to display the crack width value Ecw1 for each load step [Fig. 71].
We then ask to display the local stress value $S_{xx}$ for each load step [Fig. 73].

Figure 72: Element selection

Figure 73: Chart view: local stress diagram
3.2.5 Comparison with *fib* Constitutive Tensile Stress–CMOD Curve

We can copy the stress $S_{xx}$ and crack-width $E_{cw1}$ values to an Excel spreadsheet and plot the tensile stress–CMOD curve provided by the DIANA calculation. In the following graph a comparison with the tensile stress–CMOD curve provided by the *fib* Model Code 2010 for SFRC material is presented [Fig. 74].

![Comparison between DIANA and fib Model Code ft–CMOD curves](https://dianafea.com)
Appendix A  Additional Information

Folder: Tutorials/NotchedBeam

Number of elements ≈ 1950

Keywords:
  ANALYS: nonlin physic.
  CLASS: large.
  CONSTT: suppor.
  ELEMEN: cl12i cq16m ct12m interf pstres struct.
  LOAD: deform.
  MATERI: crack elasti frcon harden isoto parabo rotati soften totstr.
  OPTION: direct newton regula units.
  POST: binary ndiana.
  PRE: dianai.
  RESULT: cauchy crack crkwdt displa extern force green princ reacti status strain stress total.

References:


© DIANA FEA BV

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.