Linear Elastic Analysis of a Deep Beam with Web Opening
# Outline

## 1 Description

## 2 Finite Element Model

<table>
<thead>
<tr>
<th>Subsection</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1 Geometry</td>
<td>6</td>
</tr>
<tr>
<td>2.2 Properties</td>
<td>11</td>
</tr>
<tr>
<td>2.3 Boundary Conditions</td>
<td>13</td>
</tr>
<tr>
<td>2.4 Loads</td>
<td>15</td>
</tr>
<tr>
<td>2.5 Mesh</td>
<td>16</td>
</tr>
</tbody>
</table>

## 3 Structural linear static analysis

<table>
<thead>
<tr>
<th>Subsection</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1 Commands</td>
<td>17</td>
</tr>
<tr>
<td>3.2 Results</td>
<td>18</td>
</tr>
<tr>
<td>3.2.1 Reaction Forces</td>
<td>18</td>
</tr>
<tr>
<td>3.2.2 Total Displacements</td>
<td>19</td>
</tr>
<tr>
<td>3.2.3 Stress Field SXX</td>
<td>21</td>
</tr>
<tr>
<td>3.2.4 Stress Field SYY</td>
<td>23</td>
</tr>
<tr>
<td>3.2.5 Stress field SXY</td>
<td>24</td>
</tr>
<tr>
<td>3.2.6 In-Plane Principal Components – Stress Field SXX</td>
<td>25</td>
</tr>
<tr>
<td>3.2.7 Diagram – Stress Field SXX</td>
<td>26</td>
</tr>
<tr>
<td>3.2.8 Diagram – Stress Fields SYY and SXY</td>
<td>28</td>
</tr>
</tbody>
</table>

### Appendix A Additional Information

<table>
<thead>
<tr>
<th>Subsection</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Appendix A Additional Information</td>
<td>29</td>
</tr>
</tbody>
</table>
1 Description

We analyze a concrete continuous deep beam with web openings presented in (Ashour and Rishi) \(^1\) and here shown in Figure 1. The beam is 3 meters long, 625 mm height and 120 mm thick. The web opening is of 250 × 250 mm. Due to symmetry we model one half of the beam. Supports and loading plate are made of steel.

The material properties are listed in Table 1.

<table>
<thead>
<tr>
<th></th>
<th>Concrete</th>
<th>Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>25000</td>
<td>210000 N/mm²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.2</td>
<td>0.2</td>
</tr>
</tbody>
</table>

The nodal force \(P\) is equal to \(4e+05\) N.

We will study the elastic response of the beam by performing a linear elastic static analysis.

---

\(^1\) Ashour and Rishi, *Tests of Reinforced Concrete Continuous Deep Beams with Web Openings*, 2000
2 Finite Element Model

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

Figure 2: New project dialog
We choose millimeter for the unit length, ton for mass, newton for Force.

---

**Geometry browser** ➔ Reference system ➔ Units  [Fig. 3]

**Property Panel**  [Fig. 4]

---

**Figure 3: Geometry browser**

**Figure 4: Property panel - units**

---

Linear Elastic Analysis of a Deep Beam with Web Opening | [https://dianafea.com](https://dianafea.com)
2.1 Geometry

We create a sheet for the beam. We change the viewpoint to a top view, fit the shape in the workspace window and hide the working plane.

Main menu → Geometry → Create → Add polygon sheet

Main menu → Viewer → Viewpoints → Top view

Main menu → Viewer → Fit all

![Figure 5: Add Beam polygon sheet](image1)

![Figure 6: Top view of the Beam](image2)
Similarly, we create sheets for the left and right supports and the loading block. For the left support an additional vertex is created at \((160, -40, 0)\) mm to account for the constrain we apply along the \(Y\) direction. Similarly, we add a vertex at \((820, 665, 0)\) mm to the loading block where to apply the nodal vertical force.

Figure 7: Add Left support polygon sheet

Figure 8: Add right support polygon sheet

Figure 9: Add Loading block polygon sheet
Figure 10: Top view of the model
We create a sheet for the web opening.

**Main menu** ➔ Geometry ➔ Create ➔ Add polygon sheet [Fig. 11]

**Figure 11:** Add *Opening* polygon sheet

**Figure 12:** Top view of the model
We subtract the sheet *Opening* from the sheet *Beam*.

**Main menu** → **Geometry** → **Modify** → **Subtract shapes**

![Figure 13: Subtract shapes](image1)

![Figure 14: Top view of the model](image2)
2.2 Properties

We assign the element class and the material and geometrical properties to the beam. We use regular plane stress finite elements. The material is linear elastic. Regarding the geometry, we need to specify the thickness of the beam (120 mm).

DianaIE
We assign the element class and the material and geometrical properties also to the supports and the loading block. We also use regular plane stress finite elements. The material is linear elastic. The thickness is equal to that of the beam (120 mm), thus we do not need to specify a new geometry set.

Main menu → Geometry → Assign → Shape Properties [Fig. 19]
Shape Properties → Material → Add material [Fig. 20] → Edit material [Fig. 21]

Figure 19: Property assignments
Figure 20: Add new material
Figure 21: Material properties
2.3 Boundary Conditions

We support the right edge (i.e., the centerline) of the beam in $X$ direction to get the symmetry condition.

---

**Main menu** ➔ **Geometry** ➔ **Assign** ➔ **Add supports**  📒 [Fig. 22] [Fig. 23]

**Figure 22:** Apply symmetry boundary conditions  
**Figure 23:** Symmetry boundary conditions
We support the center of both support blocks in Y direction. Due to the symmetry boundary condition (the center bottom point of the Right support has already been supported in X direction), we now have a roller under the center of the left support and we pinned the center of the right support.

**Main menu** → **Geometry** → **Assign** → **Add supports**  [Fig. 24]  [Fig. 25]

Figure 24: Apply vertical (Y direction) boundary conditions

Figure 25: Vertical (Y direction) boundary conditions
2.4 Loads

We apply a nodal force of \(-4\times10^5\) N to the center of the loading block.

Figure 26: Apply vertical nodal force

Figure 27: Vertical nodal force
2.5 Mesh

We set the mesh properties with an element size of 50 mm and we generate the mesh.

Figure 28: Mesh properties

Figure 29: Finite element mesh
3 Structural linear static analysis

3.1 Commands

We perform a linear structural analysis.

Main menu ➔ Analysis ➔ Add analysis ➔ LinSta
Analysis browser ➔ Analysis1 ➔ Rename ➔ LinSta
Analysis browser ➔ LinSta ➔ Add command ➔ Structural linear static
Main menu ➔ Analysis ➔ Run selected analysis

Figure 30: Analysis window
Figure 31: Add command
Figure 32: Analysis tree
3.2 Results

3.2.1 Reaction Forces

To validate the results we check that the reaction forces are in equilibrium with the applied load. Since the load has only vertical component, we check only the reaction forces along the Y axis (FBY).

The summation of all nodal reaction forces along the Y axis is equal to 400 kN, in equilibrium with the total applied load $P = -400$ kN.
3.2.2 Total Displacements

We make a contour plot of the total displacements $Dt_{XYZ}$ [Fig. 36].

As expected, the highest total displacements $Dt_{XYZ}$ are located near the applied nodal force and in the proximity of the opening [Fig. 36].
In order to have a smooth contour plot, we choose a continuous color scale [Fig. 38].
3.2.3 Stress Field SXX

We create a contour plot for the stress SXX [Fig. 42]. We only display the element set Beam. We change the color scale to specified values with minimum value equal to -12 N/mm² and maximum value 4 N/mm² (this provides a better understanding of the stress field in the beam).
Due to the bending of the beam, the stress component \( S_{XX} \) takes the highest values (in tension) at the bottom part of the beam between the two supports and at the top right (the center of the full beam). High values of \( S_{XX} \) are also noticed at the bottom left and top right corner of the opening. On the contrary, negative values (compression) of \( S_{XX} \) are observed at the other two corners.
3.2.4 Stress Field SYY

We create a contour plot for the stress SYY [Fig. 44].

The highest absolute values of SYY are located close to the supports and the loading block. Highest concentration of stresses are also noticeable at the opening’s corners as observed for the case of SXX.
3.2.5 Stress field SXY

We create a contour plot for the stress SXY [Fig. 47]. The color scale limits are changed to the minimum and maximum values of SXY that correspond to -4 N/mm$^2$ and 4 N/mm$^2$, respectively.

The contour plot shows that the highest absolute values of SXY are in two regions with an inclination of 45° starting from the loading block. It is interesting to observe how the stresses redistribute around the opening.
3.2.6 In-Plane Principal Components – Stress Field SXX

We create a vector plot of the in-plane principal components of the SXX stresses.

**Results browser** → LinSta → Output linear static analysis → Element results → Cauchy Total Stresses → SXX  
→ Show in-plane principal components  

[Fig. 48] [Fig. 49]

Figure 48: Results browser

Figure 49: In-plane components of SXX
3.2.7 Diagram – Stress Field SXX

We create diagrams of the stress field along a vertical cut at $X = 1050$ mm (450 mm from the symmetry line). Therefore, we create a probe curve.

Property Panel → Result → Probing curve setting → Add... → Curve [Fig. 50]
Property Panel → Result → Probing curve setting → probe-curve → Number of intervals between points → 20 [Fig. 51]
Property Panel → Result → Probing curve setting → probe-curve → Add... → Point coordinates [Fig. 52]
Property Panel → Result → Probing curve setting → probe-curve → Point coordinates [Fig. 53]
We now create a diagram of $S_{XX}$ along the vertical cut at $X = 1050$ mm [Fig. 55].

The diagram clearly shows that at the bottom of the beam we have positive stresses (tension) that become negative (compression) while moving toward the top. Nevertheless, close to the top surface (near the loading plate) the stress becomes again positive.
3.2.8 Diagram – Stress Fields SYY and SXY

Similarly, we create a diagram of SYY and SXY along the vertical cut at \( X = 1050 \text{ mm} \) [Fig. 56] [Fig. 57].

Figure 56: Stress SYY along the cut at \( X = 1050 \text{ mm} \)

Figure 57: Stress SXY along the cut at \( X = 1050 \text{ mm} \)
Appendix A  Additional Information

Folder: Tutorials/DeepBeamWithWebOpening

Number of elements ≈ 350

Keywords:
   ANALYS: linear static.
   CONSTR: suppor.
   ELEMEN: cq16m pstres.
   LOAD: force node.
   MATERI: elasti isotro.
   OPTION: direct units.
   POST: binary ndiana.
   PRE: dianai.
   RESULT: cauchy displa extern force green reacti strain stress total.

References:
