Column-Beam Joint in a Portal Frame
1 Description

Column-beam joints of a portal frame are designed to transfer forces in a structure through moment resisting action. Therefore, the ultimate load carrying capacity of the connection when subjected to bending moments is often of interest.

In this tutorial, an experiment aimed at evaluating the ultimate load carrying capacity of a reinforced concrete column-beam knee joint, subjected to a closing moment under monotonic loading\(^1\), is simulated by means of a two-dimensional plane stress model. The experimental specimen is a half portal frame which is loaded horizontally. The geometry, reinforcement details, loading and boundary conditions can be seen in Figure 1, Figure 2 and Figure 3 respectively.

\(^1\)Long and Lee, *Modelling of Two Dimensional Reinforced Concrete Beam-Column Joints Subjected to Monotonic Loading*, 2015
A structural nonlinear analysis is carried out accounting for physical and geometrical nonlinear effects. The horizontal load is applied to the beam through a loading plate by means of a displacement control procedure. Displacement is prescribed in a number of load steps. The important physical nonlinear phenomena considered in the analysis are:

- cracking of concrete
- crushing of concrete
- bond-slip of longitudinal reinforcement bars

A total strain based crack (TSC) formulation is used to describe the behavior of concrete. In this approach, a one-to-one relationship is assumed between the total strain and stress at a material point.

The values for Young’s modulus and characteristic cylinder compressive strength \( f_{ck} \) are provided in the paper\(^2\). The unknown material properties for concrete are calculated from \( f_{ck} \), using the expressions given in the fib Model Code\(^3\).

The behavior in tension is described by the Hordijk tensile curve with Govindjee crack-bandwidth estimator and a damaged based Poisson’s ratio reduction model.

The compressive behavior is described through an ideal curve with a reduction in compressive strength due to lateral cracking as per the reduction model - Vecchio and Collins 1993. Additionally a stress confinement model by Selby and Vecchio is also considered.

For reinforcement steel, Von Mises plasticity with an assumption of isotropic strain hardening is used.

Two different assumptions are made for the behavior of the bond between concrete and steel depending on the type and the location of the reinforcement within the specimen. For stirrups and longitudinal reinforcements in compression, i.e. the inner bars in column and the bottom bars in the beam, an embedded formulation is utilized. This implies an assumption of perfect bond between concrete and steel.

For the longitudinal reinforcements in tension, i.e. the outer bars in the column and top bars in the beam, bond-slip properties are provided, which describe the normal and shear behavior of the interface between concrete and steel.

Linear interface elements are used between the loading plate and the beam. Nonlinear interface elements with no-tension and shear stiffness reduction are used between the base support and the column.

---


\(^3\) CEB-FIP, *fib Model Code for Concrete Structures*, 2010
The material properties assumed in the analysis are listed in Table 1.

Table 1: Material properties

<table>
<thead>
<tr>
<th>Material Type</th>
<th>Model Type</th>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Concrete</strong></td>
<td>(Total strain crack model)</td>
<td>Young’s modulus</td>
<td>30000 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Poisson’s ratio</td>
<td>0.2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Tensile strength</td>
<td>2.6464 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Mode I tensile fracture energy</td>
<td>0.137863 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Compressive strength</td>
<td>34.2 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Lower bound reduction curve</td>
<td>0.4</td>
</tr>
<tr>
<td><strong>Longitudinal reinforcement steel</strong></td>
<td>(Von Mises Plasticity)</td>
<td>Young’s modulus</td>
<td>200000 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Yield stress</td>
<td>360 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Ultimate stress</td>
<td>400 MPa</td>
</tr>
<tr>
<td><strong>Stirrup reinforcement steel</strong></td>
<td>(Von Mises Plasticity)</td>
<td>Young’s modulus</td>
<td>200000 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Yield stress</td>
<td>331 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Ultimate stress</td>
<td>365 MPa</td>
</tr>
<tr>
<td><strong>Bond-slip reinforcement interface</strong></td>
<td>(Linear elasticity)</td>
<td>Normal stiffness modulus</td>
<td>1000 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shear stiffness modulus</td>
<td>10 MPa</td>
</tr>
<tr>
<td><strong>Loading plate</strong></td>
<td>(Linear elasticity)</td>
<td>Young’s modulus</td>
<td>200000 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Poisson’s ratio</td>
<td>0</td>
</tr>
<tr>
<td><strong>Loading plate-Beam interface</strong></td>
<td>(Linear)</td>
<td>Normal stiffness modulus</td>
<td>1000 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shear stiffness modulus</td>
<td>10 MPa</td>
</tr>
<tr>
<td><strong>Base support-Column interface</strong></td>
<td>(No tension with shear stiffness reduction)</td>
<td>Normal stiffness modulus</td>
<td>1 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shear stiffness modulus</td>
<td>10000 MPa</td>
</tr>
</tbody>
</table>

Column-Beam Joint in a Portal Frame | https://dianafea.com
2 Finite Element Model

For the modeling session we start a new project for structural analysis as seen in Figure 4.

Figure 4: New project dialog
The units set are millimeter for length, ton for mass and newton for force [Fig. 5].

Figure 5: Units
2.1 Geometry

We create the geometry of the frame using polygon sheets. We start with the coordinates defined in Figure 6.

Figure 6: Add sheet - Column

Figure 7: View of the model - Column
Similarly, we create the beam and the loading plate as seen in Figure 8 and Figure 9. With this the portal frame is created.

**Main menu** ➔ Geometry ➔ Create ➔ Add polygon sheet  
*Fig. 8*  [Fig. 9]

**Figure 8**: Add sheet - Beam  
**Figure 9**: Add sheet - Loading plate  
**Figure 10**: View of the model - portal frame
Now, we make the reinforcements. We first create the vertical reinforcement bars in the column by defining lines as seen in Figure 11 and Figure 12.

**Main menu** → Geometry → Create → Add line → [Fig. 11] [Fig. 12]

**Figure 11:** Add line - *Outer column reinforcement*

**Figure 12:** Add line - *Inner column reinforcement*

**Figure 13:** View of model - column reinforcement
Similarly, we create the top and bottom longitudinal reinforcement bars in the beam as defined in Figure 14 and Figure 15 respectively.

![Figure 14: Add line - Top beam reinforcement](https://dianafea.com)

![Figure 15: Add line - Bottom beam reinforcement](https://dianafea.com)

![Figure 16: View of model - beam reinforcement](https://dianafea.com)
Further, we create the stirrups, which have different details in the column, beam and joint region. First we create the bottom-most stirrup as seen in Figure 17.

Main menu ➔ Geometry ➔ Create ➔ Add line ➔ [Fig. 17]

Figure 17: Add line - *Column stirrup*

Figure 18: View of model - *Column stirrup*
We copy this stirrup 11 times with required spacing (100 mm) [Fig. 19].

Figure 19: Array copy - Column stirrup

Figure 20: View of model - column stirrups
We create the stirrups in the joint and beam by following a similar procedure. First, we define the stirrups in the joint with a spacing of 55 mm [Fig. 21 to 23]. Since, there are two different stirrup spacings in the beam (100 mm and 150 mm), the array copy command is used twice while drawing the beam stirrups. The generation of joint and beam stirrups is shown in Figure 21 to Figure 28.
So, we first create the stirrups in the beam with 150 mm spacing.
And now the stirrups with 100 mm of spacing

Figure 27: Array copy - Beam stirrups

Figure 28: View of model - beam stirrups
We group the stirrups into a reinforcement set because they have the same geometry and material properties. For that, we create a reinforcement set called **Stirrup** and move all the stirrups to that set.

Tip: an alternative way to create the reinforcement set is to select the shapes in the geometry browser, right-click in the selection and choose the option 'New reinforcement shapeset from selection'.
2.2 Properties

We start by assigning properties to the column. For the material properties, we use a total strain based crack model with the parameters described in Table 1.

Figure 30: Property assignments

Figure 31: Add material - Concrete

Figure 32: Edit material - Concrete
Figure 33: Edit material - Concrete

Figure 34: Edit material - Concrete

Figure 35: Edit material - Concrete
For geometry properties, we use regular membrane (plane stress) elements and the thickness is 300 mm [Fig. 1].

Figure 36: Add geometry - Column

Figure 37: Edit geometry - Column
We assign properties to the beam. Since the same material *Concrete* is used, we only need to create a new geometry with a thickness of 200 mm [Fig. 1].

---

**Main menu** ➔ Geometry ➔ Assign ➔ Shape Properties 📜 [Fig. 38]
Shape Properties 📜 ➔ Geometry ➔ Add new geometry 📜 [Fig. 39] ➔ Edit geometry [Fig. 40]

---

Figure 38: Beam property assignment

Figure 39: Add geometry - *Beam*

Figure 40: Edit geometry - *Beam*
We assign properties to the loading plate. The thickness of the loading plate is assumed equal to the beam, hence the same geometry *Beam* is used. Linear elastic isotropic material model is used with the parameters listed in Table 1.

**Main menu**
- Geometry
- Assign
- Shape Properties

**Shape Properties**
- Material
- Add material

**Figure 41:** Shape property assignments - *Loading plate*

**Figure 42:** Add material - *Loading plate*

**Figure 43:** Edit material - *Loading plate*
We assign the properties to the reinforcement. The longitudinal vertical bars on the inner side of the column and bottom of the beam are considered first. These bars are considered as embedded reinforcements, implying a perfect bond between concrete and steel. For material properties, we use Von Mises plasticity with isotropic strain hardening with the parameters listed in Table 1. For the geometry properties, we reduce the geometry of the bars to a two-dimensional problem by summing the total amount of steel area present in the cross-section into a single bar with an equivalent area. We start with the inner column reinforcement. As seen in Figure 2, two bars each of diameter 25 mm and 22 mm are present in the inner column side, which amounts to a total area of 1742.01 mm².

DianaIE

<Select the correspondent reinforcement in the Geometry browser>
Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties [Fig. 44]
Reinforcement properties ➔ Material ➔ Add material [Fig. 45] ➔ Edit material [Fig. 46] - [Fig. 55]
Reinforcement properties ➔ Geometry ➔ Edit geometry [Fig. 49]
Figure 47: Edit material - Main reinforcement embedded

Figure 48: Edit material - Main reinforcement embedded

Figure 49: Edit geometry - Inner column reinforcement
We assign the properties to the bottom beam reinforcement. The same material model, *Main reinforcement embedded* is used. So we only need to define a new geometry with a cross-section of $628.32 \text{ mm}^2$ [Fig. 2].

**Select the correspondent reinforcement in the Geometry browser**

**Main menu** ➔ **Geometry** ➔ **Assign** ➔ **Reinforcement properties** [Fig. 50]

**Reinforcement properties** ➔ **Geometry** ➔ **Edit geometry** [Fig. 51]
We assign properties to the outer bars in column and top bars in the beam. These bars are considered as bond-slip reinforcements (type circular bond-slip bar) and a Von Mises plasticity model with isotropic strain hardening is used with the parameters listed in Table 1. For geometry properties, we reduce the geometry of the bars to a two-dimensional problem as done previously. Geometry of bond-slip reinforcements has to be defined using the diameter, so additionally, the equivalent area is converted to an equivalent diameter. We start with the outer bars in the column. As seen in Figure 2, two bars of diameter 25 mm and 22 mm each are present in the outer column side, which amounts to an area of 1742.01 mm². By equating this to the area of a circle, we obtain an equivalent diameter equal to 47.0956 mm. The bond-slip interface elements are treated as beam elements (reinforcement type circular bond-slip bar). We edit properties for it through the geometry browser [Fig. 52 to 57].
Figure 52: Reinforcement property assignment - Outer column reinforcement

Figure 53: Add material - Main reinforcement bond-slip

Figure 54: Edit material - Main reinforcement bond-slip
Figure 55: Edit material - *Main reinforcement bond-slip*

Figure 56: Edit material - *Main reinforcement bond-slip*

Figure 57: Add geometry - *Outer column reinforcement*
We assign the properties to the top beam reinforcement. The same material model, *Main reinforcement bond-slip* is used. So we only need to define a new geometry with a cross-section of 1472.62 mm² [Fig. 2], which yields an equivalent diameter of 43.3012 mm.

**Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties**  ![Fig. 58]

**Reinforcement properties ➔ Geometry ➔ Edit geometry**  ![Fig. 59]
We assign the properties to the stirrups, grouped in the reinforcement set Stirrup [Fig. 29]. We use embedded reinforcements with Von Mises plasticity with isotropic strain hardening with the parameters listed in Table 1. As seen in Figure 2, the diameter of the stirrups is 10 mm. We consider twice the cross-sectional area, i.e. 157.08 mm$^2$ as the equivalent area of a single stirrup bar.

<Select the correspondent reinforcement set in the Geometry browser>

**Main menu** → Geometry → Assign → Reinforcement properties [Fig. 60]

Reinforcement properties → Material → Add material [Fig. 61] → Edit material [Fig. 62] [Fig. 64]

Reinforcement properties → Geometry → Edit geometry [Fig. 65]
Figure 63: Edit Material - Stirrup

Figure 64: Edit Material - Stirrup

Figure 65: Add geometry - Stirrup
We use linear interface elements at the connection between the beam and the loading plate [Fig. 66]. We use two-dimensional line interface elements with the properties listed in Table 1. A high normal stiffness and a low shear stiffness is provided in order to ensure a smooth transfer of load.
We also provide a geometry in which we specify the thickness equal to the beam width, i.e. 200 mm [Fig. 70].

Figure 69: Add geometry - *Linear interface*

Figure 70: Edit geometry - *Linear interface*
We use nonlinear interface elements at the column base in order to allow free rotation of the column edge as in experimental specimen [Fig. 3]. We use two-dimensional line interface with no tension and shear stiffness reduction with the parameters listed in Table 1. The no-tension condition along with a low value for normal stiffness is chosen to simulate free rotation of the column base. A high value for shear stiffness is chosen to avoid translation in the horizontal direction. We also provide a geometry in which we specify the thickness equal to the column width, i.e. 300 mm [Fig. 76].
Figure 73: Edit material - *Nonlinear interface*

Figure 74: Edit material - *Nonlinear interface*
Figure 75: Add geometry - *Nonlinear interface*

Figure 76: Edit geometry - *Nonlinear interface*
2.3 Boundary Conditions

The constraints are schematically presented in Figure 3. We constrain a point on the loading plate in the $X$ and $Y$ direction [Fig. 77]. The constraint in the $X$ direction is added in order to prescribe deformation at this point, hence this support is named *Deformation support*. We then use a tying such that the deformation of this point and of the outer edge of the loading plate are always equal. In this manner, the load is distributed over the outer edge of the loading plate.

![Figure 77: Attach support - Deformation support](https://dianafea.com)

![Figure 78: Attach tying](https://dianafea.com)
Next, we constrain the base edge of the column [Fig. 79] in X and Y direction.

**Figure 79: Attach support - Column support**

**Figure 80: View of model - supports**
2.4 Loads

We apply the load as a prescribed deformation. For that, the deformation is prescribed in the X direction at the supported point of the loading plate [Fig. 81].
2.5 Mesh

We set an element size of 25 mm for the beam and column and generate the mesh.

---

**Figure 83:** Mesh properties

**Figure 84:** Finite element mesh
3 Structural Nonlinear Analysis

3.1 Commands

We add a new analysis and rename it to *Nonlinear*. We add a command for 'Structural nonlinear' [Fig. 87]. First we edit the properties of the 'Nonlinear effects' section and specify 'Geometrically nonlinear' [Fig. 86]. Further we set the load step sizes using 400 steps of factor 0.1 followed by 80 steps of factor 0.5. We also use 'Arc length control'.

---

**Main menu** ➔ **Analysis** ➔ **Add analysis**

**Analysis browser** ➔ **Analysis1** ➔ **Rename** ➔ **Nonlinear**

**Analysis browser** ➔ **Nonlinear** ➔ **Add command** ➔ **Structural nonlinear**

**Analysis browser** ➔ **Nonlinear** ➔ **Structural nonlinear** ➔ **Nonlinear effects** ➔ **Edit properties** [Fig. 85] [Fig. 86]

**Analysis browser** ➔ **Nonlinear** ➔ **Structural nonlinear** ➔ **new execute block** ➔ **Load steps** ➔ **Edit properties** [Fig. 87] [Fig. 88]
We change the maximum number of iterations to 20 and edit the 'Equilibrium iteration', 'Displacement' and 'Force' convergence norms. For settings of each of the convergence criteria we change 'No convergence' setting to 'Continue'. This means that the analysis continues even if convergence is not achieved at a particular load step.
We define the output of the analysis. We use the 'User selection' option in the properties of the 'Output' section [Fig. 92]. By clicking on 'Modify', we can select the desired output results [Fig. 93]. We choose results for displacements, reaction forces, local, global and principal values for total strains and stresses and principal crack width. For all stress and strain items we select output at 'Integration points' by clicking on 'Properties' [Fig. 94]. Finally, we run the analysis.

DianaIE

Analysis browser  ➔ Nonlinear ➔ Structural nonlinear ➔ Output ➔ Edit properties [Fig. 92] ➔ Result ➔ User selection ➔ Modify [Fig. 93]
Results Selection ➔ Stress/Strain items ➔ Properties... ➔ Result Item Properties ➔ Location ➔ Integration points [Fig. 94]
Main menu ➔ Analysis ➔ Run analysis
Figure 92: Output Properties
Figure 93: Output selection
Figure 94: Results at integration points for all stress and strain result items
3.2 Results

3.2.1 Force vs. Displacement Curve

We select load step 480 in the result browser and check the final deformed shape of the specimen through displacements TDtXYZ [Fig. 96].

Figure 95: Results browser

Figure 96: Displacements TDtXYZ for load step 480
Next, we plot the force versus displacement curve for the specimen by checking the reaction FBX at the loading point. We start by switching on the 'Node selection mode' and select the node of interest [Fig. 97]. By right clicking on reaction FBX in the result browser [Fig. 98] we obtain a chart view, from which the values for all load steps can be copied in order to compare the force vs. displacement plot with the experimental results [Fig. 99].
The authors of the experiment reported an imperfection in the boundary condition due to which a direct comparison with the experimental force-displacement curve is not possible\(^4\). Hence, a curve from a simulation performed by the same authors is compared with the obtained result in which a good agreement is observed [Fig. 100].

\[\text{Figure 100: Force vs displacement comparison}\]

\[^4\text{Long and Lee, Modelling of Two Dimensional Reinforced Concrete Beam-Column Joints Subjected to Monotonic Loading, 2015}\]
3.2.2 Concrete Cracking

For the results to follow, we switch off the deformed shape in the results browser. We also turn off the deformed mesh and switch to the undeformed mesh with feature edges for a better visualization [Fig. 101]. We examine the crack patterns at some of the load steps to study the behavior of the specimen. First, we view the crack pattern at load step 20 by selecting Ecw1 under crack-widths in the result browser and select load-step 20 [Fig. 101]. This step marks the onset of cracking and corresponds to a displacement of 1.99 mm in the load displacement curve [Fig. 100]. We set the scale of the contours by specifying a minimum value of 0 and a maximum value of 0.07 [Fig. 102].

This crack location can be attributed to the reduction of cross section thickness as we move from the beam to the column.
Further we study the development of the crack pattern by viewing crack widths at load steps 60, 100, 200, 400, 480 [Fig. 104 to 108]. The color scale limits indicated in the legend of each plot can be defined, similar to the previous plot [Fig. 101], for a better visualization.

Figure 104: Crack pattern at load step 60 (displacement: 5.9mm)

Load step 60 [Fig. 104], which corresponds to a displacement of 5.9 mm in Figure 100, indicates that the bending cracks were formed during the early stage of the analysis. These cracks widen as the load increases until diagonal cracks are initiated in the joint as observed at load step 100 [Fig. 105].
Upon further increase in the load, new diagonal cracks localize in the joint as seen at load step 200 [Fig. 106]. After this point few new cracks are formed in the joint region and the pattern is characterized by widening of the existing cracks, as can be observed from load step 400 [Fig. 107] and load step 480 [Fig. 108].
3.2.3 Concrete Crushing

Load step 400 is the point where the force versus displacement curve starts showing a plateau. We consider this as the ultimate load and check the concrete crushing through the principal stress $S_2$ [Fig. 109] and principal strain $E_2$ [Fig. 110]. Since an ideal compressive behavior is defined for concrete, the scale limit for $S_2$ is set to the compressive strength and an ultimate strain value of 0.0035 is chosen for $E_2$ in order to visualize crushing of concrete.

**Figure 109: Principal stress $S_2$ - load step 400**

**Figure 110: Principal strain $E_2$ - load step 400**
The cracking and crushing pattern in the joint at this step is then compared with the experimental observation, where a good agreement is observed [Fig. 111 to 113].

Figure 111: Principal strain E2 - load step 400

Figure 112: Crack pattern in joint region - load step 400

Figure 113: Crack pattern - experimental observation
3.2.4 Reinforcement Stresses and Strains

We also check the stresses and strains in the reinforcement at load step 400 (peak load).

Yielding of the reinforcements can be observed at the ultimate load in the stress plot in Figure 114. The strains are plotted at a scale limit of minimum -0.025 to maximum 0.025 [Fig. 115], which is usually considered as the ultimate strain limit for steel in engineering practice. Large strains in stirrups can be observed, in the joint region, where large crack localize.
Appendix A  Additional Information

Folder: Tutorials/Portal

Number of elements $\approx 2896$

Keywords:
- ANALYS: geomet nonlin physic.
- CLASS: large.
- CONSTR: suppor tying.
- ELEMEN: bar beam bondsl circle cl12i cl9be class3 cq16m cq22if interf pstres reinfo struct.
- LOAD: deform.
- MATERI: consta crack elasti epssig harden hordyk isotro multil plasti rotati soften strain totstr vonmis.
- OPTION: arclen direct lagran newton normal regula total units update.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy crkwdt displa extern force green princi reacti strain stress total tracti.

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.