Reinforced Concrete Column under Uniaxial Cyclic Loading
Outline

1 Description 3

2 Finite Element Model 6
2.1 Geometry 8
2.2 Properties 18
2.3 Boundary Conditions 25
2.4 Loads 31
2.5 Mesh 33

3 Structural Nonlinear Analysis 34
3.1 Commands 34
3.2 Results 40
   3.2.1 Deformed Shape 40
   3.2.2 Force vs Displacement Hysteresis Curve 41
   3.2.3 Crack Pattern 43
3.3 Full Analysis Results 46
   3.3.1 Hysteresis Curve 46
   3.3.2 Failure Mode 47

Appendix A Additional Information 49
1 Description

The behavior of axially loaded Reinforced Concrete (RC) columns combined with cyclic horizontal loading is recognized as an important research topic as it is essential in determining the earthquake response of a typical building structure. In this tutorial, an experiment performed on a reinforced concrete column loaded with a combination of axial and horizontal cyclic loading\(^1\), is simulated by means of a 3D solid model. The geometry, reinforcement details, loading and boundary conditions of the experiment are seen in Figure 1 and Figure 2 respectively.

\(^1\)Rodrigues et al., *Behavior of RC building columns under cyclic loading: experimental study*, 2013
The symmetry of the specimen is exploited and only half the column and foundation block is modeled. The horizontal load is applied directly to the top surface of the column by means of prescribed displacement. Hence, the height of the column above the plate of the horizontal jack is discarded. The fixing of the foundation block to the ground [Fig. 2] is modeled by means of no tension interfaces at the base of the block and the location of the bolt plates at the top surface of the block.

A structural nonlinear analysis is carried out accounting for physical and geometrical nonlinear effects. The peak displacements of the loading cycles are taken from the experimental paper. In order to restrict running time, the analysis is not set up until failure. Instead, the first six loading cycles in the experiment are considered. However, in the final section, some results from a full analysis are presented.

A total strain based crack (TSCR) 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 value for mean compressive strength ($f_{cm}$) at testing time is provided in the paper\(^2\). The unknown material properties for concrete are calculated from $f_{cm}$, using the expressions given in fib Model Code\(^3\). The behavior in tension is described by the Japanese Society of Civil Engineers Code (JSCE) tension stiffening with a damaged based Poisson’s ratio reduction model. Maekawa cracked concrete curves are used as compression curves which describe the behavior of concrete under hysteresis loading. Additionally a stress confinement model by Selby and Vecchio is also applied.

For reinforcement steel, Menegotto-Pinto plasticity model is applied. Two different assumptions are made for the behavior of bond between concrete and steel depending on the location of the reinforcement within the specimen. For reinforcements in the foundation block, an embedded formulation is utilized. This implies an assumption of perfect bond between concrete and steel. For reinforcements in the column, bond slip properties are provided, which describe the normal and shear behavior of the interface between concrete and steel.

Nonlinear interface elements with no-tension and shear stiffness reduction are used to model the fixing of the foundation block to the ground.

\(^2\)Rodrigues et al., Behavior of RC building columns under cyclic loading: experimental study, 2013
\(^3\)CEB-FIP, fib Model Code for Concrete Structures, 2010
The material properties assumed in the analysis are listed in Table 1.

<table>
<thead>
<tr>
<th>Material</th>
<th>Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Concrete (TSCR)</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>38524.5 N/mm²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.2</td>
</tr>
<tr>
<td>Mass density</td>
<td>2.4e-9 T/mm³</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>4.04607 N/mm²</td>
</tr>
<tr>
<td>Power c</td>
<td>0.4</td>
</tr>
<tr>
<td>Compressive strength</td>
<td>57.53 N/mm²</td>
</tr>
<tr>
<td><strong>Reinforcement (Menegotto-Pinto plasticity)</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>194660 N/mm²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Mass density</td>
<td>7.85e-9 T/mm³</td>
</tr>
<tr>
<td>Yield stress</td>
<td>432.63 N/mm²</td>
</tr>
<tr>
<td>Initial tangent slope ratio</td>
<td>0.00286359</td>
</tr>
<tr>
<td>Initial curvature parameter</td>
<td>20</td>
</tr>
<tr>
<td>Constant a1</td>
<td>18.5</td>
</tr>
<tr>
<td>Constant a2</td>
<td>0.15</td>
</tr>
<tr>
<td>Constant a3</td>
<td>0.01</td>
</tr>
<tr>
<td>Constant a4</td>
<td>7</td>
</tr>
<tr>
<td><strong>Bond-slip reinforcement interface (Linear)</strong></td>
<td></td>
</tr>
<tr>
<td>Normal stiffness modulus</td>
<td>1000 N/mm²</td>
</tr>
<tr>
<td>Shear stiffness modulus</td>
<td>100 N/mm²</td>
</tr>
<tr>
<td><strong>Steel plates (Linear)</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>194660 N/mm²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Mass density</td>
<td>7.85e-9 T/mm³</td>
</tr>
<tr>
<td><strong>Foundation block interface (No tension with shear stiffness reduction)</strong></td>
<td></td>
</tr>
<tr>
<td>Normal stiffness modulus</td>
<td>1000 N/mm²</td>
</tr>
<tr>
<td>Shear stiffness modulus</td>
<td>1000 N/mm²</td>
</tr>
</tbody>
</table>
2 Finite Element Model

For the modeling session we start a new project for structural analysis as seen in Figure 3.

Figure 3: New project dialog
The units set are millimeter for length, ton for mass and newton for force as seen in Figure 4.

![Figure 4: Units](https://dianafea.com)

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Unit</th>
<th>Symbol</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>millimeter</td>
<td>mm</td>
<td></td>
</tr>
<tr>
<td>Mass</td>
<td>ton</td>
<td>T</td>
<td></td>
</tr>
<tr>
<td>Force</td>
<td>newton</td>
<td>N</td>
<td></td>
</tr>
<tr>
<td>Time</td>
<td>second</td>
<td>s</td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>kelvin</td>
<td>K</td>
<td></td>
</tr>
<tr>
<td>Angle</td>
<td>degree</td>
<td>°</td>
<td></td>
</tr>
</tbody>
</table>
2.1 Geometry

We do the geometry of the specimen by creating polygon sheets and then extruding them. We start with the foundation block with coordinates defined in Figure 5 and extrusion defined in Figure 6.

---

**Main menu** ➔ **Geometry** ➔ **Create** ➔ Add polygon sheet  
[Fig. 5]

**Main menu** ➔ **Geometry** ➔ **Modify** ➔ **Extrude**  
[Fig. 6]
Similarly, we create the column as seen in Figure 8 and Figure 9. With this the concrete geometry is created.
Now, we make the reinforcements. We start with the foundation block and create the top and bottom reinforcement lines as seen in Figure 11 and Figure 12.

**Main menu ➔ Geometry ➔ Create ➔ Add line**

[Fig. 11] [Fig. 12]

Figure 11: Add line - *Base bottom reinforcement*

Figure 12: Add line - *Base top reinforcement*

Figure 13: View of model
Then we make two copies of the bottom line as seen in Figure 14 to make the left side reinforcements. Further, we copy the newly created left side lines once to create the right side reinforcements [Fig. 15].

**Main menu ➔ Geometry ➔ Modify ➔ Array copy**  [Fig. 14]  [Fig. 15]
We copy the top and bottom lines eight times as seen in Figure 17.

Figure 17: Array copy - *Base top and bottom reinforcement*

Figure 18: View of model
We create a stirrup in the foundation block using a closed polygon Figure 19 and then copy it four times along the Y direction Figure 20 to complete the foundation reinforcements.

**Main menu → Geometry → Create → Add polyline** [Fig. 19]

**Main menu → Geometry → Modify → Array copy** [Fig. 20]

Figure 19: Add polygon - *Base stirrup*

Figure 20: Array copy - *Base stirrup*

Figure 21: View of model
Next we create reinforcements in the column. We make the longitudinal bars first [Fig. 22 to 24].

**Main menu ➔ Geometry ➔ Create ➔ Add polyline** [Fig. 22]  [Fig. 23]

**Main menu ➔ Geometry ➔ Create ➔ Add line** [Fig. 24]

---

**Figure 22**: Add polygon line - *Longitudinal reinforcement* 1

**Figure 23**: Add polygon line - *Longitudinal reinforcement* 2

**Figure 24**: Add line - *Longitudinal reinforcement* 3

**Figure 25**: View of model
Then we create a stirrup in the column [Fig. 26] and copy it seven times with a spacing of 150 mm [Fig. 27].

**Figure 26:** Add polygon line - *Base stirrup*

**Figure 27:** Array copy - *Base stirrup*

**Figure 28:** View of model
Next, we copy the topmost stirrup three times with a spacing of 75 mm Figure 29. With this we finish the reinforcements according to the experimental details shown in Figure 1.

Figure 29: Array copy - topmost column stirrup

Figure 30: View of model - reinforcements
We now make the steel plates which we use to fix the foundation block to the floor as in the experiment [Fig. 2]. We create a sheet [Fig. 31] and copy it once [Fig. 32]. The geometry of the model is complete.
2.2 Properties

We start by assigning the properties to the concrete parts, i.e. Foundation block and Column [Fig. 34 to 39]. For the material properties we use a total strain based crack model with the parameters listed in Table 1.
We assign the properties to the reinforcement bars as per the experiment details seen in Figure 1. We start with the foundation block in which an assumption of embedded reinforcement is made. For material properties we use the Menegotto-Pinto plasticity model with the parameters listed in Table 1. The property assignment for these reinforcements is seen in Figure 41 to Figure 46. The assignment for the side reinforcements is done separately from the rest of the bars, since the side bars differ in diameter [Fig. 46]. The same material is used for all bars in the foundation block.

---

**Main Menu**  ➔  **Geometry**  ➔  **Assign**  ➔  **Reinforcement property assignments**  🌟  [Fig. 41]

**Shape property assignments**  ➔  **Add new material**  🎨  [Fig. 42]  ➔  [Fig. 43]

**Reinforcement property assignments**  ➔  **Add new geometry**  🔗  [Fig. 44]

---

![Figure 41: Reinforcement property assignment](image1.png)

![Figure 42: Add material - Embedded reinforcement](image2.png)

![Figure 43: Edit material - Menegotto-Pinto properties](image3.png)

![Figure 44: Add geometry - 16mm Embedded reinforcement](image4.png)
Figure 45: Reinforcement property assignment - Side bars

Figure 46: Add geometry - 10mm Embedded reinforcement
Similarly, we assign properties to the reinforcement bars in the column. For material properties we use the Menegotto-Pinto plasticity model combined with bond slip behavior with the parameters listed in Table 1. The assignment for these reinforcements is seen in Figure 47 to Figure 54. The assignment for the longitudinal bars and stirrups is done separately due to different diameters. The same material is assigned to all bars. Additionally, an element data is assigned to all bars which describes the discretization of the reinforcement bar. We use truss elements for this purpose.

**Main Menu → Geometry → Assign → Reinforcement property assignments** 
**Shape property assignments → Add new material**  
**Reinforcement property assignments → Add new geometry**

---

**Figure 47: Reinforcement property assignment - Longitudinal bars**

**Figure 48: Add material - Bondslip reinforcement**

**Figure 49: Edit material - bond slip interface**

**Figure 50: Add geometry - 12mm Bondslip reinforcement**
Next we assign the properties to the bolt plates as seen in Figure 55 to Figure 58. We set the element class to regular curved shell and use linear elastic material properties with the parameters listed in Table 1. We define a geometry with thickness of 50 mm.

Main Menu → Geometry → Assign → Shape property assignments [Fig. 55]
Shape property assignments → Add new material [Fig. 56]
Shape property assignments → Add new geometry [Fig. 57]
Shape property assignments → Edit geometry [Fig. 58]
2.3 Boundary Conditions

We fix the foundation block to the ground. We restrain the base of the foundation block in the $X$, $Y$ and $Z$ directions [Fig. 59].

**Main menu ➔ Geometry ➔ Assign ➔ Attach support**

[Fig. 59] [Fig. 60]
Next, we provide a boundary interface between the base support and the base of the foundation block [Fig. 61 to 66]. We use the material properties listed in Table 1.

**Main menu** ➔ **Geometry** ➔ **Assign** ➔ **Attach connection** [Fig. 61]
**Shape property assignments** ➔ **Add new material** [Fig. 62] - [Fig. 64]
**Shape property assignments** ➔ **Add new geometry** [Fig. 65] - [Fig. 66]
Figure 65: Add geometry - Interface

Figure 66: Edit geometry - Interface

Figure 67: View of model - Base interface
Further, we select the steel plates and reverse their orientation [Fig. 68] and imprint them on the top face of the foundation block [Fig. 69].
Further, we restrain the steel plates in X, Y, and Z directions Figure 70 and provide an interface between the plates and the imprinted faces on the top of the foundation block Figure 71. With this we fix the foundation block between the base support and the bolt plates.

**Figure 70**: Attach support - *Plate support*

**Figure 71**: Connection property assignment - *Plate interface*

**Figure 72**: View of model
Further we provide symmetry supports by restraining the displacements in the $Y$ direction along the symmetry face of foundation block and column [Fig. 73]. Since, bond slip properties are defined for the column stirrup, we also restrain the displacements of the column along the $Y$ direction on the symmetry face [Fig. 74]. We add a support in the $X$ direction at the top left point on the column symmetry face [Fig. 75] for prescribing deformation for the horizontal cyclic loading.
2.4 Loads

For the loads, we first define the self weight [Fig. 77]. Next, in the same load set, we attach the axial load as a vertical distributed force on the top face of the column [Fig. 78]. The value of the distributed force is obtained by dividing the experimental axial load by the area of the top face, i.e. \( \frac{170000}{400 \times 200} = 2.125 \text{ N/mm}^2 \).

We apply the horizontal load in a different load set. The deformation is prescribed at the column point supported in the X direction in the previous section [Fig. 79].
To complete the horizontal loading, we tie the $X$ displacements of the top face of the column to the horizontally loaded point [Fig. 81].
2.5 Mesh

We set an element size of 50 mm for the column [Fig. 83] and 100 mm for the foundation block [Fig. 84] and generate the mesh [Fig. 85].
3 Structural Nonlinear Analysis

3.1 Commands

To set up the analysis we start by adding a new analysis and rename it to *Nonlinear*. Next, we add a command for structural nonlinear analysis [Fig. 86]. We edit the nonlinear effects to account for physical and geometrical nonlinearities [Fig. 87].
Further we rename the 'new execute block' as *Permanent loads* [Fig. 88] [Fig. 89].

**Analysis browser** ➔ Nonlinear ➔ Structural nonlinear ➔ new execute block ➔ Rename ➔ Permanent loads  [Fig. 88] [Fig. 89]

---

**Figure 88:** Analysis browser

**Figure 89:** Analysis browser - rename execute load block
We edit the equilibrium iteration properties. We change the maximum number of iterations to 20 and switch on line search [Fig. 90]. For settings of each of the convergence criteria we change 'No convergence' setting to 'Continue' [Fig. 91] [Fig. 92]. This means that the analysis will continue even if convergence is not achieved at a particular load step.
Next we copy the *Permanent loads* command block and rename it to *Horizontal cyclic load* [Fig. 93] [Fig. 94].

**Analysis browser** → Nonlinear → Structural nonlinear → Permanent loads → 

**Analysis browser** → Nonlinear → Structural nonlinear → Permanent loads - copy → 

**Analysis browser** → Nonlinear → Structural nonlinear → Permanent loads 2 → 

**Analysis browser** → Nonlinear → Structural nonlinear → Permanent loads → 

**Analysis browser** → Nonlinear → Structural nonlinear → Permanent loads → 

**Analysis browser** → Nonlinear → Structural nonlinear → Permanent loads → 

**Analysis browser** → Nonlinear → Structural nonlinear → Permanent loads → 

**Analysis browser** → Nonlinear → Structural nonlinear → Solution method → 

**Analysis browser** → Nonlinear → Structural nonlinear → Solution method → 

**Analysis browser** → Nonlinear → Structural nonlinear → Output → 

**Analysis browser** → Nonlinear → Structural nonlinear → Output → 

Figure 93: Analysis browser  
Figure 94: Analysis browser - rename execute load block
In the *Horizontal cyclic load* execute block, we set the load set to *Horizontal cyclic load* and load steps to: 1(3) -1(6) 1(3) 1(5) -1(10) 1(5) 1(10) -1(20) 1(10) 1(4) -1(8) 1(4) 1(12) -1(24) 1(12) 1(15) -1(30) 1(15) [Fig. 95] [Fig. 96]. Here we are imposing horizontal displacement cycles with peak values of 3, 5, 10, 4, 12, 15 mm. These load steps are derived from the experimental paper.

**Analysis browser** ➔ **Nonlinear** ➔ **Structural nonlinear** ➔ **Horizontal cyclic load** ➔ **Load steps** ➔ ![Edit properties](Fig. 95) ➔ ![Edit properties](Fig. 96)

![Figure 95: Analysis tree](image)

![Properties](image)

**Figure 95: Analysis tree**

**Figure 96: Load steps - Horizontal cyclic load**

*DianaIE*
We now select the output items for the analysis. We use the 'User selection' option in the properties of the 'Output' section [Fig. 97]. By clicking on 'Modify', we can select the desired output results [Fig. 98]. We choose results for displacements, reaction forces, local, global and principal values for total strains and stresses and principal crack widths. For all stress and strain items we select output at 'Integration points' by clicking on 'Properties' [Fig. 99]. Finally, we run the analysis.
3.2 Results

3.2.1 Deformed Shape

We view the deformed shape of the column in the final load cycle. For positive (rightward) displacement, we set the required load step in the result browser, i.e. load step 152, (displacement: 15 mm) and set the required result, i.e. Displacements TDXYZ [Fig. 100] [Fig. 101]. We change the result case to load step 182 (displacement: -15 mm) for negative (leftward) deflection [Fig. 102].
3.2.2 Force vs Displacement Hysteresis Curve

Now we make the force vs displacement hysteresis curve for the analysis and compare it with the experimental results. We select the node of interest [Fig. 103] and show table for the result item FBX in the result browser [Fig. 104]. This enables us to view the reaction over multiple load cases [Fig. 105]. We select the required load cases in the chart view (load step 2 onwards) and copy paste the corresponding results to excel in order to make a force vs. displacement hysteresis curve. A comparison with the experiment is seen Figure 106 where a good agreement is found.

Figure 103: Select node of interest
Figure 104: Results browser
Figure 105: FBX - chart view

DianaIE
Figure 106: Force vs displacement hysteresis curve - comparison[1]
3.2.3 Crack Pattern

Now we compare the crack patterns obtained in the analysis with the experimental observations. We select two load cycles for this purpose. Load cycle of peak displacement of 5 mm and 3 mm displacement (load step 16 and load step 26) and final load cycle at 14 mm displacement (load step 151 and load step 181). We select output item Ecw1 in the result browser [Fig. 107] and set scales in the property window [Fig. 108] for better visualization. We use a maximum value of 0.05 mm for load step 16 and load step 26 and a maximum value of 0.5 mm for load step 152 and load step 182. The development of cracks in the experiment is captured by the numerical analysis [Fig. 110 to 114].

![Figure 107: Result browser](https://dianafea.com)

![Figure 108: Property window - contour plot settings](https://dianafea.com)
Figure 109: Experimental crack pattern at 5 mm load cycle (2.5 mm displacement)[1]

Figure 110: Experimental crack pattern at 5 mm load cycle (3 mm displacement)

Figure 111: Experimental crack pattern at 5 mm load cycle (-3 mm displacement)
Figure 112: Experimental crack pattern at 15 mm load cycle (13.84 mm displacement)[1]

Figure 113: Experimental crack pattern at 5 mm load cycle (14 mm displacement)

Figure 114: Experimental crack pattern at 5mm load cycle(-14 mm displacement)
3.3 Full Analysis Results

In this section results from a full analysis with load steps 1(3) -1(6) 1(3) 1(5) -1(10) 1(5) 1(10) -1(20) 1(10) 1(4) -1(8) 1(4) 1(12) -1(24) 1(12) 1(15) -1(30) 1(15) 1(7) -1(14) 1(7) 1(20) -1(40) 1(20) 1(30) -1(60) 1(30) 1(40) -1(80) 1(40) 1(50) -1(100) 1(50) 1(50) -1(100) 1(50) 1(50) -1(100) 1(50) are presented.

3.3.1 Hysteresis Curve

An early failure is obtained in the numerical analysis when compared with the experiment as seen in Figure 115. Therefore there is a slight under prediction of the capacity of the column.

![Hysteresis Curve](https://dianafea.com)

Figure 115: Hysteresis curve - comparison[1]
3.3.2 Failure Mode

The heavy cracking and spalling of concrete at the base on the column after the final load cycle in the experiment [Fig. 116] can be visualized in the analysis with a plot of principal strain $E_1$ at the final load step [Fig. 117].

![Figure 116: Crack pattern after 50 mm displacement - experiment[1]](image1)

![Figure 117: Principal strain $E_1$ after 15 mm displacement - analysis](image2)
A plot of reinforcement stresses and strains over the deformed mesh shows yielding and buckling of the longitudinal bars in the column [Fig. 118] [Fig. 119].

Figure 118: Reinforcement stress and deformation after 15 mm displacement - analysis

Figure 119: Reinforcement stress and deformation after -15 mm displacement - analysis
Appendix A  Additional Information

Folder: Tutorials/ConcreteColumn

Number of elements \( \approx 1290 \)

Keywords:
- ANALYS: geomet nonlin physic.
- CONSTR: suppor tying.
- ELEMEN: bar bondsl cable chx60 chx69i circle cl9tr cq40s cq48i ct36i ctp45 curved interf reinfo shell solid struct truss.
- LOAD: deform elmen face force weight.
- MATERI: crack elasti harden isotro jsce maekcc menegp plasti rotati soften strain totstr.
- OPTION: direct lagran linese newton regula total units.
- 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.