Concrete Slab under Cyclic Transverse Load
Outline

1 Description  ................................................................. 3
  1.1 Case Study ........................................................................ 3
  1.2 Geometry and Reinforcement Layout .............................. 4
  1.3 Material Properties ........................................................ 5
  1.4 Modeling Strategy .......................................................... 6

2 Finite Element Model .......................................................... 7
  2.1 Geometry ................................................................. 9
  2.2 Properties ..................................................................... 14
    2.2.1 Concrete Slab ......................................................... 14
    2.2.2 Reinforcement ......................................................... 18
  2.3 Boundary Conditions ..................................................... 23
    2.3.1 Symmetry .......................................................... 23
    2.3.2 No-tension Interface .............................................. 24
  2.4 Loads ......................................................................... 28
  2.5 Mesh .......................................................................... 31

3 Structural Nonlinear Analysis .............................................. 32
  3.1 Commands ................................................................. 32
  3.2 Results ...................................................................... 35
    3.2.1 Force vs. Displacement Diagram ............................ 35
    3.2.2 Displacements ...................................................... 37
    3.2.3 Interface Traction .................................................. 39
    3.2.4 Plastic Strain in Reinforcements ............................. 41

Appendix A Interface Geometry – Definition of the Element Local Axes 44

Appendix B Additional Information 45
1 Description

1.1 Case Study

The purpose of this tutorial is to model and analyze a reinforced concrete (RC) slab under cyclic transverse loading in DIANAIE [Fig. 1 to 3].

Figure 1: Test setup

Figure 2: Experimental response of the RC slab under cyclic loading

Figure 3: Crack patterns at the bottom (left) and top (right) faces of the RC slab at the end of the loading cycles

1Maekawa et al., *Nonlinear mechanics of reinforced concrete*, 2003
1.2 Geometry and Reinforcement Layout

The squared slab of side 1.4 m is reinforced at the upper and lower faces with straight steel bars in the $X$ and $Y$ directions as shown in Figure 5. Three loading cycles are applied at the center of the slab through a circular loading plate with a diameter of 0.24 m.

The slab is supported at the perimeter on a no-tension bedding (the plate can lift up at the boundaries during loading).

![Figure 4: Geometry of the model](https://dianafea.com)

![Figure 5: Reinforcement layout](https://dianafea.com)
1.3 Material Properties

The constitutive behavior of the materials is defined according to the Japanese Specifications for Concrete Structures JSCE \(^2\), as in the analysis presented in Maekawa et al. (2013)\(^3\). The total strain based crack model with rotating crack orientation is used for the concrete. The tensile behavior after cracking is defined according to the JSCE tension stiffening model while the compressive behavior to the Maekawa cracked concrete curve. The material properties are listed in Table 1.

Steel reinforcements are modeled as perfectly plastic. The hysteric response is defined according to the JSCE Standard Specifications for Concrete Structures 2012\(^2\). The material properties of the steel are also listed in Table 1.

<table>
<thead>
<tr>
<th><strong>Table 1: Material properties</strong></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>Density</td>
</tr>
<tr>
<td>Tensile strength</td>
</tr>
<tr>
<td>Compressive strength</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Steel</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Yield stress</td>
</tr>
</tbody>
</table>

---


\(^3\)Maekawa et al., *Nonlinear mechanics of reinforced concrete*, 2003
1.4 Modeling Strategy

The model of the RC slab created in DIANAIE is as similar as possible to the one presented in Maekawa et al. (2013)[1] and illustrated in Figure 6. For this purpose, the following considerations were addressed:

- only half of the model is considered due to symmetry
- the mesh, is the same as in Maekawa et al. (2013)[1], is generated by imprinting lines on the RC slab (without using the automatic mesh generator)
- the RC slab is modeled with quadratic continuum-based shell elements
- according to Maekawa et al. (2013)[1], 11 integration points are used through the thickness of the shell elements for the integration of stresses in the RC slab
- reinforcing steel bars are modeled using grid reinforcements
- since the plate can lift up at the boundaries, no-tension interface elements are used along the perimeter (green dashed lines in Figure 6);
- during the cyclic loading, vertical displacements are applied to the nodes located in the area of the loading plate (blue nodes in Figure 6);
- in order to reproduce the results presented in Maekawa et al. (2013)[1], incremental displacements $|\Delta u_z| = 0.01$ m are applied at each step as shown in Figure 7.

![Figure 6: Characteristics of the finite element mesh (dimensions in centimeters)](https://dianafea.com)

![Figure 7: Displacement steps applied at the center of the RC slab](https://dianafea.com)
2 Finite Element Model

For the modeling session we start a new project. As mentioned in the previously, we use quadratic elements.

Figure 8: New project dialog
We choose meter for the unit length, kilogram for the mass and newton for force.

**Figure 9: Geometry browser**

**Figure 10: Property Panel - units**
2.1 Geometry

Due to symmetry, we consider only half of the RC slab and its mid plane is located on the XY plane (Z = 0). We model the slab with a rectangular polygonal sheet [Fig. 12].

![Add polygon sheet](https://dianafea.com/fig11)

Figure 11: Add polygon sheet

![Isometric view 1](https://dianafea.com/fig12)

Figure 12: Isometric view 1
The rectangular sheet is copied and vertically translated four times to create the grid reinforcements. We create the first layer by translating the rectangular sheet 0.025 m in the vertical direction using the command *Array copy*. We repeat this command three more times using a vertical translation equal to 0.035, -0.025 and -0.035 m [Fig. 14].

Figure 13: Array copy 1

Figure 14: View of the model - rectangular sheets
Then, we rename the shapes as following:

- **Concrete Slab 1** ➔ **Reinforcement Top xDir**
- **Concrete Slab 2** ➔ **Reinforcement Top yDir**
- **Concrete Slab 3** ➔ **Reinforcement Bottom xDir**
- **Concrete Slab 4** ➔ **Reinforcement Bottom xDir**
In this example the finite element mesh described in Figure 6 is created by imprinting straight lines on the rectangular polygonal sheet representing the RC slab. The coordinates of the points of the lines are listed in Table 2.

**Table 2: Initial and final coordinates of the mesh lines**

<table>
<thead>
<tr>
<th>Name</th>
<th>Point 1</th>
<th>Point 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>mesh line 2</td>
<td>(-0.7, 0.41, 0) m</td>
<td>(0.7, 0.41, 0) m</td>
</tr>
<tr>
<td>mesh line 3</td>
<td>(0, 0, 0) m</td>
<td>(0, 0.7, 0) m</td>
</tr>
<tr>
<td>mesh line 4</td>
<td>(-0.41, 0, 0) m</td>
<td>(-0.41, 0.7, 0) m</td>
</tr>
<tr>
<td>mesh line 5</td>
<td>(-0.12, 0, 0) m</td>
<td>(-0.12, 0.7, 0) m</td>
</tr>
<tr>
<td>mesh line 6</td>
<td>(0.12, 0, 0) m</td>
<td>(0.12, 0.7, 0) m</td>
</tr>
<tr>
<td>mesh line 7</td>
<td>(0.41, 0, 0) m</td>
<td>(0.41, 0.7, 0) m</td>
</tr>
</tbody>
</table>

Figure 17: Add *mesh line 1*
We now imprint the mesh lines on the RC slab. In order to facilitate the procedure, we first hide the reinforcement layers and then we imprint the mesh lines on the RC slab.

**Geometry browser** ➔ Geometry ➔ Shapes ➔ [Reinforcement Top xDir ➔ Hide](https://dianafea.com)

< Repeat for all reinforcement layers. >

**Main menu** ➔ Geometry ➔ Modify ➔ Shape projection  
[Fig. 18]

**Main menu** ➔ Viewer ➔ Viewpoints ➔ Isometric view 1  
[Fig. 19]

---

Figure 18: Shape projection  
Figure 19: Slab with the imprinted lines
2.2 Properties

2.2.1 Concrete Slab

We assign the material and geometry properties mentioned in Section 1.4 to the RC slab. As we are not using the default number of integration points, a new element data set must be added. We use 11 integration points through the thickness of the shell elements in order to precisely integrate the stresses.

---

**Figure 20:** RC slab properties

**Figure 21:** Add new material - *Concrete*

**Figure 22:** Material properties for concrete

---

*Concrete Slab under Cyclic Transverse Load | https://dianafea.com*
Figure 23: Material properties for concrete

Figure 24: Material properties for concrete
According to Figure 4, the thickness of the concrete slab is set equal to 0.1 m [Fig. 26].
We set the number of integration points to be used through the thickness of the shell elements.
2.2.2 Reinforcement

We assign material and geometry properties to the reinforcement layers corresponding to the steel bars along the X direction. We create a new material Steel with the properties listed in Table 1. We create a new geometry for the reinforcement in the X direction. We start with the property assignment to the top reinforcement.

Figure 30: Assign reinforcement properties

Figure 31: Add new material - Steel

Figure 32: Edit material
Figure 33: Edit geometry - X direction
We assign the same material and geometry properties to the bottom reinforcement in the $X$ direction.

<Select the correspondent reinforcement layer in the Geometry browser >

Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties  📅 [Fig. 34]

![Image of reinforcement properties settings]

Figure 34: Assign reinforcement properties
Analogously, we define the properties to the reinforcement layers corresponding to the bars along $Y$ direction. We use the same material model Steel. We create a new geometry for the reinforcement in the $Y$ direction. We start with the property assignment to the top reinforcement.

**Main menu** ➔ Geometry ➔ Assign ➔ Reinforcement properties ➔ [Fig. 35]

**Reinforcement properties** ➔ Geometry ➔ Edit geometry ➔ [Fig. 36]

![Figure 35: Assign reinforcement properties](https://dianafea.com)

![Figure 36: Edit geometry - $Y$ direction](https://dianafea.com)
We assign the same material and geometry properties to the bottom reinforcement in the $Y$ direction.

Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties  📌 [Fig. 37]
2.3 Boundary Conditions

2.3.1 Symmetry

We now apply the symmetry boundary condition along the side of the RC slab at $Y = 0$. Along this edge we constrain the displacements in the $Y$ direction and the rotations around the $X$ axis.

Main menu ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 39] [Fig. 40]

---

**Figure 38:** Attach Support

**Figure 39:** Edge selection

**Figure 40:** Geometry view - symmetry constraints
2.3.2 No-tension Interface

We apply a no-tension boundary conditions along the outer perimeter of the RC slab. As mentioned in Section 1.4, this is achieved by adding a boundary interface to these sides. Therefore, in DIANAIE, we assign to the edges on the slab perimeter a set of boundary interfaces with material properties [Fig. 43 to 44], geometry properties [Fig. 46] and a set of boundary conditions [Fig. 47].

Figure 41: Boundary interface properties

Figure 42: Edge selection
We define the material properties for the nonlinear interface.
As illustrated in Figure 46, for the definition of the interface geometry we assume the concrete slab being on a support of thickness 0.01 m. To define correctly the no-tension behavior of the interface, the direction vector parallel to the shell plane (that determines the local axes of the interface imaginary wall) is set equal to [0 0 -1]. This implies that the supported edge of the boundary interface is located below the plate edge. Thus, the plate is resting (not hanging) on the supports. In this case, the vector parallel to the shell plane defines the compression direction in the boundary interface.

4See Appendix A for a detailed explanation on how to specify the interface local axes.
We define the vertical supports of the boundary interface.

**Main menu ➔ Geometry ➔ Assign ➔ Add supports**  [Fig. 47]

Figure 47: Attach support

Figure 48: Geometry view - vertical supports
2.4 Loads

According to Maekawa et al. (2013)[1], a cyclic vertical displacement is applied to the nodes laying on the loading plate imprint (i.e., all nodes having a distance of 0.12 m from O in Figure 6). To reproduce the load vs. displacement curve presented by Maekawa [1] [Fig. 2], the prescribed vertical displacement is applied to one single node (in this case we choose node O), while the remaining nodes are tied to it using the Master-Slave method.

This is performed as follows:

- constrain the vertical translation of node O at the center of the RC slab (coordinates (0, 0, 0) m)
- apply a vertical displacement equal to -0.001 m, referred to as Prescribed deformation, to node O
- impose the vertical displacement of all nodes located at 0.12 m from node O at (0, 0, 0) m. Since we use quadratic elements (that have middle nodes) for the RC slab, the vertical displacement is imposed to the element sides sharing node O [Fig. 54]

Figure 49: Add supports

Figure 50: Node selection (for better visualization, symmetry boundary constraints are hidden)

Figure 51: Add load
Figure 52: Add tyings

Figure 53: Add tyings - master node

Figure 54: Add tyings - slave lines
Also the deadweight of the RC slab is considered in the analysis.

**Figure 55: Global load - dead weight**
2.5 Mesh

We generate the mesh. As we defined the mesh previously with the imprint of the lines on the slab, we now choose only one division for the edges in the slab.

**Figure 56: Mesh properties**

**Figure 57: Finite element mesh**
3 Structural Nonlinear Analysis

3.1 Commands

We perform a quasi-static nonlinear analysis.
We define the load steps, convergence criteria and solution method. From the load vs. displacement curve [Fig. 6] we choose to use the following load steps as *User specified sizes* [Fig. 62]: 1.4(-1.3) 1.5(-1.5) 1.12(-1.9) 1.17(-1.10) where the notation $n(m)$ means that a load increment $n$ for the load set *Prescribed deformation* ($\Delta u_z = -0.001$ m [Fig. 51]) is repeated $m$ times consecutively.

We use the default settings for the convergence norms [Fig. 63].
We select the desired output results - displacements, reaction forces, stresses at the interfaces and plastic strains in the reinforcements - and we run the analysis.

**Analysis browser** ➔ Nonlin ➔ Structural nonlinear ➔ Output ➔ Edit properties

Properties - OUTPUT ➔ Modify ➔ Results Selection

Main menu ➔ Analysis ➔ Run selected analysis

---

**Figure 64**: Results properties

**Figure 65**: Results selection

**Figure 66**: Output properties
3.2 Results

3.2.1 Force vs. Displacement Diagram

We do a force vs. displacement diagram of node O, located at the center of the plate and compare it with the experimental results presented in Maekawa (2013)[1].

Figure 67: Results table

Figure 68: Show table
Once the table in Figure 68 is pasted in Excel, multiply the vertical displacements TDtZ by -1 and the vertical reaction force FBZ by -2. Using these results, it is possible to make the load vs. displacement graph [Fig. 69].

The blue markers in Figure 69 highlight the load cases investigated in the next sections.
3.2.2 Displacements

We create a contour plot of vertical displacements at load step 12, 17, 55 and 65 which corresponds to 60%, 3%, 100% and 7% of peak load, respectively. To compare deformed configurations and contour plots of the RC plate at different load steps, we hide the reinforcement layers and define common output view settings: absolute deformation scaling and contour plot limits. The limits of the contour plot [Fig. 72] are set to the minimum and maximum value that TDtZ takes throughout the analysis (-0.021 and 0.0045 m, respectively).

Figure 70: Mesh browser - hide reinforcements

Figure 71: Results browser

Figure 72: Output properties
We show the contour plot of the vertical displacements at the different load steps [Fig. 73 to 76] by changing the case in the output browser [Fig. 71].

Figure 73: Vertical displacement - Load step 12 (60% of peak load)

Figure 74: Vertical displacement - Load step 55 (100% of peak load)

Figure 75: Vertical displacement - Load step 17 (3% of peak load)

Figure 76: Vertical displacement - Load step 65 (7% of peak load)
3.2.3 Interface Traction

We show the diagram of stresses along the interface elements at the boundary of the plate. To understand where the interface opens, we display the interface stress component along their local $y$ axis. To facilitate the comparison of results within different load cases, we use common view settings for the diagrams. The limits of the line diagrams [Fig. 78] are set to the minimum and maximum value taken by STNy during the analysis (-4e+07 and 0 N/m², respectively).

![Results browser](https://dianafea.com)

**Figure 77: Results browser**

**Figure 78: Diagram properties**
We present the line diagrams of the interface tractions at different load steps [Fig. 79 to 82] by changing the case in the output browser [Fig. 77]. The load cases are the same as presented in Figure 73 to Figure 76 and Figure 69. The portions of the line boundary interfaces that do not show a diagram indicate that the RC plate is lifting up in those regions as suggested by the deformed configuration.
3.2.4 Plastic Strain in Reinforcements

We investigate the plastic strain in the reinforcement grids at the peak load. We start from the visualization of $E_{pXX}$ in the top reinforcement layer with the bars aligned along the $X$ axis.

---

**Mesh browser**

Mesh browser $\rightarrow$ Mesh $\rightarrow$ Shapes $\rightarrow$ Concrete Slab $\rightarrow$ Hide [Fig. 83]

**Mesh browser**

Mesh browser $\rightarrow$ Mesh $\rightarrow$ Shapes $\rightarrow$ Reinforcement Top xDir $\rightarrow$ Show [Fig. 83]

**Results browser**

Results browser $\rightarrow$ Case $\rightarrow$ Load-step 55, Load-factor 21.000, Prescribed deformation [Fig. 84]

**Results browser**

Results browser $\rightarrow$ Nonlin $\rightarrow$ Output $\rightarrow$ Reinforcements results $\rightarrow$ Reinforcement Plastic Strain $\rightarrow$ $E_{pXX}$ [Fig. 84]

**Property Panel**

Property Panel $\rightarrow$ Result $\rightarrow$ Contour plot settings $\rightarrow$ Color scale limits $\rightarrow$ Specified values [Fig. 85]

**Property Panel**

Property Panel $\rightarrow$ Result $\rightarrow$ Contour plot settings $\rightarrow$ Specified values $\rightarrow$ Minimum value $\rightarrow$ -0.021 [Fig. 85]

**Property Panel**

Property Panel $\rightarrow$ Result $\rightarrow$ Contour plot settings $\rightarrow$ Specified values $\rightarrow$ Maximum value $\rightarrow$ 0.0045 [Fig. 85]

**Property Panel**

Property Panel $\rightarrow$ Result $\rightarrow$ Diagram plot settings $\rightarrow$ Specified values $\rightarrow$ Minimum value $\rightarrow$ -2.1e-02 [Fig. 85]

**Property Panel**

Property Panel $\rightarrow$ Result $\rightarrow$ Diagram plot settings $\rightarrow$ Specified values $\rightarrow$ Maximum value $\rightarrow$ 4.5e-03 [Fig. 85]

---

*Figure 83: Mesh browser - hide elements*

*Figure 84: Results browser*

*Figure 85: Contour plot properties*
To display the plastic strain in a single reinforcement layer at the time [Fig. 86 to 89], we show/hide the reinforcements from the mesh browser. Only the bottom reinforcement in $X$ direction [Fig. 88] shows plastic strain because: i) we are considering the strain component in $X$ direction (not present in the reinforcements in $Y$ direction) and ii) the stresses at the top reinforcement in the $X$ direction are smaller due to the loading conditions.
We also investigate the plastic strain field in the $Y$ direction $\varepsilon_{pYY}$ at peak load. Similarly to what observed for $\varepsilon_{pXX}$, only the bottom reinforcement in the $Y$ direction [Fig. 93] shows plastic strain because: i) we are considering the strain component in $Y$ direction (not present in the reinforcements in $X$ direction) and ii) the stresses at the top reinforcement in the $Y$ direction are smaller due to the loading conditions.
Appendix A  Interface Geometry – Definition of the Element Local Axes

For the modeling of a no-tension line interface, it is important to correctly define the local axes of the element. For line structural shell interfaces (used in this tutorial), the default local axis ($x$ axis) coincides with the direction of the edge of the line interface [Fig. 94]. Then, the user should define one of the two remaining local axes in the geometry property [Fig. 46]: either the direction vector parallel to shell plane of the interface (local $y$ axis) or the direction vector normal to shell plane ($z$ axis). Hence, the third local axis is automatically calculated in DIANA with the “right-hand” rule.

For line structural shell interface on boundary we advice to define the direction vector parallel to shell plane ($y$ axes) because this is easier: the $y$ axis represents the direction of getting tension or compression in the interface elements.

DIANA starts with the numbering of the nodes of the interface at the edge selected for the line interface. For boundary interface elements this is always the edge belonging to the shell (slab) not the supported edge. So node 1, 2 and 3 of the interface are on the slab edge. The local $y$ axis should always point to the opposite edge of the interface, the supported edge (node 3, 4 and 5). The slab is resting on the supports (not hanging), so the local $y$ axis of the line shell interface element should be defined in this example in negative global $Z$ direction [0 0 -1] [Fig. 94].

When choosing the direction vector parallel to the shell plane of interface and defining its direction pointing from construction towards the supports, you always get the right orientation of the line structural shell elements. This is independent on the direction of the selected edge of the interface. Only the direction vector of the local $x$ and $z$ axis will be different [Fig. 95].

![Figure 94: Local axes in for boundary line interface (first case)](image1)

![Figure 95: Local axes in for boundary line interface (second case)](image2)
Appendix B  Additional Information

Folder: Tutorials/CyclicLoadedSlab

Number of elements \( \approx 20 \)

Keywords:
- ANALYS: nonlin physic.
- CONSTR: suppor tying.
- ELEMEN: cl24i cq40s curved grid interf reinfo shell struct taper.
- LOAD: deform weight.
- MATERI: crack elasti harden isotro jsce jscerh maekcc plasti rotati soften strain totstr.
- OPTION: direct newton regula units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: displa force green plasti 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.