Design and Stiffness Adaptation Analysis of a Reinforced Concrete Slab
## Outline

1 **Description**  

2 **Finite Element Model**  
   2.1 Geometry  
   2.2 Properties  
      2.2.1 Concrete Slab  
      2.2.2 Grid Reinforcement  
      2.2.3 Composed Surface  
      2.2.4 Load and Support Plates  
      2.2.5 Interfaces between Plates and Slab  
   2.3 Boundary Conditions  
   2.4 Loads  
   2.5 Mesh  

3 **Design Checking Analysis**  
   3.1 Commands  
   3.2 Results  

4 **Stiffness Adaptation Analysis**  
   4.1 Properties  
      4.1.1 Concrete Slab  
      4.1.2 Grid Reinforcement  
   4.2 Loads  
   4.3 Mesh  
   4.4 Commands  
   4.5 Results  

**Appendix A Additional Information**
1 Description

This tutorial presents a design and stiffness adaptation analysis of a reinforced concrete (RC) slab. The case study is a classical benchmark tested in laboratory by McNeice (1967) in London University\(^1\) and used by many authors to verify and validate numerical models and computation methods; some examples are Lin and Scordelis (1975)\(^2\), Crisfield (1982)\(^3\), Polak and Vecchio (1993)\(^4\).

The benchmark consists on a corner supported two-way squared RC slab tested with a central point load until failure. The characteristics of the slab are represented in Figure 1: dimensions of 914.4 mm (36 in) × 914.4 mm (36 in) and 44.45 mm (1.75 in) thick; grid reinforcement of 282 mm\(^2\)/m in the two orthogonal directions at an effective depth of 33.3 mm (1.31 in). A photo of the experimental test is presented in Figure 2.

---

\(^1\)McNeice, Elastic-plastic bending of plates and slabs by the Finite Element Method, 1967

\(^2\)Lin and Scordelis, Nonlinear analysis of reinforced concrete shells of general form, 1975

\(^3\)Crisfield, Variable step-length for nonlinear structural analysis, 1982

\(^4\)Polak. and Vecchio, Nonlinear analysis of reinforced-concrete shells, 1993

---

![Figure 1: Geometry of the McNeice slab (dimensions in mm) [1]](image-url1)

![Figure 2: Experimental test by McNeice (1967) [1]](image-url2)
We study this slab using two different approaches:\(^5\):

1. design checking analysis for load levels correspondent to service and ultimate states
2. stiffness adaptation analysis with incremental load until ultimate limit state

The following considerations were taken into account to simulate this slab:

- we model one quarter of the slab using symmetry support conditions [Fig. 1]
- we use a 3D finite element model with solid elements for the slab and embedded grid reinforcements [Fig. 3]
- for the Design Check analysis we need to define composed surface elements to calculate the distributed bending moments and shear forces
- for the Stiffness Adaptation analysis we model the support and load plates and the correspondent interfaces
- for concrete cracking consideration we use the smeared rotating crack approach

\(^5\)A nonlinear analysis until failure is presented in the Verification Report that is part of the User’s Manual. A detailed analysis is made to compare with the experimental results for verification of DIANA calculations.
Failure of the slab occurred in the experimental test for a load level around 11 kN. This value is considered in the analysis as failure (or ultimate) state. The service load is considered as the failure load divided by the factor 1.5 which corresponds to 7.3 kN.

The mechanical properties considered for concrete and steel are presented in Table 1.

<table>
<thead>
<tr>
<th></th>
<th>Concrete</th>
<th></th>
<th>Reinforcement steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus $E$</td>
<td>28.6E3 N/mm²</td>
<td>Young's modulus $E$</td>
<td>200E3 N/mm²</td>
</tr>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.15</td>
<td>Yielding strength $f_{ym}$</td>
<td>350 N/mm²</td>
</tr>
<tr>
<td>Compressive strength (average) $f_{cu}$</td>
<td>38 N/mm²</td>
<td>Ultimate strength $f_{um}$</td>
<td>400 N/mm²</td>
</tr>
<tr>
<td>Compressive strength (characteristic) $f_{ck}$</td>
<td>30 N/mm²</td>
<td>Ultimate strain $\varepsilon_{um}$</td>
<td>0.005</td>
</tr>
<tr>
<td>Tensile strength (average) $f_{tm}$</td>
<td>2.9 N/mm²</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ultimate tensile strain</td>
<td>0.001</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
2 Finite Element Model

For the modeling session we start a new project. We use quadratic hexagonal elements.

![New project dialog](https://dianafea.com)
We use mm as unit length, newton for force and ton for mass [Fig. 6].

---

**Geometry browser** → Reference system → Units  [Fig. 5]

**Property Panel** [Fig. 6]

---

**Figure 5:** Geometry browser

**Figure 6:** Property panel - units
2.1 Geometry

We simulate only one quarter of the slab by making use of the symmetry conditions. We define all the geometry needed for both the design and stiffness adaptation analysis.

We start the model by creating a sheet for the bottom surface of the slab.
We copy and move this sheet (using the array-copy tool) for the location of the reinforcement. The grid reinforcement is located at 11.15 mm from the bottom surface of the slab. We array copy the bottom surface of the slab again to the center of the slab. This will be the composed surface which is required to compute the distributed moments and section forces used in the design checks. In the Design analysis the composed surface should be located in the center of the slab to get the cross sectional forces and moments.

Figure 9: Array copy sheet for reinforcement grid

Figure 10: Array copy sheet for composed surface
We rename the sheets.

Figure 11: Geometry browser - copied sheets
Figure 12: Geometry browser - renamed sheets
Figure 13: View of the model - Sheets
We now extrude the Slab sheet 44.45 mm in the Z direction.

Main menu ➔ Geometry ➔ Modify ➔ Extrude  [Fig. 14]

Figure 14: Geometry - extrude sheet

Figure 15: View of the mode - Slab
We add two blocks: one for the support block and one for the load block.

**Main menu** ➔ Geometry ➔ Create ➔ Add block ![Fig. 16] ![Fig. 17]

**Figure 16**: Add block *Support pad*

**Figure 17**: Add block *Load pad*

**Figure 18**: Support and load blocks
We create a vertex located in the middle of the support plate and imprint this on the support block. This vertex will be supported in the $Z$ direction.

Figure 19: Geometry - add vertex

Figure 20: Geometry - imprint
2.2 Properties

2.2.1 Concrete Slab

To model the concrete we use the Concrete design codes class available in DIANA. We use the Eurocode 2 EN1992-1-1 with the concrete class C30/37 and we include total strain crack model. We use structural solids.

![Main menu](https://dianafea.com)

- Geometry
- Assign
- Shape Properties
- Shape Properties
- Material
- Add material
- Edit material

![Shape Properties](https://dianafea.com)

- Create
- Close

![Target selection](https://dianafea.com)

- Slab

![Shape Properties](https://dianafea.com)

- Element class
- Structural Solids
- Material
- Concrete
- Geometry
- Data

![Add new material](https://dianafea.com)

- Name: Concrete
- Class: Concrete design codes
- Material model: Eurocode 2 EN 1992-1-1

- Aspects to include:
  - Total Strain crack model
  - Shrinkage
  - Rayleigh damping
  - Additional dynamic 3D line mass
  - Strength reduction

- Creep
- Young hardening concrete
- Additional dynamic surface mass

![Concrete material properties](https://dianafea.com)

- Concrete type
- Aggregate type
- Cement type

- Material safety factors (SSM):
  - Young's modulus: 30000 N/m²
  - Poisson ratio ν: 0.2
  - Thermal expansion coefficient α_t: 10⁻⁶/°C
  - Density ρ: 2.4-97, 10⁹ N/m³
  - Mean uniaxial tensile strength σ_y: 2000 MPa
  - Mean compressive strength f_c: 30 MPa
2.2.2 Grid Reinforcement

We now define the properties of the reinforcement. We use embedded grid reinforcement. We define a new material for the steel using non-hardening Von Mises plasticity with the properties defined in Table 1.

For the stiffness adaptation analysis we will define hardening.
The grid of reinforcement has 282 mm\(^2/m\) which we model with bars of 6 mm diameter spaced of 100 mm in the two orthogonal directions. For the Design Check analysis we need to set that the reinforcement is considered in design.

![Edit geometry - Grid](https://dianafea.com)
2.2.3 Composed Surface

We assign the properties to the composed surface. This composed surface, related to the grid reinforcement, is needed to computed the distributed moments and section forces used in the design checks. The concrete properties are listed in Table 1.

[Figure 28: Assign properties to composed surface]

[Figure 29: Add new material - Composed]

[Figure 30: Edit material - Composed]
We define a new geometry for the composed surface with the total thickness of the slab (see Figure 1). This means that the stresses that are inside this thickness are integrated to determine the distributed moments and section forces.
2.2.4 Load and Support Plates

We define the material properties for the load and support plates. We use structural solids element class and we don’t need to define geometry and data.

**Main menu ➔ Geometry ➔ Assign ➔ Shape Properties [Fig. 32]**

**Shape Properties ➔ Material ➔ Add material [Fig. 33] ➔ Edit material [Fig. 34]**

---

**Figure 32: Assign pad properties**

**Figure 33: Add new material - Plate**

**Figure 34: Material properties - Plate**
2.2.5 Interfaces between Plates and Slab

We define linear elastic interfaces between the plates (load and support pads) and the slab with a high value for the normal stiffness and low value for the shear stiffnesses. We don’t need to define geometry and data.

Figure 35: Assign connection properties

Figure 36: Interface - add new material

Figure 37: Material properties - Interface
2.3 Boundary Conditions

Due to the symmetry condition we restrict the translations in $X$ and $Y$ directions in the symmetry faces of the slab and loading plate. We fix the translations in the $Y$ direction in the faces symmetric along the $X$ axis.

Main menu ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 38]

<Select the faces symmetric along $X$ axis>

Figure 38: Attach support

Figure 39: View of the model - symmetry support $X$
We fix the translations in the $X$ direction in the faces symmetric along the $Y$ axis.

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

<Select the faces symmetric along $Y$ axis>

Figure 40: Add face support

Figure 41: View of the model - symmetry support $Y$
We add a support in the vertex located at the middle of the support plate to fix the transverse translations (in the Z direction).

Main menu ➔ Geometry ➔ Assign ➔ Add supports [Fig. 42]

Figure 42: Add point support

Figure 43: View of the model - point support
2.4 Loads

We consider the point load in the middle of the slab as in the experimental test\(^7\). The failure in the experiment occurred for a load level of approximately 11 kN. We use this value as reference of the Ultimate Limit State (ULS). For the Serviceability Limit State (SLS) we consider the load level of 7.3 kN (determined as 11 kN divided by the factor 1.5).

For the design analysis we consider a total concentrated load of 7.3 kN relating to the service state. This corresponds to a point load of 1.825 kN in the quarter of the slab\(^8\).

\(^7\)Self-weight is not considered as the slab is tested in the vertical position being in this case not relevant for the structural response.

\(^8\)For the stiffness adaptation analysis we will change the applied load to 1 kN (to use this value as increments), which corresponds to 0.25 kN in the quarter of the slab.
2.5 Mesh

We use the same mesh for all the analysis. We use an element size of 14.2 mm. Finally we generate the mesh.

---

**Main menu ➔ Geometry ➔ Assign ➔ Mesh properties [Fig. 46]**

**Main menu ➔ Geometry ➔ Generate mesh [Fig. 47]**
3 Design Checking Analysis

3.1 Commands

We add a new analysis for Checking Design. This application checks the cross-section bending moments and forces resulting from a linear analysis against the condition that the failure stresses in the reinforcement and concrete are in equilibrium. We consider averaging of reinforcement moments and shear forces with the default options.

Main menu → Analysis → Add analysis  
Analysis browser → Analysis1 → Rename → Design

Analysis browser → Analysis1 → Add command → Checking design  
Analysis browser → Checking Design → Add command → Add average reinforcement moments and shear forces

< Rename (F2 or  ) the load combinations as SLS and ULS >  

Figure 48: Analysis browser  
Figure 49: Add command  
Figure 50: Analysis browser
We define the load combinations for the *Service Limit State* (SLS, load factor of 1.0) and for the *Ultimate Limit State* (ULS, load factor of 1.5).

**Analysis browser** ➔ **Design** ➔ **Checking design** ➔ **SLS** ➔ **Edit properties**  [Fig. 51]

**Analysis browser** ➔ **Design** ➔ **Checking design** ➔ **ULS** ➔ **Edit properties**  [Fig. 52]

---

Figure 51: Combination SLS

Figure 52: Combination ULS
We use the default design code specification.

Analysis browser → Design → Checking design → Design code specification → Edit properties  [Fig. 53]

Figure 53: Design code specification
We add two output blocks: for SLS and for ULS.
And we select the results we want for each result block. We start with the results for the SLS.

**Properties - OUTPUT** → **Device Properties**  
*Base name: SLS*  

![Figure 55: Properties output - OUT_SLS](image)

![Figure 56: Output base name SLS](image)
Now we select the desired results for the ULS. And we run the analysis.

**Figure 57:** Properties output - OUT_ULS

**Figure 58:** Output base name ULS
3.2 Results

We start with the unit checks (UC) for the reinforcement at SLS: the maximum diameter, maximum spacing and cover. If the values are less or equal than 1 it complies with the norm. We start with the diameter.

We want to see the grid of reinforcement so we show only the Grid in the mesh browser.

![Diagram showing mesh browser and results browser]

**Figure 59: Mesh browser**

**Figure 60: Results browser**

**Figure 61: Unity check maximum diameter x**

**Figure 62: Unity check maximum diameter y**
We check the maximum spacing. As the results are symmetric we only show the correspondent to the $x$ direction.

**Figure 63:** Results browser

**Figure 64:** Unity check maximum spacing $x$
And finally we check the cover. As the coverage results are not dependent on the loading they are listed in the 'Model data' case.

---

**Results browser**
- Case ➔ Model data [Fig. 65]
- OUT_SLS ➔ Reinforcement results ➔ Reinforcement UC Minimum Coverage NODES ➔ UCCOVx [Fig. 66]

---

**Figure 65: Results browser**

**Figure 66: Unity check minimum coverage x**
In the *Model data* we can also enter our own input of applied reinforcement.

**Results browser** → Case → Model data  [Fig. 67]

**Results browser** → OUT_SLS → Reinforcement results → Reinforcement Applied → ASAPLx  [Fig. 68]

**Figure 67**: Results browser

**Figure 68**: Reinforcement applied

Design
Model data
Reinforcement Applied (THICK) ASAPLx
min: 0.28mm max: 0.28mm
In the ULS we can now see the ratio between the required and applied reinforcement.

**Results browser** → Case → Combination ULS  [Fig. 69]
**Results browser** → OUT_SLS → Reinforcement results → Reinforcement Ratio Required Applied → ASRATx  [Fig. 70]
We can also see the required area per unit length. We observe a very localized increment of required reinforcement in the area of application of the load.

Design
Combination ULS
Reinforcement Required NODES (THICK) ASREQx
min: 0.02mm max: 0.22mm
In the composed surface elements, we can see the distributed bending moments. For that, we show the Composed.

**Design and Stiffness Adaptation Analysis of a Reinforced Concrete Slab**

**Mesh browser** → Mesh → Element sets → Composed → ![Check icon] Show  ![Fig. 73]

**Results browser** → Case → Combination ULS → ![Check icon] Show  ![Fig. 74]

**Results browser** → OUT_SLS → Element results → Distributed Bending Moment → Mxx → ![Fig. 75]
4 Stiffness Adaptation Analysis

4.1 Properties

For the Stiffness Adaptation analysis we consider the nonlinear material properties.

4.1.1 Concrete Slab

For concrete we use the total strain rotating crack model with linear ultimate strain based curve for tensile behavior and Thorenfeldt curve for compression with the properties defined in Table 1. For that we add a new material and assign it to the slab.

Figure 76: Assign slab properties

Figure 77: Add material for concrete

Figure 78: Edit material for concrete
Figure 79: Edit material for concrete

Figure 80: Edit material for concrete

Figure 81: Edit material for concrete
4.1.2 Grid Reinforcement

For steel we now consider hardening with the properties defined in Table 1.

Main menu ➔ Geometry ➔ Materials ➔ Edit material [Fig. 82] – [Fig. 84]

Figure 82: Material properties for steel

Figure 83: Material properties for steel

Figure 84: Strain - stress diagram
In the element geometry *Grid* we deselect the option of reinforcement grid considered in design.

**Main menu** ➔ **Geometry** ➔ **Element geometries** ➔ **Edit geometry**  

![Figure 85: Element geometry - Grid](https://dianafea.com)
4.2 Loads

We change the point load to -250 N in order to have a total load increment of 1 kN. This is to facilitate the interpretation of the analysis with increasing load until failure.

Figure 86: Attach load
4.3 Mesh

We defined the new properties in the geometry so we need to remesh the model using the same mesh properties, so we do not need to define them again.

Figure 87: Finite element model
4.4 Commands

We add a new analysis for the stiffness adaptation calculation.

Figure 88: Analysis browser - Stiffness Adaptation

Figure 89: Add command

Figure 90: Analysis browser
We define the analysis with 15 increments of load factor 0.5, which corresponds to increments of 0.5 kN, making a total of 7.5 kN (approximately the service limit load). We use the default maximum number of iterations (10).

![Figure 91: Analysis browser](image1)
![Figure 92: Load properties](image2)
With the output user selection we choose the results of displacements, strains, stresses and crack width. Finally we run the analysis.

**Analysis browser** ➔ **StiffnessAdaptation** ➔ **Stiffness adaptation analysis** ➔ **Output results** ➔ **Edit properties**

**Properties - OUTPUT** ➔ **Result** ➔ **User selection** ➔ **Modify**

**Main menu** ➔ **Analysis** ➔ **Run selected analysis**

---

**Figure 93:** Output properties

**Figure 94:** Selection of results
4.5 Results

We show the contour plot of displacements for the last load step of the analysis, correspondent to the service limit state. In order to better see the results we show only the Slab from the mesh.

Figure 95: Mesh browser

Figure 96: Results browser

Figure 97: Contour plot - displacements $\Delta tZ$
We now present crack widths for the same load level, correspondent to service state (SLS = 7.5 kN).

Figure 98: Results browser

Figure 99: Crack widths Ecw1 - SLS

Move the model to a position where we can see the bottom of the slab.

Stiffness Adaptation
Load step 15, Load-factor 7.5000
Crack-widths Ecw1
min.: -2.08e-04 mm max.: 2.11e-02 mm
We present the stresses in the reinforcement for these same load level.

The Stiffness Adaptation analysis determines the structural response with relatively high accuracy as it accounts for the nonlinear material properties. However, it is not capable to determine structural failure. For that we need to do a nonlinear structural analysis as presented in the Verification Report in the DIANA User’s Manual.
Appendix A  Additional Information

Folder: Tutorials/McNeiceSlab

Number of elements $\approx 3200$

Keywords:
- ANALYS: standap.
- CONSTR: suppor.
- ELEMEN: chx60 compos cq48i cq8cm grid interf reinfo solid struct taper.
- LOAD: force node.
- MATERI: concre crack elasti en1992 harden isotro linear multil plasti rotati soften strain thoren totstr vonmis.
- OPTION: direct units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy crkwdt displa force green princi reacti strain stress total.

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.