Fire under Concrete Slab
## Outline

1 **Description**  
1.1 Temperature Curves  
1.2 Material Properties  

2 **Finite Element Model**  
2.1 Geometry  
2.2 Properties  
2.3 Boundary Conditions  
\hspace{1em} 2.3.1 Thermal Boundary Conditions  
\hspace{1em} 2.3.2 Static Boundary Conditions  
2.4 Initial Temperatures  
2.5 Loads  
2.6 Mesh  

3 **Transient Heat Flow Analysis**  
3.1 Commands  
3.2 Results  

4 **Structural Nonlinear Analysis**  
4.1 Commands  
4.2 Results  
\hspace{1em} 4.2.1 Stresses in the Concrete Slab  
\hspace{1em} 4.2.2 Stresses in the Reinforcement Grid  
\hspace{1em} 4.2.3 Distributed Bending Moments in the Composed Surface  

Appendix A **Additional Information**
1 Description

This example shows how to perform a staggered heat flow-stress analysis on a quarter symmetric model of a reinforced concrete slab as presented in Figure 1. The effect of the fire is simulated by a temperature applied at the bottom and lateral faces. Material properties will be introduced with temperature dependencies on elastic parameters.

Figure 1: Geometry of the slab (dimensions in mm)
1.1 Temperature Curves

The temperature boundary conditions with time that are applied for the bottom face and for the lateral faces of the model of the slab are presented in Figure 2 and Figure 3, respectively. The development of ambient temperature with time is presented in Figure 4.

Figure 2: Temperature vs. time - bottom face
Figure 3: Temperature vs. time - lateral faces
Figure 4: Ambient temperature vs. time
1.2 Material Properties

The material properties for concrete and for reinforcement steel are presented in Table 1. The variation of Young’s modulus with the temperature for concrete and steel are presented in Figure 5 and Figure 6, respectively.

### Concrete
- Young’s modulus $E$ = 42000 N/m²
- Poisson’s ratio $\nu$ = 0.18
- Thermal expansion coefficient $\alpha$ = 1E-5
- Thermal conductivity = 1.32 W/m°C
- Thermal capacity = 2.3E+6 J/m³°C

### Reinforcement (Steel)
- Elastic modulus $E$ = 200000 N/m²
- Poisson’s ratio $\nu$ = 0.33
- Equivalent thickness (both directions) $t_{eq}$ = 0.48 mm

<table>
<thead>
<tr>
<th>Property</th>
<th>Concrete</th>
<th>Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus $E$</td>
<td>42000 N/m²</td>
<td>200000 N/m²</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.18</td>
<td>0.33</td>
</tr>
<tr>
<td>Thermal expansion coefficient $\alpha$</td>
<td>1E-5</td>
<td></td>
</tr>
<tr>
<td>Thermal conductivity</td>
<td>1.32 W/m°C</td>
<td></td>
</tr>
<tr>
<td>Thermal capacity</td>
<td>2.3E+6 J/m³°C</td>
<td></td>
</tr>
</tbody>
</table>

Table 1: Material properties

Figure 5: Young’s modulus of concrete vs. temperature

Figure 6: Young’s modulus of steel vs. steel
2 Finite Element Model

For the modelling session we start a new project.

Main menu ➔ File ➔ New  [Fig. 7]
We define the units to be used in the model. We choose the following units [Fig. 9]: newton for force, millimeter for length, second for time and celsius for temperature.
2.1 Geometry

We create the bottom layer of the slab [Fig. 12] that we then extrude [Fig. 11] to obtain the final geometry.

Figure 10: Add sheet
Figure 11: Extrude shape
Figure 12: Bottom layer of the slab
Figure 13: Extruded geometry of the slab
We create the grid reinforcement [Fig. 14]. For that we extract the bottom face of the slab and move it 15 mm inside the slab to the position of the reinforcement.
We create the composed surface [Fig. 19]. For that we duplicate the sheet created for the reinforcement and move it 60 mm to be in the center of the slab.

---

**Main menu** → Geometry → Shapes → Sheet 1 → Duplicate [Fig. 17]

**Main menu** → Geometry → Modify → Move shape [Fig. 18]

---

Figure 17: Geometry browser

Figure 18: Move shape

Figure 19: Geometry of the slab including composed surface and reinforcement grid
We create a line to use later for definition of the supports to fix the translations in the $Z$ direction. So we add a line beneath the slab and project it on the bottom surface of the slab.
2.2 Properties

We assign the properties to the slab [Fig. 24]. We use structural solid elements and we add a linear elastic isotropic material for concrete including thermal effects and heat flow. The parameters are described in Section 1.2.
We input the linear elastic properties, including the variation of the Young’s modulus with temperature.

Figure 26: Linear material properties

![Linear material properties](image)

Figure 27: Temperature-Young’s modulus variation

![Temperature-Young’s modulus variation](image)

Note: You can copy and paste the variation of Young modulus with respect to time from a Excel sheet
We input the heat flow properties: thermal conductivity is $1.32 \text{T/mm/s}^3\text{°C} = 1.32 \text{ W/m.°C}$, thermal capacity is $2.3 \text{T/mm}^2\text{°C} = 2300000 \text{ J/m}^3\text{°C}$.

Figure 28: Heat flow properties
We assign the properties to the grid reinforcement with the parameters described in Section 1.2 including the variation of the Young’s modulus with temperature [Fig. 32].

Select the correspondent reinforcement grid in the Geometry browser

Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties [Fig. 29]

Reinforcement properties ➔ Material ➔ Add material [Fig. 30] ➔ Edit material [Fig. 31] [Fig. 32]

< Edit the Temperature-Young’s modulus >
Figure 31: Linear elastic properties

![Figure 31: Linear elastic properties](https://dianafea.com)

Figure 32: Temperature - Young's modulus variation

![Figure 32: Temperature - Young's modulus variation](https://dianafea.com)

Note: You can copy and paste the variation of Young modulus with respect to time from an Excel sheet.
We also assign the geometry properties to the reinforcement grid, we use direct input in which we define the equivalent thickness of the grid in both local directions.

Reinforcement properties ➔ Geometry ➔ Edit geometry ➔ [Fig. 33]

Figure 33: Edit geometry
We continue by defining the material and geometry properties of the composed surface. We use this composed surface for output purposes, so we can have the distributed moments and shear forces in the slab.

Note that we do not need to fill the mechanical properties for the composed surface if we are not doing a design analysis.
2.3 Boundary Conditions

2.3.1 Thermal Boundary Conditions

We define the external temperature at the bottom face of the slab as a function of time. We set the external temperature as 1°C and we will later define the temperature curves.

[Fig. 38] [Fig. 39] [Fig. 40]

Figure 38: Thermal boundary conditions
Figure 39: Assign boundary interface at the bottom face of the slab
Figure 40: Boundary interface at the bottom face of the slab
We create a new material – *Boundary* – and introduce the heat flow properties.

Figure 41: Add new material

![Add new material](image)

We repeat the same operations to attach the thermal boundary conditions to the side and the top faces of the slab – “External Temperature Lateral” and “External Temperature Top” – respectively. The properties are the same. We need to define three different boundary conditions because we will have three correspondent temperature curves.

Figure 42: Material properties

![Material properties](image)
Time dependencies of thermal boundaries. We define the functions for temperature versus time for the bottom, lateral and top faces of the slab and assign them to the thermal boundaries created previously [Fig. 43]. We start by the temperature function for the bottom face.

Geometry browser → Boundary conditions → External Temperature Bottom → Edit time dependency [Fig. 43] [Fig. 44]

Figure 43: Geometry browser - thermal boundary conditions

Figure 44: Edit time dependent factors - bottom face

Note: when temperature quantities are changed (e.g. from Celsius to Kelvin) the External temperature parameter is converted (e.g. 1 °C = 274.15 K) but not the time-dependent factors. When you have defined a time-dependent factor of 20 with reference temperature 1 °C × 20 = 20 °C, changing to Kelvin the temperature will change to 274.15 × 20 = 548.3 K.
We repeat the process to define the time dependencies of the thermal boundary conditions on the side [Fig. 45] and the top of the slab [Fig. 46].

**Geometry browser** → Boundary conditions → External Temperature Lateral → Edit time dependency [Fig. 45]
**Geometry browser** → Boundary conditions → External Temperature Top → Edit time dependency [Fig. 46]
2.3.2 Static Boundary Conditions

We attach the supports to the slab. We first fix the translations in the $X$ and $Y$ directions in the lateral faces to make the symmetry conditions.

Main menu ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 47] [Fig. 48] [Fig. 49] [Fig. 50]

Figure 47: Symmetry supports - fix $X$ translations

Figure 48: Supported face $BCx$

Figure 49: Supported face $BCy$

Figure 50: Symmetry supports - fix $Y$ translations
We fix the translations in the $Z$ direction along the line.

**Main menu** ➔ Geometry ➔ Assign ➔ Add supports [Fig. 51] [Fig. 52]

**Figure 51:** Bottom supports - fix $Z$ translations

**Figure 52:** Supported edge
2.4 Initial Temperatures

We attach an initial field to define the initial temperature (20 °C) of the solid slab.

Main menu ➔ Geometry ➔ Assign ➔ Add initial fields 🎁 [Fig. 53] [Fig. 55]
2.5 Loads

We add the effect of the self-weight as a global load.

Figure 56: Self-weight
2.6 Mesh

We define the properties of the mesh. First we set an element size of 200 mm for all the slab with preferable Hexagonal/Quadrilateral elements [Fig. 57]. As we want 6 elements in the thickness of the slab we select one lateral edge and set the divisions to 6 [Fig. 58]. Finally we mesh the composed surface "Sheet 2" with an element size of 150 mm [Fig. 59].
And we generate the mesh [Fig. 61].

Figure 60: Geometry of the slab

Figure 61: View of the mesh
3 Transient Heat Flow Analysis

3.1 Commands

We first perform a transient heat flow analysis. For that we add a new analysis for transient heat transfer and we specify the initial temperature field as the initial conditions to be considered in the analysis.
We input the time step size as: 60(10) 300(8) 3000(4) seconds [Fig. 66]. This means that there are 10 steps of 60 seconds followed by 8 steps of 300 seconds followed by 4 steps of 3000 seconds. We chose to output the temperatures [Fig. 67] [Fig. 68] and finally we run the analysis.
3.2 Results

We display a contour plot of the temperature distribution at \( t = 250 \) minutes (= 15000 seconds).
We make a graph of the temperature distribution over the thickness of the slab. For that we define a probe curve through the thickness of the model $Z = [0, 150]$ mm at location $X = 1225$ mm, $Y = 380$ mm [Fig. 72]. We can study how the temperature varies in time other the thickness: select the result cases: "Time Step 10(600)", "Time Step 18(3000)" and "Time Step 22(15000)". At each time step show the resulting contour probe [Fig. 69] [Fig. 73].
We can copy the results to an Excel Worksheet and sort the results in Z direction [Fig. 74].

Figure 74: Temperature distribution over thickness
4 Structural Nonlinear Analysis

4.1 Commands

We now continue the analysis by performing a structural nonlinear computation that accounts for the temperature effects determined previously in the heat transfer analysis. For that we add a new execute block in the Heatflow-stress analysis for structural nonlinear. We add a second execute block to apply the time steps. Select the first execute block to apply the self-weight (SW) load in one step [Fig. 76] [Fig. 77].

![Figure 75: Analysis browser](https://dianafea.com)
![Figure 76: Analysis browser](https://dianafea.com)
![Figure 77: Load steps](https://dianafea.com)
In the second execute block, we define the time steps in the same manner we did for the transient heat transfer analysis: $60(10)$ $300(8)$ $3000(4)$ [Fig. 76] [Fig. 78]. We select displacements and stresses for output results [Fig. 79] [Fig. 80] and we run the analysis.
4.2 Results

4.2.1 Stresses in the Concrete Slab

We display a contour plot of the maximum principal stress $S_1$ in the concrete slab at $t=250$ minutes ($=15000$ seconds).

Figure 81: Results browser
Figure 82: Cauchy Total Stresses $S_1$
Figure 83: Cauchy Total Stresses $S_1$
We also make a graph for the distribution of maximum principal stress over the thickness of the slab. For that we define a probe-curve through the model at \(X=200\) mm, \(Y=380\) mm [Fig. 84]. We can study how the stresses vary in time: select the result cases “Time Step 11(600)”, “Time Step 19(3000)” and “Time Step 23(15000)” . At each time step show the resulting contour probe [Fig. 85].

![Figure 84: Probe curve settings](https://dianafea.com)

![Figure 85: Probe curve](https://dianafea.com)
We copy the results to an Excel WorkSheet and sort the results in Z direction so we can compare the results.

Figure 86: Maximum principal stress distribution over thickness
4.2.2 Stresses in the Reinforcement Grid

We observe the stresses in the reinforcement grid. From the mesh browser we show only the "Sheet 1" that corresponds to the reinforcement grid [Fig. 87].

The created reinforcement set represents the reinforcement grid in the composed surface elements. This is only required for a design analysis. Because this reinforcement set is on the same location as the general grid, we hide the reinforcement set in this analysis to plot results on the general reinforcement grid.
We show the stresses $S_{XX}$ and $S_{YY}$ for the last time step.
4.2.3 Distributed Bending Moments in the Composed Surface

In the mesh browser we show only the "Sheet 2" shape that corresponds to the composed surface [Fig. 91].

Mesh browser  →  Mesh  →  Shapes  →  Sheet 2  →  Show only  [Fig. 91]

Results browser  →  Case  →  Time-step 23,...  [Fig. 92]
We show the distributed moments $M_{xx}$ and $M_{yy}$ in the slab.

Figure 93: Distributed Moments $M_{xx}$

Figure 94: Distributed Moments $M_{yy}$
Appendix A  Additional Information

Folder: Tutorials/SlabUnderFire

Number of elements \( \approx 700 \)

Keywords:
- ANALYS: flow flowst heat nonlin physic stagge transi.
- CONSTR: initia supper temper.
- ELEMMEN: lsq4ht chx60 compos c8cm flow hx8ht potent reinfo solid taper.
- LOAD: elemen temper time weight.
- MATERI: conduc elasti isoto temper.
- OPTION: direct newton regula units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy displa force moment princi stress temper total.
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.