Creep Response of a Prestressed Concrete Beam under Sustained Load
1 Description

This tutorial presents the simulation of a creep benchmark: a partially prestressed flexural beam tested under sustained load by Espion and Halleux in 1991\(^1\).

A series of partially prestressed concrete beams were tested in the University of Brussels between 1981 and 1986 by Espion and Halleux. The experimental tests were composed by simple supported beams with 8.0 m of span and rectangular cross-sections of 0.34 m width and 0.4 m high, tested under sustained load during 4.5 years. These tests are a recognized benchmark used to validate finite element models for the time-dependent behavior of concrete beams\(^2\).

This tutorial relates to beam LT05Q. The geometric characteristics and reinforcement layout of the beam is presented in Figure 1. The prestress tendons have a linear variation with an inclination of 3.2° with the horizontal axis between the load points and the supports. Transversal reinforcement consists of 10 mm stirrups distanced of 0.125 m in the shear span and 0.25 m in the central zone.

![Figure 1: Characteristics of the creep beam test LT05Q from Espion and Halleux 1991][1]

---

\(^1\)Espion and Halleux, *Long term behavior of prestressed and partially prestressed concrete beams: experimental and numerical results*, 1991

\(^2\)Espion, *Benchmark examples for creep and shrinkage analysis computer programs. Creep and shrinkage of concrete*, 1993
The beam was cured for a period of 1 day. After that it was maintained at a constant relative humidity of 60% and constant temperature of 20 °C.

The mechanical properties of the materials are indicated in Table 1. To model concrete we use a class type of Model Code 2010 (MC2010) that better fits these properties (C25).

A prestress force of 1228 kN per tendon was applied by post-tension in one of the extremities at the age of 14 days. Two concentrated loads $Q$ were applied at different time stages ($Q = 16.5$ kN at 28 days and increased to $Q = 63.75$ kN at 84 days of age). The load history is presented in Table 2. Cracks were observed in the beam after application of the second load phase ($Q = 63.75$ kN) as reported in Espion 1993[2].

### Table 1: Material properties

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s modulus $E$</th>
<th>Compressive strength $f_{cm}$</th>
<th>Tensile strength $f_{tm}$</th>
<th>Mass density $\rho$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete</td>
<td>30.75E9 N/m²</td>
<td>33.9E6 N/m²</td>
<td>3.0E6 N/m²</td>
<td>2350 kg/m³</td>
</tr>
<tr>
<td>Reinforcement steel</td>
<td>200E9 N/m²</td>
<td></td>
<td></td>
<td>8000 kg/m³</td>
</tr>
<tr>
<td>Prestress steel</td>
<td>200E9 N/m²</td>
<td></td>
<td></td>
<td>8000 kg/m³</td>
</tr>
<tr>
<td>Steel for plates</td>
<td>210000 N/mm²</td>
<td>0.3</td>
<td>8000 kg/m³</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Time (days)</th>
<th>Self weight (kN/m)</th>
<th>Post-tension (kN/cable)</th>
<th>Load $Q$(kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 &lt; $t$ &lt; 14</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>14 &lt; $t$ &lt; 28</td>
<td>$\approx$ 3.3</td>
<td>122.8</td>
<td>16.5</td>
</tr>
<tr>
<td>28 &lt; $t$ &lt; 84</td>
<td>$\approx$ 3.3</td>
<td>122.8</td>
<td>63.75</td>
</tr>
<tr>
<td>84 &lt; $t$ &lt; 1642</td>
<td>$\approx$ 3.3</td>
<td>122.8</td>
<td></td>
</tr>
</tbody>
</table>
To model this benchmark we use a three-dimensional finite element model with solid elements for the beam and embedded truss elements for longitudinal and transversal reinforcements [Fig. 2]. The following considerations were taken into account:

1. due to symmetry, only half of the beam is modeled; we opt not to model one quarter because of the configuration of the reinforcement and prestressing
2. the concrete behavior is modeled through the MC2010 expressions available in DIANAIE (Concrete class C25), including creep and shrinkage effects; for the sake of simplicity, total strain crack models were not included
3. the reinforcement steel (passive and active) is modeled with embedded reinforcements with linear elastic behavior; we used non-bonded embedded reinforcements for the application of post-tension in the prestressing cables
4. the steels plates for load application and support (0.15 m of length and 0.035 m thick) are modeled with linear elastic material properties
5. interface elements are used between the steel plates and the concrete beam, with high normal stiffness and low shear stiffness modulus
6. a distributed load equivalent to the point load $Q$ [Fig. 1] is applied in the top face of the load steel plate
7. a structural nonlinear analysis is performed with time steps until a total time of approximately 4.5 years
8. the experimental and numerical results are compared in terms of displacements versus time

Figure 2: Three-dimensional model of the creep beam test LT05Q
2 Finite Element Model

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

Figure 3: New project dialog
We use SI units (N, m, s) with the exception of temperature that we change to Celsius. Despite the fact that it would be handy to have the time unit in days we keep it in seconds (SI) to have a consistent unit set [Fig. 5].
2.1 Geometry

We start the model by making three blocks for: beam, support plate and loading plate.

Main menu ➔ Geometry ➔ Create ➔ Add block ➔ [Fig. 6] [Fig. 7]

Figure 6: Add block

Figure 7: Geometry view - Beam
Main menu ➔ Geometry ➔ Create ➔ Add block ➔ [Fig. 8] [Fig. 9]

Figure 8: Add block

Figure 9: Geometry view - Support plate
Add block

Create  Close

Name*
Load plate
Position*
1.925 0.4 m
Size*
0.15 0.34 0.035 m

Figure 10: Add block

Figure 11: Geometry view - Loading plate
We now define the reinforcement. We start by creating 13 lines for the ordinary longitudinal reinforcement (see Figure 1). We have 4 layers of passive reinforcement in the cross-section: R1-1 means layer 1 reinforcement 1, R2-1 means layer 2 reinforcement 1, etc. The coordinates of the lines are listed in Table 3. We group the reinforcements into reinforcement sets, according to their material and geometry properties.

Table 3: Coordinates for defining the lines for passive reinforcement

<table>
<thead>
<tr>
<th>Reinforcement set</th>
<th>Name</th>
<th>Point 1</th>
<th>Point 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Passive reinforcement 8</td>
<td>R1-1</td>
<td>0, 0.035, 0.373</td>
<td>4, 0.035, 0.373</td>
</tr>
<tr>
<td></td>
<td>R1-2</td>
<td>0, 0.125, 0.373</td>
<td>4, 0.125, 0.373</td>
</tr>
<tr>
<td></td>
<td>R1-3</td>
<td>0, 0.215, 0.373</td>
<td>4, 0.215, 0.373</td>
</tr>
<tr>
<td></td>
<td>R1-4</td>
<td>0, 0.305, 0.373</td>
<td>4, 0.305, 0.373</td>
</tr>
<tr>
<td></td>
<td>R2-1</td>
<td>0, 0.035, 0.285</td>
<td>4, 0.035, 0.285</td>
</tr>
<tr>
<td></td>
<td>R2-2</td>
<td>0, 0.305, 0.285</td>
<td>4, 0.305, 0.285</td>
</tr>
<tr>
<td></td>
<td>R3-1</td>
<td>0, 0.035, 0.115</td>
<td>4, 0.035, 0.115</td>
</tr>
<tr>
<td></td>
<td>R3-2</td>
<td>0, 0.305, 0.115</td>
<td>4, 0.305, 0.115</td>
</tr>
<tr>
<td>Passive reinforcement 18</td>
<td>R4-1</td>
<td>0, 0.035, 0.035</td>
<td>4, 0.035, 0.035</td>
</tr>
<tr>
<td></td>
<td>R4-2</td>
<td>0, 0.103, 0.035</td>
<td>4, 0.103, 0.035</td>
</tr>
<tr>
<td></td>
<td>R4-3</td>
<td>0, 0.170, 0.035</td>
<td>4, 0.170, 0.035</td>
</tr>
<tr>
<td></td>
<td>R4-4</td>
<td>0, 0.238, 0.035</td>
<td>4, 0.238, 0.035</td>
</tr>
<tr>
<td></td>
<td>R4-5</td>
<td>0, 0.305, 0.035</td>
<td>4, 0.305, 0.035</td>
</tr>
</tbody>
</table>

Tip: here we can also make use of the array copy tool, for example: create R1-1 and array copy it 3 times for 0.090 m in the Y direction; create R4-1 and array copy it for 0.0675 m in the Y direction.
Figure 13: Geometry view - longitudinal reinforcement
We now define the polygon lines for the 5 prestress tendons (see Figure 1). There are 3 layers of prestressing reinforcement: line P1-1 means layer 1 prestress tendon 1, P2-1 means layer 2 prestress tendon 1, etc. The coordinates of the polygon lines are listed in Table 4.

<table>
<thead>
<tr>
<th>Reinforcement set</th>
<th>Name</th>
<th>Point 1 (X, Y, Z)</th>
<th>Point 2 (X, Y, Z)</th>
<th>Point 3 (X, Y, Z)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Prestress tendon</td>
<td>P1-1</td>
<td>0, 0.07, 0.2518</td>
<td>2, 0.07, 0.140</td>
<td>4, 0.07, 0.140</td>
</tr>
<tr>
<td></td>
<td>P1-2</td>
<td>0, 0.27, 0.2518</td>
<td>2, 0.27, 0.140</td>
<td>4, 0.27, 0.140</td>
</tr>
<tr>
<td></td>
<td>P2-1</td>
<td>0, 0.17, 0.2208</td>
<td>2, 0.17, 0.115</td>
<td>4, 0.17, 0.115</td>
</tr>
<tr>
<td></td>
<td>P3-1</td>
<td>0, 0.07, 0.2018</td>
<td>2, 0.07, 0.090</td>
<td>4, 0.07, 0.090</td>
</tr>
<tr>
<td></td>
<td>P3-2</td>
<td>0, 0.27, 0.2018</td>
<td>2, 0.27, 0.090</td>
<td>4, 0.27, 0.090</td>
</tr>
</tbody>
</table>

Table 4: Coordinates for defining the polylines for prestress tendons

*Tip: here we can also use array copy tool from P1-1 to P1-2 and from P3-1 to P3-2.
Figure 15: Geometry view - active and passive longitudinal reinforcement
We now define the transversal reinforcement (reinforcement set *Stirrups*). For that we create closed polygon lines with the dimensions of the stirrups distanced by 0.125 m in the shear span and by 0.25 m in the central zone.

We define the first double leg stirrup from the bottom to the top reinforcements and at a distance of 0.035 m from the edge of the beam. We array copy that line 15 times with a distance of 0.125 m in the $X$ direction [Fig. 17].
We copy the last stirrup (Stirrup 16) once with a distance of 0.09 m to be aligned with the load point [Fig. 18]. Finally we copy this last one (Stirrup 17) 8 times at a distance of 0.25 m in the X direction [Fig. 19].

Figure 18: Array copy stirrups
Figure 19: Array copy stirrups
The definition of the reinforcement is complete.

Figure 20: Geometry view - reinforcement
We create reinforcement sets for the different types of reinforcement present in the model: *Passive reinforcement 8, Passive reinforcement 18, Prestress reinforcement, Stirrups*. We also add a shape set called *Beam* that includes the beam and support and load plates. We select the shapes and create the new sets from the selection.

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

---

**Main menu ➔ Geometry ➔ Create ➔ Add shape set ➔ Beam** [Fig. 21]

**Main menu ➔ Geometry ➔ Create ➔ Add reinforcement set ➔ Passive reinforcement 8** [Fig. 21]

< Repeat 3 × for the other reinforcement sets >
2.2 Properties

2.2.1 Concrete Beam

To model the concrete we use the concrete design codes class available in DIANAIE. We chose the MC2010 (Class C25). We include creep and shrinkage effects. Ambient temperature is 20 °C, notional size of member ($h = 2A_c/c$) is 0.1837 and the relative humidity is 60%. We choose a non-aging creep curve type: concrete age at loading is 14 days (1209600s) and the concrete age at end of curing period is 1 day (86400s).

We use structural solid elements and we don’t need to define geometry and data.

---

Main menu ➔ Geometry ➔ Assign ➔ Shape Properties ➔ [Fig. 22]
Shape Properties ➔ Material ➔ Add material ➔ Edit material ➔ [Fig. 23] ➔ [Fig. 24] ➔ [Fig. 25]
2.2.2 Steel Plates

We define the properties of the loading and support plates. Steel is considered as linear elastic Table 1. We use structural solid elements and we don’t need to define geometry and data.

---

**Main menu** → **Geometry** → **Assign** → **Shape Properties** [Fig. 26]

**Shape Properties** → **Material** → **Add material** [Fig. 27] → **Edit material** [Fig. 28]

---

Figure 26: Plates properties

Figure 27: Add new material - Steel

Figure 28: Steel material properties
2.2.3 Passive Longitudinal Reinforcement

We define the properties of the ordinary reinforcement (reinforcement sets *Passive reinforcement 8* and *Passive reinforcement 18*). We define a new linear elastic material for the reinforcement steel [Table 1]. We need two geometry types for the \( \phi 8 \) bars (cross-section area of 5e-05 m\(^2\)) and the bars \( \phi 18 \) bars (cross-section area of 2.54e-4 m\(^2\)).

We start with the \( \phi 8 \) bars (see Figure 1).

---

<Select the correspondent reinforcement set in the Geometry browser:>

**Main menu** ➔ **Geometry** ➔ **Assign** ➔ **Reinforcement properties** ➔ **Fig. 29**

Reinforcement properties ➔ **Material** ➔ **Add material** ➔ **Fig. 30** ➔ **Edit material** ➔ **Fig. 31**

---

Figure 29: Reinforcement properties - \( \phi 8 \) bars

Figure 30: Add new material - reinforcement steel

Figure 31: Reinforcement steel properties
Figure 32: Edit new geometry - φ8 bars
We now define the φ18 bars (see Figure 1). We use the same material and we need to define a new geometry type.

Select the correspondent reinforcement set in the Geometry browser.

Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties ➔ [Fig. 33]

Reinforcement properties ➔ Geometry ➔ Edit geometry ➔ [Fig. 34]

Figure 33: Reinforcement properties - φ18 bars

Figure 34: Edit geometry - φ18 bars
2.2.4 Stirrups

We now define the properties of the transversal reinforcement (reinforcement set *Stirrups*). We use the same material as in the longitudinal reinforcement and we define a new geometry for φ10 (cross-section area of 7.85e-05 m²).

<Select the correspondent reinforcement set in the Geometry browser:>

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

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

We now define the properties of the active reinforcement (reinforcement set *Prestress reinforcement*). We need a new linear elastic material for the prestressing tendons [Table 1]. We include the bonding aspect and set that the reinforcement is not bonded to mother element in order to have a post-tension load.

<Select the correspondent reinforcement set in the Geometry browser>

**Main menu** ➔ Geometry ➔ Assign ➔ Reinforcement properties ➔ [Fig. 37]

Reinforcement properties ➔ Material ➔ Add material ➔ [Fig. 38] ➔ Edit material ➔ [Fig. 39]

![Figure 37: Reinforcement properties - prestress tendons](image1)

![Figure 38: Add new material - Prestress](image2)

![Figure 39: Material properties - Prestress](image3)
We define a new geometry for the 0.5” cables (cross-section area of 9.3e-05 m²).

Figure 40: Edit geometry - prestress tendons
2.2.6 Interfaces

We include structural surface interfaces between the steel plates and the beam. We define a no-tension interface with shear reduction. We use a high value for the normal stiffness and low values for shear stiffness.

Figure 41: Interface properties

Figure 42: Add new material - Interface

Figure 43: Material properties - Interface
2.3 Boundary Conditions

We restrict the translation in the $Z$ direction along the bottom edge of the support plate. We change the view of the model to Left view to see the point support.

Main menu → Geometry → Assign → Add supports  
Main menu → Viewer → Viewpoints → Left view
We restrict the translation in the $Y$ direction in one vertex at the bottom edge of the support plate.
Due to the symmetry condition we restrict the translation in the $X$ direction in the symmetry surface. We change the view of the model back to *Isometric view 1*.

---

**Main menu → Geometry → Assign → Add supports**  [Fig. 48]

**Main menu → Viewer → Viewpoints → Isometric view 1**

Figure 48: Add face support

Figure 49: Geometry view - symmetry support
2.4 Loads

2.4.1 Self-weight

We first set the self-weight of the model.
2.4.2 Post-tension Load

We define the post-tension load applied to one end of the prestress tendons. The anchor points are located in the extreme of the beam. The prestress force is 122.8 kN per tendon. This force was measured after application of prestress, so we don’t consider prestress losses.

Figure 51: Post-tension load

Figure 52: Geometry view - post-tension load
2.4.3 Applied Load

We define a distributed load of \(-19607 \text{ N/m}^2\) in the top surface of the loading plate. The value of the distributed load is equivalent to a point load of \(-1 \text{ kN}\) applied in the middle of the loading plate; the area of the top surface of the loading plate is \(0.15 \times 0.34 = 0.051 \text{ m}^2\). We will afterwards define the true value of the applied load (\(Q = 16.5 \text{ kN}\) and \(Q = 63.75 \text{ kN}\)) through load factors dependent on time.

Figure 53: Distributed load

Figure 54: Geometry view - distributed load
2.4.4 Load Combinations and Time Dependent Factors

We create two geometry load combinations and define time dependent functions for each of it\(^6\) (see Table 2):

- Combination 1: self-weight and post-tension loads are applied at 14 days (1209600 s) and maintained in time
- Combination 2: load is applied at 28 days (2419200 s) with the value of 16.5 kN (equivalent distributed force of 323529.41 N/m\(^2\)) and increased to 63.75 kN (equivalent distributed force of 1250000 N/m\(^2\)) at 84 days (7257600 s)

We will make an analysis until 1642 days (141868800 s) so we define the table until 1750 days (151200000 s)\(^7\).

\(^6\)Once we create geometry load combinations we can no longer use the load cases for the analysis, we can only use combinations. When we create combinations, by default DIANAIE presents as many load combination as previously defined loads. In this case we defined three loads and we only want two combinations so we need to delete one of them.

\(^7\)Make sure that the time tables are defined for a period that is always equal or larger then the time considered in the analysis.
Creep Response of a Prestressed Concrete Beam under Sustained Load | https://dianafea.com
2.5 Mesh

We set the element size as 0.1 m and generate the mesh.

Figure 58: Mesh properties

Figure 59: Finite element mesh
3 Structural Nonlinear Analysis

3.1 Commands

We perform a structural nonlinear analysis with time steps. As the load execute block is the default, we first need to remove it, and then add a new execute block with time steps.

Figure 60: Analysis browser

Figure 61: Add command

Figure 62: Analysis browser - execute time steps
We choose user specified sizes for the time steps. We use small time steps in the beginning of the analysis (because is when creep deformations are more relevant) and also immediately after the application and increasing of the load. Afterwards, we use larger time steps until completing the 1642 days of the analysis.

The considered time steps are (in seconds): 1.0 1.2096e+06 86400.0(84) 172800.0(15) 432000.0(12) 5.184e+06(25) s

---

Figure 63: Analysis browser - edit time steps

Figure 64: User specified sizes for time steps

---

*The time steps in days are: 1.157E-5 14 0.5(56) 1(84) 2(15) 5(12) 60(25).
With the output user selection we choose the results of displacements, strains, stresses and crack index.

Finally we run the analysis.
3.2 Results

3.2.1 Displacements

We start the presentation of results with the curve of displacements at midspan versus time and comparison with experimental data.

Figure 68: Results browser

Figure 69: Selected node for displacements
We can see the graph of displacements versus time steps in DIANAIE.

![Graph of displacements versus time steps in DIANAIE](https://dianafea.com)

Figure 70: Table of total displacements
In order to compare the numerical results with the experiment data we copy the displacements from DIANAIE to Excel. We change the time unit from seconds to days and the displacement unit from meter to millimeters.

In general, we can observe a good agreement between the numerical and experimental results. This fitting is specially good during the post-tensioning stage and after the application of the first load. The computed long-term response after the second increment of load at 84 days starts to deviate from the experimental results. The model presents a stiffer response than the experiment. This is due to the fact that, for the sake of simplicity of the model, we are not considering cracking in the simulation. And after the second application of load the beam is cracked, as we are going to see from the stress results. The cracking after the second load application was also observed experimentally, as reported in Espion and Halleux 1991.

Considering cracking in the creep analysis would increase significantly the complexity and computation time of the analysis.

Espion and Halleux, Long term behavior of prestressed and partially prestressed concrete beams: experimental and numerical results, 1991
We are now presenting the contour plots of several results and we first hide the steel plate parts from the mesh [Fig. 72]. We start with the displacements in the Z direction for the different stages of the analysis:

1. post-tensioning and self-weight at 14 days,
2. application of the first load at 28 days,
3. increment of the load at 84 days,
4. final time at 1716 days.

In the instant immediately after the application of the post-tensioning we can observe the curvature of the beam [Fig. 74].
This deformed configuration changes after the application of the first load [Fig. 75]. We can observe the increment of displacements with time by comparing with the values of displacements immediately before the second application of the load [Fig. 76].

Figure 75: Displacements in $Z$ direction at $t = 28$ days

Figure 76: Displacements in $Z$ direction at $t = 83$ days
Figure 77 shows the displacements after the application of the second load and Figure 78 at final time of analysis.

We can see the significant increment of displacements in the beam under constant load due to the time-dependent effects, mainly creep effects.
3.2.2 Stresses and Crack Indices

We now present the principal stresses. For principal stresses $S_1$ we make a contour plot and set the legend to a maximum of 3 MPa, to check if the beam is cracked. We also switch off the deformed shape view.

---

**Results browser** → Analysis1 → Output → Element results → Cauchy Total Stresses → $S_1$  
**Property Panel** → Result → Color scale limits  
<Set to specified values>

**Main menu** → Results → Absolute deformed results

---

Figure 79: Results browser  
Figure 80: Contour plot settings
We see the maximum principal stresses for the instant after the application of the second load at $t = 84$ days and for the final time of analysis at $t = 1716$ days. The beam presents high crack risk in some locations (in red).

Figure 81: Maximum principal stresses $S_1$ for $t = 84$ days

Figure 82: Maximum principal stresses $S_1$ for $t = 1716$ days
We can also check the crack index. For that we put the contour plot settings back to default: auto-scale to display results.

Figure 83: Results browser

Figure 84: Default contour plot settings
So we also see the crack index $F_{tu}$ for the instant after the application of the second load at $t = 84$ days and for the final time of analysis at $t = 1716$ days. The crack index $F_{tu}$ is determined as $F_{tu} = \frac{\sigma_f}{f}$. In the areas with $F_{tu} > 1$ there is risk of cracking. We can observe that after the second load application [Fig. 85] the beam is cracked, which is the reason for the difference between the numerical and experimental results appearing at this stage [Fig. 71].
We now look into the compressive stresses in the beam after the application of the second load at 84 days. It is generally accepted that creep in concrete is linear when stresses remain below 0.45fcm.

We can observe from the contour plot of minimum principal stresses S3 that the beam is mostly in the linear stage (\(\sigma < 0.45 \times 33.9 \text{ MPa} = 15.255 \text{ MPa}\)), so the linear creep strain assumption is valid. We can also observe the relaxation of stresses in time by comparing Figure 87 (after application of the second load for t = 84 days) and Figure 88 (at the end of the analysis for t = 1716 days).
Appendix A  Additional Information

Folder: Tutorials/CreepPrestressBeam

Number of elements ≈ 500

Keywords:
- ANALYS: nonlin physic.
- CLASS: large.
- CONSTR: suppor.
- ELEMEN: bar chx60 cq48i interf reinfo solid struct.
- LOAD: anchor elemen face force postte reinfo time weight.
- MATERI: concre creep elasti isotor maxwel mc2010 shrink viscoe.
- OPTION: direct newton regula units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy crack crkind displa green princi status 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.