Gas Explosion in a Tunnel
Outline

1 Description
   1.1 Dimensions and Reinforcement Layout ........................................... 4
   1.2 Material Properties ................................................................. 5
   1.3 Load ......................................................................................... 6
      1.3.1 Static Loads ....................................................................... 7
      1.3.2 Explosion ........................................................................... 7
   1.4 Modeling Assumptions ................................................................. 8

2 Finite Element Model
   2.1 Geometry ................................................................................. 9
      2.1.1 Tunnel ................................................................................ 9
      2.1.2 Reinforcements ................................................................. 10
   2.2 Properties ................................................................................ 11
   2.3 Boundary Conditions ................................................................. 12
   2.4 Loads ....................................................................................... 13
      2.4.1 Self-Weight ...................................................................... 13
      2.4.2 Hydrostatic Pressure ......................................................... 14
      2.4.3 External Pressure from Sand ............................................. 15
      2.4.4 Explosion ........................................................................ 16
      2.4.5 Load Combinations ........................................................... 16
      2.4.6 Time-Load Diagram for the Explosion Pressure .................. 18
   2.5 Mesh ....................................................................................... 19

3 Linear Static Analysis
   3.1 Commands ............................................................................. 20
   3.2 Results for the Self-Weight ......................................................... 21
      3.2.1 Displacements .................................................................. 22
   3.3 Results for the Sand and Water Pressure ................................. 23
   3.4 Results for the Initial Explosion .............................................. 24
      3.4.1 Displacements .................................................................. 25
      3.4.2 Principal Stresses ............................................................. 26

4 Reduced Model for Dynamic Analyses
   4.1 Simplified Geometry ................................................................. 27
   4.2 Properties .............................................................................. 28
   4.3 Boundary Conditions ................................................................. 29
   4.4 Mesh ....................................................................................... 30

5 Eigenvalue Analysis
   5.1 Commands ............................................................................. 31
5.2 Results
   5.2.1 Eigenfrequencies and Rayleigh Damping Parameters
   5.2.2 Eigenmodes

6 Transient Nonlinear Dynamic Analysis
   6.1 Commands
   6.2 Results after Initial Application of Static Loads
      6.2.1 Displacements
      6.2.2 Cracks
      6.2.3 Stresses in Concrete
      6.2.4 Plasticity in Reinforcement
   6.3 Results for the Explosion Load
      6.3.1 Results after Start of Explosion
      6.3.2 Results at the End of Explosion
      6.3.3 Results at the End of Analysis
   6.4 Time History Diagrams

Appendix A Additional Information
1 Description

This example involves an underwater tunnel of which a typical configuration is shown in Figure 1. For a comprehensive description of this example see Meyer (1987)\(^1\), Van Mier (1987)\(^2\) and Nauta (1991)\(^3\). The tutorial focuses on the finite element model set-up and the analyses under static and dynamic loadings. Linear static and transient nonlinear dynamic analyses are performed to assess the response of the tunnel against an internal gas explosion.

\(1\) Meyer, Analysis of underwater tunnel for internal gas explosion, 1987
\(2\) Mier, van, Examples of non-linear analysis of reinforced concrete structures with DIANA, 1987
\(3\) Nauta, Dynamische belastingen (II), computerprogramma’s voor dynamica-berekeningen – DIANA, 1991

Figure 1: Typical underwater tunnel cross-section
1.1 Dimensions and Reinforcement Layout

Figure 2 shows the tunnel cross-section, and Figure 3 represents the layout of the reinforcement. There are five reinforcement layers in the transversal cross-section of the tunnel’s walls: one at 10 cm below the external top surface, one at 5 cm below the external surface, one at 5 cm above the internal surface, and two vertical reinforcement layers which extend from the wall into the roof slab. Each layer consists of one or more sections, depending on the amount of reinforcement (diameter and number of bars).

Figure 2: Tunnel cross-section (dimensions in m)

Figure 3: Tunnel reinforcement for a 1.5 m wide section
1.2 Material Properties

The material properties used for concrete and steel are presented in the following tables. We consider a class C40 for concrete. We do not make use of a model code material, but we define a total strain crack material with C40 class properties as reported in Table 1. The mechanical properties for the reinforcement steel are presented in Table 2.

<table>
<thead>
<tr>
<th>Property</th>
<th>Concrete (C40)</th>
<th>Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus $E$</td>
<td>$2.53873E+10$</td>
<td>$2.1E+11$</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.15</td>
<td></td>
</tr>
<tr>
<td>Compressive strength $f_{cm}$</td>
<td>$4.0E+7$</td>
<td></td>
</tr>
<tr>
<td>Tensile strength $f_{tm}$</td>
<td>$2.45617E+6$</td>
<td></td>
</tr>
<tr>
<td>Fracture energy in compression $G_c$</td>
<td>37500 N/m</td>
<td></td>
</tr>
<tr>
<td>Fracture energy in tension $G_F$</td>
<td>150 N/m</td>
<td></td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>2400 kg/m$^3$</td>
<td></td>
</tr>
</tbody>
</table>

Table 1: Material properties for concrete

<table>
<thead>
<tr>
<th>Property</th>
<th>Concrete (C40)</th>
<th>Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yielding strength $f_{ym}$</td>
<td>$5.0E+8$</td>
<td></td>
</tr>
</tbody>
</table>

Table 2: Material properties for reinforcement steel
1.3 Loads

1.3.1 Static Loads

The tunnel segment is subjected to a static load (deadweight, sand and water pressure) and to a dynamic load (internal explosion) [Fig. 4]. Since the bottom slab is laying on the underwater soil and considering its thickness, for the scope of the example, the tunnel section is assumed to be supported on the lower edge in $X$ and $Y$ directions.

The distributed pressure due to the presence of the sand above the tunnel section is represented by the weight of a layer of 2 m height and a mass density $\rho = 1900 \text{ kg/m}^3$. The water has a height of 10 m above the soil level and is modeled through an hydrostatic pressure involving the external sides of the tunnel.

The pressure due to sand and water on the upper part of the tunnel is $(2 \times 1900 + 10 \times 1000) = 13800 \text{ kg/m}^2$ for a unitary tunnel section. In addition, depending on the tunnel position within the underwater sand and the water level above, the lateral pressure is also imposed, as shown in Figure 4.
1.3.2 Explosion

The explosion pressure [Fig. 5] starts with an instantaneous shock front, followed by a parabolic decrement of the pressure during 25 ms. Then the pressure remains constant during 100 ms defining the ‘pressure plateau’. In the next 25 ms the depressurize phase takes place.

Figure 5: Explosion - pressure in time
1.4 Modeling Assumptions

The following aspects are considered in the model:

For the linear static analysis:

- due to the characteristics of the problem in terms of geometry, boundaries and loading, a plane strain modeling strategy is used
- the reinforcement is modeled through embedded grid reinforcements
- the model geometry is symmetric (in terms of dimensions, reinforcement, boundary conditions and static loadings). However, due to presence of the explosion load only in one of the tunnel internal sides, the entire section is modeled
- the loads are applied as distributed forces (pressures) on the external edges of the tunnel section
- the effect of the water is taken into account by means of an hydrostatic pressure imposed on the external edges of the tunnel section, defining a reference level for the water head at the sand level and assuming an hydraulic head of 10 m
- extra pressure due to the underwater sand is taken into account as a distributed force on top (constant pressure) and on the sides (linear pressure varying with height) of the tunnel
- the conditions at the bottom side of the tunnel are simplified into fixed supports for $X$ and $Y$ translations, considering the stiffness of the bottom part of the tunnel laying on the underwater soil

For the nonlinear dynamic analysis:

- since the presence of the bottom slab is of less interest and its participation to the deformation is limited, for the transient analysis involving the explosion load, the model is simplified by cutting out the bottom part of the tunnel and imposing translational supports at the base of the tunnel’s walls
- the simplification of the model by removing the bottom slab is done by cutting the initial surface representing the tunnel cross-section. With this operation it is also shown how the previously assigned loads can be kept, without need to be reassigned due to a partial modification of the shape to which they are applied
- in order to be considered as a mass participating in the dynamic behavior, the external loads representing the soil and water are here modeled as distributed mass elements
- due to the presence of the lateral underwater soil which provides stiffness to the external vertical walls, supports in the horizontal direction are assumed since the focus of the analysis is in the upper part of the tunnel
2 Finite Element Model

For the modeling session we start a new project [Fig. 6]. We choose to work on a plane strain model and we set the dimensions of the domain equal to 100 m. Quadratic quadrilateral finite elements will be predominantly used in the analysis."
We use meter, newton and kilogram for length, force and mass, respectively. We choose to use degree for the angles.
2.1 Geometry

2.1.1 Tunnel

To model the geometry of the tunnel in DianaIE, we start by creating a sheet and we name it Tunnel [Fig. 11]. We specify the coordinates of the points from 1 to 6 as shown in Figure 9. A plane strain model is situated in the $XY$ plane, i.e., the $Z$ coordinates are zero.

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>-14.9</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>14.9</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>14.9</td>
<td>6.8778</td>
<td>0</td>
</tr>
<tr>
<td>13.9178</td>
<td>7.86</td>
<td>0</td>
</tr>
<tr>
<td>-13.9178</td>
<td>7.86</td>
<td>0</td>
</tr>
<tr>
<td>-14.9</td>
<td>6.8778</td>
<td>0</td>
</tr>
</tbody>
</table>

Figure 9: Coordinates – Tunnel
Figure 10: Add sheet – Tunnel
Figure 11: View of the model – Tunnel
Afterwards, we define three more sheets which are used to create the empty areas of the tunnel cross-section. We start by creating the first sheet and we name it *Inner_Left* [Fig. 14]. We provide the coordinates of the points as shown in Figure 12.

**Main menu ➔ Geometry ➔ Create ➔ Add polygon sheet [Fig. 13]**

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>-13.9</td>
<td>1.3887</td>
<td>0</td>
</tr>
<tr>
<td>-1</td>
<td>1.63</td>
<td>0</td>
</tr>
<tr>
<td>-1</td>
<td>6.56</td>
<td>0</td>
</tr>
<tr>
<td>-3.5</td>
<td>6.96</td>
<td>0</td>
</tr>
<tr>
<td>-11.4</td>
<td>6.96</td>
<td>0</td>
</tr>
<tr>
<td>-13.4</td>
<td>6.61</td>
<td>0</td>
</tr>
<tr>
<td>-13.9</td>
<td>6.11</td>
<td>0</td>
</tr>
</tbody>
</table>

Figure 12: Coordinates – *Inner_Left*

---

**Figure 14: View of the model – *Inner_Left***
We add a second sheet on the right side and we name it *Inner_Right* [Fig. 17]. We provide the coordinates of the points as shown in Figure 15.

Main menu ➔ Geometry ➔ Create ➔ Add polygon sheet ![Fig. 16]

![Add polygon sheet](image)

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.63</td>
<td>0</td>
</tr>
<tr>
<td>13.9</td>
<td>1.3887</td>
<td>0</td>
</tr>
<tr>
<td>13.9</td>
<td>6.11</td>
<td>0</td>
</tr>
<tr>
<td>13.4</td>
<td>6.61</td>
<td>0</td>
</tr>
<tr>
<td>11.4</td>
<td>6.96</td>
<td>0</td>
</tr>
<tr>
<td>3.5</td>
<td>6.96</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>6.56</td>
<td>0</td>
</tr>
</tbody>
</table>

Figure 15: Coordinates – *Inner_Right*

Figure 16: Add sheet – *Inner_Right*

Figure 17: View of the model – *Inner_Right*
Finally, we add a third central sheet and we name it *Inner_Center* [Fig. 19]. We provide the coordinates of the points as shown in Figure 18.

Figure 18: Add sheet – *Inner_Center*

Figure 19: View of the model – *Inner_Center*
We can now subtract the three internal sheets (which we use as tools) from the first one (which is used as a target) in order to create the internal holes of the tunnel. We do not keep the shapes used as tools [Fig. 20] [Fig. 21].

Figure 20: Subtract shapes

Figure 21: View of the model – Tunnel
2.1.2 Reinforcements

The reinforcement grids are lines in a plane strain model. We define some new lines by their start and end points, as listed in Table 3. This table presents only the $X$ and $Y$ coordinates of the points as the $Z$ coordinates are zero.

<table>
<thead>
<tr>
<th>Reinforcement line</th>
<th>Start point (m)</th>
<th>End point (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>re1</td>
<td>(0, 7.81)</td>
<td>(8.4, 7.81)</td>
</tr>
<tr>
<td>re2</td>
<td>(6.4, 7.81)</td>
<td>(12.2, 7.81)</td>
</tr>
<tr>
<td>re3</td>
<td>(10.1971, 7.81)</td>
<td>(13.8971, 7.81)</td>
</tr>
<tr>
<td>re4</td>
<td>(13.8971, 7.81)</td>
<td>(14.85, 6.8571)</td>
</tr>
<tr>
<td>re5</td>
<td>(14.85, 6.8571)</td>
<td>(14.85, 0.05)</td>
</tr>
<tr>
<td>re6</td>
<td>(10.3, 0.05)</td>
<td>(14.85, 0.05)</td>
</tr>
<tr>
<td>re7</td>
<td>(6, 0.05)</td>
<td>(12.3, 0.05)</td>
</tr>
<tr>
<td>re8</td>
<td>(0, 0.05)</td>
<td>(8, 0.05)</td>
</tr>
<tr>
<td>re9</td>
<td>(0, 7.76)</td>
<td>(6.1, 7.76)</td>
</tr>
<tr>
<td>re10</td>
<td>(1.5, 7.01)</td>
<td>(13.2, 7.01)</td>
</tr>
<tr>
<td>re11</td>
<td>(0, 1.5087)</td>
<td>(4.5, 1.5145)</td>
</tr>
<tr>
<td>re12</td>
<td>(2.5, 1.5519)</td>
<td>(14.5, 1.3275)</td>
</tr>
<tr>
<td>re13</td>
<td>(0.54, 0.53)</td>
<td>(0.54, 7.43)</td>
</tr>
<tr>
<td>re14</td>
<td>(0.96, 0.53)</td>
<td>(0.96, 7.43)</td>
</tr>
<tr>
<td>re15</td>
<td>(13.95, 0.3)</td>
<td>(13.95, 7)</td>
</tr>
<tr>
<td>re16</td>
<td>(0, 6.6094)</td>
<td>(0.9921, 6.6094)</td>
</tr>
<tr>
<td>re17</td>
<td>(0.9921, 6.6094)</td>
<td>(4.5, 7.1706)</td>
</tr>
<tr>
<td>re18</td>
<td>(12.6, 7.4807)</td>
<td>(14.85, 5.2307)</td>
</tr>
<tr>
<td>re19</td>
<td>(10.1971, 7.2213)</td>
<td>(14.3, 6.5033)</td>
</tr>
</tbody>
</table>
We now create the lines representing the reinforcements. As an example we create the first reinforcement line and we name it \textit{re1} [Fig. 22]–[Fig. 23]. Based on the coordinates listed in Table 3, the other lines are created by repeating the same command [Fig. 24].

Note that, in this phase, only the reinforcement lines present in the left part of the tunnel are created. The reinforcement is symmetric within the tunnel section, therefore it can be easily made on the right side of the tunnel through the mirroring command, which we apply after the reinforcement property assignment for practical reasons.
2.2 Properties

We assign the material and geometry properties to the tunnel [Fig. 25]. Since the aim of this tutorial is to carry out a shock analysis with an explosion loading in time, the corresponding safety coefficients provided in case of dynamic loading conditions are considered for the strength properties of concrete and steel in order to take into account the strain-rate effect.

We use class concrete C40 class concrete. We create a total strain based crack material and we name it Concrete [Fig. 26]. The linear material parameters are \( E = 2.53873 \times 10^9 \text{ N/m}^2 \) and \( \nu = 0.15 \) [Fig. 27].
We choose the rotating option for the total strain crack model [Fig. 28]. Afterwards, we define the nonlinear properties for the tensile and the compressive behaviors, as shown in Figure 29 and in Figure 30, respectively. For tension, we choose a softening law according to Hordijk and a damage based reduction model for Poisson’s ratio [Fig. 29]. For compression, we use a parabolic law based on fracture energy [Fig. 30].
In the dynamic analysis we assume 10% damping for the frequency $\omega_1 = 125$ Rad/s and $\omega_2 = 1250$ Rad/s (corresponding to periods of about $T_2 = 0.005$ s and $T_1 = 0.05$ s). Therefore, we assign the corresponding Rayleigh damping parameters as $a = 22.7$ 1/s and $b = 0.00145$ s to the tunnel material [Fig. 31].

Figure 31: Edit material – *Concrete*
We create a new element geometry and we name it *Tunnel* [Fig. 32]. We explicitly define the orientation of the local $x$ axis [Fig. 33].
We create a new material for the reinforcements and we name it *reinfo* [Fig. 34]. The Young’s modulus for steel is $E = 2.1 \times 10^{11} \text{ N/m}^2$ [Fig. 35], and the yield stress is $f_y = 5.0 \times 10^8 \text{ N/m}^2$ [Fig. 36]. We assume for the reinforcement steel the same Rayleigh damping parameters assigned to the concrete.
We now create different geometry properties in order to define the equivalent thickness of each reinforcement grid present in the tunnel. Therefore we need to define nine geometries, \( t_{re1} \) to \( t_{re9} \). For each geometry we provide the values for the equivalent thicknesses of the grids in the two directions. A summary of the reinforcement amount, the diameter and the number of bars for a 1.5 m thick tunnel section for each grid is given in Table 4. The corresponding equivalent thickness for unit length is also presented.

We need to create nine reinforcement sets, one for each reinforcement section type. We move the lines created for the reinforcements [Table 3] for the respective reinforcement sets. The list of the reinforcement lines to which the nine geometry properties are assigned and the respective reinforcement sets that they belong to are listed in Table 5.

### Table 4: Reinforcement amount per grid section

<table>
<thead>
<tr>
<th>Reinforcement section type</th>
<th>( A_s )</th>
<th>nr. of bars</th>
<th>( d ) (mm)</th>
<th>( t_{eq} ) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( t_{re1} )</td>
<td>10</td>
<td>10</td>
<td>40</td>
<td>8.378e-3</td>
</tr>
<tr>
<td>( t_{re2} )</td>
<td>9</td>
<td>9</td>
<td>40</td>
<td>7.540e-3</td>
</tr>
<tr>
<td>( t_{re3} )</td>
<td>8</td>
<td>8</td>
<td>40</td>
<td>6.702e-3</td>
</tr>
<tr>
<td>( t_{re4} )</td>
<td>10</td>
<td>10</td>
<td>30</td>
<td>4.712e-3</td>
</tr>
<tr>
<td>( t_{re5} )</td>
<td>8</td>
<td>8</td>
<td>30</td>
<td>3.770e-3</td>
</tr>
<tr>
<td>( t_{re6} )</td>
<td>7</td>
<td>7</td>
<td>30</td>
<td>3.299e-3</td>
</tr>
<tr>
<td>( t_{re7} )</td>
<td>6</td>
<td>6</td>
<td>30</td>
<td>2.827e-3</td>
</tr>
<tr>
<td>( t_{re8} )</td>
<td>20</td>
<td>20</td>
<td>30</td>
<td>9.425e-3</td>
</tr>
<tr>
<td>( t_{re9} )</td>
<td>4</td>
<td>4 + 3</td>
<td>30/40</td>
<td>4.398e-3</td>
</tr>
</tbody>
</table>

### Table 5: Reinforcement properties and corresponding reinforcement lines and sets

<table>
<thead>
<tr>
<th>Reinforcement section type</th>
<th>Reinforcement lines</th>
<th>Reinforcement sets</th>
</tr>
</thead>
<tbody>
<tr>
<td>( t_{re1} )</td>
<td>re1, re2</td>
<td>reinfo sec 1</td>
</tr>
<tr>
<td>( t_{re2} )</td>
<td>re3, re4, re5, re6, re10</td>
<td>reinfo sec 2</td>
</tr>
<tr>
<td>( t_{re3} )</td>
<td>re12</td>
<td>reinfo sec 3</td>
</tr>
<tr>
<td>( t_{re4} )</td>
<td>re9</td>
<td>reinfo sec 4</td>
</tr>
<tr>
<td>( t_{re5} )</td>
<td>re11</td>
<td>reinfo sec 5</td>
</tr>
<tr>
<td>( t_{re6} )</td>
<td>re7, re8, re19</td>
<td>reinfo sec 6</td>
</tr>
<tr>
<td>( t_{re7} )</td>
<td>re2, re15</td>
<td>reinfo sec 7</td>
</tr>
<tr>
<td>( t_{re8} )</td>
<td>re13, re14</td>
<td>reinfo sec 8</td>
</tr>
<tr>
<td>( t_{re9} )</td>
<td>re16, re17</td>
<td>reinfo sec 9</td>
</tr>
</tbody>
</table>
We add the reinforcement sets and move the lines in the geometry browser for the correspondent reinforcement sets as listed in Table 5.

Tip: an alternative way to create the reinforcement sets is to select the respective lines in the geometry browser, right-click in the selection and choose the option 'New reinforcement shapeset from selection'.
We now create the element geometry properties for the reinforcements. As an example we create the first reinforcement property and we name it \( t_{re,1} \) [Fig. 38] [Fig. 39]. Based on the information reported in Table 4, the other reinforcement properties are created by repeating the same command.

Main menu → Geometry → Element geometries → Add element geometry [Fig. 38] [Fig. 39]

Figure 38: Add new geometry – \( t_{re,1} \)

Figure 39: Edit geometry – \( t_{re,1} \)
We now assign the material and geometry properties to the reinforcement sets. We show it for the first reinforcement set *reinfo sec 1* in Figure 40. We continue with the property assignment for the other reinforcement sets based on the information reported in Table 5.

<Select the correspondent reinforcement set in the Geometry browser >

**Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties**  📂 [Fig. 40]

---

**Figure 40:** Reinforcement property assignments

**Figure 41:** Selection of reinforcement lines
Once the reinforcements are defined for the right side of the tunnel, we mirror all the reinforcement lines on the left side of tunnel [Fig. 42]. The same properties are kept for the corresponding reinforcements after the mirroring operation.

Figure 42: Mirror reinforcement shapes

Figure 43: Selection of reinforcement shapes to mirror
2.3 Boundary Conditions

The translation of the bottom edge of the tunnel is assumed to be supported in the $X$ and $Y$ direction [Fig. 44].

Main menu ➔ Geometry ➔ Assign ➔ Add supports [Fig. 44]

Figure 44: Attach support - Base

Figure 45: Selected edge

Figure 46: Translational supports on bottom edge
2.4 Loads

2.4.1 Self-Weight

We create a load case for the self-weight [Fig. 47].

Main menu → Geometry → Assign → Add global loads [Fig. 47]

![Edit global loads](image)

Figure 47: Global load – Self weight
2.4.2 Hydrostatic Pressure

We define the pressure imposed by the water on the external edges of the tunnel section through the definition of an hydrostatic pressure load. In order to illustrate the possibility to change the reference point for total head [Fig. 48], we choose to set the reference level in correspondence with the top level of the submerged sand, which is located 2 m over the roof of the tunnel. Therefore, we define the reference level at $Y = 7.86$ m (height of the tunnel section) + 2 m = 9.86 m [Fig. 49].

![Geometry browser](image1.png)

**Geometry browser** ➔ **Reference system** ➔ **Definitions**
**Property Panel** ➔ **Reference point for total head** ➔ 0.0. 9.86 0.0  

[Fig. 48] [Fig. 49]

Note: the reference point for total head could also be kept as default (origin) and, in this case, the total head for hydrostatic pressure would be defined as $Y = 7.86$ m (height of the tunnel) + 2 m (sand) + 10 m (water level) = 19.86 m.

![Figure 48](image2.png)  
**Figure 48:** Reference system - definitions

![Figure 49](image3.png)  
**Figure 49:** Property panel - reference point for total head
We create a new load which we name $P_{\text{Water}}$ and we include it in the Water loadcase [Fig. 50]. We assign it to the external edges of the tunnel [Fig. 51] [Fig. 52].

Main menu ➔ Geometry ➔ Assign ➔ Add loads 🌦️ [Fig. 50]

Figure 50: Attach load – $P_{\text{Water}}$

Figure 51: Selected edges

Figure 52: Water pressure varying with height
2.4.3 External Pressure from Sand

We add the pressure of the submerged sand on the upper and lateral edges of the tunnel. We do this through distributed force loads. We first create the load $P_{Sand, Top}$ for the top edges of the tunnel and we add it to the Pressure loadcase [Fig. 53].

Main menu → Geometry → Assign → Add loads 🏷️ [Fig. 53]
Afterwards, we add the same type of load to the two sides of the tunnel. Since the lateral pressure imposed by the sand is varying with depth, we first define a depth function and we name it \( f_{\text{side}} \), where the two extreme values correspond to the base and top values of pressure, respectively [Fig. 56]. This function is applied to a unitary load over the lateral edges of the tunnel.

Figure 56: Edit function – \( f_{\text{side}} \)
We now create the load $P_{Side}$ for the top edges of the tunnel and we add it to the Pressure loadcase. We input a unitary value for the distributed force and we select the previously defined function [Fig. 57].

Figure 57: Attach load – $P_{Side}$

Figure 58: Selected edges
2.4.4 Explosion

The initial explosion pressure on the internal edges of the right part of the tunnel is defined through the load $P_{\text{Explosion}}$ within the Explosion loadcase [Fig. 59]. Due to the ‘Normal’ option for the load direction, the distributed load always act perpendicularly to the shape edges, even along the inclined lines [Fig. 61].
2.4.5 Load Combinations

We define load combinations based on the defined load cases. In particular, in the *Geometry load combination 2* we combine the external pressure from water and sand, while the *Geometry load combination 4* includes all the load cases [Fig. 62].
2.4.6 Time-Load Diagram for the Explosion Pressure

Finally, we specify the time–load diagram for the explosion pressure (see Figure 5). This is done via a time dependent curve with the points listed in Table 6. We attach the time curve to the Geometry load combination 3 to define the explosion pressure in time [Fig. 63] [Fig. 64].

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>1.E-6</td>
<td>2.000</td>
</tr>
<tr>
<td>0.001</td>
<td>1.8824</td>
</tr>
<tr>
<td>0.002</td>
<td>1.7896</td>
</tr>
<tr>
<td>0.003</td>
<td>1.6616</td>
</tr>
<tr>
<td>0.004</td>
<td>1.5584</td>
</tr>
<tr>
<td>0.005</td>
<td>1.4600</td>
</tr>
<tr>
<td>0.006</td>
<td>1.3664</td>
</tr>
<tr>
<td>0.007</td>
<td>1.2776</td>
</tr>
<tr>
<td>0.008</td>
<td>1.1936</td>
</tr>
<tr>
<td>0.009</td>
<td>1.1144</td>
</tr>
<tr>
<td>0.010</td>
<td>1.0400</td>
</tr>
<tr>
<td>0.011</td>
<td>0.9704</td>
</tr>
<tr>
<td>0.012</td>
<td>0.9056</td>
</tr>
<tr>
<td>0.013</td>
<td>0.8456</td>
</tr>
<tr>
<td>0.014</td>
<td>0.7904</td>
</tr>
<tr>
<td>0.015</td>
<td>0.7400</td>
</tr>
<tr>
<td>0.016</td>
<td>0.6944</td>
</tr>
<tr>
<td>0.017</td>
<td>0.6536</td>
</tr>
<tr>
<td>0.018</td>
<td>0.6176</td>
</tr>
<tr>
<td>0.019</td>
<td>0.5864</td>
</tr>
<tr>
<td>0.020</td>
<td>0.5600</td>
</tr>
<tr>
<td>0.021</td>
<td>0.5384</td>
</tr>
<tr>
<td>0.022</td>
<td>0.5216</td>
</tr>
<tr>
<td>0.023</td>
<td>0.5096</td>
</tr>
<tr>
<td>0.024</td>
<td>0.5024</td>
</tr>
<tr>
<td>0.025</td>
<td>0.5000</td>
</tr>
<tr>
<td>0.125</td>
<td>0.5000</td>
</tr>
<tr>
<td>0.150</td>
<td>0.0000</td>
</tr>
<tr>
<td>1.000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

Table 6: Time-load diagram

Gas Explosion in a Tunnel | https://dianafea.com
2.5 Mesh

We assign the mesh properties to the tunnel by defining an element size of 0.3 m [Fig. 65]. Afterwards, we generate the finite element mesh [Fig. 66]. To check the applied loads, and particularly their direction, we display them on the mesh [Fig. 67].

Main menu → Geometry → Assign → Mesh properties [Fig. 65]

Mesh properties → Preview

Main menu → Geometry → Generate mesh [Fig. 66]

Main menu → Mesh → Loads → Show all loads [Fig. 67]
3 Linear Static Analysis

3.1 Commands

We start by adding a new analysis (that we call Linear Static) [Fig. 68]. We add to it a Structural linear static command to perform a linear static analysis [Fig. 69] [Fig. 70].

---

**Main menu** ➔ **Analysis** ➔ **Add analysis**

**Analysis browser** ➔ **Analysis1** ➔ **Rename** ➔ **Linear Static** [Fig. 68]

**Analysis browser** ➔ **Linear Static** ➔ **Add command** ➔ **Structural linear static** [Fig. 69] [Fig. 70]

---

![Figure 68: Analysis browser](image1)

![Figure 69: Command menu](image2)

![Figure 70: Analysis browser](image3)
For the *Linear Static* analysis we choose for *User selection* output [Fig. 71] and we add the following types of results [Fig. 72]. A summary of the required output is presented in Table 7.

We run the analysis.

---

**Table 7: Required output results**

<table>
<thead>
<tr>
<th>Total displacements</th>
<th>DISPLA TOTAL TRANS GLOBAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reaction forces</td>
<td>FORCE REACTI TRANS GLOBAL</td>
</tr>
<tr>
<td>External forces</td>
<td>FORCE EXTERN TRANS GLOBAL</td>
</tr>
<tr>
<td>Total stresses</td>
<td>STRESS TOTAL CAUCHY GLOBAL</td>
</tr>
<tr>
<td></td>
<td>STRESS TOTAL CAUCHY LOCAL</td>
</tr>
<tr>
<td></td>
<td>STRESS TOTAL CAUCHY PRINCI</td>
</tr>
<tr>
<td>Total strains</td>
<td>STRAIN TOTAL GREEN GLOBAL</td>
</tr>
<tr>
<td></td>
<td>STRAIN TOTAL GREEN LOCAL</td>
</tr>
<tr>
<td></td>
<td>STRAIN TOTAL GREEN PRINCI</td>
</tr>
</tbody>
</table>

**Figure 71: Output properties**

**Figure 72: Results selection**
3.2 Results for the Self-Weight

3.2.1 Displacements

We present the contour plot of the displacements $D_{tXY}$ due to the self-weight, by selecting the Load-combination 1 [Fig. 73]. The results indicate a maximum displacement of the roof of about 1 mm.

![Results browser](image1)

**Figure 73:** Results browser

![Deformation due to self-weight](image2)

**Figure 74:** Deformation due to self-weight
3.3 Results for the Sand and Water Pressure

3.3.1 Displacements

For the deformation due to the sand and water pressure we select the *Load-combination 2* [Fig. 75]. The maximum displacement is about 6 mm. We notice that the deformations due to self-weight and to the sand and water pressure are in the same order.

---

Figure 75: Results browser

Figure 76: Deformation due to sand and water pressure
3.4 Results for the Initial Explosion

3.4.1 Displacements

For the results of the initial explosion pressure we select the *Load-combination 3* [Fig. 77]. Here we observe a larger maximum displacement of about 50 mm.

![Figure 77: Results browser](https://dianafea.com)

![Figure 78: Deformation due to initial explosion](https://dianafea.com)
3.4.2 Principal Stresses

We show the stresses due to the initial explosion in combination with all the static loads applied on the tunnel. For this we select the Load-combination 4 and we present the contour plot of the principal stresses $S_1$ [Fig. 79]. The results indicate a maximum stress of more than 70 MPa, which is far beyond the tensile strength of the concrete, evidencing that a nonlinear analysis is required.

Figure 79: Results browser

Figure 80: Principal stresses
4 Reduced Model for Dynamic Analyses

4.1 Simplified Geometry

In order to simplify the model, the bottom slab of the tunnel is cut out from the initial surface, since its presence is of less interest. As shown in the preliminary linear static analysis its participation to the deformation response is limited. For this scope, we start by creating a polyline [Fig. 84]. We specify the coordinates of the points as shown in Figure 81. Afterwards, we extrude it in the direction orthogonal to the plane [Fig. 83].

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>-14.9</td>
<td>1.3887</td>
<td>0</td>
</tr>
<tr>
<td>-13.9</td>
<td>1.3887</td>
<td>0</td>
</tr>
<tr>
<td>-1</td>
<td>1.63</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>1.63</td>
<td>0</td>
</tr>
<tr>
<td>13.9</td>
<td>1.3887</td>
<td>0</td>
</tr>
<tr>
<td>14.9</td>
<td>1.3887</td>
<td>0</td>
</tr>
</tbody>
</table>

Figure 81: Coordinates – Polyline
Figure 82: Add polyline – Polyline
Figure 83: Extrude
Figure 84: View of the model – Polyline 1
Before cutting the tunnel shape, we move the previously created surface [Fig. 85].

**Figure 85**: Move shape

**Figure 86**: View of the cutting shape
We now trim the tunnel section [Fig. 87].

Figure 87: Trim shapes

Figure 88: Trimmed shape
We trim also the reinforcement lines extending into the bottom part of the tunnel [Fig. 89] [Fig. 90]. Finally, we select and remove the reinforcement lines within the bottom slab [Fig. 91].

**Main menu**  ➔  Geometry  ➔  Modify  ➔  Trim sheets and wires  🔍 [Fig. 89]
In order to take into account the participation of the soil and water layers above the tunnel in terms of mass, the geometry is completed with an extra polygon line \textit{mass} [Fig. 92], to which an equivalent boundary surface with free field characteristics is assigned.

\textbf{Main menu} \xrightarrow{\text{Geometry}} \xrightarrow{\text{Create}} \xrightarrow{\text{Add polyline}} \xrightarrow{\text{[Fig. 92]}}
4.2 Properties

We assign the material and properties to the top mass line [Fig. 94]. We use the distributed mass element class. We create a new material for mass elements class and we name it mass. For the line mass 2D material we choose distributed mass with 0 kg/m in tangential direction and 13800 kg/m in normal direction. The distributed mass represents 10 m of water and 2 m of sand as described in Section 1.3.1 [Fig. 96].

![Image](https://dianafea.com)
4.3 Boundary Conditions

Translational supports are imposed at the base of the tunnel’s walls [Fig. 97].

Main menu ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 97]
The lateral underwater soil is providing stiffness to the external vertical walls of the tunnel. For the scope of the example, this aspect is simplified by assuming the lateral edges supported in the horizontal direction [Fig. 100], since the focus of the analysis is on the upper part of the tunnel.

---

Main menu ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 100]
4.4 Mesh

Once the geometry is complete we generate the new mesh [Fig. 103]. There is no need to explicitly reassign the mesh properties since the elements size is kept.

Note that the cutting process allowed to keep all the previously assigned loads on the remaining portion of the initial surface. To check the applied load cases, we display them on the mesh [Fig. 104].

Figure 103: Mesh

Figure 104: Show loads in the mesh
5 Eigenvalue Analysis

5.1 Commands

To verify the dynamic behavior of the finite element model we perform an eigenvalue analysis. We add a new analysis and we call it Eigenvalue [Fig. 105]. We add to it a Structural eigenvalue command [Fig. 106] [Fig. 107].
The default values for parameters of an eigenvalue analysis are appropriate in most cases. Here we ask for some more eigenpairs than the default first one. We ask to calculate eight eigenfrequencies [Fig. 108]. In addition, we ask to take into account the self-weight of the structure, included in the Geometry load combination 1, within the free vibration analysis properties [Fig. 109].
We add to the Structural eigenvalue execute block the Calculate Rayleigh parameters command [Fig. 110]. Here we ask to calculate the Rayleigh parameters corresponding to 10% of damping for the frequency $\omega_1 = 125$ Rad/s and $\omega_2 = 1250$ Rad/s, in order to check the values previously provided for the materials assigned to the tunnel. The frequency values correspond to about 20 Hz and 200 Hz, respectively. Therefore, we explicitly input the desired values of frequency and corresponding damping ratios [Fig. 111].

We now run the analysis.

---

**Analysis browser** ➔ **Eigenvalue** ➔ Add command ➔ Structural eigenvalue Calculate Rayleigh parameters [Fig. 110]

**Analysis browser** ➔ **Eigenvalue** ➔ Structural eigenvalue ➔ Calculate Rayleigh parameters ➔ Edit properties [Fig. 111]
5.2 Results

5.2.1 Eigenfrequencies and Rayleigh Damping Parameters

When the analysis is finished, we can read in the messages window the eigenfrequency values [Fig. 112]. The analysis provides also the Rayleigh damping parameters for the frequency values defined during the analysis setup in correspondence with 10% damping [Fig. 112].
5.2.2 Eigenmodes

We present the different eigenmodes [Fig. 113] [Fig. 114].

Results browser ➔ Case ➔ Mode 1, Eigen frequency 6.3004 Hz, Top Load [Fig. 113]
Results browser ➔ Eigenvalue ➔ Output eigenvalue analysis ➔ Nodal results ➔ Displacements ➔ $D_{tXYZ}$ [Fig. 113] [Fig. 114]

Figure 113: Results browser

Figure 114: Mode 1

Eigenvalue
Mode 1, Eigen frequency 6.3711 Hz
Displacements $D_{tXYZ}$
min: 0.00m max: 1.00m
We repeat the operation for each eigenmode. We show here the modes from 2 to 7 [Fig. 115]–[Fig. 120].

Figure 115: Mode 2

Figure 116: Mode 4

Figure 117: Mode 6

Figure 118: Mode 3

Figure 119: Mode 5

Figure 120: Mode 7
6 Transient Nonlinear Dynamic Analysis

6.1 Commands

We add now a new analysis and we call it *Transient Nonlinear* [Fig. 121]. We add to it a *Structural nonlinear* command [Fig. 122] [Fig. 123].
We make the *Transient effects* active for the nonlinear dynamic analysis [Fig. 124]. Within the dynamic effects, we use a consistent mass and damping matrix for which we use the current stiffness for Rayleigh damping [Fig. 125].
It is necessary that the transient nonlinear analysis starts (at time $t = 0$) from a static equilibrium situation. Therefore an initial nonlinear analysis is required to apply the static loads (self-weight and sand and water pressure). Therefore, we use the first execute block to apply the self-weight, included in the *Geometry load combination 1*. We rename it as Self-weight [Fig. 126]. We apply the *Geometry load combination 1* and we ask to perform two load steps of size 0.5 [Fig. 127]. It is good practice to insure convergence during the preliminary application of the static loads, therefore we ask to satisfy all the convergence norms [Fig. 128].
We need a second load steps execute block in order to apply the external pressure due to sand and water loads. We do this by duplicating the first execute block. We rename the new one as Pressure [Fig. 129]. Here we apply the Geometry load combination 2, containing the load cases related to water and sand pressure, and we ask to perform two load steps of size 0.5 [Fig. 130]. In order to insure the convergence we ask to satisfy all the convergence norms and we set a higher number of iterations [Fig. 131].

**Analysis browser** ➔ Transient Nonlinear ➔ Structural nonlinear ➔ Self-weight ➔ Duplicate [Fig. 129]

**Analysis browser** ➔ Transient Nonlinear ➔ Structural nonlinear ➔ Self-weight - copy ➔ Rename ➔ Pressure [Fig. 129]

**Analysis browser** ➔ Transient Nonlinear ➔ Structural nonlinear ➔ Pressure ➔ Load steps ➔ Edit properties [Fig. 130]

**Analysis browser** ➔ Transient Nonlinear ➔ Structural nonlinear ➔ Pressure ➔ Equilibrium iteration ➔ Edit properties [Fig. 131]
We add now a time steps execute block *Explosion* [Fig. 132] for the application of the transient load. We choose user specified time steps [Fig. 133].
For the dynamic load we use a convergence norm based on energy and we choose to continue the analysis even if convergence is not achieved [Fig. 134] [Fig. 135].

Figure 134: Equilibrium iteration properties

Figure 135: Convergence norm
For the *Transient Nonlinear* analysis we choose a *User selection* output [Fig. 136] and we add the following types of results Table 8. Finally, we run the analysis.

**Analysis browser** ➔ *Transient Nonlinear* ➔ *Structural nonlinear* ➔ Output ➔ Edit properties [Fig. 136]

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

| Total displacements       | DISPLA TOTAL TRANSL GLOBAL | –  |
| Total velocities          | VELOCI TOTAL TRANSL GLOBAL | –  |
| Reaction forces           | FORCE REACTI TRANS GLOBAL  | –  |
| External forces           | FORCE EXTERN TRANS GLOBAL  | –  |
| Total stresses            | STRESS TOTAL CAUCHY GLOBAL | Integration points |
|                          | STRESS TOTAL CAUCHY LOCAL  | Integration points |
|                          | STRESS TOTAL CAUCHY PRINCI | Integration points |
| Total strains             | STRAIN TOTAL GREEN GLOBAL  | Integration points |
|                          | STRAIN TOTAL GREEN LOCAL   | Integration points |
|                          | STRAIN TOTAL GREEN PRINCI  | Integration points |
| Plastic strains           | STRAIN PLASTI GREEN LOCAL  | Integration points |
| Crack strains             | STRAIN CRACK GREEN         | Integration points |
| Crack widths              | STRAIN CRKWDT GREEN PRINCI | Integration points |

Table 8: Required output data

Figure 136: Output properties

Figure 137: Results selection
6.2 Results after Initial Application of Static Loads

6.2.1 Displacements

We present the contour plot of the displacements $D_{tXYZ}$ due to the application of static loads [Fig. 138]. Here we see a maximum displacement of about 7 mm.
6.2.2 Cracks

We assess the crack formation by first presenting the normal crack strains $E_{knn}$ [Fig. 140]. Via the property view settings, we ask to scale the symbol size for the crack plot by their value [Fig. 141].
We also show the contour plot of the crack widths Ecw1 [Fig. 143]. Cracks appear in the roof’s outer side in correspondence with the connection of the middle vertical walls and on the two external corners. Other cracks appear in the middle areas of the bottom side of the roof, in accordance with the deformation.

Figure 143: Results browser

Figure 144: Crack widths
6.2.3 Stresses in Concrete

We present the stress distribution in the concrete due to the initial static loading by showing the contour plots of the principal tensile $S_1$ and compressive stresses $S_2$ [Fig. 145] [Fig. 147].

Figure 145: Results browser

Figure 146: Principal stresses $S_1$
Results browser ➔ Transient Nonlinear ➔ Output ➔ Element results ➔ Cauchy Total Stresses ➔ S2  

Figure 147: Results browser

Figure 148: Principal stresses $S_2$

Transient Nonlinear
Load-step 4, Load-factor 1.0000
Cauchy Total Stresses $S_2$
min: $-2.55e+06$N/m² max: $1.09e+06$N/m²

Gas Explosion in a Tunnel | https://dianafea.com
6.2.4 Plasticity in Reinforcement

We display the stresses in the reinforcements along their axial direction $S_{xx}$ [Fig. 149]. We notice that, compared to the crack plot shown in Figure 144, we can see higher tensile stress in the reinforcements at places where cracks arise. The yield stress has nowhere been reached. We confirm this by displaying the plastic strain $E_{pxx}$ [Fig. 151]. The contour plot shows all reinforcements entirely in blue [Fig. 152], indicating that for the applied initial static loads the reinforcements remain elastic.
6.3 Results for the Explosion Load

6.3.1 Results after Start of Explosion

6.3.1.1 Displacements

At time $t = 25$ ms the explosion is at the beginning of the ‘pressure plateau’ (see Figure 5). We show the results for this time by selecting the appropriate step. We hide all the reinforcements and we display the contour plot of the displacements [Fig. 153]. We notice that the internal wall already experienced significant deformations and it is probably not adequate to withstand the explosion load, requiring some strengthening measures [Fig. 154].
In order to easily remove from the view a portion of the model, as in this case one of the central walls, we select from the model window the elements that we wish to hide [Fig. 155]. Then, we create a new element set using the selected elements and we rename it as `Right_Wall` [Fig. 157] [Fig. 158].
We now hide the right central wall from the view and analyze the rest of the structure. We display again the contour plot of the displacements [Fig. 159], showing a maximum displacement of about 18 mm [Fig. 160].

**Figure 159:** Results browser

**Figure 160:** Displacements
6.3.1.2 Velocity Field

We show the velocity field as a vector plot [Fig. 161]. The maximum velocity is about 1.6 m/s [Fig. 162].
6.3.1.3 Cracks

We first show the contour plot of the crack widths $\text{Ecw1}$ over the whole structure in order to visualize the damage in the right central wall of the tunnel [Fig. 163] [Fig. 164]. Afterwards, in order to have a clear view of the cracking pattern on the roof of the tunnel, we hide the damaged central wall [Fig. 165] [Fig. 166].
6.3.2 Results at the End of Explosion

6.3.2.1 Displacements

At time $t = 150$ ms the explosion has come to an end (see Figure 5). We show the results for this time by selecting the appropriate step and then display the contour plot of the displacements [Fig. 167]. The maximum displacement has increased to 29 cm [Fig. 168].
6.3.2.2 Velocity Field

We show the velocity field as a vector plot [Fig. 169]. The maximum velocity is about 2 m/s [Fig. 170].
6.3.2.3 Cracks

We show the contour plot of the crack widths Ecw1 [Fig. 171]. We notice that the cracking pattern is characterized by a main vertical crack in the top middle section of the tunnel's roof and by inclined cracks in correspondence to the connection of the roof slab with the vertical walls [Fig. 172].

**Figure 171:** Results browser

**Figure 172:** Crack widths
6.3.3 Results at the End of Analysis

6.3.3.1 Displacements

We select the last step to assess the results at the end of the transient analysis, i.e., for \( t = 250 \) ms. We show the contour plot of the displacements [Fig. 173]. The maximum displacement has reached 38 cm [Fig. 174].
6.3.3.2 Velocity Field

We show the velocity field [Fig. 175]. The maximum velocity is about 1 m/s and its direction is now mainly pointing inward with respect to the explosion load, indicating that the structure is on the way back [Fig. 176].
6.3.2.3 Cracks

We show the contour plot of the crack widths for the final step Ecw1 [Fig. 177] [Fig. 178].

Figure 177: Results browser

Figure 178: Crack widths
6.3.3.4 Plasticity in Reinforcements

We display the stress \( S_{xx} \) [Fig. 179] [Fig. 180] and the plastic strains \( E_{pxx} \) [Fig. 181] [Fig. 182] in the reinforcements in the final stage of the analysis. The maximum stress is 500 MPa, corresponding to the steel yielding stress. The yielded areas of the reinforcement are in accordance with the crack opening distribution.
6.4 Time History Diagrams

The presentation of results in the previous sections gives a reasonable impression of their development in time. However, a more precise insight requires the presentation of the results in the form of a time history diagram. Therefore, we select a node in the top midspan of the roof slab [Fig. 183], and we make a time history plot for the vertical displacement TDtY. In the chart view window, we select all the time steps [Fig. 184].

Figure 183: Node selection

Figure 184: Vertical displacement time history plot
Similarly, we make a time history plot for the vertical velocity TVtY [Fig. 186].

Note that the two time history plots confirm the development of the displacement and velocity as obtained in the previous sections.
Appendix A  Additional Information

Folder: Tutorials/ExplosionTunnel

Number of elements ≈ 850

Keywords:
- ANALYS: dynami nonlin physic transi.
- CLASS: large.
- CONSTR: suppor.
- ELEMET: c6tm cq16e ct12e grid mass pstrai reinfo taper.
- LOAD: edge elemen force functi hydro time weight.
- MATERI: crack dampin elasti harden hordyk isotro massli parabo plast rotati soften strain totstr viscou vonmis.
- OPTION: direct newmar newton regula units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy crack crkwdt displa extern force green plast princi reacti strain stress total veloci.

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.