Thermal-Stress Analysis of a Buttress Dam
Outline

1 Description
   1.1 Case Study .................................................. 3
   1.2 Geometry Layout ........................................... 4
   1.3 Material Properties ....................................... 5

2 Finite Element Model ........................................ 6
   2.1 Geometry .................................................... 8
   2.2 Properties .................................................. 27
      2.2.1 Concrete ............................................... 27
      2.2.2 Rock ................................................... 29
      2.2.3 Steel ................................................... 30
      2.2.4 Interface ............................................... 38
   2.3 Boundary Conditions ....................................... 41
      2.3.1 Static .................................................. 41
      2.3.2 Thermal ................................................ 44
   2.4 Loads ....................................................... 86
   2.5 Mesh ......................................................... 87

3 Staggered Analysis ........................................... 89
   3.1 Commands .................................................... 89
   3.2 Results ...................................................... 100
      3.2.1 Temperature ............................................ 100
      3.2.2 Structural Nonlinear .................................. 102

Appendix A Additional Information .......................... 108
1 Description

1.1 Case Study

This case study presents a staggered analysis for a concrete monolith formed by a front plate with a supporting buttress. This structure is subjected to the following load sets: self-weight, hydrostatic pressure and variation of seasonal temperatures.

Maximum and minimum mean temperatures are considered during Summer and Winter in a steady state context. However, to simulate the cyclic effect of thermal stresses due to seasonal changes on the monolith (e.g. cracking) we perform a pseudo heat transient analysis followed by a nonlinear structural analysis.

The structure is simulated under two different conditions:

- before installation of an insulation wall (period of two years)
- after installation of an insulation wall (period of three years)

Concrete material model is defined as total strain crack model with linear tension softening and Thorenfeldt compression curve. The impact of thermal load is only considered for the concrete and the bottom rock foundation. Therefore, the effect of variation of temperature on the grid reinforcement is ignored.
1.2 Geometry Layout

The geometry of the buttress dam is illustrated in Figure 1.

Figure 1: Geometry of the buttress dam: side view (left) and 3D view (right)
1.3 Material Properties

We define the material and physical properties of the model for the concrete, rock and steel. Nonlinearities are considered only for concrete.

Table 1: Linear material properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Concrete</th>
<th>Rock</th>
<th>Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>2.5e10</td>
<td>6e10</td>
<td>2.06e11</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.2</td>
<td>0.2</td>
<td>0.3</td>
</tr>
<tr>
<td>Thermal expansion coefficient</td>
<td>1e-5</td>
<td>1e-5</td>
<td>1e-5</td>
</tr>
<tr>
<td>Mass density</td>
<td>2300</td>
<td>2500</td>
<td>7800</td>
</tr>
<tr>
<td>Conductivity*</td>
<td>2.5</td>
<td>2.5</td>
<td>-</td>
</tr>
<tr>
<td>Thermal capacity</td>
<td>2.3e6</td>
<td>2.3e6</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 2: Nonlinear material properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Concrete</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile curve</td>
<td>Linear-crack energy</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>2.5e6</td>
</tr>
<tr>
<td>Mode-I tensile fracture energy</td>
<td>120</td>
</tr>
<tr>
<td>Compression curve</td>
<td>Thorenfeldt</td>
</tr>
<tr>
<td>Compressive strength</td>
<td>2.85e7</td>
</tr>
</tbody>
</table>

Note: users are advised to keep units consistent throughout the parameters entry. In this example we use transient analysis features. Therefore, it is important that the time unit is consistent with the time function, which we define later.
2 Finite Element Model

For the modeling session we start a new project for structural and heat flow analysis [Fig. 2]. The dimensions of the domain for the 3D model are set equal to 100 m. We use quadratic hexagonal finite elements with linear interpolation for the mid-side nodes.
We choose meter for the unit length, kilogram for mass, newton for force, second for time, celsius for temperature.

Figure 3: Geometry browser

Figure 4: Property panel - units
2.1 Geometry

First we create a sheet for each part of the model which represents its cross section. Then we extrude them in order to give the proper width converting them into solid shapes. Finally we move the solid shapes to the correct location. We split the model into three main parts: the wall, the buttress and the rock.

The wall consists of two parts: the bottom and top wall.
We extrude the two sheets to create the solid shapes of the wall.

**Figure 8:** Geometry - extrude polygon sheet

**Figure 9:** Geometry - initial position of Wall
We select the solid shapes of the wall and we move them to have symmetry in the $XZ$ plane. We change the color of these solid shapes that belong to the wall in order to easily identify the different parts of the model.

**Main menu** ➔ Geometry ➔ Modify ➔ Move shape [Fig. 10]

**Geometry browser** ➔ Geometry ➔ Shapes ➔ Wall Bottom, Wall Top ➔ Click on the color box and select the new color from the panel [Fig. 11] [Fig. 12]

---

**Figure 10:** Geometry - move shape

**Figure 11:** Geometry browser - change color of Wall

**Figure 12:** Geometry - final position of Wall
The buttress consists of three parts: the upstream and downstream buttress and the insulation wall. As before, we create a sheet for each part.

Main menu ➔ Geometry ➔ Create ➔ Add polygon sheet 🗓️ [Fig. 13] - [Fig. 16]

![Fig. 13: Geometry - add polygon sheet](image1)

![Fig. 14: Geometry - add polygon sheet](image2)

![Fig. 15: Geometry - add polygon sheet](image3)

![Fig. 16: Geometry - front view of the three sheets for the buttress](image4)
We extrude each sheet to create the solid shapes of the buttress.

**Figure 17:** Geometry - extrude polygon sheet

**Figure 18:** Geometry - initial position of Buttress
We move the solid shapes of the buttress to have symmetry in the $XZ$ plane. We change the colors of the buttress shapes to identify them more easily. The process for changing the color is the same as for the wall: we go to the geometry browser, click the rectangle which is on the right hand side of the respective shape and choose the desired color from the color menu [Fig. 11].

Main menu → Geometry → Modify → Move shape  
Geometry browser → Geometry → Shapes → Buttress Upstream, Isolation Wall, Buttress Downstream → Click on the color box and select the new color from the panel  

Figure 19: Geometry - move shape

Figure 20: Geometry - final position of Buttress
The rock consists of one part and we create it by adding a sheet. Then we extrude it.

Figure 21: Geometry - add polygon sheet

Figure 22: Geometry - front view of the sheet for the rock
We extrude the sheet of the rock to create the solid shape. Then, we move the two faces to extend the borders of the rock.
We move the solid shape of the rock to have symmetry in the $XZ$ plane. We also change the color of the rock shape based on the process described in the previous steps [Fig. 11].

![Figure 27: Geometry - move shape](image)

![Figure 28: Geometry - final position of Rock](image)
We create the grid reinforcement for the front side of the wall. First, we create the grid reinforcement for Wall Bottom. We switch the geometry selection to faces and from the model viewer we choose the front face of Wall Bottom. We extract the face and a new sheet is shown in the geometry browser. We rename Sheet 1 to Grid Bot Plate 1.

---

**Main menu** → Viewer → Selection mode → Face selection → Select the front face of shape Wall Bottom  
**Geometry browser** → Geometry → Shapes → Sheet 1 → Rename → Grid Bot Plate 1  

[Fig. 29]  
[Fig. 30]  
[Fig. 31]  

---

**Figure 29:** Model viewer - select face  
**Figure 30:** Geometry browser - extract face  
**Figure 31:** Geometry browser - rename sheet

---
We use the latter process to create the grid reinforcement for *Wall Top*.

**Main menu** → Viewer → Selection mode → Face selection → Select the front face of shape *Wall Top*  
**Main menu** → Geometry → Modify → Extract → Rename → Grid Bot Plate 1  

[Fig. 32]  

**Geometry browser** → Geometry → Shapes → Sheet 1 → Rename → Grid Bot Plate 1  

[Fig. 33]  

**Geometry browser** → Geometry → Shapes  

[Fig. 34]  

**Figure 32**: Model viewer - select face  
**Figure 33**: Geometry browser - extract face  
**Figure 34**: Geometry browser - rename sheet
The concrete coverage is 50 mm, so we move the extracted faces inside the wall.

---

**Main menu** ➔ Geometry ➔ Modify ➔ Move shape  🌆 [Fig. 35] -  [Fig. 37]
We create the grid reinforcement for the back side of the wall. First we create the front grid reinforcement of *Wall Bottom*.

**Main menu** → **Viewer** → **Selection mode** → **Face selection** → Select the back face of shape *Wall Bottom* [Fig. 38]

**Main menu** → **Geometry** → **Modify** → **Extract**

**Geometry browser** → **Geometry** → **Shapes** → **Sheet 1** → Rename → **Grid Bot Plate 2** [Fig. 39] [Fig. 40]

---

**Figure 38:** Model viewer - select face

**Figure 39:** Geometry browser - extract face

**Figure 40:** Geometry browser - rename sheet

---

Thermal-Stress Analysis of a Buttress Dam | https://dianafea.com
We create the back grid reinforcement of *Wall Top*.

**Fig. 41** Model viewer - select face

**Fig. 42** Geometry browser - extract face

**Fig. 43** Geometry browser - rename sheet
The concrete coverage is 50 mm, so we move the extracted faces inside the wall.
We create the grid reinforcement for the left and right side of the buttress.

Figure 47: Model viewer - select faces

Figure 48: Geometry browser - extract faces

Figure 49: Geometry - unite shapes
The concrete coverage is 50 mm, so we move the extracted faces inside the buttress.

Figure 50: Geometry browser - rename sheet
Figure 51: Geometry - move shape
Figure 52: Geometry - move shape
Figure 53: Coverage buttress grid
We create the additional grid reinforcement for the buttress. The concrete cover is 50 mm.
We imprint the *Buttress Upstream* shape to *Wall Bottom* and *Wall Top* to split the back faces of the wall into three sub-faces. In this way, we distinguish between the faces which are in contact with the buttress and the air.

**Main menu** ➔ Geometry ➔ Modify ➔ Imprint shapes 🔄 [Fig. 57] [Fig. 58]
2.2 Properties

2.2.1 Concrete

We assign the material properties to Wall Bottom, Wall Top, Buttress Upstream, Isolation Wall and Buttress Downstream. The linear and nonlinear properties of the concrete are reported in Table 1 and Table 2 respectively.
2.2.2 Rock

We assign the material properties to Rock. The properties of the rock are reported in Table 1.
2.2.3 Steel

We assign the material properties to the grid reinforcements. Each grid has different geometry, so they are defined separately. We start with the reinforcement of Wall Bottom [Fig. 70].

Select the correspondent reinforcement grid in the Geometry browser.

Main menu → Geometry → Assign → Reinforcement properties [Fig. 71]

Reinforcement properties → Material → Add material [Fig. 72] → Edit material [Fig. 71]

Reinforcement properties → Geometry → Edit geometry [Fig. 73]

Figure 70: Wall Bottom - reinforcement grid
Figure 71: Wall Bottom - assign properties
Figure 72: Steel - add new material
Figure 73: Wall Bottom - geometry
We use the same material and geometry for the *Grid Bot plate 2*.

---

*Select the correspondent reinforcement grid in the Geometry browser>*

**Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties**  

![Figure 74: Wall Bottom - assign properties](image)

Note: You can also group the two *Grid Bot Plates* in a reinforcement shape set and assign the properties directly to the shape set.
We assign the same material properties to the reinforcement grid of Wall Top [Fig. 76] and define a different geometry.
We use the same material and geometry for the Grid Top Plate 2

Note: You can also group the two Grid Top Plates in a reinforcement shape set and assign the properties directly to the shape set.
We assign the properties to the reinforcement grid of Buttress: we start with the Grid Buttress 1 [Fig. 79].

<Select the correspondent reinforcement grid in the Geometry browser >
Main menu → Geometry → Assign → Reinforcement properties [Fig. 80]
Reinforcement properties → Geometry → Edit geometry [Fig. 81]
We continue with the Grid Buttress 2 [Fig. 83], that has the same material and geometry as Grid Buttress 1.

Note: You can also group the two Grid Buttress in a reinforcement shape set and assign the properties directly to the shape set.
We assign the properties to the reinforcement grid of Link [Fig. 83]. First we create a new direction based on the inclined sheet and then we set the geometry of the grid reinforcement. We start with Grid Link 1

Based on Figure 83, in order to set the new direction we compute the sine and cosine of the angle of the sheet, \( \sin(57^\circ) = 0.83867 \) and \( \cos(180^\circ + 57^\circ) = -0.54464 \). We fill the dialog accordingly [Fig. 85].
We continue with the Grid Link 2 [Fig. 83], that has the same material and geometry as Grid Link 1.

<Select the correspondent reinforcement grid in the Geometry browser >

Main menu ➔ Geometry ➔ Assign ➔ Reinforcement properties 🕖 [Fig. 87]

![Reinforcement properties](image)

Figure 87: Buttress - assign properties

Note: You can also group the two Grid Link in a reinforcement shape set and assign the properties directly to the shape set.
2.2.4 Interface

We introduce an interface between the buttress and the rock. We can calculate the normal $K_n$ and tangential $K_t$ stiffness parameters for the plane interface by these general guidelines:

$$K_n \approx 100 \sim 1000 \frac{E_{\text{adj}}}{l_{\text{el}}}$$
$$K_t \approx \frac{K_n}{100} \text{ (Generally in order of 10 lower than } K_n)$$

where $E_{\text{adj}}$ is the elastic modulus of the adjacent mesh set and $l_{\text{el}}$ is the characteristic length of an element.

Since this interface is not going to be used as an insulation layer, we are going to introduce a conduction coefficient determined by the general guidelines:

$$k \approx 100 \sim 1000 \frac{k_{\text{adj}}}{l_{\text{el}}}$$

where $k_{\text{adj}}$ is the conduction coefficient of the adjacent mesh set and $l_{\text{el}}$ is the characteristic length of an element.

So, we calculate the initial parameters of the plane interface as follows:

$$K_n \approx 100 \frac{E_{\text{concrete}}}{l_{\text{el}}} = \frac{100 \times 2.5 \times 10^{-6}}{0.5} = 5 \times 12 \ N/m^3$$
$$K_t \approx 5 \times 8 \ N/m^3 \text{ (In this example we consider a low tangential stiffness in order to simulate sliding effect.)}$$
$$k \approx 1000 \ W/m^2\degree C \text{ (This parameter will be introduced to DIANA through material assigned to the interface.)}$$

*Please note that these formulas are only used as the initial guess and should be calibrated later on.

*Please note that if we want to have full conductivity in the interface, the value for the conduction coefficient $k$ should be high enough. Lower value of the conduction coefficient $k$ means less thermal exchange between two sides of the interface.
Figure 88: Interface between buttress and rock

Figure 89: Assign interface properties

Figure 90: Interface - add new material
Figure 91: Interface - linear properties

Figure 92: Interface - heat flow
2.3 Boundary Conditions

2.3.1 Static

We restrict the translation in the $X$ direction for the front and back face of the rock.

![Figure 93: Front view - x constraints](image)

![Figure 94: Add face supports - x constraints](image)
We restrict the translation in the Z direction for the bottom faces of the rock.

Figure 95: Front view - z constraints

Figure 96: Add face supports - z constraints
We restrict the translation in the $Y$ direction for the side faces of the rock and wall.

Figure 97: Left view - $y$ constraints

Figure 98: Add face supports - $y$ constraints
2.3.2 Thermal

The idea for this analysis is to perform a steady state analysis. However, to simulate the resultant cracks due to temperature variation in Summer and Winter, we are going to introduce pseudo time functions and perform heat transient analysis. In addition, in a staggered analysis, choosing a transient analysis has an advantage of passing the resultant forces to the structural analysis automatically compared to the steady state analysis.

To this end, we define thermal boundary conditions to the model under two conditions:

1. without insulation wall
2. with insulation wall

In Figure 99 and Figure 100 we can see the impact of the temperature on the model during Winter and Summer respectively.
We introduce the cyclic effect of seasonal temperature changes (5 years in total) by a series of factorial functions in time. Also we use the same time frame to introduce the insulation wall to the model after two years. In other words, during the first two years we use the temperatures assigned to the model without insulation wall and afterward we consider the temperatures assigned to the model with insulation wall [Fig. 101].

*Temperature initialization* - To avoid any cracking in the dam structure due to resultant compressive stresses (summer temperature), we initialize the temperature in winter (starting from tensile behavior).

![Figure 101: Reference functions - thermal factors](https://dianafea.com)
Figure 102 and Figure 103 show the Winter and Summer temperatures in the structure, water and rock foundation. We create four different thermal boundary condition sets based on the season and the presence of the insulation wall. For each boundary condition set prescribed temperatures based on the two figures are applied. Also we assign the corresponding time function to simulate the variation of temperature from Winter to Summer [Fig. 101].

1. Boundary condition set: Winter no wall [Fig. 102]
2. Boundary condition set: Summer no wall [Fig. 102]
3. Boundary condition set: Winter with wall [Fig. 103]
4. Boundary condition set: Summer with wall [Fig. 103]
To set prescribed temperatures to the faces of the model, it is first required to attach a fixed temperature condition along these faces. This will keep temperature constant during the analysis.

Main menu ➔ Geometry ➔ Assign ➔ Add fixed temperatures [Fig. 104] [Fig. 105]
1. Boundary condition set: Winter no wall

We set the prescribed temperature of the buttress faces to -15 °C.

**Main menu ➔ Geometry ➔ Assign ➔ Add thermal conditions**

**[Fig. 106] [Fig. 107]**

**Figure 106: Buttress faces**

**Figure 107: Thermal boundary conditions - buttress - winter no wall**
We set the prescribed temperature of the bottom wall front face to 4 °C.

Figure 108: Wall bottom front face

Figure 109: Thermal boundary conditions - wall bottom - winter no wall
We set the prescribed temperature of the top wall front face to 2.5 °C.

Figure 110: Wall top front face

Figure 111: Thermal boundary conditions - wall top - winter no wall
We set the prescribed temperature of the wall back faces to -15 °C.

Figure 112: Wall top front face

Figure 113: Thermal boundary conditions - wall - winter no wall
We set the prescribed temperature of the buttress base faces to -2.5 °C.

**Main menu** ➔ Geometry ➔ Assign ➔ Add thermal conditions [Fig. 114] [Fig. 115]

**Figure 114:** Buttress base faces

**Figure 115:** Thermal boundary conditions - buttress base - winter no wall
To avoid any temperature gradient in the rock foundation, we assign thermal loadings to the bottom rock faces. The value of the temperature is equal to the value used for the buttress base. We set the prescribed temperature of the rock bottom faces to -2.5 °C.

Figure 116: Rock bottom faces

Figure 117: Thermal boundary conditions - rock - winter no wall
The temperature of a common edge between two surface boundary conditions with different temperature must be defined separately or else there will be a conflict for the boundary values along this line.

We set the prescribed temperature of the top wall edge to 2.5 °C.

**Figure 118:** Wall top edge

**Figure 119:** Thermal boundary conditions - wall top - winter no wall
The temperature of a common edge between two surface boundary conditions with different temperature must be defined separately or else there will be a conflict for the boundary values along this line.

We set the prescribed temperature of the buttress base edges to -2.5 °C.
Because we are about to perform a transient analysis, we must also specify the development of the ambient temperature with time. The time function [Fig. 123] is derived from Figure 101 and it shows the factors used to multiply the values of the prescribed temperature for this boundary condition set.

**Figure 122**: Boundary conditions set - winter no wall

**Figure 123**: Time function - winter no wall
2. Boundary condition set: Summer no wall

We set the prescribed temperature of the buttress faces to 25 °C.
We set the prescribed temperature of the bottom wall front face to 4 °C.

Figure 126: Wall bottom front face

Figure 127: Thermal boundary conditions - wall bottom - summer no wall
We set the prescribed temperature of the top wall front face to 7.5 °C.

Figure 128: Wall top front face

Figure 129: Thermal boundary conditions - wall top - summer no wall
We set the prescribed temperature of the wall back faces to 25 °C.

Figure 130: Wall back faces

Figure 131: Thermal boundary conditions - wall - summer no wall
We set the prescribed temperature of the buttress base faces to 7.5 °C.

Figure 132: Buttress base faces

Figure 133: Thermal boundary conditions - buttress base - summer no wall
To avoid any temperature gradient in the rock foundation, we assign thermal loadings to the bottom rock faces. The value of the temperature is equal to the value used for the buttress base. We set the prescribed temperature of the rock bottom faces to 7.5 °C.

[Fig. 134] [Fig. 135]
The temperature of a common edge between two surface boundary conditions with different temperature must be defined separately or else there will be a conflict for the boundary values along this line.

We set the prescribed temperature of the top wall edge to 7.5 °C.
The temperature of a common edge between two surface boundary conditions with different temperature must be defined separately or else there will be a conflict for the boundary values along this line.

We set the prescribed temperature of the buttress base edges to 7.5 °C.
We must also specify the development of the ambient temperature with time. The time function [Fig. 141] is derived from Figure 101 and it shows the factors used to multiply the values of the prescribed temperature for this boundary condition set.
3. Boundary condition set: Winter with wall

We set the prescribed temperature of the upstream buttress faces to 3 °C.

Figure 142: Buttress upstream faces

Figure 143: Thermal boundary conditions - buttress upstream - winter with wall
We set the prescribed temperature of the downstream buttress faces to -15 °C.
We set the prescribed temperature of the bottom wall front face to 4 °C.

Figure 146: Wall bottom front face

Figure 147: Thermal boundary conditions - wall bottom - winter with wall
We set the prescribed temperature of the top wall front face to 2.5 °C.

Figure 148: Wall top front face

Figure 149: Thermal boundary conditions - wall top - winter with wall
We set the prescribed temperature of the wall back faces to 3 °C.

Figure 150: Wall back faces

Figure 151: Thermal boundary conditions - wall - winter with wall
We set the prescribed temperature of the buttress base faces to -2.5 °C.
To avoid any temperature gradient in the rock foundation, we assign thermal loadings to the bottom rock faces. The value of the temperature is equal to the value used for the buttress base.

We set the prescribed temperature of the rock bottom faces to -2.5 °C.

**Figure 154:** Rock bottom faces

**Figure 155:** Thermal boundary conditions - rock - winter with wall
The temperature of a common edge between two surface boundary conditions with different temperature must be defined separately or else there will be a conflict for the boundary values along this line.

We set the prescribed temperature of the top wall edge to 2.5 °C.

Figure 156: Wall top edge

Figure 157: Thermal boundary conditions - wall top - winter with wall
The temperature of a common edge between two surface boundary conditions with different temperature must be defined separately or else there will be a conflict for the boundary values along this line.

We set the prescribed temperature of the buttress base edges to -2.5 °C. Note that the edges of the insulation wall are not taken into account.
We must also specify the development of the ambient temperature with time. The time function [Fig. 161] is derived from Figure 101 and it shows the factors used to multiply the values of the prescribed temperature for this boundary condition set.
4. Boundary condition set: Summer with wall

We set the prescribed temperature of the upstream buttress faces to 5 °C.
We set the prescribed temperature of the downstream buttress faces to 25 °C.

Figure 164: Buttress downstream faces

Figure 165: Thermal boundary conditions - buttress downstream - summer with wall
We set the prescribed temperature of the bottom wall front face to 4 °C.

Figure 166: Wall bottom front face

Figure 167: Thermal boundary conditions - wall bottom - summer with wall
We set the prescribed temperature of the top wall front face to 7.5 °C.

![Figure 168: Wall top front face](image)

![Figure 169: Thermal boundary conditions - wall top - summer with wall](image)
We set the prescribed temperature of the wall back faces to 5 °C.

**Figure 170: Wall back faces**

**Figure 171: Thermal boundary conditions - wall - summer with wall**
We set the prescribed temperature of the buttress base faces to 7.5 °C.

**Figure 172:** Buttress base faces

**Figure 173:** Thermal boundary conditions - buttress base - winter with wall
To avoid any temperature gradient in the rock foundation, we assign thermal loadings to the bottom rock faces. The value of the temperature is equal to the value used for the buttress base. We set the prescribed temperature of the rock bottom faces to 7.5 °C.
The temperature of a common edge between two surface boundary conditions with different temperature must be defined separately or else there will be a conflict for the boundary values along this line.

We set the prescribed temperature of the top wall edge to 7.5 °C.

Figure 176: Wall top edge

Figure 177: Thermal boundary conditions - wall top - summer with wall
The temperature of a common edge between two surface boundary conditions with different temperature must be defined separately or else there will be a conflict for the boundary values along this line.

We set the prescribed temperature of the buttress base edges to 7.5 °C. Note that the edges of the insulation wall are not taken into account.
We must also specify the development of the ambient temperature with time. The time function [Fig. 181] is derived from Figure 101 and it shows the factors used to multiply the values of the prescribed temperature for this boundary condition set.

**Geometry browser → Boundary conditions → Summer with wall → Edit time dependency  ☺ [Fig. 180]  [Fig. 181]**

**Figure 180**: Boundary conditions set - summer with wall

**Figure 181**: Time function - summer with wall
2.4 Loads

We apply the self-weight and the hydrostatic water pressure. The hydraulic head is 38.5 m based on Figure 1. The hydrostatic water pressure is applied on the front faces of the wall and the front top face of the rock.
2.5 Mesh

The mesh will be defined by setting the characteristic size of the elements as 0.5 m for all the solid shapes. Furthermore, the size of the rock vertical edges are set to 1.0 m resulting to a coarser mesh along its height. No properties are given to the grid reinforcements because they will follow the mesh size of the respective solid shapes.

Main menu ➔ Geometry ➔ Assign ➔ Mesh properties [Fig. 185] [Fig. 187]
We generate the mesh.

Figure 188: Mesh view of the solid shapes

Figure 189: Mesh view of the reinforcement grids
3 Staggered Analysis

3.1 Commands

To perform a staggered analysis, first we introduce a heat transient analysis followed by a structural nonlinear analysis. In this manner the resultant thermal forces will be automatically considered as part of active forces in the subsequent structural nonlinear analysis.
In the dialog of **Evaluate model** [Fig. 193], we check off the option for assembling heat capacity matrix (consistent/lumped) and the transient effect of temperature on elements is ignored. Please note that this analysis is pseudo transient analysis and we simulate the cyclic effect of temperature variation in different seasons by a time factor.

In the dialog of **Initial conditions** [Fig. 194], we check on **Initial temperature field** to start with temperatures as calculated in a steady-state analysis. If temperatures are available from a previous steady-state analysis, these temperatures are used. Otherwise a linear steady-state analysis is performed. In **Boundary case** [Fig. 194] we select load/boundary case 3, *Winter no wall* as the reference temperatures for initialization. The reason for selecting this load case is to start with concrete behavior in tension rather than compression (i.e. tensile behavior of concrete in downstream due to difference between default temperature and winter temperatures). This way the initiation and propagation of cracks become realistic. In **Type** [Fig. 194] we select **Nonlinear analysis** to initiate a nonlinear analysis, i.e., with time or temperature dependent material properties.
In the dialog of *Execute analysis* [Fig. 195], the option *Step sizes* is the number of time intervals that we will consider for variation of seasons. In this example we consider every two seasonal intervals (e.g. summer and winter) with 6 months laps. To be consistent with the defined time functions and time dependent material parameters units, we use a step of 15768000 seconds to represent a cycle of 6 months period. Since this cycle remains constant we multiply this step by the number of seasonal changes for this analysis (e.g. 9 cycles).

In the dialog of *Analysis output* [Fig. 196], we select the results we want to obtain [Fig. 197], which is the temperature.
In the same analysis we add *Structural Nonlinear* analysis command.

The process for structural nonlinear analysis is as followings:

1. *Stress initialization* due to self-weight and reset the resultant displacements to zero.
2. Imposing *hydrostatic water pressure* as part of mechanical load cases.
3. Defining *9 time dependent execute blocks* for considering the results/effect of heat transient analysis on the dam structure every 6 months period (e.g. 15768000 sec.)

First, we remove the default execute block because it is for load steps, we add *Execute steps - Start steps* and we rename it to *Self-Weight*.
In the Self-Weight execute block we add the Physic nonlinear options.

We set the Start steps [Fig. 202], Physic nonlinear options [Fig. 205], Equilibrium iteration [Fig. 206] and Output [Fig. 208] [Fig. 209] options.
For the equilibrium iteration procedure we use the Newton-Raphson method with a maximum of 20 iterations. For the convergence norm we use only force [Fig. 206]. With the user selection we choose to output results of displacements, forces, crack, plasticity, strains and stresses [Fig. 209].

Figure 206: Equilibrium iteration
Figure 207: Analysis browser
Figure 208: Output options
Figure 209: Output user selection
In the Structural nonlinear analysis command we add Execute steps - Load steps and we rename it to Hydrostatic Pressure.

We set the Load steps [Fig. 212], Equilibrium iteration [Fig. 213] and Output [Fig. 216] [Fig. 217] options.

For the load steps we choose the load set water pressure to be applied in two steps 0.5(2).
For the equilibrium iteration procedure we use the Newton-Raphson method with a maximum of 20 iterations. For the convergence norm we use the default options.
With the user selection we choose to output results of displacements, forces, crack, plasticity, strains and stresses [Fig. 217].
In the *Structural nonlinear* analysis command we add *Execute steps - Time steps* and we rename it to *Temperature1*.

We set the *Time steps* [Fig. 220], *Equilibrium iteration* [Fig. 221] and *Output* [Fig. 224] [Fig. 225] options.

The time steps used for this execution block is $1.57680 \times 10^7$ which is equal to 6 months.

For the equilibrium iteration procedure we use the Newton-Raphson method with a maximum of 20 iterations. For the convergence norm we use only displacement.
With the user selection we choose to output results of displacements, forces, crack, plasticity, strains and stresses [Fig. 225].

**Analysis browser** ➔ Staggered ➔ Structural nonlinear ➔ Temperature1 ➔ Add... ➔ Output - User [Fig. 222] [Fig. 223] [Fig. 224] [Fig. 225]

**Analysis browser** ➔ Staggered ➔ Structural nonlinear ➔ Temperature1 ➔ Output ➔ Edit properties [Fig. 222] [Fig. 223] [Fig. 224] [Fig. 225]
To create the remaining execute blocks we copy and rename *Temperature1* eight times. Finally we run the analysis.

**Analysis browser** → Staggered → Structural nonlinear → Temperature1 → Duplicate [Fig. 226]

**Analysis browser** → Staggered → Structural nonlinear → Temperature1 → Rename → Temperature2 [Fig. 227]

< Repeat the last two steps and rename accordingly [Fig. 228] >

**Analysis browser** → Staggered → Run selected analysis

---

Figure 226: Duplicate Temperature 1

Figure 227: Analysis browser

Figure 228: Analysis browser
3.2 Results

3.2.1 Temperature

We investigate the temperature field (PTE) in the model.

We show the contour plots of the temperature in the model for the different seasons with and without insulation wall. The contours are in accordance with the respective temperature values in Figure 102 and Figure 103.
Results browser → Case → Time-step 6, ...
Fig. 229
Results browser → Case → Time-step 5, ...
Fig. 229

Figure 232: Winter with wall

Figure 233: Summer with wall
3.2.2 Structural Nonlinear

We show the contour plots of the displacements TDtXYZ on deformed shape for the time steps 4, 11 and 12.
Staggered
Time-step 12, Time 0.14191E+09, Self weight
Displacements TDtXYZ
min: 0.00e+00m max: 1.55e-02m

Figure 237: Time step 12 - Displacements TDtXYZ
We show the contour plots of the maximum principal strains on deformed shape for the time steps 4, 11 and 12 to highlight the direction of the crack propagation. We change the number of contour levels from 8 to 16 for better visualization.

Results browser → Staggered → Output → Element results → Total strains → E1  
Property Panel → Result → Contour plot settings → Equidistant levels options → Number of contour levels → 15  
Results browser → Case → Time-step 4, ...  
Results browser → Case → Time-step 11, ...

Figure 238: Results browser E1  
Figure 239: Property panel - number of contour levels  
Figure 240: Time step 4 - Crack strains E1
Results browser ➔ Case ➔ Time-step 12, ...

**Figure 241: Time step 11 - Crack strains E1**

Staggered
Time-step 11, Time 0.12614E+09, Self weight
Total Strains E1
min: -6.01e-06 max: 4.10e-03

**Figure 242: Time step 12 - Crack strains E1**

Staggered
Time-step 12, Time 0.14191E+09, Self weight
Total Strains E1
min: -2.25e-06 max: 3.48e-03
We show the contour plots of the crack patterns on deformed shape for the time steps 4, 11 and 12 to illustrate the normal crack strains. The initial cracks are observed in the front faces of the wall and the buttress base, while when the insulation wall diagonal cracks occur around the opening in the buttress.
Figure 246: Time step 12 - Crack strains Eknn

Staggered
Time-step 12, Time 0.14191E+09, Self weight
Crack Strains Eknn
min: 0.00e+00 max: 2.81e-03
Appendix A  Additional Information

Folder: Tutorials/ButtressDam

Number of elements ≈ 28000

Keywords:
- ANALYS: flow flowst heat nonlin physic stagge transi.
- CLASS: large.
- CONSTR: suppor temper.
- ELEMEN: chx60 cq48i ctp45 flow grid hx8ht interf iq8ht potent reinfo solid struct taper tp6ht.
- LOAD: elemen face hydro node temper time weight.
- MATERI: conduc crack elasti harden isotro linear porosi rotati soften thoren totstr.
- OPTION: direct newton regula units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauhy crack displa force green plasti princ reacti status strain 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.