Early-age Hygro-Thermal Cracking of a Concrete Dam

| For LUSAS version:	18.0
For software product(s):	Any Plus version
With product option(s):	Nonlinear, Dynamic, Heat of Hydration, Fast Solvers.

Description

A small concrete dam has a height of 6m; a width at the base of 3m and a width at the top of 2m. It has a length of 20m along its vertical face which spans an arc of 30° between two rock faces. It is to be constructed in three stages, each 2m high, and with 10 days between stages.

For the purposes of this example the dam is assumed to be symmetrical about its vertical axis to reduce the computation time. In doing so, it is accepted that, if calculated, any crack pattern may not be symmetrical.

Ground beneath and to the side of the dam is modelled to account for temperature dissipation through it and to better model the restraints to the thermal expansion that will decrease as the ground is heated.

Units used are N, m, kg, s, C throughout.
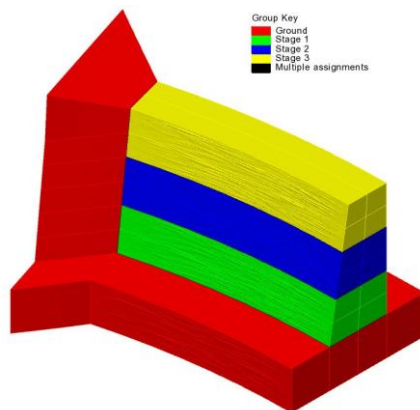
Objectives

The required output from the analysis consists of:

- ❑ Temperature and water saturation in the dam during construction.
- ❑ Prediction of cracking and crack widths.
- ❑ Change of curing conditions to reduce cracking.

Steps involved

- ❑ **Step 1: Modelling the staged construction process to obtain early-age hygro-thermal cracking and structural results**

- ❑ **Step 2: Crack width reduction by improving curing conditions**

- ❑ **Step 3: Crack width reduction by pre-cooling the concrete.**

Note that solving a hygro-thermal analysis at each stage will take an appreciable time.

Keywords

3D, Dam, Concrete, Coupled, Early-age, Hygro-Thermal, Staged Construction, Activate, Deactivate, Heat of hydration, Semi-coupled analysis, Crack widths

Associated Files

- ❑ **concrete_dam_3D_geometric_modelling.mdl** contains the basic model geometry. The example will use this model as a starting point.
- ❑ **concrete_dam_3D_modelling.vbs** carries out the complete modelling.

Hygro-Thermal / Structural Analysis

Concrete is generally unique among structural materials in that it interacts with its environment undergoing unavoidable physical and chemical volume changes. Concrete exhibits certain characteristics such as ageing, creep and shrinkage, which are collectively known as time dependent deformations; these are dealt with in other examples in this manual. See the example 'Staged Construction of a Concrete Tower with Creep' if age and creep is of interest to you.

The early-age hygro-thermal analysis considers the process of hydration. It is an exothermic chemical reaction that releases large amounts of heat. During the curing process the water saturation in concrete decreases due to self–desiccation (water becomes chemically bounded in the hydration products) and drying from exposure to the environment. The decrease of the internal water saturation causes shrinkage which may potentially cause the concrete to crack.

The semi-coupled structural analysis uses the temperature, water saturation and degree of hydration results from the hygro-thermal analysis to compute the evolution of the material properties with curing and the thermal and shrinkage strains. The resulting stresses can cause cracking. A specialised LUSAS concrete material model is used to model the crack distribution and compute the crack widths.

Model supplied / meshing considerations

Model creation

The half dam model supplied for this example was created by initially defining a 2D section through the dam in the XZ plane. The 3D half-model was created by sweeping the 2D surfaces through a rotation of 15° about the Z axis to create volumes representing the dam wall.

Line directions

To model the thermal and hygro gradients at each construction stage the directions of the lines defining each surface all point to the corners of the staged construction cross sections and the line mesh assigned to these lines are graded to give smaller elements to these regions.
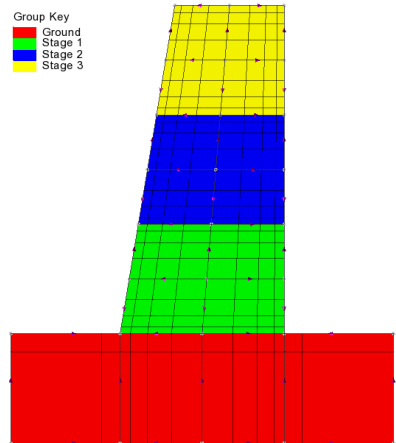
Line mesh

❑ A line mesh of **4** elements with a spacing ratio of first to last element of **3** is assigned to all lines of the surfaces representing the concrete.

❑ A line mesh of **2** elements with a spacing ration of first to last element of **5** is assigned to the lines defining the surfaces representing the ground.

The resulting mesh arrangement as shown on the preceding image will be seen as soon as a volume mesh has been assigned to the model.

The spacing of the mesh in the vertical Z direction makes elements smaller towards the top surfaces which are exposed to several hygro-thermal loadings. If the elements are too big along Z (i.e. the direction of heat and moisture transfer) spurious oscillations could be computed. A simple rule is 'bigger elements require a bigger time step'. However, the length of the time step is predetermined by the timescale of the heat of hydration. Time steps in the range of 0.5 to 3 hours (depending also on the time variations of the boundary condition) are recommended for the first day of curing. If oscillations and obvious physically incorrect results are computed elements should be made smaller. Ultimately, as in all finite element solutions, a mesh sensitivity assessment should be carried out to arrive at an optimum mesh density and arrangement.

After the 2D section was swept to create a 3D half-model:

❑ A uniform line mesh of **25** divisions was assigned to the arc lines defining the curvature of the dam faces.

❑ A sweep and rotation at 3° of the end surfaces and a subsequent enlargement of the end surfaces of the abutment created the ground at the abutment to provide a larger volume for the thermal dissipation.

Groups

The use of Groups to identify particular elements of the model is recommended.

❑ Groups representing the side faces and tops of each construction stage are predefined in the supplied model. These allow for easy assignment of attributes, and for viewing results on a slice through the dam.

❑ Groups representing each construction stage are to be user-defined as part of working through this example.

Concrete material models

LUSAS nonlinear concrete material models 105 and 109 are based on the smeared crack concept. This imposes limitations on the size of the element based on material properties elastic modulus, tensile strength, and fracture energy – which should not exceed approximately 0.35±0.05m. If it does, the actual value for the fracture energy is increased internally.

Step 1: Modelling the staged construction process

Running LUSAS Modeller

For details of how to run LUSAS Modeller see the heading *Running LUSAS Modeller* in the *Introduction to LUSAS Worked Examples* document

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Loading the model

File
 Open…

To start this example, open the read-only file:
concrete_dam_3d_geometric_modelling.mdl that is located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Depending upon your access rights it may be necessary to copy this file to another folder before it can be opened, and remove the read-only protection before it can be saved to the new folder. Once opened:
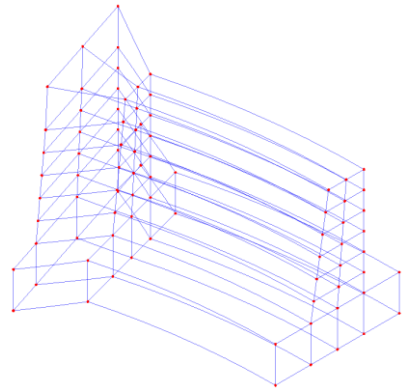
The basic dam geometry will be displayed.

File
  Save As…

- Save the file into a folder named **\<LUSAS Installation Folder>\Projects\dam**

- Save the model into this new folder as **Concrete_Dam_3D**

Note. Use the Undo button to correct any mistakes made since the last save was done.

Defining and Assigning Mesh Attributes

Attributes
  Mesh   >
    Volume…

- On the Volume Mesh dialog accept the default structural, hexahedral, linear element **HX8**. On the Hygro-thermal tab, the corresponding hygro-thermal element HHT8 will be automatically selected.

- Ensure a **Regular** mesh with an **Automatic** spacing is selected. Enter the dataset name as **HTM Coupled Bricks** and click **OK.**

- In the view window select the whole model and drag and drop the **HTM Coupled Bricks** mesh from the
  Treeview onto the selected features. This will generate the mesh shown.
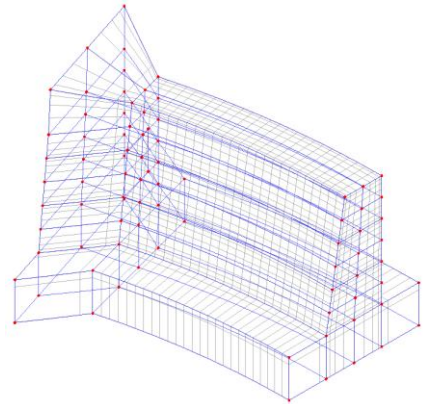
Defining Groups

It is useful in a staged construction analysis to make use of the group facility in LUSAS. This allows parts of the model to be displayed in isolation according to the construction stage being modelled and simplifies the assignment of attributes and the viewing of results.

- Rotate or view the model from a suitable angle and, by holding-down the **V** key to select only Volume features, box-select all the volumes representing the layer of ground beneath the dam. Then press the **Shift + V** keys to box-select the volume of the side wall and add it to the initial boxed selection.
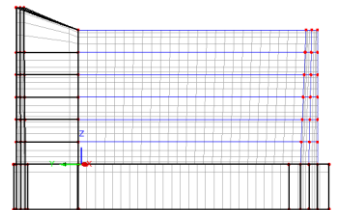
Geometry
| Group | > |
| New Group | |

- 🖲 Enter the group name as **Ground** and click **OK** to finish.

- Repeat the procedure, holding-down the **V** key to box-select only the volumes representing stage 1 (as shown), to create a group named **Stage 1.**

- Repeat again box-selecting the appropriate volumes to create a group called **Stage 2**

- Repeat again to create another called **Stage 3**.
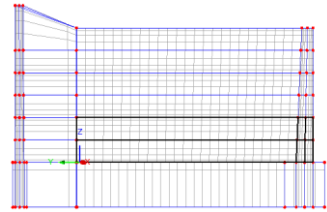
✏️ **Tip.** To see the contents of each group, in the Groups 🖲 Treeview right-click on a group name and select **Select members.**

Defining Material Properties

In a coupled hygro-thermal analysis, both hygro-thermal and structural material properties must be defined.

Concrete (Hygro-thermal)

Attributes
Material	>
Hygro-Thermal	>
Isotropic....	

Default values for all **hygro-thermal properties** are provided in the dialog.

- Change Cement type to **Type V** (a sulphate resistant concrete).

- Enter name Concrete Hygro-Thermal and click OK.

Isotropic

Hygro-thermal

Material type   Hygro-thermal concrete ▼

Input density and specific heat capacity for
- ⦿ Kiln dry concrete
- ○ Concrete at ambient conditions

Porosity
- ⦿ Automatically determined
- ○ Specify

	Value
Thermal conductivity	2.0
Specific heat capacity	900.0
Density	2.4E3
Permeability	0.1E-18
Entrapped air volume ratio	0.023

Advanced...

Exothermic behaviour
- ○ None
- ⦿ Concrete Heat of Hydration

Cement type   Type V ▼

	Value
Mass of cement per unit volume	400.0
Water/Cementitious mass ratio	0.4
Mass of slag per unit volume	0.0
Mass of fly ash per unit volume	0.0
CaO content of fly ash (%)	0.0
Delay for heat of hydration	0.0

Name Concrete Hygro-Thermal ▼ ▲ (new)

OK   Cancel   Apply   Help

Note. Concrete heat of hydration loading is a type of internal heat generation. From the input parameters LUSAS automatically calculates the amount and rate of this internal heat generation based on formulae presented in published research by Schindler and Folliard. For more information see reference [S18] in the *LUSAS Theory Manual.* In addition to this LUSAS calculates the degree of hydration and the amount of water used during the hydration (self desiccation) which would be used in the structural analysis to compute the shrinkage strains.

Note. Porosity is computed automatically taking into account the entrapped volume ratio, the mass of all cementitious material and the water/cementitious mass ratio.

Ground (Hygro-thermal)

Attributes	
Material	>
Hygro-Thermal	>
Isotropic....	

To model the porous ground (rock) the same material properties as concrete may be used.

- Select **None** for the Exothermic behaviour.

- Enter name **Ground Hygro-Thermal** and click **OK**.

Concrete (Structural)

Attributes	
Material	>
Structural	>
Concrete (Nonlinear)...	

- Select a **Transient smooth multi-crack** concrete model is selected, and select (even if it is visible by default) the Creep model **CEB-FIP 1990**, and specify that shrinkage is **Included**.

- Enter a Young's modulus of **30E9** Pa, a Poisson's ratio of **0.2**, and a mass density of **2.4E3** kg/m^3**,** and a Coefficient of thermal expansion of **10E-6**.

- Leave all other values as their default settings and enter the name **Concrete Structural** and click **OK**.

Ground (Structural)

Attributes
  Material   >
    Structural >
      Isotropic....

- Enter a Young's modulus of **40E9** Pa, a Poisson's ratio of **0.2**, and a mass density of **2.0E3** (kg/m^3). Click the **Thermal expansion** check box and set a Coefficient of thermal expansion of **10E-6**.

- Enter name **Ground Structural** and click **OK**.

Assign the Material Properties

The hygro-thermal and structural materials must be assigned to all elements.

- In the Groups 🔲 layer right click on **Ground** and select **Invisible.**

- Press the **Ctrl** + **A** keys to select all the visible features.

- Drag and drop **Concrete Hygro-Thermal** material from the 🔴 Treeview. Ensure **Assign to volumes** is selected for **Analysis 1 (Hygro-Thermal)** and click **OK**.

- With the features still selected, drag and drop **Concrete Structural** material from the 🔴 Treeview. Ensure **Assign to volumes** is selected for **Analysis 1 (Structural)** and click **OK**.

- In the Groups 🔲 layer right click on **Ground** and select **Set as Only Visible.**

- Press the **Ctrl** + **A** keys to select all the visible features.

- Drag and drop **Ground Hygro-Thermal** material from the 🔴 Treeview. Ensure **Assign to volumes** is selected for **Analysis 1 (Hygro-Thermal)** and click **OK**.
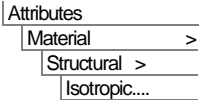
- With the features still selected, drag and drop **Ground Structural** material from the 🎯 Treeview. Ensure **Assign to volumes** is selected for **Analysis 1 (Structural)** and click **OK**.

- In the view window right-click and select **All visible** to re-display the hidden featuires

Initial conditions

Attributes
  Loading
    Hygro-thermal…

- Ensure **Initial conditions** is selected and click **Next**. Set the temperature to be **20** (^{o}C). Ensure the **Cured concrete** option is selected and enter the concrete relative humidity as **0.9**. Enter the name **Initial conditions Ground**. Click **Apply**.

- Then deselect the **Cured concrete** option. Enter the name **Initial conditions Concrete**. Click **Finish**.

Thermal Loading
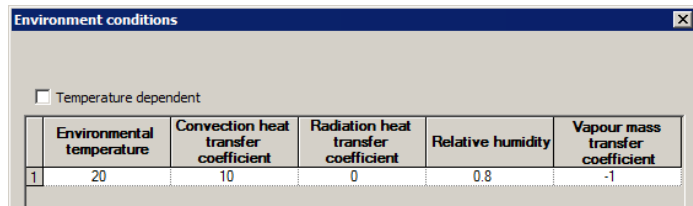
Concrete will cool and dry when in contact with the air, which can be modelled by defining environmental conditions. For accurate modelling of the thermal exchange with the ground, the latter has to be discretised explicitly. For simplicity, the effect of the ground will also be modelled by specifying environmental conditions.

Environmental condition for surfaces exposed to air

Attributes
  Loading
    Hygro-thermal…

- Select **Environmental Conditions** and click **Next**.

Enter the following data:

- Environmental temperature: **20.0** (ºC)

- Convection heat transfer coefficient **10.0** (J/s·m^2·°C)

- Radiation heat transfer coefficient **0.0** (or leave blank)

- Relative humidity: **0.8**

- Vapour mass transfer coefficient: **-1.0**

	Environmental temperature	Convection heat transfer coefficient	Radiation heat transfer coefficient	Relative humidity	Vapour mass transfer coefficient
1	20	10	0	0.8	-1

Environment conditions

☐ Temperature dependent

- Enter the attribute name as **ENVT – Air** and click **Apply** to allow another loading to be defined.

Environmental condition for formwork surfaces

Change the following parameters to the values shown:

- Convection heat transfer coefficient: **7.0** (J/s·m^2·ºC)

- Vapour mass transfer coefficient: **0.0**

- Enter the attribute name to **ENVT – Formwork** and click **Finish.**

Note. A negative vapour mass transfer coefficient will invoke its computation based on the Chilton-Colburn analogy. A zero vapour mass transfer coefficient implies a perfectly isolated hygro boundary and the given value of relative humidity is of no importance. If necessary see the relevant online help topics for more details.

Prescribed temperature

Attributes
Loading   >
      Hygro-Thermal…

- Select **Prescribed** and click **Next**. Set Temperature **Fixed** and enter **20** ºC. Leave the Concrete relative humidity set to **Free**. Enter a name of **Prescribed T=20** and click **Finish.**

Loading will be assigned when the appropriate loadcases have been defined later in the example.

Modelling Staged Construction

To correctly model the staged construction of the dam, volumes (and hence elements) in the model must only be included in the analysis after they have been constructed. Similarly, loading attributes (such as surface heat transfer) will need to be applied only for a certain portion of the analysis to reflect the construction stage being considered. In general, model attributes are assigned and changed using Loadcases. Loading attributes can either be applied for each loadcase if they remain constant during the loadcase (note that when such a loadcase finishes its loads are 'forgotten') or via load curves if the loads vary in time.

A hygro-thermal transient analysis may simulate concrete behaviour over a period of several hours or even years. For an analysis covering a long period, varying the time step would be required. For example, during the hydration process a time step of about 1 hour would be needed, but for a creep analysis a time step of several days would be required. The magnitude of the transient time step can be changed if a new loadcase is defined.

Creating Activation and Deactivation datasets

Modelling of staged construction processes that require activation and deactivation of elements is carried out in LUSAS using the Activate and Deactivate facility.

Attributes
    Activate and
    Deactivate…

- Choose the **Activate** option and click **Next.**

- Enter the attribute name as **Activate** and click **Apply**. Then click **Back** so the dialog can be reused to define the deactivate attribute.

- Choose the **Deactivate** option and click **Next.**

- Enter the attribute name as **Deactivate**, select **Percentage to redistribute** and leave the value as **100**%, then select **Finish.**

Defining Loadcase Properties

In the ⏰ Treeview two analysis entries can be seen: Analysis 1 (Hygro-Thermal) and Analysis 1 (Structural). These are created automatically as a result of selecting a Coupled Hygro-thermal / Structural user interface option at the beginning of the example.

Loadcases need to be created for each construction stage to be modelled – for both the structural loadings and for the thermal loadings.

Defining Hygro-thermal Loadcases

At least three load cases are needed to cover the duration of the three construction stages, which each last for 10 days. An initial time step of 0.05 days can be used at the start of each construction stage when the poured concrete generates heat of hydration then, as the analysis progresses, larger time steps of 0.1, 0.2 and 0.5 can be used to capture the behaviour for the remaining duration of each stage.

For this example four time steps of a stated duration will be used for each stage, and for these the total number of the expected time steps per stage would be 39. This does not include the initial one at 'Time Step 0', which approximates the initial conditions before the analysis starts. So the total number of time steps that will be solved is 117.

The magnitude of the time step and the total response time for all stages is stated in the following Table. Note all values are specified in <u>days</u>.

Time interval from start of each stage	Initial time step	Total response time for stage 1	Total response time for stage 2	Total response time for stage 3	Number of time steps for each stage
0 to 0.5	**0.05**	**0.5**	**10.5**	**20.5**	10
0.5 to 1.0	**0.1**	**1.0**	**11.0**	**21.0**	5
1.0 to 3.0	**0.2**	**3.0**	**13.0**	**23.0**	10
3.0 to 10.0	**0.5**	**10.0**	**20.0**	**30.0**	14

Time step and response times for Stages 1, 2 and 3

Stage 1 loadcases

- In the ⌖ Treeview, in Analysis 1 (Hygro-Thermal), right-click on **Loadcase 1** and rename it to **Hygro Stage 1a**

- Select **Hygro Stage 1a** using the right-hand mouse button and from the **Controls** menu select the **Nonlinear and Transient** option.

- Enter the Initial time step as **0.05**, Total response time as **0.5.** Leave Max time steps or increments to **0** (default, i.e. ignore it)

- Click **OK** to return to the Modeller window.

🖉   **Note.** An alternative would be to define a Total response time of **0** and set the Max time steps or increments to **10**. A difference between the two would arise if there are convergence difficulties and a step reduction is used. If the number of increments is specified (as done in this example) this number will be carried out regardless of what the actual response time is.

- In the ⌖ Treeview right click on **Hygro Stage 1a** and select **Copy**. Right click on **Hygro Stage 1a** and select **Paste**. Rename the new loadcase to **Hygro Stage 1b.**

- Repeat the paste on top of the previously created loadcase each time and rename the new loadcases to create **Hygro Stage 1c** and **Hygro Stage 1d**

- In the ⌖ Treeview double click on the **Nonlinear and Transient** option for each of these 3 new loadcases and set the **Initial time step** and **Total response time** for each as shown in the previous table.

Stage 2 loadcases

- In the ⌖ Treeview select **Hygro Stage 1a** and add **Hygro Stage 1b, Hygro Stage 1c,** and **Hygro Stage 1d** to the selection. Right click on **Hygro Stage 1d** and select **Copy**.

- Click in a blank area in the treeview, then right click on **Hygro Stage 1d** and select **Paste.**

- Rename the four new loadcases to create **Hygro Stage 2a**, **Hygro Stage 2b**, **Hygro Stage 2c**, and **Hygro Stage 2d**.

- Double click on the **Nonlinear and Transient** option for each of these 3 new loadcases. Leave the 'Initial time step' unchanged but set the **Total response time** for each as shown in the previous table.

Stage 3 loadcases

- In the ⌖ Treeview select **Hygro Stage 2a** and add **Hygro Stage 2b**, **Hygro Stage 2c**, and **Hygro Stage 2d** to the selection. Right click on **Hygro Stage 2d** and select **Copy**.

- Click in a blank area in the treeview, then right click on **Hygro Stage 2d** and select **Paste.**

- Rename the four new loadcases to create **Hygro Stage 3a**, **Hygro Stage 3b**, **Hygro Stage 3c**, and **Hygro Stage 3d**.

- Double click on the **Nonlinear and Transient** option for each of these 3 new loadcases. Leave the 'Initial time step' unchanged but set the **Total response time** for each as shown in the previous table.

Assigning Activation and Deactivation attributes

Construction Stage 1

The elements not required for the first construction stage must be deactivated.

- In the treeview click on **Stage 2**, and add **Stage 3** to the selection, then right-click and choose **Select members**.

- From the Treeview drag and drop the deactivation attribute **Deactivate** onto the selected features. On the assignment dialog choose Analysis: **Analysis 1(Hygro-thermal)** and Loadcase: **Hygro Stage 1a**, and click **OK.**

- Click in the view window to deselect members
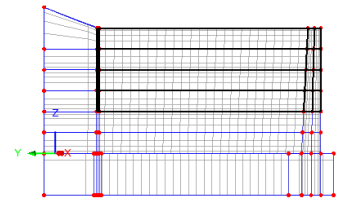
Construction Stage 2

The elements required for **Stage 2** must be activated.

- In the Groups treeview right-click on **Stage 2** and choose **Select members**.

- From the Treeview drag and drop the activation attribute Ac**tivate** onto the selected features. On the assignment dialog choose Analysis: **Analysis 1(Hygro-thermal)** and Loadcase: **Hygro Stage 2a**, and click **OK.**

- Click in the view window to deselect members
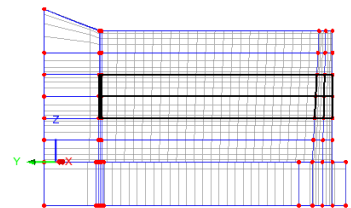
Construction Stage 3

- In the Groups 🔲 treeview right-click on **Stage 3** and choose **Select members**.

- From the 🔴 Treeview drag and drop the activation attribute Ac**tivate** onto the selected features. On the assignment dialog choose Analysis: **Analysis 1(Hygro-thermal)** and Loadcase: **Hygro Stage 3a**, and click **OK.**

- Click in the view window to deselect members
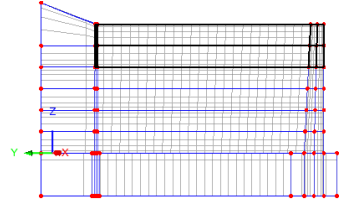
Defining structural loadcases

The structural loadcases can be created by copying and pasting the Hygro-thermal loadcases that are already defined, into the Structural analysis section within the 🔴 Treeview, beneath the existing loadcase entry, and the removing that default entry.

- In the 🔴 Treeview select **Hygro Stage 1a** and then individually add all loadcases from **Hygro Stage 1b** to **Hygro Stage 3d** to the selection. Right click on **Hygro Stage 3d** and select **Copy**.

- Click in a blank area in the treeview, then right click on **Loadcase 2** and select **Paste** to paste the copied loadcases beneath it.

Note. The Deactivation and Activation hygro-thermal assignments will also be copied to the structural loadcases.

The initial Structural Stage 1 loadcase is no longer required:

- Right-click on the **Loadcase 2** loadcase select **Delete**

The twelve newly pasted loadcases now need to be renamed:

- Rename the loadcases in Analysis 1 (Structural) by replacing the word 'Hygro' with the word '**Structural'** (and also removing the word **(Copy)** so that the names will now range from **Structural Stage 1a** to **Structural Stage 3d.**

The time step values (as previously set in the Nonlinear and Transient options for Thermal loadcases) can be left as they are for these structural loadcases, but for each loadcase the Time domain setting must be set to be 'Viscous', as follows:

- In the 🔴 Treeview, double click on the Nonlinear & Transient object 🐝 for loadcase **Structural Stage 1a,** and in the time domain panel select **Viscous**.

- Repeat the above to set the time domain to **Viscous** for all the other structural loadcases.

Structural Supports

Note. The interaction between the dam and the ground is modelled explicitly. Fully fixed boundary conditions are needed to 'fix' the ground.
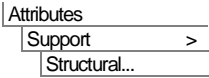
- Set translations **Y** to **Fixed**. Enter name **Symmetry** and click **Apply.**

- Set all the translation in **X, Y** and **Z** to be **Fixed**. Enter name **Fixed** and click **OK.**

**Tip.** In the next steps, hold-down the **Shift + S k**eys and box-select the surfaces required.

Assign symmetry support

- View the model along the X-axis and select the surfaces at the plane of symmetry

- Then, from the 🔵 Treeview, drag and drop the support **Symmetry, Y=0** onto the selected features, and click **OK** to assign.
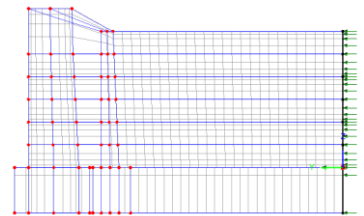
Assign fixed supports

- Rotate or view the model from a suitable angle and, by holding-down the **S** key to select only Surface features, select the surfaces that define extent of the ground beneath the dam and behind the side wall.

- Then, from the 🔵 Treeview, drag and drop the support **Fixed** onto the selected features.
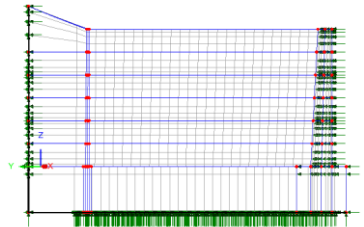
Assigning Hygro-thermal initial conditions to each stage

- In the 🔲 treeview right-click on group **Ground**, and choose **Select members**.

- From the 🔵 Treeview drag and drop **Initial conditions Ground** to loadcase **Hygro Stage 1a**. Click **OK.** Click in the view window to deselect members.

- In the 🔲 treeview right-click on group **Stage 1** and choose **Select members**.
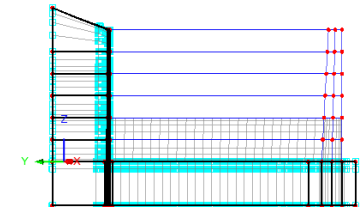
- From the 🎯 Treeview, drag and drop **Initial conditions Concrete** to loadcase **Hygro Stage 1a**. Click **OK.**

- Click in the view window to deselect members.

- In the 🔲 treeview right-click on group **Stage 2** and choose **Select members**.

- From the 🎯 Treeview, drag and drop **Initial conditions Concrete** to loadcase **Hygro Stage 2a**. Click **OK.**

- Click in the view window to deselect members.

- In the 🔲 treeview right-click on group **Stage 3** and choose **Select members**.

- From the 🎯 Treeview, drag and drop **Initial conditions Concrete** to loadcase **Hygro Stage 3a**. Click **OK.**

- Click in the view window to deselect members.
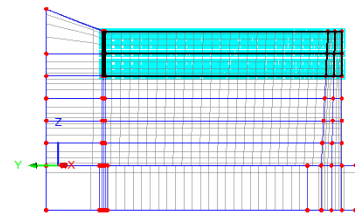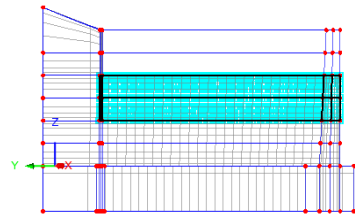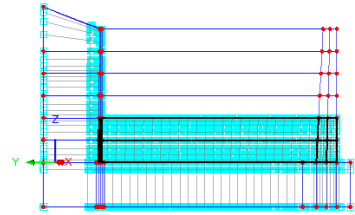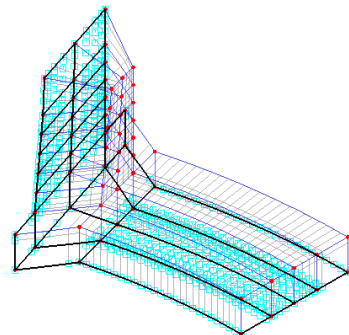
Assigning hygro-thermal loads

To assign hygro-thermal thermal loads regions of the model need to be selected to allow the appropriate loading to be assigned. This can be done either by selecting features that have assigned other attributes, or by using groups.

Ground Temperature

- ⊕ View the model in an isometric view

With the loadcase '**Hygro Stage 3a**' still active:

- In the 🔲 Treeview, right-click on **Ground**, and select **Set as Only Visible**.

- In the 🎯 Treeview, in the Supports folder, right-click on **Fixed** and choose **Select Assignments**. Choose **All Assignments** and click **OK**.

- From the 🎯 Treeview, drag and drop the hygro-thermal load **Prescribed T=20** onto the selected features. On the assignment dialog ensure **Assign to surfaces** is selected for **Analysis 1**

**(Hygro-thermal)**, then select the option **All loadcases** and click **OK**.

- Click in the view window to deselect members.

Note. To see the assigned prescribed loading in isolation it may be necessary to visit the 🔩 Treeview and deselect 'Visualise assignments' for other loadings that were assigned and that are visible for this active loadcase, and select 'Visualise assignments' for this loading. This applies to the following assignments also.

### Environmental load assignments for each construction stage

When a construction stage is activated its side surfaces will be covered by formwork for the duration of the stage, and its top will be exposed to the air. When the next construction stage begins the formwork from the previous stage will be removed and the side surfaces of the stage just cast will be exposed to air. This is modelled as follows:
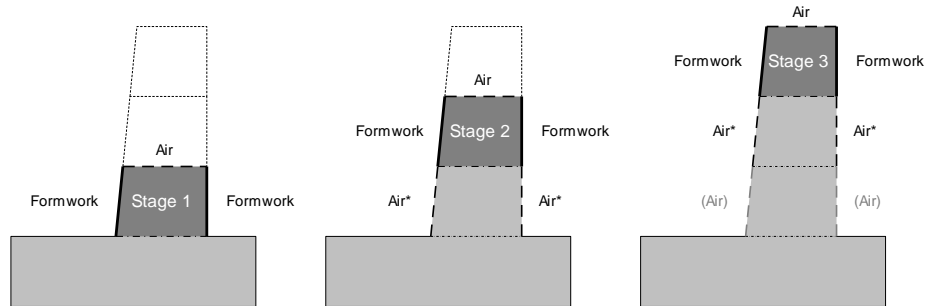
* Environmental air loading conditions for surfaces previously covered by formwork are assigned in the previous stage and are stated to apply until the end of the analysis.

Stage 1 – side walls

- In the 🔲 Treeview right-click on **Stage 1**, and click **Visible.**

- In the 🔲 Treeview, right-click on **Concrete Side Faces Stage 1** and choose **Select Assignments**.

- In the 🕒 Treeview ensure that loadcase H**ygro Stage 1a** is active.

- From the 🔩 Treeview, drag and drop hygro-thermal load **ENVT** - **Formwork** onto the selected features. On the dialog presented, click the **More…** button and specify the loading applies for a range of loadcases from **Hygro Stage 1a** until (and including) a specified loadcase of **Hygro Stage 1d**. Click **OK**.
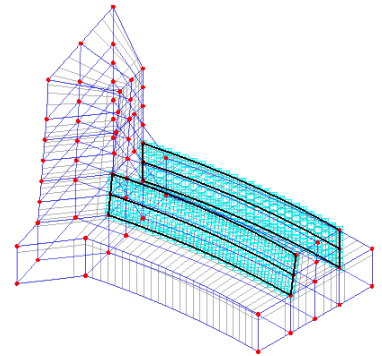
- Then, from the ⚛ Treeview, drag and drop hygro-thermal load **ENVT** - **Air** onto the same features. On the dialog, specify the loading applies for a range of loadcases from **Hygro Stage 2a** to **Until end of analysis**. Click **OK**.

- Click in the view window to deselect members.

Stage 1 - top face

- In the 🔲 Treeview, right-click on **Concrete Top Face Stage 1** and choose **Select Members**.

- From the ⚛ Treeview, drag and drop hygro-thermal load **ENVT** - **Air** onto the selected features. On the dialog presented, specify the loading applies for a range of loadcases from **Hygro Stage 1a** until (and including) a specified loadcase of **Hygro Stage 1d**. Click **OK**.

- Click in the view window to deselect members.
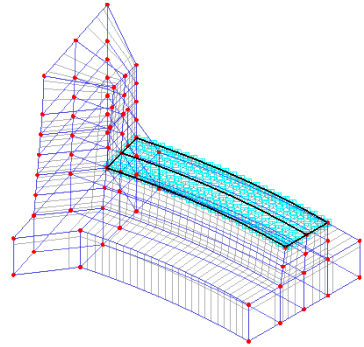
Stage 2 – side walls

- In the 🔲 Treeview right-click on **Stage 2**, and click **Visible.**

- In the 🔲 Treeview, right-click on **Concrete Side Faces Stage 2** and choose **Select Members**.

- In the 🕒 Treeview set active the loadcase **Hygro Stage 2a**

- From the ⚛ Treeview, drag and drop hygro-thermal load **ENVT** - **Formwork** onto the selected features. On the dialog presented, click the **More…** button and specify the loading applies for a range of loadcases from **Hygro Stage 2a** until (and including) a specified loadcase of **Hygro Stage 2d**. Click **OK**.

- Then, from the ⚛ Treeview, drag and drop hygro-thermal load **ENVT** - **Air** onto the same features. On the dialog, specify the loading applies for a range of loadcases from **Hygro Stage 3a** to **Until end of analysis**. Click **OK**.

- Click in the view window to deselect members.

Stage 2 - top face

- In the ⬚ Treeview, right-click on **Concrete Top Face Stage 2** and choose **Select members**.

- From the 🔬 Treeview, drag and drop hygro-thermal load **ENVT** - **Air** onto the selected features. On the dialog presented, specify the loading applies for a range of loadcases from **Hygro Stage 2a** until (and including) a specified loadcase of **Hygro Stage 2d**. Click **OK**.
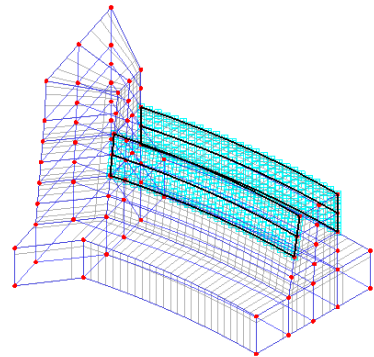
- Click in the view window to deselect members.
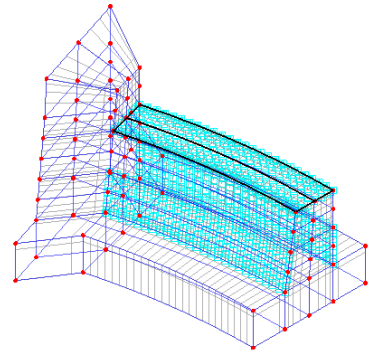
Stage 3 – side walls

- In the ⬚ Treeview right-click on **Stage 3**, and click **Visible.**

- In the ⬚ Treeview, right-click on **Concrete Side Faces Stage 3** and choose **Select Members**.

- In the 🕒 Treeview set active the loadcase **Hygro Stage 3a**

- From the 🔬 Treeview, drag and drop hygro-thermal load **ENVT** - **Formwork** onto the selected features. On the dialog presented, click the **More…** button and specify the loading applies for a range of loadcases from **Hygro Stage 3a** until (and including) a specified loadcase of **Hygro Stage 3d**. Click **OK**.
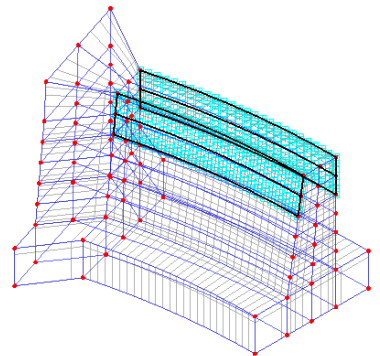
📝 **Note.** No 'ENVT – Air' loading is assigned to the concrete side faces for stage 3 since this example does not consider any effects after the formwork for this stage has been removed.

- Click in the view window to deselect members.

Stage 3 - top face

- In the ⬚ Treeview, right-click on **Concrete Top Face Stage 3** and choose **Select Members**.

- From the ⬚ Treeview, drag and drop hygro-thermal load **ENVT** - **Air** onto the selected features. On the dialog presented, specify the loading applies for a range of loadcases from **Hygro Stage 3a** until (and including) a specified loadcase of **Hygro Stage 3d**. Click **OK**.

- Click in the view window to deselect members.
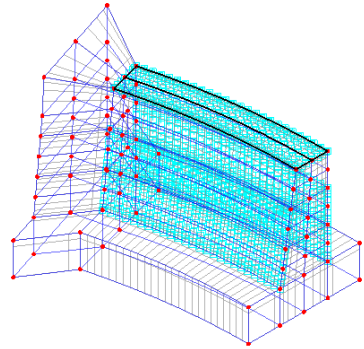
Recap of hygro-thermal load assignments made
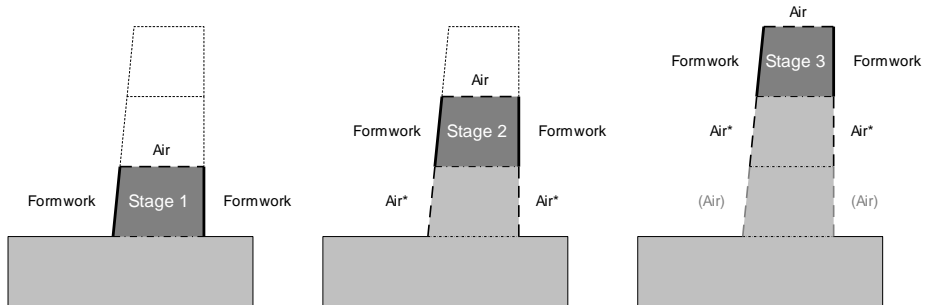
* Environmental air loading conditions for surfaces previously covered by formwork are assigned in the previous stage and are stated to apply until the end of the analysis.

Group to be Set as Only Visible	Select Assignments for Members of this Group	Loading to be assigned to selected group	Loading applies from loadcase	Loading applies until (and including) loadcase
Stage 1	Concrete Side Faces Stage 1	ENVT - Formwork	Hygro Stage 1a	Hygro Stage 1d
Stage 1	Concrete Side Faces Stage 1	ENVT - Air	Hygro Stage 2a	Until end of analysis
Stage 1	Concrete Top Face Stage 1	ENVT - Air	Hygro Stage 1a	Hygro Stage 1d
Stage 2	Concrete Side Faces Stage 2	ENVT - Formwork	Hygro Stage 2a	Hygro Stage 2d

Group to be Set as Only Visible	Select Assignments for Members of this Group	Loading to be assigned to selected group	Loading applies from loadcase	Loading applies until (and including) loadcase
Stage 2	Concrete Side Faces Stage 2	ENVT - Air	Hygro Stage 3a	Until end of analysis
Stage 2	Concrete Top Face Stage 2	ENVT - Air	Hygro Stage 2a	Hygro Stage 2d
Stage 3	Concrete Side Faces Stage 3	ENVT - Formwork	Hygro Stage 3a	Hygro Stage 3d
Stage 3	Concrete Top Face Stage 3	ENVT - Air	Hygro Stage 3a	Until end of analysis

The model is now complete.

| File |
| Save |

- 🖫 Save the model file.

✎ **Note.** The computed temperature and water saturation in the hygro-thermal analysis will introduce thermal and shrinkage strains into the structural analysis. If there were other structural loads they can be defined and applied to the required Structural Loadcase.

Checking the Attributes and Loadcase Treeviews

In defining the staged construction modelling requirements for the dam the Attributes 🔩 Treeview and the 🕒 Treeview now contain all the information for an analysis to be carried out.

For a successful analysis your model should contain treeviews similar to those shown on the following page. Right-clicking on attribute and loadcase data allows any defined values to be checked. If your model has potential errors in your Attribute data, or in your Loadcase assignments, a file is provided to enable you to re-create the model from scratch and run the analysis successfully.

✎ **Note.** The thermal and structural loadcases are always tabulated and solved in 🕒 Treeview order from top to bottom.

Attributes treeview

Attributes
Layers | Attri... | Grou... | Anal... | Repo... | Utilit...

- Concrete_Dam_3D.mdl
  - Attributes (18)
    - Mesh (5)
      - Line (4)
        - 1:X_lines
        - 2:Y_lines
        - 3:Z_lines
        - 5:lines=2_spaced
      - Volume (1)
        - 4:HTM Coupled Bricks
    - Material (4)
      - Isotropic (4)
        - 1:Concrete Hygro-Thermal
        - 2:Ground Hygro-Thermal
        - 3:Concrete Structural
        - 4:Ground Structural
    - Supports (2)
      - 1:Symmetry
      - 2:Fixed
    - Loading (5)
      - Hygro-thermal (5)
        - 1:Initial conditions Ground
        - 2:Initial conditions Concrete
        - 3:ENVT – Air
        - 4:ENVT – Formwork
        - 5:Prescribed T=20
    - Activate elements (1)
      - 1:Activate
    - Deactivate elements (1)
      - 1:Deactivate

Analyses treeview

Analyses
Layers | Attri... | Grou... | Anal... | Repo... | Utilit...

- Concrete_Dam_3D.mdl
  - Hygro analyses
    - Analysis 1 (Hygro-thermal)
      - Material
      - 12:Hygro Stage 1a
        - Loading
          - 1:Initial conditions Ground (x 1.0)
          - 2:Initial conditions Concrete (x 1.0)
          - 3:ENVT – Air (x 1.0) (range)
          - 4:ENVT – Formwork (x 1.0) (range)
          - 5:Prescribed T=20 (x 1.0) (range)
        - Deactivate
          - 1:Deactivate
        - Nonlinear and Transient
      - 13:Hygro Stage 1b
        - Loading
          - 3:ENVT – Air (x 1.0) (range)
          - 4:ENVT – Formwork (x 1.0) (range)
          - 5:Prescribed T=20 (x 1.0) (range)
        - Nonlinear and Transient
      - 14:Hygro Stage 1c
        - Loading
          - 3:ENVT – Air (x 1.0) (range)
          - 4:ENVT – Formwork (x 1.0) (range)
          - 5:Prescribed T=20 (x 1.0) (range)
        - Nonlinear and Transient
      - 15:Hygro Stage 1d
        - Loading
          - 3:ENVT – Air (x 1.0) (range)
          - 4:ENVT – Formwork (x 1.0) (range)
          - 5:Prescribed T=20 (x 1.0) (range)
        - Nonlinear and Transient
      - 16:Hygro Stage 2a
        - Loading
          - 2:Initial conditions Concrete (x 1.0)
          - 3:ENVT – Air (x 1.0) (range)
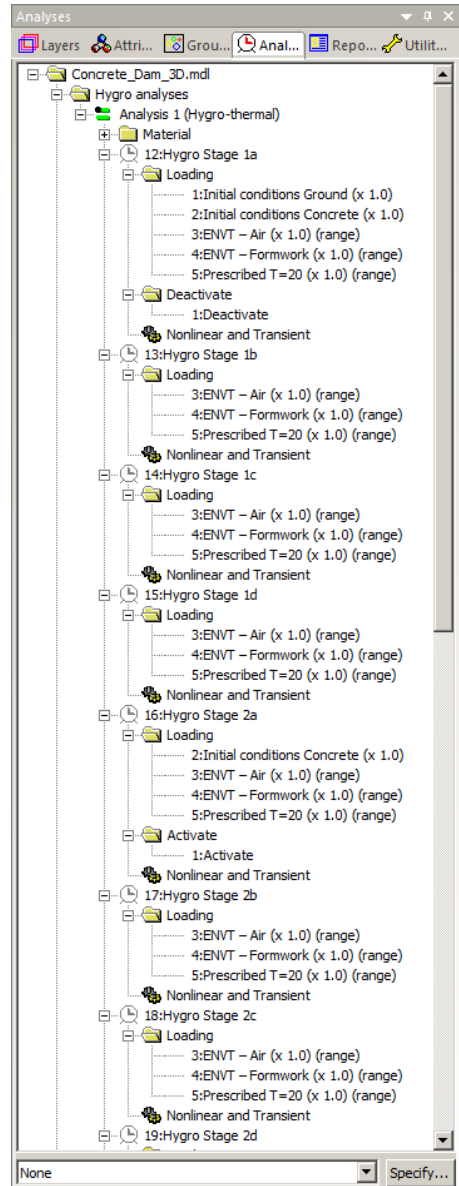   - 4:ENVT – Formwork (x 1.0) (range)
          - 5:Prescribed T=20 (x 1.0) (range)
        - Activate
          - 1:Activate
        - Nonlinear and Transient
      - 17:Hygro Stage 2b
        - Loading
          - 3:ENVT – Air (x 1.0) (range)
          - 4:ENVT – Formwork (x 1.0) (range)
          - 5:Prescribed T=20 (x 1.0) (range)
        - Nonlinear and Transient
      - 18:Hygro Stage 2c
        - Loading
          - 3:ENVT – Air (x 1.0) (range)
          - 4:ENVT – Formwork (x 1.0) (range)
          - 5:Prescribed T=20 (x 1.0) (range)
        - Nonlinear and Transient
      - 19:Hygro Stage 2d

None | Specify...

Running the Analysis

Note. Hygro-thermal / structural coupled analysis is computationally intensive, so the use of the **Fast Solver** option and the **Fast Parallel Direct Solver** is recommended for this example. Solver options can be accessed via the **Model Properties** control object 🦾 in the Analyses 🕐Treeview.

With the model loaded:

 Open the **Solve Now** dialog.

- Ensure that the **Hygro-Thermal** and **Structural** analyses are selected, and press **OK** to run the analysis.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

Solution times

By its very nature a hygro-thermal / structural coupled analysis is computationally expensive. The time required to solve these types of models is directly related to the number of elements present, the number of time steps to be solved, and the Solver that is used. For this example, and for each analysis, 117 time steps need to be solved. The Hygro-thermal analysis is solved first, followed by the Structural analysis. An estimate of the time required to solve the model can be obtained by watching the Solver output window.

If the analysis is successful...

Analysis loadcase results are added to the 🕐 Treeview.

In addition, four files will be created in the Associated Model Data directory where the model file resides:

❑ **Concrete_Dam_3D~Analysis 1 (Hygro-thermal).out**
   this output file contains details of model data, assigned attributes and selected statistics of the thermal analysis.

❑ **Concrete_Dam_3D~Analysis 1 (Structural).out**
   this output file contains details of model data, assigned attributes and selected statistics of the structural analysis.

❑ **Concrete_Dam_3D~Analysis 1 (Hygro-thermal).mys**
   this is the LUSAS results file from the thermal analysis which is loaded automatically into the 🕐 Treeview to allow results to be viewed.

❑ **Concrete_Dam_3D~Analysis 1 (Structural).mys**
   this is the LUSAS results file from the structural analysis which is loaded automatically into the 🕐 Treeview to allow results to be viewed.

If the analysis fails…

If the analysis fails, information relating to the nature of the error encountered is written to a output files in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

If it proves impossible for you to correct the errors reported the following file is provided to enable you to re-create the model from scratch and run an analysis successfully:

❑ **concrete_dam_3D_modelling.vbs** carries out the entire modelling of the example ready for an initial solve to be undertaken.

File	
New…	

 Start a new model file. If an existing model is open, Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **Concrete_Dam_3D**

- Select units of **N,m,kg,s,C**

- Ensure the **Coupled** user interface is selected and click the **OK** button

File	
Script	>
Run Script...	

- To recreate the model, select the file **concrete_dam_3D_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.
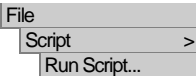
 Run the analysis to generate the results.

Viewing the Results

Analysis results for each time step are present in the Treeview. The time step result for the last thermal loadcase to be solved is set active by default.

Temperature contours throughout the dam are to be investigated for each stage of the construction process. Animations of the change in temperature and of the stress in the dam will be created.
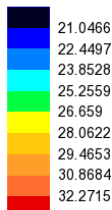
Hygro-Thermal Results

To illustrate the variation of temperature and water saturation throughout the model contour plots will be drawn for the final time step of each construction stage. To view the temperature in the longitudinal middle sections of the dam one half of the model will be made visible.

- In the ⊛ Treeview, right-click on **Half_Dam_Upstream** and choose **Set as Only Visible**.

- In the ▣ Treeview turn-off the display of the **Geometry** layer

- In the ⊙ Treeview, right-click on the last results loadcase for **Hygro Stage 1d**, for which **Time = 10.000** and choose **Set Active**

- With no features selected click the right-hand mouse button in a blank part of the Graphics window and select **Contours....**

- Ensure that Entity **Temperature** and Component **Temp** are selected. Click **OK**.

A plot showing the temperature distribution at the end of **Stage 1d** will be displayed.

Analysis: Analysis 1 (Hygro-thermal)
Loadcase: 15:Hygro Stage 1d, 40:Time Step 39 Time = 10.0000
Results file: Concrete_dam__3D_~Analysis 1 (Hygro-thermal).mys
Response time: 10.0
Entity: Temperature
Component: Temp (Units: C)

21.0466
22.4497
23.8528
25.2559
26.659
28.0622
29.4653
30.8684
32.2715

Maximum 32.628 at node 1637
Minimum 20.0 at node 7
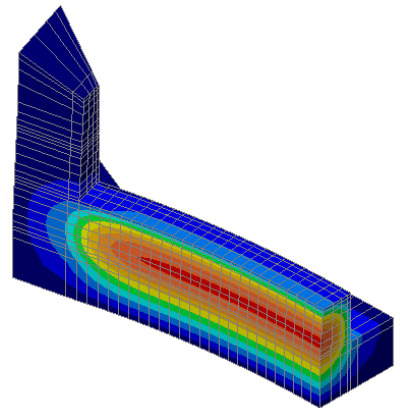
- In the ▣ Treeview double-click on the **Contours** layer. Select 'Entity' **Hygro-thermal results** and 'Component' **Hr** (relative humidity). Click **OK**.

A plot showing the water saturation distribution at the end of Stage 1will be displayed.

Analysis: Analysis 1 (Hygro-thermal)
Loadcase: 15:Hygro Stage 1d, 40:Time Step 39 Time = 10.0000
Results fle: Concrete_dam__3D_~Analysis 1 (Hygro-thermal).mys
Response time: 10.0
Entity: Hygro-thermal results
Component: Hr

0.741782
0.760327
0.778871
0.797416
0.81596
0.834505
0.853049
0.871594
0.890138
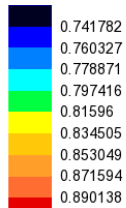
Maximum 0.905682 at node 3340
Minimum 0.738781 at node 2861
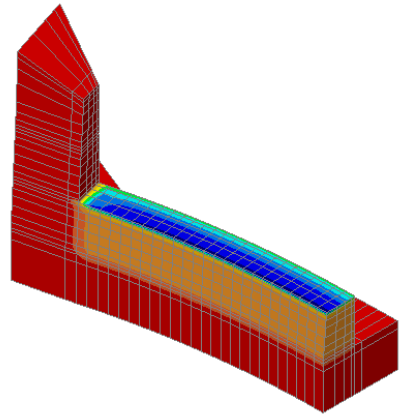
In a similar way, the results for any time step can be displayed. Two examples follow:

Analysis: Analysis 1 (Hygro-thermal)
Loadcase: 23:Hygro Stage 3d, 118:Time Step 117 Time = 30.0000
Results fle: Concrete_dam__3D_~Analysis 1 (Hygro-thermal).mys
Response time: 30.0
Entity: Temperature
Component: Temp (Units: C)

20.7545
21.9753
23.1962
24.417
25.6379
26.8587
28.0796
29.3004
30.5213

Maximum 30.9877 at node 5399
Minimum 20.0 at node 7
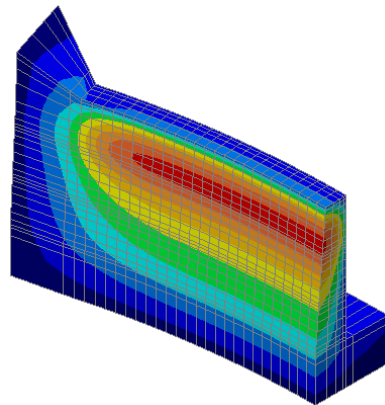
Temperature distribution at the end of Stage 3d.

Analysis: Analysis 1 (Hygro-thermal)
Loadcase: 23:Hygro Stage 3d, 118: Time Step 117 Time = 30.0000
Results file: Concrete_dam__3D_~Analysis 1 (Hygro-thermal).mys
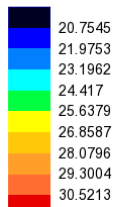Response time: 30.0
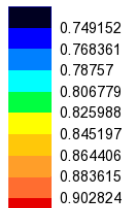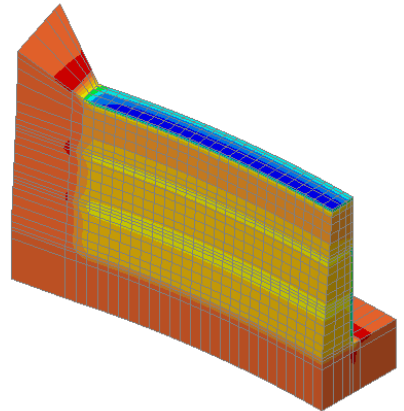Entity: Hygro-thermal results
Component: Hr

0.749152
0.768361
0.78757
0.806779
0.825988
0.845197
0.864406
0.883615
0.902824

Maximum 0.908619 at node 2847
Minimum 0.735737 at node 4629

Hygro-thermal results of water saturation distribution at the end of Stage 3d.

Animating Thermal Results

By setting a Time Step active for the end of the construction stage under consideration (as done for the previous contour plots) an estimate of the maximum and minimum values required for the contour key can normally be obtained. However, when animating thermal results sometimes the maximum value obtained may occur for a time-step other than the last time-step in an analysis, as in this case. By animating a range of timesteps the maximum value of temperature reached in any part of the dam can be seen.

- In the Treeview double-click on the **Contours** layer.

- Ensure that Entity **Temperature** and Component **Temp** are selected.

- Select the **Contour Range** tab and ensure that the contour **Interval** contour is set to **2.0**. Set the **Maximum** and **Minimum** values to **45** and **20** respectively. Click **OK**

Note. The viewing parameters (e.g. the view angle and contour options) used in the animation are those that are currently specified in the window when the animation is loaded. It is therefore important to have an appropriate view visible in the Modeller window when running the animation wizard.

To create and save an animation of the temperature variation on selected surfaces of the dam throughout the whole analysis, complete the following procedure:

Tools
  Animation Wizard…

- Select the **Load history** radio button and click **Next**

Note. In some analyses very small time increments are required for solution convergence but are not required for animation purposes. It is possible to reduce the size of (and the time required to produce) an animation by using the 'Step' option.

- In the 'Available' panel click on the **Analysis 1 (Hygro-thermal)** entry, then, in the Step field, enter **2**.
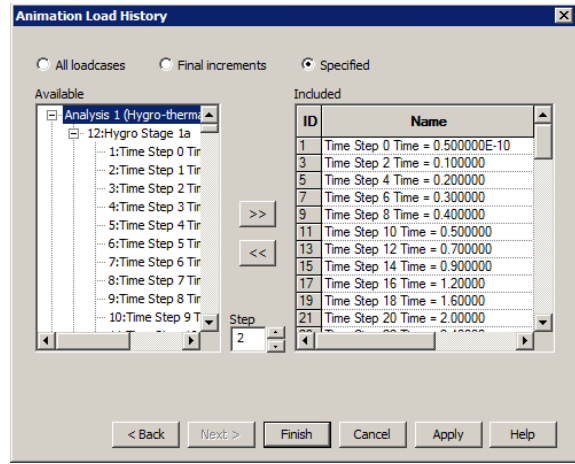
- Press the 'Add to' button, and click **Finish**

The first and every second Time Step of each construction stage will be added to the 'Included' panel of the dialog.

Once LUSAS has processed the selected time step results a new window will show an animation of the variation of temperature with time.
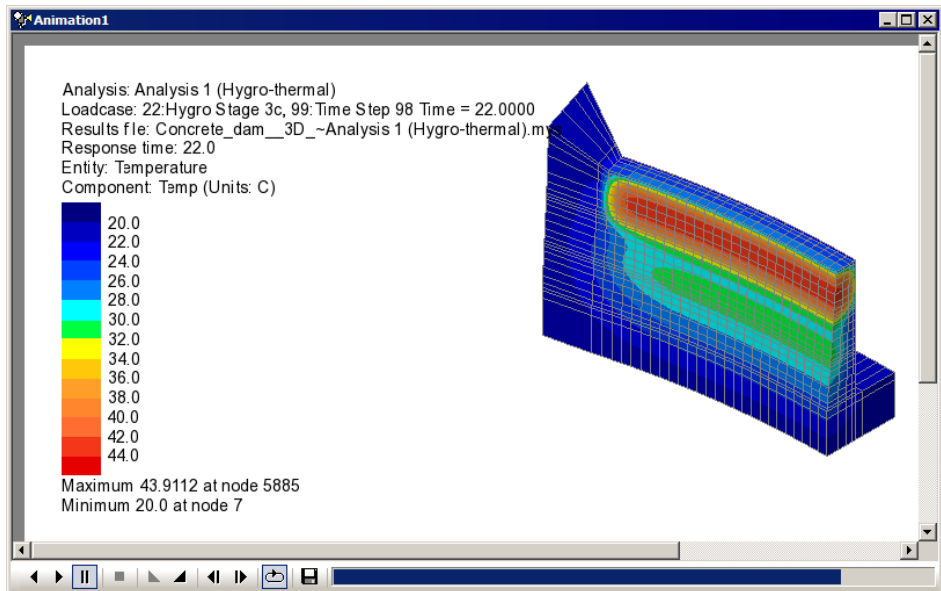
Animations may be saved in compressed AVI format for playback in other applications.

- Save the animation and enter **Concrete_dam_thermal.avi** for the filename and press **Save**

- Close the Animation window.

Graphing of Results at Selected Nodes

In analyses of this type it may also be useful to graph the changing temperature at a selected node (or nodes) as the construction progresses. This can be done by using the Graph Wizard.

- In the 🕒 Treeview, ensure that the last results loadcase for **Hygro Stage 3d** for which **Time = 30.000** is **Set Active**

- Turn-off the display of the contour layer.

- Click in an empty region of the view window and holding down the **Shift + N** keys, and on the surfaces on the axis of symmetry of the dam, select the node in the middle of each of the three construction stages.
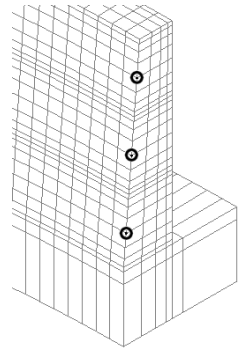
Graphing temperature for selected locations and stages
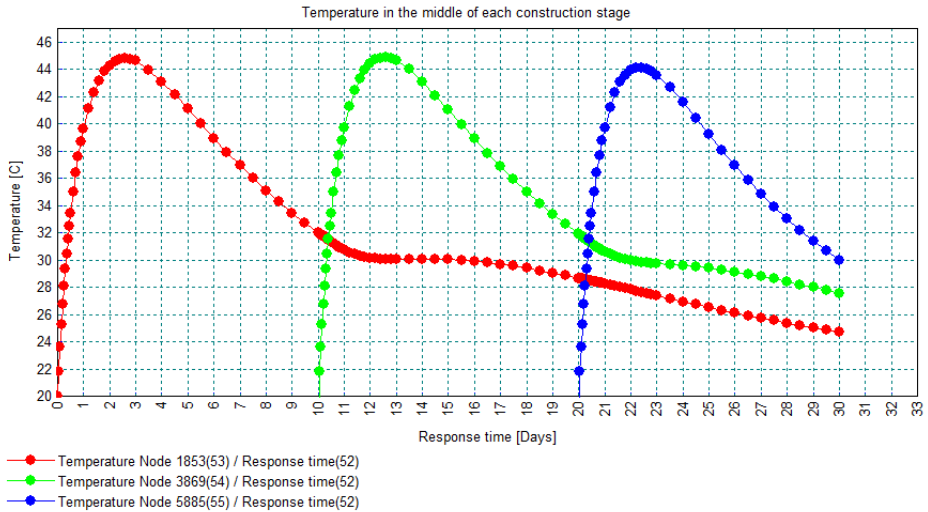
- Select **Time history** and click **Next**.

To define the graph X axis details:

- Select **Named** entity data.

- In the Sample loadcases panel ensure **Whole analysis and Analysis 1(Hygro-thermal)** is selected. Click **Next** to continue.

- Select **Response time** for the data to be plotted and click **Next.**

To define the graph Y axis details:

- Select **Nodal** entity data and click **Next**.

- Ensure Entity **Temperature,** Component **Temp,** and Extent **Selected nodes only** are selected. In the Selected nodes panel select **Multiple**. Click **Next**.

- Enter the title as **Temperature in the middle of each construction stage**

- Enter the X axis title as **Response time [Days]**

- Enter the Y axis as **Temperature [C]**

- For the Y scale choose **Manual** and enter min = **20** and max = **47** since the extent of the temperature variation is roughly known. Click **Finish**.

A graph showing the temperature time history at the selected nodes will be displayed.

Temperature in the middle of each construction stage

● Temperature Node 1853(53) / Response time(52)
● Temperature Node 3869(54) / Response time(52)
● Temperature Node 5885(55) / Response time(52)

Graphing relative humidity for selected locations

A graph of response time against relative humidity can be similarly obtained.

- Click in an empty region of the view window to clear the previous selections

- Select the node in the middle of Stage 1 (Node 1853) and holding down the **Shift + N** key, select the node in the middle of the top surface of Stage 1 (Node 2861).

<table>
<tr><td>Utilities</td></tr>
<tr><td>Graph wizard…</td></tr>
</table>

- For **Time history**, **Named** results, for the **Whole analysis**, define the X axis to be **Response time**

- For **Nodal** results define the Y axis to be **Hygro-thermal results** of Component **Hr.**

- Set the Extent to be **Selected nodes only**. In the Selected nodes panel select **Multiple**. Click **Next**.

- Enter the title as **Relative humidity, stage 1, middle and top.**

- Enter the X axis title as **Response time [Days].**

- Enter the Y axis as **Relative humidity.**

- For the Y scale choose **Automatic** and click **Finish** to create the graph.

Relative humidity, stage 1, middle and top

— Hygro thermal results Hr Node 1853(57) / Response time(56)
— Hygro thermal results Hr Node 2861(58) / Response time(56)
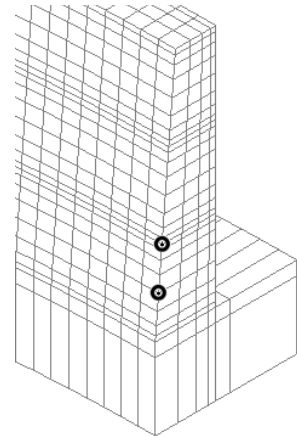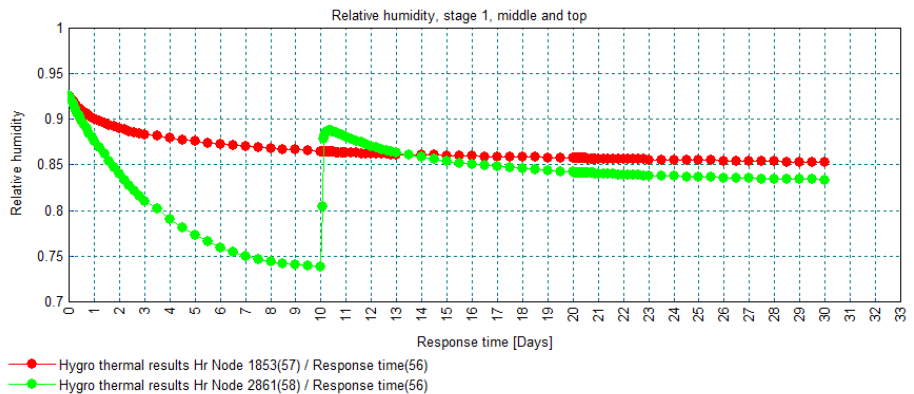
Drying out at the uncovered top surface of stage 1 is evident.

Note. The direction of the water vapour transport (i.e. from the environment to the concrete or vice versa) depends on the vapour content in the environmental air and the concrete pore air. The vapour content depends on the relative humidity and the temperature. Since concrete temperature is higher (due to the heat of hydration) water vapour transport from the concrete to the environment (i.e. drying out) can occur when concrete relative humidity is lower than the environmental one.

Structural Results

Viewing and animating structural results is done in a similar manner to that described for thermal results, ensuring that only structural loadcase results time steps are set active or used in an animation.

In addition to the usual structural results the coupled hygro-thermal-structural analysis offers results for the evolution of the material properties during the curing process and the thermal, shrinkage and creep strains. It can also calculate the width of the cracks that could develop due to high thermal and shrinkage strains during curing or any service loads afterwards. The additional results can be found under the entity name **Concrete Results.**

LUSAS assumes that at a given node a maximum of three mutually perpendicular crack planes can occur. The maximum crack width from these three crack planes is obtained by plotting results component CWmax.

- Right click in a blank part of the view window and select **All Visible**

- In the 🕑 Treeview right-click on the last results loadcase for **Structural Stage 3d**, for which **Time = 30.000** and choose **Set Active**

- Double-click the contours layer in the 🔲 Treeview. On the dialog, set the results entity to **Concrete Results** and component **CWmax**
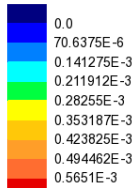
- Select the **Contour Range** tab and set the Number of contours required to be **9** and deselect all other options. Click **OK.**
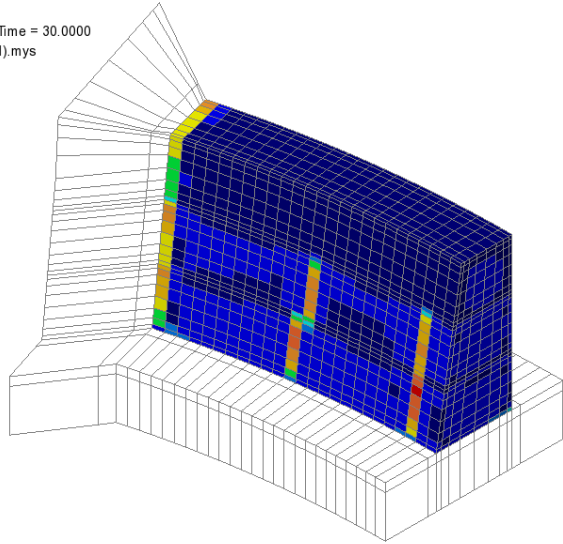
The location and width of the predicted cracks will be displayed.

A region of cracking adjacent to the abutment is noted, as well as in the face. The maximum crack width somewhere in the dam is reported to be in the order of **0.63** mm.

- If the Geometry layer is turned off, by hovering the cursor over elements of interest the largest reported crack width in the downstream face can be seen to be in the order of **0.57** mm.

- In the ⬡ Treeview, right-click on **Half_Dam_Upstream** and choose **Set as Only Visible**.

Analysis: Analysis 1 (Structural)
Loadcase: 25:Structural Stage 3d, 118:Time Step 117 Time = 30.0000
Results file: Concrete_Dam_3D~Analysis 1 (Structural).mys
Response time: 30.0
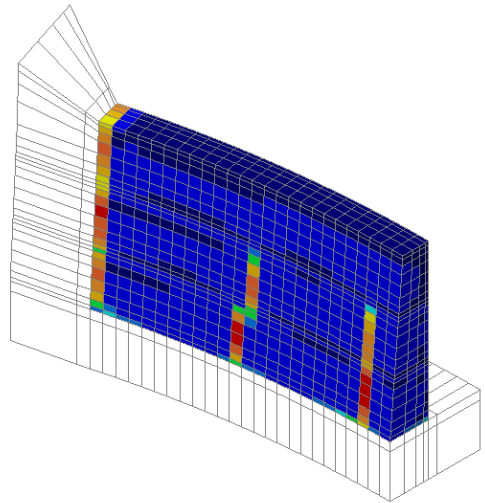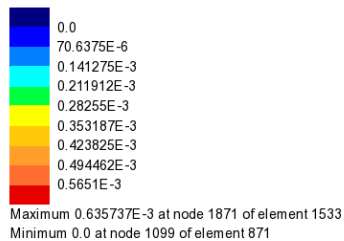Entity: Concrete Results - Solids
Component: CWMax (Units: m)

0.0
70.6375E-6
0.141275E-3
0.211912E-3
0.28255E-3
0.353187E-3
0.423825E-3
0.494462E-3
0.5651E-3

Maximum 0.635737E-3 at node 1871 of element 1533
Minimum 0.0 at node 1099 of element 871

- With this 'sectional' view of the model, the internal location of the maximum crack width that was calculated can be seen.

Note. Numerically cracks form when the stress strain response is in the strain softening phase and the strain normal to the crack plane localizes in a band of single elements. For a more accurate localization and accurate crack width prediction a denser mesh than the one used for a usual structural analysis is required.

Note. When localization of a crack plane and crack growth occur a sufficient number of time steps should be used. Ideally a time step sensitivity study should be done. A converged solution with a small number of solution steps may not predict the crack widths accurately.

Graphing of Concrete Results at Selected Elements

Cracking has occurred in the dam due to high tensile stresses. These stresses are caused by the combined effect of high thermal and shrinkage strains in a restrained structure. Early creep, that reduces these stresses, is also taken into account in the analysis.

The thermal and/or shrinkage strains need to be reduced in order to reduce and even eliminate cracking. Which of these strains is the main cause of cracking can be ascertained by plotting their time history graph for the element along with the maximum crack width, CWmax.

With the model viewed in an isometric view:

- Hold down the **Shift + E** keys to allow selection of only elements and select the element in the external face with the highest value of CWmax – as shown in the adjacent image.
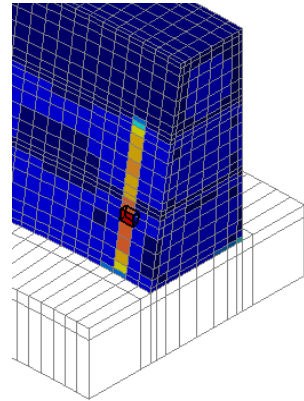
Graphing shrinkage for a selected element

- Select **Time history** and click **Next**.

To define the graph X axis details:

- Select **Named** entity data in the Entity data panel.

- In the Sample loadcases panel ensure **Whole analysis and Analysis 1(Structural)** is selected. Click **Next** to continue.

- Select **Response time** for the data to be plotted and click **Next.**

To define the graph Y axis details:

- Select **Gauss Point** entity data and click **Next**.

The number of the selected element is shown along with one of eight Gauss points that are available for the element.
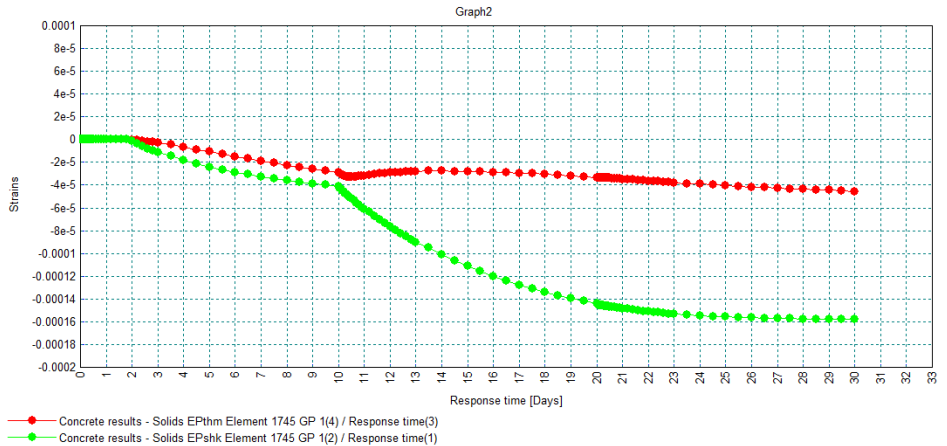
- Ensure Gauss point **1** is selected, select entity **Concrete Results** Component **EPshk** (shrinkage strain), click **Next**.

- Enter the title as **Shrinkage and Thermal Strains**

- Enter the X axis title as **Response time [Days]**, the Y axis as **Strains** and click **Finish**.

A graph plot showing the time history of the **EPshk** is plotted. Note the number of the graph in the Utilities treeview.

Graphing thermal strain for a selected element

- Repeat the previous sequence to create a new graph of **Response time** against thermal strain **EPthm**, making sure to include the graph number of the graph that plotted **EPshk**, then click **Finish**

Graph2

Concrete results - Solids EPthm Element 1745 GP 1(4) / Response time(3)
Concrete results - Solids EPshk Element 1745 GP 1(2) / Response time(1)

🖊 **Note.** It would seem that increasing EPthm to counteract EPshk could be a valid strategy to reduce cracking, but it is not. EPthm is not the real thermal strain loading in the concrete because the initially liquid concrete solidifies in a stress and strain free state after a certain degree of hydration has been reached; the temperature at this moment in time is higher than the initial temperature. This is taken into account during the calculations but it is not included in the thermal strain.

Step 2: Crack width reduction by improving curing conditions

Reduction of the shrinkage strain can be achieved by keeping the air exposed concrete surfaces wet. In LUSAS this is achieved by applying a prescribed relative humidity loading to those surfaces to represent the contact with water. This also requires a free 'support' to be assigned to remove the loading after it is no longer required.
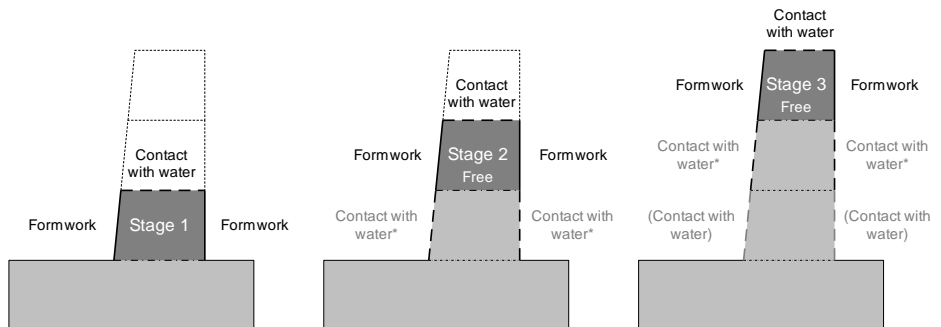
* Prescribed 'Contact with water' conditions for surfaces previously covered by formwork, and the subsequent setting of Free 'supports' are both assigned in the previous stage and stated to apply until the end of the analysis.

Continuing with the current model:

Prescribed relative humidity

- Select **Prescribed.**

- Leave 'Temperature' **Free.**

- Set 'Concrete relative humidity' to **Contact with water (1.0)**

- Enter a name of **Prescribed - Contact with water** and click **Finish**.

Note. Defining the range of application of the Prescribed Load, say from Stage 1a to Stage 1d, is not enough to stop the load after Stage 1d. It will set the prescribed value to zero (i.e. the load will be converted to a 'Support'). To stop the load a 'Support' condition **Free** should be assigned at the next stage (Stage 2a).

- Ensure all 'supports' are set to **Free**

- Enter the name as **Free - Prescribed hygro** and click **OK**

Assigning prescribed relative humidity

- In the 🟤 Treeview, double click to edit the loading attribute **ENVT** - **Air** and set the vapour mass transfer coefficient to **0**. Click '**OK**'.

Note. This simply specifies that no convection mass transfer is allowed to take place. However, even if the coefficient was not set to zero the prescribed relative humidity that would be assigned next would supersede and remove the stated convective mass transfer.

- In the 🔲 Treeview, ensure that the **Geometry** layer is present and is turned **On**.

For stage 1 side faces

- In the 🔷 Treeview right click on **Concrete Side Faces Stage 1** and choose **Select members** (these are the surfaces that were assigned the ENVT - Formwork loading earlier).

- From the 🟤 Treeview, drag and drop the loading **Prescribed - Contact with water** onto the selected features. Select the **More >>** button and set the **Range of loadcases** from **Hygro Stage 2a** to **Until end of analysis**. Click O**K**.

For stage 1 top face

- In the 🔷 Treeview right click on **Concrete Top Face Stage 1** and choose **Select members** (these are the surfaces that were assigned the ENVT - Air loading earlier). If asked, choose **Yes** to deselect previously selected members.

- From the 🟤 Treeview, drag and drop the loading **Prescribed - Contact with water** onto the selected features. Select the **More >>** button and set the **Range of loadcases** from **Hygro Stage 1a** to **Hygro Stage 1d**.

- From the 🔗 Treeview, drag and drop the support **Free - Prescribed hygro** onto the selected features. Select the **From loadcase** option and assign to **Analysis 1 (Hygro-thermal)** and **Hygro Stage 2a**. Click **OK**.

For stages 2 and 3

- Repeat the above sequence, selecting the appropriate groups to isolate the concrete side faces and top surfaces for stages 2 and 3, making the appropriate assignments as shown in this table:

Select group	Assign attribute	To:
Concrete Side Faces Stage 2	Prescribed - Contact with water	Hygro Stage 3a to Until end of analysis
Concrete Top Face Stage 2	Prescribed - Contact with water	Hygro Stage 2a to Hygro Stage 2d
Concrete Top Face Stage 2	Free - Prescribed hygro	Analysis 1 (Hygro-thermal) and Hygro Stage 3a
Concrete Side Faces Stage 3	(None to be assigned)	See Note.
Concrete Top Face Stage 3	Prescribed - Contact with water	Hygro Stage 3a to Hygro Stage 3d

Note. No 'Prescribed - Contact with water' loading is assigned to the concrete side faces for stage 3 because this example does not consider any effects after the formwork for this stage has been removed, meaning that this analysis finishes with 'Prescribed - Contact with water' still applied to the side faces for stage 3.

Note. No 'Free – Prescribed hygro' load is to be assigned to the concrete top face for stage 3 because the example does not consider any additional stages, meaning that this analysis finishes with 'Prescribed - Contact with water' still applied to the top face.

Save the revised model

File
Save As…

- Save the file into a folder named **\<LUSAS Installation Folder>\Projects\dam**

- Save the model into this new folder as **Concrete_Dam_3D_Improved_Curing**

Running the Analysis
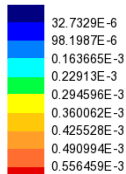
Run the analysis to generate the results.

- In the 🔲 Treeview, ensure that the **Geometry** layer is now turned **Off**.
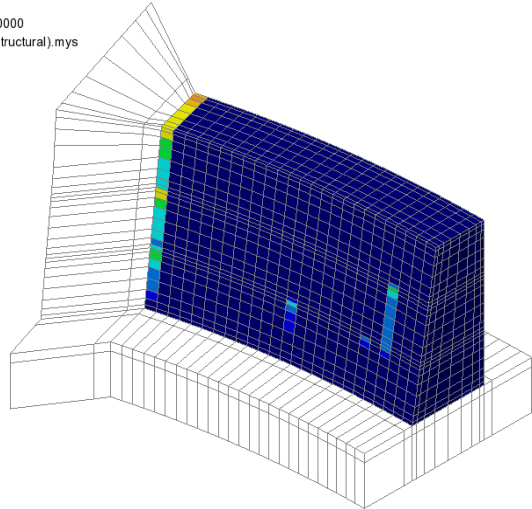
- In the 🕒 Treeview, ensure the last results loadcase for **Structural Stage 3d**, for which **Time=30.000** is **Set Active.**

The location and width of the predicted cracks will be displayed on the Contours layer.

Analysis: Analysis 1 (Structural)
Loadcase: 25:Structural Stage 3d, 118: Time Step 117 Time = 30.0000
Results file: Concrete_Dam_3D_Improved_Curing~Analysis 1 (Structural).mys
Response time: 30.0
Entity: Concrete Results - Solids
Component: CWMax (Units: m)

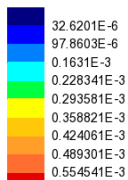|	
	32.7329E-6
	98.1987E-6
	0.163665E-3
	0.22913E-3
	0.294596E-3
	0.360062E-3
	0.425528E-3
	0.490994E-3
	0.556459E-3

Maximum 0.589192E-3 at node 3643 of element 3052
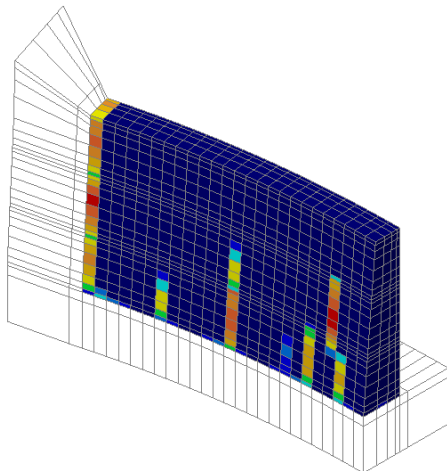Minimum 0.0 at node 2354 of element 1946

A region of cracking adjacent to the abutment is still present, but the cracking in the face is now reduced. The maximum crack width is reported to be **0.59** mm but this is now internal.

- In the 🔷 Treeview, right-click on **Half_Dam_Upstream** and choose **Set as Only Visible**.

Analysis: Analysis 1 (Structural)
Loadcase: 25:Structural Stage 3d, 118: Time Step 117 Time = 30.0000
Results file: Concrete_Dam_3D_Improved_Curing~Analysis 1 (Structural).mys
Response time: 30.0
Entity: Concrete Results - Solids
Component: CWMax (Units: m)

	32.6201E-6
	97.8603E-6
	0.1631E-3
	0.228341E-3
	0.293581E-3
	0.358821E-3
	0.424061E-3
	0.489301E-3
	0.554541E-3

Maximum 0.587162E-3 at node 3644 of element 3053
Minimum 0.0 at node 2359 of element 1951

With this 'sectional' view of the model, the internal location of the maximum crack width calculated can be seen.

Step 3: Crack width reduction by pre-cooling the concrete

A further reduction in crack width may possibly be achieved by reducing the thermal strain that is present. This can be obtained by pre-cooling the concrete.

Continuing with the current model:

- In the 🎯 Treeview right click on the loading **Initial Conditions Concrete** and change the temperature to **10** °C

Save the revised model

File
   Save As…

- Save the file into a folder named **\<LUSAS Installation Folder>\Projects\dam**

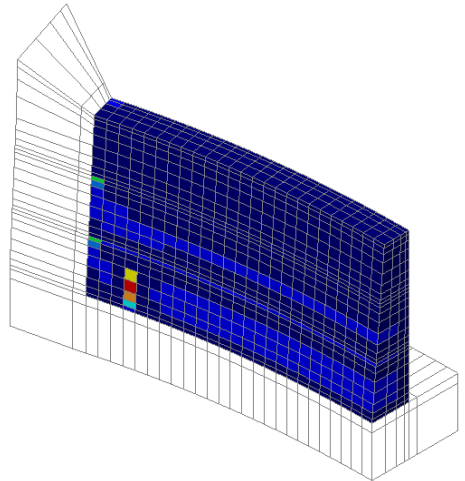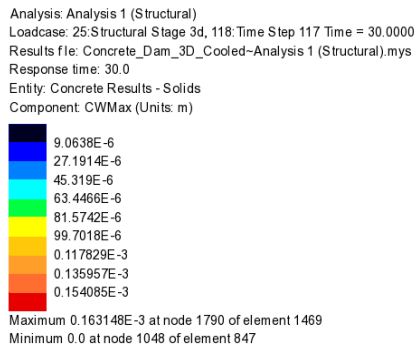- Save the model into this new folder as **Concrete_Dam_3D_Cooled**

Running the Analysis

Run the analysis to generate the results.

- In the 🕐 Treeview, ensure the last results loadcase for **Structural Stage 3d**, for which **Time=30.000** is **Set Active.**

The location and width of the predicted cracks will be displayed.

Analysis: Analysis 1 (Structural)
Loadcase: 25:Structural Stage 3d, 118: Time Step 117 Time = 30.0000
Results file: Concrete_Dam_3D_Cooled~Analysis 1 (Structural).mys
Response time: 30.0
Entity: Concrete Results - Solids
Component: CWMax (Units: m)

9.0638E-6
27.1914E-6
45.319E-6
63.4466E-6
81.5742E-6
99.7018E-6
0.117829E-3
0.135957E-3
0.154085E-3

Maximum 0.163148E-3 at node 1790 of element 1469
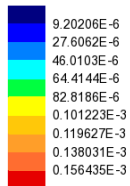Minimum 0.0 at node 1048 of element 847

From this sectioned view of the model, the cracking adjacent to the abutment is reduced and maximum crack width can be seen to have been reduced to **0.16** mm.

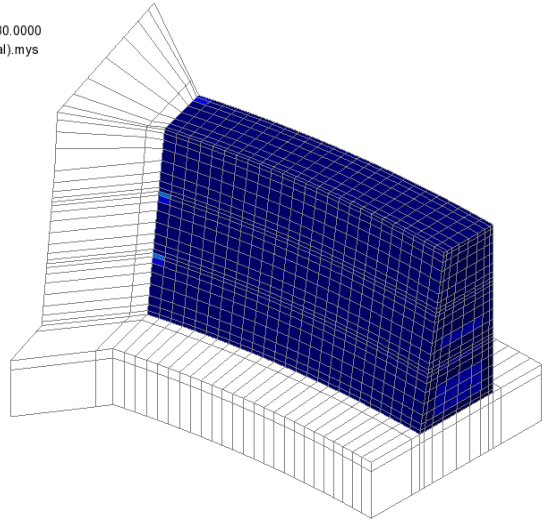Lastly, to check the crack widths in the surface of the dam:

- In the view window right-click in a blank region and select **All Visible**.

Analysis: Analysis 1 (Structural)
Loadcase: 25:Structural Stage 3d, 118:Time Step 117 Time = 30.0000
Results file: Concrete_Dam_3D_Cooled~Analysis 1 (Structural).mys
Response time: 30.0
Entity: Concrete Results - Solids
Component: CWMax (Units: m)

9.20206E-6
27.6062E-6
46.0103E-6
64.4144E-6
82.8186E-6
0.101223E-3
0.119627E-3
0.138031E-3
0.156435E-3

Maximum 0.165637E-3 at node 1788 of element 1468
Minimum 0.0 at node 1043 of element 843

No significant exterior cracking is present.

This completes the example.