Staged Construction of a Concrete Dam

For LUSAS version:	19.0
For software product(s):	Any Plus version
With product option(s):	Nonlinear, Dynamic, Thermal, Heat of Hydration

Description

A 30m high concrete dam, tapering from 10m thick at the base to 3.1735m thick at the crest is to be constructed in three stages, each 10 metres high. Each stage is added 10 days after the previous stage, and the analysis runs for a total of 30 days.

Analysis: Thermal
Loadcase: 61:Thermal Stage 3, Time Step 60 Time = 23.5391
Results file: dam_from_vbs~Thermal.mys
Response time: 23.5391
Entity: Potential
Component: PHI (Units: C)

20.0
25.0
30.0
35.0
40.0
45.0
50.0
55.0
60.0

Maximum 68.4665 at node 734
Minimum 22.0616 at node 330
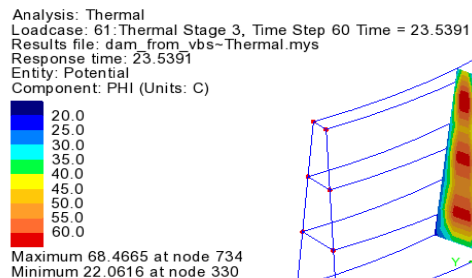
Animations are created showing the variation of temperature and stress during its construction.

Simplified geometry is used to allow the example to concentrate on the definition of the concrete heat of hydration loading and staged construction techniques required. Ground is not modelled explicitly in this example, but should be in a real-life scenario.

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

Objectives

The required output from the analysis consists of:

❑ A time history of the temperature throughout the dam during construction.

❑ A time history of the distribution of maximum principal stress (S1) throughout the dam.

Keywords

3D, Coupled, Thermal, Staged Construction, Load curve, Activate, Deactivate, Heat of hydration, Semi-coupled analysis, Concrete.

Associated Files

❑ **concrete_dam_geometric_modelling.vbs** carries out the geometric modelling of the example.

❑ **concrete_dam_modelling.vbs** carries out the complete modelling of the example ready for a solve to be carried out.

Discussion

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 Concrete Tower example if age and creep is of interest to you.

The process of hydration is an exothermic chemical reaction, which leads to large amounts of heat generation. This leads to thermal stresses, which may potentially cause the concrete to crack. A semi-coupled analysis can be used to investigate the connection between the heat generated due to concrete hydration and the thermal stresses induced as a result.

Modelling

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.

Creating a new model

- Enter a File name of **Concrete_Dam**

- Use the default User-defined working folder.

- Ensure an Analysis type of **Coupled Thermal/Structural** is set.

- Select an Analysis Category of **3D**

- Set Model units of **N,m,kg,s,C**

- Leave the Timescale units as **Days**

- Select a Startup template of **None**

- Ensure the Layout grid is set as **None**

- Enter a Title of **Concrete dam example to model heat of hydration**

- Click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Defining the Geometry

The base of the dam is 10m thick, 54.97m along its centreline, and subtends an angle of 30 degrees. The thickness of the dam tapers with increasing height in a nonlinear manner (such that the thickness at the abutments is very slightly larger than at the centre of the dam).

Since the geometry of the dam is not the focus of this example a script file has been provided which will automatically create the model.

File
  Script   >
    Run Script…
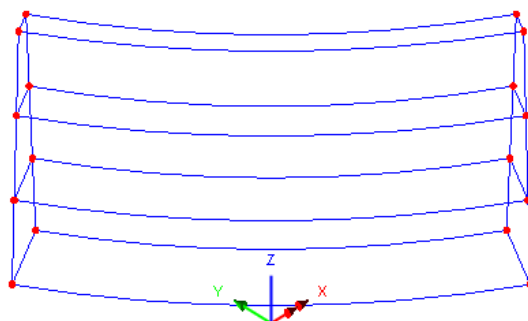
- Select the file **concrete_dam_geometric_modelling.vbs** which is located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory and click **OK**
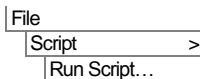
⊕ Select the isometric button which, because of the way the geometry has been defined, will give a front view of the dam. The three volumes represent the three stages of construction.

Defining Groups

It is useful in 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 viewing of results.

- Select the lowest volume representing the concrete in construction stage 1.

Geometry	
Group	>
New Group	

🔲 Enter the group name as **Stage 1** and click **OK** to finish defining the group.

- With the previous volume still selected, hold the Shift key down and add the middle volume representing construction stage 2 to the selection.

Geometry	
Group	>
New Group	

🔲 Enter the group name as **Stage 2** and click **OK** to finish defining the group.

- With the previous two volumes still selected, hold the Shift key down and add the upper volume representing construction stage 3 to the selection.

Geometry	
Group	>
New Group	

🔲 Enter the group name as **Stage 3** and click **OK** to finish defining the group.

Defining and Assigning Mesh Attributes

To ensure a well-proportioned mesh, line mesh attributes will be assigned to define a mesh with three elements through the thickness of the dam, and eight elements across the width of the dam. Vertically, lines defining each volume will be split into two divisions. To create this mesh arrangement both default and individual line mesh divisions will be used.

File
Model properties…

- On the Model Properties dialog select the **Meshing** tab. Change the default line divisions to **3** and click **OK.**

This specifies the default number of mesh divisions to be used along a line unless a line mesh attribute is subsequently assigned.

Attributes	
Mesh	>
Line…	

- On the Line Mesh dialog ensure that the Structural element type is set to **None**. Set the number of divisions to **2**, name the attribute **Divisions = 2** and click **Apply** to create the attribute.

- Change the number of divisions to **8**, change the name to **Divisions = 8** and click **OK**.

🖉   **Tip.** To assign these mesh divisions to the model the selection of lines could be done individually, but lines can also be selected by holding-down the **L** key while box-selecting with the mouse. When this is done any part of a line that is within the area dragged is selected.

- Select the twelve near-vertical lines that define the four corners of the dam at the abutments. Assign the line mesh attribute **Divisions = 2** from the 🔴 Treeview onto the selected features. Take care to not select the horizontal lines at these abutments.

- Select the eight horizontal curved lines from the model. Assign the line mesh attribute **Divisions = 8** from the 🔴 Treeview onto the selected features.

Attributes
Mesh   >
Volume…
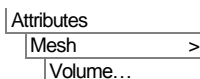
- The dam is to be modelled with **Stress, Hexahedral, Linear** elements (HX8M elements) in the structural analysis. By clicking on the Thermal tab it can be seen that these elements are automatically coupled to **Field, Hexahedral, Linear** elements (HF8 elements) in the thermal analysis. Ensure that a **Regular** mesh is used. (The mesh spacing will be governed by the assigned line mesh attributes). Enter the dataset name as **Coupled Brick Elements** and click **OK**
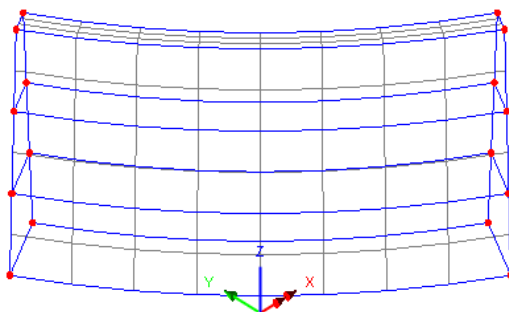
- Select all volumes in the model (or use the **Ctrl + A** keys to select all features) and drag and drop the **Coupled Brick Elements** mesh from the 🔴 Treeview onto the selected features.

Material Properties

The structural material will be defined first.

Attributes
Material
Structural
Isotropic…
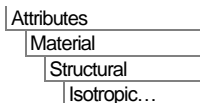
- Enter a Young's modulus of **30e9**, a Poisson's ratio of **0.2**, and a mass density of **2.4e3**. Click the **Thermal expansion** check box and enter a Coefficient of thermal expansion of **10e-6**. Change the name to be **Concrete Ungraded (Structural)** and Click **OK**.

The heat of concrete hydration thermal material will now be defined.

Attributes
Material
Thermal
Isotropic…
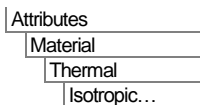
- Leave the Phase change state set to **None** and enter a Thermal conductivity of **2.0** (J/m·s·ºC), a Specific heat capacity of **1.0448e3** (J/kg·ºC), and a Density of **2.4e3**

- Ensure **Concrete Heat of Hydration** is selected in the Exothermic behaviour panel, and select Cement type **Type I.** Note that the timescale units have already been set when the model was first created and will be used in any heat of hydration analysis.

- Set the remaining values required in the dialog as follows: Mass of cement per unit volume **307**, Water/Cementitious ratio **0.47**, Mass of slag per unit volume **0**, Mass of fly ash per unit volume **0**, CaO content of fly ash (%) **0**, Delay for heat of hydration **0**.

- Change the attribute name to **Concrete Ungraded (Thermal)** and click **OK**.

Note. When computing the heat due to the rate of hydration of concrete, hours or days are convenient units to use for the time step and elapsed time. Due to the way the solution algorithms are formulated this is independent of the fundamental units of seconds and Newtons that are defined for the model.

Note. The 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. Reference [S18] *LUSAS Theory Manual.*

Assign the Material Properties

The concrete material properties now need assigning to the appropriate analyses:

- With all three volumes in the model selected, drag and drop **Concrete Ungraded (Structural)** from the Treeview onto the selected volumes. Press **OK** to assign to **Analysis 1 (Structural).**

- With all three volumes in the model selected, drag and drop **Concrete Ungraded (Thermal)** from the Treeview onto the selected volumes. Press **OK** to assign to **Analysis 1 (Thermal).**

Structural Supports

Attributes	
Support	>
Structural...	

A fully fixed support is required for the base and abutments of the dam.

- All degrees of **Translation** in the **X**, **Y** and **Z**-axes must therefore be set as **Fixed**. Enter the attribute name as **Fixed** and click **OK**
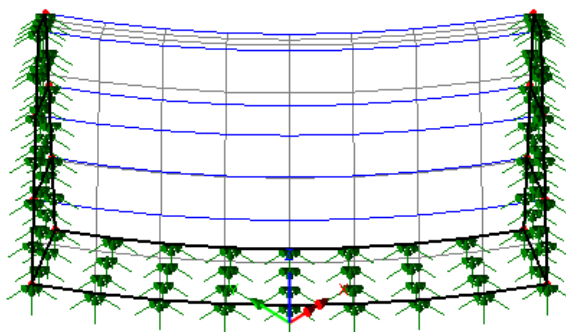
- Select the surfaces that define the base and both abutments of the dam

Tip. Hold-down the **Shift +S** keys and box-select the surfaces required.

- Then, from the Treeview, drag and drop the support attribute **Fixed** onto the selected

features, ensure the **Analysis 1 (Structural)** and the **All analysis loadcases** options are selected, and click **OK.**

Thermal Supports

A thermal support needs to be defined for use later in the example.

Attributes
  Support   >
    Thermal...

- Ensure the temperature support is set to **Free** and enter the attribute name as **Free** and click **OK.**

Do not assign this support to the model yet**.** It is to be used during the activation and deactivation of the thermal loadcases to free-up elements that have had their temperature restrained at the concrete placement temperature.
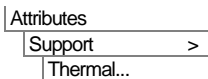
Thermal Loading

Several thermal loads need to be defined for applying to the model later. Environmental temperature loads for the air and ground temperatures will be defined first, followed by the initial concrete placement temperature.

Air temperature

Attributes
  Loading   >
    Thermal...

- Select **Environmental conditions** and click **Next**

- Enter a value of **21.1** (ºC) for the environmental temperature, and **8.333** (J/m^2·s·ºC) for the convection heat transfer coefficient. Leave the other two inputs blank. Enter the attribute name as **Air temperature**. Click **Apply**
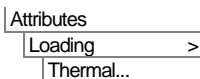
Ground temperature

- In the first row of the dialog, keep the value of **21.1** (ºC) for the environmental temperature, but over-type a value of **2.777** (J/m^2·s·ºC) for the convection heat transfer coefficient. Change the attribute name to **Ground temperature** and click **Finish**

Note. The ground is not modelled explicitly in this example and in real –life should be. The use of a environmental load is a simplification that will result in unconservative results.

Initial concrete temperature

The initial concrete temperature will be defined:

Attributes
| Loading >
| | Thermal...

- Select **Prescribed** and click **Next.**

- Select **Fixed** and enter a value of **21.1** (ºC). Ensure the **Total** option is selected. Enter the attribute name as **Initial concrete temperature**. Click **Finish**
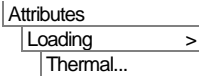
Modelling Staged Construction

In order 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, whilst loading attributes that apply during particular times during an analysis are applied to selected features of the model using Load curves. Modelling of staged construction processes is carried out in LUSAS by activating using the activation and deactivation facility.

Creating Activation and Deactivation datasets

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**

When building staged construction models it is often very helpful to see the elements that are actually activated in the loadcase being viewed:

- In the 🔲 Treeview, double-click the **View properties** control, and under the **View** tab ensure the **Show only activated elements** option is selected, and click **Close.**
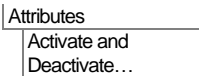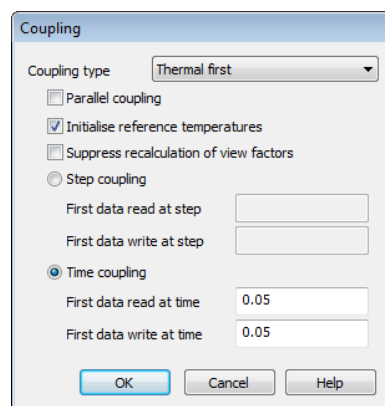
Set coupled analysis options

In the 🕒 Treeview, two analysis entries can be seen: Analysis 1 (Thermal) and Analysis 1 (Structural). These are created automatically as a result of selecting a Coupled user interface option at the beginning of the example. There is also a 🐞 Coupled analysis options object, which permits coupling options to be specified.

In this analysis, the thermal results affect the structural behaviour (that is, temperatures cause thermal stresses), but the structural results do not affect the thermal behaviour. A semi-coupled analysis is therefore required, with a coupling type of 'Thermal first' and other appropriate coupled analysis specfied.

- Double click on the 🐞 **Coupled analysis options** object in the 🕒 Treeview.

- Set the coupling type as **Thermal first** from the drop down list.

- Select the **Initialise reference temperatures** option.

- Ensure the **Time coupling** radio button is selected.

- Set the 'First data read at time' to **0.05** and the 'First data write at time' to **0.05.**

- Click **OK**

Note. Coupling options that are set on this dialog dictate the coupling options that can be set via the Nonlinear & Transient options dialog.

Defining Loadcase properties

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

Defining a structural initialisation loadcase

To make the model easier to manipulate, the default analysis names will be changed to something more descriptive.

Defining a structural initialisation loadcase

- In the Structural analysis, right-click on **Loadcase 1** and rename it to **Structural Initialisation**

- Select **Structural Initialisation** using the right-hand mouse button and from the **Controls** menu select the **Nonlinear and Transient** option.

- On the Nonlinear & Transient dialog select the **Nonlinear** option and leave the incrementation type as **Manual**

- Select the **Time domain** option. Choose a **Viscous** time domain from the drop-down list. Enter the Initial time step as **1e-6**, leave the Total response time as its default value, and set the Max time steps or increments to **1**

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

Note. The small initial time step of 1e-6 has been used to minimise the curing effects during this loadcase. In this instance the large default total response time has no significance because only one time step is processed. This load case will finish after the first small time step.

Defining a structural loadcase for construction stage 1

Analysis
  Loadcase

- Ensure the **Analysis 1 (Structural)** analysis is selected, then enter a loadcase name of **Structural stage 1** and click **OK**

- Select **Structural stage 1** using the right-hand mouse button and from the **Controls** menu select the **Nonlinear and Transient** option.

- On the Nonlinear & Transient dialog select the **Nonlinear** option and leave the incrementation type as **Manual**

- Select the **Time domain** option. Choose a **Viscous** time domain from the drop-down list. Enter the Initial time step as **0.05**, the Total response time as **10**, and set the Max time steps or increments to **100**

- Select the **Advanced** button, and on the Advanced Time Step Parameters dialog set the Time step increment restriction factor to **1.5**, the Minimum time step to **0.05** and the Maximum time step to **0.5** This will allow the time step to increase from its initial value of 0.05 up to the maximum value of 0.5 and thus reduce the number of increments required.

- Click **OK** to close the Advanced Time Step Parameters dialog and return to the Nonlinear & Transient dialog.

- On the Nonlinear & Transient dialog press the **Coupling** button and on the Coupling Data dialog change both the 'Time between coupled reads' to **0.05**, and the 'Time between coupled writes' to **0.05**, and click **OK**.

- Back on the Nonlinear and Transient dialog click **OK** to return to the view window.
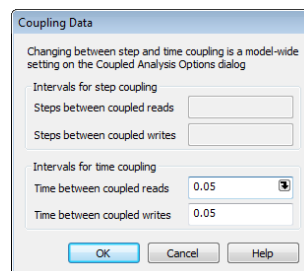
Defining a structural loadcase for construction stage 2

Loadcases can be copied and pasted in the Treeview. This saves having to re-enter similar details for each loadcase. Then, just the different values can be defined for each newly copied loadcase.

- In the ⬜ Treeview, click on loadcase **Structural stage 1**. Press the 📋 copy toolbar button, followed by the 📋 paste toolbar button. This will create a new loadcase that is an exact copy named **Structural stage 1 (Copy 1)**. Rename this new structural loadcase to **Structural stage 2**

- Double-click the 🌀 **Nonlinear and Transient** object for **Structural stage 2** and set the total response time as **20**

Defining a structural loadcase for construction stage 3

- Repeat the copy and paste procedure to create copy of the Structural stage 2 and rename it to be **Structural stage 3**, andset the total response time as **30**

Defining a thermal initialisation loadcase

- In the ⬜ Treeview, in Analysis 1 (Thermal), right-click on the thermal Loadcase 1 and rename it to **Thermal Initialisation**

- Select **Thermal Initialisation** using the right-hand mouse button and from the **Controls** menu select the **Nonlinear and Transient** option.

- On the Nonlinear & Transient dialog select the **Nonlinear** option and leave the incrementation type as **Manual**

- Select the **Time domain** option. Choose a **Thermal** time domain from the drop-down list. Enter the Initial time step as **1e-6**, leave the Total response time as its default value, and set the Max time steps or increments to **1**

- In the Solution Strategy section set the 'Max number of iterations' to **15**

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

Note. The small initial time step of 1e-6 has been used to minimise the curing effects during this loadcase. In this instance, again, the large default total response time has no significance because only one time step is processed. This load case will finish after the first small time step.

Defining a thermal loadcase for construction stage 1

Analyses
  Loadcase...

- Select **Analysis 1 (Thermal)** from the drop-down list of analysis names.

- Enter **Thermal stage 1** (overtype the default loadcase name).

- Click on loadcase **Thermal stage 1** using the right-hand mouse button and from the **Controls** menu choose the **Nonlinear and Transient** option.

- On the Nonlinear & Transient dialog tick the **Nonlinear** option and leave the incrementation type as **Manual**

- Select the **Time domain** option and note that the **Thermal** time domain option is the only one available for selection. Enter the Initial time step as **0.05**, the Total response time as **10**, and set the Max time steps or increments to **100**

- Select the **Advanced** button, and on the Advanced Time Step Parameters dialog set the Time step increment restriction factor to **1.5**, the minimum time step to **0.05** and the Maximum time step to **0.5**

- Click **OK** to close the Advanced Time Step Parameters dialog and return to the Nonlinear & Transient dialog. On this dialog, pick the **Coupling** button and change the 'Time between coupled reads' to **0.05**, and the 'Time between coupled reads' to **0.05**, and click **OK**. Click **OK** to return to the view window.

Now create a copy of this loadcase:

Defining a thermal loadcase for construction stage 2

- Click on loadcase **Thermal stage 1**. Press the copy toolbar button, followed by the paste toolbar button. This will create a new loadcase that is an exact copy named **Thermal stage 1 (Copy 1)**. Rename this new structural loadcase to **Thermal stage 2**

- Double-click the Nonlinear and Transient object for **Thermal stage 2**, set the total response time as **20**

Defining a thermal loadcase for construction stage 3

- Repeat the copy and paste procedure to create copy of the Structural stage 2 and rename it to be **Thermal stage 3**, and set the total response time as **30**

All loadcases have now been defined. Element activations and model attributes will be assigned to these loadcases later.

Note. The total response time in each loadcase represents the total time at which that loadcase in the analysis ends, and the next one begins.

Load Curves

Load curves are used to describe the variation of the loading in nonlinear, transient and Fourier analyses. In a transient thermal analysis, such as in this example, the environmental temperature loading on specific surfaces of the dam will change with time. As the construction progresses some surfaces that were initially external surfaces (that could dissipate heat to the outside environment) become internal ones that can only dissipate heat to other parts of the structure.

Defining Load Curves

- Right-click on the **Analysis 1 (Thermal)** ⬄ icon, and choose **New** and then **Load Curve** to open the Load Curve dialog.

Defining a concrete placement load curve:

- In the User-defined section of the dialog enter a Time of **0** and a Factor **1**, press the **Tab** key to create a new row.

- On this new row enter a Time of **30** and a Factor of **1**.

- Leave the default values of Activation time (0.0) and Scaling factor (1.0) unchanged.

- Name the load curve **Concrete placement** and click **Apply**

- With reference to data that follows, repeat this procedure four more times to add four more load curves called **External surface stage 1**, **External surface stage 2**, **External**

surface stage 3 and **Base and abutments**. The User-defined area of each of the dialog boxes should be filled-in according to the tables that follow.

Time	Factor
0	1
9.999	1

Table 1. External surface stage 1

Time	Factor
10	1
19.999	1

Table 2. External surface stage 2

Time	Factor
20	1
30	1

Table 3. External surface stage 3

Time	Factor
0	1
30	1

Table 4. Base and abutments

Assigning Activation and Deactivation attributes

Construction Stage 1

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

- In the View window select the top two volumes of the dam.
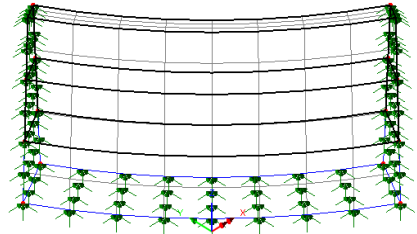
Structural activation/deactivation

- Drag and drop the deactivation attribute **Deactivate** from the 🔴 Treeview, ensuring that it is assigned to **Analysis 1 (Structural)** and loadcase **Structural Initialisation** and then click **OK.**

Because the option to show the activated mesh only was set earlier in the example the view window will update to show only the mesh elements for the lowest volume.

Thermal activation/deactivation

- With the top two volumes still selected, assign the deactivation attribute **Deactivate** from the 🔴 Treeview onto the selected features ensuring that it is assigned to **Analysis 1 (Thermal)** and loadcase **Thermal Initialisation**

Verifying self weight and activation assignments

- If, during the course of this example, you need to check when particular elements become active in an analysis, right click on an attribute in the 🔵 Treeview and select **Edit Assignments**.

Construction Stage 2

The elements in the second construction stage now need to be activated.

- In the view window select only the second stage of the dam (i.e. the middle volume).

Structural activation/deactivation

- Assign the activation attribute **Activate** from the 🔵 Treeview, ensuring that it is assigned to **Analysis 1 (Structural)** and loadcase **Structural Stage 2**, and click **OK.**
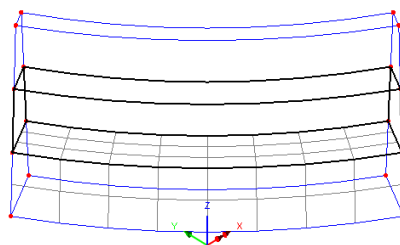
Thermal activation/deactivation

- With the middle volume still selected, assign the activation attribute **Activate** from the 🔵 Treeview ensuring that it is assigned to **Analysis 1 (Thermal)** and loadcase **Thermal Stage 2**

Construction Stage 3

The elements in the third construction stage now need to be activated.

- In the view window select only the top volume of the dam.

Structural activation/deactivation

- Assign the activation dataset **Activate** from the 🔵 Treeview, ensuring that it is assigned to **Analysis 1 (Structural)** and loadcase **Structural stage 3**, and click **OK**
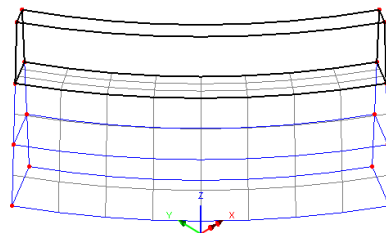
Thermal activation/deactivation

- With the top volume still selected, assign the activation attribute **Activate** from the 🔵 Treeview ensuring that it is assigned to analysis **Analysis 1 (Thermal)** and loadcase **Thermal stage 3**

Adding self-weight

- In the 🔍 Treeview right-click on the 🟰 **Analysis 1(Structural)** icon and select **Add gravity** to apply self-weight loading to all the structural loadcases in the analysis.

✏️ **Note.** Loadcases with automatic self-weight loading can easily be identified by the 🔍 gravity loadcase icon.

Assigning thermal loading (load curves)

Load curves were defined to model the thermal loading and these need to be assigned to appropriate features of the model.

Assigning the Initial Concrete Temperature

The initial concrete temperature attribute will apply to all the stages of the model as they are activated.

- Select the whole model and assign the thermal loading attribute **Initial concrete temperature** from the 🔵 Treeview, ensuring that it is assigned to Load curve **Concrete Placement** in the drop-down list before clicking **OK.**
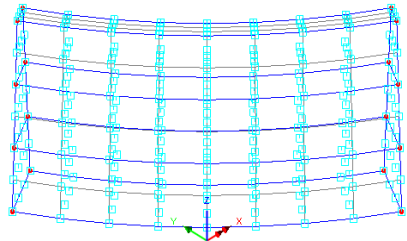
Assigning ground temperature

Ground temperature loading only applies to selected surfaces. These could be selected one-by-one but, as an alternative, they can also be selected by their support assignment. First, a structural loadcase must be set active to allow selection of an assigned structural attribute:

- In the 🔍 Treeview, right-click on the **Structural Initialisation** loadcase and select **Set active.**

- In the 🔵 Treeview right-click on the Supports entry **Fixed** and choose **Select Assignments.** Click **OK** to clear the previously selected features.

- With the bottom and side surfaces only selected, assign the thermal loading attribute **Ground temperature** from the 🔴 Treeview, ensuring that <u>each of the environmental load conditions</u> are assigned to Load curve **Base and abutments** from the drop-down list, before clicking **OK**.
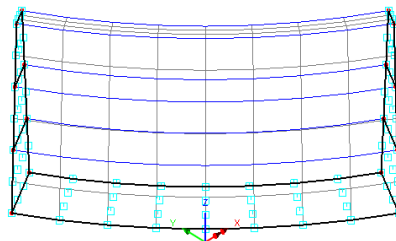
Assigning air temperatures and thermal supports

The surfaces in contact with the air change as the construction progresses. To help ensure the correct surfaces are selected prior to assigning air temperature loading the pre-defined Groups (that were set-up at the start of the example to mimic the construction process) will be used. Thermal supports also need to be assigned to the model at each construction stage to free-up nodes in the model that are restrained at a prescribed temperature by the LUSAS Modeller.
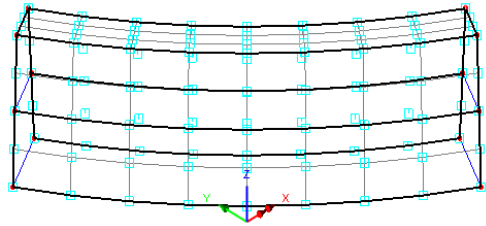
Construction Stage 1

- In the 🔵 Treeview right-click on the group name **Stage 1** and select the **Set as Only Visible** option.

- Click in the view window to allow a keyboard short-cut to be used.

- Holding-down the **Shift** + **S** keys select the front, back and top Surfaces of the concrete that forms stage 1

- Assign the thermal loading attribute

   Air temperature from the 🔴 Treeview, ensuring <u>each of the environmental load conditions</u> are assigned to Load curve **External surface stage 1** from the drop down list before clicking **OK**

- Select the Volume and assign the thermal support **Free** from the 🔴 Treeview, ensuring that it is assigned to volumes from loadcase **Thermal stage 1** and click **OK**
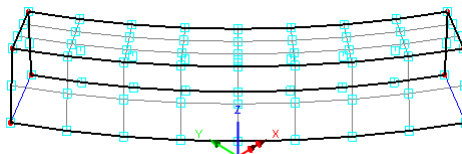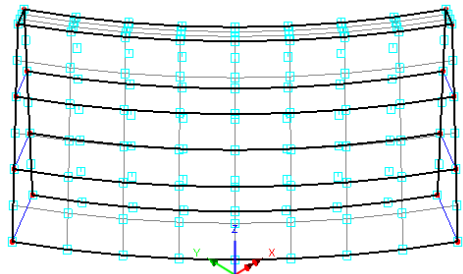
Construction Stage 2

- In the Treeview right-click on the group name **Stage 2** and select the **Set as Only Visible** option.

- Click in the view window to allow a keyboard short-cut to be used.

- Holding-down the **Shift + S** keys select the front, back and top surfaces of the concrete that form stage 2 (5 surfaces in total)

- Assign the thermal loading attribute **Air temperature** from the Treeview, ensuring <u>each of the environmental load conditions</u> are assigned to Load curve **External surface stage 2** from the drop down list before clicking **OK**

- Select the two Volumes representing this construction stage and assign the thermal support **Free** from the Treeview, ensuring that it is assigned to volumes from loadcase **Thermal stage 2** and click **OK.**

Construction Stage 3

- In the Treeview right-click on the group name **Stage 3** and select the **Set as Only Visible** option.

- Click in the view window to allow a keyboard short-cut to be used.

- Holding-down the **Shift + S** keys select the front, back and top surfaces of the concrete that forms stage 3 (7 surfaces in total)

- Assign the thermal loading attribute **Air temperature** from the Treeview, ensuring that it is assigned to Load curve **External surface stage 3** in the drop down menu before clicking **OK**

- Select the three Volumes representing this construction stage and assign the thermal support **Free** from the Treeview, ensuring that it is assigned to volumes from loadcase **Thermal stage 3** before clicking **OK**

The model is now complete.

File
  Save

🖫 Save the model file.

Checking the Attributes and Loadcase Treeviews

In defining and modelling the staged construction of the dam the Attributes 🎛 Treeview and the Analyses 🔍 Treeview will now contain all the information for the LUSAS Solver to carry out an analysis.

For a successful analysis your model should contain treeviews similar to those shown in the following images.
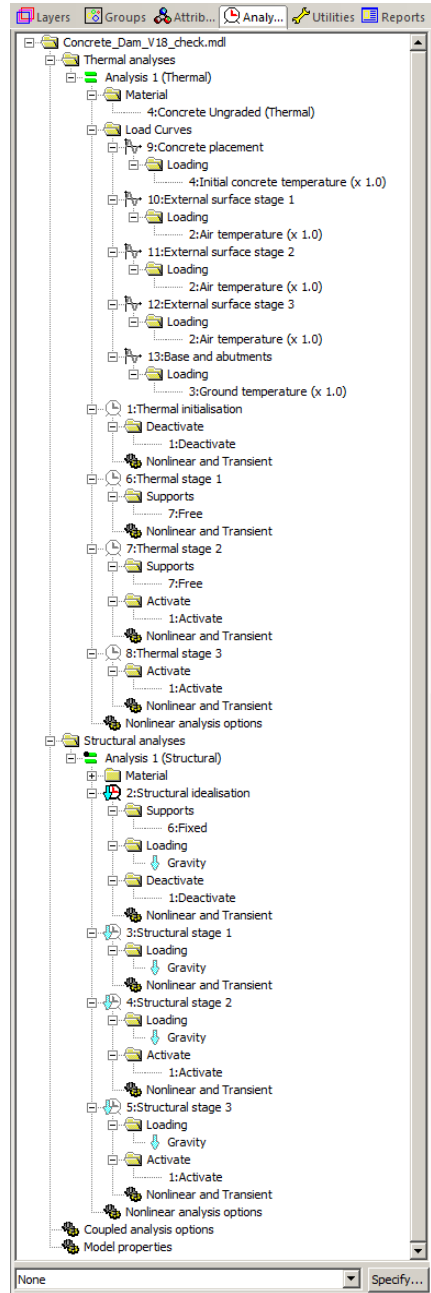
Attributes treeview
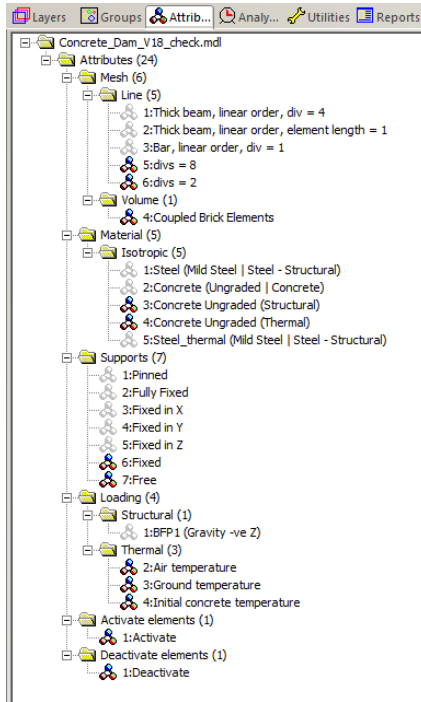
Note that right-clicking on attribute and loadcase data allows any defined values to be checked.

If your model has potential errors in your Attribute data or in your Analyses Treeviews, a file is provided to enable you to re-create the model from scratch and run the analysis successfully. See the next page for details.

Analyses treeview

Running the Analysis

With the model loaded:

 Open the **Solve Now** dialog. Ensure that the **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.

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

In addition, 4 files will be created in the Associated Model Data directory where the model file resides:

- ❑ **concrete_dam_therm.out** this output file contains details of model data, assigned attributes and selected statistics of the thermal analysis.
- ❑ **concrete_dam_struct.out** this output file contains details of model data, assigned attributes and selected statistics of the structural analysis.
- ❑ **concrete_dam_therm.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_struct.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_modelling.vbs** carries out the modelling of the example.

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**

- 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_modelling.vbs** located in the **\\<LUSAS Installation Folder>\Examples\Modeller** directory.

🟰 Run the analysis to generate the results.

Viewing the Results

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

Thermal Results

To illustrate the variation of temperature throughout the model with time, an animation will be created showing contours at different time steps. Firstly, contours of temperature for the current time step will be viewed.

- Turn off the display of the **Mesh, Deformed Mesh** and **Attributes** layers in the 🗖 Treeview.

- In the 🕒 Treeview right-click on the Thermal loadcase results for **Time Step 1** and **Set Active**

- With no features selected click the right-hand mouse button in a blank part of the View window and select the **Contour** option to add the contour layer to the 🗖 Treeview. Select **Potential** from the entity drop down list and ensure **PHI** is selected in the component drop down list. Click **OK**.

A plot showing initial contours at ambient temperature will be displayed.

When creating animations of contours it is preferable to have a static contour scale. By setting a Time Step active for the end of the construction stage under consideration an

estimate of the maximum and minimum values required for the contour key can be obtained.

- In the ⏰ Treeview right-click on the Thermal loadcase results for **Time Step 25** and **Set Active**

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

- Select the **Contour Range** tab and ensure that the contour **Interval** contour is set to **5.0**. Set the **Maximum** and **Minimum** values to **60.0** and **20.0** respectively. Click **OK**

Analysis: Thermal
Loadcase: 26:Thermal Stage 1, Time Step 25 Time = 10.0000
Results file: concrete_dam~Thermal.mys
Response time: 10.0
Entity: Potential
Component: PHI (Units: C)

| 20.0 |
| 25.0 |
| 30.0 |
| 35.0 |
| 40.0 |
| 45.0 |
| 50.0 |
| 55.0 |
| 60.0 |
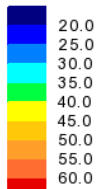
Maximum 78.1401 at node 41
Minimum 22.3072 at node 73
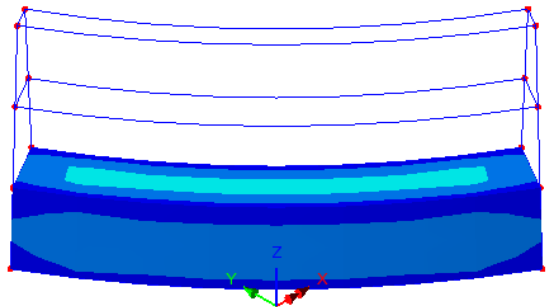
Animating Thermal Results

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 the external 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 **Thermal** entry, then in the Step field enter **2**.
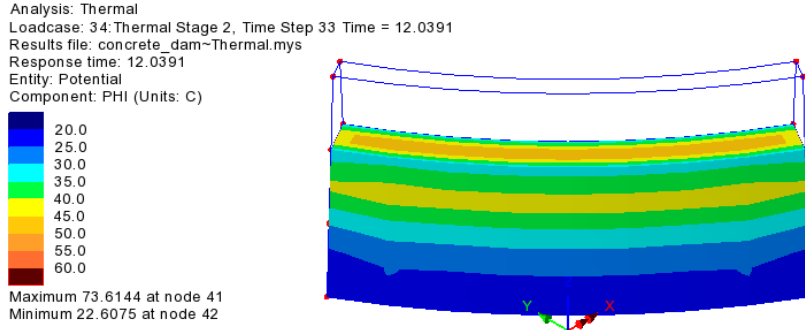
- Press the 'Add to' ⟩⟩ button, and click **Finish**

The first and every 2nd Time Step of each construction stage will be added to the 'Included' area of the dialog.

Analysis: Thermal
Loadcase: 34: Thermal Stage 2, Time Step 33 Time = 12.0391
Results file: concrete_dam~Thermal.mys
Response time: 12.0391
Entity: Potential
Component: PHI (Units: C)

20.0
25.0
30.0
35.0
40.0
45.0
50.0
55.0
60.0

Maximum 73.6144 at node 41
Minimum 22.6075 at node 42

LUSAS will load-in the selected time step results and open a window showing an animation of the variation of temperature with time. Animations may be saved in compressed AVI format for playback in other applications.

File
Save As AVI…

- Save the animation to your projects folder and enter **concrete_dam_thermal.avi** for the filename and press **Save**

- Close the Animation window.

## Animating Thermal Results on a Slice Section

With heat of hydration analysis the maximum temperatures reached will occur within the concrete. To plot the change of temperature over time inside the dam slice sections are used:

Note. When creating animations using slice sections the animation results are only created for the elements that are sliced. This means that for this example one of the Thermal loadcase Time Step results for Stage 3 must be set active so that the complete mesh for the dam can be seen.

- In the Treeview right-click on the Thermal Stage 3 loadcase results for **Time Step 60** and select **Set Active**

Analysis: Thermal
Loadcase: 60:Thermal Stage 3, Time Step 59 Time = 23.0391
Results file: concrete_dam~Thermal.mys
Response time: 23.0391
Entity: Potential
Component: PHI (Units: C)

20.0
25.0
30.0
35.0
40.0
45.0
50.0
55.0
60.0
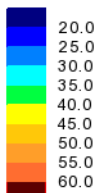
Maximum 70.0859 at node 186
Minimum 21.6096 at node 7
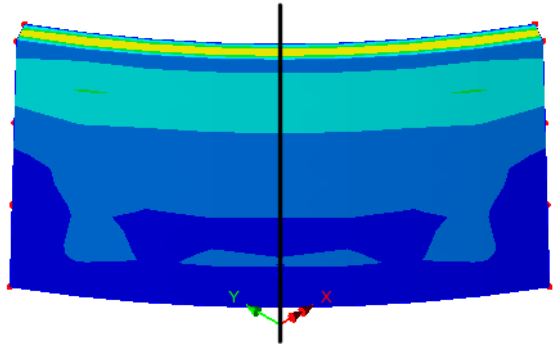
Utilities
Section Through 3D…

- Click **OK** to accept a grid size of 1

- Position the cursor above the dam in-line with the Z-axis and click and drag vertically downwards to define a slice section right through the centre of the dam. A group named Slice 1 will be created in the 🔷 Treeview.

- In the 🔲 Treeview select double-click the **Contour** option, ensure the **Display on slice(s)** option is selected and click **OK**
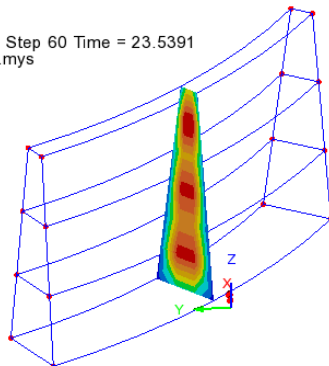
🔧 Use the Dynamic rotate button to rotate the model around to the view shown.

Analysis: Thermal
Loadcase: 61:Thermal Stage 3, Time Step 60 Time = 23.5391
Results file: dam_from_vbs~Thermal.mys
Response time: 23.5391
Entity: Potential
Component: PHI (Units: C)

20.0
25.0
30.0
35.0
40.0
45.0
50.0
55.0
60.0
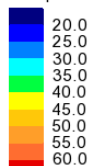
Maximum 68.4665 at node 734
Minimum 22.0616 at node 330

To create and save an animation of the temperature variation on this slice section through the dam, complete the following procedure:

Tools
Animation Wizard…

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

- In the 'Available' panel click on the **Thermal** entry, then in the Step field enter **4**.

- Press the 'Add to' [>>] button, and click **Finish**

Save the animation to your projects folder and enter **concrete_dam_thermal_slice.avi** for the filename. Click **OK**

- Close the Animation window.

Plotting contours on the whole model after slicing

- In the ▣ Treeview select double-click the **Contour** option and <u>deselect</u> the **Display on slice(s)** option and click **OK**.

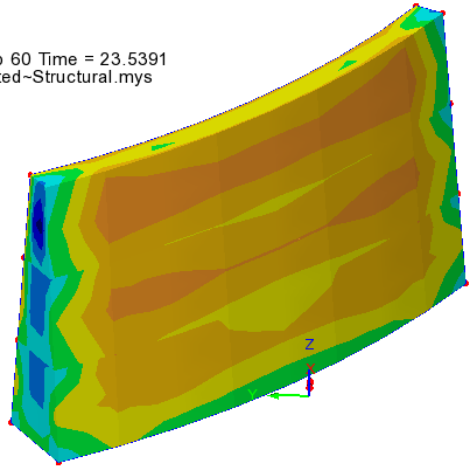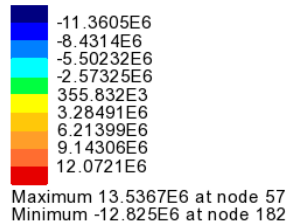The display will revert to show contours of Potential of entity PHI on the whole model.

- Remove any slices generated by picking the **Slices** group from the ▣ Treeview, clicking right-hand mouse button and choosing the **Delete** menu item from the context menu.

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 the ▣ Treeview right-click on the structural results for **Time Step 60** and **Set Active**

- Double-click the contours layer in the ▣ Treeview. On the dialog, set the results entity to **Stress – Solids** and component **S1**.

- Select the **Contour Range** tab and ensure that an **Automatic** contour range using **9** contours is being used. Deselect the **Maximum** and **Minimum** values and click **OK**

Analysis: Structural
Loadcase: 5:Structural Stage 3, 61:Time Step 60 Time = 23.5391
Results file: concretedamAl_coupling_corrected~Structural.mys
Response time: 23.5391
Entity: Stress - Solids
Component: S1 (Units: N/m²)

-11.3605E6
-8.4314E6
-5.50232E6
-2.57325E6
355.832E3
3.28491E6
6.21399E6
9.14306E6
12.0721E6

Maximum 13.5367E6 at node 57
Minimum -12.825E6 at node 182

This will show contours of the maximum principal stress (S1) at the current time step.

## Graphing of Results at a Selected Node

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 selecting a node of interest and using the Graph Wizard - making sure that only thermal results and thermal time steps are selected for plotting.

- In the ⬚ Treeview, turn off the **Contour** and **Geometry** layers and turn on the **Mesh** layer.

- In the 🕒 Treeview right-click on **Time Step 60** in the **Thermal Stage 3** loadcase results and **Set Active**

- Double-click the **Mesh** layer and select **Hidden parts** and deselect **Dotted** to view all the mesh.

- Select the stage 1 node for results graphing. (See the next image)

Select this node (only) for stage 3 results graphing

Select this node (only) for stage 2 results graphing
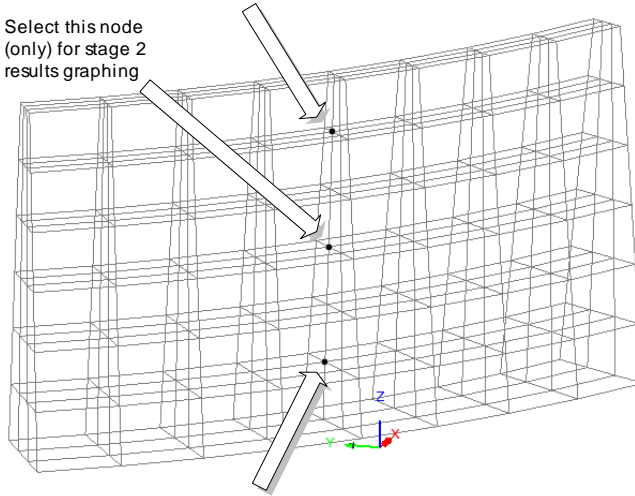
Select this node (only) for stage 1 results graphing

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

To define the graph X axis details:

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

- In the Sample loadcases panel ensure **Whole analysis** is selected and choose the **Thermal** analysis from the drop-down box. 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 that entity **Potential** and results component **PHI** are selected for the node shown on the dialog and click **Next**.

- Enter the title as **Thermal results for selected internal nodes**

- Enter the X axis title as **Response time (Days)**

- Enter the Y axis as **Temperature (Degrees)** and click **Finish** to create the graph.

Thermal results for selected internal nodes

Temperature Node 55(2) / Response time(1)
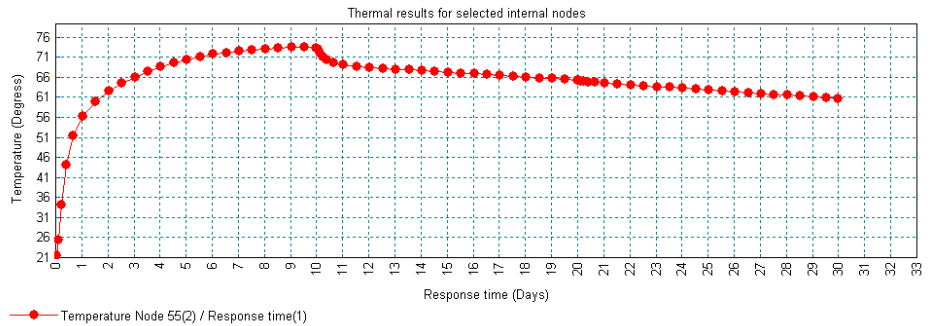
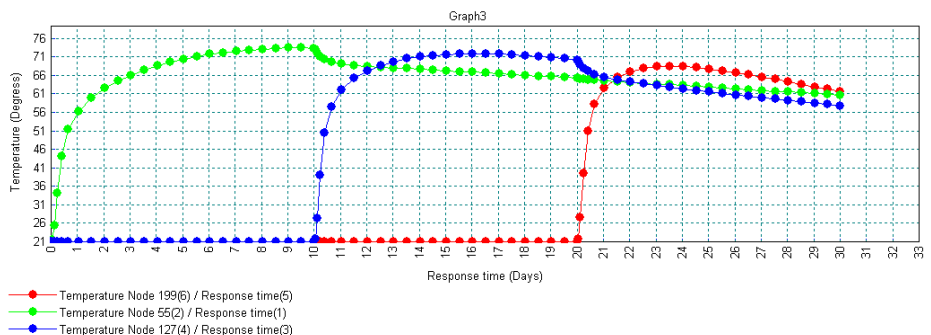• Without deleting the Graph Window return to the Model Window

Graphing Results for Other Nodes

When a node in the mid-section of the dam for the other two construction stages is selected the following graph can be created, which clearly shows the initial rise and subsequent reduction of temperature after each stage of concrete is constructed.

Utilities
   Graph wizard…

• For each additional node identified, select it, and run the Graph Wizard. Repeat the above steps, but this time select **Previously defined** for the X-Attribute Entity Data and then re-use the **Response time** dataset by selecting that option on the dialog presented.

• Make sure that results for previously created graphs are plotted onto this latest graph by selecting the **Include existing graph** option for both previously created graphs at the final stage of the wizard.

Graph3

Temperature Node 199(6) / Response time(5)
Temperature Node 55(2) / Response time(1)
Temperature Node 127(4) / Response time(3)

Note. When including graphs into graphs as done here, the original colour of the included graph cannot be retained. But graph properties (colours, symbols and the editing of axis or curve titles) can be edited by right-clicking on the Graph and choosing **Edit Graph Properties**

This completes the example.