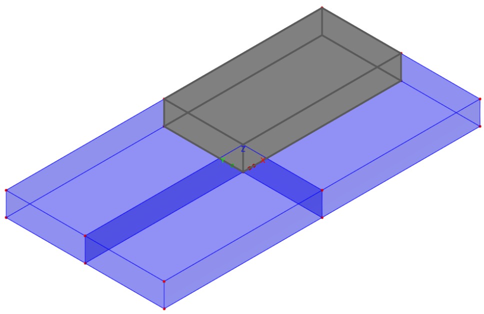
Heat of hydration modelling of a concrete base slab

For LUSAS version:	22.0
For software product(s):	Any Plus version
With product option(s):	Nonlinear, Dynamic, Heat of Hydration.

Description



A concrete base slab has a length of 40m, a width of 20m and a height of 3m. For the purposes of this example it is assumed to be cast in one pour.

The temperature effects from modelling casting the slab with a constant ambient temperature and a varying ambient temperature are investigated on a quarter model of the slab, as shown coloured in grey in the adjacent image. Side formwork is modelled as being present from 0 to 3 days and then removed for the duration of the investigation.

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

Objectives

The required output from the analysis consists of:

- A graph showing temperature against response time for a selected location within the slab for the ambient temperature conditions investigated.

Keywords

3D, Slab, Concrete, Heat of Hydration, Loadcase, Load curve, Temperature, Graphing.

Associated Files

- **slab_HHC_modelling.lvb** carries out the modelling for Analysis 1 – constant ambient temperature.
- **slab_HHC_loadcurves.lvb** carries out the modelling for Analysis 1 and Analysis 2 – varying ambient temperature.

Analysis 1 – Constant ambient temperature

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

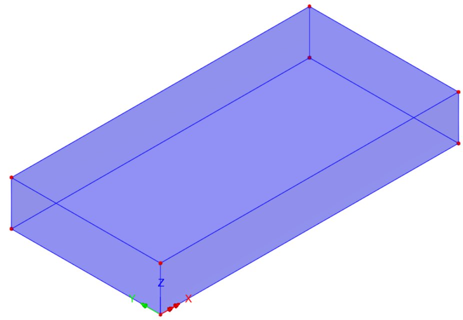
File
New…

- Enter a file name of **Slab_HHC**
- Use the default **User-defined** working folder.
- Set analysis type to be **Thermal**.
- Set the analysis category to be **3D**.
- Set modelling units to be **N,m,kg,s,C**.
- Set timescale units of **Days**.
- Enter a title of **Heat of hydration modelling of a concrete slab**
- Click the **OK** button.

Defining the slab geometry

Geometry
Shape Wizard…

- Select a **Cuboid** shape of type **Volume** and click **Next**.
- Enter a length in X of **20**, in Y of **10** and in Z of **3** and click **Finish**.
- Press the isometric button to view the volume model generated.



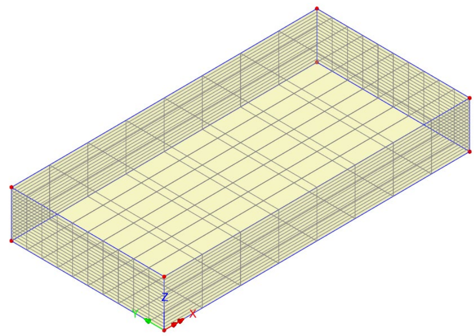
Defining and assigning the mesh

Attributes
Mesh >
Volume…

- Ensure Element type is set to **Field**, Shape is **Hexahedral** and Interpolation order is set to **Quadratic.**
- Deselect the **Automatic** divisions checkbox and enter divisions in x, y and z of **16**, **10** and **8** respectively, and click **OK**.

With the whole model selected:

- From the treeview drag and drop the mesh attribute **Vmsh1** onto the model.
- Click in an empty region of the view window to deselect the selection and view the mesh.



Defining and assigning the material properties

Attributes
Material >
Isotropic…

- On the Thermal tab enter a Thermal conductivity of **2**, a Specific heat capacity of **1.0448E3** and a Density of **2400** – noting that the units in use for each can be seen when hovering over the cells.
- In the Exothermic behaviour section of the dialog select **Concrete Heat of Hydration**.
- Select the 'Cement Type' to be **Type I**, define the 'Mass of cement per unit volume' to be **307** and enter a 'Water/Cementitious mass ratio' of **0.47** and click **OK**.

With the whole model selected:

- From the treeview drag and drop the material attribute **Iso1** onto the model, assigning it to **Volumes** for **All analysis loadcases**. Click **OK**.

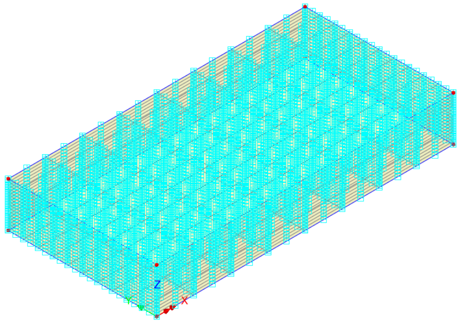
Defining the temperature loading

Attributes
Loading...

- Select **Prescribed** and click **Next.**
- Ensure that **Total**, **Fixed**, temperature loading of **20** degrees is specified to represent the stress-free ambient temperature. Enter a name of **Initial Temp** and click **Finish**.

With the whole model still selected:

- From the treeview drag and drop the loading attribute **Initial Temp** onto the model, assigning it to **Volumes** for **Loadcase 1**. Click **OK**.
- In the Treeview, rename Loadcase 1 to be **Initialisation**



Defining and assigning the supports

Attributes
Support...

- Select **Free**, enter a name of **Free** and click **OK**.

With the whole model still selected:

- From the treeview drag and drop the support attribute **Free** onto the model assigning it to **Volumes**. Select the **From loadcase** radio button and type **Formwork for 3 days** into the loadcase field to create a new loadcase. Click **OK**.

Note. This frees up the temperatures from the prescribed temperature loading which "locked" the temperatures in the Initialisation loadcase.

Defining environmental conditions

These will define the ambient temperatures and rate of heat transfer on the faces of the volume. Any face not assigned environmental conditions will be treated as an internal concrete face, i.e. a mirror plane.

Attributes
Loading...

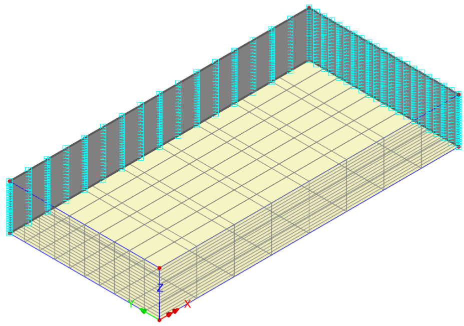
- Select **Environmental conditions** and click **Next**.
- Untick the **Temperature dependent** checkbox.

- Enter an Environmental temperature of **20** and a Convection heat transfer coefficient of **2.777**. Enter a name of **Face Ground Temperature** and click **Apply**.
- Leaving the environmental temperature set to **20**, change the convection coefficient to be **8.333**. Change the name to be **Face Air Temperature** and click **Apply**.
- Lastly, leaving the environmental temperature set to **20**, change the convection coefficient to be **7**. Change the name to be **Face Formwork Temperature** and click **Finish**.
- Click in an empty region of the view window to deselect the volumes selected.

Assigning environmental conditions

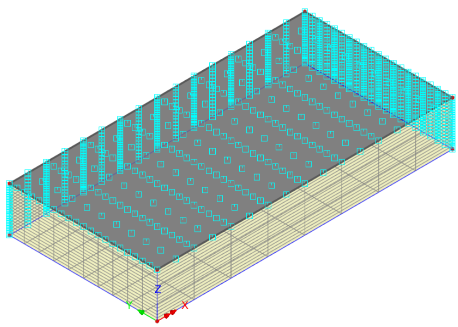
Side faces

- Select the two outer vertical faces of the base slab.
- From the treeview, drag and drop the **Face Formwork Temperature** onto the selection assigning it to the loadcase **Formwork for 3 days**.
- From the treeview, drag and drop the **Face Air Temperature** onto the selection and assigning it to a new loadcase by over-typing the name with **Exposed until 30 days**.



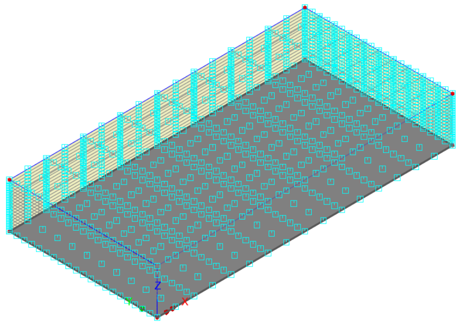
Top face

- Select the top face of the slab.
- From the treeview, drag and drop the **Face Air Temperature** onto the selection assigning it to the loadcase **Formwork for 3 days**.
- From the treeview, drag and drop the **Face Air Temperature** onto the selection and assigning it to the loadcase **Exposed until 30 days**..



Bottom of slab

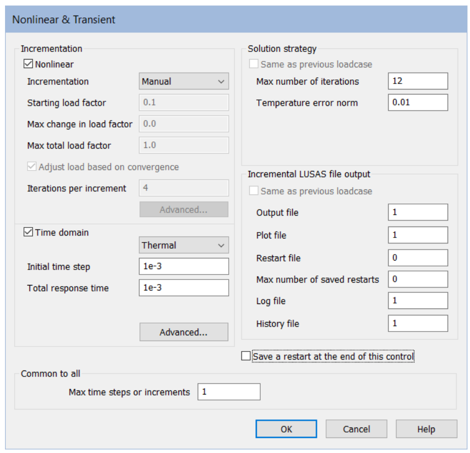
- Select the bottom face of the slab.
- From the treeview, drag and drop the **Face Ground Temperature** onto the selection assigning it to the loadcase **Formwork for 3 days**.
- From the treeview, drag and drop the **Face Ground Temperature** onto the selection assigning it to the loadcase **Exposed until 30 days**.



Define nonlinear and transient controls

'Initialisation' loadcase

- In the Treeview, right-click on the **Initialisation** loadcase and select **Controls** then **Nonlinear and transient**.
- In the Incrementation section tick **Nonlinear** to turn it on.
- Select **Time Domain**, enter an initial timestep of **1e-3,** a total response time of **1e-3**, a maximum number of time steps of **1**, and click **OK**.



'Formwork for 3 days' loadcase

- In the Treeview, right-click on the **Formwork for 3 days** loadcase and select **Controls** then **Nonlinear and transient**.
- In the Incrementation section tick **Nonlinear** to turn it on.
- Select **Time Domain**, enter an initial timestep of **0.05,** a total response time of **3**

- Click the **Advanced** button and change the Time step increment factor to **1.25** and click **OK**. Then **OK** again to exit the main dialog.

'Exposed for 30 days' loadcase

- In the Treeview, right-click on the **Exposed until 30 days** loadcase and select **Controls** then **Nonlinear and transient**.
- In the Incrementation section tick **Nonlinear** to turn it on.
- Select **Time Domain**, enter an initial timestep of **0.5,** a total response time of **30**
- Click the **Advanced** button and change the Time step increment factor to **1.25** and click **OK**. Then **OK** again to exit the main dialog.

The model is now complete.

Turn off the display of the loading

Press the loading on-off button to turn off the loading.

Save the model

File
Save

Save the model file.

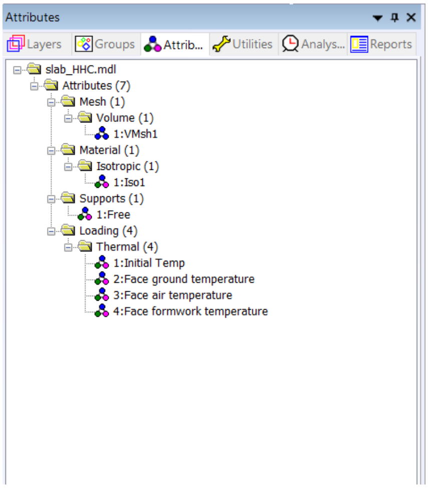
Checking the Attributes and Analyses Treeviews

The Attributes Treeview and the Analyses Treeview now contain all the information for an analysis to be carried out.

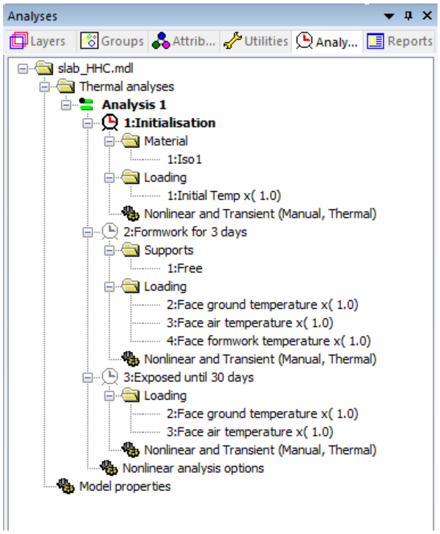
For a successful analysis your model should contain treeviews similar to those that follow. Right-clicking on attribute and loadcase data allows any defined values to be checked. If your model has potential errors in your Attribute data, or in your Loadcase assignments that you cannot fix, a file is provided to enable you to re-create the model from scratch and run the analysis successfully.

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

Attributes treeview



Analyses treeview



Running the Analysis

With the model loaded:

Open the **Solve Now** dialog.

- Ensure that the **Analysis 1** is 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.

Associated files will be created in the LUSASFiles\<model_name> folder.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered is written to an output file 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:

- ❑ **slab_HHC_modelling.lvb** carries out the modelling this first part 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 **slab_HHC**
- Select units of **N,m,kg,s,C**
- Ensure the **Thermal** user interface is selected and click the **OK** button

File
Script >
Run Script...

- To recreate the model, open the file **slab_HHC_modelling.lvb** that was downloaded and placed in a folder of your choosing.

Run the analysis to generate the results.

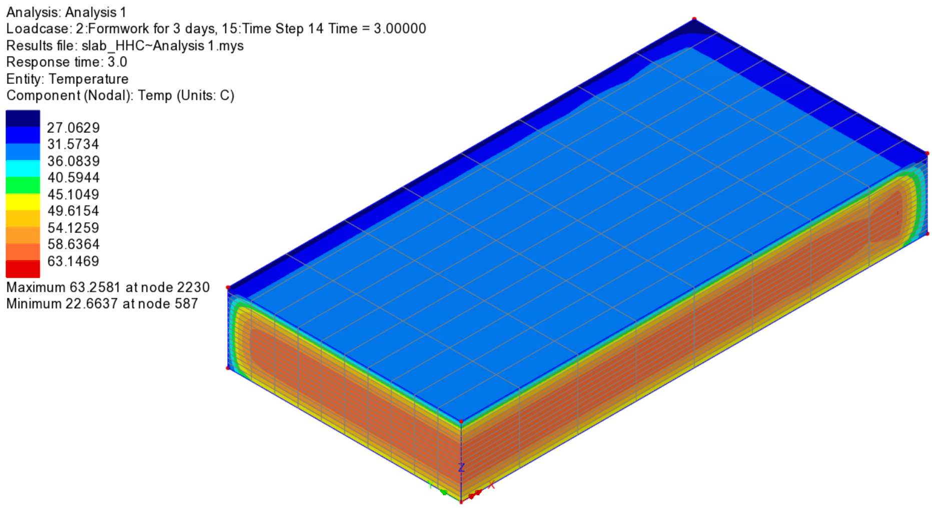
Viewing the Results

Analysis results for each time step are present in the Treeview. The time step result for the last thermal loadcase to be solved (Time = 30.0000) is set active by default.

Plotting contours of temperature after 3 days

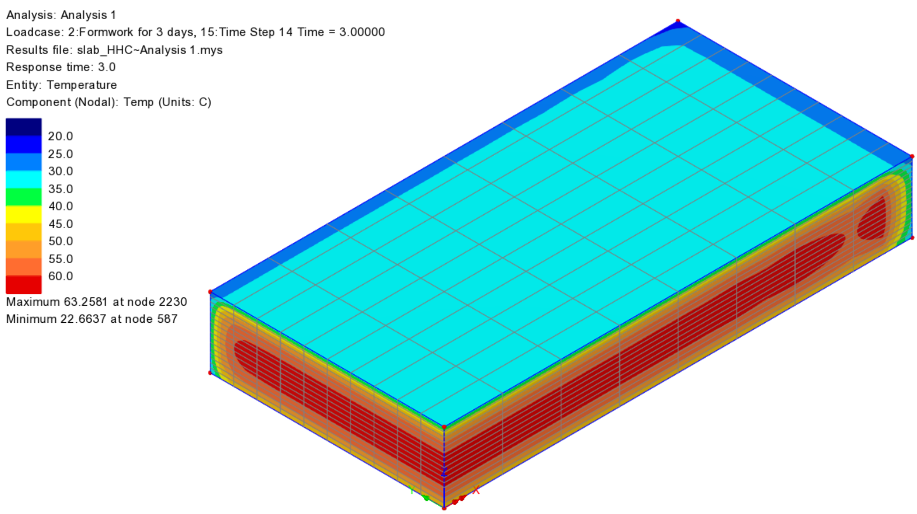
- In the Treeview, in the **Formwork for 3 days** analysis, set active the loadcase **Time Step 14 Time = 3.000**
- With no features selected click the right-hand mouse button in a blank part of the Graphics window and select **Contours....**
- Ensure that Entity **Temperature** and Component **Temp** and **Nodal** results are selected.

A plot showing the temperature distribution at ‘Time = 3 days’ will be displayed.



Change the contour range

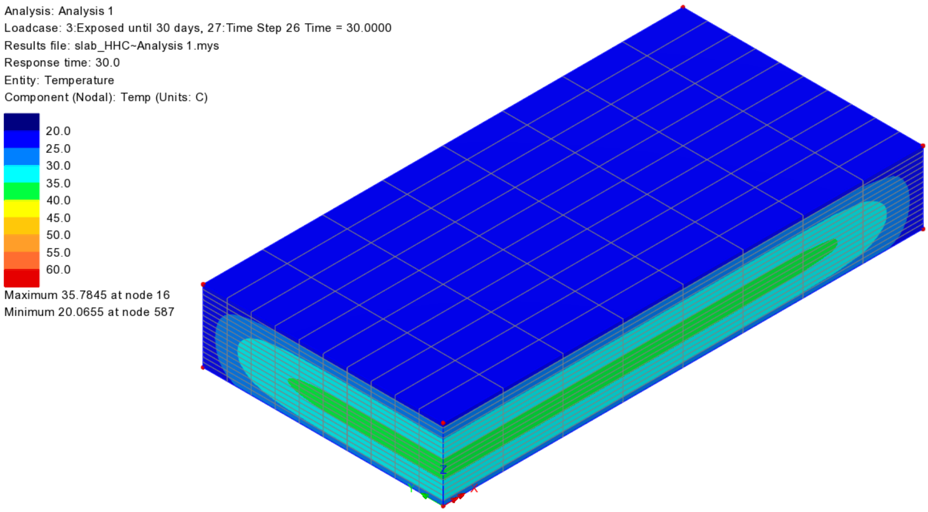
- In the Treeview, double click the **Contours** entry. On the properties dialog, click the **Appearance** tab and choose the **Set** button adjacent to the Classic contour option to set the maximum and minimum contour values to **60** and **20** respectively. Click **OK** to return to parent dialog and **OK** again to finish.



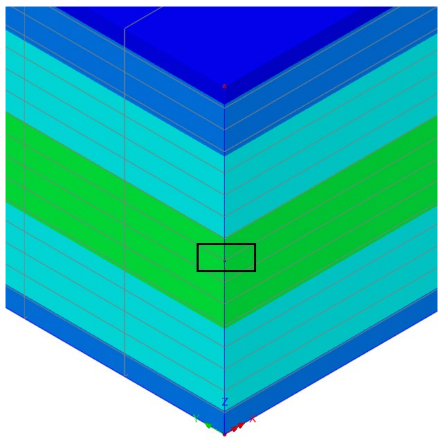
Note. Setting a fixed contour range is useful when wanting to view a consistent set of colours for a range of temperatures, and when animating load increments (not done in this example).

Plotting contours of temperature after 30 days

- In the Treeview, in the **Exposed until 30 days** analysis, set active the loadcase **Time Step 26 Time = 30.000**



Graphing of temperature at a selected node

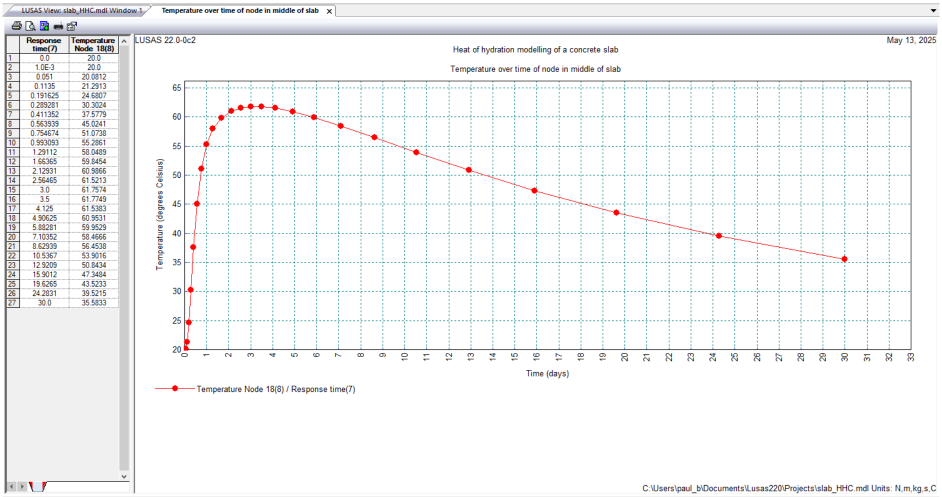


- With reference to the adjacent image, select the node in the middle of the slab as highlighted – the one at half height, at the corner between the two internal faces, at coordinates (0,0,1.5). Note that by selecting a point and then hovering the cursor over it its coordinates may be seen.

Utilities
Graph wizard

- For the X axis, select **Time history** and press **Next**.
- Select **Named** results for the **Whole Analysis** for **Analysis 1** and click **Next.**
- Select **Response time** and click **Next**.
- For the Y-axis, select **Nodal** and click **Next.**

- Select entity **Temperature** and Component **Temp**, for the specified single node and click **Next**.
- Title the graph **Temperature over time of node in middle of slab**. Name the X axis to be **Time (days)** and the Y axis to be **Temperature (degrees Celsius)**. Click **Finish.**



	Response time(7)	Temperature Node 18(8)
1	0.0	20.0
2	1.0E-3	20.0
3	0.051	20.0812
4	0.1135	21.2913
5	0.191625	24.6807
6	0.289281	30.3024
7	0.411352	37.5779
8	0.563939	45.0241
9	0.754674	51.0738
10	0.993093	55.2861
11	1.29112	58.0489
12	1.66365	59.8454
13	2.12931	60.9866
14	2.56465	61.5213
15	3.0	61.7574
16	3.5	61.7749
17	4.125	61.5383
18	4.90625	60.9531
19	5.88281	59.9529
20	7.10352	58.4666
21	8.62939	56.4538
22	10.5367	53.9016
23	12.9209	50.8434
24	15.9012	47.3484
25	19.6265	43.5233
26	24.2831	39.5215
27	30.0	35.5833

From the table data it can be seen that a peak temperature of 61.8 degrees is achieved after 3.5 days. After this time, the temperature is continually cooling but has not returned to ambient within 30 days.

Analysis 2 – Varying ambient temperature

Varying the ambient temperature using loadcurves

The use of a constant ambient temperature, as done in the previous analysis, is not very realistic. In reality the air temperature will vary between day and night, which will affect the slab temperature. A new analysis (Analysis 2) will be created to show a more realistic modelling approach. Analysis 1 will be retained for reference.

Create a new analysis

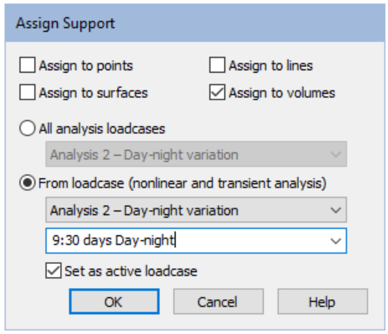
Analyses
Thermal Analysis

- Enter the name to be **Analysis 2 – Day-night variation** and press **OK**.
- Rename the first loadcase in it to be **Initialisation Day-night**

Defining and assigning the supports

With the whole model selected:

- From the treeview drag and drop the support attribute **Free** onto the model ensuring **Assign to volumes** is selected.
- With the 'From loadcase' radio button selected for **Analysis 2 – Day-night variation** ensure that a **New** loadcase called **30 days Day-night** is created and click **OK**.



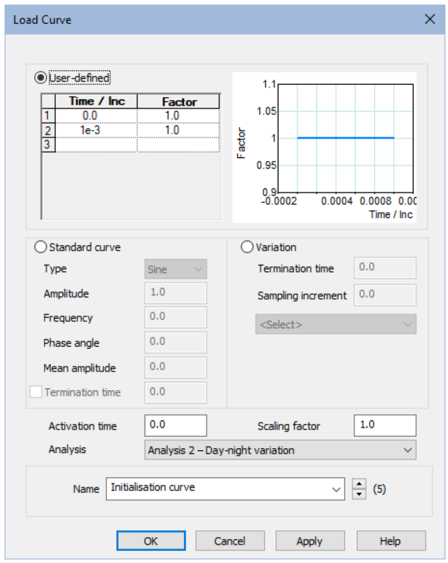
Note. This frees up the temperatures from the prescribed temperature loading which has "locked" the temperatures in the original Initialisation loadcase.

Defining load curves

Load curves can be used to describe the variation of loading with time.

Analyses
Load Curve

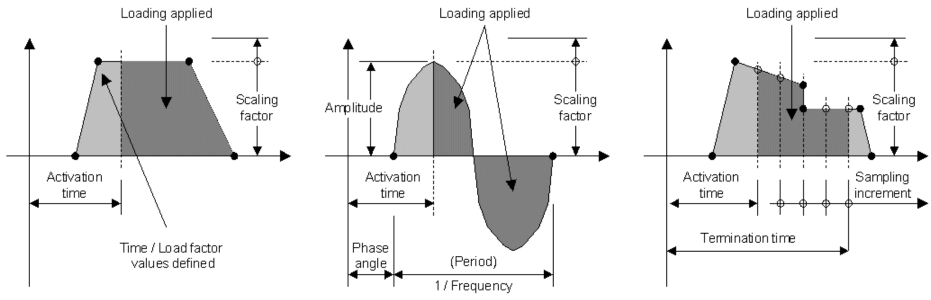
- Create a **User-defined** loadcurve by entering a time of **0** and a factor of **1**, then press the **Tab** key to get a new line in the table. Enter a time of **1e-3** and a factor of **1**. (This will give a constant load factor of 1 for time = 0 to time = 0.001 days). Ensure it is in **Analysis 2**, give it a name **Initialisation curve** and click **Apply.**
- Next, change the time in the second row to be **30**, change the name to **Constant curve** and click **Apply**.
- Now select the **Standard curve** option of type **Cosine**, with Amplitude **10**, Frequency **1** and Mean amplitude **20**.



(This will give a curve with a mean factor of 20, a maximum factor of 20+10 = 30, a minimum factor of 20-10 = 10; varying with a cosine function starting at max amplitude, getting to minimum amplitude at 1/2 = 0.5 days, back up to max amplitude at 1 day and repeating every 1 day.)

- Specify a Termination time of **3** days, give it the name **Day-night curve 3 days** and click **Apply**.
- Lastly, untick the Termination time, change the Activation time to **3** days, enter a name of **Day-night curve 3 to 30 days** and click **OK**.

Loadcurve types explained



User-defined load curve **Standard curve** **Variation-based**

See the LUSAS Modeller help topic for more information.

Defining environmental conditions

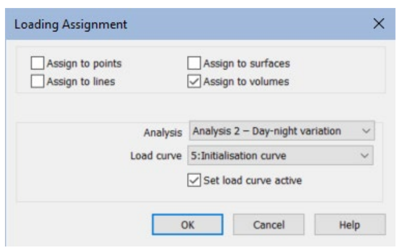
Attributes
Loading...

- Select **Environmental conditions** and click **Next.**
- Uncheck **Temperature dependent**.
- Enter an Environmental temperature of **1** and a Convection heat transfer coefficient of **8.333**. Enter a name of **Unit Face Air Temperature** and **Apply**.
- Leave the Environmental temperature as **1** and change the Convection heat transfer coefficient to be **7**. Enter a name of **Unit Face Formwork Temperature** and click **Finish**.

Note. When using loadcurves, loads are assigned to the loadcurves rather than to the loadcases. The time controls within the loadcases then determine what loads are actually present, based on the load curves.

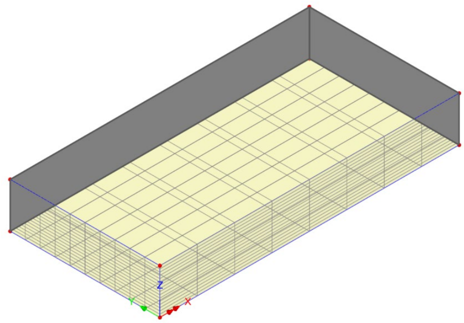
Assigning environmental conditions

With the whole model selected:



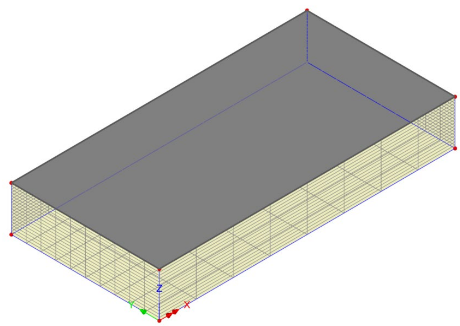
- From the treeview, drag and drop the **Initial Temp** loading onto the volumes selected, for **Analysis 2 – Day-night variation**, assigning it to the load curve named **Initialisation curve**.
- Click in an empty region of the view window to deselect the selection.

Side faces



- Select the two outer vertical faces of the base slab.
- From the treeview, drag and drop the **Unit Face Formwork Temperature** onto the selected surfaces for **Analysis 2 – Day-night variation**. For the Environmental temperature select **Day-night curve 3 days** and click **OK**. This will take the defined temperature (1 degree Celsius) and multiply it by the varying load factor (10 to 30) to give a temperature of 10 to 30 degrees Celsius.
- From the treeview, drag and drop the **Unit Face Air Temperature** onto the selected surfaces for **Analysis 2 – Day-night variation**. For the Environmental temperature select **Day-night curve 3 to 30 days** and click **OK**.

Top face

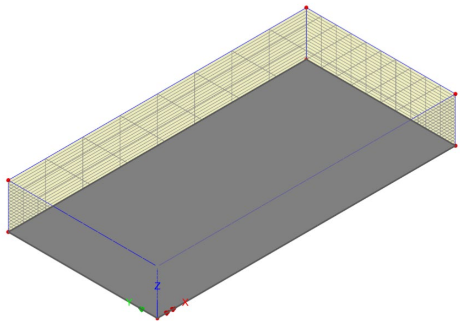


- Select only the top face of the slab
- From the treeview, drag and drop the **Unit Face Air Temperature** onto the selection for **Analysis 2 – Day-night variation**. For the Environmental temperature select **Day-night curve 3 days** and click **OK**.
- From the treeview, drag and drop the **Unit Face Air Temperature** onto the selected surfaces for **Analysis 2 – Day-night variation**. For the Environmental temperature select **Day-night curve 3 to 30 days** and click **OK**.

Bottom of slab

- Select only the bottom face of the slab.
- From the treeview, drag and drop the **Face Ground Temperature** onto the selection. For the Environmental temperature select **Constant curve** and click **OK**.

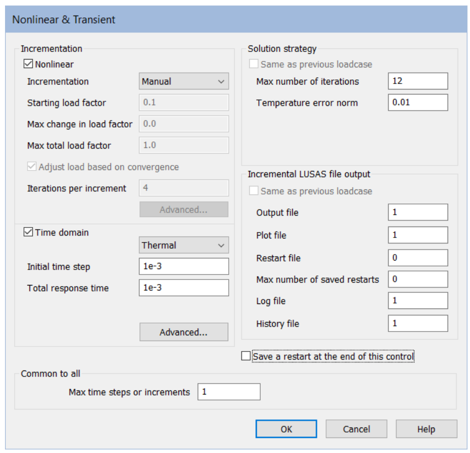
The ground temperature does not vary significantly between day and night, so the coefficients can be left as Constant (no load curve).



Define nonlinear and transient controls

'Initialisation day-night' loadcase

- In the Treeview, right-click on the **Initialisation Day-night** loadcase and select **Controls** then **Nonlinear and transient**.
- In the Incrementation section tick **Nonlinear** to turn it on.
- Select **Time Domain**, enter an initial timestep of **1e-3,** a total response time of **1e-3**, a maximum number of time steps of **1**, and click **OK**.



'30 Days day-night' loadcase

- In the Treeview, right-click on the **30 days Day-night** loadcase and select **Controls** then **Nonlinear and transient**.
- In the Incrementation section tick **Nonlinear** to turn it on.
- Select **Time Domain**, enter an initial timestep of **0.05,** a total response time of **30**

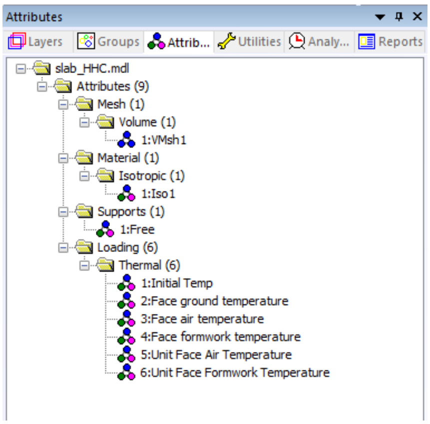
- Click the **Advanced** button and change the Time step increment factor to **1.25** and click **OK**. Then **OK** again to exit the main dialog.

Checking the Attributes and Analyses Treeviews

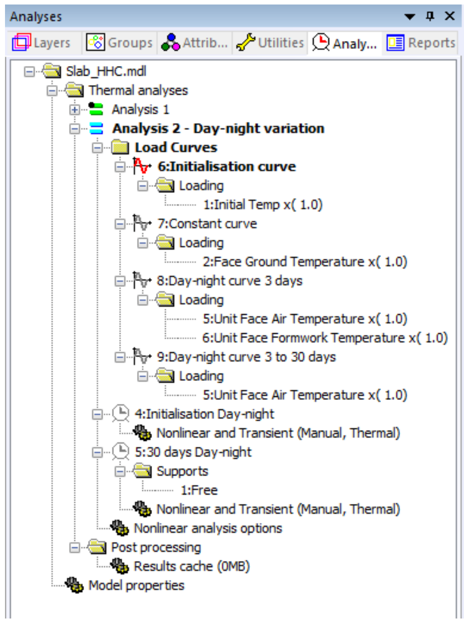
The Attributes Treeview and the Analyses Treeview now contain all the information for an analysis to be carried out.

For a successful analysis your model should contain treeviews similar to those that follow. 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.

Attributes treeview



Analyses treeview



Running the Analysis

With the model loaded:

Open the **Solve Now** dialog.

- Ensure that **Analysis 2** is selected and press **OK** to run the analysis.

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

Associated files will be created in the LUSASFiles\<model_name> folder.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered is written to an output file 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:

- **slab_HHC_loadcurves.lvb** carries out the modelling the whole example ready for a 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 **slab_HHC**
- Select units of **N,m,kg,s,C**
- Ensure the **Thermal** user interface is selected and click the **OK** button

File
Script >
Run Script...

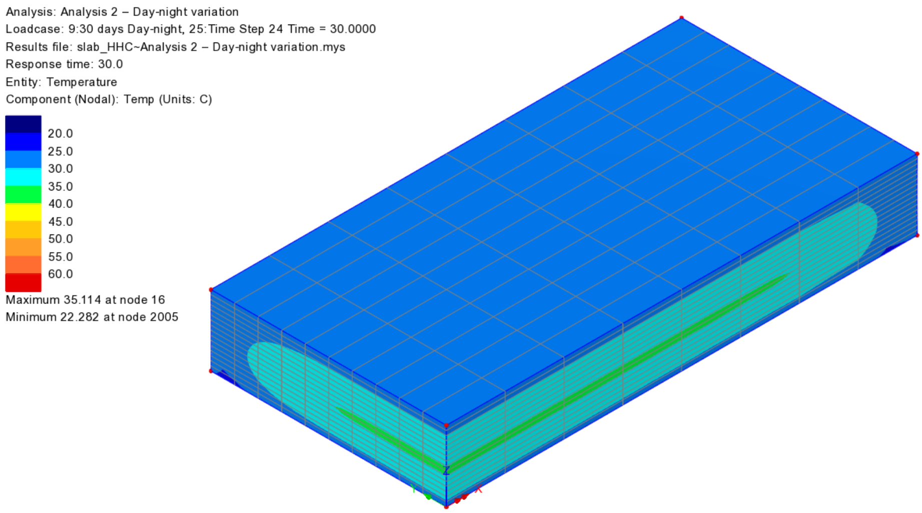
- To recreate the model, open the file **slab_HHC_loadcurves.lvb** that was downloaded and placed in a folder of your choosing.

Run the analysis to generate the results.

Viewing the Results

Analysis results for each time step are present in the Treeview.

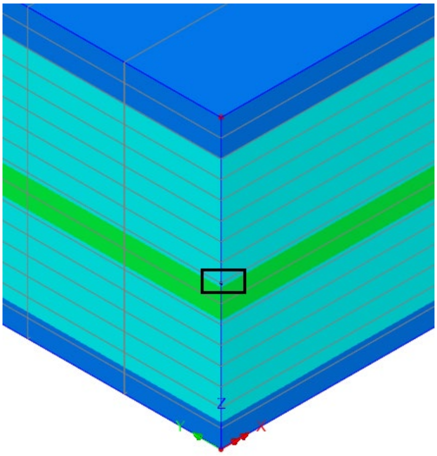
- In the Treeview, in the **Analysis 2 – Day-night variation** analysis, for the **30 days Day-night** loadcase, set active the loadcase **Time Step 24 Time = 30.000**



A reduced maximum temperature can be seen. Results can be compared against the previous graphed results in a similar manner to that done previously.

Graphing of temperature at a selected node

A graph is required for the same node as selected previously.

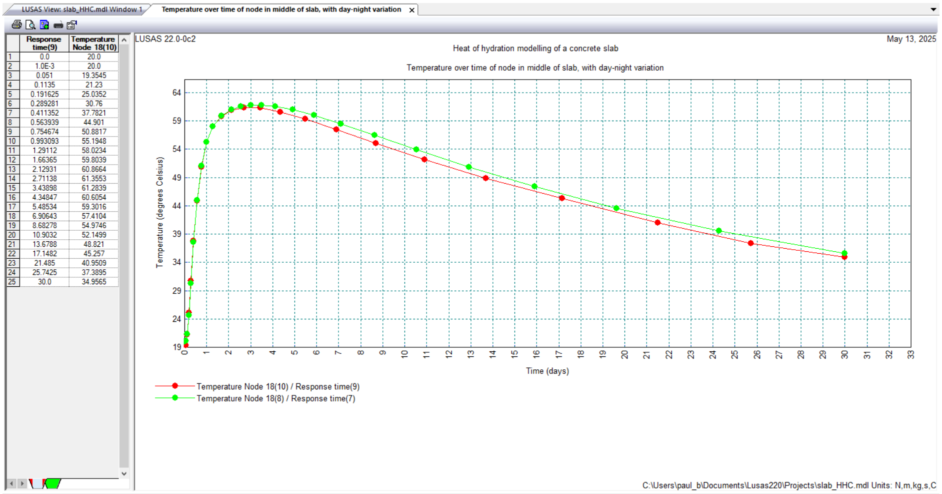


- If not still selected, select the node in the middle of the slab (i.e. at half height, at the corner between the two internal faces, at coordinates (0,0,1.5).

Utilities
Graph wizard

- For the X axis, select **Time history** and press **Next**.
- Select **Named** results for the **Whole Analysis** for **Analysis 2 – day-night variation** and click **Next.**
- Select **Response time** and click **Next**.
- For the Y-axis, select **Nodal** and click **Next.**
- Select entity **Temperature** and Component **Temp**, for the specified single node and click **Next**.
- Title the graph **Temperature over time of node in middle of slab, with day-night variation**. Name the X axis **Time (days)** and the Y axis **Temperature (degrees**

Celsius). Tick to include the previous graph (**Graph 1**) for comparison. Click **Finish.**



On the graph the red line now shows results for Analysis 2, the green line shows the previously obtained results for Analysis 1.

By selecting the red data tab (which is to the left of the green tab on the graph above), the more realistic modelling of the ambient temperature can be seen to provide slightly lower concrete temperatures from 1 day onwards, including a slightly lower peak temperature of 61.4 degrees after 2.7 days.

This completes the example.