Thermal Analysis of a Pipe

For LUSAS version:	16.0
For software product(s):	Any Plus version
With product option(s):	Thermal, Dynamics (for the second stage of the example)

Description

This example provides an introduction to performing a thermal analysis with LUSAS.

A continuous steel pipe is exposed to an atmospheric temperature of 25°C. Oil at 150°C is to be pumped through the pipe. The pipe has a thermal conductivity of 60 J s^{-1} m^{-1} $^{\circ}C^{-1}$, a specific heat capacity of 482 J kg^{-1} $^{\circ}C^{-1}$ and a density of 7800 kg/m^3. Two analyses will be carried out. A steady state analysis is required to determine the maximum temperature of the outer surface of the pipe and a transient thermal analysis is then performed to find out the time it will take the surface to reach this temperature once pumping begins.

0.1m   0.05m

The units of the analysis are N, m, kg, s, C throughout.
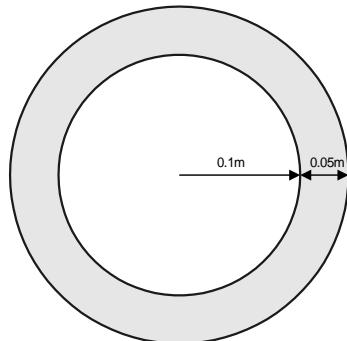
Note. There are three transport mechanisms for heat energy; conduction, convection and radiation. The first of these is defined as a material parameter, the others are defined within the load attributes as environmental variable and environmental temperatures. In this example, the effects of radiation are ignored.

Objectives

The objectives of the analysis are:

❑ To determine the maximum temperature the outer surface of the pipe reaches during continual pumping.

❑ To determine how long it will take for the maximum temperature to be reached once pumping of the oil begins.

Keywords

2D, Inplane, Thermal, Steady State, Transient, Environmental Temperature, Prescribed Temperature.

Associated Files

❑ **pipe_modelling.vbs** carries out the modelling of the example.

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.

Creating a new model

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 **pipe**

• Use the default user-defined working folder.

• Change the Analysis Type to **Thermal**

• Select an Analysis Category of **2D Inplane**

• Set the model units as **N,m,kg,s,C**

• Leave the Timescale units as **Seconds**

• No startup template or Loading grid are to be used.

• Enter the title as **Steady State Thermal Analysis of Pipe** and 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.

Feature Geometry

The pipe geometry will be generated by defining a vertical line that will be swept into a quarter segment. This segment will then be copied to generate the full model.

Geometry
| Line | > |
| By Coords... | |
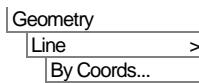
Enter coordinates of (**0, 0.1**) and (**0, 0.15**) to define a vertical line and click the **OK** button.

- Select the line

Geometry
| Surface | > |
| By Sweeping... | |
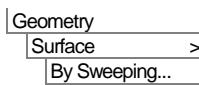
On the Sweep dialog, choose the **Rotate** option and enter a rotation angle of **90** about the **Z-axis.**

- Leave the other options and click the **OK** button.

- Select the Surface

Geometry
| Surface | > |
| Copy... | |
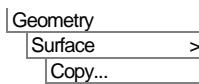
In the Copy dialog, select the **Rotate** option.

- Enter an Angle of **90** about the **Z-axis**

- Enter the number of copies as **3**

- Click **OK** to create the full pipe cross section.
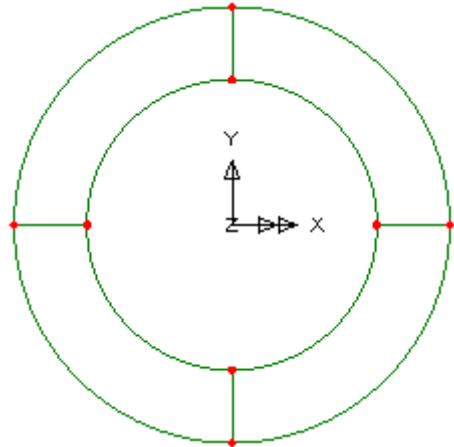
Meshing

Plane field elements are to be used for this analysis. These elements are used to model the cross section of 'infinite' components since they only model heat flow in the XY plane.

Attributes	
Mesh	>
Surface..	

- Select **Plane field, Quadrilateral** shaped**, Linear** elements.

- Deselect **Allow transition pattern**

- Enter the attribute name as **2D Thermal Mesh** and click the **OK** button.
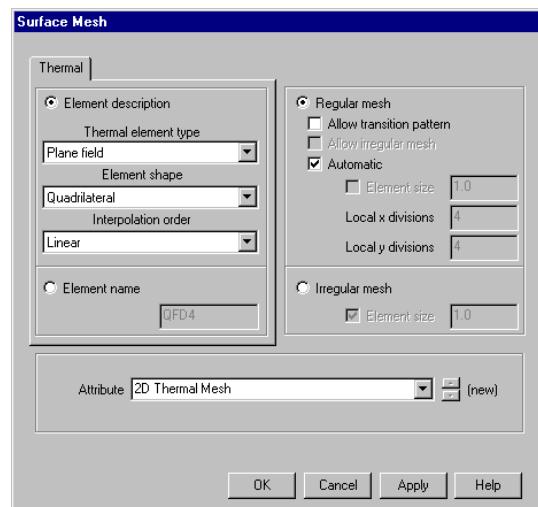
- Use **Ctrl + A** to select all the features.

- Drag and drop the mesh attribute **2D Thermal Mesh** from the 🔵 Treeview onto the selection to assign the mesh to the selected surfaces.
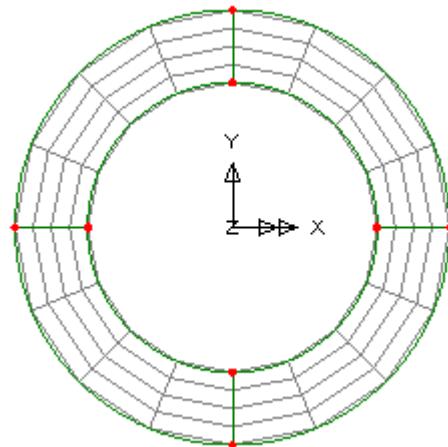
Geometric Properties

Geometric properties are used to define the thickness of the pipe. Since the pipe is of infinite length a unit length is modelled.

Attributes	
Geometric	>
Surface...	

- Enter a thickness of **1** and leave the eccentricity blank.

- Enter the attribute name as **Thickness** and click the **OK** button.

- Select all the surfaces and assign the attribute **Thickness**

⚡ Select the fleshing on/off button to turn-off geometric property visualisation.

Material Properties

Within LUSAS the specific heat is defined as a massless quantity. In order to calculate this quantity, the standard specific heat capacity for a material is multiplied by the density. The result is a material parameter in the correct massless units, in this case $J\ m^{-3}\ C^{-1}$.

The materials in this example have properties of steel.

- On the Isotropic dialog enter the thermal conductivity as **60**

- Enter the specific heat as **3.7596E6**

- Enter the attribute name as **Steel (J,m,C)** and click the **OK** button.

- Assign the material attribute **Steel (J,m,C)** to the surfaces.

Boundary Conditions

Unsupported nodes in thermal analyses are assumed to be perfect insulators. The environmental conditions are defined using environmental loading. This loading defines the amount of convection to the environment that occurs.

- On the first dialog, select the **Environment Conditions** option and click **Next**

- On the Environmental Conditions dialog enter the environmental temperature as **25**

- Enter the convective heat transfer coefficient as **500**

- Since radiation is to be ignored set the radiation heat transfer coefficient to **0**

Environmental Temperature

	Environmental temperature	Convection heat transfer coefficient	Radiation heat transfer coefficient	Reference temperature
1	25	500	0	

Attribute Environmental Temperature   [new]

< Back   Next >   Finish   Cancel   Apply   Help

- Enter the attribute name as **Environmental Temperature** and click the **Finish** button.
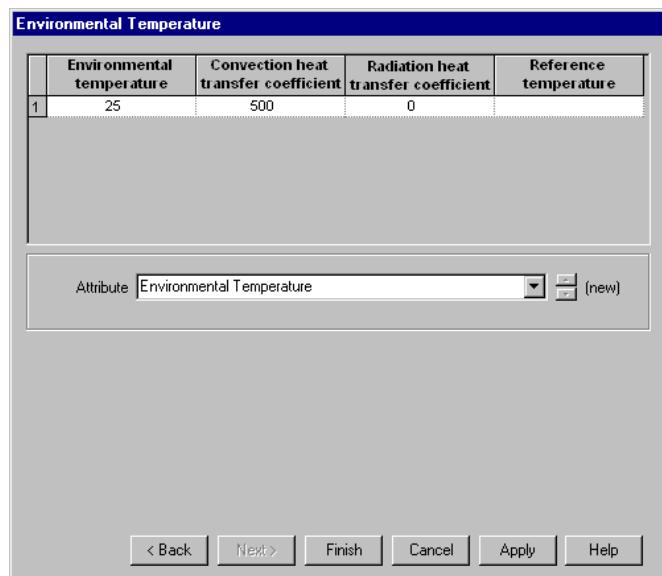
- Select the 4 lines defining the outside of the pipe and assign the attribute **Environmental Temperature** to the these lines Ensure the option to **Assign to lines** is selected. Ensure **Analysis 1** and **Loadcase 1** are selected. Click **OK** to finish the assignment.

Attributes
Loading…

The pipe is heated by the oil passing along inside the pipe. This is modelled using a prescribed heat input assigned to the lines defining the inner surface of the pipe.

- On the first dialog, select the **Prescribed** option and click **Next**

- On the Prescribed Temperature dialog enter a **Fixed** temperature of **150**

- Ensure that the **Total** prescribed loading option is selected.

- Enter an attribute name of **Oil Temperature** and click the **Finish** button.

- Select the 4 lines defining the inner surface of the pipe and assign the attribute **Oil Temperature** to the these lines. Ensure the option to **Assign to lines** is selected for **Analysis 1** and **Loadcase 1**. Click **OK** to finish the assignment

Saving the model

File
Save

Save the model file.

Running the Analysis

File
LUSAS Datafile...

Open the **Solve Now** dialog and press **OK** to run the analysis.

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

❑ **Pipe.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.

❑ **Pipe.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered can be 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 a file is provided to enable you to re-create the model from scratch and run an analysis successfully.

❑ **Pipe_modelling.vbs** carries out the modelling of the example.
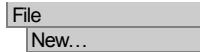
File
New…

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 **pipe**

- Change the user interface to **Thermal**

File
Script >
Run Script...
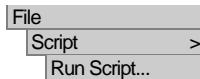
To recreate the model, select the file **pipe_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results

Viewing the Results

Analysis loadcase results are present in the Treeview, and the results for loadcase 1 will be set active by default.

- Turn off the **Geometry** layer in the Treeview.

- If necessary any visualised thermal loadings can be removed by deselecting **Visualise Assignments** from both thermal loading datasets in the Treeview.

Temperature Contours

With no features selected, click the right-hand mouse button in a blank part of the Graphics window and select the **Contours** option to add the **Contours** layer to the ▣ Treeview.

Loadcase: "1:Loadcase 1"
Results file: pipe.mys
Entity: Potential
Component: PHI

110.614
115.223
119.832
124.441
129.05
133.658
138.267
142.876
147.485
Maximum 150.0 at node 1
Minimum 108.52 at node 26

The contour plot properties will be displayed.

- Select **Potential** contour results component **PHI**

- Click the **OK** button.
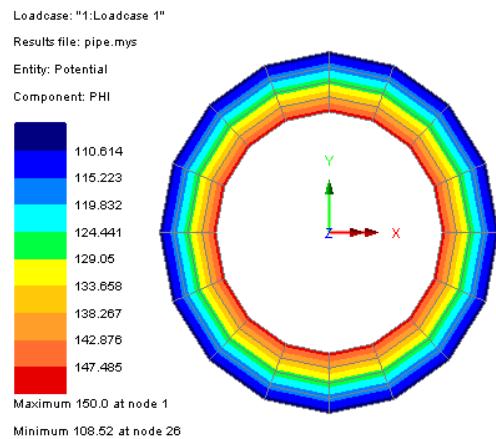
From the analysis it can be seen that the maximum temperature the outer surface of the pipe reaches is 108.5 $^{\circ}$C.

Note. If Plane field elements with quadratic interpolation order were to be used instead of elements with linear interpolation order the true shape of the pipe would be seen – as shown on the image to the right.

Loadcase: "1:Loadcase 1"
Results file: pipe.mys
Entity: Potential
Component: PHI

112.429
117.114
121.798
126.483
131.167
135.852
140.536
145.221
149.905
Maximum 150.0 at node 1
Minimum 107.839 at node 17

The difference between the results, however, is negligible for this example.
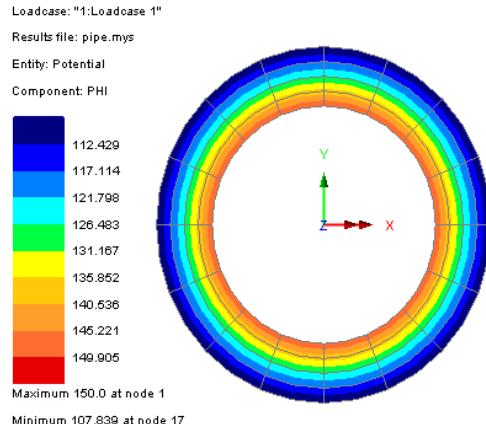
This completes the steady state part of the example.

Transient Thermal Analysis

For LUSAS version:	16.0
For software product(s):	Any Plus version
With product option(s):	Thermal, Dynamics.

This part of the example extends the previously defined pipe model used for the steady state analysis. A file is supplied that can be used to recreate the model if required.

- If you are continuing from the first part of the example you may save your model file as **pipe_transient.mdl** and continue from the heading 'Setting up the Starting Conditions'.

Otherwise you can use one of the supplied scripts to re-create a suitable model as follows:

Creating a new model (if required)
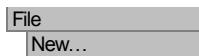
File
  New…

⬜ 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.
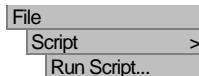
- Enter the file name as **pipe_transient**

File
  Script   >
    Run Script...

- To create the model, import the file **pipe_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Setting up the Starting Conditions

The first step of a transient analysis is used to establish the steady state conditions before any heat is input. In this example this means the oil temperature needs to be removed from the first loadcase to allow the pipe to reach the environmental temperature before the oil temperature is introduced and the transient analysis begins.

- If present, turn off the Contours layer in the 🔲 Treeview

- Ensure the **Geometry**, **Mesh** and **Attributes** layers are present and turned on in the 🔲 Treeview

- In the 🌐 Treeview click the right-hand mouse button on the **Oil Temperature** attribute and choose the **Deassign> From all** option.

Now we define how the transient analysis should take place:

- Expand **Analysis 1** then, using the right-hand mouse button, click on **Loadcase 1** in the 🕐 Treeview and from the **Controls** menu select the **Nonlinear & Transient** entry.
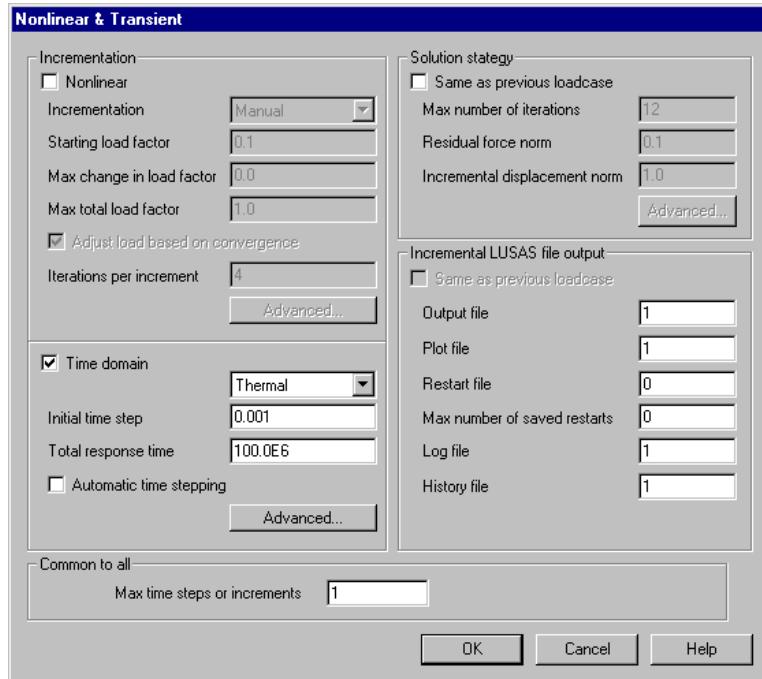
- Select the **Time domain** option.

- Ensure **Thermal** is selected in the drop down list.

- Enter an **Initial time step** of **0.001**

- Leave the Total response time set to **100E6**

- In the Common to all section, enter the **Max time steps or increments** as **1**

- Click the **OK** button.

Setting up the Transient Analysis

Once the starting conditions have been established the transient analysis can begin. The heat input is assigned to the inner surface of the pipe and the time stepping regime is defined.

Firstly reapply the environment temperature, which represents the atmospheric temperature.

- In the 🔩 Treeview click on the attribute **Environmental Temperature** with the right-hand mouse button and choose the **Select Assignments** option. This will select the lines that are currently using this attribute.

- Drag and drop the attribute **Environmental Temperature** onto the graphics window to assign the attribute to **Analysis 1** and edit the loadcase name in the drop down list to be **Loadcase 2** by overtyping. Click the **OK** button. This will introduce Loadcase 2 into the 🕐 Treeview.

Now apply the Oil Temperature.

- Select the four lines defining the inner surface of the pipe.

- Assign the dataset **Oil Temperature** to these Lines selecting **Loadcase 2**. Click **OK** to finish the assignment.

- Using the right-hand mouse button click on **Loadcase 2** in the 🕐 Treeview and from the **Controls** menu select the **Nonlinear & Transient** entry.
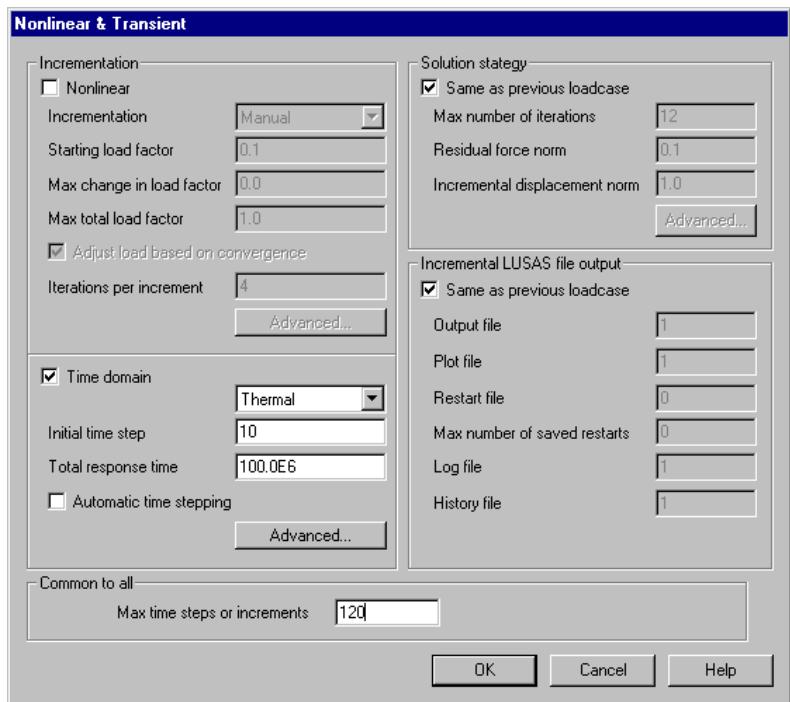
- Select the **Time domain** option.

- Ensure **Thermal** is selected in the drop down list.

- Enter an **Initial time step** of **10**

- Leave the Total response time set to **100E6**

- In the Common to all section enter the **Max time steps or increments** as **120**

- Click the **OK** button.

Saving the model

🖫 Save the model file.

Running the Analysis

▤ Open the **Solve Now** dialog and press **OK** to re-run the analysis.

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

Viewing the Results

The loadcase results can be seen in the 🕒 Treeview. For a nonlinear analysis the last solved loadcase increment will be set active by default.

To establish the time taken to reach the steady state condition a graph of external temperature verse response time is to be generated.

- First, select the node on the outside of the pipe as shown.

- Choose the **Time history** option and click on the **Next** button

Firstly we define the X axis data.

- Select the **Named** option and click the **Next** button.

- Choose **Response Time** from the drop down list and click the **Next** button.

Then we define the Y axis data

- Select the **Nodal** option and click the **Next**

Select this Node
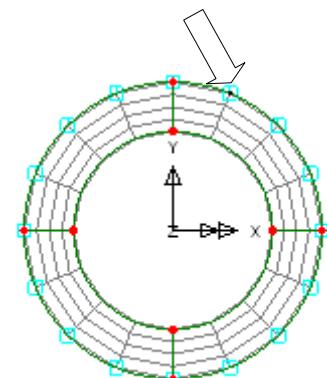
button.

- Select Entity **Potential** component **PHI**

- Select **Specified single node** from the Extent drop down list and the selected node number will appear in the Selected Node drop down list.

- Click the **Next** button.

It is not necessary to input the graph titles at this stage. They can always be modified later.

- Ensure **Use logarithmic scale** is not selected.

- Deselect the **Show symbols** option

- Click on the **Finish** button to display the graph showing the variation of temperature on the outer surface of the pipe with time.
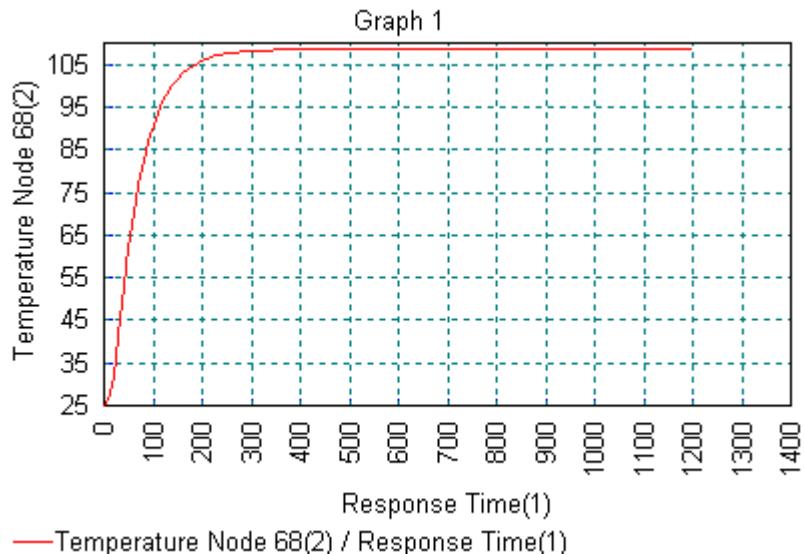
Graph 1

From the graph it can be seen that the outside of the pipe reaches its steady state condition after approximately 300 seconds.

This completes the transient thermal example.