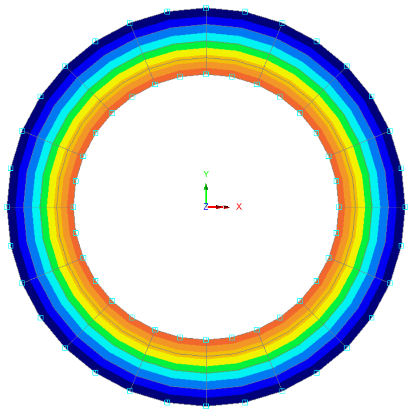
Thermal Analysis of a Pipe

For LUSAS version:	22.0
For software product(s):	Any Plus version
With product option(s):	Thermal, Dynamics (for the second stage of the example)

Description



This example introduces carrying out thermal analysis with LUSAS.

A 0.05m thick steel pipe of 0.2m internal diameter is exposed to an atmospheric temperature of 25ºC. Oil at 150ºC is to be pumped through the pipe. The pipe has a thermal conductivity of 60 J s^{-1} m^{-1} $^{o}C^{-1}$, a specific heat capacity of 482 J kg^{-1} $^{o}C^{-1}$ and a density of 7800 kg/m^3.

Two analyses are to be carried out.

- A steady state analysis is used to determine the maximum temperature of the outer surface of the pipe.
- A transient thermal analysis is also performed to find out the time it will take the surface to reach the maximum temperature once pumping begins.

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

Associated files can be downloaded from the user area of the LUSAS website.

- **pipe_modelling.lvb** carries out the modelling of the example.

Part 1: Modelling for a Steady State Analysis

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.

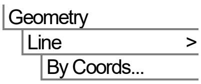
- Enter the file name as **pipe**
- Use the default user-defined working folder.
- Change the Analysis Type to **Thermal**
- Select an Analysis Category of **2D Inplane**
- Set the model units as **N,m,kg,s,C**
- Leave the Timescale units as **Seconds**
- No startup template or Loading grid is to be used.

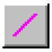
- Enter the title as **Steady State Thermal Analysis of Pipe** and click the **OK** button.

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

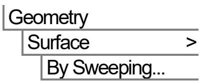
Feature Geometry

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.



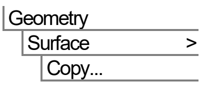
 Enter coordinates of (**0, 0.1**) and (**0, 0.15**) to define a vertical line and click the **OK** button.

- Select the line



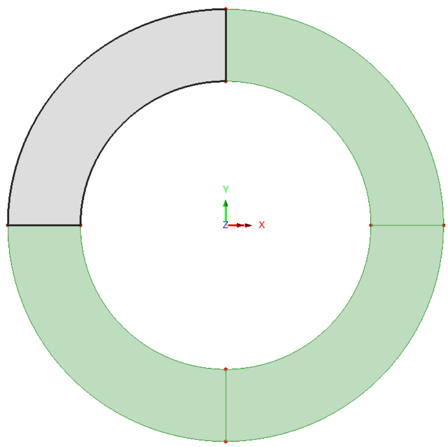
 On the Sweep dialog, choose the **Rotate** option and enter a rotation angle of **90** about the **Z-axis.**

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



 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.

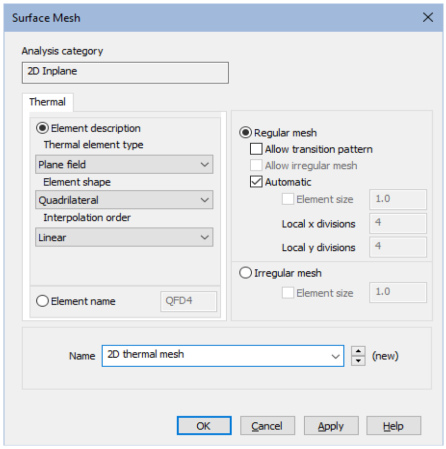


Meshing

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.

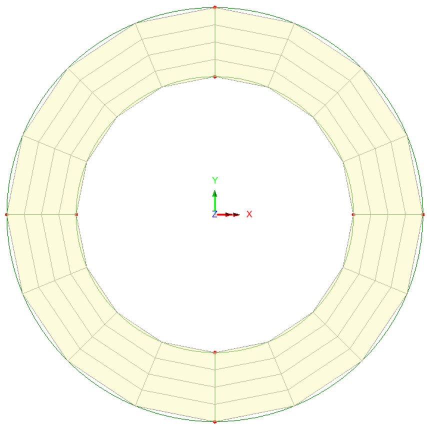
Attributes
Mesh >
Surface...

- Select **Plane field**, **Quadrilateral** shaped, **Linear** elements.
- <u>Deselect</u> **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 selected surfaces.

Note that using elements with linear interpolation order does not model the true shape of the pipe, but this will be shown to have negligible effect on the results obtained later in the example.



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

The materials in this example have properties of steel.

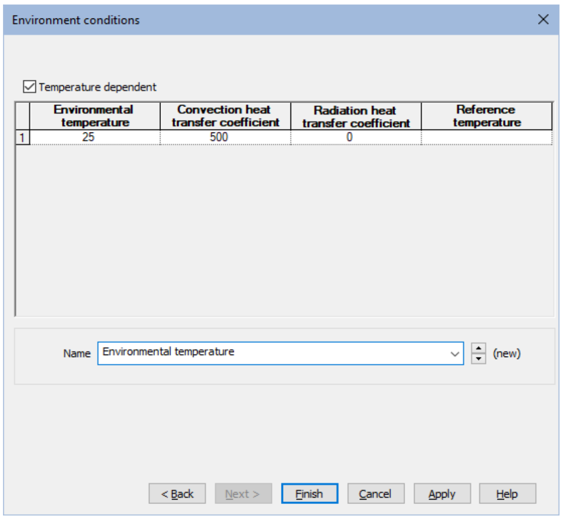
Attributes
Material >
Isotropic…

- On the Isotropic dialog enter the thermal conductivity as **60**
- Enter the specific heat capacity as **482**
- Enter the density as **7800**
- Enter the attribute name as **Steel** and click the **OK** button.
- Assign the material attribute **Steel** 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.

Attributes
Loading…



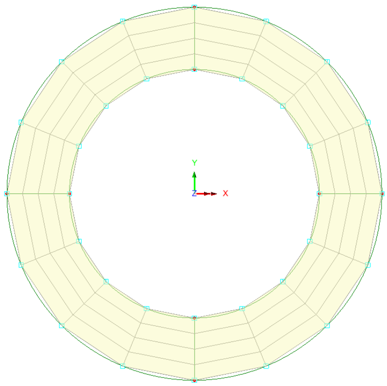
- On the first dialog, select the **Environment Conditions** option and click **Next**
- On the Environmental Conditions dialog enter the environmental temperature as **25**
- Enter the convective heat transfer coefficient as **500**
- Since radiation is to be ignored, set the radiation heat transfer coefficient to **0**
- 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 these lines. Ensure the option to **Assign to lines** is selected for **Analysis 1** and **Loadcase 1**. Click **OK** to finish the assignment.

The thermal loading assignments are visualised using circles at nodes.

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 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, these files will be created in the LUSASFiles\<model_name> folder:

- ❑ **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. You may download this file from the user area of the LUSAS website.

- **Pipe_modelling.lvb** 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 analysis type to **Thermal**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **2D Inplane** is set.
- Click the **OK** button

Note. There is no need to enter any other new model details when a script is run to build a model, since the contents of the script will overwrite any other settings made.

File
Script >
Run Script...

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

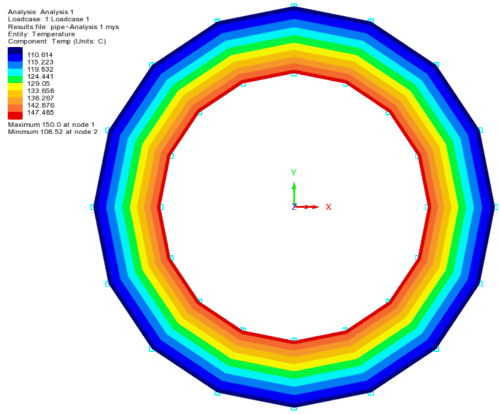
Solve the model to generate the results.

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 View window and select the **Contours** option to add the **Contours** layer to the Treeview.

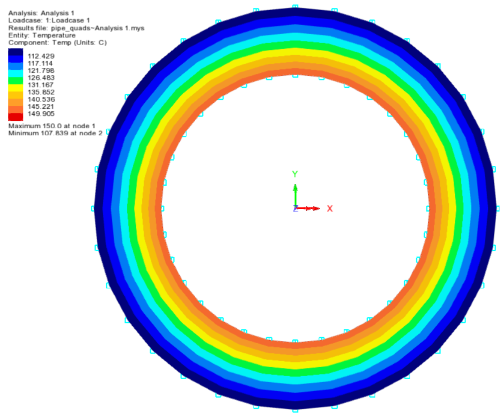
The contour plot properties will be displayed.

- Select **Temperature** contour results of component **emp** and display **Nodal** results
- 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 ^{o}C.

This completes the steady state part of the example.

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.



From an analysis using elements with a quadratic order, a maximum temperature of the outer surface is 107.8 ^{o}C would be obtained.

The difference between the linear results and these, therefore, is negligible for this example.

Part 2: Transient Thermal Analysis

For LUSAS version:	22.0
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 associated scripts to re-create a suitable model as follows:

Creating a new model (only if required)

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. You may download this file from the user area of the LUSAS website.

- ❑ **Pipe_modelling.lvb** This 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_transient**
- Change the analysis type to **Thermal**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **2D Inplane** is set.
- Click the **OK** button

Note. There is no need to enter any other new model details when a script is run to build a model, since the contents of the script will overwrite any other settings made.

File
Script >
Run Script...

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

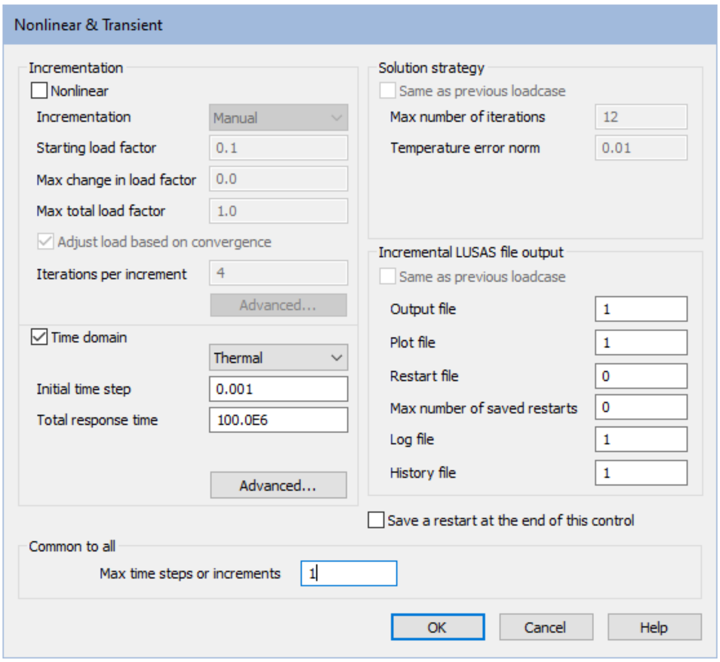
Setting up the Starting Conditions

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:

- In the Treeview, expand **Analysis 1** then, using the right-hand mouse button, click on **Loadcase 1** and from the **Controls** menu select the **Nonlinear & Transient** entry.



- Select the **Time domain** option.
- Ensure **Thermal** is selected in the drop 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

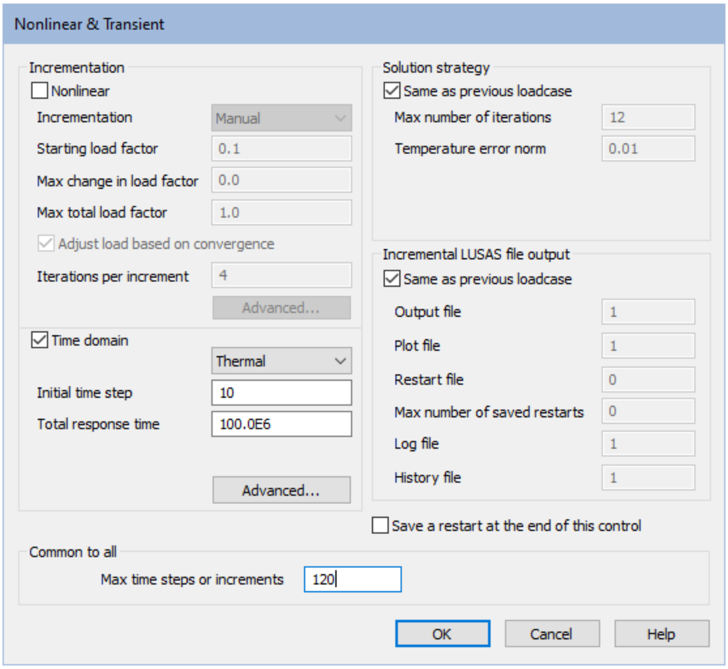
Once the starting conditions have been established the transient analysis can begin. The heat input is assigned to the inner surface of the pipe and the time stepping regime is defined.

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 view 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.
- In the Treeview, using the right-hand mouse button, click on **Loadcase 2** 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** to be **120**
- Click the **OK** button.

Saving the model

File
Save

Save the model file.

Running the Analysis

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

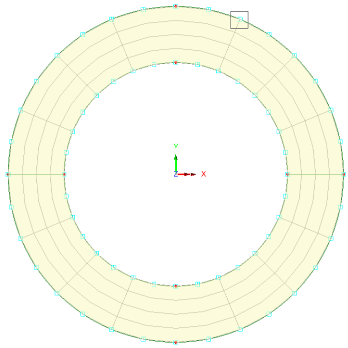
A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

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 top-right outside of the pipe as shown.



Utilities
Graph Wizard...

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

Firstly, we define the X axis data.

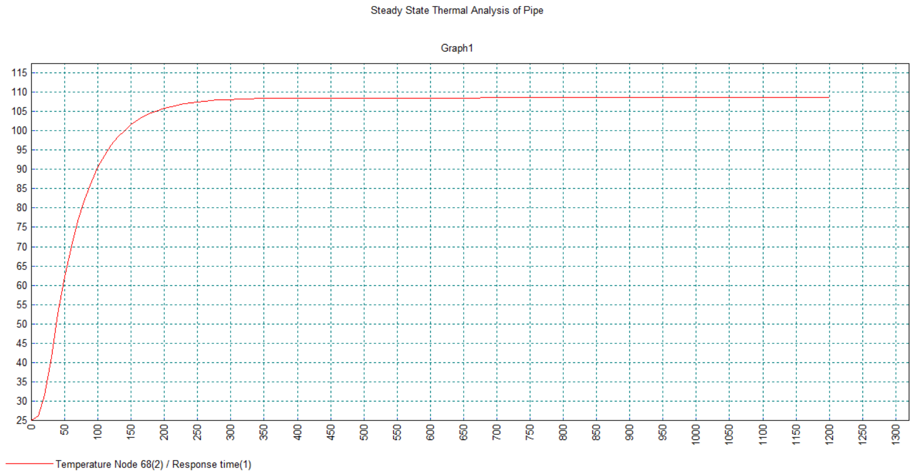
- Select the **Named** option and click the **Next** button.
- Choose **Response Time** from the drop-down list and click the **Next** button.

Then we define the Y axis data.

- Select the **Nodal** option and click the **Next** button.
- Select Entity **Temperature** component **Temp**
- Because only one node has been selected, **Specified single node** will be automatically present in the Extent drop down list. 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 <u>not</u> selected.
- <u>Deselect</u> 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.



From the graph it can be seen that the outside of the pipe reaches its steady state condition after approximately 350 seconds.

This completes the transient thermal example.