Maximus®

Quick start guide

Maximus®

REVISION NUMBER	7.6 - 2
DATE	20 August 2024

A Yokogawa Company

KBC ADVANCED TECHNOLOGIES LIMITED, HEAD OFFICE AND REGISTERED OFFICE

42-50 Hersham Road
Walton on Thames
Surrey KT12 1RZ
UK

T: +44 (0) 1932 242 424
F: +44 (0) 1932 224 214
E: info@kbc.global
www.kbc.global

Registered in England and Wales. Registered number 01357958

Bringing DECARBONIZATION to Life™

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

1.0 Introduction

This quick start guide is intended to give new users the ability to be able to use some of the basic functions within Maximus®. No prior knowledge of Maximus is required.
This guide will demonstrate how to set up a simple steady-state network, import a Multiflash® fluid, run the model and extract results.

The general workflow for building a model in Maximus is described below in Figure **1**:

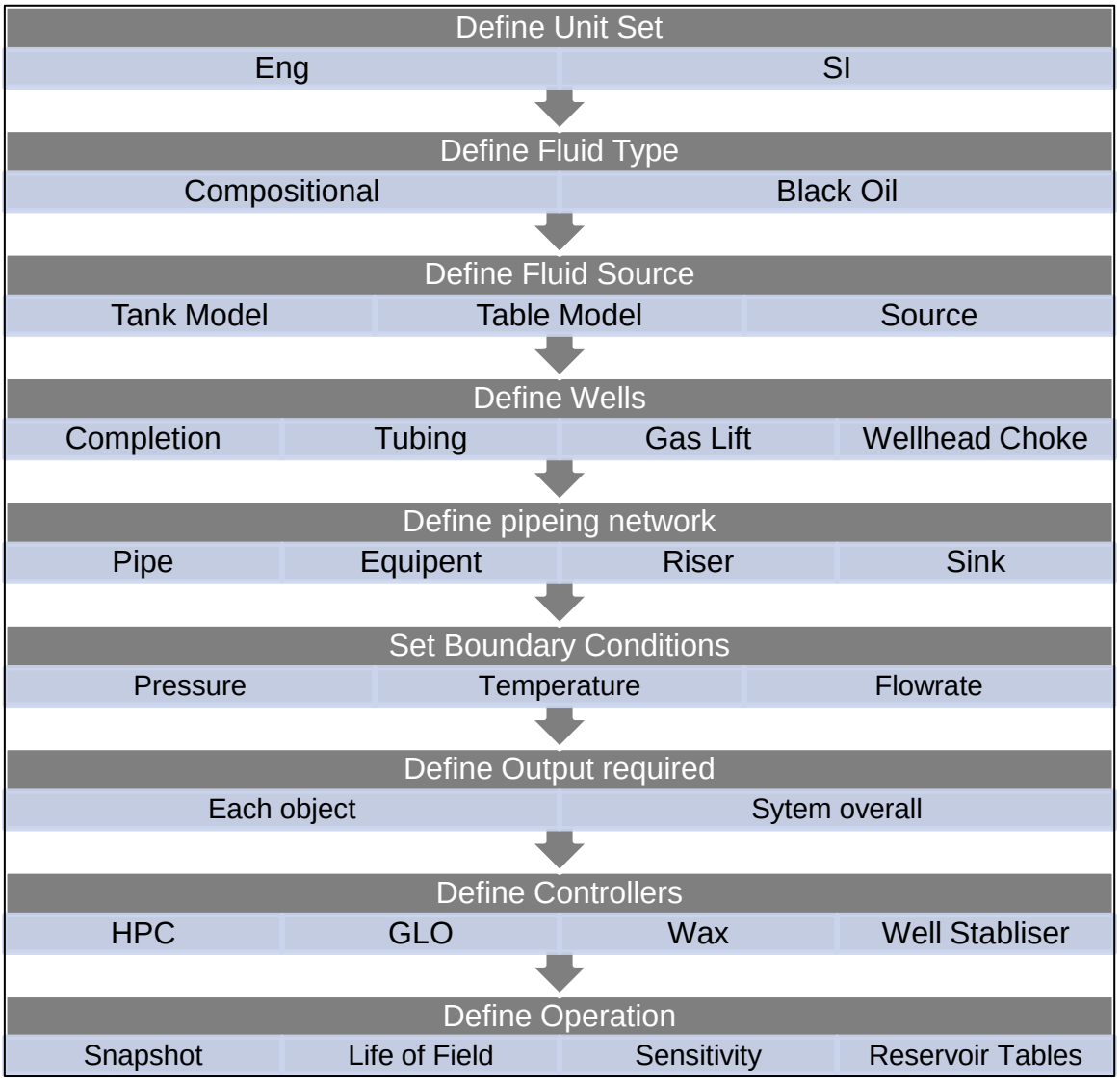

Figure 1: General Workflow for Maximus

2.0 The Maximus Interface

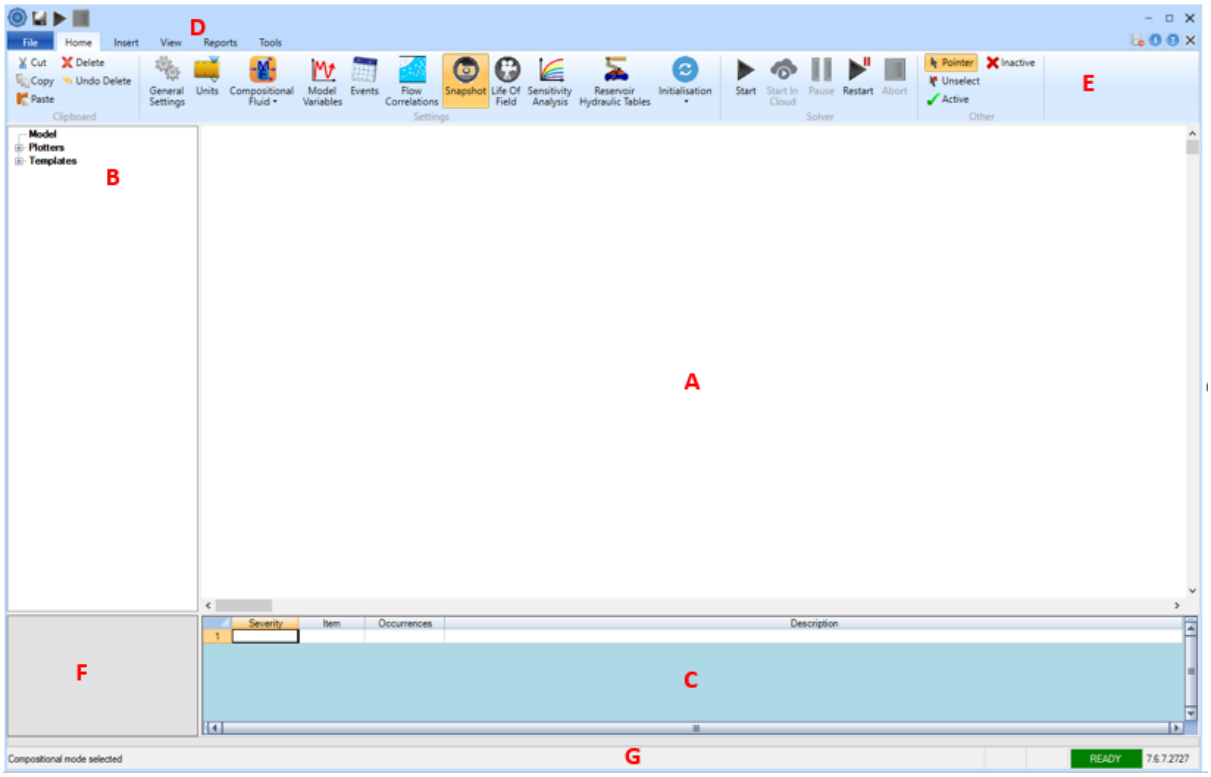

Figure 2: Maximus main GUI

When Maximus is opened, the window shown in Figure 1 will be visible. This is the new default view (Ribbon) for version 7.6. The main areas in the window are as follows:

A. This is the model canvas where the pipe network is constructed using objects from the objects bar.
B. This is the model tree which lists objects and functions within the model.
C. This is the dynamic solution messages panel where messages and errors are displayed in real time during simulation runs.
D. This is the object toolbar where the tabs can be selected for
    a. File
    b. Home
    c. Insert (objects)
    d. View
    e. Reports
    f. Tools
    g. Information (about the current version) and Help button
E. This is the ribbon that will change according to the tab selected simulation toolbar where the following options are available:
    a. Access other Maximus windows
    b. Control the model including start, pause, restart, and stop simulation.
F. This is the mini map where the whole network is shown to the user and where the model canvas is zoomed into. This can be used to navigate the model canvas easily by dragging the black rectangle.

G. This is the status bar showing the current statuses of the model and the progress of any simulation runs.

The object and simulation toolbars can be configured to have a ribbon bar view (default) or the original Classis view instead of the above. To change to the classic view, use the **Switch to Classis** option under View and the view will change to:

Figure 3: Maximus classic view

To switch from Classic to Ribbon view then, under **Setup>General** select the GUI settings tab and select “show ribbon bar”.

Note: Some screenshots may differ slightly in their appearance to the version that you are using.

3.0 Setting up a Simple Network

This section will take users through a step-by-step guide to set up a simple pipe network including setting up objects used in the network.

- At any point the Maximus help system can be viewed for the current dialog. Press **F1** to view in a browser.
- Set the units to be used for data entry. Although an overall unit set is defined each value can be entered in any units by using the local unit definition. For this example, we will set up in SI units, **Home->Units** and set to SI and press **OK**, Figure **4**.

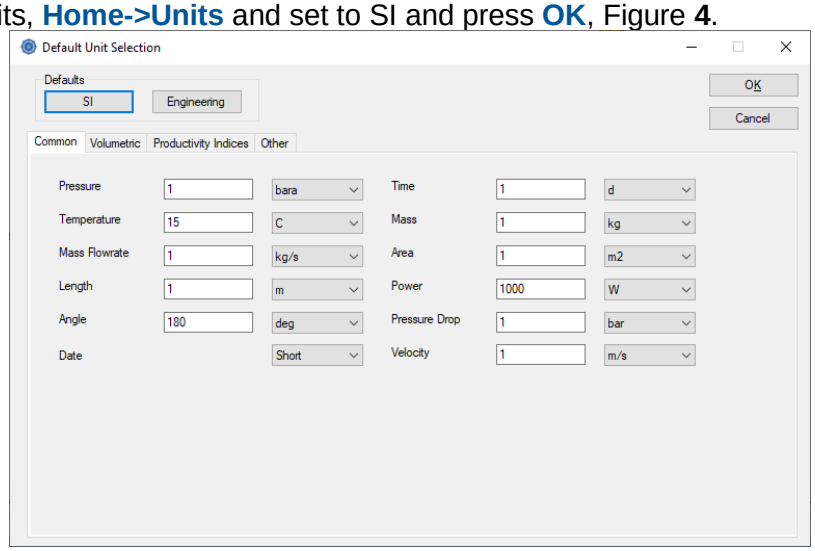

Figure 4: Units set.

- Next, we set the fluid type; options for fluid modelling are either Black Oil or Compositional. Ensure that **Home->Compositional Fluid** is selected as opposed to Black Oil. Use the down arrow to show the options.

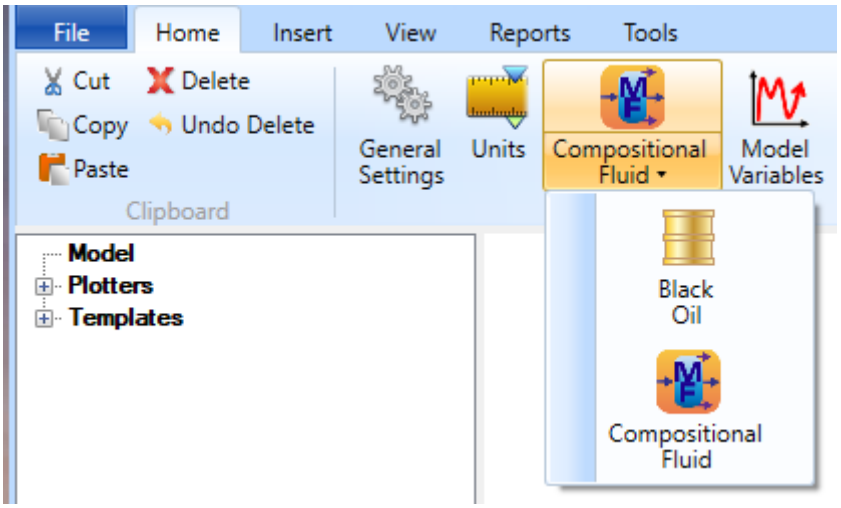

Figure 5: Fluid selection via dropdown options.

- Now we can start to build the model. Select the pipe object from the **Insert tab** as shown in Figure **6**. Click on the model canvas to place the pipe object. Extend the pipe by drag and dropping one of the black squares at the end of the pipe object. If you want to move the pipe, drag-and-drop the black arrow in the middle of the pipe. The red circle indicates that the object is either not connected to another object or is missing data.

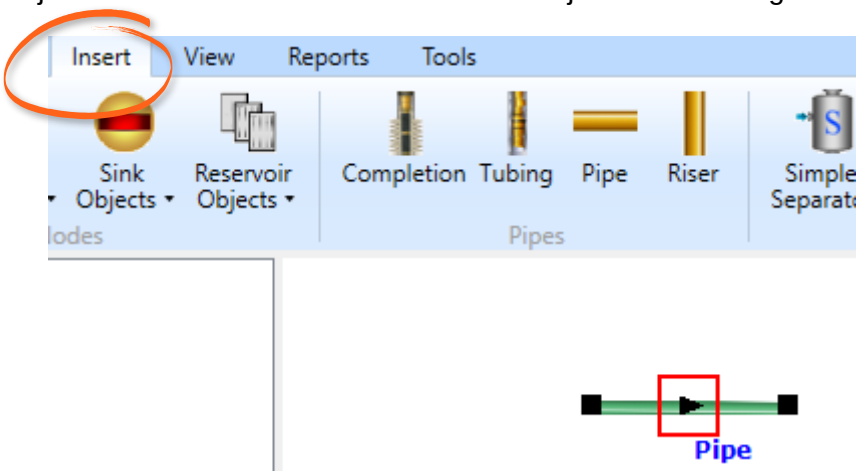

Figure 6: Adding a pipe object to the model canvas.

- Open the pipe object window by double clicking on the black arrow. In this window, it is possible to enter the parameters of the pipeline such as dimensions, models etc. *Note*: *Some fields are pre-populated*
- To begin with, change the name of the pipe to Trunkline (each object must be uniquely named) by using the box at the top of the window. *Note*: *Object names are not case sensitive, so PIPE is the same as pipe.*
- Set the internal diameter to 12" (use the drop-down menu to change the units from mm to inches) and the roughness to 0.05 mm. If you enter the value before changing the units, the value will change so it is the equivalent in the new units. To prevent this from happening, hold shift when selecting the units. It is also possible to use the pipe schedule button to add pipe properties from the Maximus database.

- The small square in the number field can be used to store comments about the data entered, Figure **7**. This option can be disabled by un-checking the "Display comments button on input" option under **Home -> General Settings -> GUI Settings.** The models history and notable changes can also be recorded under **Reports -> Revision History.**

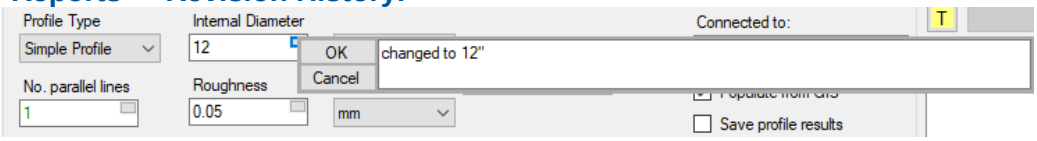

Figure 7: Notes entry for data entry fields.

- Copy the data from Table 1 (in the Excel file provided) and paste it into the profile table in the bottom right of the window. After the data has been entered the window should look like Figure 8. *Note: Actual pipeline profiles can easily be obtained by using the GIS view.*

Horizontal Distance (km)	Elevation (m)	Ambient Temperature (°C)
0	0	5
5	5	5
10	-30	5
15	10	5
20	10	5
25	0	5
30	10	5
35	7	5
40	-11	5
45	-20	5
50	2	5
55	-13	5
60	-15	5
65	-3	5
70	-27	5
75	-20	5
80	8	5
85	-9	5
90	-17	5
95	-2	5
100	-26	5

Table 1: Trunkline elevation profile and ambient conditions.

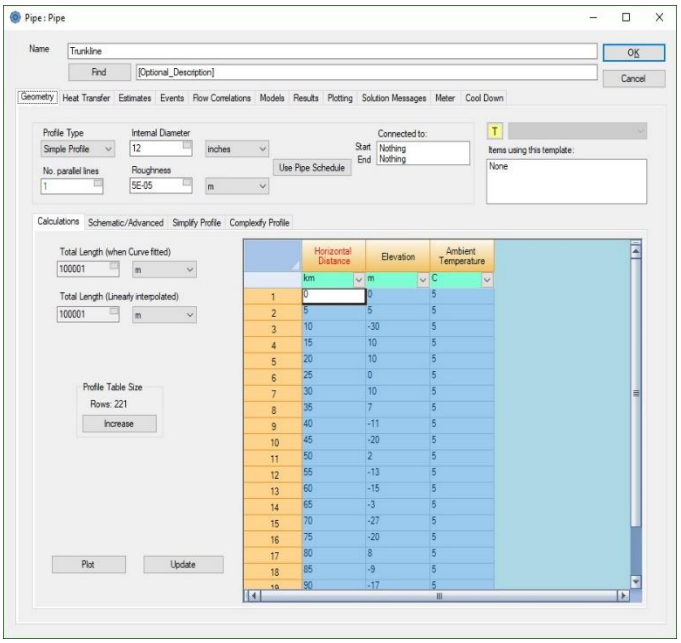

Figure 8: Main trunkline settings

- Click on **Plot** and the window shown in Figure 9 will appear. This is a graphical representation of the elevation of the trunkline over its length. The plot can be enlarged, modified, exported, or printed via the tabs and buttons.

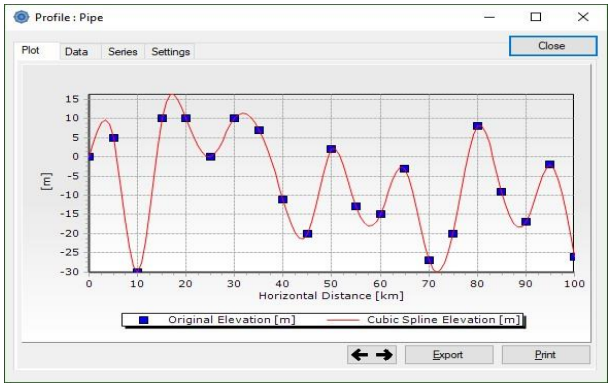

Figure 9: Trunkline elevation profile

- Click on **Close** to return to the main pipe object window.
- Click on the **Heat Transfer** tab. There are two options for calculating heat transfer where a U value can either be specified calculated based on the properties of the pipe, insulation materials and the pipe surroundings including burial, water etc.
- We will not calculate the U value but specify it instead with a value of 4.5 W/m^2/K (average ground value),

- Figure 10. The U value is referenced to the pipe outer diameter, so does not include the pipe wall. So, the pipe wall thickness needs to be set, in this case 25 mm.

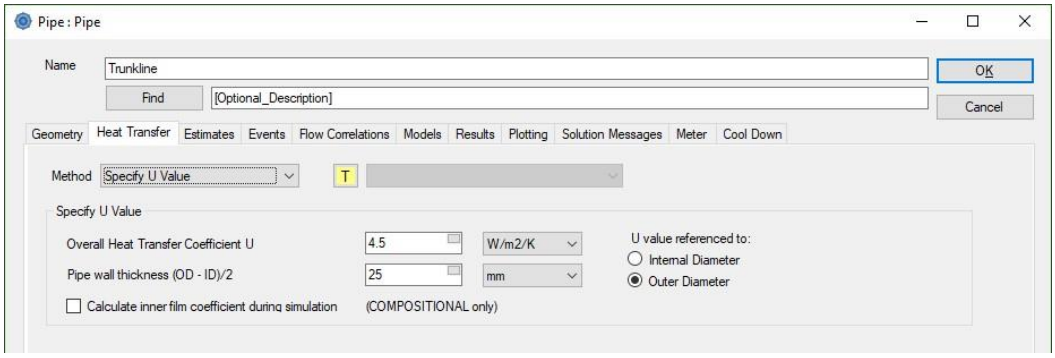

Figure 10: Heat transfer options.

- We can ignore **Estimates, Events** and **Models** for this example, so click on the **Flow Correlations** tab. This page allows the user to select the correlations used to model the flow within the pipe. Leave the correlations to the default. Advantages and a description of each correlation can be found in the Maximus help files.

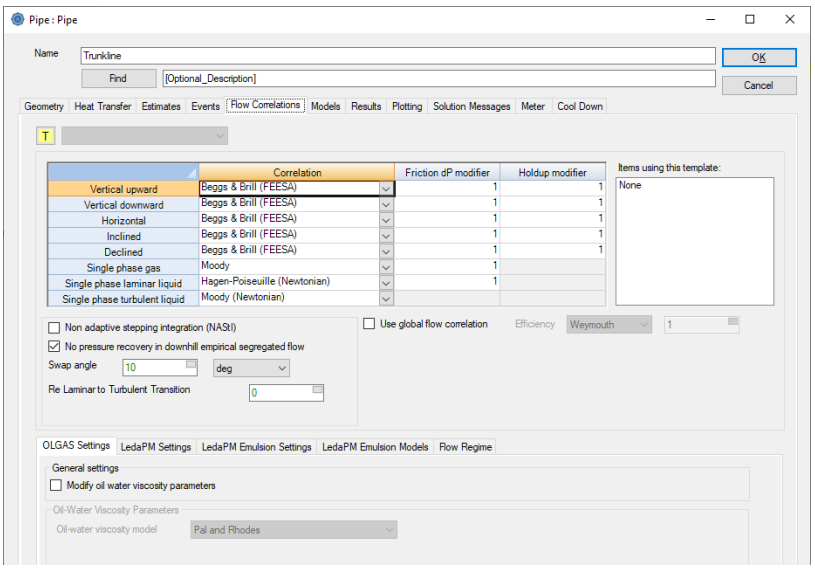

Figure 11: Default flow correlations

- Click on **OK** to save the changes and close the window. *Note*: Selecting Cancel on any top-level dialog will cancel all changes made in sub-dialogs.
- Next a second pipe segment will be added. Add a new pipe to the model canvas and place it to the left of the trunkline. To attach the new pipe, drag and drop the right-hand black square to the left-hand black square of the trunkline. If they have connected correctly, a yellow circle should appear.
- This yellow circle is a node and specifically a junction node. This junction is used to connect pipes together and may have more than two connections. It is assumed that no mass is produced or lost at a junction, i.e., the sum of the flowrate of the fluids that enter the junction is equal to the flow of the fluids that leave the junction.
- The model should look like Figure 12.

Figure 12: Model with two pipes connected.

- Open the new pipe and enter the following details:
  a. Name: Trunkline2. *Note: object names can have spaces*
  b. Internal diameter: 12"
  c. Pipeline profile as per Table 2.
  d. Roughness: .05 mm
  e. Overall heat transfer coefficient U: 4.5 W/m^2/K
  f. Wall thickness: 25 mm
  g. Leave flow correlations to default.

Horizontal Distance (km)	Elevation (m)	Ambient Temperature (°C)
0	0	5
5	20	5
10	30	5
15	10	5
20	-10	5

Table 2: Trunkline2 elevation profile and ambient conditions.

- Now add three more pipes (Branch 1,2 & 3) so the model looks like

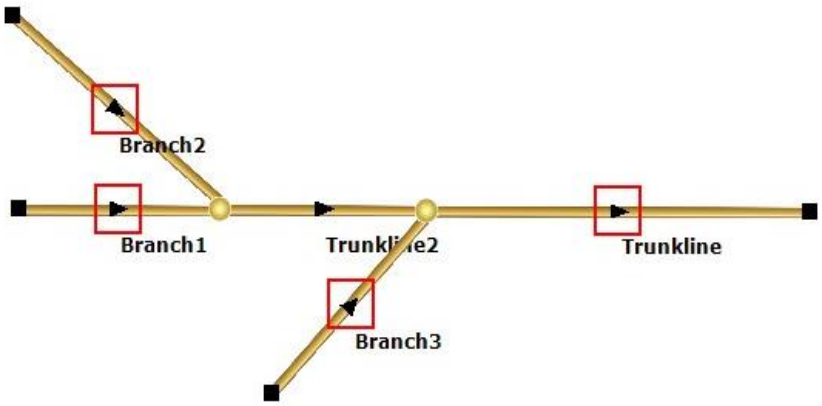

Figure 13: Network configuration.

- For all three pipes, enter the following details:

  a. Rename the pipes so they match **Error! Reference source not found.**.
  b. Internal diameter: 6"
  c. Pipeline profile as per Table 3.
  d. Roughness: 5×10^{-5} m
  e. Overall heat transfer coefficient U: 4.5 W/m^2/K

f. Wall thickness: 25 mm
g. Leave flow correlations to default.

Horizontal Distance (m)	Elevation (m)	Ambient Temperature (°C)
0	0	5
50	10	5
100	5	5
150	-10	5

Table 3: Branch elevation profile and ambient conditions.

This concludes setting up the pipe network.

4.0 Setting up sources and sinks

Now the pipe network has been set up, it is now necessary to introduce fluids to the network at the inputs (the three branches) and remove fluids at the outputs (the end of the trunkline). This is achieved using sources and sinks.

A source is a node object which is used to add fluids to the network. This is where the composition of the fluid is specified which can be achieved using the internal Maximus tool or by importing an Multiflash MFL file. There are also options to specify the fluid flowrate, pressure, and temperature. It is possible to have sources as junctions, i.e., in the middle of a pipe network.

An example where this might be used is to add hydrate inhibitor or gas lift gas.

A sink is a node object where fluids complete their journey through the pipe network. Similarly, to a source object, pressure and flowrates can be specified.
Note, unlike process simulators, such as Petro-SIM® and HYSYS® which are specified from the inlet, Maximus can calculate backwards. For example, it is possible to specify a source flowrate and a sink pressure. In this case, Maximus will back calculate the source pressure to maintain the specified flowrate.

To begin with, add a source object to the model canvas using the same method to add a pipe. The source object can be found on the **Insert** tab.

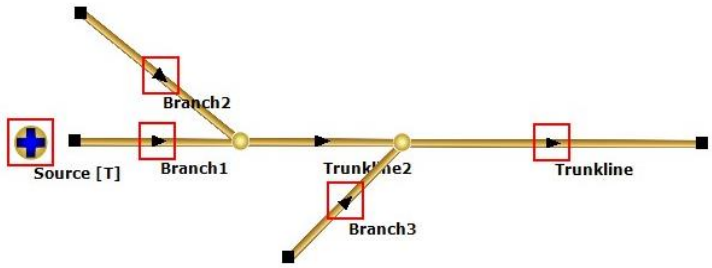

Figure 14: The source object added to the model canvas.

1. Open the source window by double clicking on the object.

2. Under Source fluid, click on the dropdown menu starting with **Launch Multiflash**.

The three options, Figure **15**, allowed to define a fluid composition are either:

a. Launch Multiflash, using the Multiflash GUI,
b. Load MFL file, loading a pre-created fluid file, or
c. Advanced, using the legacy Maximus GUI functionalities.

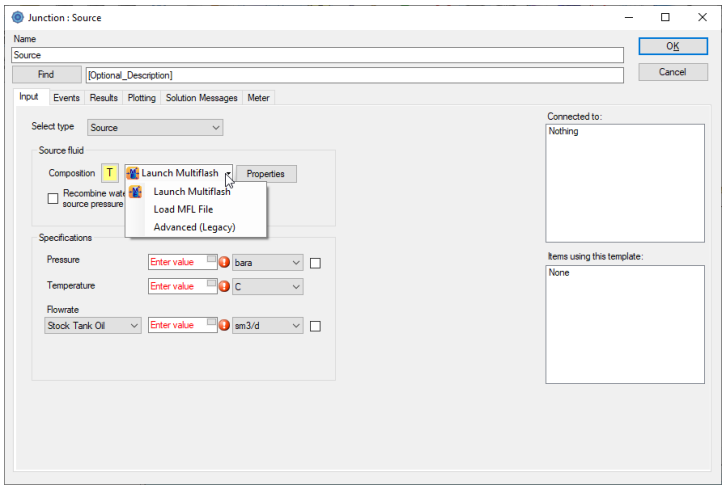

Figure 15: Fluid properties definition choice

3. Select **Load MFL File** and browse for the *Condensate.MFL* file which was supplied with this guide. When the file is loaded, a pop-up confirms that the file was loaded successfully – click **OK**.

4. Click on **Properties** to verify that the composition is loaded correctly. In the **Composition** tab, inspect the component list. In the **Properties** tab, select the appropriate Phase Boundaries and **Calculate** the phase diagram.

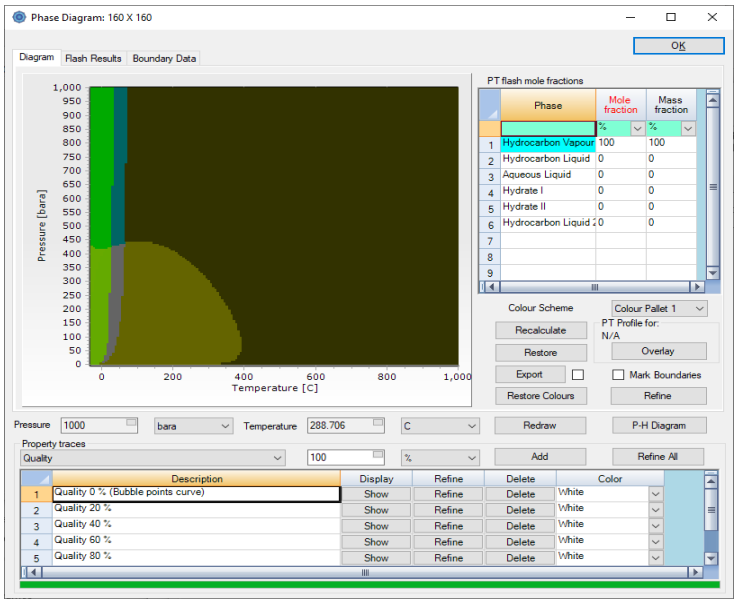

Figure 16: Phase diagram

5. Click **OK** on the Phase diagram and then the Properties window to close them.

6. Back in the Source window:
    a. Enter a temperature of 50°C.
    b. Under flowrate, change the dropdown to Stock Tank Gas and set the flowrate to 10 mmscf/d.
    c. Tick the box next to the flowrate. When ticked this means that this parameter is specified for the model meaning the flowrate from this source will always be used.
    d. Leave the pressure and corresponding tick box blank. This will be calculated by Maximus.

7. The source window should look like Figure **17**. Click **OK** to close the window and save the changes.

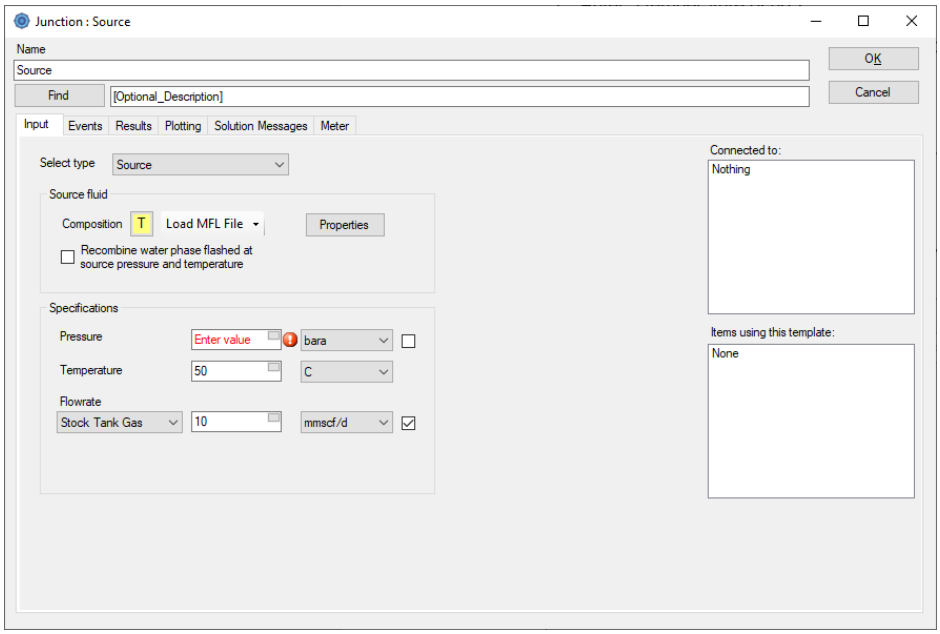

Figure 17: Complete source window

8. Duplicate the source object by **right clicking** on it and select **copy**. Then right click on the canvas and select paste where you want to place the duplicated object. Repeat this so there is a third source, one for each branch.

9. Drag and drop the ends of the three branches onto the relevant sources to connect them.

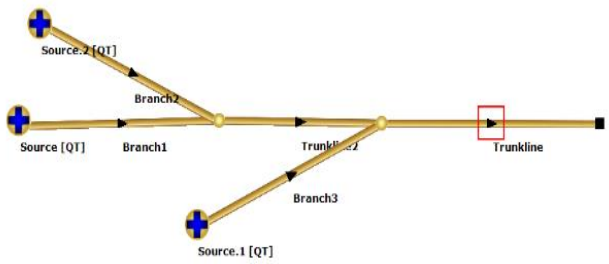

Figure 18: Connected sources

10. Finally, a sink is required. Select a sink object from the **Insert** tab and place it onto the canvas. To make the model more visually appealing the sink object can be any of the sink type objects, i.e., sink, FPSO, Platform or LNG Plant: They all have the same data input requirements and functionality.

11. *Don't forget to regularly save your model!*

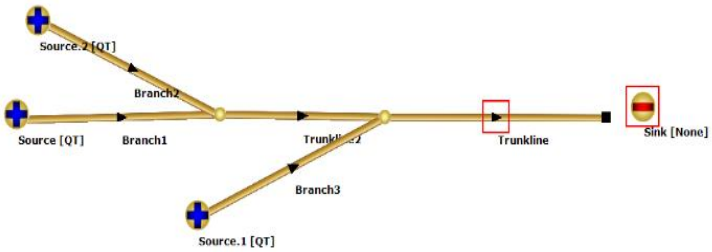

Figure 19: Sink added to the model canvas.

12. Open the sink and in the window, specify an arrival Pressure of 40 bara remembering to tick the box corresponding to pressure. Leave the sink temperature and flowrate blank as these will be calculated. Click **OK** to close the window and save the changes.

13. At various points while setting up the network, red squares will have surrounded the objects. The red square means that there is an issue with the object such as information is missing, or the object needs to be connected to something. In Figure 19, there is a red square around the trunkline and the newly added sink. The trunkline square is because there is nothing connected to its outlet and the sink square is because it is not connected to anything.

14. Attach the end of the trunkline to the sink and both red squares should disappear. If everything has been set up correctly there should be no red squares on the canvas and should look like Figure 16.

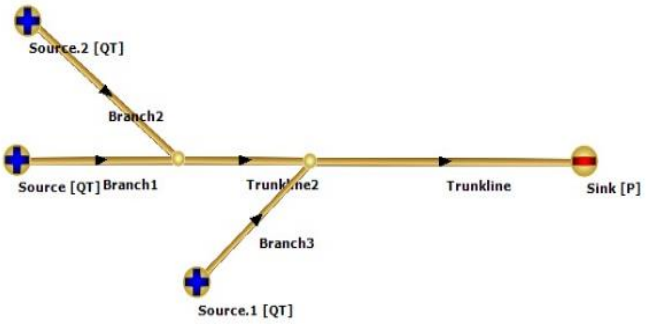

Figure 20: The completed network with sources and sink.

5.0 Setting up and Running the Simulation

Now the network is complete and fully specified, the properties of the simulation need to be set up. This section will demonstrate some of the settings available to the user.

5.1 Snapshot and Life of Field

As discussed earlier, there are two main types of simulation including Life of Field and Snapshot. In this example, snapshot will be used where a single steady-state timestep is used. Other specific operations can be selected and setup from the **Insert** menu. These include:

Gas Lift Optimiser.	Optimise the production when a field has limited is gas lift gas for injection and/or has field constraints.
Hydraulic Potential Controller	
FloWax Branch module	
Look Up table	
Well Stabiliser	
History Matching	
Tuning Tool	
6X Simulator	
UWA Hydrate Formation	Utilise the UWA hydrate model

To ensure the model is run in snapshot mode, click on the snapshot symbol on the **Home** tab, see Figure **21**.

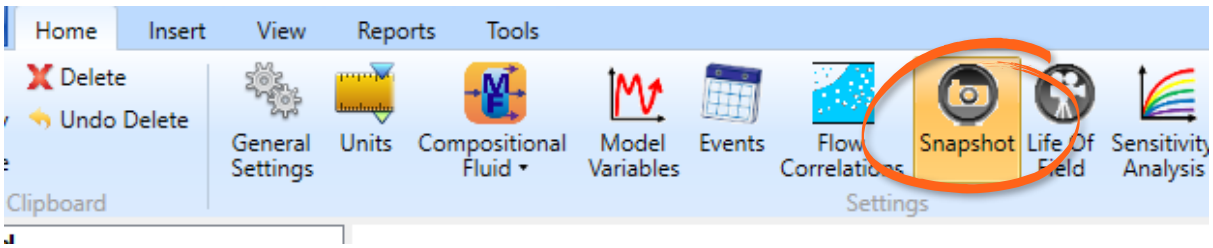

Figure 21: The snapshot mode button.

5.2 General Settings

General settings can be accessed by clicking on the **Home** tab and then **General Settings**. This allows various Maximus settings to be altered to suit the model/user.

On the primary settings tab, change the tolerance from 0.01% to 0.002%. This will improve the accuracy of the results.

As the simulation will be run as a snapshot, there is no need to change the life of field settings.

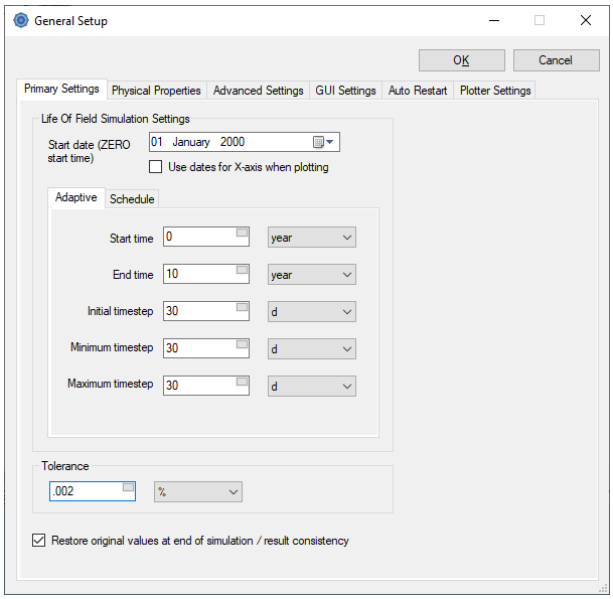

Figure 22: General settings window.

5.3 Fluid Correlations and Models

The fluid correlations and models window can be accessed by clicking on **Home** tab and then selecting **Compositional Fluid**.

This window allows the user to specify the compositional models used in the simulation such as the thermodynamic model and the transport properties. When the MFL file was imported, these settings were overwritten to match the MFL file, therefore, there is no need to change any of the settings.

Click on the **phase diagram** tab. This is where the thermodynamic limits of the model can be set. Restricting this window reduces the amount of computation required. The default maximum pressure is 1,000 bara. It is extremely unlikely that the model will reach this pressure so this can be reduced to 100 bar. Similarly, the temperature will not reach 1,000°C anywhere in the model so reduce this to 100°C.

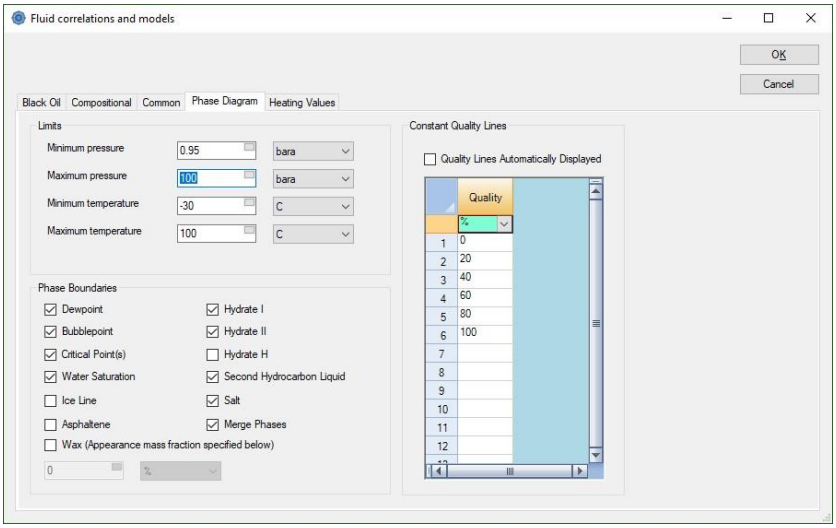

Figure 23: Updated phase diagram limits.

5.4 Running the model

To run the model, click on the play symbol on the **Home** tab or in the title bar.

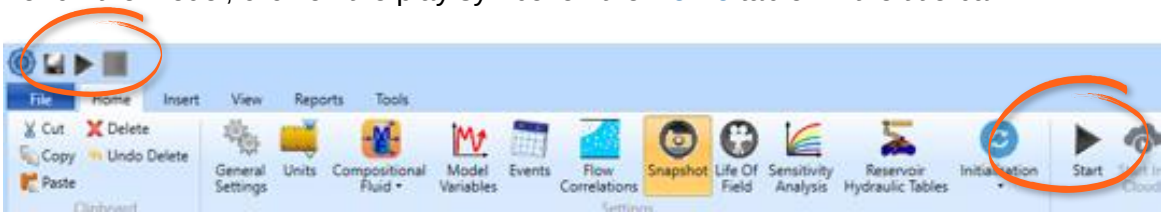

Figure 24: The run button.

If the model has been set up correctly the model should run without any errors and a message box like Figure **25** should appear. Click on **OK** to close the message box.

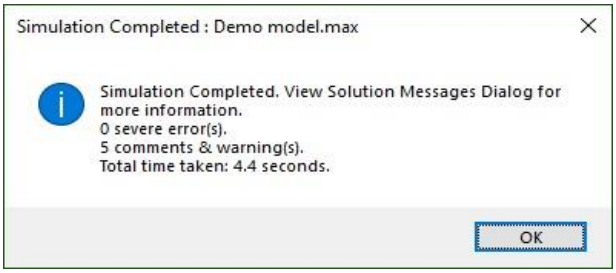

Figure 25: Simulation Completed message.

There should be messages in the dynamic solutions box like Figure **26**. Any messages shown in red are severe errors and should be addressed.

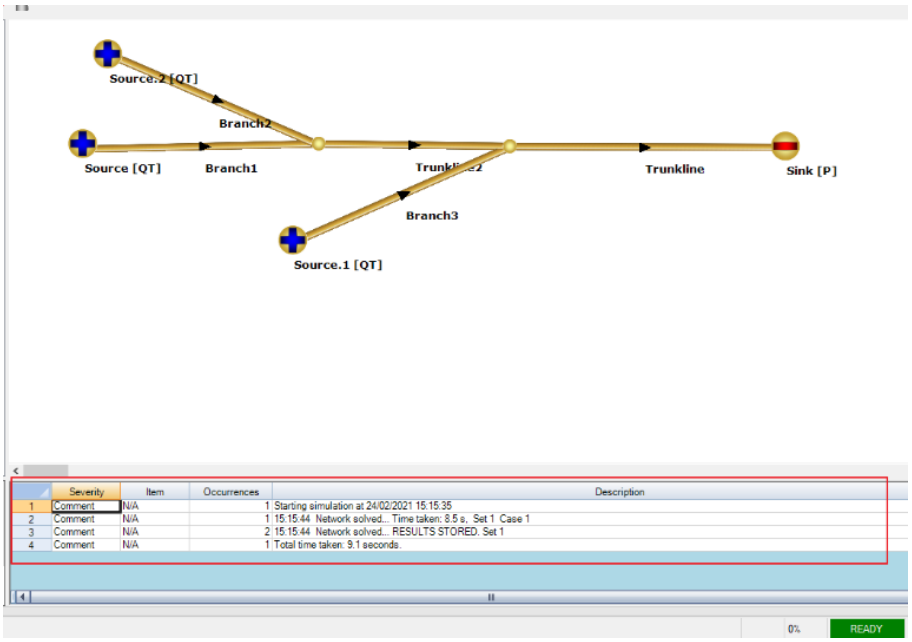

Figure 26: Completed run.

6.0 Extracting results

6.1 Nodes

Now the model has been run it is possible to view the results of the simulation. The model has been set up so that the feed flowrates and temperatures have been set as well as the arrival pressure. This means a result of interest would be the calculated feed pressure. To see this, open the source object called Source and select the **Results** tab.

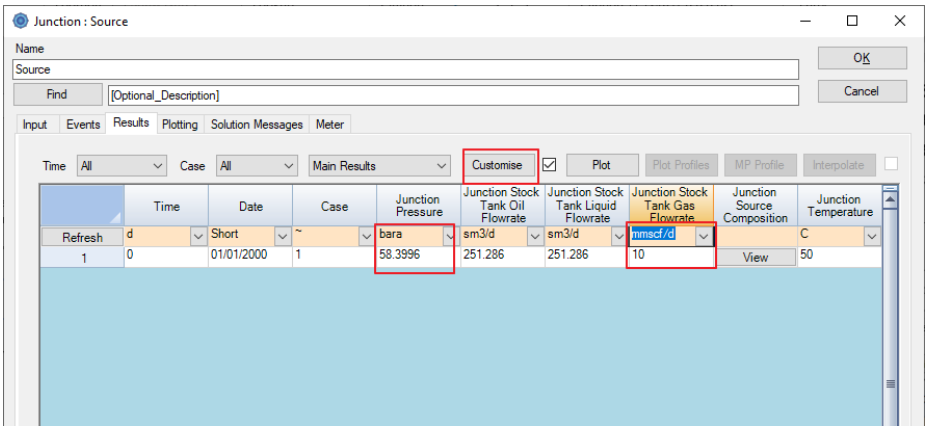

Figure 27: Source results

The pressure at this point to maintain an arrival pressure of 40 bar and a flowrate of 10 MMscfd is about 58 bara. Under Junction Stock Tank Gas Flowrate, change the units to mmscf/d and the value should change to 10 which is what was specified.

It is possible to add other results to this table by clicking on Customise.

Next open the Sink and go to the results tab. The pressure is 40 bara as expected. To view the arrival temperature of the fluid, click on **Customise**. Select Junction Temperature in the pane on the left and click Add. Click on **OK** and message will appear. Click **Yes**. A new column will be added to the right of the existing results showing the temperature of 4.46°C.

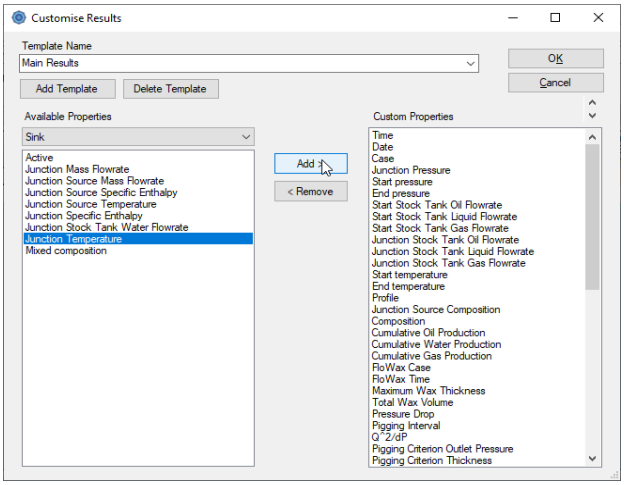

Figure 28: Customising results.

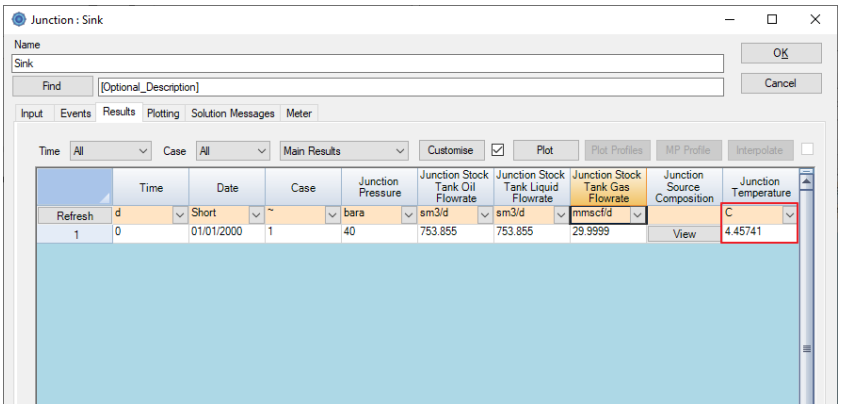

Figure 29: Sink results with temperature added.

As expected, the total volume of gas through the sink is 30 mmscf/d given by the sum of the contribution of each source.

6.2 Branches and profiles

The results that have been looked at so far are node results meaning there is a single result at a specific point. Next, we will have a look at profile results which show multiple results, for example, over the length of a pipeline. Close the sink, open the trunkline window, and go to the results tab. It will look similar to the source and sink results page; however, it will show single results for the inlet and outlet of the pipeline and there is the ability to plot profiles. Locate and tick the tick box under **Profile** and then click **Plot Profiles**.

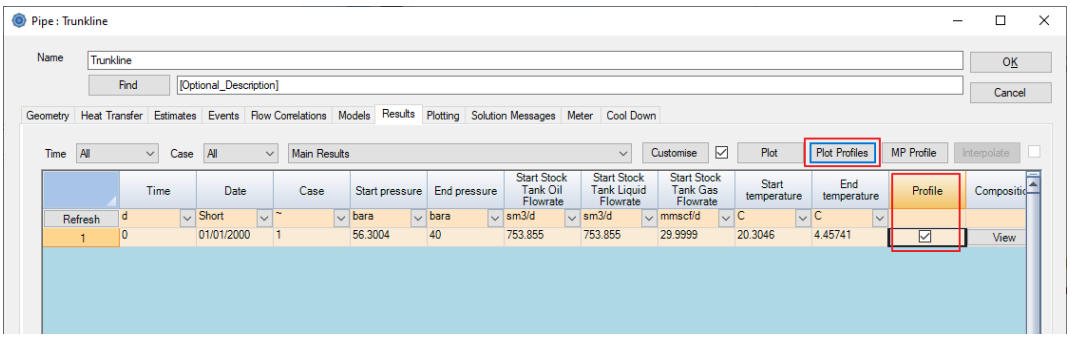

Figure 30: Trunkline results window.

A window will appear showing the default profile results which are temperature and pressure over the pipeline.

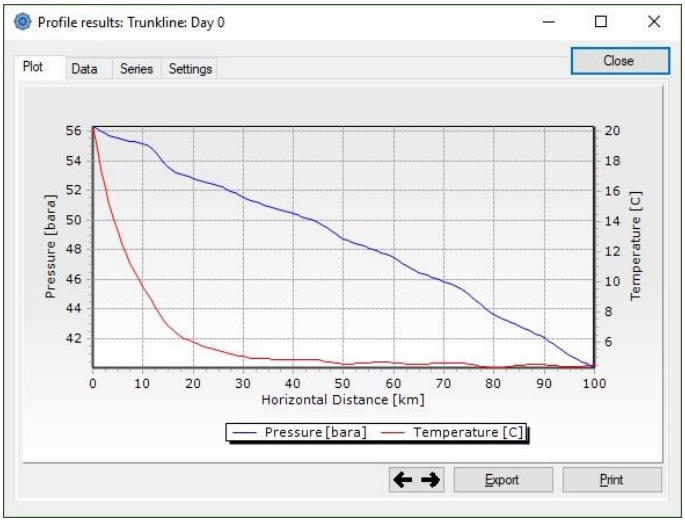

Figure 31: Trunkline temperature and pressure profile.

The raw data can be viewed by clicking on the Data tab. This data can also be copied and pasted for use in other programs such as spreadsheets.

It is possible to change the data represented in the profile chart. We will substitute temperature with liquid holdup. First, select the Series tab. The column called Shown is used to select which data series are included. Untick temperature and tick liquid holdup. The next column, Axis, is used to choose which axis will be used to represent the selected data. For liquid holdup, ensure right is selected. There are also further options to change how the data is represented such as it is possible to change to a stacked chart. Return to the Plot tab and the chart should look like Figure 28.

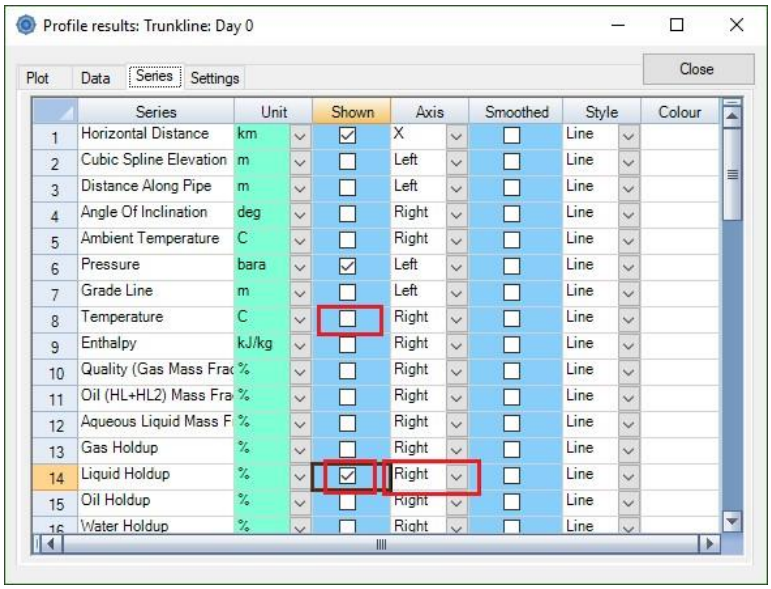

Figure 32: Chart data series selection

.

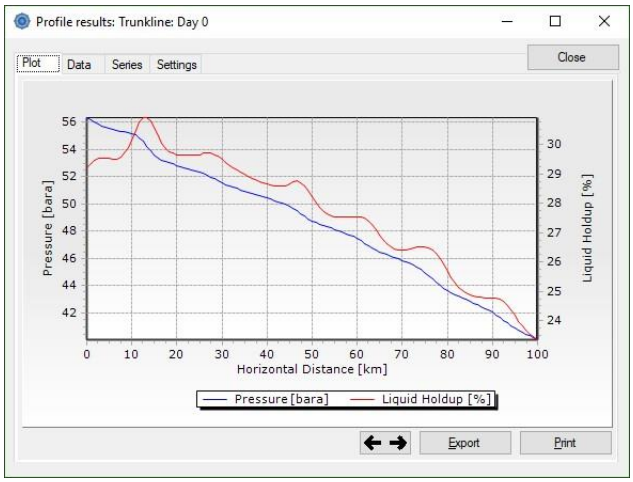

Figure 33: Trunkline profile with pressure and liquid holdup.

In the results tab, by clicking the “View” button in the Composition column in Figure 30, next to the profile checkbox, it is possible to visualise the composition of the fluid at the sink and its properties. By right clicking the View button, the phase diagram is automatically computed. In this case, the composition would be identical to the one at the sources, since the three sources were all created by uploading the same .mfl file. This is not necessarily going to be the same however, in case of fluids with different composition in each source.

7.0 Events Configurator

An event can be connected to any object within the model and consists of 3 elements:

- If (the condention is meet)
- Then set (a particular variable)
- To (a numerical value or status)

The event configurator allows the setting up of the “If” condition for complex events via a dedicated dialogue. It can be turn on or off globally via the option *Use Event Configurator* under **Home->General Settings**.

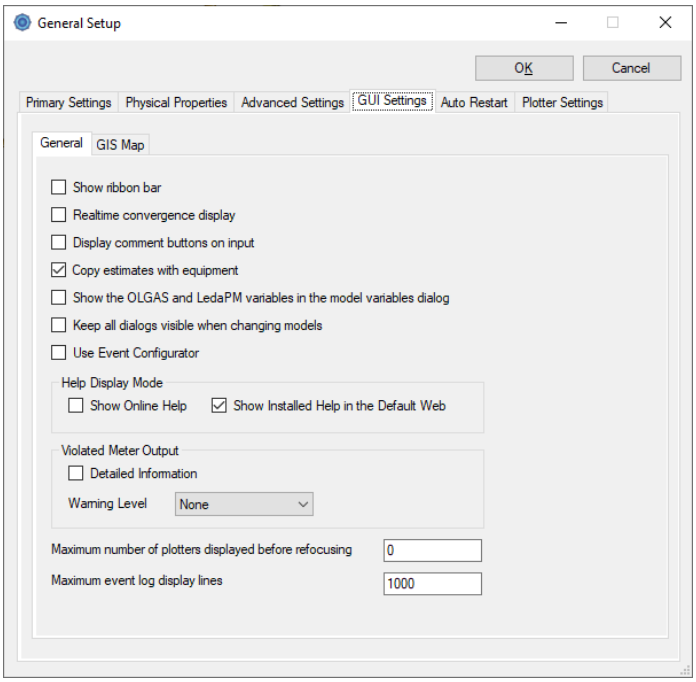

Figure 34: General Setup

And activated or deactivated locally on the events dialog as in Figure 35

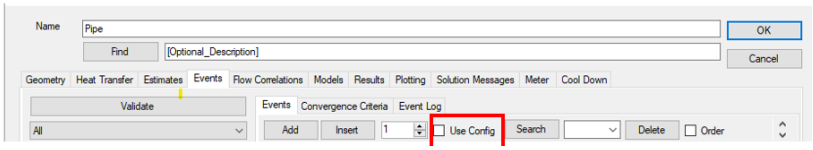

Figure 35: Events Setup

Once selected the Add event will activate the events configurator as in Figure 36

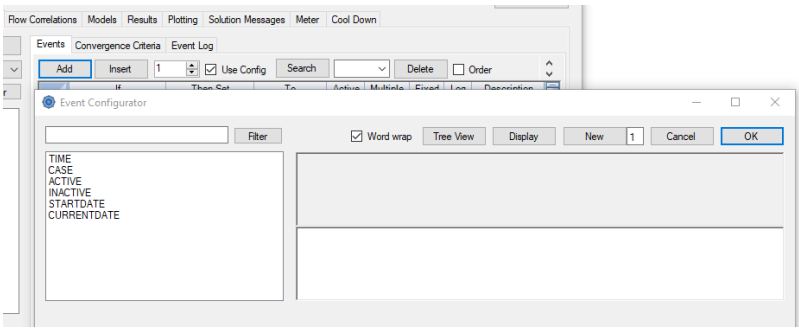

Figure 36: Events Configurator (initial screen)

The recommend workflow is:

- Select the number of levels required (default is 1)
- Select New from Figure 36 and the dialog changes to Figure 37 with 1 new level

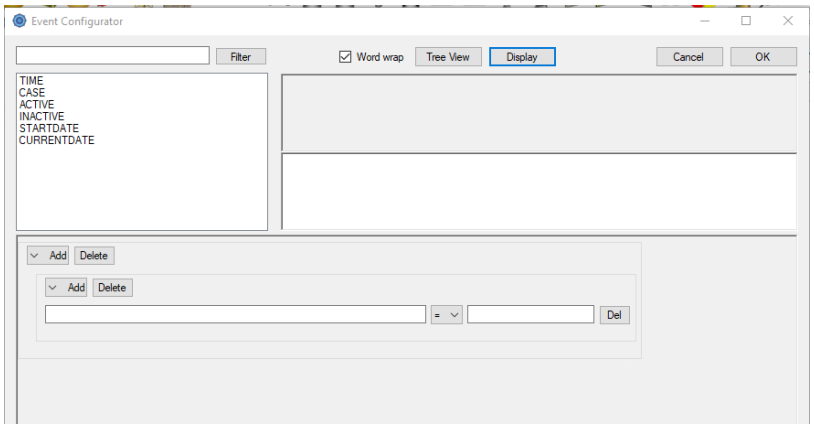

Figure 37: Events Configurator (after New)

- Use of and/or conditions can be specified

Figure 38 shows the use of:

- Multiple conditions
- Tree View
- Model variables will in the list (this is from Exercise 13)

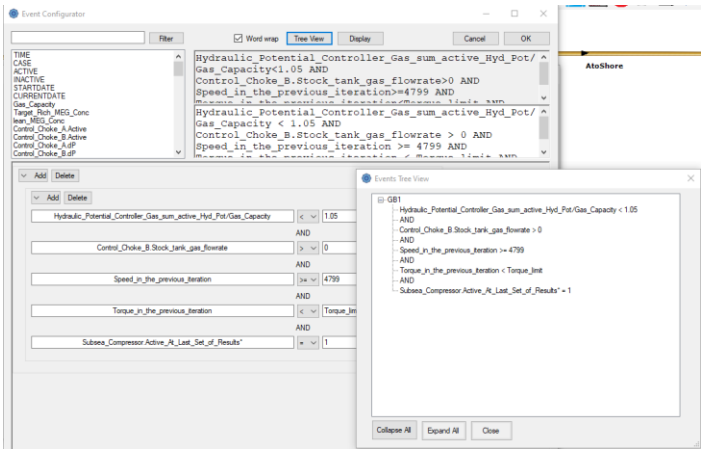

Figure 38: Typical usage

8.0 Shortcuts

Useful shortcuts:

Where	Action	Result
Any spreadsheet	CTRL + mouse wheel	Zoom in and out
Unit description	CTRL + change units	Numbers do not get converted
Canvas	Mouse wheel	Zoom in and out
Canvas	Press and hold mouse wheel	Pan model
Canvas	Shift	Move multiple objects
Anywhere	Press F1 key	Brings up help on that dialog
About menu	About - > Web Portal	Brings up latest information on a Wiki page
About menu	About - > About Maximus	Licensed Features
Canvas	Drag-and-drop a file from Windows Explorer	Files loaded

9.0 Menu structure

File	Home	Insert	View	Reports	Tools
New	Cut/Copy/Paste	Source	GIS map	Solution messages	Excel Add-in
Open->	Delete/Undo Delete	Junction	Switch to classis	Phase Diagram	Batch run
Open Example	General	Sink	Switch Window	Revision history	Licences
Save	Units	Reservoir	Zoom/Find	Plots	Open Interface
Save As	Compositional or Black Oil Fluid	Completion	Icon view	Reporting options	Web Portal
Export	Model variables	Tubing	Toggle Options	Historian	
Print->	Events	Pipe	Full messages	Check Meter	
Close Model	Flow correlations	Riser	Specifications		
Exit	Snapshot	Separators	Runtime displays		
	Life of Field	Field Equipment	Plotter control		
	Sensitivity Analysis	Connector	Templates		
	Reservoir Tables	Chokes			
	Initialisation	Heater/Cooler			
	Start/Abort	Wizard			
	Restart	Operations			
	Pointer/unselect	Utilities			
	Active/inactive				

Trademark/Registered Trademark Reference