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: <u>info@kbc.global</u> www.kbc.global

Registered in England and Wales. Registered number 01357958

© KBC Advanced Technologies Limited 2024. All rights reserved.

Proprietary Information: By accepting this document, the recipient confirms that all information contained herein will be kept confidential, and this information will not be disclosed to third parties without the prior written agreement of KBC Advanced Technologies Limited.



Page 2 of 27



TABLE OF CONTENTS

1.0	Intro	oduction	
2.0	The	Maximus Interface	5
3.0	Sett	ting up a Simple Network	6
4.0	Sett	ting up sources and sinks	
5.0	Sett	ting up and Running the Simulation	
	5.1	Snapshot and Life of Field	
	5.2	General Settings	
	5.3	Fluid Correlations and Models	
	5.4	Running the model	
6.0	Extr	racting results	
	6.1	Nodes	
	6.2	Branches and profiles	
7.0	Eve	nts Configurator	
8.0	Sho	prtcuts	
9.0	Mer	nu structure	





LIST OF FIGURES

Figure 1: General Workflow for Maximus	4
Figure 2: Maximus main GUI	5
Figure 3: Maximus classic view	6
Figure 4: Units set	6
Figure 5: Fluid selection via dropdown options	7
Figure 6: Adding a pipe object to the model canvas.	7
Figure 7: Notes entry for data entry fields	8
Figure 8: Main trunkline settings	9
Figure 9: Trunkline elevation profile	9
Figure 10: Heat transfer options	10
Figure 11: Default flow correlations	10
Figure 12: Model with two pipes connected	11
Figure 13: Network configuration.	11
Figure 14: The source object added to the model canvas.	12
Figure 15: Fluid properties definition choice	13
Figure 16: Phase diagram	13
Figure 17: Complete source window	14
Figure 18: Connected sources	14
Figure 19: Sink added to the model canvas.	15
Figure 20: The completed network with sources and sink	15
Figure 21: The snapshot mode button	16
Figure 22: General settings window.	17
Figure 23: Updated phase diagram limits.	18
Figure 24: The run button	18
Figure 25: Simulation Completed message	18
Figure 26: Completed run.	19
Figure 27: Source results	20
Figure 28: Customising results.	20
Figure 29: Sink results with temperature added.	21
Figure 30: Trunkline results window.	21
Figure 31: Trunkline temperature and pressure profile.	22
Figure 32: Chart data series selection	22
Figure 33: Trunkline profile with pressure and liquid holdup	23
Figure 34: General Setup	24
Figure 35: Events Setup	24
Figure 36: Events Configurator (initial screen)	24
Figure 37: Events Configurator (after New)	25
Figure 38: Typical usage	25
- ··· •	

LIST OF TABLES

Table 1: Trunkline elevation profile and ambient conditions	8
Table 2: Trunkline2 elevation profile and ambient conditions.	11
Table 3: Branch elevation profile and ambient conditions	12



Page 4 of 27



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:

	Define	Unit Set		
E	ng		SI	
	Define F	luid Type		
Compo	sitional		Black Oil	
	Define Flu	uid Source		
Tank Model	Table	Model	S	Source
	Define	Wells		
Completion	Tubing	Gas Lift	W	ellhead Choke
	Define pipe	ing network		
Pipe	Equipent	Riser		Sink
	Set Boundar	y Conditions		
Pressure	Tempe	erature	F	lowrate
	Define Out	out required		
Each	object	S	ytem over	all
	Define C	ontrollers		
HPC	GLO	Wax	V	Vell Stabliser
	Define C	operation		
Snapshot	Life of Field	Sensitivity	Re	eservoir Tables

Figure 1: General Workflow for Maximus







	Your Poone Insert Your ≧ Cut X Delete Image: Section 200 Image: Section 200 Image: Section 200 Clipboard Clipboard Section 200 Section 200 Section 200	Imports Notes Imports Notes Imports Imports	a o o x
F Concostional mode selected G BEEKTY 76.7277	⊢ Model ⊕ Potters ⊛ Templates B	A	
Compositional mode selected G PERATY 7.6.7.2227	F	c Description	× 14 == v
	Compositional mode selected	G	7.6.7.2727

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:





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.

Oefault Unit Select	ion				-	×
Defaults SI	Engineering Productivity Indiaco	Other				O <u>K</u> Cancel
Volumetric	Productivity indices	Other				
Pressure	1	bara 🗸 🗸	Time	1	d 🗸	
Temperature	15	C ~	Mass	1	kg 🗸 🗸	
Mass Flowrate	1	kg∕s ∨	Area	1	m2 ~	
Length	1	m ~	Power	1000	W ~	
Angle	180	deg \sim	Pressure Drop	1	bar 🗸 🗸	
Date		Short \checkmark	Velocity	1	m/s 🗸	

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.



Title: Quick start guide



File	Home	Insert	View	Repo	orts	Tools	
🐰 Cut	🗙 Delet	e	2000			- <u>M</u> -	MA.
Copy	🦘 Undo	Delete	General Settings	Units	Con	npositional Fluid •	Model Variables
Model ⊕Plotter ⊕Templ	lipboard rs ates					Black Oil	
						Compositie Fluid	onal

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.



Page 8 of 27



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.

Profile Type	Internal Diameter	Connected to:
Simple Profile \sim	12 OK changed to 12"	
No. parallel lines	Roughness	
1	0.05 mm V	Save profile results

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.





Pipe : Pipe	e									- 0
Name	Trunkline									OK
	Find	[Optional_Des	cription]							Court
eometry	Heat Transfer E	Estimates Events	Flow Correlati	ons Mod	els R	eaults Plotting	Solu	tion Messages.	Meter Cool	Down
Dutte	. T	Internal Directo						c		T
Profile	e type la Paulite	12	H inches				Start	Connected to:		L.
Ship	le Flotte	14	incries		line D	en Cabadala	End	Nothing		None
No.p	arallel lines	Houghness			use P	the occuednie				THAT IN
1		0E-00 .	m	~						
1	Total Length (whe 100001	n Curve fitted)	~			Horizontal Distance		Elevation	Ambient Temperature	<u> </u>
1	100001	m	~			km			C	
1	Total Length (Line	anly interpolated)		1		0	0		5	
1	100001	m	~	2		5	5		5	
1				3		10	2	0	5	
				4		15	10)	5	
				5		20	10)	5	
	Dulls Tell	. 0		6		25	0		5	
	Prote Tabl	e 3ize		7		30	10)	5	=
	NUNS. 22			8		35	7		5	
	Incre	ase		9	1	40	-		0	
				1	0	50	2		5	-
				-	2	55	-1	3	5	
				1	2	60	-1	5	5	
				1	4	65	-3		5	
				-	•	70	-2	7	5	
				1	5				6.5	
				1	5 6	75	-2	0	5	
	_			1	5 6 7	75 80	-2 8	0	5	
	Plot	Upd	ate	1 1 1	5 6 7 8	75 80 85	-7 8 -9	0	5 5 5	

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/m²/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.





Name	Trunkline								0 <u>K</u>
	Find	[Optional_Description	1						Cancel
	Up at Taxadan a			Modele Res	sults Platting	Solution I	Messages Meter	Cool Down	
eometry	Heat Transfer	stimates Events How	orrelations	Modela Mea	rate ristarig				
eometry	Reat transfer	stimates Events Flow			The starty				
eometry Method	Specify U Value	stimates Events How	T	MOUCIA INC.	into protong				
eometry Method Specify	Specify U Value	stimates Events How		Mouela rica	, intering		1		
eometry Method Specify Overa	Specify U Value U Value	istimates Events How	T 4.5		W/m2/K	~	U value referer	nced to:	

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.

	Trunkline				OF
	Find [Ontion	al Description]			
	Colored Colored				Can
metry	Heat Transfer Estimates E	vents How Correlations Models Results	Plotting Solution Messages	Meter Cool Down	
т		\sim			
		Correlation	Friction dP modifier	Holdup modifier	Items using this template:
	Vertical upward	Beggs & Brill (FEESA)	1	1	None
	Vertical downward	Beggs & Brill (FEESA)	1	1	
	Horizontal	Beggs & Brill (FEESA)	1	1	
	Inclined	Beggs & Brill (FEESA)	1	1	
	Declined	Beggs & Brill (FEESA)	1	1	
	Single phase gas	Moody	1		
Sin	igle phase laminar liquid	Hagen-Poiseuille (Newtonian)	1		
Sing	gle phase turbulent liquid	Moody (Newtonian)			
	n adaptive stepping integratio	n (NASt)	e global flow correlation	Efficiency Weymouth	1 × 1
	pressure recovery in downhi	emoined accordented flow			
	pressure recovery in downini	rempindar segregated how			
Swap a	ingle 10	deg 🗸			
	ninar to Turbulent Transition	0			
Re Lan					
Re Lan					
Re Lan			n Modele Row Regime		
Re Lan	Settings LadaPM Sattings	LedaPM Emulsion Settings LedaPM Emulsion	IT PRODUCES TROPP TROUBLE		
Re Lan	Settings LedaPM Settings	LedaPM Emulsion Settings LedaPM Emulsion			
Re Lan OLGAS	Settings LedaPM Settings al settings	LedaPM Emulsion Settings LedaPM Emulsion			
Re Lan OLGAS Gener	Settings LedaPM Settings al settings dify oil water viscosity param	LedaPM Emulsion Settings LedaPM Emulsive			

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/m²/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/m²/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.

Iunction : Source	- 🗆 ×
Name Source	0 <u>K</u>
Find [Optional_Description] Input Events Results Plotting Solution Messages Meter	Cancel
Select type Source Mid Source Mid Composition T Sel Launch Multiflash Recombine wark Select Milliflash Lead MFL File Advanced (Legacy)	Connected to: Nothing
Presure Enter value Image: bara Image: bara Temperature Enter value Image: bara Image: bara Bowrate Stock Tank Of Enter value Image: bara	Rema using this template : None

Figure 15: Fluid properties definition choice

- 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.
- 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.

Ø Junction : Source	- 🗆 X
Name	0 <u>K</u>
Find [Ontional Description]	Cancel
Input Events Results Platting Solution Messages Meter	
Select type Source V Source fluid	Connected to: Nothing
Composition T Load MFL File - Properties Recombine water phase flashed at source pressure and temperature	
Specifications	Items using this template:
Temperature 50 C V	None
Rowrate Stock Tank Gas V 10 mmscf/d V	

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**.

	Home	Insert	View	Repo	orts Tools						
	💢 Delete	e	202	THE REAL PROPERTY.	.M.	MA.					1
,	🔸 Undo	Delete	- C 0 0		E,	L V,	ШĐ	<u></u>	U		
			General Settings	Units	Compositional Fluid •	Model Variables	Events	Flow Correlations	Snapshot	Life Of Field	Sensitivity Analysis
	Clipboard							Setting	gs		
ŝ											

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.







•							
						0 <u>K</u>	Cancel
rimary Settings Physical	Properties	Advance	d Settings	GUI Settings	Auto Restart	Plotter Settin	ngs
Life Of Field Simulation	Settings						
Start date (ZERO start time)	01 Januar Use date	y 2000 es for X-axi	s when plot				
Adaptive Schedule	e						
Start time	e 0		year	\sim			
End time	e 10		year	~			
Initial timester	30		d	~			
Minimum timester	30		d	~			
Maximum unester	30		d	~			
Tolerance							
.002	%	~					
_							

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.







luid correlations and mode	els					- 0
ck Oil Compositional Comm	non Phase Diagram	Heating Values				O <u>K</u> Cancel
Limits			Constant	Quality Lines		
Minimum pressure	0.95	bara 🗸	[] Qu	uality Lines Automatic	ally Displayed	
Maximum pressure	100	bara 🗸				
Minimum temperature	-30	C ~		Quality	^	
Maximum temperature	100	c ×		% ~		
			1	20		
Dhave Davidarias			3	40		
Priase boundaries			4	60		
			5	80	=	
Bubblepoint	Hydrate	1	6	100		
Critical Point(s)	Hydrate	4	7			
Water Saturation	Second	Hydrocarbon Liquid	8			
C Ice Line	Salt		9			
Asphaltene	Merge Pl	lases	10			
Wax (Appearance mass	fraction specified belo	w)	12			
		.,			*	
0 %			4			

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.



Page 20 of 27



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.

ر	unction : So	ource										-		×
Nam	e												ОК	
Sour	ce													
	Find	[Option	al_Descriptio	n]									Cancel	
Inp	ut Events	Result	Plotting	Solution Me	ssages	Meter								
	Time All		 ✓ Case 	All	\sim	Main Results	~	Customise	Plot	Plot Profiles	MP Profile	Inter	polate	
			Time	Date		Case	Junction Pressure	Junction Stock Tank Oil Flowrate	Junction Stock Tank Liquid Flowrate	Junction Stock Tank Gas Flowrate	Junction Source Composition	Jur Temj	nction perature	
	Refresh	d	~	Short	~ ~	~	bara 🔍	sm3/d 🗸	sm3/d 🧹	mmscf/d 🧹		С	~	
	1	0		01/01/2000	1		58.3996	251.286	251.286	10	View	50		
										·				111

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.

Template Name OK Main Results OK Add Template Cancel Available Properties Cative Sink Cative Aunction Sure Reserve Stat Pressure Junction Sure Specific Erthalpy Add the Case Junction Sure Specific Erthalpy Stat Stock Tark Class Howrate Junction Sure Specific Erthalpy Stat Stock Tark Class Howrate Junction Sure Stock Tark Water Howrate Stat Stock Tark Class Howrate Meed composition Stat Stock Tark Class Howrate Meed composition Stat temperature Profile Junction Source Composition Composition Case Composition	ustomise Results	-		×
Junction Source Specific Erthalpy Junction Source Temperature Junction Specific Erthalpy Junction Specific Erthalpy Model Composition Model Composition	Jate Name Results dd Template Delete Template ilable Properties c ~ ~ ver ction Mass Rowrate ction Source Mass Rowrate	Custom Properties Time Date Case Start pressure Start pressure	O <u>K</u>	×
Cumulative Gas Production RoVax Care BoWax Time Maximum Wax Thickness Total Wax Volume Pressure Drop Pegging Interval Q 12/dP	cton Source Specific Enthalpy cton Source Temperature cton Specific Enthalpy cton Stock Tent Water Rowrate <u>Elem Hemoencure</u> d composition	c Hemove End pressure Stat Stock Tank (DI Rowrate Stat Stock Tank (DI Rowrate Stat Stock Tank (Di Rowrate Junction Stock Tank (Di Rowrate Junction Stock Tank (Di Rowrate Stat Stock) Tank (Di Rowrate Stat temperature Profile Junction Source Composition Composition Cumulative OI Production Cumulative OI Production Cumulative VMet Production ReViva Case Maximum Wax Thickness Total Wax Volume Pressure Drop Prigging Interval Q" 2/d" P		







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**.

Fine eometry Heat Transfe	fer Estimates	ional_Description	n] Correlations M	lodels Results Plotting	Solution Messages Meter Cool I	Down		Car
ometry Heat Transfe	fer Estimates	Events Flow	Correlations M	lodels Results Plotting	g Solution Messages Meter Cool I	Down		
Time								
Time All								
nine Ai	 Case 	Al v	Main Result	s	~	Customise 🗹 Plot	Plot Profiles MP Profile	
	Time	Date	Case	Start pressure End p	pressure Tank Oil Tank Liq Flowrate Flowra	ck Start Stock Start uid Tank Gas temperati	End temperature Profile	Compos
Refresh d	~	Short 🗸	~ 🗸	bara 🧹 bara	🧹 sm3/d 🔍 sm3/d	✓ mmscf/d ✓ C	~ C ~	
1 0		01/01/2000	1	56.3004 40	753.855 753.855	29.9999 20.3046	4.45741	View

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.

	Data Setting	S								Close	;
	Series	Uni	t	Shown	Axi	s	Smoothed	Sty	le	Colour	E
1	Horizontal Distance	km	~		Х	V		Line	~		Н
2	Cubic Spline Elevation	m	~		Left	~		Line	~		
3	Distance Along Pipe	m	~		Left	~		Line	~		1
4	Angle Of Inclination	deg	~		Right	~		Line	~		L
5	Ambient Temperature	С	~		Right	~		Line	~		
6	Pressure	bara	~		Left	~		Line	~		
7	Grade Line	m	~		Left	~		Line	~		
8	Temperature	С	~		Right	~		Line	~		
9	Enthalpy	kJ/kg	~		Right	~		Line	~		
10	Quality (Gas Mass Frac	%	~		Right	~		Line	~		
11	Oil (HL+HL2) Mass Fra	%	~		Right	~		Line	~		
12	Aqueous Liquid Mass F	%	~		Right	~		Line	~		
13	Gas Holdup	%	~		Right	~		Line	~		
14	Liquid Holdup	%	~		Right	~		Line	~		
15	Oil Holdup	%	~		Right	~		Line	~		

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**.







		0 <u>K</u>	Canc	el
nary Settings Physical Properties Advanced Settings GUI Settings A	Auto Restart	Plotter Setti	ngs	
GIS Map				
Show ribbon bar				
Realtime convergence display				
Display comment buttons on input				
Copy estimates with equipment				
Show the OLGAS and LedaPM variables in the model variables dialog	a			
Keep all dialogs visible when changing models	-			
Use Event Configurator				
Use Event Configurator				
Use Event Configurator Help Display Mode Share Online Male Share Installed Male in the Default Wate				
Use Event Configurator Help Display Mode Show Online Help Show Installed Help in the Default Web	2			
Use Event Configurator Help Display Mode Show Online Help Show Installed Help in the Default Web Violated Meter Output	5			
Use Event Configurator Help Display Mode Show Online Help Show Installed Help in the Default Web Violated Meter Output Detailed Information	2			
Use Event Configurator Help Display Mode Show Online Help Show Installed Help in the Default Web Violated Meter Output Detailed Information Warning Level None	2			
Use Event Configurator Help Display Mode Show Online Help Show Installed Help in the Default Web Volated Meter Output Detailed Information Warning Level None Maximum number of plotters displayed before refocusing 0				
Use Event Configurator Use Event Configurator Use Show Online Help Show Installed Help in the Default Web Use Heter Output Detailed Information Warning Level None Maximum number of plotters displayed before refocusing Maximum event log display lines 1000				



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

Name	Pipe							OK						
	Find	[Opt	ional_Desc	ription	1]									Cancel
Geometry	Heat Transfer	Estimates	Events	Flow	Correlation	Models	Results	Plotting	Solution Mess	ages Meter	Cool Down			
	Vali	jate -			Events	Convergen	ce Criteria	Event L	og	_				
All				\sim	Add	Ins	ert 1	-	Use Config	Search		✓ Delete	Order	÷
								_	cost coring					

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





Title: Quick start guide



Citor -	Ward wara	Canada OK
Filter	Word wrap Tree View Display	Cancel OK
TIME CASE		
ACTIVE INACTIVE		
CURRENTDATE		
V Add Delete		
✓ Add Delete		
	= ~ De	el
		el

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	Files loaded
	Windows Explorer	





Page 27 of 27



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

® Maximus is a trademark of KBC Advanced Technologies Limited and is registered in various territories.
® Multiflash is a trademark of KBC Advanced Technologies Limited and is registered in various territories.
® Petro-SIM is a trademark of KBC Advanced Technologies Limited and is registered in various territories.
® HYSYS is a registered trademark of Aspen Tech in the United States and/or other countries.

