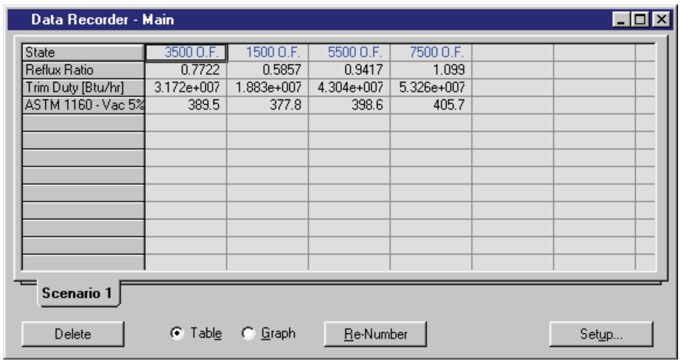
12. Click the **View** button and the **Data Recorder** appears, displaying the new values of the variables.

Figure 2.143

State	3500 O.F.	1500 O.F.
Reflux Ratio	0.7722	0.5857
Trim Duty [Btu/hr]	3.172e+007	1.883e+007
ASTM 1160 - Vac 5%	389.5	377.8

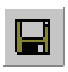
13. Record the process variables for **Overflash** rates of **5500** and **7500** barrels/day. Enter names for these variable states of **5500 O.F.** and **7500 O.F.**, respectively. The final **Data Recorder** appears below.

Figure 2.144



State	3500 O.F.	1500 O.F.	5500 O.F.	7500 O.F.
Reflux Ratio	0.7722	0.5857	0.9417	1.099
Trim Duty [Btu/hr]	3.172e+007	1.883e+007	4.304e+007	5.326e+007
ASTM 1160 - Vac 5%	389.5	377.8	398.6	405.7

14. Save your case by doing one of the following:
    - press **CTRL S**.
    - from the **File** menu, select **Save**.
    - click the **Save** icon.



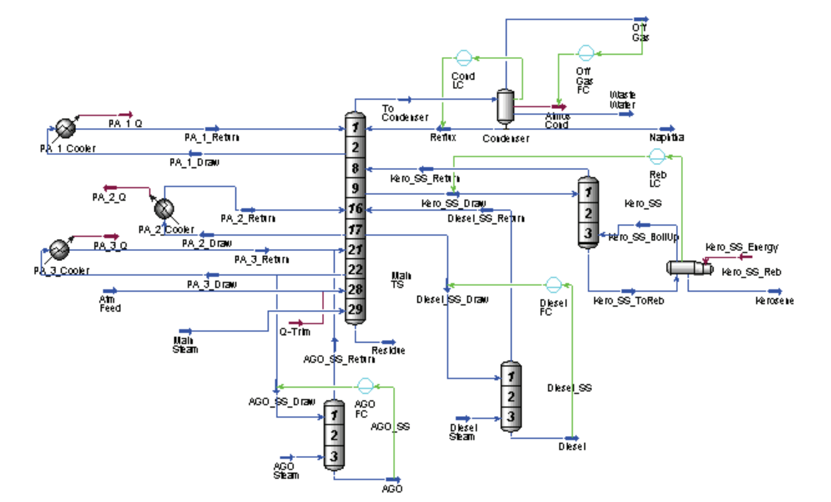
Save icon

2.3 Dynamic Simulation

This complete dynamic case has been pre-built and is located in the file **DynTUT2.usc** in your UniSim Design\Samples directory.

In this tutorial, the dynamic capabilities of UniSim Design are incorporated into a basic steady state oil refining model. A simple fractionation facility produces naphtha, kerosene, diesel, atmospheric gas oil, and atmospheric residue products from a heavy crude feed. In the steady state refining tutorial, preheated crude was fed into a pre-flash drum which separated the liquid crude from the vapour. The liquid crude was heated in a furnace and recombined with the vapour. The combined stream was then fed to the atmospheric crude column for fractionation. The dynamic refining tutorial only considers the crude column. That is, the crude preheat train is deleted from the flowsheet and only the crude column in the steady state refining tutorial is converted to dynamics.

Figure 2.145



The main purpose of this tutorial is to provide you with adequate knowledge in converting an existing steady state column to a dynamics column. The tutorial provides a single way of preparing a steady state case for dynamics mode, however, you can also choose to use the Dynamic Assistant to set pressure specifications, size the equipment in the plant, and/or add additional equipment to the simulation flowsheet.

This tutorial comprehensively guides you through the steps required to add dynamic functionality to a steady state oil refinery simulation. To help navigate these detailed procedures, the following milestones have been established for this tutorial.

In this tutorial, you follow this basic procedure in building the dynamic model.

1. Obtain a simplified steady state model to be converted to dynamics.
2. Implement a tray sizing utility for sizing the column and the side stripper tray sections.
3. Install and define the appropriate controllers.
4. Add the appropriate pressure-flow specifications.
5. Set up the Databook. Make changes to key variables in the process and observe the dynamic behaviour of the model.

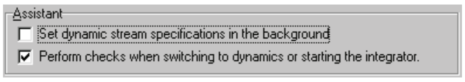
2.3.1 Simplifying the Steady State Flowsheet

In this section, you will delete the preflash train in the steady state simulation case R-1.usc:

1. Open the pre-built case file **R-1.usc**. The crude column simulation file R-1.usc is located in your UniSim Design\Samples directory.
2. Press **F4** to make the Object Palette visible.

   For the purpose of this example, the Session Preferences are set so that the Dynamic Assistant will not manipulate the dynamic specifications.
3. From the **Tools** menu, select **Preferences**. The Session Preference view appears.
4. On the **Simulation** tab, select the **Dynamics** page.
5. Deactivate the **Set dynamic stream specifications in the background** checkbox.

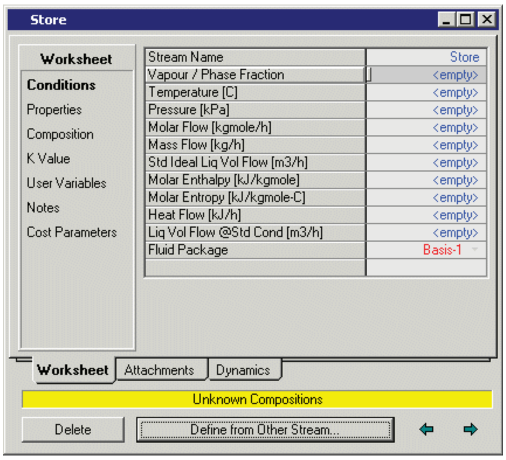
Figure 2.146



In this tutorial, you are working with SI units. The units are changed by entering the Preferences property view in the Tools menu bar. In the Units tab, specify SI in the Current Unit Set group.

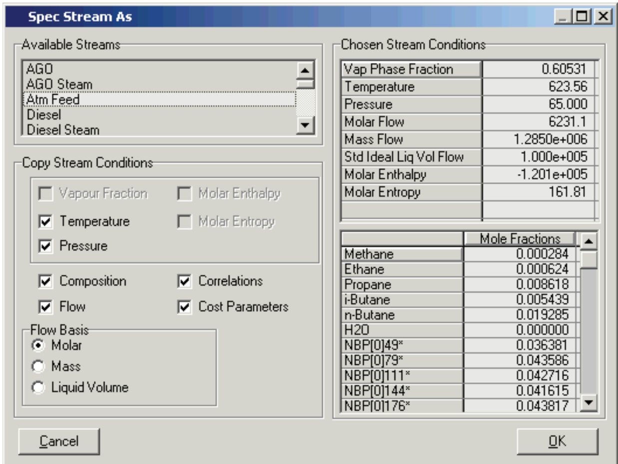
6. Click the **Variables** tab, then select the **Units** page.
7. In the Available Unit Sets group, select SI.
8. Click the **Close** icon to close the Session Preferences view. Close all other views except for the PFD view.
9. Add a material stream to the PFD by doing one of the following:
   - From the **Flowsheet** menu, select **Add Stream**.
   - Double-click the **Material Stream** icon on the Object Palette.
10. In the **Stream Name** cell, type **Store**. This stream will be used to store information from the Atm Feed stream.

Figure 2.147



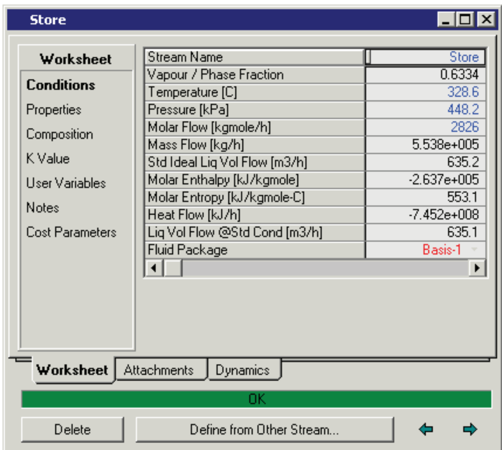
11. In the Store stream property view, click the **Define from Other Stream** button. The Spec Stream As view appears.
12. Select **Atm Feed** in the Source stream group.

Figure 2.148



13. Click on the **OK** button to copy the Atm Feed stream information to the Store stream.

Figure 2.149



14. Close the Store stream view.

When you delete a stream, unit or logical operation from the flowsheet, UniSim Design asks you to confirm the deletion. If you want to delete the object, click the **Yes** button. If not, click the **No** button.

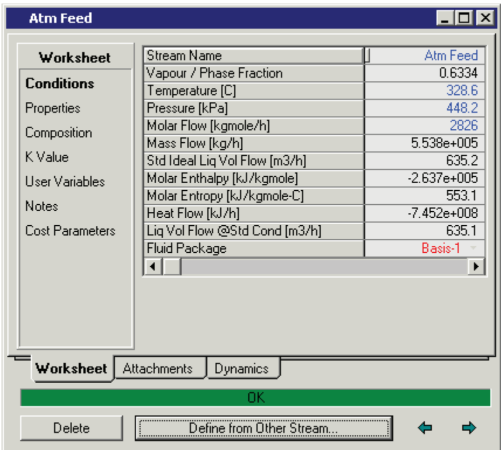
15. Delete all material streams and unit operations upstream of the Atm Feed stream. The following eight items should be deleted:

Items to be deleted		
Material Streams	**Energy Streams**	**Unit Operations**
Hot Crude	Crude Duty	PreFlash Separator
PreFlsh Liq		Crude Heater
PreFlsh Vap		Mixer
Raw Crude		

After you delete the above items, stream Atm Feed is not fully specified.

16. Double-click the **Atm Feed** stream icon to open its property view.
17. Click the **Define from Other Stream** button. The Spec Stream As property view appears.
18. Select **Store** in the Source Stream group and click **OK**.

Figure 2.150



19. Close the Atm Feed stream view, then delete the stream **Store**.

This steady state case now contains the crude column without the preflash train. Since the identical stream information was copied to stream Atm Feed, the crude column operates the same as before the deletion of the preflash train.

20. Save the case as **DynTUT2-1.usc**.

For more information on Session Preferences please refer to **Section 12.5 - Files Tab** in the **UniSim Design User Guide**.

Make sure that the **Standard Windows file picker** radio button is selected on the **File** page in the Session Preferences view.

2.3.2 Adding Equipment & Sizing Columns

In preparation for dynamic operation, the column and side stripper tray sections and surrounding equipment must be sized. In the steady state scenario, column pressure drop is user specified. In dynamics, it is calculated using dynamic hydraulic calculations. Complications arise in the transition from steady state to dynamics if the steady state pressure profile across the column is very different from that calculated by the Dynamic Pressure-Flow solver.

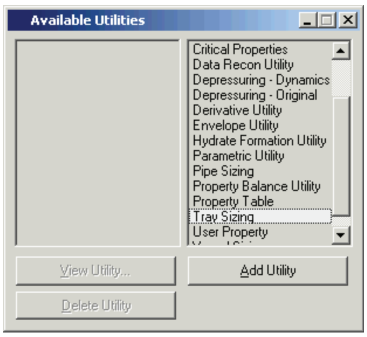
The Cooler operations in the pump arounds are not specified with the Pressure Flow or Delta P option, however, each cooler must be specified with a volume in order to run properly in dynamic mode.

Column Tray Sizing

1. Open the Utilities property view by pressing **CTRL U**.

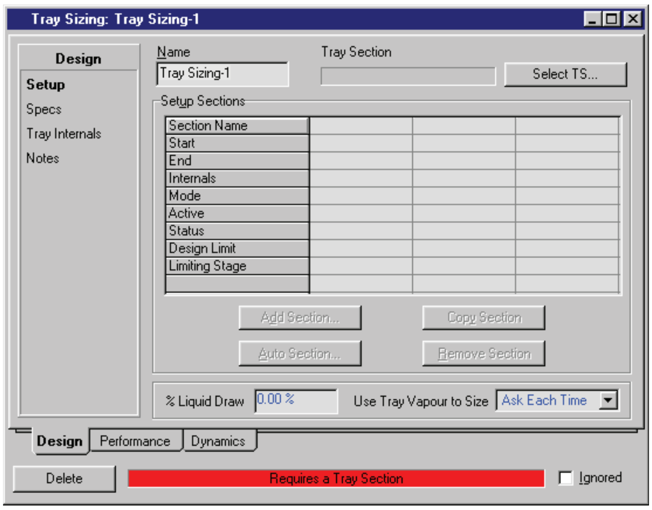
   The Available Utilities view appears.
2. Scroll down the list of available utilities until the Tray Sizing utility is visible.

Figure 2.151



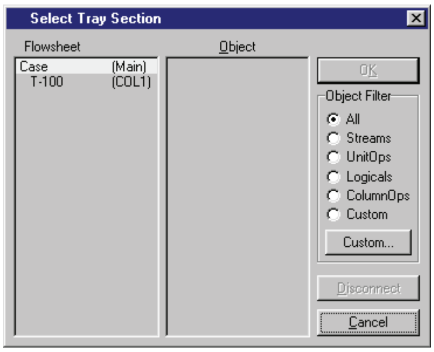
3. Select Tray Sizing, then click the **Add Utility** button. The Tray Sizing view appears.

Figure 2.152



4. In the **Name** field, change the name to **Main TS**.
5. Click the **Select TS** button. The Select Tray Section view appears.

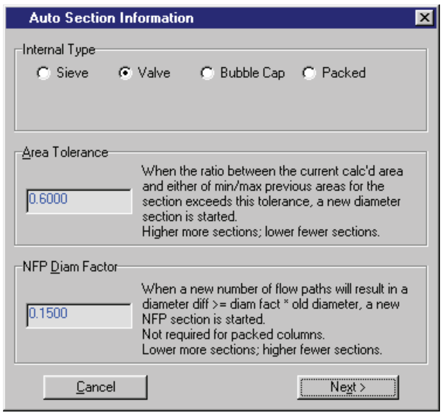
Figure 2.153



6. In the Flowsheet list, select **T-100**, then select **Main TS** in the Object list. Click the **OK** button.
7. In the Use Tray Vapour to Size drop-down list, select **Always Yes**.

8. Click the **Auto Section** button. The Auto Section Information view appears. The default tray internal types appear with the **Valve** type selected.

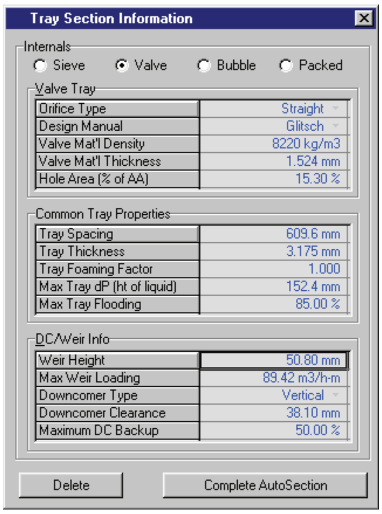
Figure 2.154



The Valve tray type is selected as the default option. This option is entered into the Main TS property view.

9. Keep the default values and click **Next**. The next view displays the specific dimensions of the valve-type trays.
10. Keep the default values and click the **Complete AutoSection** button.

Figure 2.155

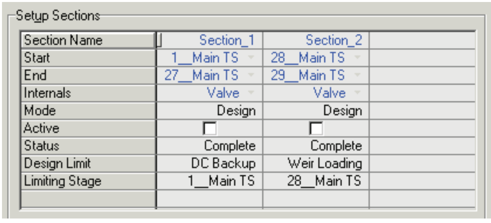


UniSim Design calculates the Main TS tray sizing parameters based on the steady state flow conditions of the column and the desired tray types.

Two tray section sizes, Section_1 and Section_2, appear in the

Setup page of the Design tab. Section_1 includes trays 1 to 27; Section_2 includes trays 28 and 29. Since there are different volumetric flow conditions at each of these sections, two different tray section types are necessary.

Figure 2.156



11. Click the **Design** tab, then select the **Specs** page.
12. In the **Number of Flow Paths** cell, enter **3** for both Section_1 and Section_2.
13. Click the **Performance** tab, then select the **Results** page to see the dimensions and configuration of the trays for Section_1 and Section_2. Since Section_1 is sized as having the largest tray diameter, its tray section parameters should be recorded.
14. Confirm the following tray section parameters for Section_1.

Variable	Value
Section Diameter	5.639 m
Weir Height	0.0508 m
Tray Spacing	0.6096 m
Total Weir Length	13.31 m

The number of flow paths for the vapour is 3. The Actual Weir length is therefore the Total Weir Length recorded/3.

15. Calculate the Actual Weir length:

Variable	Value
Actual Weir Length (Total Weir Length/3)	4.44 m

16. Confirm the Maximum Pressure Drop/Tray and check the number of trays in the Main TS column. The Total Section Pressure drop is calculated by multiplying the number of trays by the Maximum Pressure Drop/Tray.

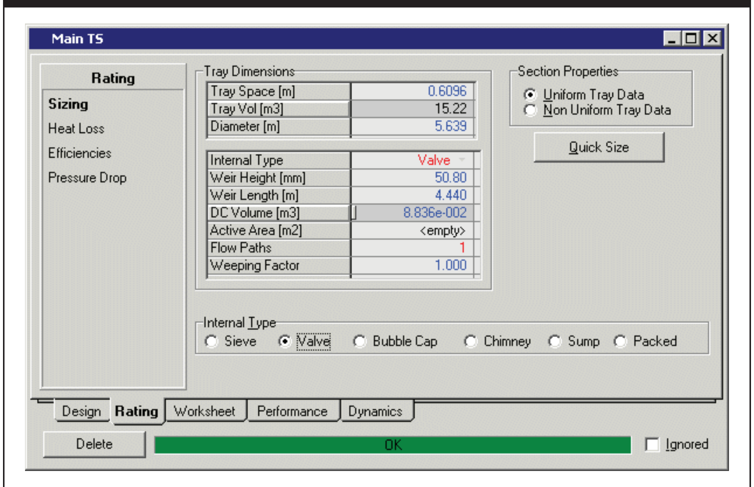
Variable	Value
Maximum Pressure Drop/Tray	0.86 kPa
Number of Trays	29
Total Section DeltaP	24.94 kPa

17. Close the Tray Sizing: Main TS and Available Utilities views.
18. Double-click on the Column T-100 icon in the PFD, then click the **Column Environment** button to enter the Column subflowsheet.
19. On the column PFD, double-click the Main TS Column icon to enter the Main TS property view.
20. Click the **Rating** tab, then select the **Sizing** page.

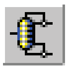
Be aware that the default units for each tray section parameter may not be consistent with the units provided in the tray sizing utility. You can select the units you want from the drop-down list that appears beside each input cell.

21. Enter the previous calculated values into the following tray section parameters:
    - Diameter 5.639m
    - Tray Spacing 0.6096m
    - Weir Height 0.0508m
    - Weir Length (Actual Weir Length) 4.44m
22. In the Internal Type group, select the **Valve** radio button.

Figure 2.157



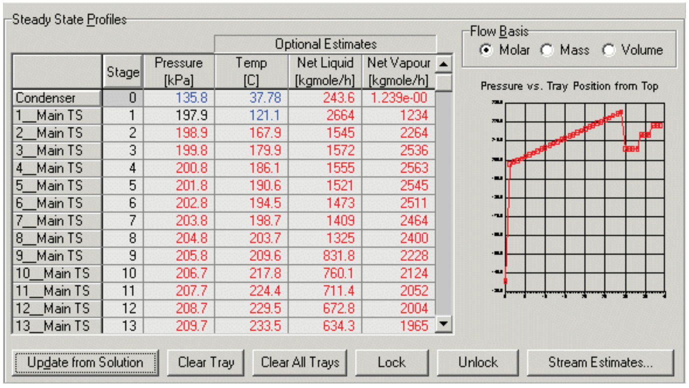
23. Close the Main TS property view.
24. Access the Column property view by clicking the **Column Runner** icon in the tool bar.



Column Runner icon

25. Click the **Parameters** tab, then select the **Profiles** page. Observe the steady state pressure profile across the column.

Figure 2.158



26. Record the top stage pressure (1_Main TS). Calculate the theoretical bottom stage pressure as follows:

$$\text{Bottom Stage Pressure} = \text{Top Stage Pressure} + \text{Total Section Pressure Drop} \tag{2.1}$$

Variable	Value
Top Stage Pressure	197.9 kPa
Total Section Pressure Drop	24.94 kPa
Bottom Stage Pressure	222.84 kPa

27. In the **Pressure** column of the Profiles group, specify a bottom stage pressure (29_Main TS) of 222.84 kPa.

Run Column Solver icon

28. Converge the Column sub-flowsheet by clicking the **Run Column Solver** icon in the tool bar.
29. Close the Column property view.

Side Stripper Tray Sizing

In this section, you will size the following side stripper operations using the tray sizing utility as described in the Column Tray Sizing section.

- Kero_SS
- Diesel_SS
- AGO_SS

1. From the **Tools** menu, select **Utilities**. The Utilities property view appears.
2. Double-click on the Tray Sizing utility. The Tray Sizing view appears.
3. In the **Name** field, change the name to **Kero_SS TS**.

4. Click the **Select TS** button. The Select Tray Section view appears.
5. From the Flowsheet list, select **T-100**, then select **Kero_SS** from the Object list. Click the **OK** button.
6. Click the **Auto Section** button. The Auto Section Information view appears.
7. Select the **Valve** radio button and click the **Next** button.
8. Click the **Complete AutoSection** button to calculate the Kero_SS TS tray sizing parameters.
9. Record the following tray section parameters available on the **Performance** tab in the **Results** page:

Variable	Kero_SS
Section Diameter	1.676 m
Weir Height	0.0508 m
Tray Spacing	0.6096 m
Total Weir Length	1.362 m
Number of Flow Paths	1
Actual Weir Length (calc)	1.362 m

10. Close the Kero_SS TS tray sizing utility.
11. Repeat steps #2-#8 to size the **Diesel_SS** and **AGO_SS** side strippers.
12. Click the **Performance** tab, select the **Results** page, then confirm that the following tray section parameters match the table below:

Variable	Diesel_SS	AGO_SS 1	AGO_SS 2
Section Diameter	1.676 m	1.067 m	0.6096 m
Weir Height	0.0508 m	0.0508 m	0.0508 m
Tray Spacing	0.6096 m	0.6096 m	0.6096 m
Total Weir Length	3.029 m	0.7038 m	0.5542 m
Number of Flow Paths	2	1	1
Actual Weir Length (calc)	1.515 m	0.7038 m	0.5542 m

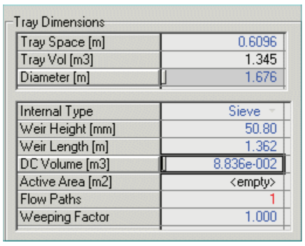
The pressure drop rating information found in the side stripper tray sizing utilities is not used to specify the pressure profile of the Side Stripper unit operations. Since there are only three trays in each side stripper, the pressure drop across the respective tray sections is small. Keeping the pressure profile across the side strippers constant does not greatly impact the transition from steady state mode to dynamics.

13. Close the Available Utilities view.

You should still be in the Column sub-flowsheet environment. If not, double-click the Column T-100 and then click the **Column Environment** button on the bottom of the Column property view.

14. In the PFD, double-click the Kero_SS side stripper icon to open its property view.
15. Click the **Rating** tab, then select the **Sizing** page.
16. Specify the following tray section parameters that were calculated in the previous table:
    - Section Diameter
    - Tray Spacing
    - Weir Height
    - Actual Weir Length

Figure 2.159



17. Close the Kero_SS property view.
18. Double-click the **Diesel_SS** icon, then specify the tray rating information using the table on the previous page. Close the property view when you are done.
19. Repeat the same procedure to specify the tray rating information for **AGO_SS**.

Run Column Solver icon

20. After the column has been specified with the tray rating information, converge the column by clicking the **Run Column Solver** icon in the toolbar.
21. Save the case as **DynTUT2-2.usc**.

Vessel Sizing

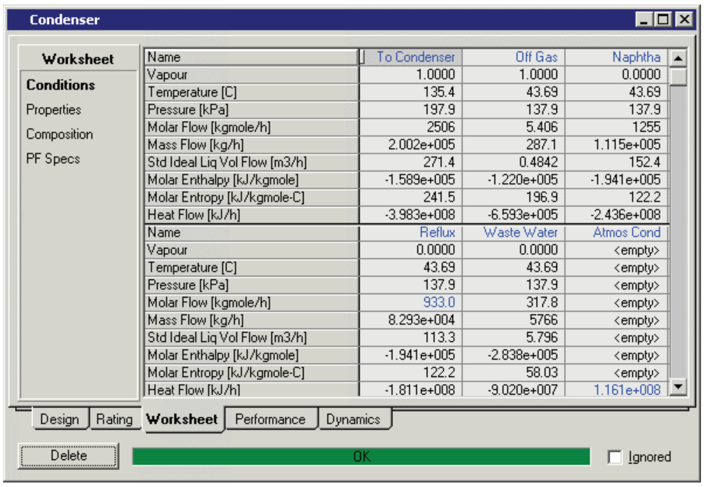
The Condenser and Kero_SS_Reb operations require proper sizing before they can operate effectively in dynamic mode. The volumes of these vessel operations are determined based on a 10 minute liquid residence time.

1. Double-click the Condenser icon on the PFD to open its property

view.

2. Click the **Worksheet** tab, then select the **Conditions** page.

Figure 2.160



3. On the **Conditions** page, confirm the following Liquid Volumetric Flow (Std Ideal Liq Vol Flow) of the following streams:

Liquid Volumetric Flow Rate (m3/h)	Value
Reflux	106.7
Naphtha	152.4
Waste Water	5.736
To Condenser	264.8

4. Calculate the vessel volume as follows, assuming a 50% liquid level residence volume and a 10 min. residence time:

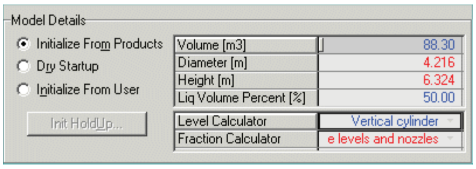
$$Vessel\ Volume = \frac{Total\ Liquid\ Exit\ Flow \times Residence\ Time}{0.5} \tag{2.2}$$

The vessel volume calculated for the Condenser is **88.3 m**^{3}.

5. Click the **Dynamics** tab, then select the **Specs** page.

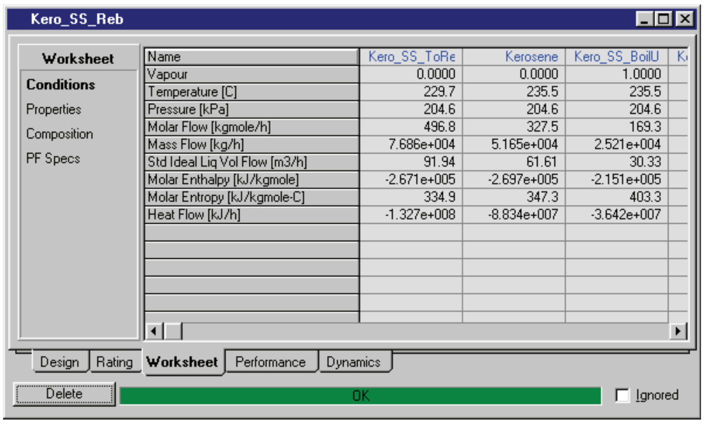
6. In the Model Details group, specify the vessel **Volume** as **88.3 m**^{3} and the Level Calculator as a Vertical Cylinder.

Figure 2.161



7. Close the Condenser property view.
8. In the PFD, double-click the **Kero_SS_Reb** icon to open its property view.
9. Click the **Worksheet** tab, then select the **Conditions** page.

Figure 2.162



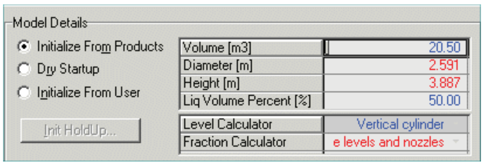
10. In the **Conditions** page, confirm that the Liquid Volumetric Flow (Std Ideal Liq Vol Flow) for Kerosene is 61.61 m^3/h.

    Assume a 10 minute of residence time and a 50% liquid level residence volume. The vessel volume calculated for the Kero_SS_Reb is **20.5 m**^{3}.

11. Click the **Dynamics** tab, then select the **Specs** page.

12. In the **Volume** cell, enter **20.5 m^3**. In the Level Calculator cell, select **Horizontal Cylinder** from the drop-down list.

Figure 2.163



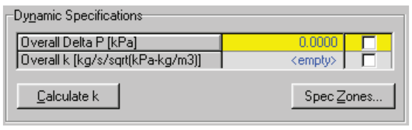
13. Close the Kero_SS_Reb property view.

Cooler Volume Sizing

UniSim Design assigns a default volume to each Cooler unit operation in the Column sub-flowsheet. In this section you will modify each pump around cooler to initialize with a default vessel volume.

1. Double-click the **PA_1_Cooler** operation in the PFD to open the property view.
2. Click the **Dynamics** tab, then select the **Specs** page.
3. In the Model Details group, click in the **Volume** cell, then press **DELETE**. The default volume of 0.10 m^3 appears.
4. In the Dynamic Specifications group, ensure that all the specification checkboxes are inactive. No dynamic specifications should be set for the pump around coolers.

Figure 2.164



5. Close the PA_1_Cooler view.
6. Repeat this process for the PA_2_Cooler and the PA_3_Cooler operations.
7. Save the case as **DynTUT2-3.usc**.

2.3.3 Adding Controller Operations

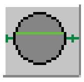
Controller operations can be added before or after the transition to dynamic mode. Key control loops are identified and controlled using

PID Controller logical operations. Although these controllers are not required to run the design in dynamic mode, they increase the realism of the model and provide more stability.

Adding a Level Controller

In this section you will add level controllers to the simulation flowsheet to control the levels of the condenser and reboiler.

First you will install the Condenser controller.

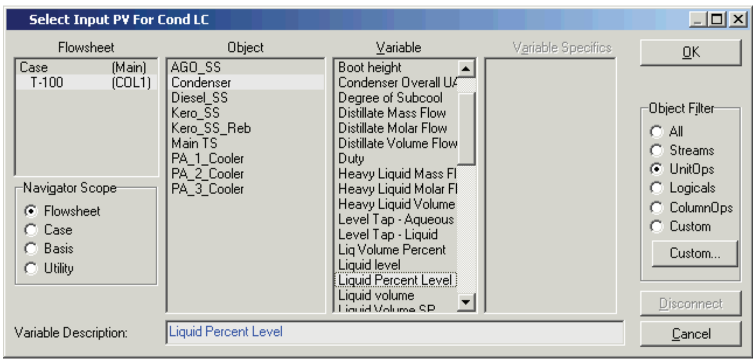


PID Controller icon

For more information regarding PID Controller, see **Section 12.4.4 - PID Controller** of the **UniSim Design Operations Guide**.

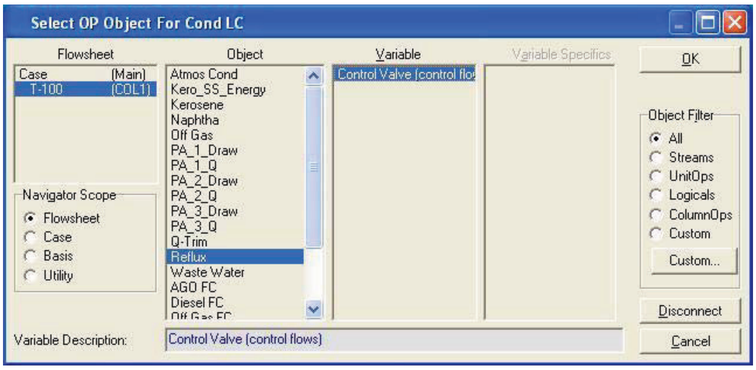
1. If the Object Palette is not visible, press **F4**.
2. In the Object Palette, click the **PID Controller** icon.
3. In the PFD, click near the Condenser operation. The controller icon, named IC-100, appears in the PFD.
4. Double-click the IC-100 icon to open the controller property view.
5. On the **Connections** tab, click in the **Name** field and change the name of the Controller to Cond LC.
6. In the Process Variable Source group, click the **Select PV** button, then select the information as shown in the figure below. Click the **OK** button when you are done.

Figure 2.165



7. In the Output Target Object group, click the **Select OP** button, then select the information as shown in the figure below. Click the **OK** button when you are finished.

Figure 2.166

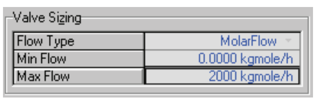


8. Click the **Parameters** tab, then select the **Configuration** page.
9. Supply the following for the **Configuration** page:

In this cell...	Enter...
Action	Direct
Kc	4
Ti	5 minutes
PV Minimum	0%
PV Maximum	100%

10. Click the **Control Valve** button. The FCV for Reflux view appears.
11. In the **Max Flow** cell of the Valve Sizing group, enter 2000 kgmole/h.

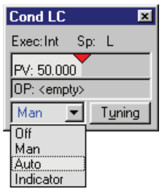
Figure 2.167



12. Close the FCV for Reflux view.
13. Click the **Face Plate** button. The face plate for Cond LC appears.

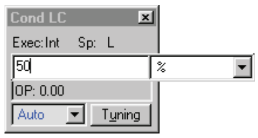
For more information regarding Face Plates, see **Section 12.13 - Controller Face Plate** in the **UniSim Design Operations Guide**.

Figure 2.168



14. Change the controller mode to Auto on the face plate by opening the drop-down list and selecting Auto.
15. Double-click the **PV** cell, then input the set point at 50%.

Figure 2.169



16. Close the Cond LC property view, but leave the face plate view open.
17. Repeat the procedures you just learned to add a PID Controller operation which serves as the Kero_SS_Reb level controller. Specify the following:

If you cannot locate a stream or operation in the Select Input for PV view, select the **All** radio button in the Object Filter group and look again.

Tab [Page]	In this cell...	Enter...
Connections	Name	Reb LC
	Process Variable Source	Kero_SS_Reb, Liq Percent Level
	Output Target Object	Kero_SS_Draw
Parameters [Configuration]	Action	Reverse
	Kc	1
	Ti	5 minutes
	PV Minimum	0%
	PV Maximum	100%

18. Click the **Control Valve** button. The FCV for Kero_SS_Draw view appears.
19. In the Valve Sizing group, enter the following

In this cell...	Enter...
Flow Type	MolarFlow
Minimum Flow	0 kgmole/h
Maximum Flow	1000 kgmole/h

20. Close the FCV for Kero_SS_Draw view.
21. Click the **Face Plate** button. Change the controller mode to Auto on the face plate, then input a set point of 50%. Leave the face plate view open.
22. Close the Reb LC property view.

Adding a Flow Controller

In this section you will add flow controllers to the product streams of the column. These controllers ensure that sufficient material is leaving the column.

1. Click the PID Controller icon in the Object Palette, then click in the PFD near the Off Gas stream. The controller icon appears.
2. Double-click the controller icon to access the property view. Specify the following details:

Tab [Page]	In this cell...	Enter...
Connections	Name	Off Gas FC
	Process Variable Source	Off Gas, Molar Flow
	Output Target Object	Atmos Cond
Parameters [Configuration]3	Action	Direct
	Kc	0.01
	Ti	5 minutes
	PV Minimum	0 kgmole/h
	PV Maximum	100 kgmole/h

3. Click the **Control Valve** button. The FCV for Atmos Cond view appears.
4. In the Duty Source group, ensure that the **Direct Q** radio button is selected.
5. In the Direct Q group, enter the following details:

In this cell...	Enter...
Minimum Available	0 kJ/h
Maximum Available	2 x 10^8 kJ/h

6. Close the FCV for Atmos Cond view.
7. Click the **Face Plate** button. The Off Gas FC face plate view appears. Change the controller mode to Auto, then input a set point of **5** kgmole/h.
8. Close the Off Gas FC property view, but leave the face plate view open.
9. In the Object Palette, click the **PID Controller** icon, then click in the PFD near the Diesel stream. The controller icon appears in the PFD.
10. Double-click the controller icon to access the property view. then specify the following details:

Tab [Page]	In this cell...	Enter...
Connections	Name	Diesel FC
	Process Variable Source	Diesel, Liq Vol Flow@Std Cond
	Output Target Object	Diesel_SS_Draw

Tab [Page]	In this cell...	Enter...
Parameters [Configuration]	Action	Reverse
	Kc	1
	Ti	5 minutes
	PV Minimum	0 m3/h
	PV Maximum	250 m3/h

11. Click the **Control Valve** button. The FCV for Diesel_SS_Draw view appears.
12. In the Valve Sizing group, enter the following details:

In this cell...	Enter...
Flow Type	MolarFlow
Minimum Flow	0 kgmole/h
Maximum Flow	1200 kgmole/h

13. Close the FCV for Diesel_SS_Draw view.
14. Click the **Face Plate** button. The Diesel FC face plate view appears. Change the controller mode to Auto and input a set point of 127.5 m3/h.
15. Close the property view, but leave the face plate view open.
16. Click the **PID Controller** icon in the Object Palette, then click near the AGO stream on the PFD. The controller icon appears.
17. Double-click the controller icon, then specify the following details:

Tab [Page]	In this cell...	Enter...
Connections	Name	AGO FC
	Process Variable Source	AGO, Liq Vol Flow@Std Cond
	Output Target Object	AGO_SS_Draw
Parameters [Configuration]	Action	Reverse
	Kc	0.7
	Ti	3 minutes
	PV Minimum	0 m3/h
	PV Maximum	60 m3/h

18. Click the **Control Valve** button. The FCV for AGO_SS_Draw view appears.
19. In the Valve Sizing group, enter the following details:

In this cell...	Enter...
Flow Type	MolarFlow
Minimum Flow	0 kgmole/h
Maximum Flow	250 kgmole/h

20. Close the FCV for AGO_SS_Draw view.

21. Click the **Face Plate** button. The AGO FC face plate view appears. Change the controller mode to Auto and input a set point of 29.8 m3/h.
22. Close the property view, but leave the face plate view open.
23. Save the case as **DynTUT2-4.usc**.

2.3.4 Adding Pressure-Flow Specifications

Before integration can begin in UniSim Design, the degrees of freedom for the flowsheet must be reduced to zero by setting the pressure-flow specifications. Normally, you make one pressure-flow specification per flowsheet boundary stream, however, there are exceptions to the rule. One extra pressure flow specification is required for every condenser or side stripper unit operation attached to the main column. This rule applies only if there are no pieces of equipment attached to the reflux stream of the condenser or the draw stream of the side strippers. Without other pieces of equipment (i.e., pumps, coolers, valves) to define the pressure flow relation of these streams, they must be specified with a flow specification.

Pressure-flow specifications for this case will be added to the following boundary streams:

- Atm Feed
- Main Steam
- AGO Steam
- Diesel Steam
- Off Gas
- Waste Water
- Naphtha
- Kerosene
- Diesel
- AGO
- Residue

For more information regarding Pressure Flow specifications in Column unit operations see **Chapter 8 - Column** in **UniSim Design Operations Guide**.

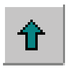
This simplified column has all the feed streams specified with a flow specification. The Off Gas stream has a pressure specification which defines the pressure of the condenser and consequently the entire column. The liquid exit streams of the column and the side stripper operations require pressure specifications since there are no attached pieces of equipment in these streams. All the other exit streams associated with the column require flow specifications.

The following pump around streams require flow specifications since both the Pressure Flow and the Delta P specifications are not set for the pump around coolers.

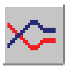
- PA_1_Draw
- PA_2_Draw
- PA_3_Draw

The following streams have their flow specifications defined by PID Controller operations.

- Reflux
- Kero_SS_Draw
- Diesel_SS_Draw
- AGO_SS_Draw



Enter Parent Simulation Environment icon

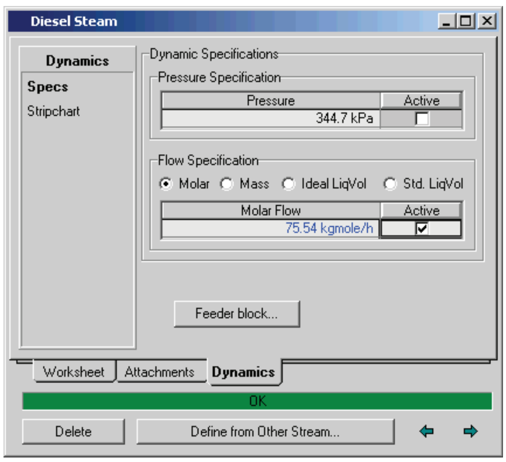


Dynamic Mode icon

1. Enter the Main Flowsheet environment. Close the column property view if it is still open.
2. Switch to dynamic mode by clicking the **Dynamic Mode** icon. When asked if you want to allow dynamics assistant to identify items which are needed to be addressed before proceeding into dynamics, click the **No** button.

   Every material stream in the Main Flowsheet requires either a pressure or flow specification.
3. Double-click the Diesel Steam icon to enter its property view.
4. Click the **Dynamics** tab, then select the **Specs** page.
5. In the Pressure Specification group, clear the **Active** checkbox.
6. In the Flow Specification group, select the **Molar** radio button, then activate the **Active** checkbox.
7. In the **Molar Flow** cell, enter 75.54 kgmole/h if required.

Figure 2.170



Once a pressure or flow specification has been made active, the stream value turns blue and can be modified.

Set the following pressure or flow specifications for the following streams in the Main Flowsheet.

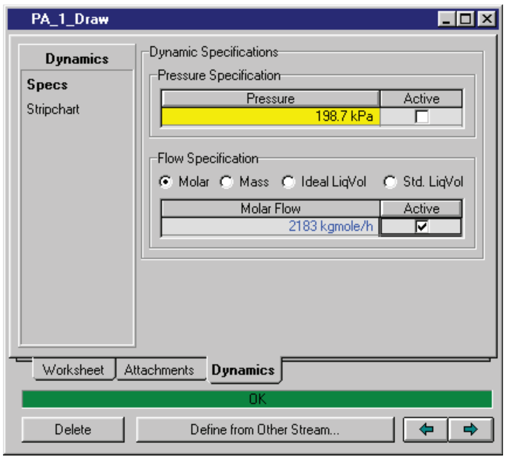
Material Stream	Pressure Specification	Flow Specification	Value
Atm Feed	Inactive	Molar Flow, Active	2826 kgmole/h
Main Steam	Inactive	Molar Flow, Active	188.8 kgmole/h
AGO Steam	Inactive	Molar Flow, Active	62.95 kgmole/h
Off Gas	Active	Inactive	135.8 kPa
Waste Water	Inactive	Molar Flow, Active	317.8 kgmole/h
Naphtha	Inactive	Ideal LiqVol, Active	152.4 m3/h
Kerosene	Inactive	Ideal LiqVol, Active	61.61 m3/h
Diesel	Active	Inactive	211.4 kPa
AGO	Active	Inactive	215.6 kPa
Residue	Active	Inactive	221.6 kPa

8. Use the Object Navigator to enter the Column subflowsheet environment. Click the **Object Navigator** icon in the tool bar. The Object Navigator view appears. In the Flowsheets group, double-click T-100.

   Every material stream in the column environment also requires either a pressure or flow specification. Use the following procedure to set a pressure-flow specification for the PA_1_Draw stream.

9. In the PFD, double-click the PA_1_Draw stream icon to open the property view.
10. Click the **Dynamics** tab, then select the **Specs** page.
11. In the Flow Specification group, select the **Molar** radio button, then activate the **Active** checkbox.

Figure 2.171



12. Close the PA_1_Draw property view.

13. Activate the following flow specifications for the following streams in the Column sub-flowsheet.

Material Stream	Pressure-Flow Specification	Value
PA_2_Draw	Molar Flow	830.2 kgmole/h
PA_3_Draw	Molar Flow	648.0 kgmole/h
Reflux	Molar Flow	879.7 kgmole/h
Kero_SS_Draw	Molar Flow	426.6 kgmole/h
Diesel_SS_Draw	Molar Flow	616.8 kgmole/h
AGO_SS_Draw	Molar Flow	124.8 kgmole/h

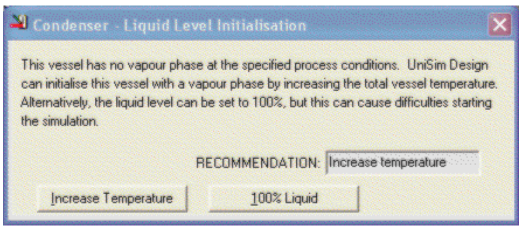
14. Save the case as **DynTUT2-5.usc**.
15. Close all the views except the face plates.
16. To arrange the face plates, select the **Arrange Desktop** command from the **Windows** menu.

Start Integrator icon

17. The integrator can be run at this point. Click the **Start Integrator** icon. When you are given the option to run dynamic assistant, select **No**.

    When the integrator initially runs, UniSim Design detects that no vapour phase exists in the Condenser at the specified process conditions. It displays the following message:

Figure 2.172



    UniSim Design recommends that you increase the temperature setting to create a vapour phase. You can also create a non-equilibrium vapour phase or set the liquid level to be 100%. For the sake of this example, select the default recommendation.

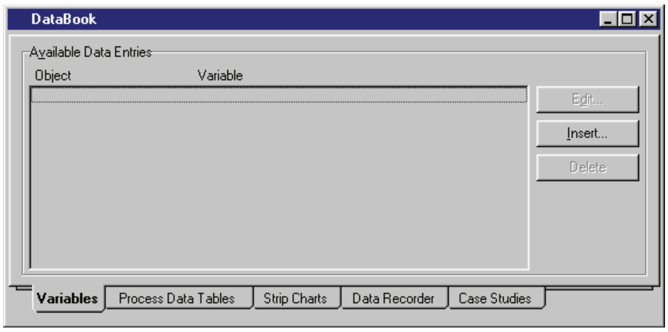
18. Click the **Increase Temperature** button.
19. Let the integrator run for few minutes so all the values can propagate through the column. Observe the value changes on the face plate view.
20. To stop the integrator, click the **Stop Integrator** icon.

2.3.5 Monitoring in Dynamics

Now that the model is ready to run in dynamic mode, the next step is to install a strip chart to monitor the general trends of key variables. The following is a general procedure for installing strip charts in UniSim Design.

1. Open the Databook by using the hot key combination **CTRL D**.

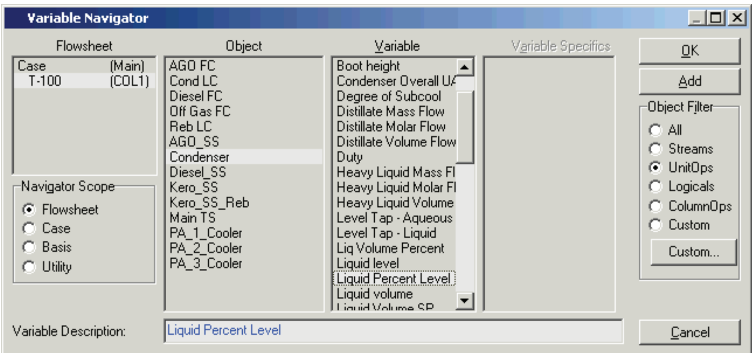
Figure 2.173



The Variable Navigator is used extensively in UniSim Design for locating and selecting variables. The Navigator operates in a left-to-right manner- the selected Flowsheet determines the Object list, the chosen Object dictates the Variable list, and the selected variable determines whether any Variable Specifics are available.

2. On the **Variables** tab, click the **Insert** button. The Variable Navigator view appears.
3. In the Flowsheet list, select the Column T-100.
4. In the Object Filter group, select the **UnitOps** radio button. The Object list is filtered to show unit operations only.
5. In the Object list, select the Condenser. The Variable list available for the column appears to the right of the Object list.
6. In the Variable list, select Liquid Percent Level.

Figure 2.174



7. Click the **OK** button. The variable now appears in the Databook.

If you can't find an Object in the Variable Navigator view, select the **All** radio button in the Object Filter group, then select Case (Main) in the Flowsheet group. All operations and streams for the design will appear in the Object list.

8. Add the following variables to the Databook. If you select the top variable in the list of Available Data Entries before inserting a new variable, the new variable will always be added to the top of the list.

Object	Variable
Kero_SS_Reb	Liquid Percent Level
Off Gas	Molar Flow
Condenser	Vessel Temperature

The next task is to create a Strip Chart to monitor the dynamics behaviour of the selected variables.

9. Click the **Strip Charts** tab in the Databook view.
10. Click the **Add** button. UniSim Design creates a new Strip Chart with the name DataLogger1.
11. Click in the blank **Active** checkbox beside the Condenser/Liquid Percent Level variable.