

Design Procedure for a Heat Exchanger on the AspenPlus Software

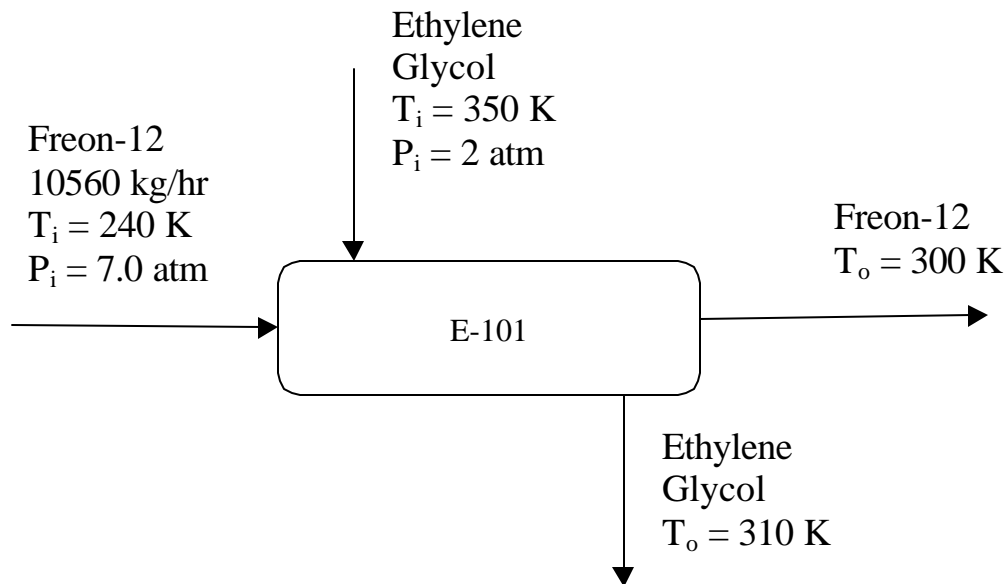
(no phase change)

author: Jim Lang (©SDSM&T, 2000)

This manual will show the steps necessary to create and design a heat exchanger unit on AspenPlus. The manual also includes tips and recommendations for the design process. The following example will be used:

Problem statement—Freon-12, at a flow rate of 10560 kg/hr, needs to be heated from 240 K to 300 K. Ethylene glycol is available at 350 K. A typical shell and tube heat exchanger will be used. The plant manager recommends that the minimum temperature approach should be at least 10 K. He also recommends using 20 BWG carbon-steel tubing with a pressure drop not exceeding 10 psig (0.67 atm) for either the shell or tube side.

Schematic:



Before starting on Aspen, you should read some of the background information behind heat exchanger design. Here are some recommended readings:

Perry's 7th edition—pg. 5-12 through 5-17, 11-4 through 11-11, and 11-33 through 11-45

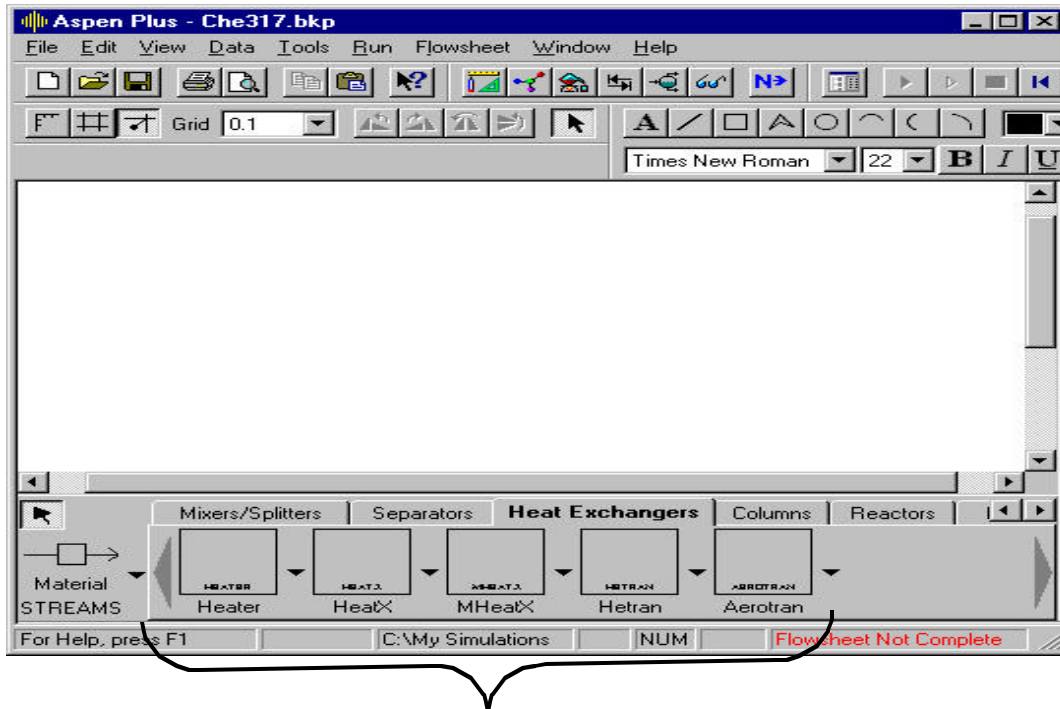
Incropera and DeWitt—pg. 582 through 597

Coulson and Richardson---

Geankoplis—pg. 267 through 275

Procedure

Logon to the AspenPlus system and start a blank simulation. The flowsheet area should appear. (Refer to “Aspen Plus Setup for a Flow Simulation” if you need help)



Shown above is the Heat Exchanger subdirectory. It shows the five options of heat exchanger types. These represent five different calculation methods, not physically different heat exchangers.

Heater – Basic heat exchanger, it performs simple energy balance calculations; requires only one process stream

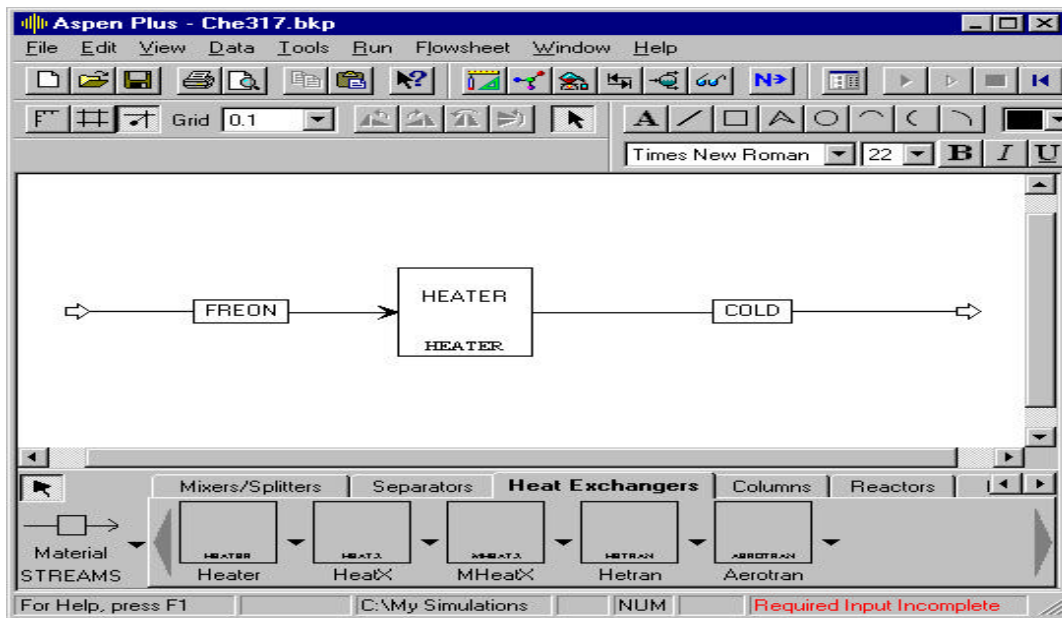
HeatX – Fundamental heat exchanger algorithms, used in rigorous design, will calculate energy balances, pressure drops, exchanger area, velocities, etc.; requires two process streams—a hot and a cold. *This block will be used for our design calculations.*

MheatX – Just like the previous block but will accept more process streams

Hetran – Heat exchanger algorithm that uses the B-JAC Hetran heat exchanger program. This will be not used since the B-JAC program is unavailable

Aerotran – Another block that uses the B-JAC program

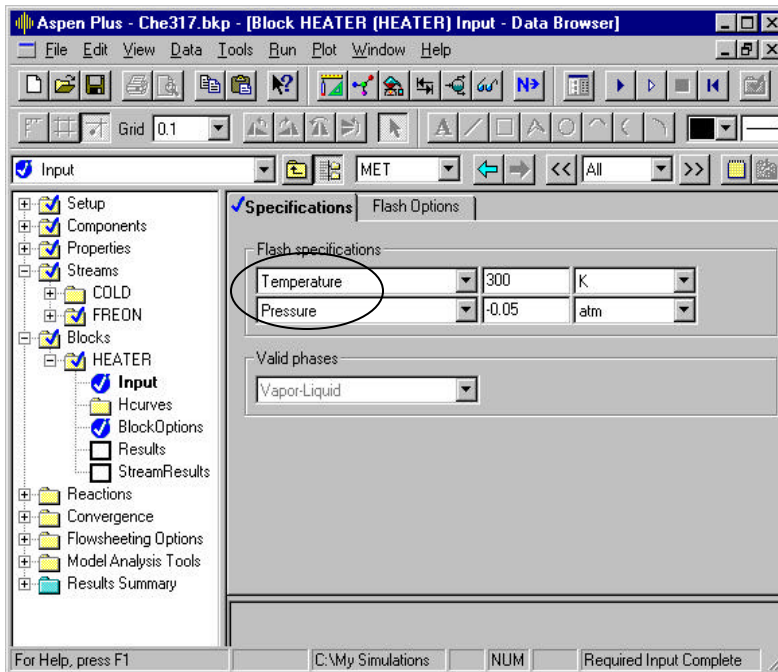
For the first step in the design, create a schematic using the **Heater** block. (Refer to “Aspen Plus Setup for a Flow Simulation” if you need help). If you click on the down arrow next to the block a set of icons will pop up. These are just different diagrams of heater blocks, they all do the same calculations and they are just for schematic purposes only. Also note that the heater block only needs one process stream for the calculations. Choose the Freon-12 stream since this is the objective of the problem statement.



Once the flowsheet is complete (Shown above), click the Next button and the title screen should appear. Give the example a title and change the units from English to Metric on the same screen. The components screen should appear next. Enter in the species used in the example. (Freon-12 is CCl_2F_2 and Ethylene Glycol is $\text{C}_2\text{H}_6\text{O}_2$) Furthermore, choose a property method for the calculations. (This example will use NRTL-RK) All of these steps are in the manual “Aspen Plus Setup for a Flow Simulation”. Click Next and the input sheet for the Freon stream will appear.

Component	Value
FREON-12	1
GLYCOL	

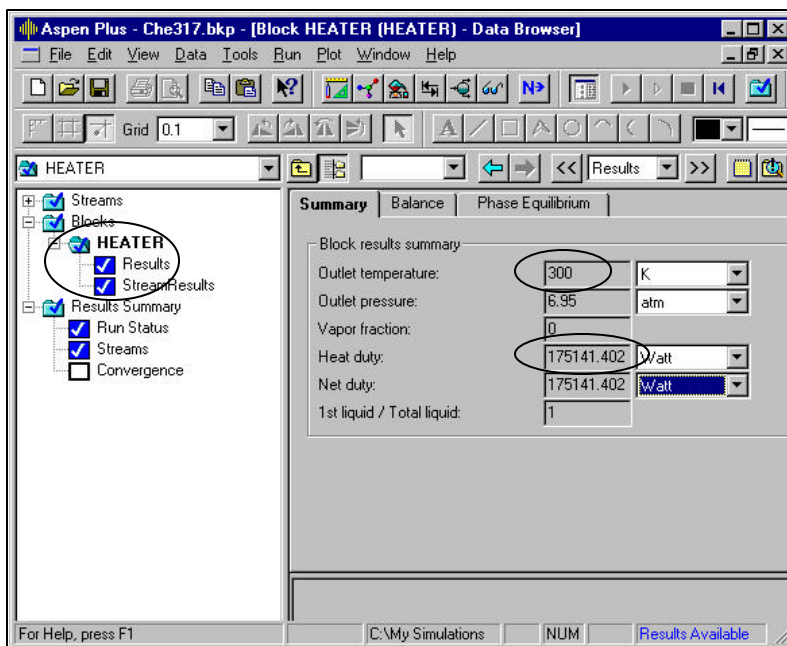
Shown at left is the input page for the Freon-12 stream. Everything is known so enter all the data from the problem statement: temperature, pressure, and the flow rate. Notice the flow rate is entered here as a molar flow rate, not a mass flow rate. Either value is fine. Click Next.



The input sheet for the Heater block will appear next. As shown at left, the block only needs two of three specifications: the outlet temperature, the outlet vapor fraction or the pressure. (shown in oval) Enter in the outlet temperature from the problem statement first. For pressure, you have two options. You can enter in the outlet pressure or enter in the pressure drop for the unit. Shown at left is the pressure drop. This value is just an initial estimate of the pressure drop. Notice that

the value is a negative number; Aspen knows the negative number means pressure drop. On the other hand a positive number would mean a value for the actual outlet pressure. Keep the “Valid Phases” set to Vapor-Liquid.

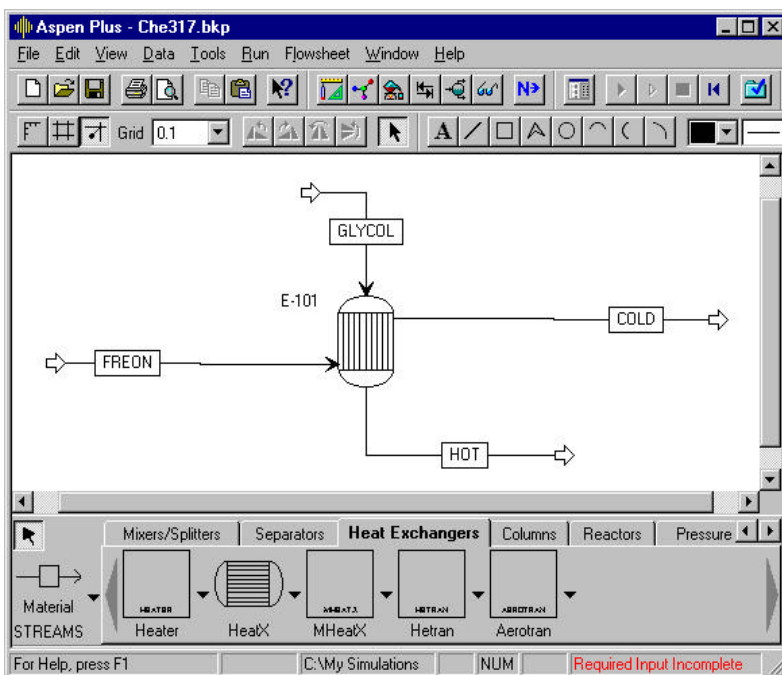
Once all the information is entered, run the simulation. Check the block results by clicking on the block folder and then the Heater folder in the results section (shown in circle). Shown below is the Heater block results page. As said before, the Heater block does simple energy balance calculations and the heat duty is shown here. (175141 Watts) You should always check this result with hand calculations. In addition, check the outlet temperature of the process stream.



Outlet temperature

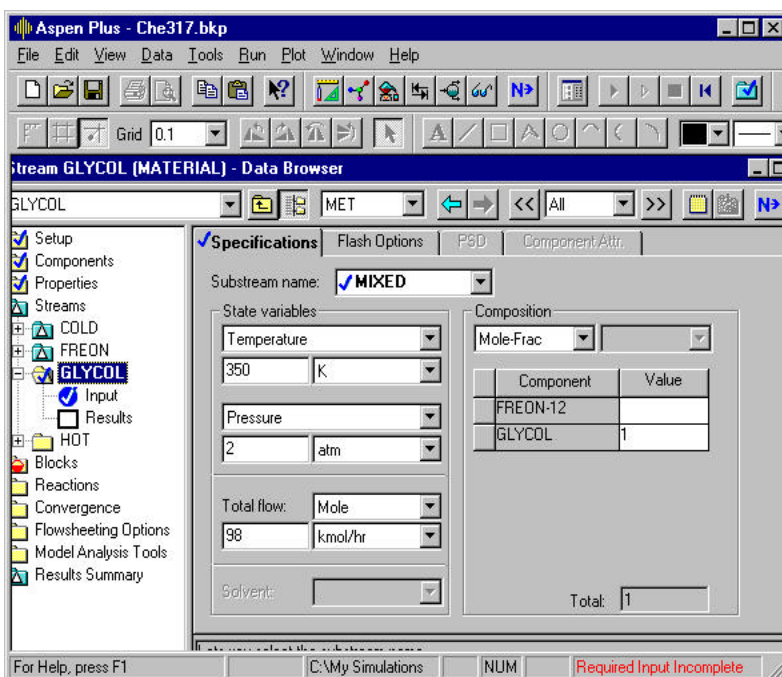
Heat Duty

Now delete the Heater block and replace with a block from the **HeatX** set of icons. Once again, you can click on the down arrow to the right of the **HeatX** space and a set of



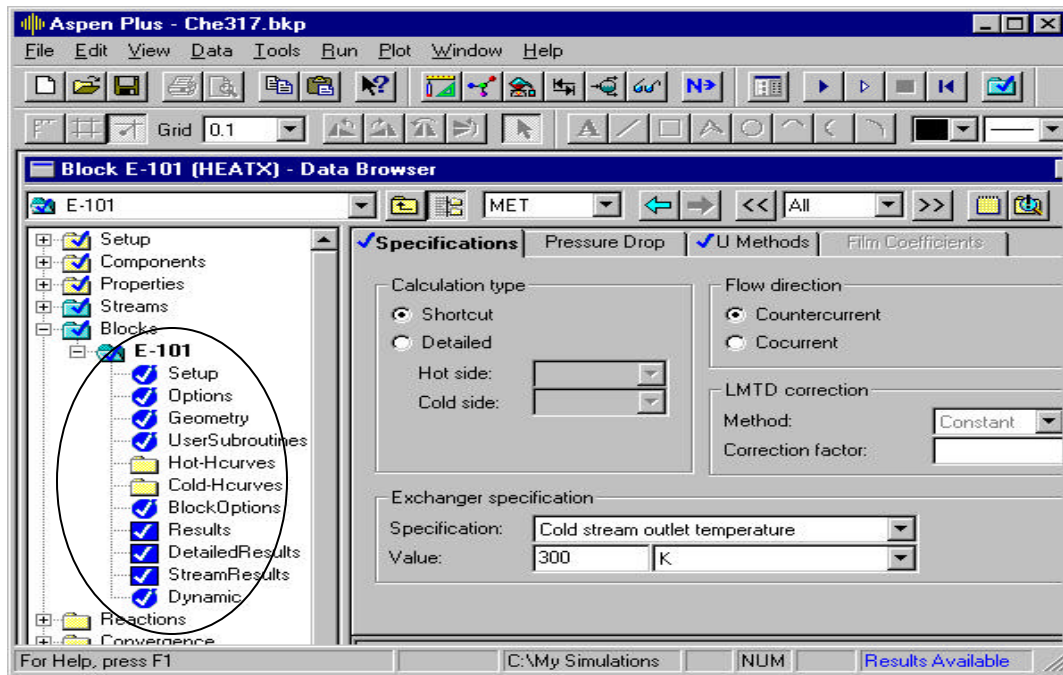
different icons pops up. Each icon represents a different TEMA shell type. Examples of these shell types can be seen in Perry's 7th edition, pg. 11-34.

For the **HeatX** block, the glycol stream will need to be added. Make sure that the proper stream is connected to the correct port of the block. The Freon-12 stream will be the cold stream and the glycol stream will be the hot side of the heat exchanger. Click Next.



Since a new stream is added, data for the new stream must be entered. (The input sheet for the glycol stream is shown at left) From the problem statement, the temperature and pressure of the glycol stream are known. However, the flow rate is unknown. This can be found using the heat duty and the heat capacity of Ethylene Glycol. Aspen can provide both of these values. (Refer to the "Estimating Properties on Aspen" manual for help) The flow rate can also be

found by using a Design-spec. Enter in the necessary information and click Next.



Shown above are the first input data sheets for the heat exchanger. All the information needed for the calculations are entered into these pages. First, let's give an overview of each subfolder. (Shown in the circle) Then each sheet will be explained in more detail.

Setup—(shown in picture) this page shows the calculation type: shortcut or detailed, the exchanger specification, whether the exchanger is countercurrent or cocurrent, and defines how Aspen will calculate the heat transfer coefficients.

Options—shows the valid phases for each stream (e.g. Vapor-liquid), also defines the convergence for calculations

Geometry—shows the input pages for the exchanger layout: tubes, diameters, baffles, etc. *Used only in the detailed calculations.*

User Subroutine—provides an interface so the user can create an algorithm to execute calculations for design (involves FORTRAN programming)

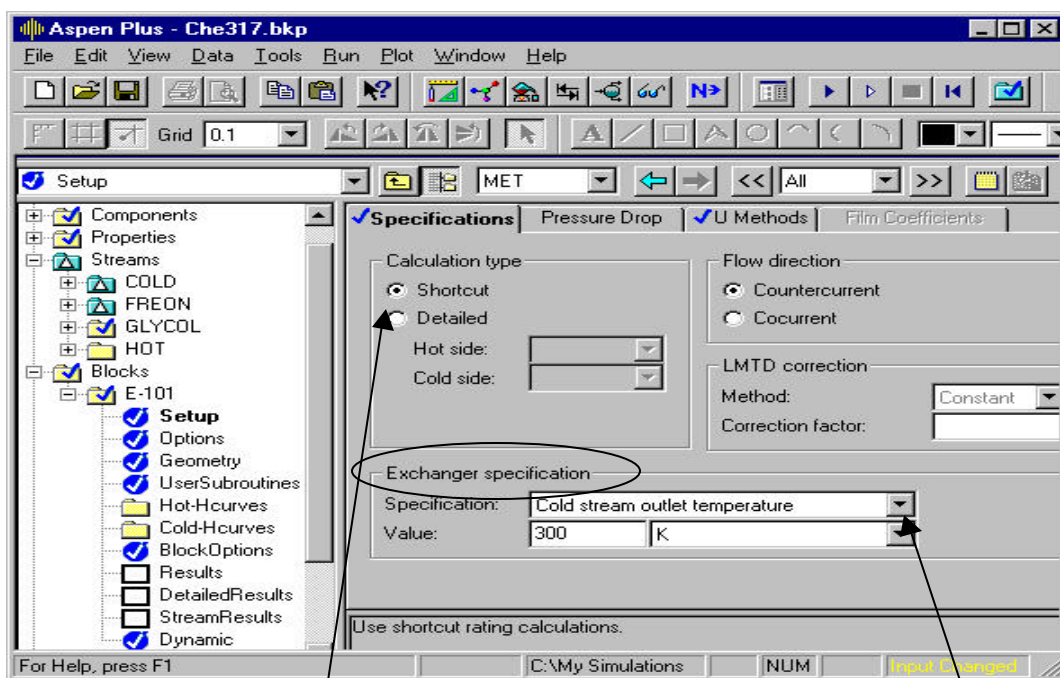
Hot H-curves—shows input pages so that Aspen will create enthalpy profiles for the hot stream, very useful for boiling/condensation processes and provides some insight on the heat transfer

Cold H-curves—same as the Hot H-curves, but creates profiles for the cold stream

Block Options—pages that show the property methods, and simulation options for the block

Dynamic—used only for dynamic mode, not steady-state

For the first calculations, we will be using the shortcut method for the calculation type.



Return to the **Setup** pages and click on the shortcut method.

The next step is to define the “Exchanger specification.” (in circle) If you click on the down arrow a list of choices will appear. Each specification gives a different calculation for Aspen to execute.

They are:

Hot/Cold stream outlet temperature—specify the outlet temperature of one of the streams, *used for situations where there is not any phase change*

Hot/Cold stream temperature change—specify the temperature increase or decrease of one of the streams

Hot outlet temperature approach—specify the temperature difference between the hot stream outlet temperature and the cold stream inlet temperature, *used with countercurrent flow*

Hot/Cold stream degrees superheat/subcool—specify the outlet temperature below the dew point or above the boiling point of a certain stream, *used in boiling and condensation*

Hot/Cold stream vapor fraction—specify the outlet vapor fraction for a certain stream (1.0 = sat. vapor and 0.0 = sat. liquid), *used for boiling and condensation design*

Cold outlet temperature approach—specify the temperature difference between the cold stream outlet temperature and the hot stream inlet temperature, *used with countercurrent flow*

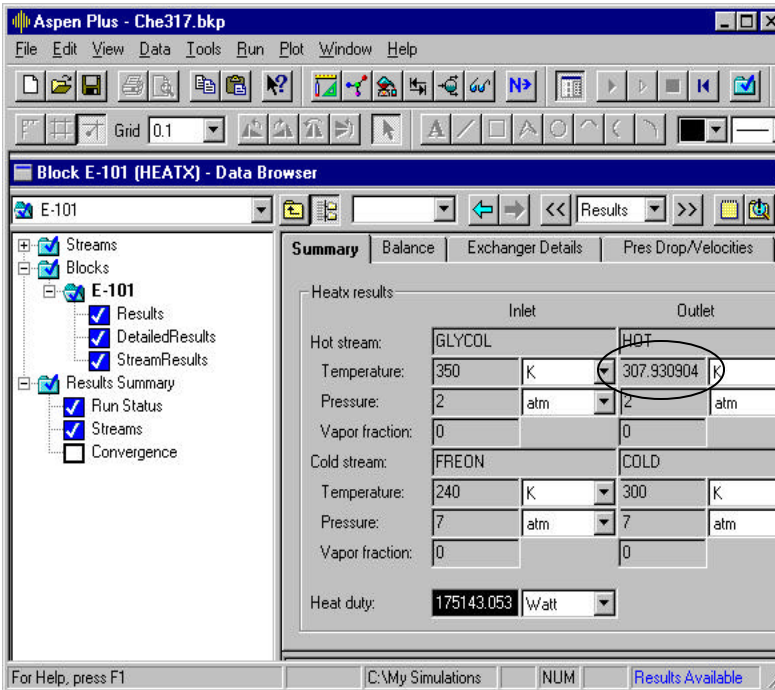
Heat Transfer Area—specify the area of the exchanger, *good for problems where the size of the heat exchanger is fixed*

Heat Duty—specify the amount of energy transferred from one stream to another

Geometry—Calculations based on exchanger layout, *good for use as a check on calculations* (Note*, you need to choose the detailed calculation type to access the geometry option)

For the example, we want the Freon-12 stream to reach a temperature of 300 K. The best specification for the calculation is “Cold stream outlet temperature”. Choose that

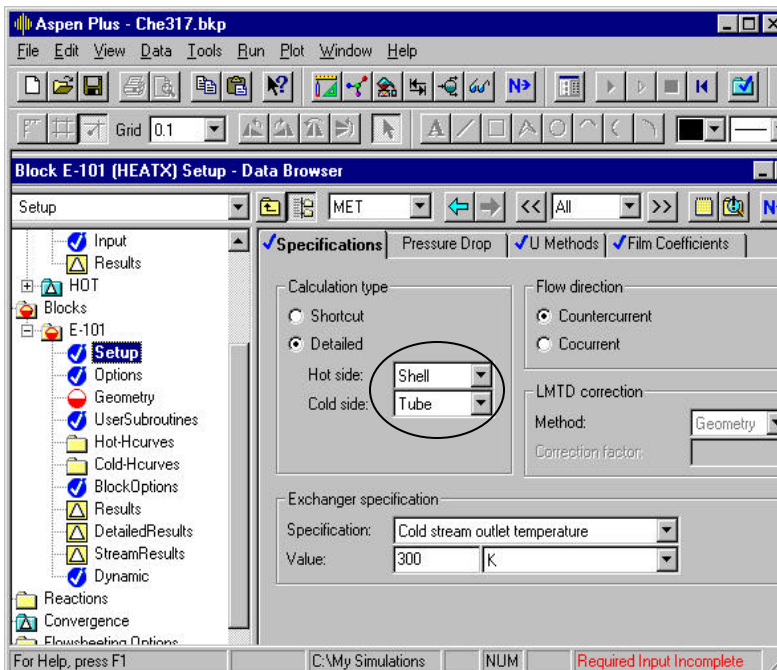
option and enter in the specified value of 300 K. (Shown above in picture) You also need to specify whether the exchanger will be countercurrent or cocurrent flow. This example will use countercurrent flow. That is all the information needed to run the shortcut method. Now run the simulation and check the block results.



The sheet shown at left is the overall results for the exchanger and can be found under the block folder of the results section. **Always** check this page each time the simulation is run. Check the inlet and outlet temperatures for both streams for temperature crossover. Check the vapor fractions as well. As you can see, the glycol stream fell below the design temperature of 310K. (in circle) The flow rate of this stream

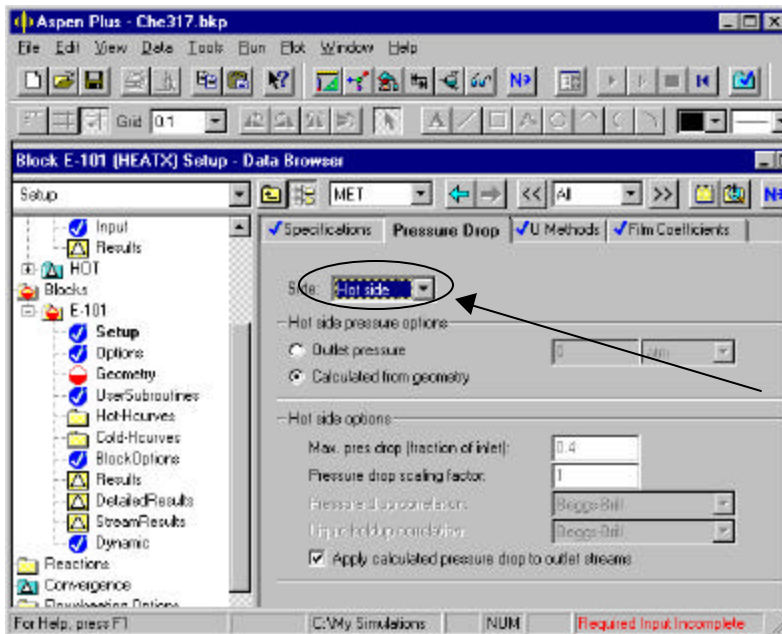
needs to be increased to keep the outlet temperature difference at 10 K. (A design-spec may be applied to find the correct flow rate) The flow rate for the glycol stream turns out to be 102.87 kmol/hr.

Once the flow rates and the heat duty are nailed down, it is time to start the detailed calculations. So return to the **Setup** page of the heat exchanger input pages.



The **Setup** page is shown at left. First of all, change the calculation type from shortcut to the detailed method. Immediately, Aspen will ask you to identify which fluid goes where, shell or tube? For this example, use what is entered in the picture. In addition, Aspen will now calculate a LMTD correction for the exchanger. The default setting is the correction based on the geometry and that will be fine.

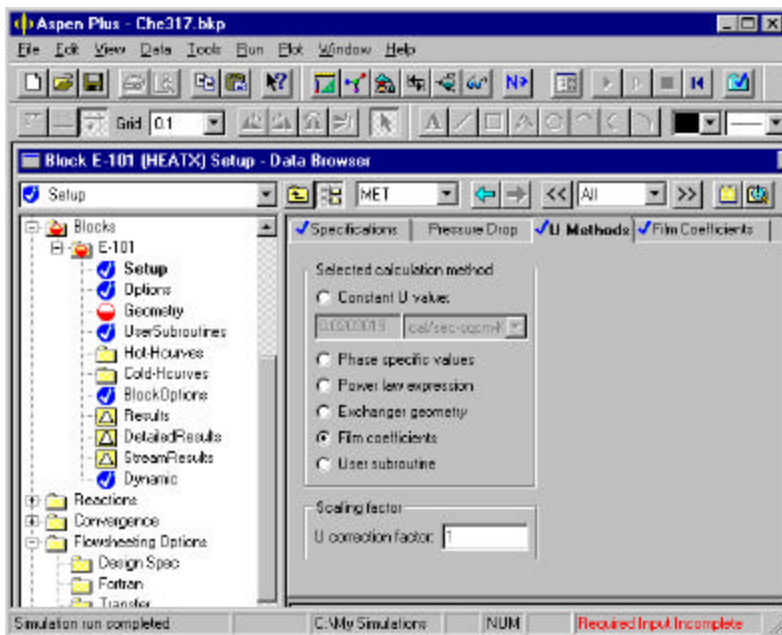
Now click on the **Pressure drop** tab at the top of the page and a new input page will



appear. Here, you need to specify how Aspen will calculate the pressure drop of the heat exchanger. We will have Aspen calculate pressure drop from the geometry of the heat exchanger. This option is shown at left and is the preferred option.

Note*: Both the hot and cold side have to be specified for the pressure drop calculations.

Now click on the **U methods** tab at the top of the page.



The input page for the overall heat transfer coefficient appears next. (Shown at left) You need to specify how Aspen will calculate the U-value for the exchanger. There are several options explained below. For this example, the U-value will be calculated from "[Film coefficients."

This option will require more input and that page will be shown next.

Constant U-value—Aspen uses a constant value in calculations; the value is supplied by the user

Phase specific values—Aspen will use default values given for specific heat transfer situations (e.g. boiling liquids, liquid in shell to liquid in tubes, condensation, etc.; these default values can also be changed by the user to apply to new situations)

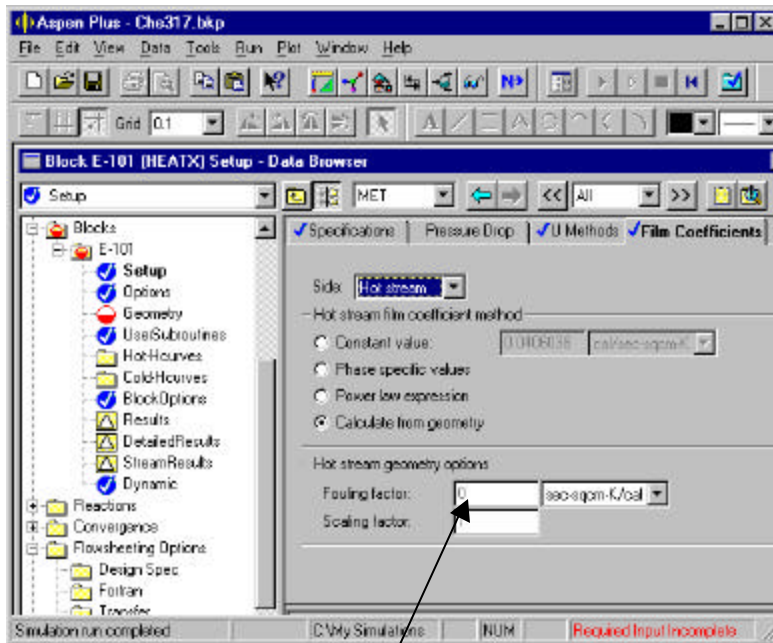
Power law expression—Aspen uses a scaling factor with a known overall H.T. coefficient and a known flow rate from a similar heat transfer situation; the user must supply both values

Exchanger geometry—Aspen calculates an average U-value from the exchanger geometry using exchanger algorithms

Film coefficients—the U-value is calculated from individual heat transfer coefficients (h_o , h_i); Note*-more input is needed on another page for this option

User Subroutine—gives input pages so the user can supply a FORTRAN algorithm for U calculations

Now click on the **Film coefficients** tab at the top of the screen.



The input page is shown at left. Here is where you need to specify how Aspen will calculate the individual heat transfer coefficients. (Note*: This page only needs to be filled out if the U-value is calculated from “Film coefficients”.) This page is very similar to the **U methods** input page and fortunately the calculation options are the same. The example will use the “Calculate from geometry” option.

Notice, however, that both

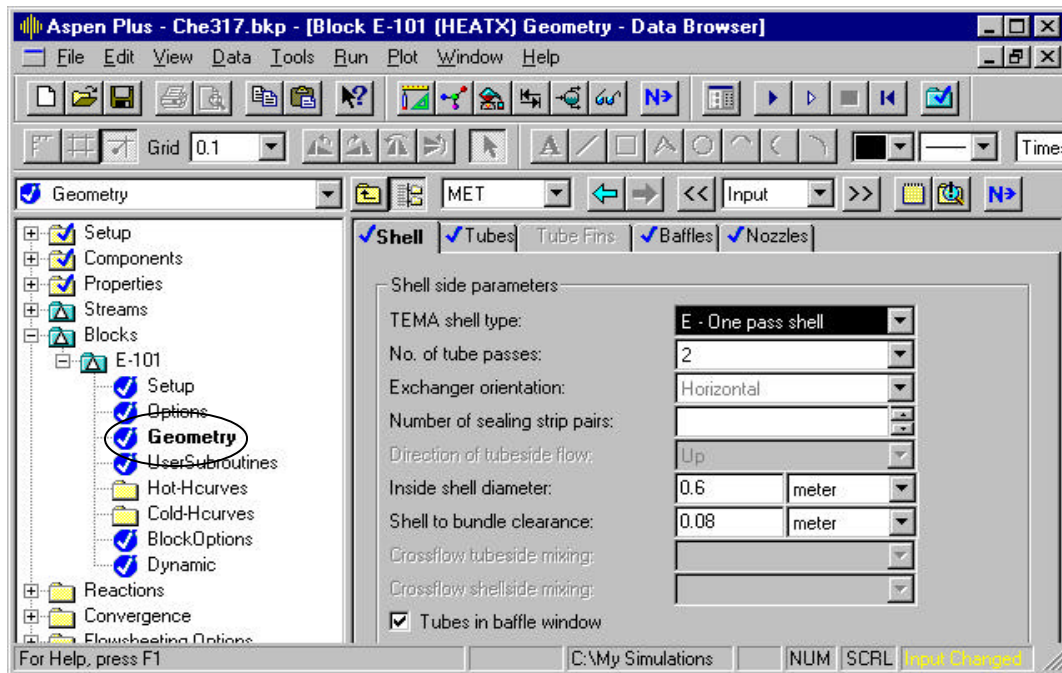
sides (hot and cold) of the exchanger need to be specified. It is also a good idea to add some fouling factors for each stream in the space provided. The example does not use any fouling factors, but if some values for the system can be found in the literature, enter them here.

After the heat transfer coefficient calculations are specified, the next step is to set out the geometry of the heat exchanger. Since Aspen does not do every calculation, hand computations will be needed. You, the user, must supply the number of tube passes, the shell diameter, the number of tubes, the length of the tubes, the inside and outside diameters of the tubes, the pitch, the material of the tubes, the number of baffles and baffle spacing. Some of the previous material is arbitrary and doesn't really need to be calculated. However, the heat transfer area needs to be estimated, at least for the first simulation run, in order to find the number of tubes. Area can be found by using the simple equation of $A = Q / U \Delta T_{LM}$. The heat duty (Q) was found by Aspen and the log mean temperature can be easily calculated. (Refer to reference five) To find the area, you need to find a value for the overall heat transfer coefficient. Refer to the literature for U-value estimates for the situation in the example. (See references 3, 4, 5, and 6) A typical value for our situation was around $150 \text{ W/m}^2 \text{ K}$ in Coulson and Richardson. This gave an area of approximately 20 m^2 . The next step is to find the number of tubes and shell diameter. Set the dimensions of one tube by specifying the tube size and length. Usually you should start with 1-inch pipe if the fluid is a liquid and 1.5 inch pipe if the

fluid is a vapor. Since Freon-12 is in the liquid state in this example, 1-inch tubes will be used. The length is not important, but make it practical. (i.e. the length should not be five miles! ☺ A typical range is 8-20 ft, 2-8 m) Once the dimensions of one pipe are set, the number of tubes can easily be found by dividing the total area by the area of one tube. The shell diameter will be calculated based on the number of tubes. One correlation given by Coulson and Richardson is:

$$D(\text{tube bundle}) = \text{O.D.} \cdot (N_t / k)^{1/n}$$

Where O.D. is the outside diameter of one tube, N_t is the number of tubes, and k and n are constants dependent on the number of tube passes. Since this example uses two tube passes: $k = 0.249$ and $n = 2.207$. This calculation only gives the size of the tube bundle and not the shell diameter. The shell diameter is now found from adding the tube bundle diameter and the clearance between the tube bundle and the shell. The clearance space depends on the exchanger type, but typically ranges from 10 mm to 90 mm. Refer to reference three or five for clearance values.



Now begin the layout by clicking on the geometry subfolder at the left-hand side of the screen. The screen above is the input page for the shell dimensions. Enter in the appropriate information.

TEMA shell type—choose the type of shell, one-pass, two-pass, split-flow, etc.

Examples of each shell type can be seen on page 11-34 of Perry's 7th edition.

of tube passes—choose the number of tube passes, *two is generally used*

Exchanger orientation—choose either horizontal or vertical tube orientation. (Note* if vertical orientation is chosen, the user must also specify the fluid flow direction)

of sealing strip pairs—not needed for calculations

Inside shell diameter—enter the shell diameter with the appropriate units

Shell to bundle clearance—enter the clearance between the inside shell diameter and the tube bundle, refer to literature for appropriate spacing

The tube-input screen will be next (shown below). Values are entered in the spaces provided as done before.

Aspen Plus - Che317.bkp - [Block E-101 (HEATX) Geometry - Data Browser]

File Edit View Data Tools Run Plot Window Help

Grid 0.1

Geometry

Setup Components Properties Streams Blocks E-101

Setup Options Geometry UserSubroutines Hot-Hcurves Cold-Hcurves BlockOptions Results DetailedResults StreamResults Dynamic

Shell Tubes Tube Fins Baffles Nozzles

Select tube type

☒ Bare tubes ☐ Finned tubes

Tube layout

Total number: 26 Length: 5 meter

Pattern: Triangle Pitch: 0.04175 meter

Material: Carbon Steel Conductivity: kcal-m/hr-sqm-K

Tube size

☐ Actual ☒ Nominal

Inner diameter: 1.049 in Diameter: 1.0-IN

Outer diameter: 1.315 in Birmingham wire gauge (BWG): 20

Tube thickness: meter

For Help, press F1

C:\My Simulations NUM SCRL Input Changed

Select tube type—either bare or finned type tubes; *generally, bare tubes are used*

Total number—specify the total number of tubes

Length—specify the total tube length, includes all passes

Pattern—specify the tube layout, either triangular or square pitch, *generally, triangular pitch is used*

Pitch—enter in the distance between tube centers within the tube bundle; *generally, the pitch is 1.25 times the outside diameter of one tube*

Material—choose the material of the tubes; Aspen has many materials to choose from or one can be provided by the user

Conductivity—enter in the thermal conductivity of the material chosen. Note* if this space is left empty, Aspen will use a default conductivity for the material from its data-banks

Tube size—specify the inner and outer diameter of one tube. Notice that nominal pipe sizes can be used; the user only need to supply the size and schedule number. (Note*: Aspen has a small data-bank of nominal pipe sizes so literature values may have to be used, *the best recommendation is to use literature values for pipe sizes*, see reference 6 pg. 10-72 through 10-74)

After entering information on the tube input page, specifying the baffle characteristics will be the next step.

Shown below is the input sheet for the baffles. If at all possible, results from hand calculations should be entered in the spaces provided. If not, there are simple “rules of thumb” for design on Aspen. Each space is explained below.

The screenshot shows the Aspen Plus software interface for the 'Block E-101 (HEATX) Geometry - Data Browser'. The 'Baffles' tab is active, displaying the following input fields:

- Baffle type:** Segmental baffle (selected), Rod baffle
- Segmental baffle:**
 - No. of baffles, all passes: 10
 - Baffle cut (fraction of shell diameter): 0.25
 - Tubesheet to 1st baffle spacing: 0.5 meter
 - Baffle to baffle spacing: (blank) meter
 - Last baffle to tubesheet spacing: 0.5 meter
 - Shell-Baffle clearance: (blank) meter
 - Tube-Baffle clearance: (blank) meter

Baffle type—choose either segmental baffle or rod baffle; segmental baffles are typical ‘wall’ baffles while rod type are simply rods that cause more turbulent flow within the shell; each baffle type has a separate input sheet (the segmental baffle input is shown above); examples of baffle types can be seen on page 11-42 of Perry’s 7th edition.

No. of baffles, all passes—specify the number of baffles in the exchanger, if the exact design number is not known a good starting value is twice the length of the tubes in meters, for example if the tube length is five meters then enter in ten baffles. More baffles can be added to increase the heat transfer coefficients, but make sure the pressure drop is kept within acceptable limits.

Baffle cut—specify the fraction of cross-sectional area for shell fluid flow; for example, shown above is a value of 0.25 which means that one baffle covers 75% of the shell cross-sectional area while 25% is left for fluid flow. *The baffle cut must be between 0 and 0.5.*

Tubesheet to 1st baffle spacing—enter in the length between the tube sheet and the 1st baffle

Baffle to Baffle spacing—specify the baffle spacing (See aside below)

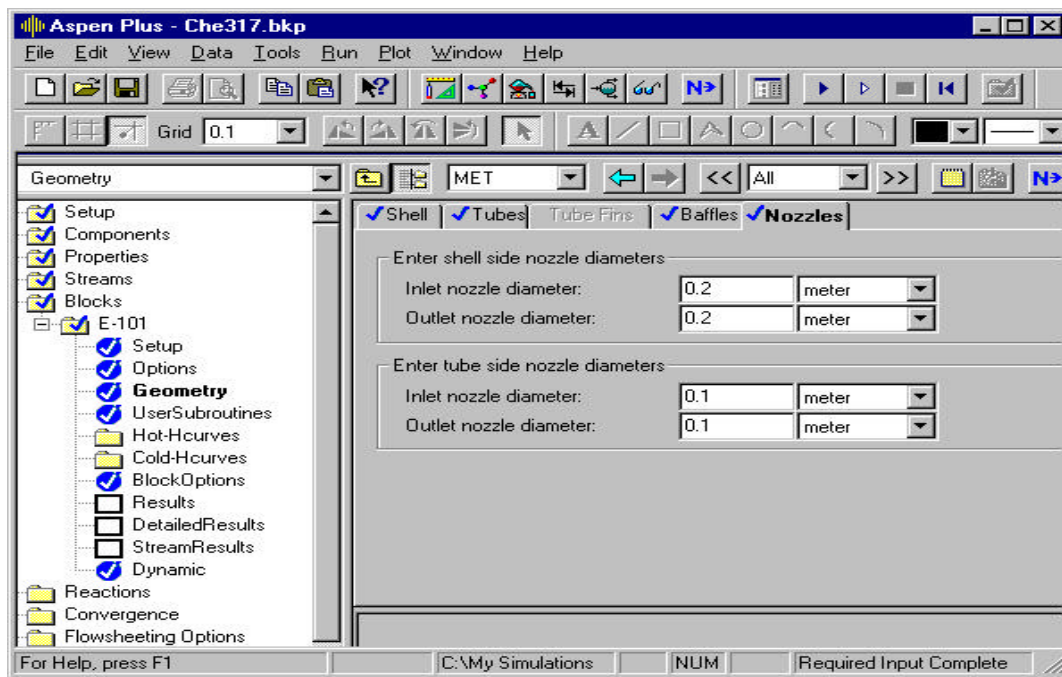
Last Baffle to tubesheet spacing—enter in the length between the last baffle and the tubesheet

(Aside –Baffle spacing in general—the previous three spaces setup the baffle spacing, two of the three spaces need to be filled for the simulation to run. If the baffle spacing is not known at the start of the simulation, the best way is to choose spacing between the tubesheet and the first/last baffle. Then Aspen will automatically calculate the inner baffle spacing)

Shell-Baffle clearance—specify the diametric distance between the shell and the outside of the baffle; this information is not necessary for simulation and can be left blank

Tube-Baffle clearance—specify the diametric distance between the tube and the tube hole in the baffle; this information is not necessary for simulation and can be left blank

When a blue check mark appears near the “baffle” tab, the input sheet is complete. The next page is the “nozzles” input sheet (shown below). Each space is explained below. Once again, if at all possible, use results from hand calculations.



Shell inlet nozzle diameter—specify the diameter of the shell inlet nozzle; refer to reference two for the diameter; if literature values are unavailable, a good starting value is one fourth the shell diameter for liquid flow and half the shell diameter for vapor flow

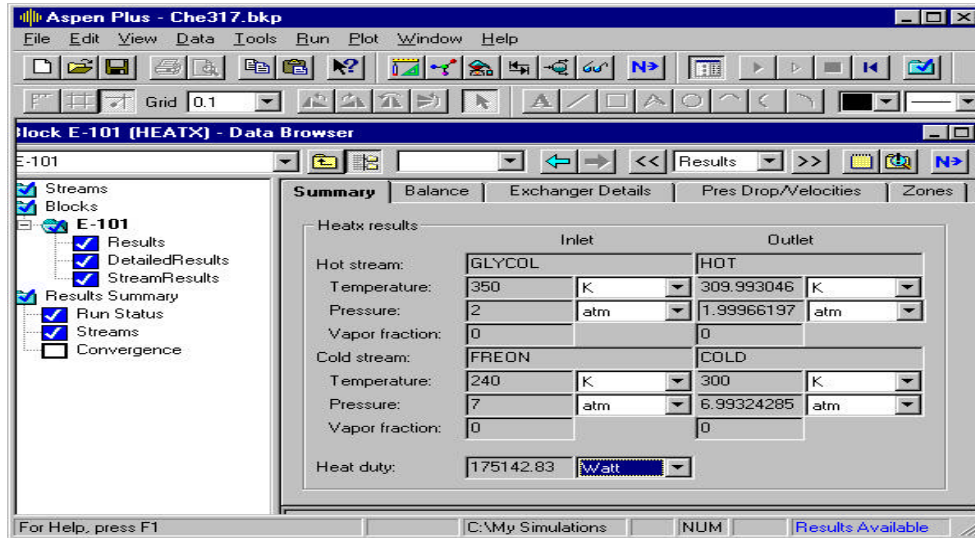
Shell outlet nozzle diameter—specify the diameter of the shell outlet nozzle; *this value should be the same as the inlet diameter when there is not a phase change*

Tube inlet nozzle diameter—specify the diameter of the tube inlet nozzle; again refer to reference two; if literature values are unavailable, a good starting value is one fifth the shell diameter for liquid flow and one fourth the shell diameter for vapor flow

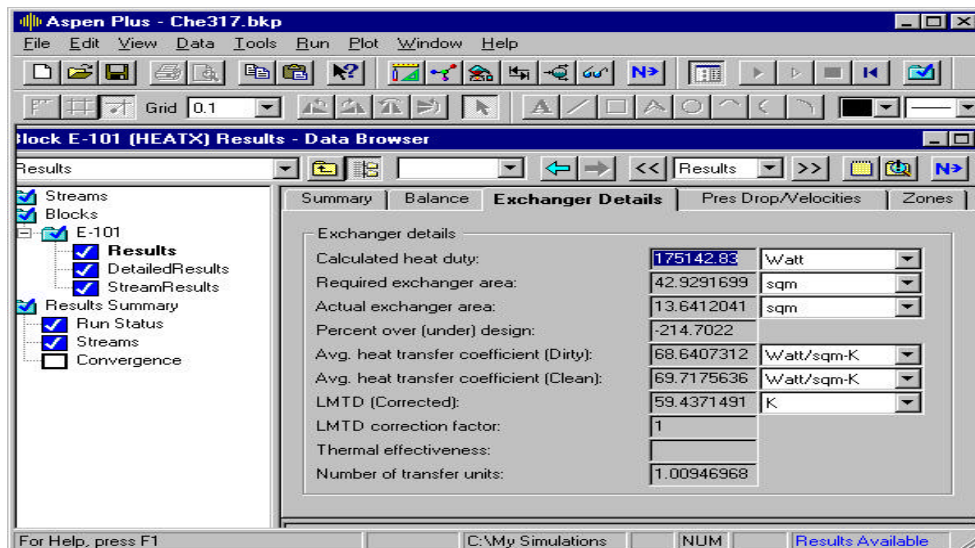
Tube outlet nozzle diameter—specify the diameter of the tube outlet nozzle; *this value should be the same as the inlet diameter if there isn't any phase change*

Enter values for each space and the nozzle-input sheet will be complete. The simulation is now ready to run. Click Next and run the simulation. (If some of the required input is incomplete, then Aspen will take you to the appropriate input sheet where the input is needed)

The **Summary** results page for the heat exchanger is shown below. You should **always** check this page. Make sure the inlet and outlet temperatures are as desired, as well as the vapor fractions. In this example, there was not a phase change so the inlet and outlet vapor fractions should be the same.

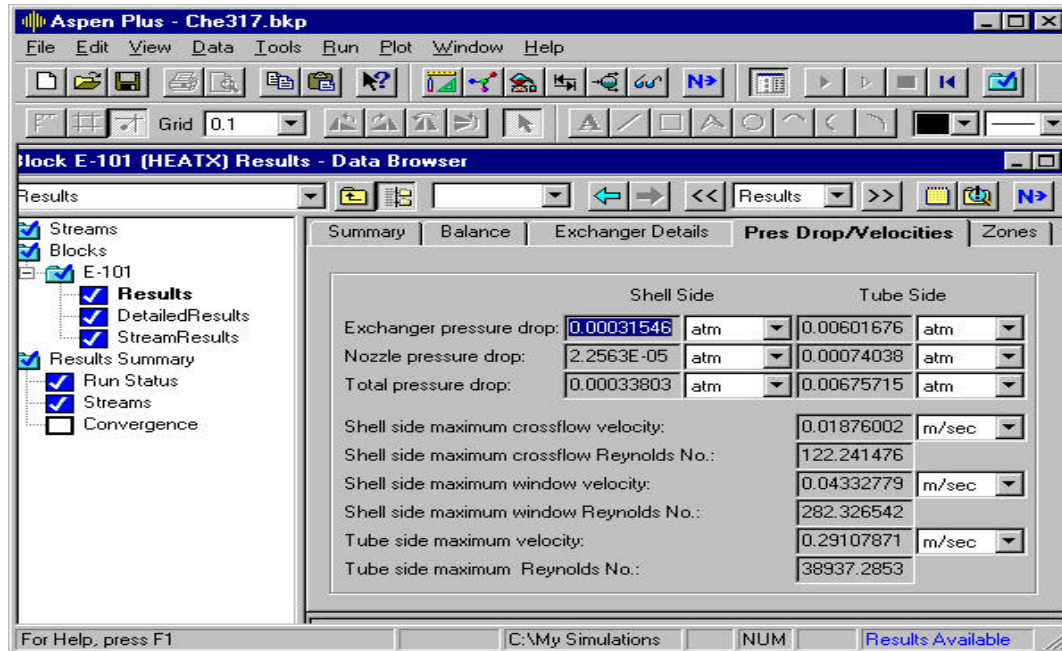


Now check the **Exchanger Details** page (shown below). Make sure to look at the required exchanger area and the actual exchanger area. As you can see, the actual area is



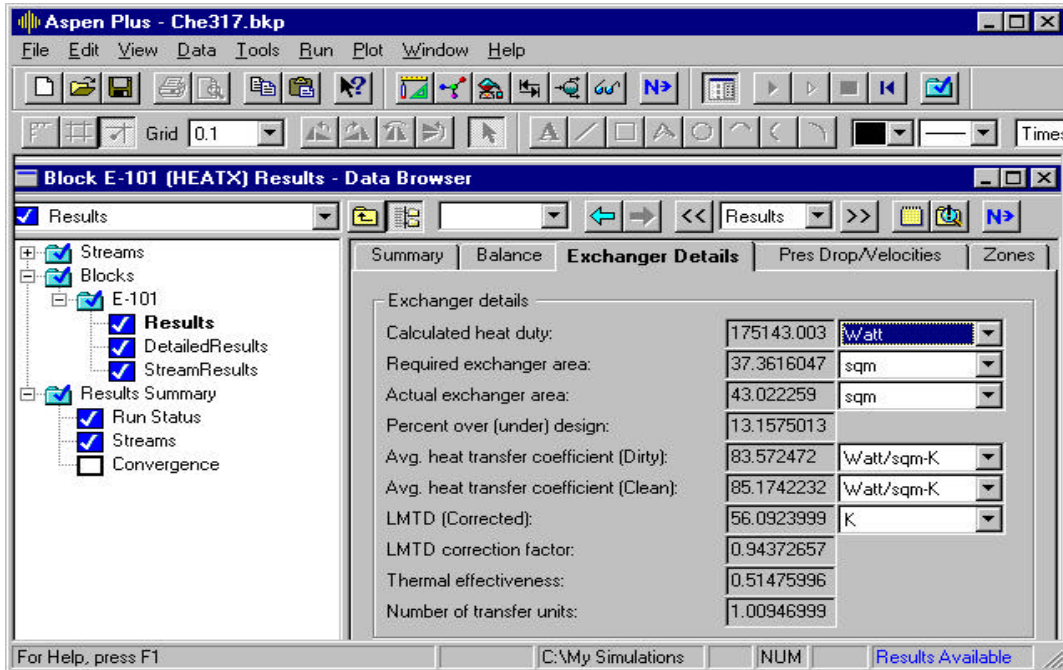
well below the required area. You can also notice the calculated heat transfer coefficient. Aspen calculated a value of $68.6 \text{ W/m}^2 \text{ K}$, which is below the estimated value of $150 \text{ W/m}^2 \text{ K}$. The results show that more area is needed for heat transfer and the heat transfer coefficients need to be increased. However, before changing the geometry of the exchanger, you need to check the rest of the result pages, starting with the pressure drop.

The **Pressure Drop/Velocities** results page is shown below. **Always** check this page to check that the pressure drops are within limits. In addition, check the velocities for both the tube and shell sides. Refer to the literature to find recommended velocities for flows through heat exchangers. For this example, these results turned out well. In fact, the shell side velocity can be increased to increase heat transfer.

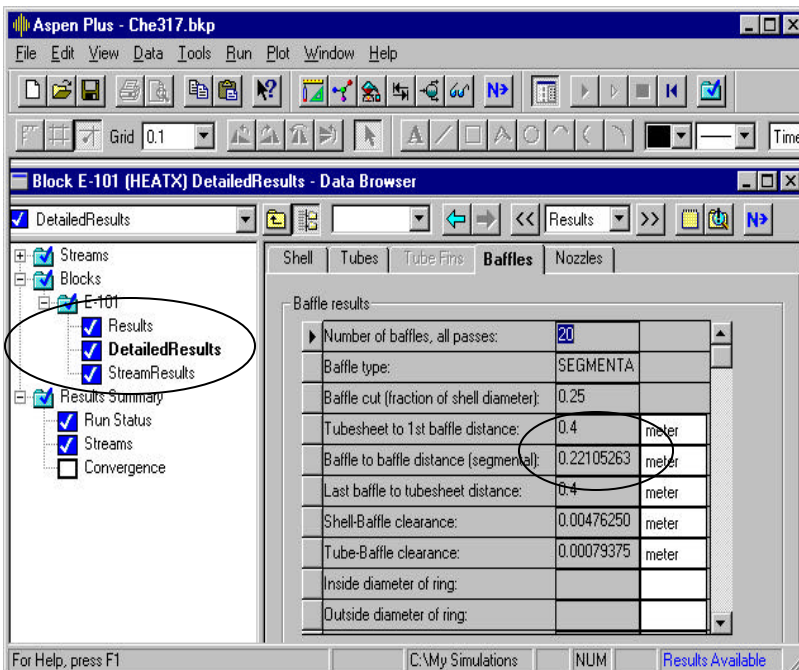


Now we need to go back and change the exchanger layout to increase the heat transfer area. Use the required area that Aspen has calculated to find a new value for the number of tubes needed. (Note*, it is possible that iterations will be needed to find the design area. To converge on the design area faster it is a good idea to use 10% over the required area.) Another good idea would be to change the baffle layout to increase the heat transfer coefficients. Once that is done, run the simulation again and recheck **all** the results pages. Make the necessary changes to the exchanger until the actual area is equal to or greater than the required area, the pressure drops are within limits, and most importantly, the design outlet temperatures are met.

Here is the “Exchanger Details” page for one final design option. The actual area is 13% over the required area so the geometry of the exchanger is finished. (It is always smart to over design by 10-20%, especially on Aspen) The heat transfer coefficient seems fine; however, this value should be checked with hand calculations.



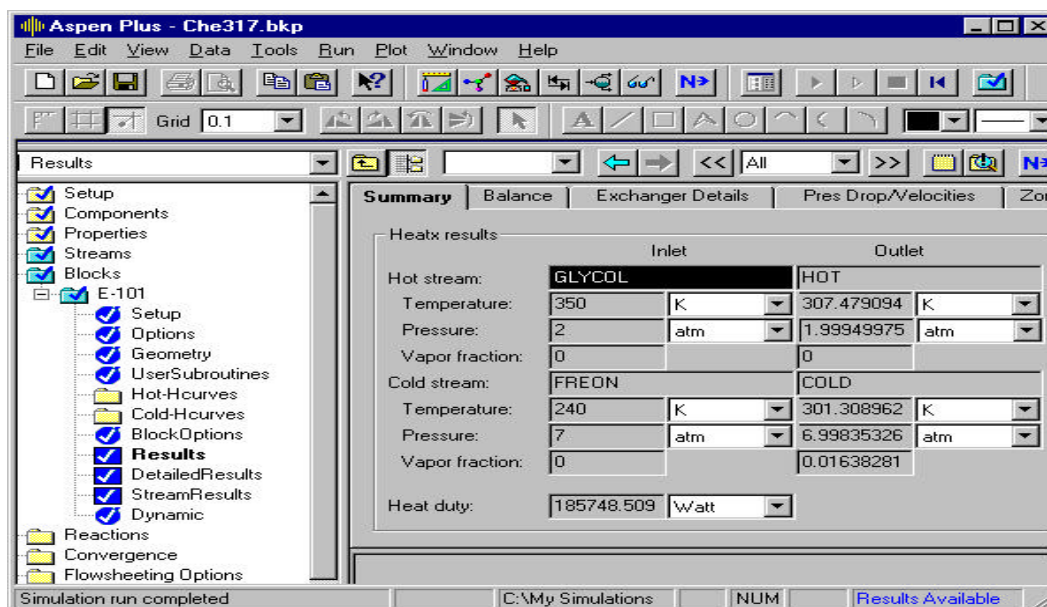
More results for the exchanger can be seen under the “Detailed Results” section under the Results page. Shown below is the baffle results page.



As said before, Aspen calculated the baffle spacing. (BS = 0.22 meters) Detailed Results also show results for the shell, tubes, and nozzles.

Now to check if the geometry is accurate, the simulation should be run with the calculations based on the geometry. Return to the **Setup** input page for the heat exchanger. Change the “Exchanger Specification” to Geometry and rerun the simulation. Now the calculations will be based on the area and layout of the exchanger. Check the results, if they are not the same as before, then the layout needs to be changed accordingly. Granted, the results will not be exactly the same since the area is 10% over designed. This will cause more energy transfer between the two fluids; so the outlet temperatures will be slightly different than the design temperatures.

The example results for the geometry calculations are shown below. As you can see, the outlet temperatures changed slightly, and the Freon stream is starting to vaporize. However, the objectives of the problem statement were still accomplished.



Once the exchanger layout is checked, return to the **Setup** page and change the Exchanger Specification back to “Cold stream outlet temperature.” You may want to change the Exchanger Specification to another option, set that value, and rerun the simulation. For example, you may choose “Hot stream outlet temperature” and set that to the design outlet temperature of the Ethylene glycol, which is 310 K. Rerun the simulation and check the results. Once again the results should be very similar. This is a good way to check your design. Remember, after each simulation run, always check the **Summary**, **Exchanger Details**, and the **Pres. Drop/Velocities** pages to make sure the exchanger is within design limits. Once the design is finished, print out the input pages as well as the results for the heat exchanger for the design report.

The next section will look at boiling and condensation unit operations. These two processes are very important in Chemical Engineering plant design and there are some “tricks” to running them on Aspen. For some reason Aspen deals with these processes the same as a normal heat exchanger when boiling and condensation are very complicated forms of heat transfer.

References

1. Aspen Plus Simulator 10.0-1. User Interface (1998).
2. Branan, Carl. Rules of Thumb for Chemical Engineers. 2nd ed., Gulf Publishing Company, 1998.
3. Coulson and Richardson. Chemical Engineering Fluid Flow, Heat Transfer and Mass Transfer. Volume 1, 5th ed., Butterworth and Heinemann, 1996.
4. Geankoplis, Christie J. Transport Processes and Unit Operations, 3rd ed., Prentice Hall, 1993.
5. Incropera and Dewitt. Fundamentals of Heat and Mass Transfer. 4th ed., John Wiley and Sons, 1996.
6. Perry, P.H. and Green, D. Perry's Chemical Engineering Handbook. 7th ed., McGraw-Hill Co., 1987.
7. Reid, Prausnitz, and Poling. The Properties of Gases and Liquids. 4th ed., McGraw-Hill Book Co., 1987.