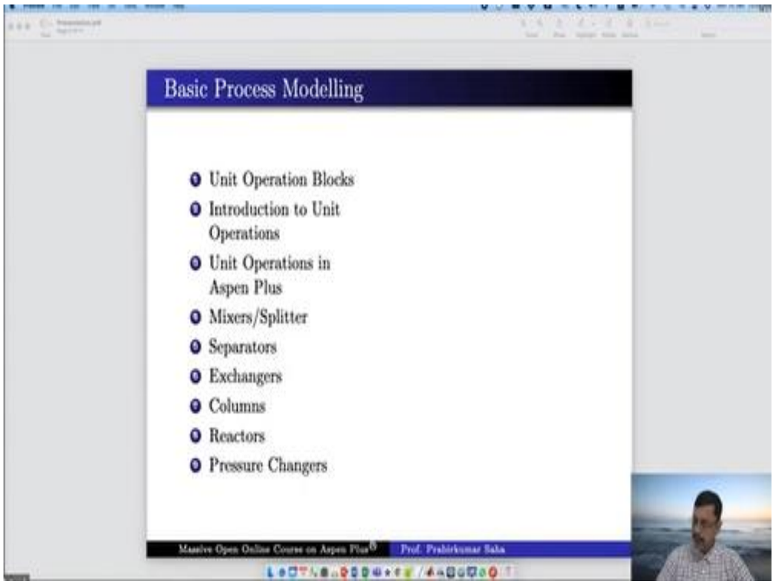
Aspen Plus Simulation Software - a Basic Course for Beginners
Prof. Prabirkumar Saha
Department of Chemical Engineering
Indian Institute of Technology, Guwahati

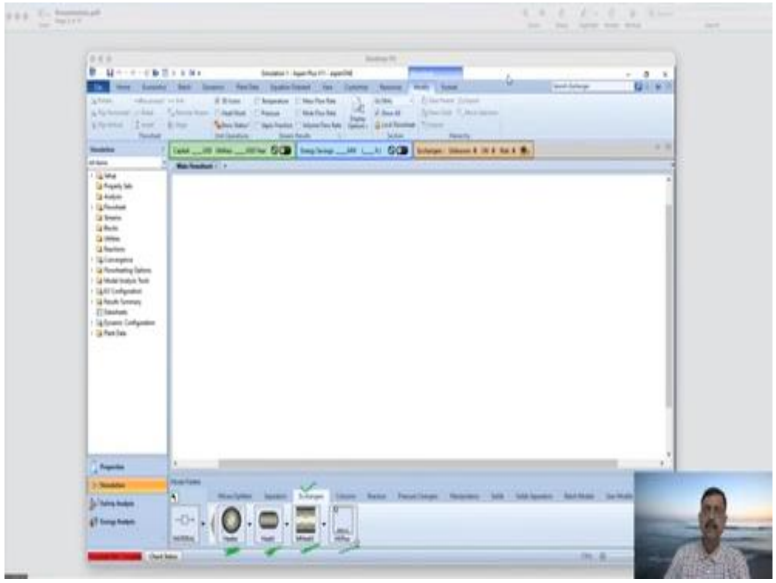
Lecture - 05
Using Model Pallete – Exchangers

(**Refer Slide Time: 00:36)**



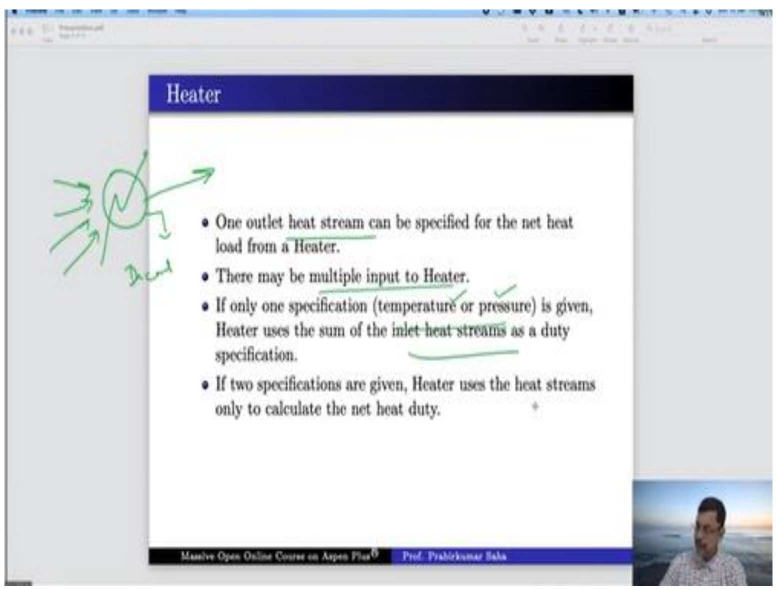
Welcome to the massive open online course on Aspen plus. In today's lecture we will cover the portion of exchangers that means the heat exchangers.

(Refer Slide Time: 00:47)



If you look at the Aspen plus simulation window, you will find the exchangers over here. There are four types of exchangers Heater, Heat X, M Heat X, and HX Flux. We will discuss them one by one. First, we will begin with heater.

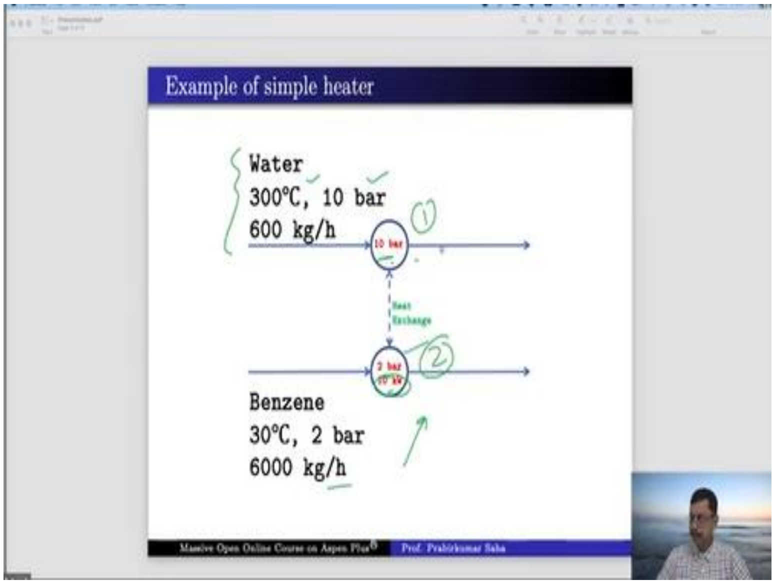
(Refer Slide Time: 01:09)



A heater can be represented by the symbol of this. It has only one outlet heat stream that can be specified for net heat load. There may be a second outlet stream, but that will be a decant stream, not the process stream. And there may be multiple inputs to the heater. Now, if only one specification is given, either temperature or pressure, then heater uses some of the inlet heat streams as a heat duty specification.

But if both of them are given, then heater uses the heat streams only to calculate the net heat duty. We will understand this heater through an example problem.

(Refer Slide Time: 02:05)



So, this is the example of a heater. Actually, we have two heaters heater 1 and heater 2. There are two inlet streams in these two heaters. The heater 1 it has water at 300 °C and 10 bar, the flow rate of water is 600 kg/hr that is going into the heater 1. And the heater 1 operates at 10 bar. So, the inlet stream and the heater both are at the same pressure.

So, there is no pressure drop; the same is true for second heater, which operates at 2 bar, and heat duty of 10 kilowatts is given into the heater. Now the second inlet stream is benzene which is at 30 °C and 2 bar, same as the pressure of the second heater, and its flow rate is 6000 kg/hr. Now we have to see how much of temperature increase that takes place from 30 °C?

Obviously, by using a 10 kilowatt of heat duty into the second one, so, first we will simulate the second heater, and then we will combine it with the first heater.

(Video Starts: 03:35)

So, we go back to the Aspen plus simulation window. First, we have to set the properties. Let us take benzene and water. Press next use the NRTL method; press next run the property method. Now go to simulation and bring the heater over here. The material stream in material stream outlet us rename them it is a benzene stream. So, write benzene in benzene out, and this is benzene heater. Press next. It will probably ask for the heater material input. Yes, it is.

So, we have to fix the temperature and pressure, which is 30 ℃ and 2 bar to work pressure, and the flow rate is 6000 kg/hr of benzene, no water. Press next, so, it is asking for the heater input. So, heater condition is 2 bar, and 10-kilowatt heat duty will fix heat duty over here, 2 bar and 10 kilowatts of heat duty. So, specification is all done. We can now run the simulation.

Let us see how much of temperature increase happens for the benzene. Now results are available. So, go to result summary here you can see benzene out. So, inlet temperature of benzene is 30 ℃, and outlet temperature of benzene is 33.3498 ℃. So, you can see even with a 10 kilowatt of heat duty, a heater can increase the temperature of benzene only from 30 degrees to 33 point, only 3 ℃ 3.35 ℃ temperature rise occurs.

So, you can see even with 10 kilowatts of heat duty, a heater can increase the temperature of benzene only 3.35 ℃. Why? Because the flow rate is very high. It has to cater to 6000 kg/hr of benzene. So, obviously, the temperature rise will not occur that much. Now we will fix the second heater. So, the heat exchange will take place. Now the question is, from where these 10 kilowatts will come?

So, we have water stream that is having 300 ℃ temperature. So, water is at higher temperature. So, it can supply heat to the benzene stream. So, we have to look for a second heater over here. And then press the material stream this is water rename WATIN, WATOUT, WATHEAT and we have to give a heat stream. So, let us say heat stream between these two this. So, this is a heat stream.

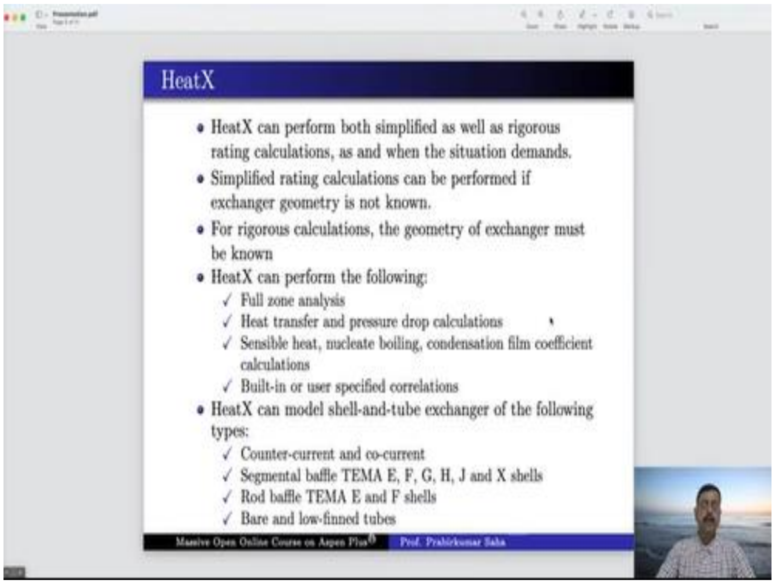
Press next so, what are inlet input? 300 ℃, 10 bar, 600 kg/hour. So, 300 ℃, 10 bar, and 600 kg/hr. Press next so, water heater, in this case, we will fix it at 10 bar pressure. There is no pressure drop, no pressure loss. Now the entire simulation is balanced, we can run it. So, we are running the simulation. Let us see what happens. So, result summary is available. Go to the stream.

Now you see the benzene output is 33.349 ℃. But water temperature reduces from 300 ℃ to 269.859 ℃. This is the outlet temperature of water. Why there is a decrease of temperature over here? Because it has given that; heat to the benzene stream. So, that is the amount of heat exchange that occurred, and that is at the rate of 10 kilowatts. If you go to the water heater, you can see the heat duty of (-)10 kilowatts.

Whereas benzene, you will see the heat duty of (+)10 kilowatts that means (-)10 kilowatts that means 10 kilowatts of heat is being sent from water heater to the benzene heater. So, this is the heat exchange is taking place. 10 kilowatt of heat is being passed from water heater to the benzene heater.

(Video Ends: 10:06)

(Refer Slide Time: 10:07)

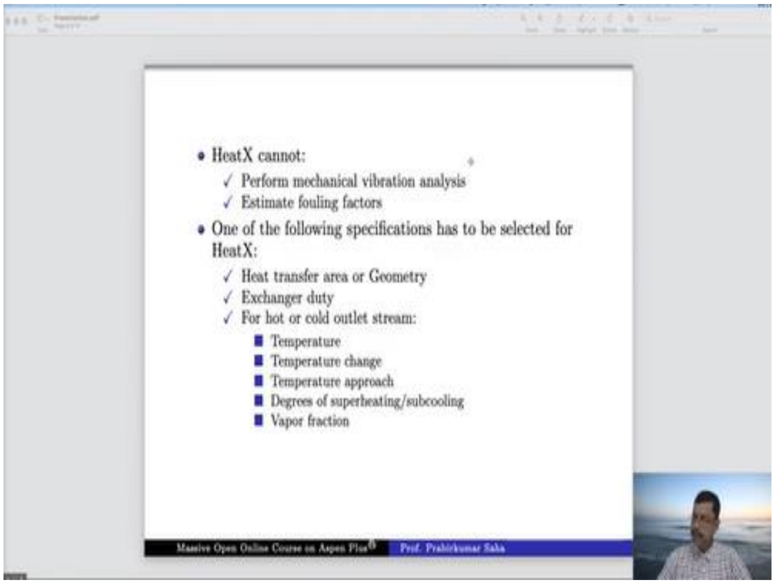


Next, we go to Heat X. That is this particular instrument. Heat X can perform both simplified as well as rigorous rating calculations as and when the situation demands. Simplified rating calculations can be performed if exchanger geometry is not known, whereas rigorous calculation can be done when geometry of exchanger is known. Heat exchanger or heat X model can perform the following.

It can do full zone analysis, heat transfer and pressure drop calculation, sensible heat nuclear boiling condensation, film coefficient calculation, built-in or user-specified correlation. And heat

X can model shell-and-tube exchangers of the following types. Co-current and counter-current, Segmental baffle TEMA E, F, G, H, J and X shells, Rod baffle TEMA E and F shells, Bare and low-finned tubes.

(Refer Slide Time: 11: 23)



Heat X cannot perform mechanical vibration, analysis and it cannot estimate the fouling factors either. Now one of the following specifications has to be selected for heat X. Heat transfer area or geometry, exchanger duty and for hot and cold streams these are the thing that has to be specified. Now it is not possible for me to give examples in each case. But I will give only one example for heat X the rest you can try yourself.

So, this is how a simple heater can be modelled in Aspen plus simulation window. Next, we will use the same example but in a proper heat exchanger model. For that, we have to use the model heat X. Heat X can perform both simplified as well as rigorous rating calculations as and when the situation demands. For simplified rating, the calculation can be performed if the exchanger geometry is unknown.

Whereas for rigorous calculation, the geometry of the exchanger must be known. We will do both of them in the example. Now, these are the things which heat X can perform. It can do full zone analysis, it can calculate the heat transfer and pressure drop of the process, it can calculate the sensible heat, nuclear boiling, condensation film coefficient, and it also can do built-in or user-specified correlations.

The heat X model, shell, and tube exchanger which is a very common type of exchanger that we use in the process industry. It can model the following types; Co-current counter-current, segmental baffle TEMA E, F, G, H, J, and X shells. Those were convergent on heat exchanger types. They know what we are specifying over here. They are all available in the heat exchanger handbook, rod baffle TEMA and F shells, bare and low-finned tubes.

But heat X cannot perform mechanical vibration analysis, and it cannot estimate the fouling factor either. One of the following specifications has to be selected for heat X among them. Either heat transfer area or the geometry the exchanger duty and for hot and cold streams, you have to fix the temperature or temperature change or temperature approach or degrees of superheating and sub cooling and or vapour fraction.

We will discuss all of them through the example of heat exchanger. Now we take up the same problem for heat X modelling. We go to the Aspen plus simulation window.

(Video Starts: 14:38)

First, we have to mention benzene and water. Press next, choose NRTL_RK, next run the property calculation. Go to simulation. Here, you have to bring in heat X. Now here you will find cold required and here hot required. So, Aspen plus wants you to feed in the colder fluid through this and hotter fluid through this. If you toggle the fluid inlets, then the calculation will be completely wrong. So, we connect the stream first, and then we have to rename.

We rename the cold fluid as benzene in, benzene out, water in, water out. Press next. It is asking for benzene in input. So, this is 30 °C, 2 bar pressure, and flow rate is 6000 kg/hr, so, it is 6000 kg/hr. Press next, the water inlet 300, 10, 600. Press next here. We get a lot of options open for us. We can do shortcut calculations. We can do shell and tube detail calculation, kettle reboiler thermosiphon, air-cooled and plate.

Now we will do the shell and tube exchanger design in this lecture. You can try out other things yourself. Now in the shortcut design, you have to specify one of them. You have to either specify the hot stream outlet temperature or hot stream outlet temperature decrease or hot outlet or cold inlet temperature difference, degrees of sub cooling vapour fraction, etcetera. Let us fix the exchanger duty, because here we have fixed the exchanger duty that is 10 kilowatts.

We wanted hour heater to operate under this condition. So, we fix up the exchanger duty as 10 kilowatts. As soon as we fix the 10-kilowatt value, it is ready to run. So, we run the simulation, and we go to see the thermal result it is ready. So, here you find water temperature reduces from 300 to 270 ℃. Whereas benzene temperature, as expected, increases from 30 ℃ to 33.3913 ℃.

And this figure it matches with the one which we calculated through heater model. Now we go to the exchanger details here, we find the required exchanger area 0.04. Now let us go to the main flowsheet and press setup, because shortcut calculation is over. Now let us go to the detailed calculation of the shell and tube exchanger. The moment I press shell and tube exchanger; it gives me the option to convert to rigorous exchanger because now it is no longer shortcut. It will be rigorous calculation.

So, these are the options I have, shell and tube kettle reboiler, thermosiphon, air-cooled, and plate. Let us work on shell and tube exchanger. Now, this select conversion method, you can ask the Aspen plus to size the exchanger itself, or you can specify the exchanger geometry which is preferred by you. Now my personal choice is let the Aspen plus size the exchanger by itself depending upon the prevailing condition. It knows about your type of fluid, the temperature difference, how much voltage you expect, etcetera.

And once the exchanger sizing is decided by the Aspen plus software you can customize, you can change the values a bit here and there to suit your purpose. So, I keep it as it is size the exchanger shell, and tube size interactively. So, I press convert so here it sizes itself. So, here it gives the option design, sizing. So, TEMA type fluid location of hot fluid it is passing through shell side.

If you want, you can keep it to tube side as well. You can change the tube, OD or pinch, tube pattern whether you want the triangular or rotated triangular or square tubes are in baffled window, yes baffle type, single segmental. You have other options also. You can choose whether the baffle cut orientation is horizontal or vertical. Now I keep all of them default. I am not changing anything at this stage.

I just press size now, let it do the optimization and size itself. And here it is, giving me an option that shell ID 8 inch, tube length 47 inches, baffle size 5.315 inch, and number of tubes per pass 47. So, it is saying we require 47 tubes per pass. And this is the kind of exchanger that it is proposing to me. Now at this moment, I accept the design and run my simulation and see the result. After seeing the result, I will take the decision where to change, so I accept this design.

Now I will do rating. What is rating? Rating will tell us whether the exchanger has been over-designed or under-designed. So, this information will be available through rating. So, I press rating and run it, so let it run. So, run result is available, so go to thermal results. You can check the voltage and other things are same. I mean, this is still 10 kilowatts, and the temperature reduction everything is done; temperature increase or temperature reduction they are also same.

Only you will find a very little decrease in pressure. Now you go to exchanger details. Here you find the required exchanger area is 0.252146 m^2. On the other hand, your actual exchanger area is 3.15866 m^2 which is quite high, 1152% over design. So, you may like to change the geometry of your exchanger just to fit your purpose. So, you go back to your setup and press specify geometry, input key geometry.

So, these are the geometry open for you. Now you may say my tubes are very long I may shorten it maybe I can work with 37 inches tube, not 47 inches. And as we have changed it, we can run the simulation once again with the reduced tube length. So, again the simulation has been done go to thermal results. So, you see, the percent over design has been reduced from 1152 to 917% still, it is quite far off. So, what we can do? We can go back here and change the number of tubes.

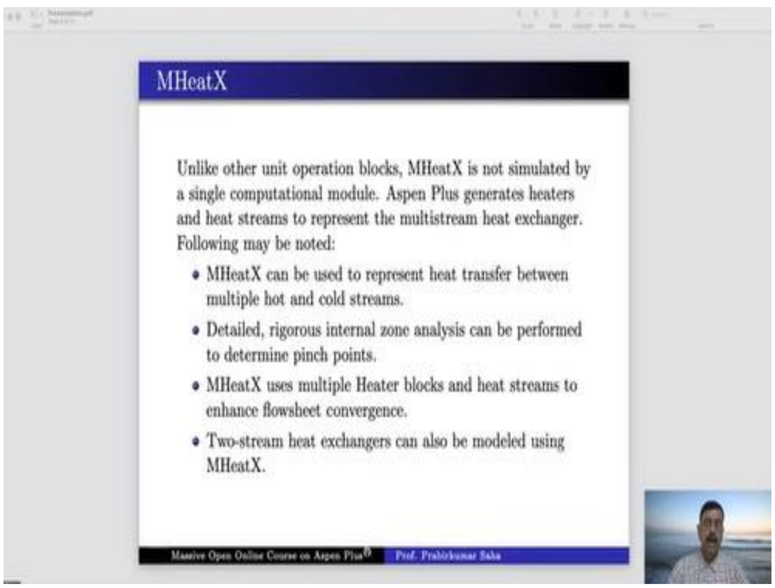
So, you may say you might say that well number of tubes 47 it is increasing the area quite high, heat transfer area. So, let us decrease it to 37. If you decrease the number of tubes, then obviously, the surface area will decrease a lot. Then again, you do the simulation, fine. So, go to thermal results it has reduced to 765. So, you can try out various other things. You can try out changing the tube ID, shell ID, changing the shell ID baffle spacing, and so many other things.

So, entire geometry of exchanger is available with you. So, you can try out various combinations, and you can try to match the required exchanger area and actual exchange area. When these two data are very close to each other, then your design will be acceptable. So, this is how you can design your heat exchanger. And later, once you are happy with your heat exchanger design, you can go and run the simulation.

So, you run the simulation and check the thermal result here, you see the required area and actual area. They have come very close to each other. Now the next block, we will have M Heat X. Now, what is M Heat X? If you over around, it will say multi-stream heat exchanger model, LNG exchanger, cold boxes, etcetera and performs zone analysis.

(Video Ends: 27:08)

(Refer Slide Time: 27:09)

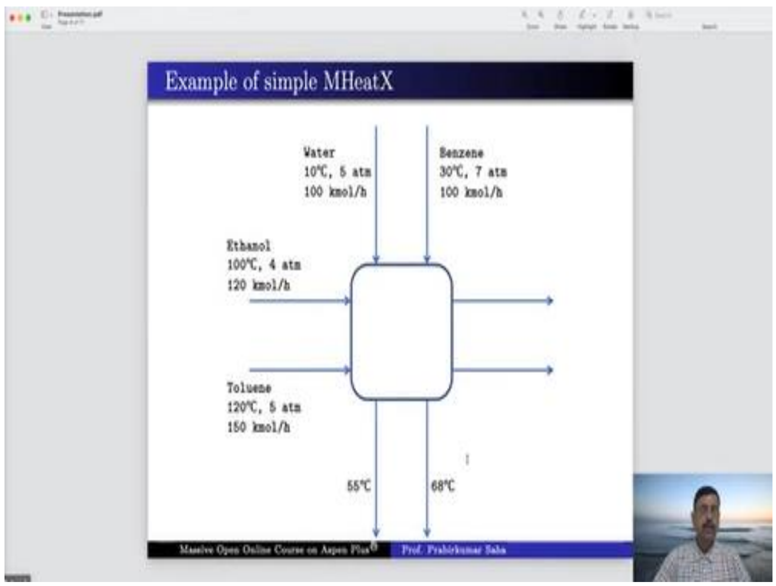


Unlike other unit operation blocks, M Heat X is not simulated by a single computational module. Rather Aspen plus generates heaters and heat streams to represent multi-stream heat exchangers.

Following may be noted; M Heat X can be used to represent heat transfer between multiple hot and cold streams. The detailed, rigorous internal zone analysis can be performed to determine the pinch points. I hope you know what is the pinch point?

M Heat X uses multiple heater blocks and heat streams to enhance flow sheet convergence, and two-stream heat exchangers can also be modelled using an M Heat X.

(Refer Slide Time: 27:55)



Now, this is the example of M Heat X that we will work with. Here there will be two streams water and benzene, which are cold streams and ethanol and toluene are hot streams; their temperature is higher than them. We will use M Heat X to analyse this multi-stream heat exchanger.

(Video Starts: 28:28)

For that, let us go back to our aspen simulation window. We open a new simulation window, blank simulation create, close the new one, save changes no. So, here we have benzene, toluene, water, and ethanol, four components. Press next, use the NRTL_RK method, press next, run press simulation. Then we take out this block over here. So, this one is hot feed. We have to enter hot feed over here.

There are two hot feeds, and this is the hot product out, two hot products. This is cold feed, the second cold feed cold product, and this is the second cold product. Now please remember you should not use this one because this is cold water decant, this is a decant stream, same is here. This is a decant stream. This is not a process stream. So, by mistake, do not use these two lines. Now this one we have to connect to this.

Now we have to rename them this is ethanol in, let it be ethanol out, let it be toluene in, toluene out, this is water in, water out, benzene in, and benzene out. So, we have made the things clear. Now just to make the simulation a bit reasonable, we do certain cosmetic changes just to make the system look better. Now we are ready to send the input. So, press next, so, benzene is 3700. So, 30 ℃, 7 atmosphere and Mole-flow rate 100 kmol/hr, we can keep here.

Next, it is asking for ethanol. So, ethanol is 100, 4, and 120. 100 ℃, 4 atmosphere we can keep it to bar also, not a problem because atmosphere and bar are same, Mole-flow rate 120 kmol/hr. Next asking for toluene, toluene is 120, 5 and 150, and finally water that is 10, 5 and 100. Now as there are multiple streams, Aspen plus is dumb. It does not know which stream has to be connected with what. Now here, ethanol in is from this side. We have given the name ETHAOUT.

But Aspen does not know that eth in corresponds to ETHAOUT that we have to mention. So, here ETHAIN has to be connected with ETHAOUT, not TOLOUT WATIN has to be connected with WATOUT. BENIN has to be connected with BENOUT and TOLIN has to be connected with TOLOUT. And you have to specify certain things like these are the things that you can specify. You can specify the temperature, vapour fraction, temperature change, degree of super-heating, degree of sub cooling, heat duty, molar heat duty, mass heat duty, fraction of heat duty.

So, these are the things either of them you can specify, and out of four, you have to specify at least two to make the equation balanced. So, we have this information with us water and benzene has to be let out at 55 ℃ and 68 ℃, respectively. So, water out temperature, we fix it at 55 ℃ and benzene temperature, we fix it at 68 ℃.

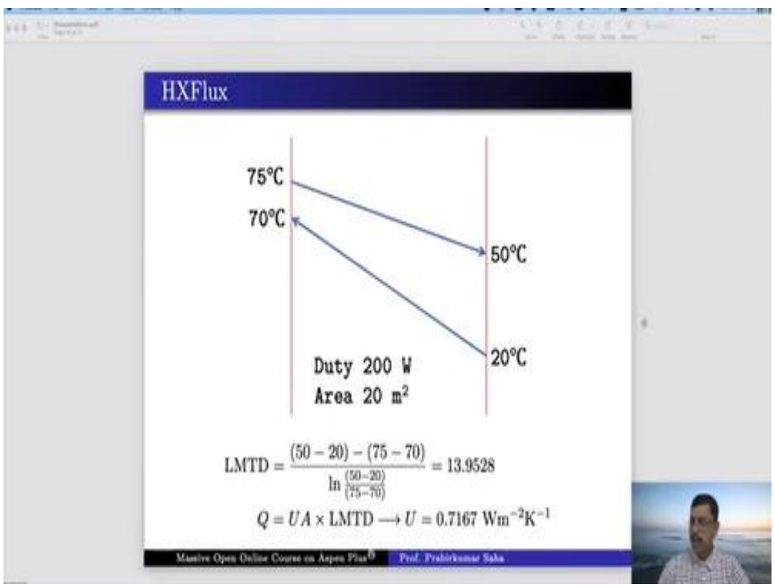
The moment we press 68 this one will become blue ticked. That means all the equations are now balanced. Now it is ready to run the simulation. So, we run the simulation now, so simulation is over. We can see the stream results. So, these are the stream results that we have. The temperature of ethanol it reduces from 100 ℃to 93.6325 ℃. Following reduces from 120 ℃to again 93.6325 ℃.

Now here one thing we have to understand that we have fixed the temperature of water and benzene outlet thereby, the equations are balanced. We did not have to specify anything for ethanol and toluene. Aspen plus assumes that all the remaining streams will go out at the same temperature. We can check that outlet temperature of ethanol is 93.6325, and it is same for toluene outlet as well. Finally, we are left out with only one block that is HX flux.

(Video Ends: 37:12)

HX flux is not actually a block for simulation purpose. It is a block for calculation purpose.

(Refer Slide Time: 37:21)



It can calculate the overall heat transfer coefficient of a process. For instance, here we have flow of say water or something it is a counter current flow. So, the temperature of one fluid reduces from 75 ℃ to 50 ℃. Whereas the counter current fluid it increases from 20 ℃ to 70 ℃. So, it is easy to calculate the LMTD which is 13.9528 degree centigrade.

$$LMTD = \frac{(50-20)-(75-70)}{ln\frac{(50-20)}{(75-70)}} = 13.9528\,℃$$

Now, if we have a heat duty of

$$Q = U \times A \times \text{LMTD}$$

$$200 = U \times 20 \times 13.9528$$

$$U = 0.7167\ W/m^2.K$$

The same thing can be done with HX flux.

(Video Starts: 38:40)

Here we delete everything will not open any new window; delete this object. And we bring in over here. Press next, it is for stream heat temperature inlet hot stream it is 75, inlet cold stream it is 20, outlet hot stream it is 50 ℃ and outlet cold stream is 70. Heat duty we fix up at 200 watts, so we press what 200 and either of them either you can give U or area. So, we have given an area of 20 m^2 the moment we give an area of 20 m^2, it is ready to run the simulation, and the results will be available.

So, we find the overall heat transfer coefficient if we write it in a $W/m^2.K$ 0.7167, which is the same as that we have calculate. That way you can find out all of them. So, this log mean temperature difference is 13.9528 ℃. So, this is one which we had calculated. Now why this is important? Because you can try out with various options like you can give the input of overall heat transfer coefficient 0.9 W/m^2-K, you can run and calculate the heat transfer area.

So, you can say that this much of heat transfer area is required for the process. So, you can very well understand how much heat transfer area is required for your design.

(Video Ends: 41:05)

So, this information is very, very essential for proper design of heat transfer equipment. This is how you can use various features of exchanger design. Thank you.