**Water Supply Engineering**
**Prof. Manoj Kumar Tiwari**
**School of Water Resources, IIT Kharagpur**

**Module No # 11**
**Lecture No # 53**
**Demonstration on EPANET and Water GEMS**

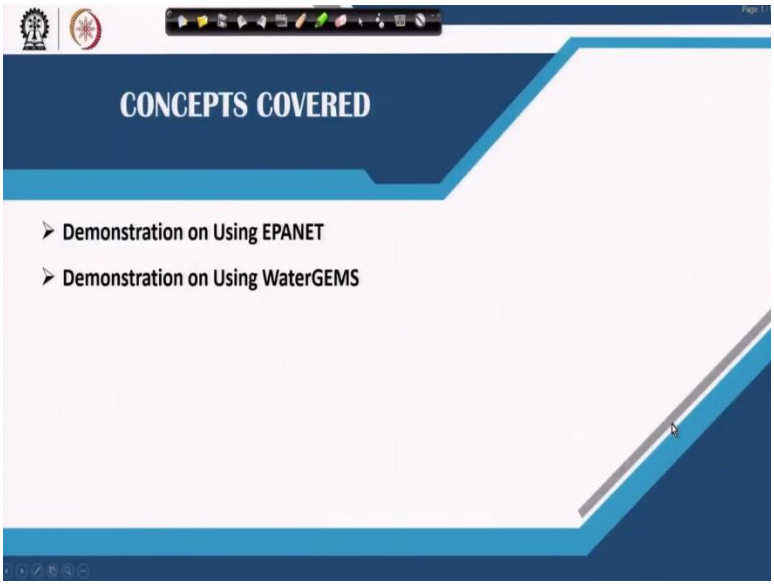
Welcome back friends now in the last we talk about the various software's that are available for design and analysis of water distribution network. And as discussed there are actually quite a few free software's available and there are certain commercial packages are also available. As we discussed that 2 of the more popular packages are EPANET and water GEMS the EPANET is a software from the USEPA and as discussed we did talk about the various functionaries of the EPANET in the last class so far the water GEMS as well.

So in this class we are going to have demonstration on the uses of these 2 software's. These are the most popular packages available in the market in free version of software's available EPANET is by far more popular than any other software packages available. The loop and other package which are available they are used at a very limited scale whereas EPANET has far wider reach and far more options also in terms of designing and analyzing the water distribution network.

The other package that we are going to discuss in this class is the water GEMS. Water GEMS is a commercial package and it is from Bentley systems and in category of commercial packages again water GEMS is considered as the most robust and much better package available as opposed to the other commercial packages available which are for specific needs whether water GEMS has much wider range of applications.

So in this particular class we are going to have some basic demonstration of course like going on a much larger network or a real network the simulation needs lot of time. But we will have just basic demonstration on these 2 packages the EPANET and water GEMS.

**(Refer Slide Time 02:25)**




So we have one of our senior researcher scholar Dina Zaman with us. And she will guide you in this particular lecture about how to use these software's EPANET and water GEMS from the basic. How basically we can setup a network and how we can analyze the network using these 2 packages. So I will handover this to Dina for taking forward from here. So hello everyone so I am Dina Zaman I am a research scholar presently with IIT Kharagpur. So today I will be demonstrating how to carry out a basic hydraulic analysis of water distribution system using EPANET and then in water GEMS.

**(Video Starts 03:11)**

So for any basic hydraulic analysis of a water distribution system we have 3 stages. The first stage is defining the project next will be model building and then of course the hydraulic analysis. So if we go to the first stage let us defining a project it will again have some basic steps to be carried out which we will looking into now. So if you can see in your screen that this is the basic interface of a EPANET software.

So we will have our basic functions in the top tool box and then we have our model building functions here. And this is the space where we will build our network and to the right is the browser. So let me start with the first step which is the projects setup. So for setting up a new project you will go to the file and then click here and then click on new. So this way you will have a new fresh interface for building your model.

So let us start so after setting up a new page for our model building we will go to project. So in project we have a defaults tab where we will have to click. So in a default step you can see that the first step is ID labels. So here we will insert some basic functions which are same for the entire model. So for example patterns we will select 1. So I will be telling why it is selected as 1 later and in ID increment also we will select it as 1.

So this is because if you selected it as 1 then all the objects that we are building in the network will be named here after as from starting from 1. So for example if you have nodes so they will be names as 1, 2, 3 and similarly for pipes as 1, 2, 3. So going to the next step which is properties so here if you already know that our water distribution system has some has the same objects properties we can select it directly here.

So suppose we know that the node elevation for the system is 3 then we can directly give the value as 3 here. Similarly if we have a tank in the water distribution system we can directly select the diameter it is height. And then a similarly for the pipe length and also the pipe dia and pipe roughness. So after we make this selection's we have to click on the save as default for all new projects. Then we come to the hydraulics part again here we see that we can select the flow units the head loss formula for entire distribution system.

For example for the flow units we have all this options from which I am selecting liter per second for the present one. Again in the head loss formula also we have 3 options out of which I am choosing the Hazen Williams formula. And all others you can keep it as default. Again you have to save as defaults for all the projects and then click on ok. So the first step of defining the default values is done. So our next step will be to go to views and then to options. Here we will be selecting all the options of the map that we are going to build.

Let how the map will display itself so coming to the first option is the option for the nodes. So here we can select the size of the node and also it is display property like will it have a border or will the junction be displayed etc., Similarly for the links we can select the link size here. Again coming to the labels so all the labels if you want all the labels to be displayed on the screen itself then you have to click here and here.

Then coming to the notation, I will suggest that display node ID and display link ID should be always selected so that we can have the IDs in the diagram itself. Again you can also select the size of the display that you want here now coming to the symbols. So here I would suggest that you would select the display for tanks, pumps, valves, emitters and all other sources in this tab. Then again you also have a selection for the flow arrows which you can make from this step.

Further you also have a selection of background color so for now let us select the defaults background color which is white and then we tab on ok. Now the third step will be to set up the map scale. So for setting up the map scale you will go to view and then to dimensions. So here you can see that the map dimensions are already set up. So you can use this as this is the basic dimension that is standard dimension.

But for the map units you can select on meters or any other units that you want. So again we click on ok and so the first step of defining the project is been complete. So now we will come to the second stage which is the model building part. So model building part is consist of 2 steps. First is building the network and then inserting all the input information for each of the objects so now coming to the first part which is the model building part. So for model building part these tools are very essential that you can see on the top of the map interface.

So you will be using these to create the network. Suppose you cannot see this tool box here so you will then go to view and then in the tool bars you will click on standard and map and this will be shown in the interface itself. So now we will start with the reserver of the system. So we click on the reserver button here and then we come and click on the map in this way. So you can see that the reserver has been selected.

So we as we press the escape button so that we can select the next tab that we want. So next we go to the junction and then select the junction button. Then I press it on the map so that I can create the junctions in the following manner. So this is how I am creating the junctions for my entire system. Now I also want to have a tank in my system so I click on the tank button and have a tank on this part of the network.

So the nodes part is complete now. Now we have to link this nodes through the pipes. So now I select the pipe from here and then click on each of the junctions between which I won this pipe.

So this is how I will be connecting all the nodes of the system. So you can see that the first loop is created. So I will be creating another loop for this system from here to here and then I will directly connect it here.

Now we still have to connect the reserver to the node and also the tank. So the tank can be simply connected through a pipe like this but from the reserver I will be giving a pump source. So here I click on the pump and then click on the reserver so as you can see I am yet to select a pipe between the reserver and the rest of the network. So here I want to select the pump as a energy source for the network. So you will have to click on the add pump tab and then come over the reserver and then click once.

So this is how you get the pump link between the reserver and the node. So I have already now selected the pump so the basic model creation in EPANET is now over. So what I will do is that I will name the important objects in my network using the next step that is the add label tab. So you click on the add level tab and then you can add any text into your network. So here I am going to add this as reserver then next to its site I am going to add a tab for the pump and similarly for the overhead tank that we have.

So here you can tab the labels and then also you can also set the levels as per your requirement. So once you are finished with the labeling you can again take the cursor select the cursor here and then move the labels according to your requirement. So you can go here and then press it here similarly with the overhead tank. So now you can see that the basic objects are labeled. So you can also although this is not important you can also move the nodes in the network and then give it a shape that look more aesthetically pleasing but this will not have any like impact on the hydraulic analysis.

So this is how you can place your pipes in different locations by just clicking on the nodes and then move it in the moving the cursor. So this is done so now you can see that I have a triangular shape at loop here. So suppose you want to make it a curve pipe so this also can be done so you have to click on the link and then select this tab just select vertex tab here. So then again click on the link again and then right click here to add a vertex in this form. So now a vertex has been added so you can move the vertex in this manner so that you can create a curve pipe.

So for a curve pipe you will need a multiple vertex which you will be adding in this fashion. So as you can see I have added 3 vertices and now I am moving the vertices in the location so that I have a curved pipe. So when you are done you can just click on enter and you will have a network ready. So now this is the basic network that I have but the model building part is not over until and unless I insert the properties of each of the objects that I have built in the network.

So we will be going to each of the objects one by one and then inserting the properties. So for example in the reserver my main input property will be the elevation of the reserver. So here you tab on the you double click on the reserver button and then click on the elevation so now for inserting the input information for every object we will click on each of the object that we are intending. So first we go to the reserver we click on the reserver and then a property tab opens.

So here in the total head you have to select the elevation of the reservoir for example I keep this as 700. In the similar manner we will go to each junction and also give the properties of the junctions. So if you remember in the initial setup of the project I had selected a default elevation for the junctions S3. So you can see that the 3 has been selected for all the junctions in the network. So if you already know that the junctions are in the same elevation you can do it in this way so that it is easier.

Similarly for the pipes also if you remember I had already selected the length of 500 and diameter of 300 with the roughness of 100. So this has also been inserted in to each of the pipes. So you can select some random pipes and check that these have been selected as default values. Now coming to the overhead tank so in the same manner you can give an elevation for the tank for example as 4 meters the initial elevation can be at 4.5. The minimum level can be 4.5 again and then the maximum level can be 6.

So as of now we are keeping an diameter of 20 for this diameter of 12 for this tank. So that tank is also done. Now we are left only with the pump so for the pump we have to insert a curve that the pump curve for this. So for this you have to come to the browser and then here you have to select the curve step. So in that curve step you have to insert or add another curve which will be named as 1. So immediately a curves editor will be open where you can select the flow and head required for the pump.

So for this pump I am selecting the value of flow as 600 and the head is 150. Once you are done click on the ok button and the pump curve is selected. So, more or less our model building part is complete. So now we will check if you have missed any information. So to check this we will carry out a steady state analysis. For this we will be clicking on the project button and then run analysis tab.

So when we run the analysis it will gives a message as run was unsuccessful see the status report for reason. So now when we click ok we find it there are 2 status reports. The first is no head curve applied for pump 9 and one or more other errors. So now we will be looking in to each of the errors and how to like eliminate these errors. So it says that there is no head curve for the pump 9. So now we go to the pump 9 tab and we check each of the properties if we have inserted all the information.

And then we see that the pump curve has not been selected. So here we select the pump curve is 1 as we have done in the browser. So let us see if we can have a successful run now. Again we go the project and run analysis step and it says warning message were generated. So let us see the warning message so the warning message is maximum trials exceeded and the system may be unstable. So there are some instability in the system which we will have to remove.

Again we do a run analysis and check if we are getting this same yes we are again getting the same message. So we will again check the other tabs that can be mis-inserted. Now going to the options we will check what is the time that is selected. So the time duration is as you can see the total 0 duration selected is 0 which is wrong we have to go for 1. So once we are done with this again we will run the analysis through this button.

So now the second stage of our model building is complete. Now we have to go to the final and the most important step that is the analysis part. So our hydraulic analysis will consist of 2 parts. First will be steady state analysis and then the extended period analysis. For the steady state analysis we actually check if there is any un-stability or some missing information in the model. For doing this we will simple go to the project tab and then click on run analysis.

So if you click on this you will get a run status which means it will show you if a run was successful or not. As you can see in that tab it is showing that the run was successful. So you can click ok here and then proceed to the next step which is the extended period analysis. So you may be knowing that steady state analysis is only for checking the stability of the model but it will not give us more information about how the model will run in according to the real time.

As, you will be knowing that we will have intermittent or 24 into 7 systems in the field. So we have to analyze our model also keeping in mind those parameter. Suppose you have an intermittent system which runs for 4 hours daily. So you have to model the network also a for 4 hours in a day. In other on the hand if you have a continuous system you have to modeled a network for a continuously running system. For this what you do is you will select pattern for the demand flow in each of the demand nodes of the network.

So this is a more of a realistic analysis where the time pattern is created at each node and this makes the demanded the nodes vary in a periodic form. So to do this you will go to the options tab you will click here and you will go to the options tab which is at the last and then click here again. So you will see that there is several tab is open up from which you will click on times. So here in times you can select the duration for which you want to carry out an extended period analysis.

For example you want the extended period analysis for 2 days that is 48 hours you will have to insert 48 hours here. Similarly you have to give a hydraulic times step that means after each hour if you give the hydraulic times tables one after each hour the steady state analysis will be carried out and the combination of all steady state analysis after each hour for 48 hours will give you the final results for the extended period analysis.

Similarly you can also choose the other tabs here I am choosing the default except for the pattern time step. So here I am selecting a pattern time step of 6. So once this is done you can close this tab here and then you can insert the pattern so here you can again go to the data tab on the browser and then click on the pattern. So you see that pattern 1 is being selected. So click here and you will have the pattern editor ready. So here what you have to do is you have to provide the multipliers for each of the time period.

So example I have selected a multiplier of 0.5 for the first step 1.3 for the second and then continuing as 1.0 and 1.2 for the third and fourth step. Once this is done click on the ok button and your pattern is ready. So now we will carry out an extended period analysis. So for this again we will go to the project and defaults. So here first we will check if the pattern 1 is selected and then click ok.

So now, again we will go to the run analysis button and try to carry out a extended period analysis. So click here and you will have a run status again similar to the steady state analysis. So here you can see that the run was successful. So this is how you know that both steady state and extended period analysis has been successfully run for your model. So now you can see that the flow direction of water running in your system is displayed on the screen itself.

So if you want to see how the flow direction changes or develops with time for the 48 hours you can go to the browser again then select on the nodes whatever element you want to check suppose for example I am selecting head for the nodes and for the links I am selecting flow. So this is how you can see the actual change in the values for the node and the links in this manner. You can also run this on real time by clicking on this button in the right side.

This is similar to a record player that we see. So you can click here and it will play how it changes with time. Similarly you can also stop this at any moment where you want to analyze and you can get the values of head and flow for this contour map. Again if you want to change these contour values you can right click on them and you can check the changes contour values and also the colors if you go to the color M.

So now I am clicking checking it on the defaults and keeping it same. So this is how you can use this contour maps to have better evaluation of your system. Again along with this graphical or visual maps you can also have tabular data. So if you want to have the tabular data you have to go to the report tab and then click on the table. So here if you want to check the table for networks network nodes you check the flow first step and then click ok.

So in this manner you will have the network table for the nodes. As you can see I have not provided any demand for the nodes so it has come as 0. But the head after the simulation is being

shown here. Similarly the pressure is also shown. So this is a basic hydraulic analysis with only steady state and an extended period if you are creating a quality analysis you will also have the quality parameters here.

Again in the same manner you can also go to the table step and select on the node links and in this manner you will have a table for the node flow. So here since I have not inserted any demand value it is showing as 0. So this is how you can carry out the basic analysis for a water distribution network using EPANET. So we have completed our basic hydraulic analysis in EPANET. We will try to do the same analysis using a commercial software packages that is water GEMS.

So for water GEMS it is similar to EPANET but it is more of a real time analysis system which can be easily applied to the actual water distribution networks. Water GEMS is more robust than the EPANET network and it can carry out more simulations and scenarios effectively. So coming to the first step that is the initial set up of the project. So in water GEMS this is the interface that you will see the first part is again the tools bar that you will be using and here you can see that there are many more options than the EPANET.

So first we will go to the tools object I have already selected a new network for building. So in tools you will go to the option more and then click on the options. So in options you can see that there are many default value selected so we will use the default values as it is. But in the drawing tab we will select the drawing mode as scale and then plot scale factor as 1 centimeter = 40 meter. This you can change as per your requirement.

Then we will go to the units step. In the unit step we will be using the SI units for all our measurements. So here you can go to defaults reset defaults and then select SI. As you can see all the units in the labels are now in SI. So we will move forward to our second stage that is the model building part. The model building part is again very similar to that we have used in EPANET. We will go to the layout tab and we can see that all the tabs that we need for model building are available here.

So first we will catch select the reserver button and then paste a reserver on a network. There as the reserver is done after that we have 2 choices either we can model the nodes and pipes

separately one after other as I will be showing here. So you can select on a junction tab and then click on the junctions that you need. And then you can select on the pipe and insert a pipe between this junctions. However there is also a simpler method to do this that is we can model the junctions and the pipes both at the same time by selecting the pipes button.

So take the pipes button and you place the junctions wherever you need and the pipes between the junctions will be selected automatically as I am showing you on the network. So this is how you model the pipes and the junction together. So we will be needing a few more junctions which I am selecting here then one will again come here and now we will be connecting the remaining junctions to form the entire system.

If you remember in our earlier system we had a tank. So similarly we will also try to create a overhead tank here. So we will click on the tab button and then place a tank here. Again we will need a pipe to connect the network from the tank to the last node like this. Here also we will use a pump as the energy source for the system. So we will click on the pump tab and then you do not select it at the same time what we can do is tick on the pump.

And then place it here so by mistake I have actually deleted the reserver so I will be adding the reserver again this way. And then we will be connecting these by pipes. So from the reserver you will have the pipe to the pump and then from the pump to the rest of the network. So as you can see the basic network building is over so here also similar to EPANET's we can move around the move around every object and place it where ever we want to make it aesthetically pleasing.

So but again this will have no impact on your hydraulic analysis so this is how we have placed our network. So now what we will do is that we will try to create a curve pipe similar to what we have done in EPANET. So you click on the pipe and then right click and select a bend. So in the bend step you will again have a selection of add bend. So you will add a bend here and you can see that a bend has come on the specific pipe. So you can move around the bend to create a curve pipe.

Similarly I will be adding a few more bends so that I can have a curved pipe in this manner. So I will do one more so that it is similar to the diagram I had made for EPANET. So once the bend is done you can click outside the network and you will have a required pipe in this manner. So we

can see that the network looks ready one more thing you can do is that you can select the selector names and place it where ever you want on the top of it so that it is easily visible.

For example the pump you can put over the pump and then also the pipe rotation over or below it. Similarly for the junctions you can click on the junctions label and take it wherever you want to place it in this manner. So you can do it for the whole junction. So now we will come go to the next step of model building which is inserting the input information for each. So in a water GEMS we have 2 options.

Either we can insert the information one by one for each of the objects. So to do this you will have to click on it and then a properties editor will open. So this is the properties editor for reserver as you know that the reserver you will have only one input information which is the elevation. So let me select an elevation of 198 meter. So here I have in put the input and information and I have closed it.

Similarly I can go to a node and then input the information that I need here. So I can put the elevation as 184 for example and I can also select the yes this is done. But this will be very clumsy if we have a lot of nodes or pipes in the system. So there is another option in water GEMS which is not there in EPANET that is we can insert all the information at one time from one place. So to do that, you have to go to the view step then to the flex table and select whatever object you need.

For example I am selecting the junctions. As you can see all the junctions are shown here in the same table itself. From here you can actually insert a information for each of the junctions one by one. For example I am doing one here 187 and the next one I am doing as 178 etc., but here is another more easy option to do this. What you will do is that you can select you can click on the elevation step here and then right click.

Here you will have a function called global edit where you can go and set all the parameters at one time. Suppose I am saying I know that all my nodes are at an elevation of 175. So I will be inserting 175 here. As you can see all the nodes have been set to this elevation 1 at a time. Similarly I can also do it with the zones. So all the nodes can be put a into the same zone or we can keep it as none if it is just one zone.

You have to also remember that the tabs that are shown in yellow cannot be inserted those are model generated. Only the tabs that are in white background can be inserted. So I am closing this table now and doing this similar insert information for the links or the pipes. Again for the pipes you can see that all the information is available in the single cable. So here we go to the diameter. Now the default diameter of 150 2.4 has been selected for all the pipes.

So this one I can this diameter value I can change similarly with global edit to any value that I want suppose I keep it as 150. So this is how the diameters can be set. Now coming to the material so if you want to change the material of the pipe. So you have to click on the materials then right click on it and then click on global edits. So in the global edit you can select any material that you want for example here we have selected it as ductile iron. Again there are more pipes that you can select from.

If you want to select other pipes then you have to click on this right tab and you will see a materials library opening here. So in this materials library you will have enormous number of pipes that you can select from you can select any of these aspire requirement. So I am again selecting ductile iron from here. So as soon as I select ductile iron the Hazen's William coefficient inserted for the ductile iron in the library is also selected automatically.

So this is how you can select all the pipes from here it can also be of different nature like you can select some pipes of PVC if it is your requirement and in this metal manner you select all the pipes. So right now I am doing a global edit for this so that all the pipes are of the same material. Again coming to the next step similarly you can also use the default values of Hazen William co efficient from the library or you can set it yourself say for example if I want this pipe to be 100 so I can select the value here.

So now coming to the user define length. So if you have the length of the pipe so you will have to click on the user defined length one by one or you can go for global edit. So here I am editing it globally and then inserting the pipe lengths. So again I am taking a pipe length of 500 meter each and inserting it in global edit itself. So the pipes part is also done. So now we will come to the tank. In the tank we have to insert lots of information one by one.

So let us keep the elevation of the tank as 200 then we come to the then we will have to also select the base elevation which is again kept as 200. Then we will have a minimum elevation of 220 and an initial elevation of 225 and also we select a maximum elevation suppose of 226. Now we can also select the shape of the section of the tank now here we have selected this as a circular and we have also assumed a diameter of 12 meter.

So now the tank is also done. Now coming to the pump so, for the pump again we have to similarly insert a pump curve for this for operating it. So here we again click on the pump step and a pump editor is open. So here we will go to the pump editor and then define the pump. So for defining the pump you have to go to the tab pump definition and then click again on this underscore. So here you will see two one function called edit pump definition. So it will again double click here and the pump definition editor will open.

So you will need a new definition for this pump so again click on new and you will have a new pump definition. So here we are selecting a basic standard 3 point pump definition for this we will insert the values of flow and head as follows. So initially it will be at flow 0 the head we will keep as 30 then will come to the next design flow of suppose we are selecting a value of 3800 for the corresponding head for this flow will select as suppose 27.4 now coming to the maximum operating flow we will keep it as round 7500 and the head is 24.

So now you can see that the basic pump definition is done. So we will close this editor and also close the pump definition. So now this is done. We will check if the pump has actually taken this definition so you can again go to the pump definition and select the definition that you have selected just now for this pump. So now our second stage of model building is also done. Now we come to the third stage that is the hydraulic analysis.

Again similar to the EPANET analysis we will again we will carry out a steady state analysis first and then move on to the extended period analysis. For is steady state analysis we will first go to the analysis step and then the options button. In the options button we have a again another tab which says steady state EPANET EPS solver. So here we will click on the basic calculation option and then fill up the required tab. So here you can see that it is already selected as steady state we will keep it that way and then close the editor.

So to compute or carry out the analysis you have to click on this button once. As you can see the analyze is complete you have no error message which will be shown here. Since you have no error message you can understand that the network is build properly it has no missing information and also that it is hydraulically stable. So once we have this information we can move towards the extended period analysis. Now again similar to the previous demonstration you will have to select a pattern for the demands in this network.

So to select the patterns we can go we can again do it in two ways. We can click on one of the nodes and then here you can see a tab here you will have to go to the demand collection tab and then right click on this button. So here you can again click on the pattern demand tab. So this is how you can extract patterns creator. So for the hydraulic analysis you will again create a new hydraulic pattern 1.

So you will keep the start time as 12 am and you can select the starting multiplier as 0.5. You can either have a continuous pattern or a stepwise. So for this one I am selecting a stepwise pattern. So now you will have to give all the multipliers early multipliers for the 24 hours of the day. So for this example I am selecting the multipliers as follows. So I am keeping a first time inserting the time in multiples of 3 as follows. So you will have to do this according to the time steps that you are selecting.

So I have done it till 24 hours in this pattern. So for each of the time step you will have to also provide the multipliers. For example I am selecting the following multipliers. So for the first 3 as its 0.8 then I will select one for 0.9 it will increase to 1.2 then 1.5 and then again there will be a diagonal decrease 0.9 0.7 and then again an increase to 1 and the last will be 1.9. So this is an example pattern that I have created for the for this system.

So now once the pattern is created we can close the tabs. Now we will have to select this pattern for each of the nodes. So what we can do is that we can go to each of the node and then select this pattern. So if you go if you do it this way then you will have to again open the nodes tab and then select this demand collection from here as hydraulic pattern 1 and also you can select the demand that you want to insert on this node.

Suppose I am selecting demand of 34 liters per second. But this is again clumsy if you have this lot of nodes in this system. So there is another way to do it which is through the component step in the component step you will have a demand center where you will click on the demand control center. So you can see that one of the junction is already selected. So you will have to also insert the other junctions similarly.

So you will go to the new tab and then initialize demand for all elements. So once you click on this you will see all the junctions of the network of the same time. So now you can either carry out the global edit in this manner or you can individually select the demands for each of the networks. So here I am doing a global edit and putting a demand of 34 liters per second. Similarly you will also have to select the hydraulic pattern which he can also do through the global edit.

So now the demands have been selected and the pattern has also been selected so we will go for a EPS analysis. For this again we will go to the analyze tab the option step inside it and then again to the solver. So in the solver we will select the basic calculation options and here we will change the steady state analysis to a EPS. So once you select the EPS you will have 2 more option that is the start time of the EPS and the duration.

So let us skip the start time as 12 am but the duration we can change to suppose 48 hours. So once this is done we will compute the network. So you can see that the engineers has there is message saying that the calculation engine has failed and to obtain a complete problem choose analysis or validate. So there are some problems with the EPS which we will have to solve one by one. So what is says is that the pattern the last multiplier in a pattern must match with the starting multiplier.

So there is a problem with the pattern that we have selected. So for this we can go to the pattern step again through any of the nodes of the system click here and then select the pattern from the demand collection. So here we can go and change the tabs as per necessity. So according to the message the last multiplier is not matching with the first one. So we can make some changes here we can edit this part and see if we get a correct result.

So we have changed the multipliers and we will compute it again. So now that we have inserted the demand multiplier pattern for all the nodes we will carry out a EPS analysis. So we will again click on the compute button and see how the result goes. So as you can see there are many messages here which are not these are not error messages but warning messages. Error messages will come when you have a some severe modeling errors in the model itself and those you have to correct before you move forward.

Warning messages are aware you are warmed about some inconsistence in the system. So as you can see we have all messages that we have are warning messages. So this is saying that the tank is empty. So this may be because we have selected a less lesser flow rate or in the nodes. So if you increase the flow rates in the nodes then the tank will be filled. So you can also again go to the demands section and then increase the demand and see what happens.

So let us do that we can co to the component section and then again the demand control center and we increase the base demand to a higher value. So suppose instead of 34 we take a value of **70** 67. So let us see how it runs now. So again we compute and see that the tank is empty. So that means the tank has been over designed so you will have to change the tank parameters to a smaller tank and you have to carry out trail and analysis to get the expected result.

So if you want to change at time dimension you can go to the tanks options again and change the diameter of the tank to a lower value. For example here I am selecting it as 8. So again it is showing as a tank empty. So it means that this will mean in this network maybe the tank is not necessary and we have overestimated and we have put a tank here. So, you can also do the same simulations by deleting in a tank and see how the results show.

So similarly to the EPANET modeling here also you can view the results at the end of the simulations through the view tabs and again go to the flex table if you want to see the elevation in if you want to see the pressure in the network you can go to this and the end you will have the 2 hydraulic grid and the pressure will be shown here.

Then similarly you can go to the views flex table again and in the pipes you will be here it will be shown what is the flow velocity and the head loss gradient in each of the pipes. So since you can see a negative value this means that the flow is in the opposite direction of what you have

chosen in this pipes. So with this I come to the end of the basic hydraulics analysis of water distribution network using water GEMS.

Of course there are many functions on many more scenarios that you can operate using the water GEMS software and you can also build a very large network city wide network and even have a DMA's and also run water quality analysis using this platform. So those are advance analysis which you can do based on this basic analysis of steady state and extended period. So with this I would like to conclude this lecture thank you.

**(Video Ends: 50:46)**

So friends we just had Dina explaining about how we use this software's some of the software's like water GEMS and EPANET which are the most popular software's in the domain of water network analysis. Now of course she explain some basic features of the software's and the people who are more interested in learning these software so EPANET is free they can it can be downloaded from the EPA website and with the practice somebody can get more acquainted to the software's and learn more features of it develop more skills on that.

So with this we conclude the discussions in this week when where we had basically talked about some of the features related to the water network designing and operation. We did talk about the concept of 24 7 system and how can we design or convert an existing intermittent system to 24 7 system. We also discussed about the sectorization of the larger network. So how we can put the zoning and segregate them into the smaller networks which distinct metered area or DMA is popularly known.

And we discussed about application of the popular water network design and analysis software's. So we conclude this particular weeks discussion; and next week we will talk about some more features on the advanced system like a mostly about the use of the automation in the system so that will be discussing next week. So thank you, thank you for joining and look forward to seeing you in the next week.