[**fel3**](https://www.eng-tips.com/userinfo.cfm?member=fel3) (Civil/Environmental)7 Feb 17 01:55

I would need to see a diagram of your system to give you the most complete advice. However, let me offer this general method that I use which you may be able to apply to your system.  
  
Let's say you have, in simplest terms, [reservoir]->[suction pipe or piping network]->[pump]->[discharge pipe or piping network]->[reservoir], which is a typical situation for a booster pumping station in a municipal water system. The only things hydraulically simpler I deal with are wet-well-only sewage and storm drainage lift stations and water wells, none of which have suction piping.  
  
The first step is to split the system at the pump to create two systems. To do this, replace the pump with regular nodes, one for pump suction and one for pump discharge. You now have a suction-side system consisting of [reservoir]->[suction pipe or piping network]->[pump suction node] and a discharge-side system consisting of [pump discharge node]->[discharge pipe or piping network]->[reservoir],  
  
EPANET can handle two unconnected systems in the same model, so I recommend you keep your suction-side system and your discharge-side system in the same model.  
  
The next step is to create an extended period simulation (EPS) where time doesn't matter (though it might in your system). EPS is simply the easiest way to handle flow steps through a pump being represented by two nodes. What I do is this: I attach a demand to the [pump suction node] and an inflow of the same magnitude to the [pump discharge node]. I then create stepped peaking factors that apply only to these two nodes. For example, I might use ±1 gpm for the demand and inflow and peaking factors of 0, 100, 200, 300, etc.; or, ±100 gpm for the demand and inflow and peaking factors of 0, 1, 2, 3, etc. Either way, I have just created flow steps of ±0 gpm, ±100 gpm, ±200 gpm, ±300, gpm, etc. (I'm using the ± to emphasize that the pump suction node gets a demand the pump discharge node gets an inflow.)  
  
Finally, run the EPS and from the results grab the HGLs for [pump suction node} and [pump discharge node] for each flow step. Then, for each flow step, TDH = HGL[pump discharge node] - HGL[pump suction node]. This family of points represents your system curve. For some systems, you might also need to look at high and low reservoir levels on both sides of the pump, varying systems demands on both sides of the pump, high and low roughness coefficients, etc. In such cases, you end up generating a system envelope.  
  
For more complicated systems, it usually takes 2 to 4 runs to get the system envelope. However, one really complicated system I modeled years ago took 16 runs to handle all of the operating conditions of interest. I had system curves crossing system curves, so the system envelope had a few bends in it.  
  
I hope this helps.

[**Steven - Civil**](https://www.eng-tips.com/userinfo.cfm?member=Steven+-+Civil) (Civil/Environmental)

**(OP)**

16 Sep 19 16:06

I'm trying to generate a system curve from EPANet. (as above)

The problem is that my "Discharge" node has a base demand of -1 and I'm getting a "system ill-conditioned at node Discharge" I think it's because there's no reservoir going into the node. If I do put a reservoir there, only the pipe between the Discharge node and the reservoir actually gets any flow at all. So maybe I'm misunderstanding Fel3's post, but for me it doesn't seem to be working. Is there something I'm doing wrong or a better way to generate a system curve?  
  
One thing I was thinking was just to put a pump in place of the nodes and run it at different HP's. Couldn't the system curve then be generated from the total losses + elevation head differences at each flow?  
  
Thanks for any advice! Model is attached

Uma imagem com file, diagrama

Descrição gerada automaticamente

* [https://files.engineering.com/getfile.aspx?folder=7fde2891-612d-4f2d-b198-e](https://files.engineering.com/getfile.aspx?folder=7fde2891-612d-4f2d-b198-eb523e129743&file=System_Flow_Analysis_6%27%27_Pipe.net)

[**fel3**](https://www.eng-tips.com/userinfo.cfm?member=fel3) (Civil/Environmental)16 Sep 19 23:57

Steven....  
  
Each disconnected part of a water model needs its own boundary condition, meaning a fixed HGL at a tank or reservoir. Without a fixed HGL, there is no way to completely solve the network because there is nothing to tie the system HGLs to.  
  
The smaller portion of the model looks right (reservoir--pipes--pump suction), but the larger portion of the model is missing its reservoir. But, the reservoir needs to be at the far end from the pump (e.g. Node 1), not at the near end per your post. The larger system needs to be pump discharge--pipes--reservoir.  
  
Fred  
  
==========

[**fel3**](https://www.eng-tips.com/userinfo.cfm?member=fel3) (Civil/Environmental)17 Sep 19 18:28

Steven....  
  
I have attached a model I built last year as part of training a young engineer in the fine art of pumping station design. The model consists of an EPANET file and an EXCEL spreadsheet. For EPANET, I included both the .inp and the .net files because it's easier to read the Project Summary in the .inp file (it's just a text file). You can run the .net file directly, but you can also edit the .inp file and start from there. The model duplicates fairly closely what I explained in the thread you referenced above. The model generates system curves for one pump operating alone and two pumps operating in parallel.  
  
The procedure is pretty simple:  
-- Edit the EPANET file to suit, then run it.  
-- Take the HGLs for the nodes representing the pump suction and pump discharge and copy them into the spreadsheet. This will automatically plot the system curves.  
-- Add the design point and the pump curves (my spreadsheet has room for two pumps, but you can add more) and you are done.  
  
I hope this helps,  
Fred

==========  
[https://files.engineering.com/getfile.aspx?folder=f07a001a-1301-47dc-8a36-6](https://files.engineering.com/getfile.aspx?folder=f07a001a-1301-47dc-8a36-68724dae5496&file=FEL_Simple_Pumping_Station_(spreadsheet).zip)

==========

The best way I have found to create a system curve at a pump station is to do the following:

1) Shut off the pumps using initial controls

2) disable the pump controls from turning on

3) Place a negative demand on the discharge of the pump station and an equal demand on the suction side.

4) Run the model.

5) the difference in head between the suction and discharge node is the head required to push that flow

6) If one progressively run a series of flows at 500 to 1000 gpm intervals to the max flow desired, you can generate the specific head curve for that pump station

7) be aware that you need to account for varying water levels in the tanks controlling the suction and discharge head in each zone.  I have in the past set the suction tank to its lowest operational level and the discharge tank to just 0.01 ft below overflow as one condition ( Highest static lift condition) and ran a second series with the suction tank 0.01 ft from overflow and the receiving tank at its lowest (lowest static lift curve) These two curves will represent the range of operating conditions for the system curve. Lastly if you run the highest static lift under zero demands (forcing all water to the tank you have the highest system curve that will be experienced. If you run the lowest static lift condition with the highest system demand you will get the lowest system curve condition. Any operation of the pumps will fall in-between the two curves with the pumps generally operating more often near the lower portion of the two curves as the pumps are more often running when the receiving tank is low. You can then overlay pump curves on the system curve and use this to guide you in pump selection.

Hope this helps,

Pat Moore