# KM 331 PROSES BENZETİM PROGRAMLARI DERS NOTLARI [1-4]

Kaynaklar

1.   Chemcad User Guide and Tutorial, Chemstations, Inc. Version 6.1.
2.   Aspen Technology, Inc., Apsen HYSYS ® Version 7.
3.   ChemCad Eğitim Notları , Chemstations, Inc- Houston,TX,USA.
4. A Guide for Getting Started in Aspen HYSYS
   Dinu Ajikutira, Sr. Director, Engineering Product Marketing, Aspen Technology, Inc.

ChemCad 'de Borulandirma ve Pompa Uygulamalari

Problem Statement:
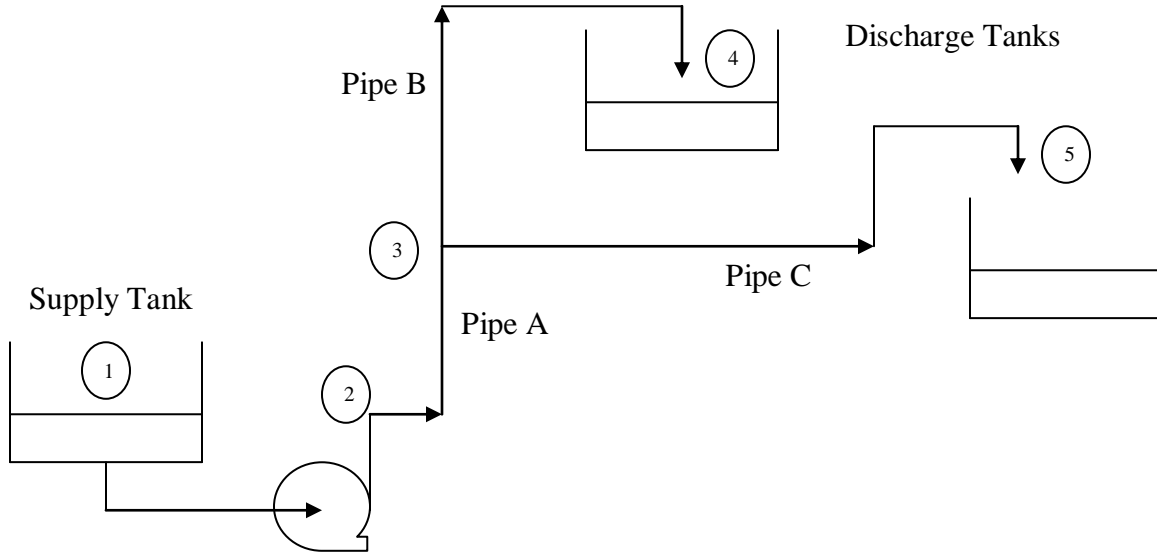
In the piping network shown above, water at 77F and 14.7 psi is fed at 2509 lbmol/h to a pump whose characteristic equation is as shown below:

$$\Delta p = a - bQ_A^2, a = 72\,psi, b = 0.0042\,psi/(gpm)^2$$

Pipes A, B and C are made of schedule 40, commercial steel and the pipeline parameters and elevations are as shown below:

Node	Elevation, ft
1	0
2	0
3	25
4	80
5	60

Pipe	D, in	L, ft
A	3.068	80
B	2.067	300
C	2.067	500

If the pressure at inlets of the two discharge tanks (nodes 4 and 5 in the diagram above) is 14.7 psi, report the flowrates and pressures at each of the five nodes using CHEMCAD simulation.

Procedure:

Step 1: Creating the flow sheet
Step 2: Entering the components and formatting engineering units
Step 3: Entering the feed stream composition
Step 4: Entering the pump specs
Step 5: Entering the pipe simulator specs
Step 6: Entering the node specs
Step 7: Running the simulation and retrieving the results

Step 1: Creating the flow sheet

The supply tank is just a source of water for the pump and can be ignored for the purposes of the simulation. Select and click *feed, pump, pipe simulators (three)* and *product(two)* icons on the workspace. Also click on *node* (icon in the 10th row and third column). This *node* icon needs to be present before and after each pump and pipe when simulating a piping network. So we need five nodes for our flowsheet consisting of three pipes. For the purposes of this simulation, we can ignore the discharge tanks and just use product icons in their place. Connect all the icons using *stream* as shown in the following figure. Click once on the **S**/**G** icon on the menu bar to switch the simulation mode from *Edit Simulation* to *Run Simulation.*
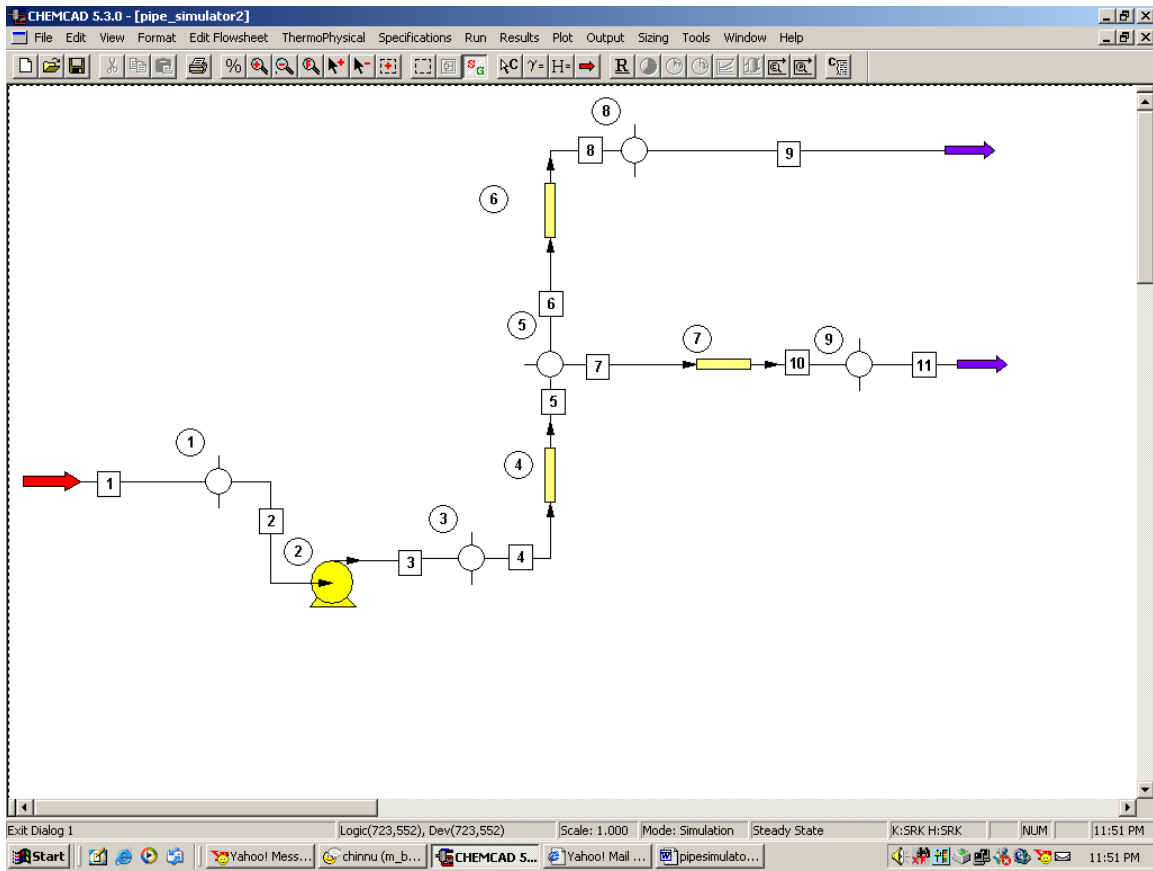
Step 2: Entering the components and formatting engineering units

Go to the *Thermophysical* on the menu bar and click on *Components List.* water from the CHEMCAD components list and add it to the component list. Go to the *Format* menu and click on *Engineering Units* and select the desired units for such properties as temperature, pressure etc. Use *SI* option to convert all units at the same time. Click OK to continue.

## Step 3: Entering the feed stream composition

Double click on the feed stream and enter the feed information (temperature=77 F, pressure=14.7 psi, water flow rate=2509 lbmol/h) given in the problem statement. Click once on *Flash* to get the feed stream enthalpy and vapor fraction in feed at the feed conditions.
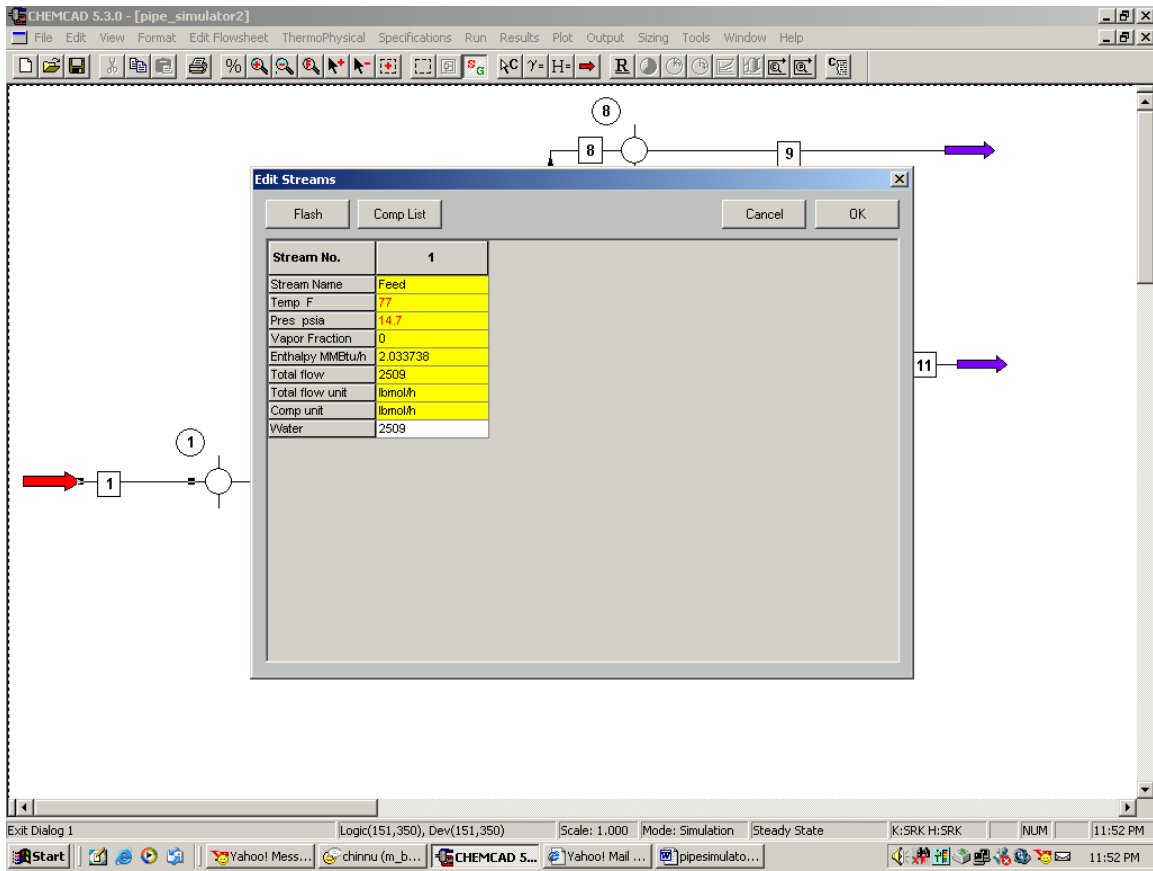
Step 4: Entering Pump Specs

Double click on *Pump.*

Under the specifications page, select *Enter characteristic eqn* for *mode.* Select *psia* for *Pressure units* and *gpm* for *liq. vol. rate units.* Enter 72 for A and 0.0042 for B. Efficiency can be left blank as that information is not provided in the problem statement. (the default value for efficiency is 100%). Click on *OK* to continue.
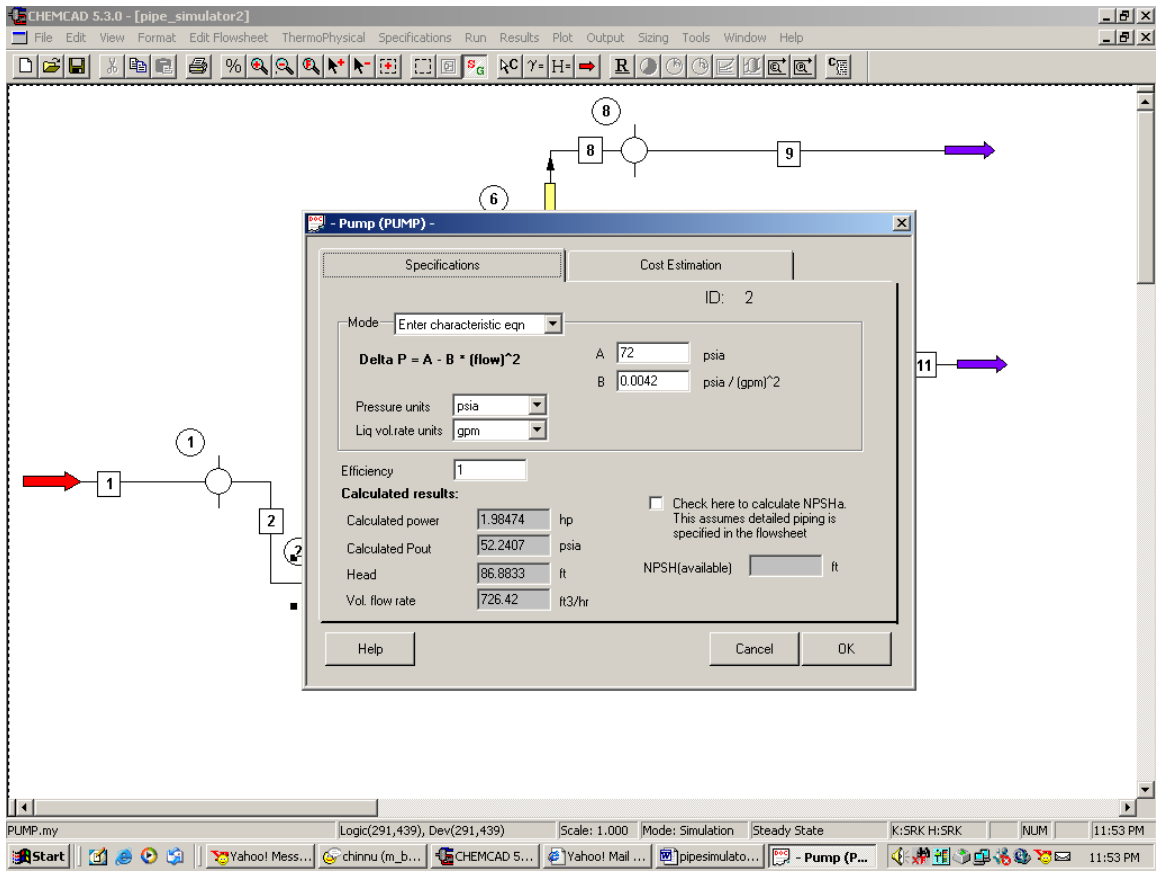
Step 5: Entering the pipe simulator specs

Double click on the first pipe simulator (Pipe A according to problem statement and unit operation 4 in my CHEMCAD simulation).

Specifications:
Method: Select option 2, *Single phase flow*

Sizing option: Select option 5, *Given size, Pin and Pout, calc flow rate*

Pipe Diameter: Convert the given diameter for Pipe A into feet using F6 key on the keyboard and enter the value in feet in this box (0.256 ft)

Pipe Schedule: Enter 40
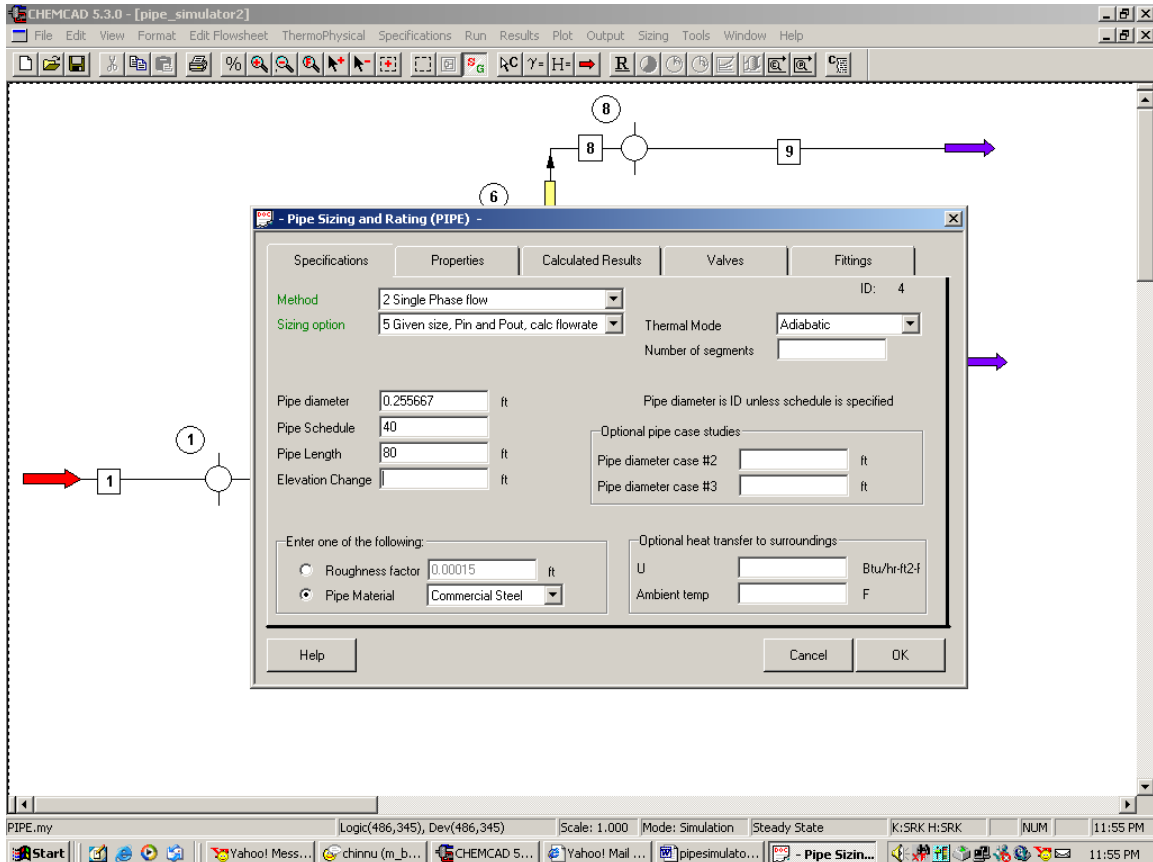
Pipe Length: Enter 80 ft

Elevation change: Leave this blank, CHEMCAD calculates this automatically based on the specs given to *nodes.*

Roughness factor: Leave this blank

Pipe Material: Select *commercial steel*

Click on *OK* to continue.

Similarly, complete step 5 for pipes B and C.

Step 6: Entering node specs

There are five nodes on our flow sheet, specifications of each one of the five are different and dependent on the two unit ops they are connected to. Usually, when the node is in the middle of the flow sheet, *Flow set by unit op* is preferred as the flowrate spec. When the node is in the beginning or end of the flow line, one can select *Free inlet stream* and *Free outlet stream* respectively. If the input node flowrate of the stream is the same as that of the output from the previous unit op, *Use current stream rate* is preferred. For more information on how each of these options specifically function, click on *help.* Provide 0.001 psi and 100 psi as the minimum and maximum bounds for pressure as and when

applicable. Providing the minimum and maximum bounds will speed up the calculations. However, these are optional inputs and can be left blank.

First node (Unit Op 1 on my flow sheet):
Double click on *node.* Choose *variable pressure* for *mode.* There is no elevation for this node according to the problem statement.
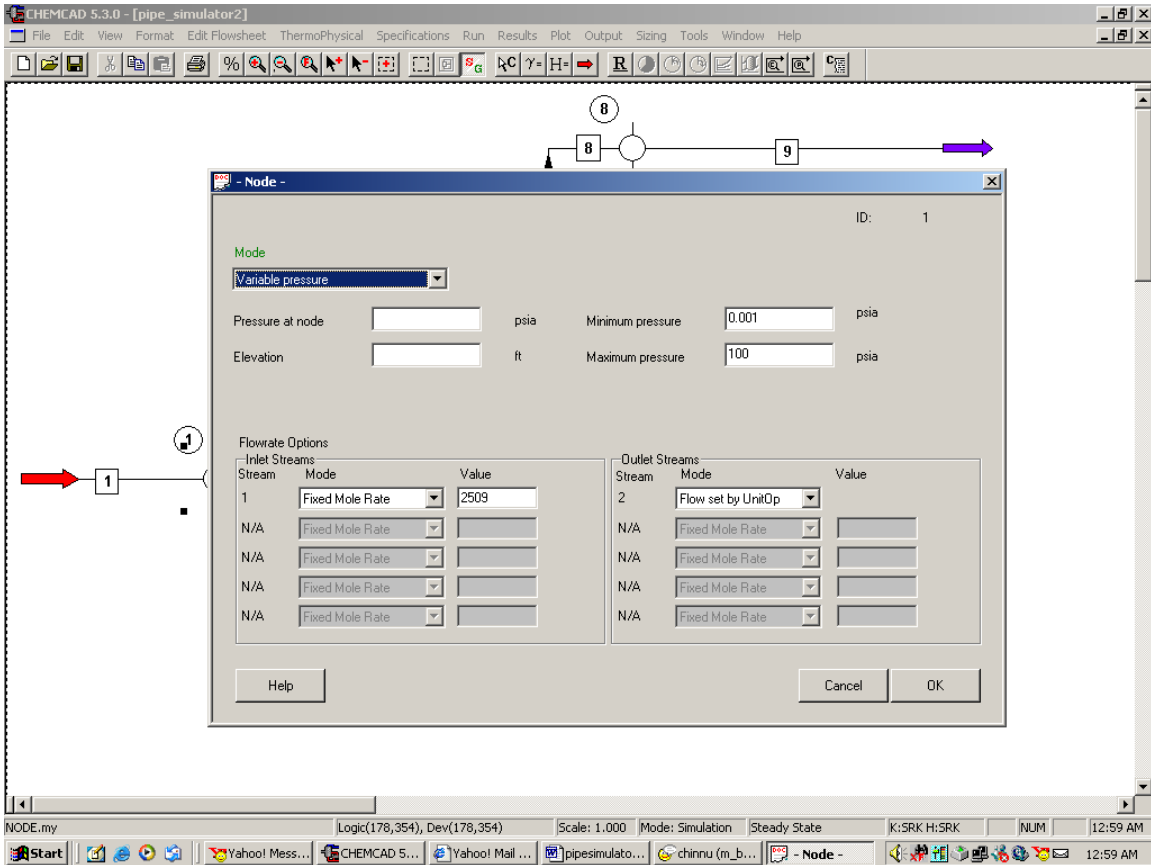
Flowrate Options:
Inlet Stream: Stream 1
Select *Fixed mole rate* option and enter the value 2059 lbmol/hr as stated in the problem
Outlet Stream: Stream 2
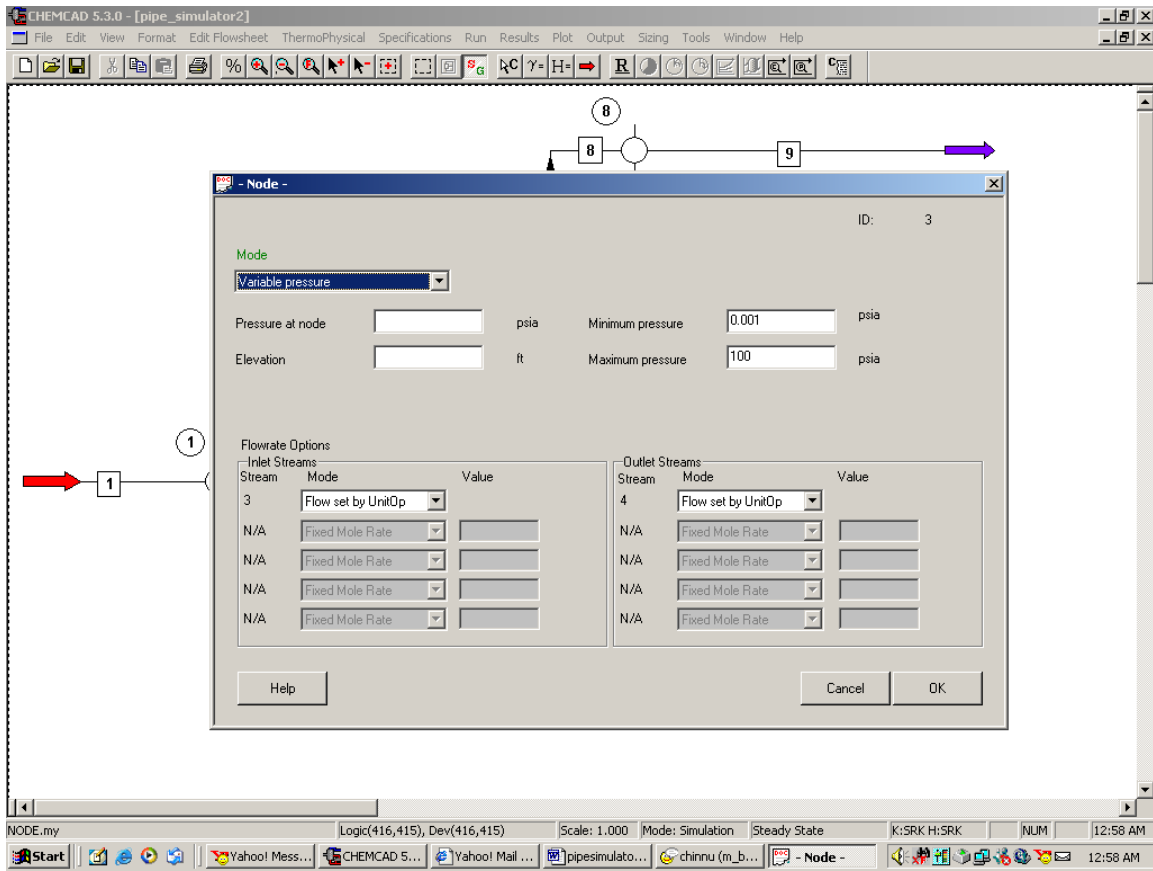Select *Flow set by UnitOp* option
Click *OK* to continue. (Ignore the *no pressure estimation* warning)
Second node (Unit Op 3 on my flow sheet):
Double click on *node.* Choose *variable pressure* for *mode.* There is no elevation for this node according to the problem statement. Under *flowrate options,* select *Flow set by unit op* for both node inlet and node outlet as shown in the following figure.

Click *OK* to continue. (Ignore the *no pressure estimation* warning)

Third node (Unit Op 5 on my flow sheet):
Double click on *node.* Choose *variable pressure* for *mode.* There is an elevation of 25 ft for this node according to the problem statement. Enter this number in the corresponding box. Under *flowrate options,* select *Flow set by unit op* for both node inlet and the two node outlets as shown in the following figure.
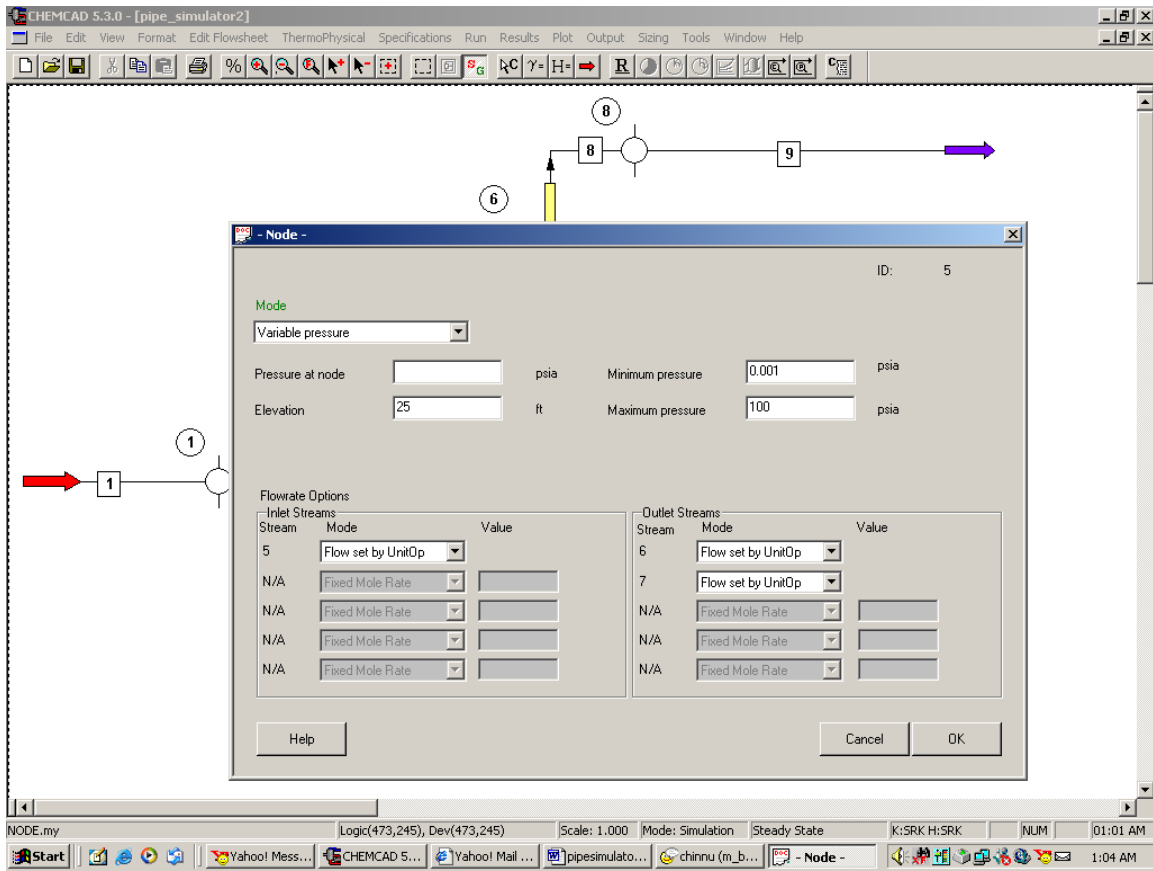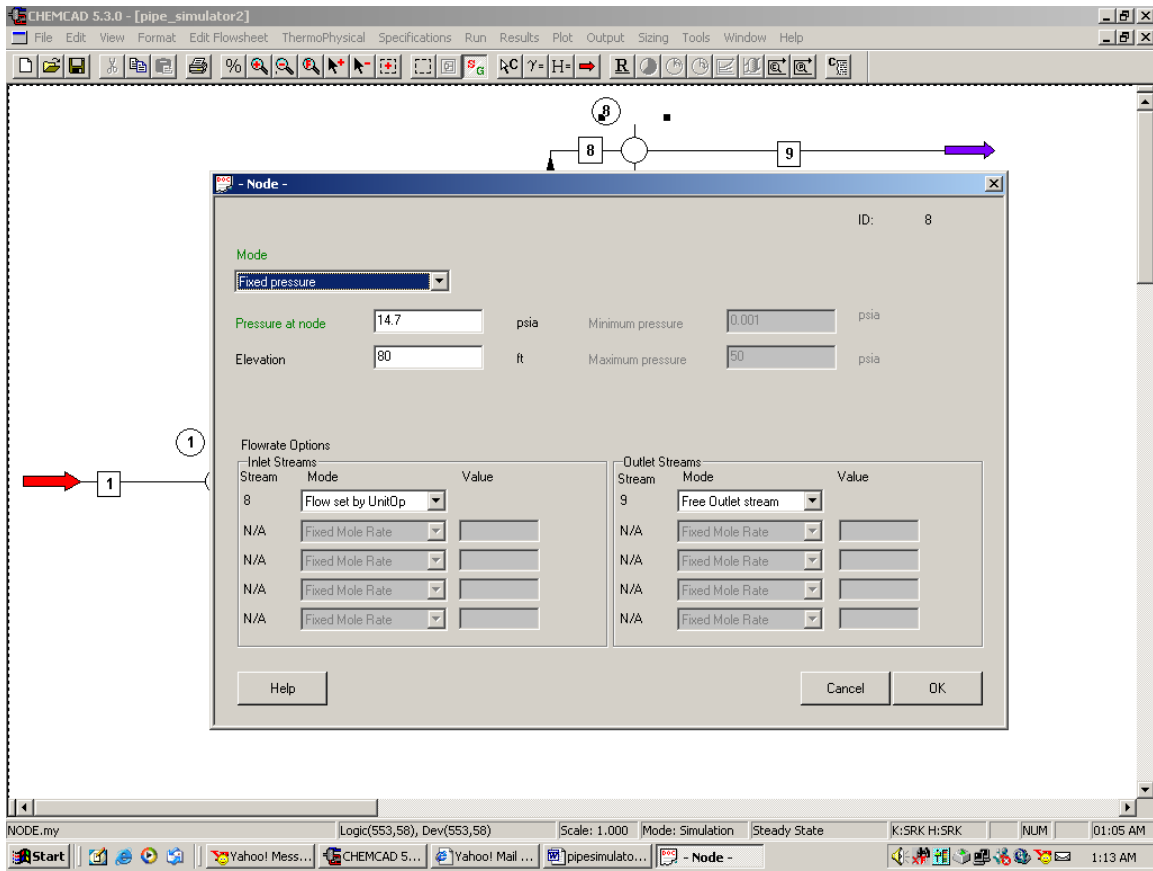
Click *OK* to continue. (Ignore the *no pressure estimation* warning)

Fourth node (Unit Op 8 on my flowsheet):

Double click on *node*. Choose *fixed pressure* for *mode*. Enter 14.7 psi for the pressure at node as stated in the problem. There is an elevation of 80 ft for this node according to the problem statement. Enter this number in the corresponding box. Under *flowrate options,* select *Flow set by unit op* for node inlet and *Free Outlet Stream* for node outlet as shown in the following figure.

Fifth node (Unit Op 9 on my flowsheet):

Double click on *node*. Choose *fixed pressure* for *mode*. Enter 14.7 psi for the pressure at node as stated in the problem. There is an elevation of 60 ft for this node according to the problem statement. Enter this number in the corresponding box. Under *flowrate options,* select *Flow set by unit op* for node inlet and *Free Outlet Stream* for node outlet as shown in the following figure.
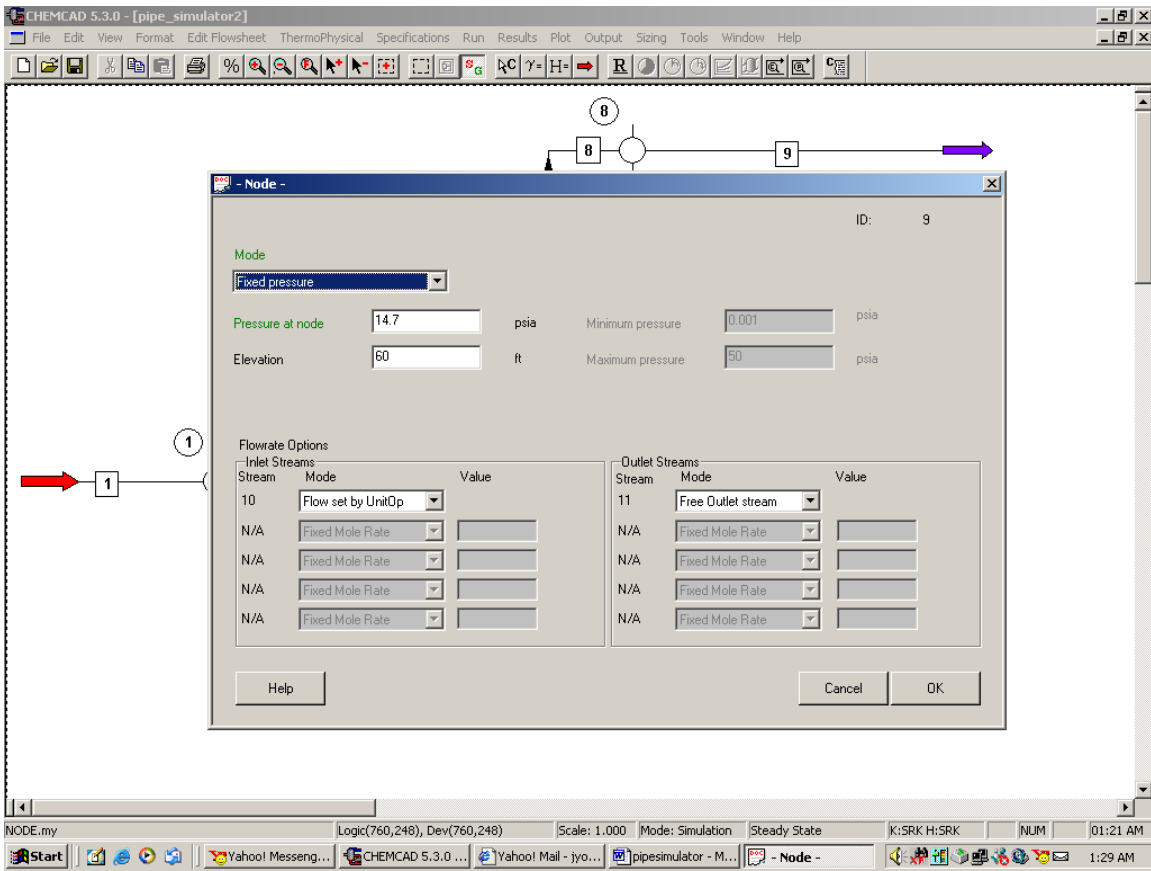
Step 7: Running the simulation and retrieving the results

Now the simulation is ready to be run. Click once on *R* to run the simulation. Alternatively, one can run the simulation by clicking on *Run* on the menu bar and selecting *Run all.* The status of the simulation can be found at the bottom left hand corner of the screen. The message, *Run Finished* appears in this place if the run is successfully completed. The stream properties can be retrieved by clicking *Results,* selecting *Stream compositions* and then *All streams.* The results obtained are shown below.

File Edit View Insert Format Help

Courier New | 10 | | B | I | U | | | | | |

```
Stream No.                   1           2           3           4
Stream Name               Feed
Temp  F                 77.0000*    77.0000     77.1118     77.1118
Pres  psia              14.7000*    14.6982     52.2406     52.2406
Enth  MMBtu/h            2.0337      2.0337      2.0388      2.0386
Vapor mole fraction     0.00000     0.00000     0.00000     0.00000
Total lbmol/h           2509.0000   2509.0005   2509.0005   2508.8281
Total lb/h              45199.6328  45199.6406  45199.6406  45196.5391
Total std L ft3/hr      724.1454    724.1455    724.1455    724.0958
Total std V scfh        952111.88   952112.06   952112.06   952046.63
Flowrates in lb/h
Water                   45199.6328  45199.6406  45199.6406  45196.5391


Stream No.                   5           6           7           8
Stream Name
Temp  F                 77.1118     77.1118     77.1118     77.1118
Pres  psia              40.8299     40.8299     40.8299     14.7000
Enth  MMBtu/h            2.0387      0.74718     1.2916      0.74718
Vapor mole fraction     0.00000     0.00000     0.00000     0.00000
Total lbmol/h           2508.8281   919.5068    1589.4916   919.5068
Total lb/h              45196.5391  16564.9141  28634.6895  16564.9141
Total std L ft3/hr      724.0958    265.3872    458.7577    265.3872
Total std V scfh        952046.63   348933.16   603178.06   348933.16
Flowrates in lb/h
Water                   45196.5391  16564.9141  28634.6895  16564.9141


Stream No.                   9          10          11
Stream Name
Temp  F                 77.1118     77.1118     77.1118
Pres  psia              14.7000     14.7000     14.7000
Enth  MMBtu/h            0.74718     1.2916      1.2916
Vapor mole fraction     0.00000     0.00000     0.00000
Total lbmol/h           919.5068    1589.4916   1589.4916
Total lb/h              16564.9141  28634.6895  28634.6895
Total std L ft3/hr      265.3872    458.7577    458.7577
Total std V scfh        348933.16   603178.06   603178.06
Flowrates in lb/h
```

For Help, press F1

Start | Yahoo! Messeng... | CHEMCAD 5.3.0 ... | Yahoo! Mail - jyo... | pipesimulator - M... | pipe_simulato... | 1:34 AM