Absorption Column Design Using HYSYS 3.2 – Tutorial

This tutorial introduces the use of HYSYS 3.2 to model a continuous gas absorption process in a packed column.

The following problem is solved in HYSYS.

Problem statement: CO\textsubscript{2} is absorbed into propylene carbonate in a packed column. The inlet gas stream is 20 mol\% CO\textsubscript{2} and 80 mol\% methane. The gas stream flows at a rate of 2 m\textsuperscript{3}/s and the column operates at 60 °C and 60.1 atm. The inlet solvent flow is 2000 kmol/hr. Use HYSYS to determine the concentration of CO\textsubscript{2} (mole \%) in the exit gas stream, the column height (m) and the column diameter (m).

1. Accessing HYSYS 3.2:

- Click on the ‘Start’ Button

- [In the computer labs in Swearingen Engineering Center and in 300, Main Street] Go to Programs > ChemE Apps> Hyprotech > HYSYS 3.2 > HYSYS (see figure 1b).
2. Starting a new case:

To start a new case, either go to File > New > Case or hold down ‘Ctrl N’ or press the New icon as indicated in figure 2.

(Figure 2)

3. Selecting the components:

After you open a new case, you will see a window called ‘Simulation Basis Manager’ (see figure 3a). Now, you need to add Methane (CH₄), Carbon dioxide (CO₂) and Propylene carbonate into your simulation. To add a component, click on ‘Add’ and in the ‘Match’ box, type the name of the component. Once you locate the component that you need to select, click on ‘<...Add Pure’ and the component will be selected (see figure 3b). Go through this process until you have selected all three components (i.e. CO₂, CH₄ and Propylene Carbonate). Close the window after you have completed the components selection.

(figure 3a)
(Figure 3b)
4. Selecting a ‘Fluid Package’:

Now click on “Fluid Pkgs” tab. (Figure 4a)

(Figure 4a)

Click on the ‘Add’ button. This will open a new window, ‘Fluid Package: Basics-1’, (Figure 4b) which shows the property packages that are available in the database. The Fluid Packages are in-built sub-routines that calculate the component properties as a function of system parameters (like temperature and pressure). The ideal gas law (i.e. \( PV = nRT \)) would be the simplest case. In the ‘Property Package Selection’ Window, scroll down and select ‘Sour PR’ (as indicated in figure 4b). Close this window.

(Figure 4b)
5. Entering the Simulation Environment:

Now click on the ‘EnterSimulation Environment’ button on the ‘Simulation Basic Manager’ window. When you do this, HYSYS opens up a ‘PFD’ window. All the equipments of the system should be represented in the PFD window. Along with the ‘PFD’ window, HYSYS opens up an object palette window (see figure 5). If the object palette window does not open, press ‘F4’ or go to ‘Flow Sheet > Open Object Palette’.

(Figure 5)
6. Setting up the Simulation Environment:

By placing the mouse over each icon on the object palette, one can see what each icon represents. Click and select the icon that says “Absorber”, which looks like this:

After selecting the “Absorber” icon, click on the PFD window to insert the absorption column into the simulation environment. By default, HYSYS names it as “T-100”. Double-click on “T-100” to open up the ‘Absorber Column Input Expert’ window. Name the streams that connect to the column. To name a stream, click within the box corresponding to that stream and type the name (e.g. Solvent In, Gases In, Gases Out etc.). Name all the streams as indicated in figure 6b.

(Figure 6b)

Only when all the streams are labeled, the ‘Next’ button is activated. Click ‘Next’. The window changes to the one that requires the pressure at the top and the bottom of the column. Enter both pressures as 60.1 atmospheres (please note that the pressure unit is ‘kPa’ by default – you need to change it to ‘atm’ by selecting from the list of units on the drop-down menus at the top-right corner of the window as indicated in figure 6c). No matter what units you use to represent pressure HYSYS converts it into ‘kPa’ and shows the value in ‘kPa’. After entering the pressures of the top and the bottom, click on ‘Next’. This will change the window to one where the estimates of the temperature of the column top and bottom are asked for. Enter both temperatures as 60 °C. Then click on the ‘Done’ button. By clicking on the done button, HYSYS will bring up a window pertaining to the column (It is actually a schematic of the column). Please note that it shows 10 stages (trays). Later we will see how to change those “trays” to “packing”. For now, close that window and return to PFD.
Now you are back to the ‘PFD – Case [Main]’ window. If your PFD looks untidy or tiny, go to the PFD menu and select “Auto Position All”. You should note that all the streams on the PFD are light blue in color. That indicates that HYSYS has not solved for the properties of the stream. And this is because the components of the streams have not been defined.
7. Defining stream component specifications:

On the PFD, double-click on the blue line that says “Solvent in” (stream). A “Solvent In” window will pop up. Enter the values for the Temperature (60 °C), the Pressure (60.1 atm) and the Molar Flow (2000 kgmole/h) (see figure 7a).

(Figure 7a)

After entering the temperature, the pressure and the solvent flow rate, click on “Composition”. The window will transform into one that is requesting the mole fraction of the specified components in the “Solvent In” stream. Since this stream consists only of propylene carbonate, input ‘1’ next to ‘C3=Carbonate’ and press “Enter” on the keyboard. Another window should quickly pop up after the input is made. In this window, set the other mole fractions to zero. Here, one can specify the composition in various bases. Verify that “mole fraction” is selected. Once you have specified all the mole fractions, click the ‘OK’ button (see figure 7b).

(Figure 7b)
Once you specify the composition, the yellow ‘Unknown Compositions’ tab on the bottom should turn to a green ‘OK’ tab.

(Figure 7c)

This green ‘OK’ tab indicates that all the required information for the stream is specified. Close the “Solvent In” window and return to the PFD. Note that the “Solvent In” stream is of a darker shade blue than all the other streams. The other inlet stream “Gases In” should also be defined in a similar way. For the “Gases In” stream, enter the composition as methane – 0.8, CO$_2$ – 0.2 & propylene carbonate – 0. Specify Molar Flow as 7200 m$^3$/hr (Note that HYSYS will convert the flow rate to kgmole/h as indicated in figure 7d). See to that the “Gases In” stream also changes to dark blue in color.
8. Running the simulation:

On the PFD, double click on "T-100". When the column window pops up, click on the “Run” button located near the bottom of the window. The red “Unconverged” box should turn to green “Converged” if all the above procedure was followed. However, the results that are obtained at this point do not represent a true model for our gas absorption column because the simulation was run using trays, not packing. Now, let’s see how to replace trays with packing.

9. Changing Trays to Packing:

Go to the ‘Tools’ menu and select ‘Utilities’. Scroll down and select ‘Tray Sizing’. Click on the ‘Add Utilities’ button. A tray-sizing window should pop up. Name the utility as “Packing”. Click on the ‘Select TS…’ button. (Please see figure 9a)

(Figure 9a)

Once you select the “Select TS…” button, a window should pop up similar to that of figure 9b. Make all the selections as in the figure and then click “OK”.

(Figure 9b)
After selecting the Tray section, one will return to the ‘Tray Sizing’ window. Click on the button that says ‘Auto Section’. For the tray internal, select ‘Packed’. When ‘Packed’ is selected, a drop down menu box will appear in the window. Scroll the drop down menu box for the desired packing type. For this case choose, Raschig Rings (Ceramic) ¼ inch. You should not worry if your selection doesn’t show up on the drop down menu box. When the selection is made, click on the ‘Next’ button. In the next window that appears, click on ‘Complete auto section’. The ‘Tray Sizing’ window should appear. Now close this window and go to the ‘PFD’ window. Double-click on ‘T-100’ and run the simulation again.
10. **Getting the design parameters:**

Go to “Tools” menu and click on “Utilities”. A window named “Available Utilities” will pop up. Select ‘Packing’ and click on “View Utility” button. On the window that pops-up, click on ‘Auto Section’ and change the internal type selection from ‘Valve’ to ‘Packed’. You don’t have to select the type of packing again. Click on ‘Next’ and then on ‘Complete Auto Section’. Now, click on the ‘Performance’ tab and select “Packed”. In the section results, you can see the diameter and the height of the section (see figure 10). Enter these values on the handout. Now, go back to the PFD window and double-click on the “Gases Out” stream and note the composition of CO$_2$ on the handout.  

(Figure 10)

11. **New Run:**

To change the solvent flow rate, go back to the PFD window and double-click on the blue line that says ‘Solvent In’. Change the flow rate from 2000 kmol/hr to 2500 kmol/hr. Run the simulation and see how the column dimension and exit concentration of CO$_2$ have changed. You will notice that the exit concentration of CO$_2$ decreases significantly with the increase in the inlet solvent flow rate but the column dimensions do not change appreciably. Record these values in your answer sheet!

12. **Saving your case:**

Once you have finished your simulation using HYSYS, save your case as YourLastName.hsc for future references.

*Prepared by,*  
Vijay Anand Sethuraman (sethuram@engr.sc.edu)  
Revision 4, April 11, 2005