File: g:\courses\spring2012\303\IC_3dPspicePlot.doc RWN update of 02/13/12

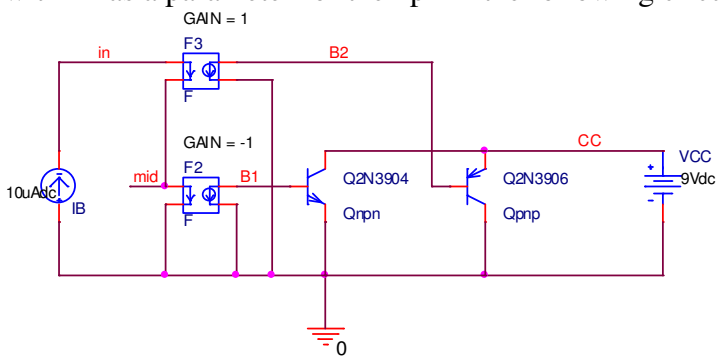
In the most recent version of PSice Run PSpice and in Probe add three traces. In the file menu select export and in that to a csv file.
Open MatLab.
In Excel open the csv file and select the full column of data for the x axis (click on the top of the column, and shift-click on the bottom of the column) and copy.
In Matlab, enter x=[paste the copied column];
Repeat on the column for y=[ ]; and again on the column for the vertical z=[ ];
In MatLab select a mesh grid sizing

gx=gx_min:gx_inc:gx_max; gy=gxy_min:gy_inc:gy_max;

These can be numbers entered in the Pspice configuration file for the run or any others as desired.
Continue in MatLab with

[x1,y1]=meshgrid(gx,gy);
z1=griddata(x,y,z,x1,y1)
mesh(x1,y1,z1); hold;
surf(x1,y1,z1)

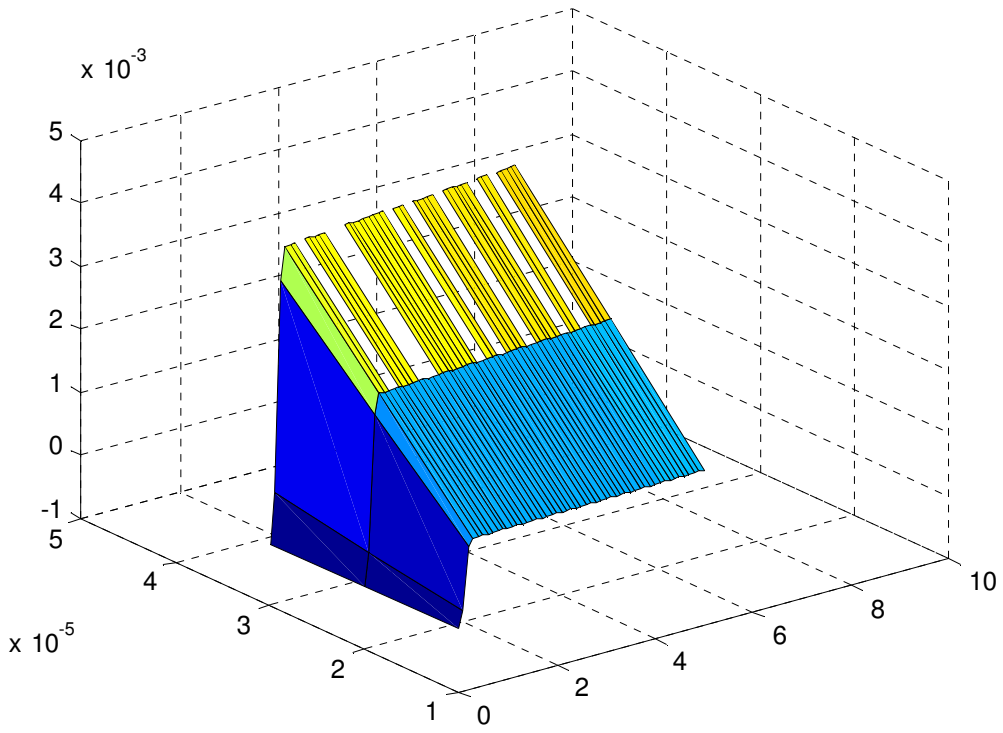
As an example; use the UMCP ECE web VPN PSpice. Make a 3D plot of the IC vs Vcc with IB as a parameter for the npn in the following circuit,



Do a DC sweep on VCC from 0 to 5V in 0.01V steps with a secondary sweep on IB from 0 to 40uA in 10uA steps. In the Probe window add I(IB) and IC(Qnpn) and add a plot with V_VCC added as a trace. Then at the file menu under export choose Column Separated File (.cvs); the chosen traces will be shown on the left (you can add or delete as desired). Click on the upper … to give a file name and then save. Open Excel with that file and also open MatLab. Select the x, y, z columns as V_VCC, I(IB), IC(Qnpn) and copy them into Matlab inside square brackets. In matlab enter

```
gx=0:0.1:5; gy=10e-6:10e-6:40e-6;
[x1,y1]=meshgrid(gx,gy);
z1=griddata(x,y,z,x1,y1);
mesh(x1,y1,z1);
surf(x1,y1,z1)
```

This will give a 3D figure like the following showing IC vs VC and IB.:


x 10^-3
5
4
3
2
1
0
-1
5
4
3
2
1
x 10^-5
0
2
4
6
8
10