Calculate the theoretical values I1, I2, I3, I4, V1, V2, V3, and V4 for the circuit shown in figure 4.1.The libraries are named as: abm, analog, analog_p, breakout, eval, evalp, source, sourcstm, special.Capture the circuit shown in figure 4 by following the given steps: a) Start -> Programs -> OrCad 16 Demo -> OrCadCapture CIS Demo.Once the simulation has run successfully, the component values can be seen by clicking on 'V' and 'I' on the tool bar of the 'Schematic1' page.This opens an 'OrCadCapture CIS - Demo Edition' window.Under the location, create a sub-directory called as CTA under TOOLS directory, by clicking on 'Browse'.The path that appears looks like: C:\ORCAD\ORCAD_16.0_DEMO\TOOLS\CTA.Select all the libraries that are in 'PSpice' folder.If a component has to be rotated such as R3 and R4 in figure 4.1, click on the component, right click on the mouse and select 'Rotate'.To select, do the following: PSpice -> Edit Simulation Profile.c) To select resistor, click on ANALOG.You would observe that a resistor symbol is attached to the curser.Once all the resistors are placed on the page, right click the mouse and select 'end mode'.Click on OK. Voltage source symbol is attached to the curser now.Select 0/CAPSYM and click on OK button.and the value for a DC source is 0 V. To change the value of a component, double click on the default value.Simulation Settings - CTA_Exp4 window opens.Under analysis, select 'Bias Point'.Click on Apply and then on OK. j) To run the simulation, do the following: PSpice -> Run.(Show the theoretical calculations, here) 2.Select 'Analog or Mixed A/D' radio button.Select 'Create a Blank project' radio button and then click on 'OK'. b) Now you are ready to capture the schematic.In the Schematic1 window, select Place -> Part.