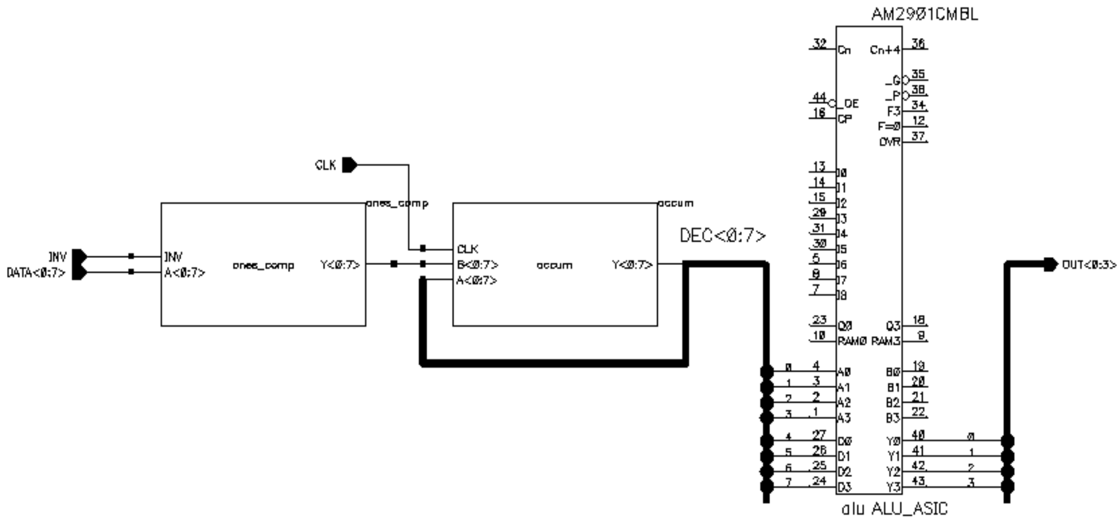
Introduction to Design Entry

1.0 Creating the Top-Level Schematic

In this section you will create the top-level schematic shown below. It contains three main blocks:

- A ones complement function based on TTL parts
- An accumulator based on TTL parts
- An ALU implemented as an ASIC



First you will read the top-level schematic and place it in the top left corner. This is done so that you can refer to the complete schematic whenever you need to do so.

1.1 Open the ***top-level*** schematic of the library ***master*** as described earlier in the Getting-Started walk-through.

1.2 Reduce the size of the schematic window by:

 1.2.1 Moving the mouse to the lower right corner of the schematic window

 1.2.2 Click on the left mouse button, drag it in, and release the button.

1.3 Press ***f*** to ***fit*** the schematic to the window.

1.4 Move the whole window to the top left corner of the screen by grabbing the ***title bar*** and dragging it across.

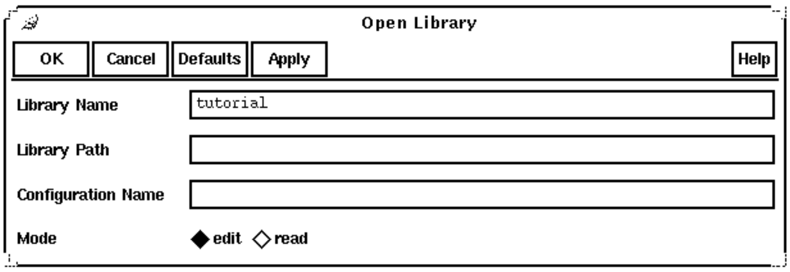
Note: You may want to store the complete schematic as an icon. Then when you need to refer to it you can click on the icon to restore the window.

2.0 Creating a New Design Cell and Schematic Window

The first step in creating the top-level design is to create a new design cell and cellview. This new cell will be placed in the library ***tutorial*** which you should have created in the first document CADEN-1. If you have not yet created the Library ***tutorial*** then follow the steps below, if you have then jump to section 2.6.

2.1 Click on ***Open->Library*** on the CIW menu banner.

2.2 Fill the ***Library Name*** with ***tutorial.***



2.3 Click on ***OK***.

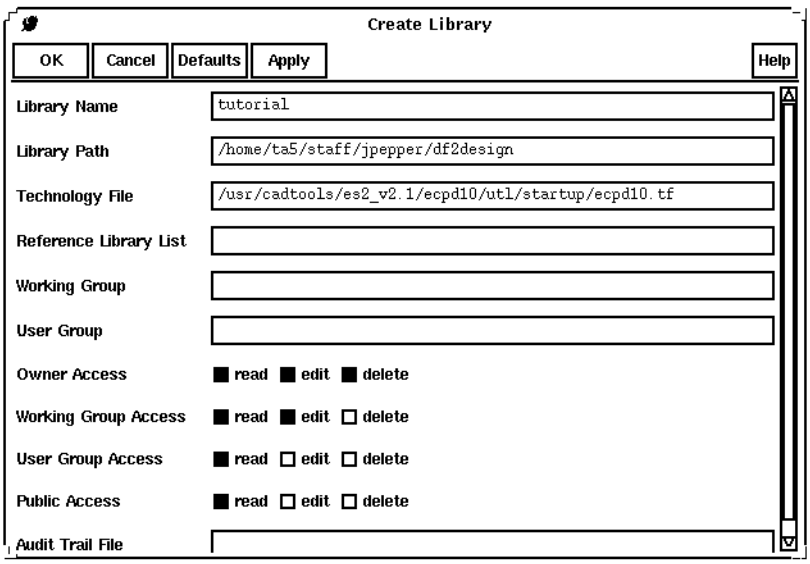
A dialog box appears with the following message:

‘Library tutorial was not found

Do you want to create it?’

2.4 Click on ***OK*** in the dialog box.

2.5 Another form appears, accept the values entered by clicking on ***OK.***



Wait while the library is created..You now have to create the cell to which will hold the schematic.

2.6 Click on ***Open->Design*** on the CIW menu banner.

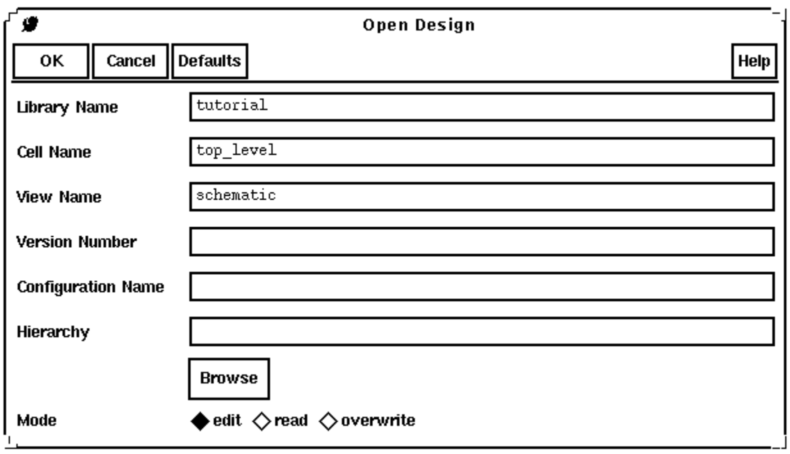
The Open Design form appears ready to be filled in.

2.7 Fill in the form as follows:

Library Name tutorial

Cell Name top_level

View Name schematic



Note: To move from field to field use the mouse to click on the field or press the ***Tab*** or ***Shift-Tab***. ***Do not*** press Enter or Return to move from field to field as this acts the same as the OK button.

2.8 Click on ***OK*** once the form has been filled in.

A dialog box appears with the following message:

‘tutorial top-level schematic does not exist

Do you want to create it?’

2.9 Click on ***OK*** on the dialog box.

A blank schematic window now appears, ready for your work.

3.0 Creating Symbols

Before you can place the instances of the ones complement and accumulator, you must first create cellviews for them.

3.1 Click on ***Create->Cellview->From Pin List***

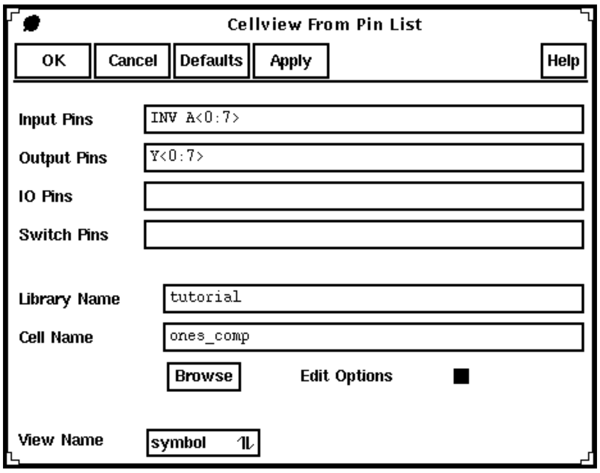
A Cellview From Pin List form appears for you to fill in.

3.2 Fill in the form as follows.

Input Pins INV A<0:7>

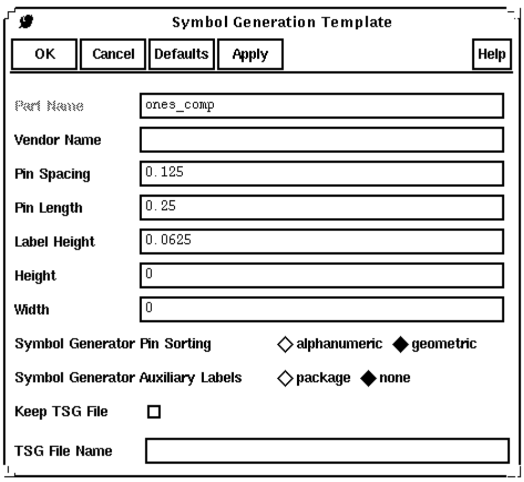
Output Pins Y<0:7>

Cell Name ones_comp



3.3 Also click on the ***Edit Options*** button to turn it on. This allows you to specify graphic data for the symbol.

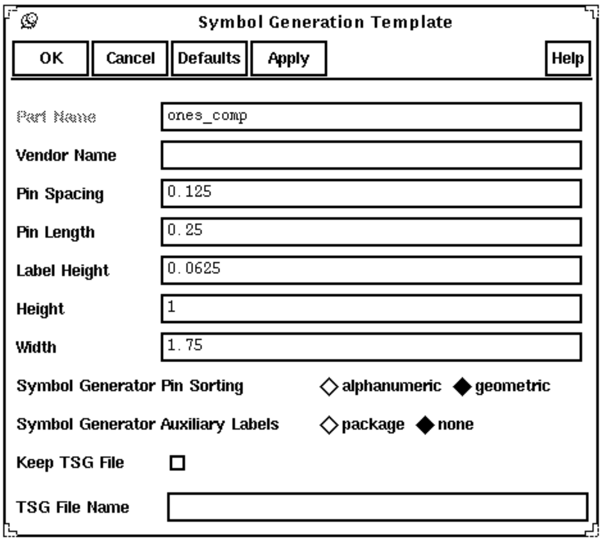
3.4 Click on ***OK***.



The default values for the height and width will need to be altered to what you require. Fill in the Height and Width:

Height 1

Width 1.75



3.5 Check that the geometric button is set to on. Click on it if not on.

3.6 Now click on ***OK*** to create the symbol cellview.

3.7 Repeat exercise 3.1 to3.6 for the accumulator, except filling in the form with:

Input Pins CLK B<0:7> A<0:7>

Output Pins Y<0:7>

Cell Name accum

4.0 Creating the Instances

These cellviews for the ***oness complement*** and ***accumulator*** can now be used to place instances on your schematic, which will be a copy of the schematic shown in section 1.

4.1 To select the ***Create Instance*** command.

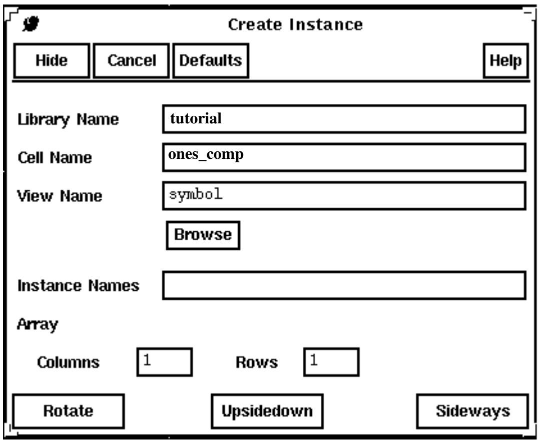
4.1.1 Move the cursor onto the schematic window.

4.1.2 Hold down the middle mouse button to reveal the menu.

4.1.3 Slide the cursor to ***Create Instance.***

4.1.4 Release the middle mouse button.

The Create Instance form now appears. Since the symbol for this instance already exists, you can use the Library Browser to fill in the form



4.2 Click on ***Browse***.

4.3 On the Library Browser, click on ***tutorial.***

The software now inserts ***tutorial*** into the Library Name field on the form.

4.4 Similarly, click on ***ones_comp***.

The ***ones_comp*** symbol cellview appears, and the software inserts ***ones_comp*** into the Cell Name field.

The symbol can now be placed onto the schematic. The ***Library Browser*** and ***Create Instance*** form may block most of the schematic where you wish to place the instance. If this is the case then you can ***Hide*** the form for later use so that you can see the whole schematic window.

4.5 Click on ***Hide*** to hide the Create Instance form.

4.6 Move the cursor onto the schematic window and click to place the one's complement instance.

4.7 Press ***F3*** to redisplay the ***Create Instance*** form.

4.8 On the Library Browser, click on the ***accum*** to change the symbol.

4.9 Place the accumulator instance onto the schematic by clicking the left mouse button.

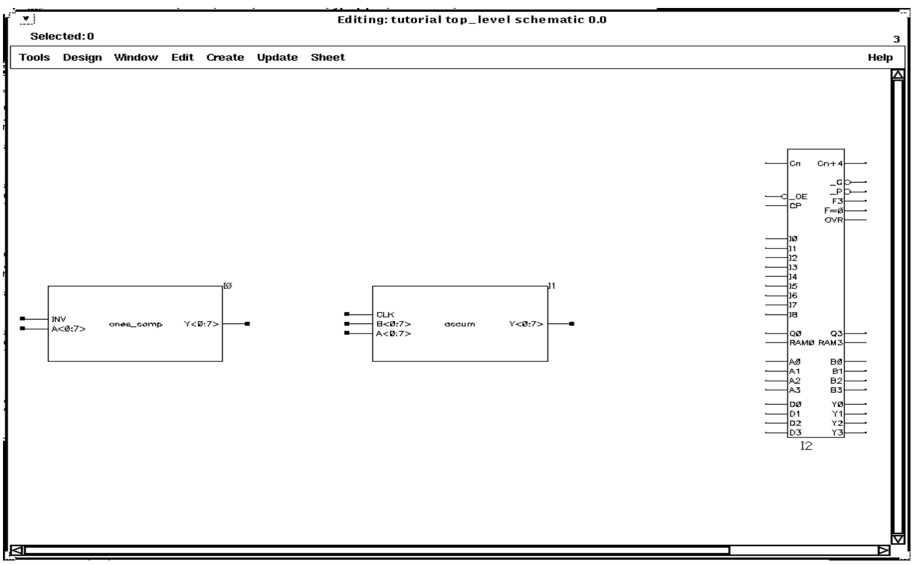
4.10 On the Library Browser, Click on the ***user_ASIC*** library. Click on ***OK*** in the dialog box to add the library to the reference list.

4.11 Click on ***ALU_ASIC*** to change the Cell Name to ALU_ASIC, the cursor now takes on the shape of the ALU instance.

4.12 Place the ALU instance onto the schematic.

4.13 With the cursor on the schematic window, press ***Esc*** to cancel the ***Create Instance*** command.

You should now have the three symbols placed on the schematic as in the diagram below.



Note: ***Esc*** cancels the current active modal command. When you press ***Esc***, the prompt on the CIW prompt line disappears. ***Esc*** works only when the cursor is in the schematic window.

5.0 Saving Your Work

Save your work regularly to ensure that you do not loose it.

5.1 Click on ***Design->Save*** on the schematic menu banner.

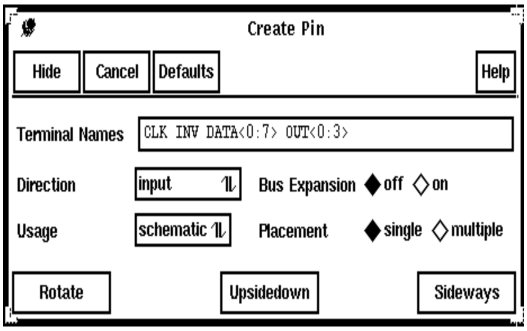
6.0 Creating the Input and Output Pins

6.1 Click on ***Create->Pin*** on the schematic menu banner.

A form will appear for you to fill in.

6.2 Fill in the ***Terminal Names*** field with the pin names as follows:

CLK INV DATA<0:7> OUT<0:3>



Note: Composer places the pins in the order you type them. So you will need to change the direction on the form to ***output*** just before you place the output pin.

6.3 Click on ***Hide*** to hide the Create Pin form.

6.4 Move the cursor to the location for the first pin (CLK).

6.5 Click to Place the pin.

If you now press ***F3*** to redisplay the Create Pin form you will notice that ***CLK*** is no longer on the list of pins. Each time you place a pin, the name is removed from the list.Click on ***Hide*** again to hide the form.

6.6 Click twice to place the remaining two input pins in the required location. ***Do not*** click yet to place the output pin.

6.7 Press ***F3*** to redisplay the from.

6.8 Change the ***direction*** on the Create Pin form by

6.8.1 Move the cursor onto ***input*** in the Direction field.

6.8.2 Hold down the left mouse button to pop up the available values.

6.8.3 Slide to ***output***

6.8.4 Release the mouse button.

6.9 Click on ***Hide*** to hide the Create Pin form.

6.10 Move the cursor to the right of the ALU and click to place the output pin.

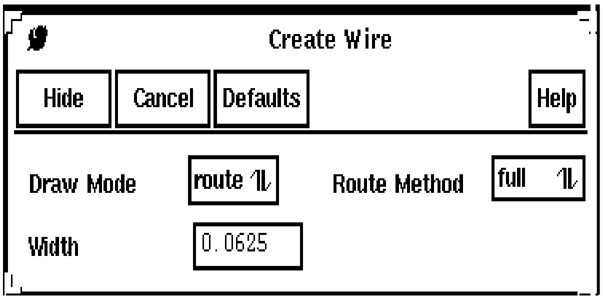
6.11 With the cursor on the schematic window, press ***ESC*** to cancel the ***Create Pin*** command.

7.0 Creating the Buses

The top-level schematic contains two buses, the DEC bus and the bus connecting the schematic output pin.

7.1 Click on ***Create->Wire(wide)*** on the schematic menu banner.

A form will appear for you to fill in



7.2 To set the form for drawing the bus in straight lines

7.2.1 Move the cursor onto the ***Draw Mode*** box,press the left mouse button to display the options, slide to ***route***, and release the mouse button.

7.2.2 Replace the width value of 1 with 0.0625

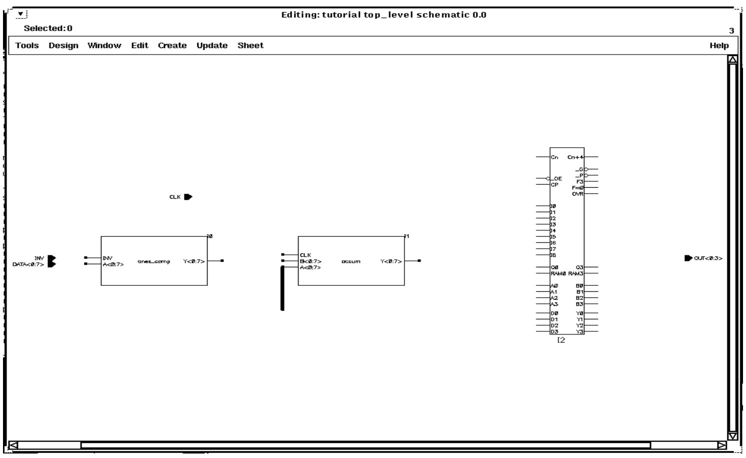
The CIW prompts you to start drawing the bus by displaying:

‘Point at first point of wire’

7.3 Click on ***Hide*** to hide the form.

7.4 Click on the ***accumulator A<0:7>*** pin to start drawing the ***DEC*** bus.

7.5 Click at the end of the first segment of the bus.



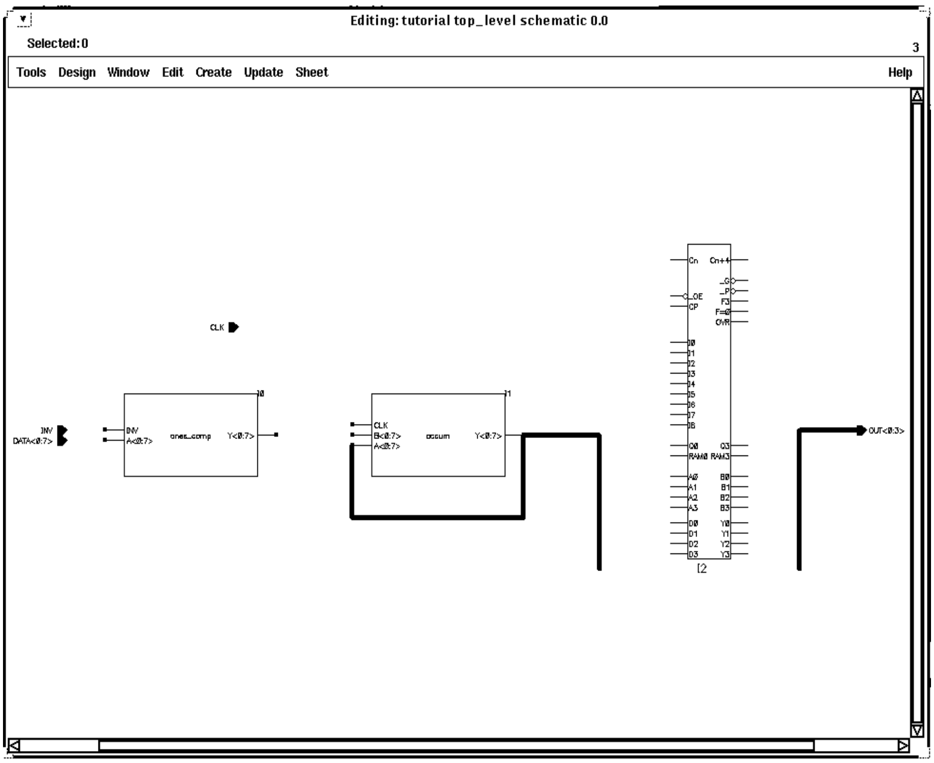
7.6 Click at the end of the second segment of the bus.

7.7 Click on the ***Y<0:7>*** pin to complete the first part of the ***DEC*** bus.

7.8 Similarly, create the rest of the ***DEC*** bus and the ***output*** bus. Double click to end these buses. A double click is used to end a bus when it is not connected to a pin.

7.9 Press ***ESC*** to cancel the ***Create Wire*** command.

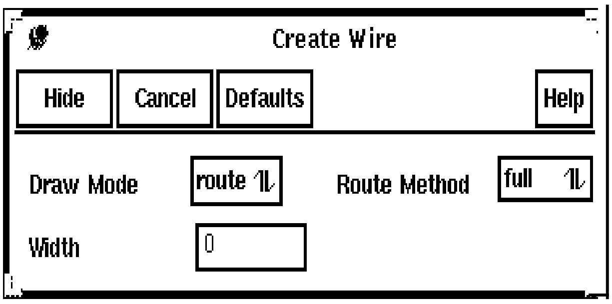
You should now have a schematic that looks similar to the figure below



8.0 Creating the Wires

8.1 Click on ***Create-Wire(narrow)*** on the schematic menu banner.

The ***Create Wire*** form appears.



8.2 Move the cursor onto the ***Draw Mode*** box, press the left mouse button to display the options, slide to ***route***, and release the mouse button.

8.3 Click on ***Hide*** to hide the form.

8.4 To create the wire

8.4.1 Click on the schematic ***CLK input pin.***

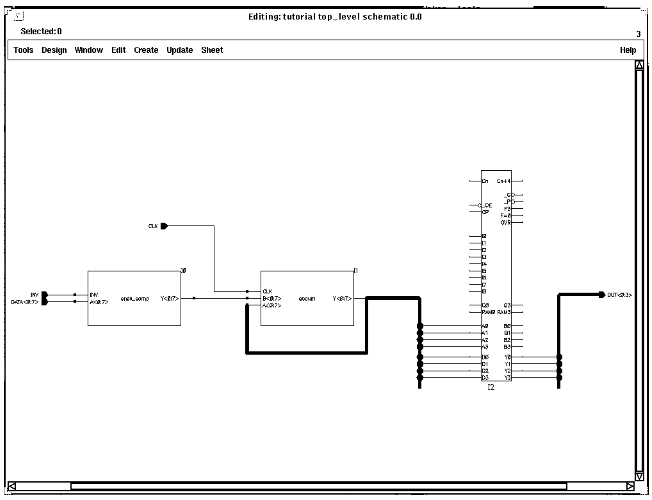
8.4.2 Move the cursor onto the ***accumulator CLK input pin***. The wire appears as a solid white line that follows the cursor, and shows the wire route.

8.4.3 Click to place the wire. The wire is automatically routed.

8.5 Similarly, create the remaining wires.

8.6 Press ***Esc*** to cancel the Create Wire command.

Your design should now look like the diagram below.



Note: You may need to ***zoom*** into an area so that you can see to connect the wires correctly. To do this press ***z*** then use the mouse to specify the area to zoom in on.

9.0 Labelling the Wires

9.1 Zoom in on the wires attached to the ALU.

9.2 Click on ***Create->Wire Label*** on the schematic menu banner

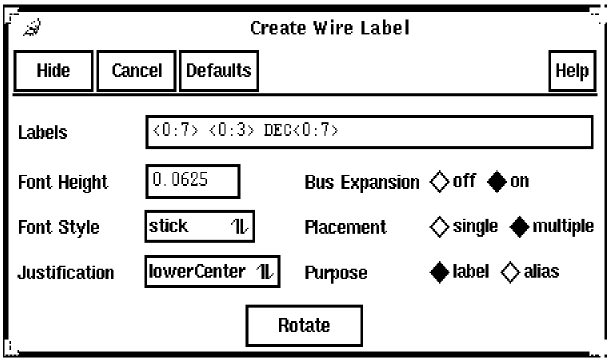
A Create Wire Label form appears.

9.3 Fill in the form as follows:

Labels: <0:7> <0:3> DEC<0:7>

Click on ***Bus Expansion on*** to turn it on.

Click on ***Placement multiple*** to turn it on.

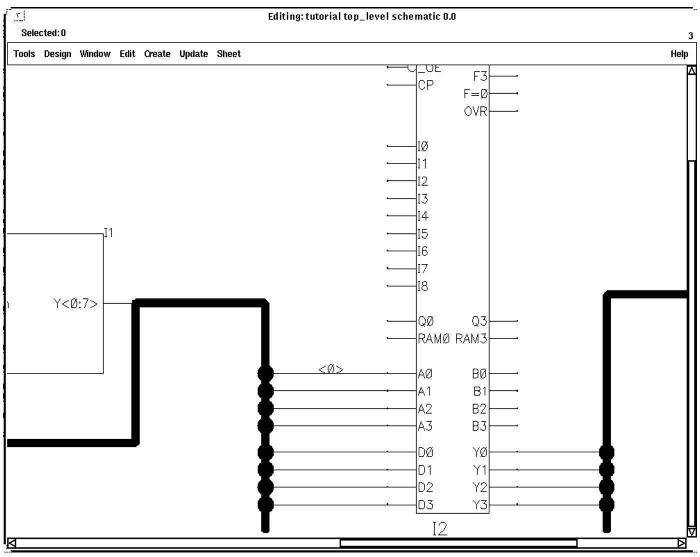


Note: The Bus Expansion button tells the software to extract individual labels from the bus names. The first label will be <0>, the second will be <1>, and so on.

9.4 Click on ***Hide*** to hide the form.

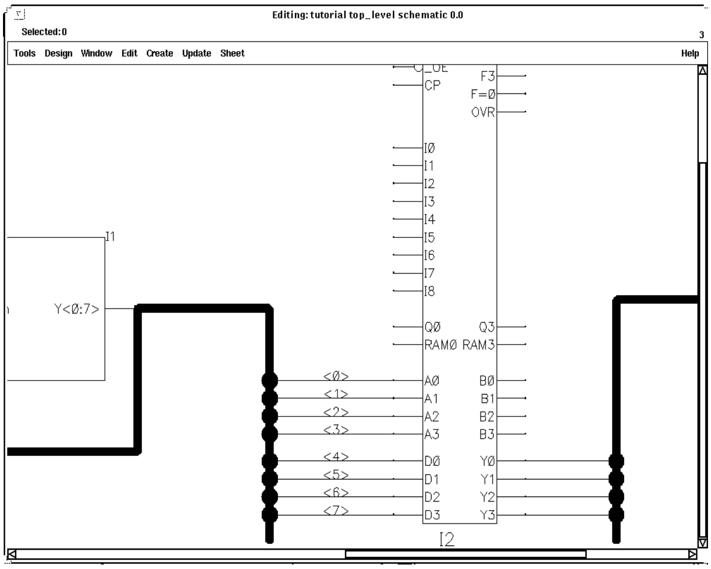
9.5 Click to place the first ***<0>*** label above the wire connecting the ***A0*** pin.

9.6 Click on the wire to glue the label.

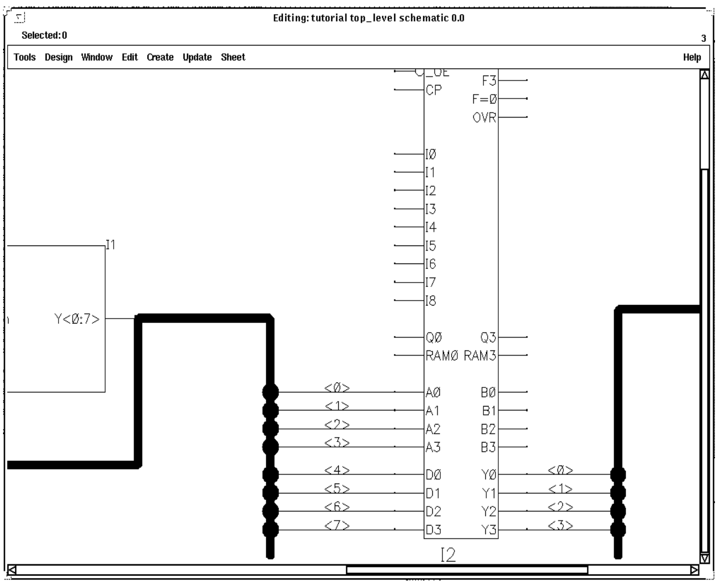


9.7 Move the cursor to the bottom wire ***D3*** to display and position the remaining labels.

9.8 Click on the bottom wire to glue the remaining labels to the wires.



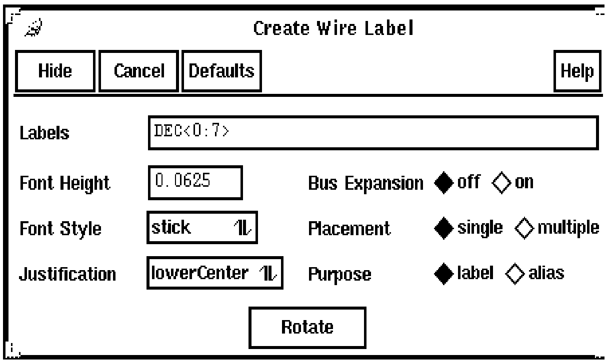
9.9 Repeat the process 9.5 to 9.8 for the ***Y*** pin usingthe output labels ***<0:3>***.



9.10 Press ***F3*** to redisplay the Create Wire Label form.

9.11 Click on ***Bus Expansion off*** to indicate that the ***DEC<0:7>*** label is not expanded

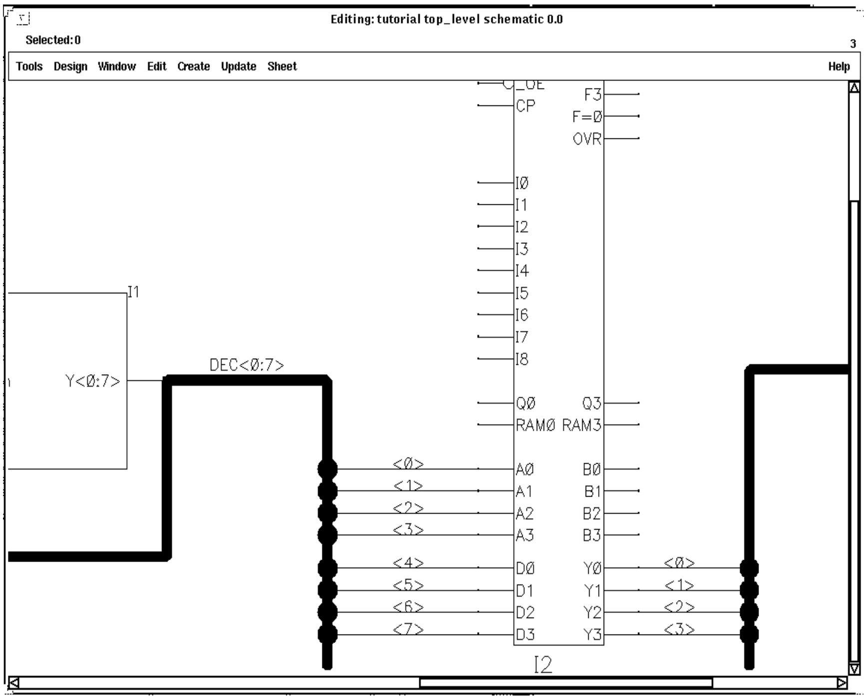
9.12 Set the ***Placement*** to ***single***.



9.13 ***Hide*** the form.

9.14 Click to position the label above the ***DEC*** bus.

9.15 Click on the bus to glue the label to it.



9.16 Press ***Esc*** to cancel the Create Wire Label.

10.0 Checking the Schematic

10.1 Click on ***Design->Check and Save*** on the schematic menu banner.

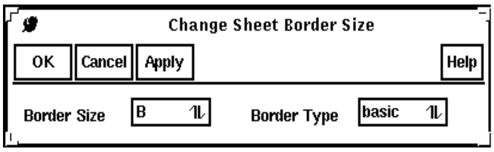
10.2 If any errors occur then a dialog box appears indicating the number of errors and warnings found. Click on close in the dialog box.

10.3 If no errors occur then the design will be saved automatically.

11.0 Attaching the Border

11.1 Click on ***Sheet->Edit->Size*** from the schematic menu banner.

11.2 Click on Border Size, silde down the options to size B, release the button.



11.3 Click on ***OK*** to accept this value.

'Some of the design may fall outside of the new sheet'

11.4 Press ***f*** to select the ***fit*** command

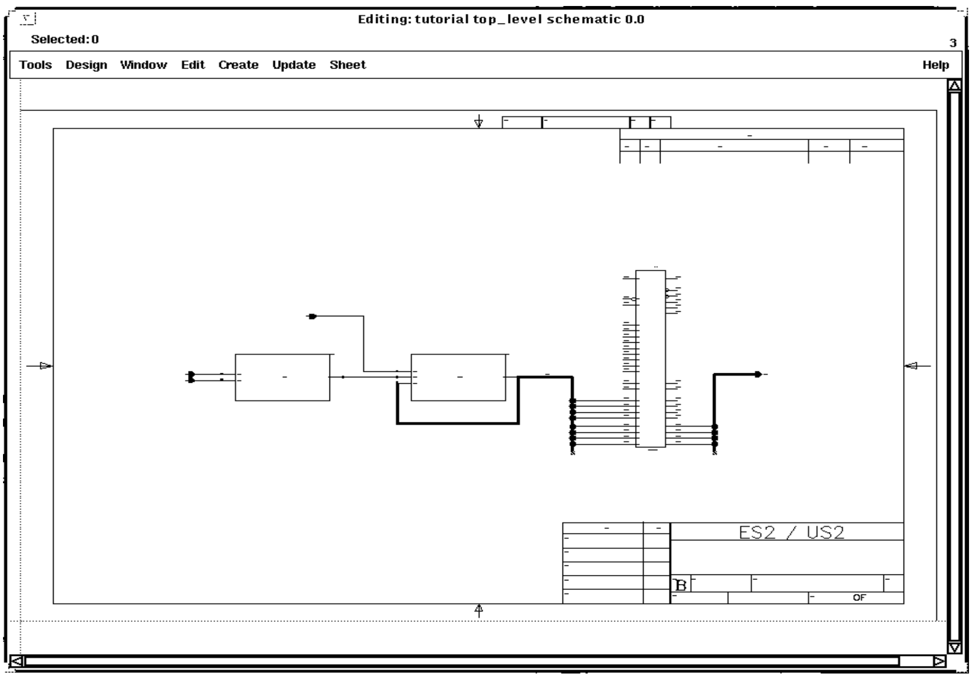
11.5 Hold the left mouse button down, move the cursor to draw a selection box around the design, release the button.The design should be selected.

11.6 Hold down the middle mouse button to reveal the menu

11.7 Select the ***Move*** option

11.8 Click anywhere on the selected design

11.9 Click to place the design within the border.

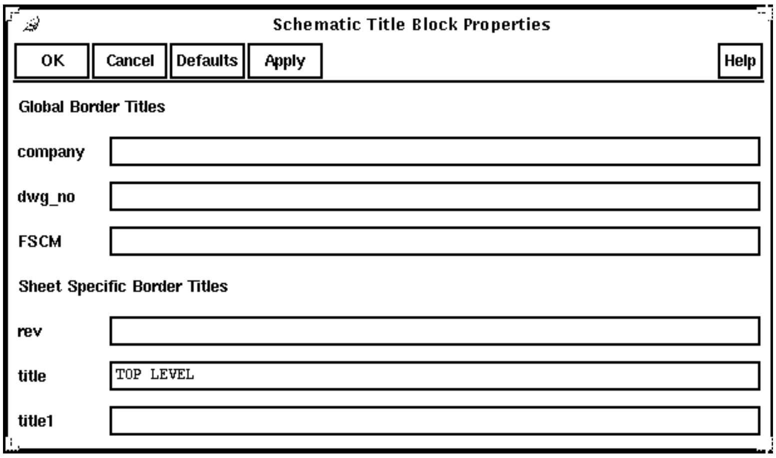


11.10 Click on the background of the schematic window to deselect the design.

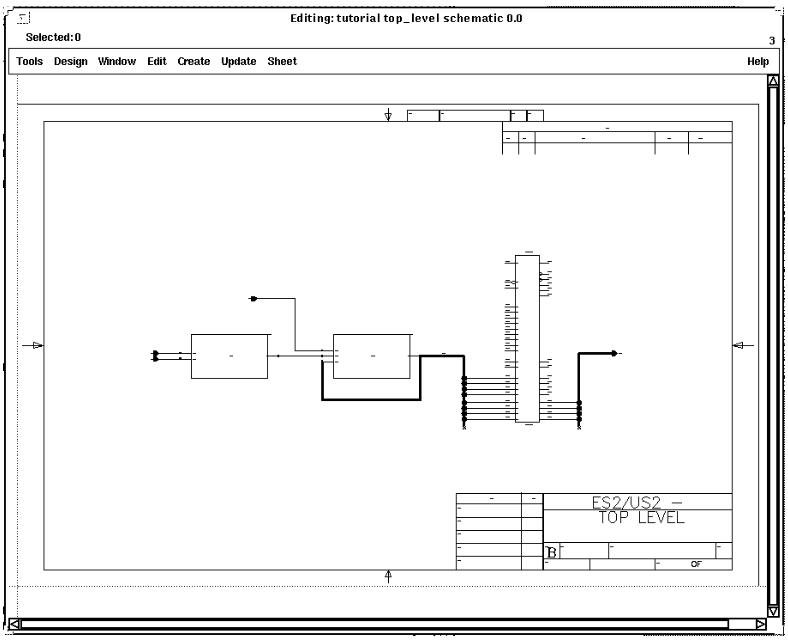
11.11 Click on ***Sheet->Edit->Title*** from the schematic menu banner.

A form appears for you to fill in.

11.12 Fill in the ***Title field***.



11.13 Click on ***OK***.



11.14 Click on ***Design->Save*** to save your design.

11.15 Click on ***Window->Close*** to close the schematic window.

12.0 Checking in Your Design

Checkin saves your design to a new version number. The schematic window must be closed to be able to checkin. This provides a convenient method for saving design milestones.

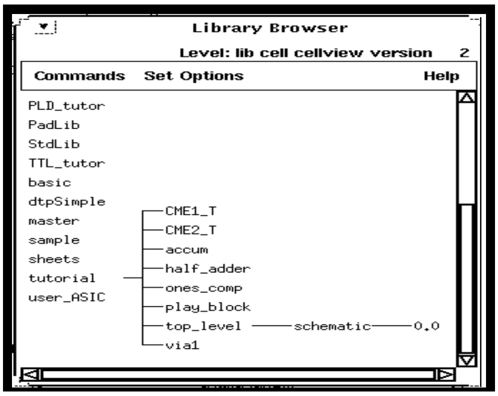
12.1 Click on ***Design Manager->Library*** Browser on the ***CIW.***

The Library Browser now appears.

12.2 Click on ***tutorial***

12.3 Click on ***top_level***

12.4 Click on ***schematic***

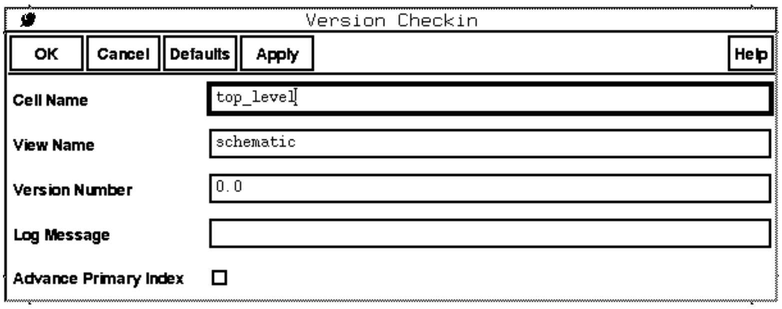


12.5 Hold down the middle mouse button on ***0.0*** to pop up the version-level menu.

12.6 Slide to ***Checkin***

12.7 Release the mouse button.

The ***Version Checkin*** form appears.You do not need to change any values on the form



12.8 Click on ***OK***.

The new version number is displayed on the Library Browser. Since you did not select ***Advance Primary Index*** on the V***ersion Checkin*** form, the new version number is ***0.1***

12.9 Click on ***Commands->Close Browser*** to close the Library Browser.

13.0 Creating the accum Schematic Cellview

In this section you will descend edit the Accumulator symbol to add the schematic.

13.1 Start the DF2 software, unless it is already running.

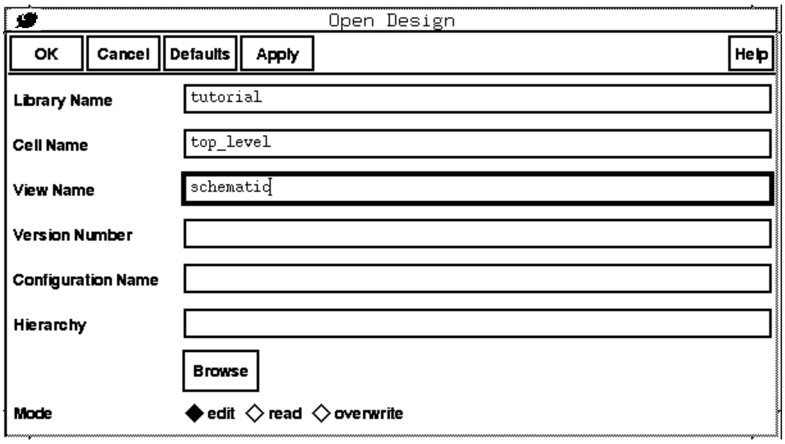
13.2 Click on ***Open->Design*** on the CIW menu banner.

13.3 Fill in the form as follows:

Library Name tutorial

Cell Name top_level

View Name schematic



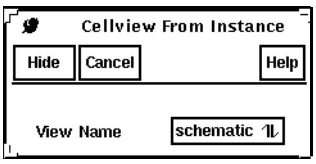
Note: you can fill in the form by typing in the values or using the Library Browser

13.4 Click on ***OK*** on the form.

The ***top_level*** schematic window appears.

13.5 Click on ***Create->Cellview->From Instance*** on the schematic command banner.

13.6 Check that the View Name on the Cellview From Instance form is set to ***schematic.***



13.7 Click on the ***accumulator instance*** in the top_level design to generate the schematic cellview.

13.8 Press ***Esc*** to cancel the Create Cellview From Instance command.

14.0 Descending the Schematic Hierarchy

14.1 Move the mouse onto the ***accumulator*** instance.

14.2 Hold down the middle button to pop up the Instance menu.

14.3 Choose the ***Descend Edit*** option.

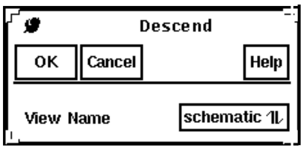
The first Descend form appears and the CIW displays the prompt:

‘ Point at instance to descend into’

14.4 If you want to hide the descend form, click on ***Hide***.

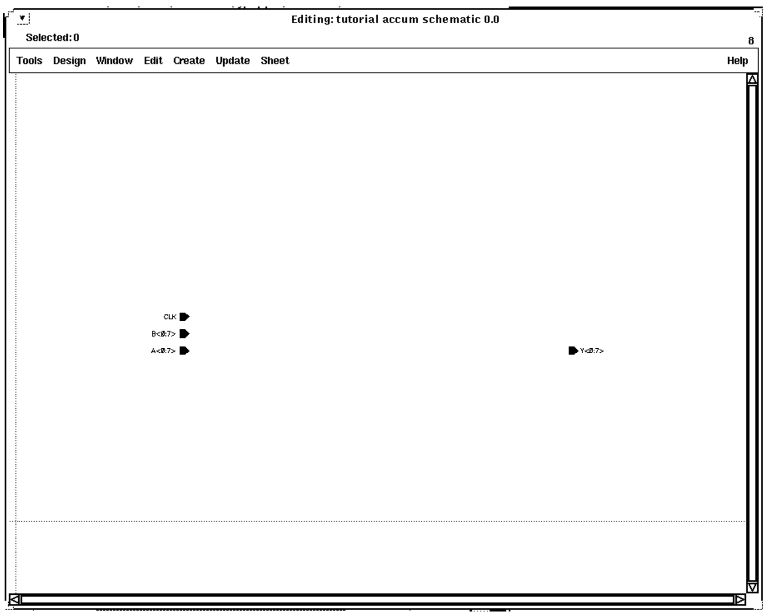
14.5 Click on the ***accumulator*** instance.

The second Descend form appears. Check that the ***View Name*** is set to schematic.



14.6 Click on ***OK*** on the second Descend form

The ***accum*** schematic appears. It contains the input and output pins created by the Create Cellview From Instance command.



15.0 Moving the Input and Output Pins

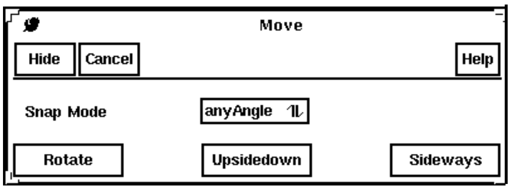
The three input pins need to be moved to the top left of the schematic. The output pin also needs to be moved up so that theY bus will fit in.

15.1 Hold down the Shift key and click on the ***CLK***, ***B<0:7>***, and ***A<0:7>*** pins to select them.

15.2 Hold down the middle mouse button to reveal the ***Multiple Menu.***

15.3 Choose the ***Move*** option.

The CIW asks you to point at the reference point for move and the move form appears.



15.4 If you want to hide the form, click on ***Hide***.

15.5 Click anywhere on the selected pins.

15.6 Move the cursor to position the pins.

15.7 Click to place the pins.

15.8 Press ***Control-d*** to deselect the pins

Note: Control-d is the bind key for deselecting all objects

15.9 Press a capital ***M*** to select the ***Move*** command.

The Move form appears and the CIW asks you to point to the object to move.

15.10 If you want to hide the from, click on ***Hide***.

15.11 Click on the output pin.

15.12 Move the cursor to position the pin.

15.13 Click to place the pin.

16.0 Creating and Placing the TTL Instances

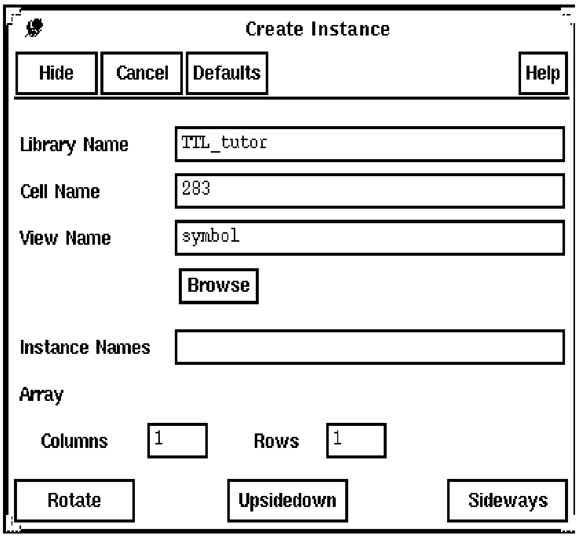
16.1 Select the ***Create Instance*** command from the menu banner or from the pop up schematic menu.

16.2 Fill in the form as follows:

Library name TTL_tutor

Cell Name 283

View Name symbol



Note : When you move to the ***Cell Name*** field a dialog box appears, prompting you to add TTL_tutor to the reference list. Click on ***OK*** to accept this.

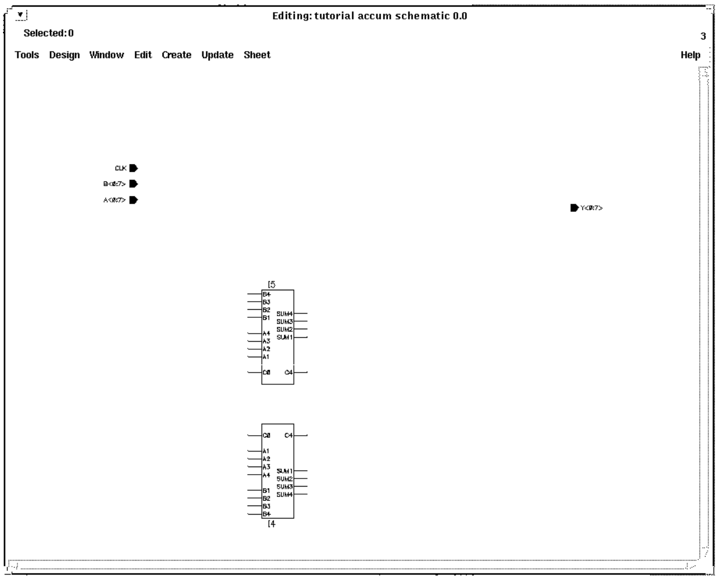
16.3 Click on ***Hide*** to hide the form.

16.4 Click to place the instance at the bottom of the screen.

16.5 Press ***F3*** to redisplay the Create Instance form.

16.6 Click on ***Upsidedown*** on the form.

16.7 Click to place the instance above the first 283 instance.



16.8 Press ***F3*** to redisplay the Create Instance form.

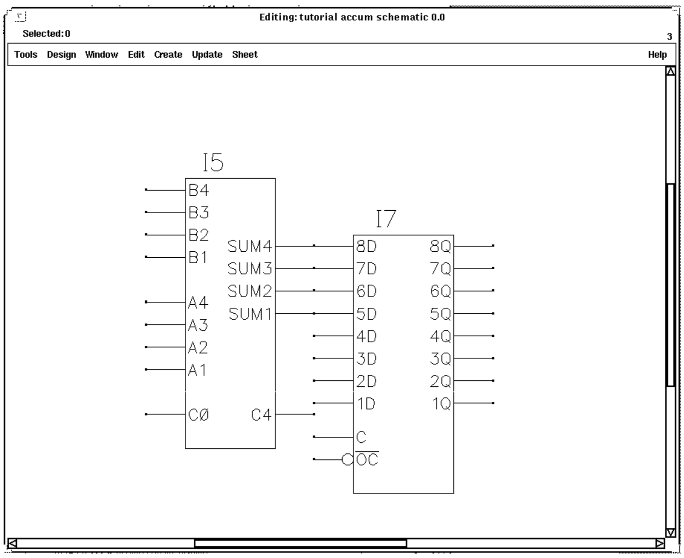
16.9 Fill in the ***Cell Name*** field with ***373***.

16.10 Click on ***Hide*** to hide the form.

16.11 Press *z* to select ***Zoom In***.

16.12 Zoom in on the top 283 instance.

16.13 Click to place the ***373*** instance so that the top four input pins are touching the four SUM output pins of the top ***283*** instance.



16.14 Press ***Esc*** to cancel Create Instance.

16.15 Press ***f*** to redisplay the entire design.

17.0 Stretching the 373 TTL instance

By stretching the 373 instance you automatically create and route the wires connecting the two instances.

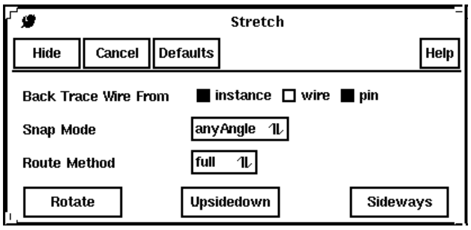
17.1 Move the cursor onto the ***373*** instance.

17.2 Hold down the middle button and select the ***Stretch*** option from the menu.

The CIW prompts you:

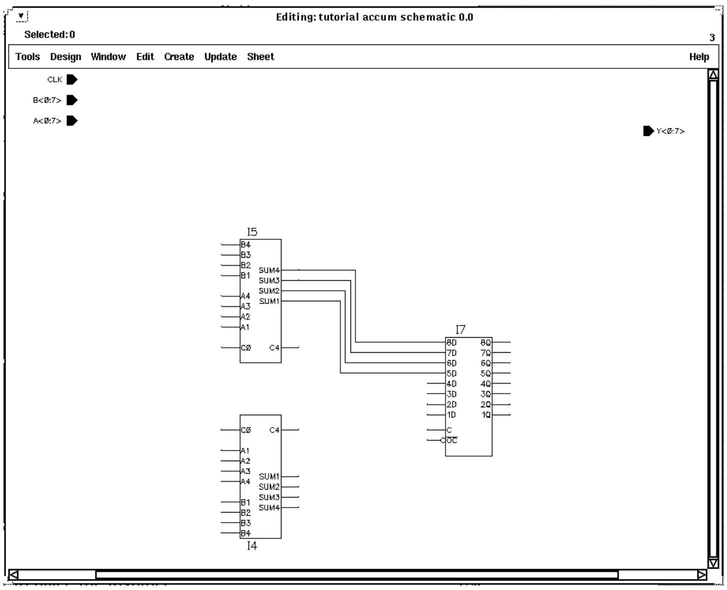
‘ Point at object to stretch’

The stretch form also appears. Use the default values on the form.



17.3 Click on the ***373*** instance to select it for the stretch.

17.4 Move the cursor to reposition the 373 instance.



17.5 Click to place the instance.

17.6 Press ***Esc*** to cancel the Stretch command.

18.0 Creating the Power and Ground Instances

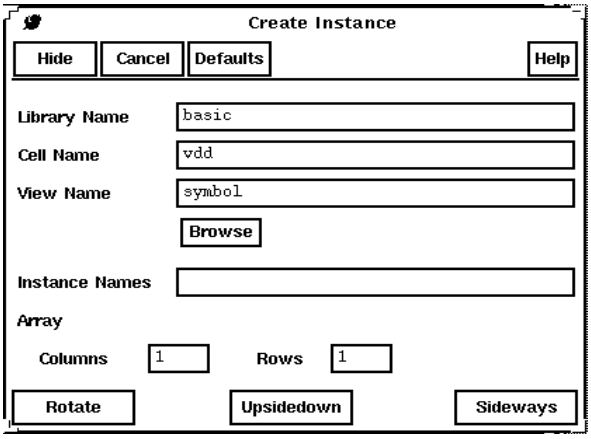
18.1 Select the ***Create Instance*** command from the menu banner or the pop up schematic menu.

18.2 Fill in the form as follows:

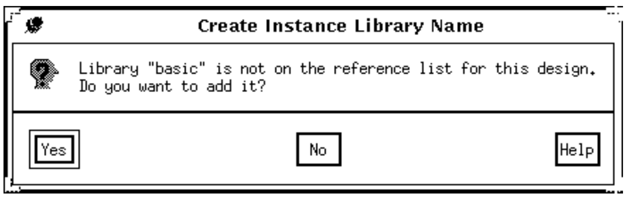
Library Name basic

Cell Name vdd

View Name symbol



Note: You will be asked to add basic to the reference list again, Click on ***OK***.

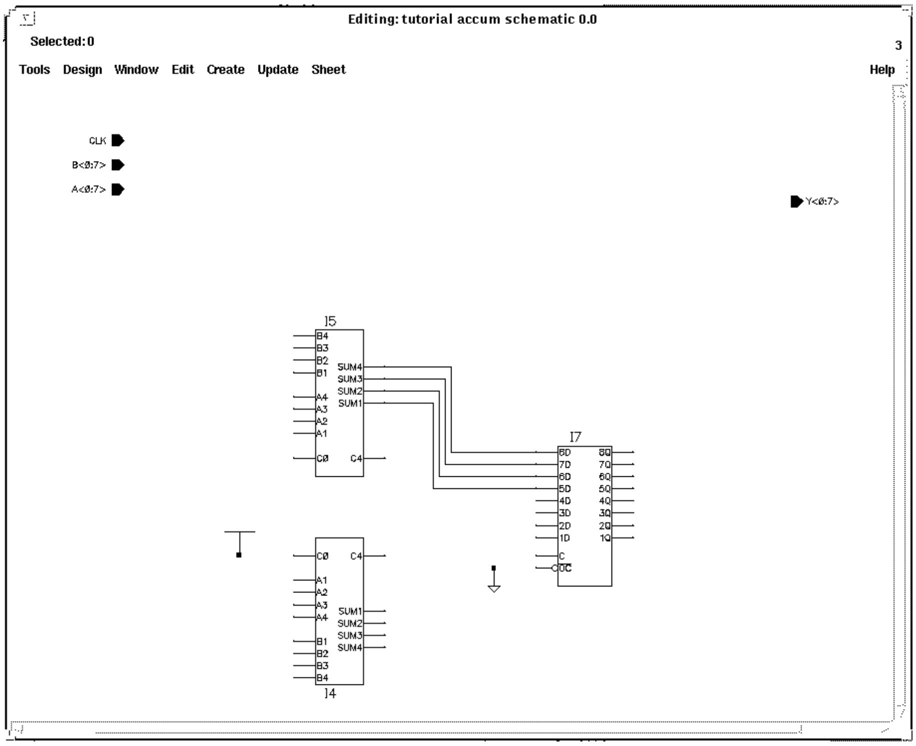


18.3 Click to place the instance close to the C0 pin on the bottom 283 instance. You may need to redisplay the Create Instance form to click on ***Rotate*** or ***Upsidedown***.

18.4 Press ***F3*** to redisplay the form.

18.5 Change the Cell Name to gnd.

18.6 Click to place the ***ground*** instance close to the bottom left pin on the 373 instance.



18.7 Press ***Esc*** to cancel the command.

19.0 Creating the Wires to the TTL instances

19.1 Press *z* to select ***Zoom In***

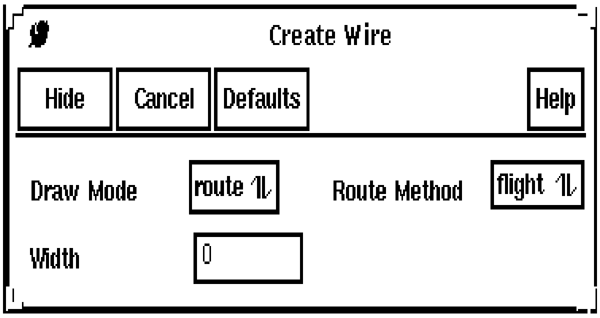
19.2 Zoom in on the three instances.

19.3 Move the cursor onto any instance pin.

19.4 Hold down the middle mouse button to pop up the ***Instance Pin*** menu.

19.5 Select the ***Wire(narrow)*** option.

19.6 On the form, change the Route Method to ***flight***.



19.7 Create the following flight lines by clicking on the pins:

SUM4 - 4D

SUM3 - 3D

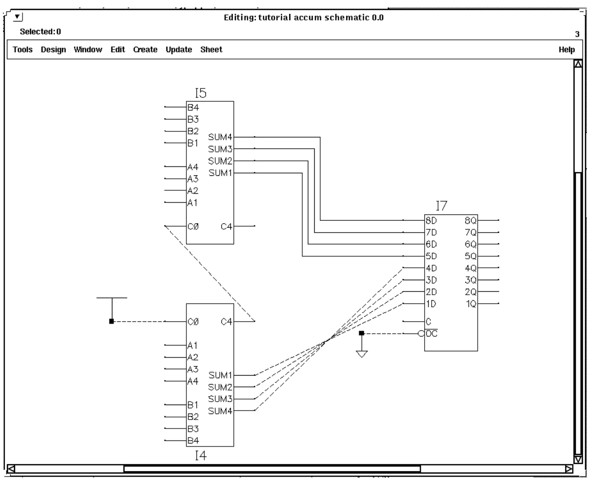
SUM2 - 2D

SUM1 - 1D

C0 (on the top 283) - C4 (on the bottom 283)

PWR - C0 (on the bottom 283)

GND - notOC (on the 373)



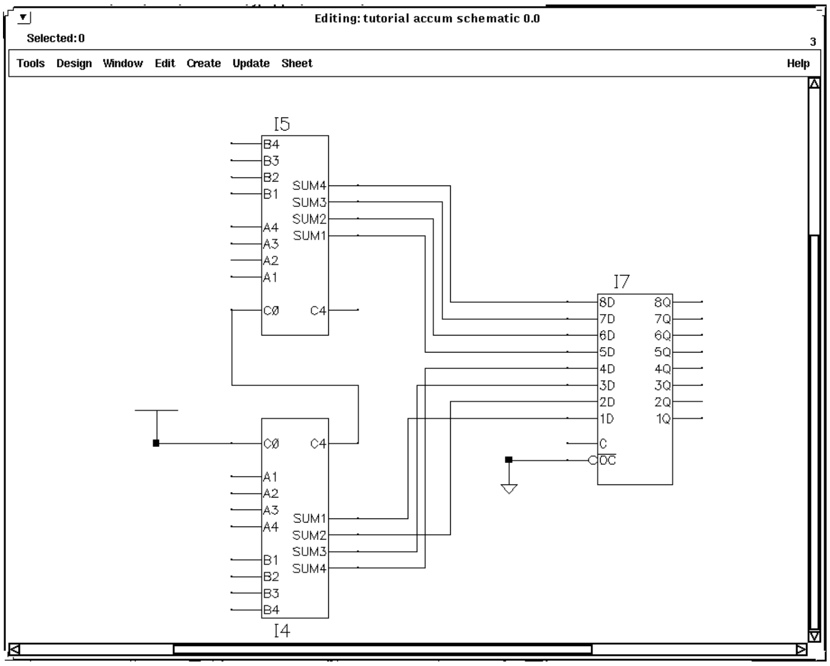
19.8 Click on ***Update->Route Flight*** to select the Route Flight command.

The Route Flight form appears. You must select the flight lines to route them. Route Flight then executes to route the selected flight lines.

19.9 Press ***Control-a*** to select everything on the design

The software automatically routes the wires.

Note: ***Control-a*** is the bind key for selecting everything on the design. Since Route Flight acts on flight lines only, you can select everything.



19.10 Press ***Esc*** to cancel Route Flight.

19.11 Press ***f*** to redisplay the entire design.

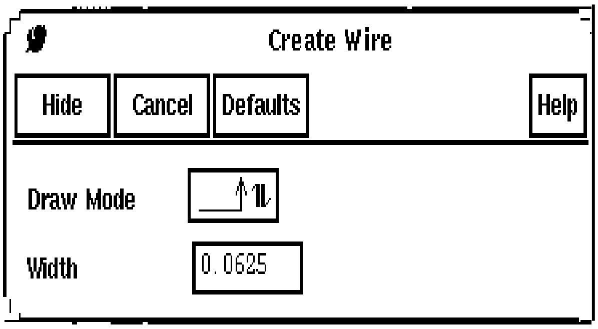
20.0 Creating the Buses

20.1 Move the cursor onto one of the schematic input or output pins.

20.2 Hold down the middle mouse button to reveal the ***Pin*** menu.

20.3 Choose ***Wire(wide).***

The Create Wire form appears. We want to change the form so that it will draw in L-shaped lines.



20.4 Change the Draw Node to that above.

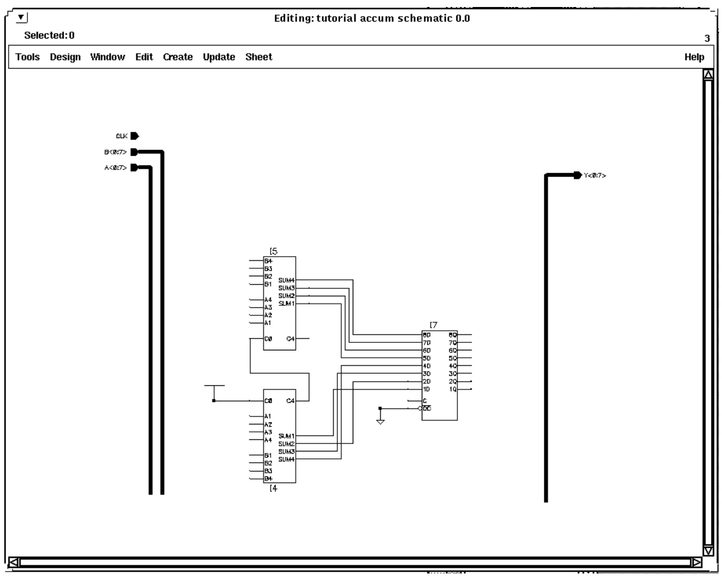
20.5 Change the Witdh value to 0.0625.

20.6 Click on the ***A<0:7>*** input pin.

20.7 Move the cursor to position the bus.

20.8 Double click to place the A bus.

20.9 Similarly, create the remaining buses.



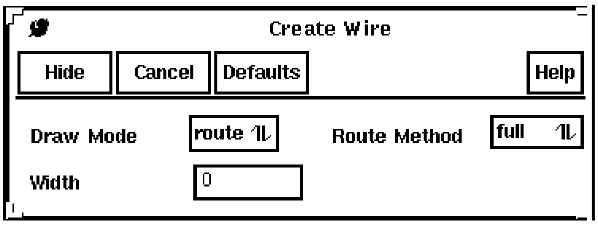
20.10 Press ***Esc*** to cancel Create Wire.

21.0 Creating the Wire to the CLK Pin

21.1 Click on ***Create->Wire(narrow)*** on the schematic menu banner.

21.2 Change the Draw Mode to ***route*** on the form.

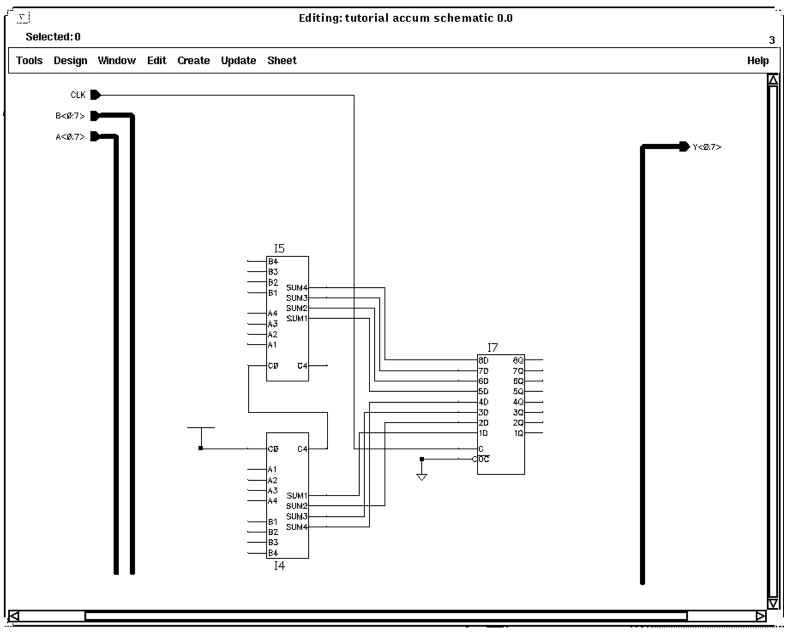
21.3 Change the Route Method to ***full***.



21.4 Click on the ***CLK*** pin.

21.5 Click on the ***C*** input pin on the ***373*** instance to route the lines.

Note: Remember, your schematic might not look exactly like the one shown here. This does not matter, as long as the pins are connected properly.



21.6 Press ***Esc*** to cancel the command.

22.0 Wiring the Instances to the Buses

22.1 Click on ***Create->wire(narrow)*** from the schematic menu banner.

22.2 Check that Draw Mode = ***route*** and Route Method = ***full***.

22.3 Click to create these wires:

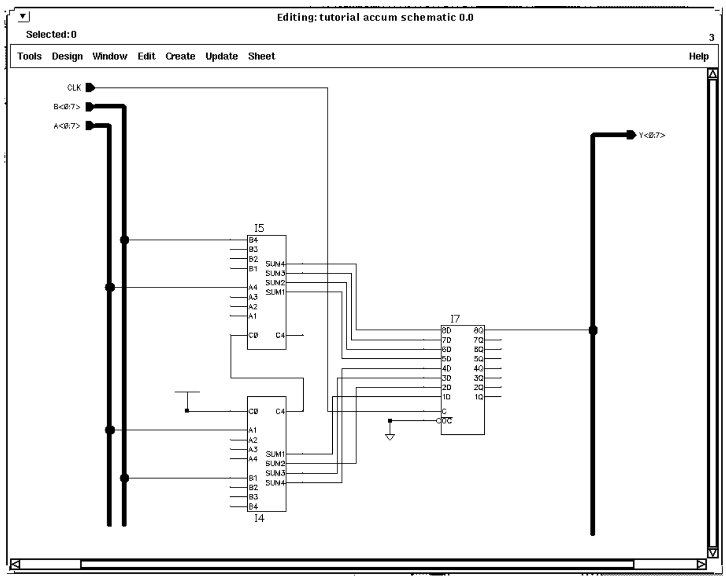
B4 to B bus on the top 283 instance

A4 to A bus on the top 283 instance

A1 to A bus on the bottom 283 instance

B1 to B bus on the bottom 283 instance

8Q to Y bus on the 373 instance



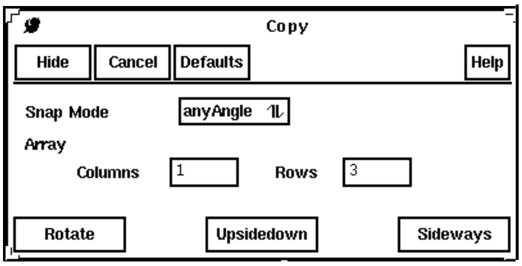
22.4 Press ***Esc*** to cancel the Create Wire command.

22.5 Click on ***Edit->Copy*** on the schematic menu banner.

The Copy form appears. The prompt appears in the CIW:

‘Point at object to copy’

22.6 Type ***3*** in the ***Rows Field*** on the Copy form to indicate that you want to make three copies of the wire.



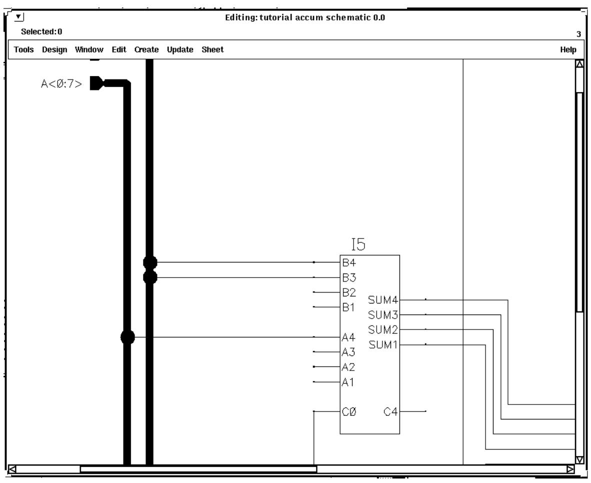
22.7 Click on the top input wire on the top 283 instance to indicate that you want to copy it.

The CIW displays the prompt:

‘Point at destination point for first element of array copy.’

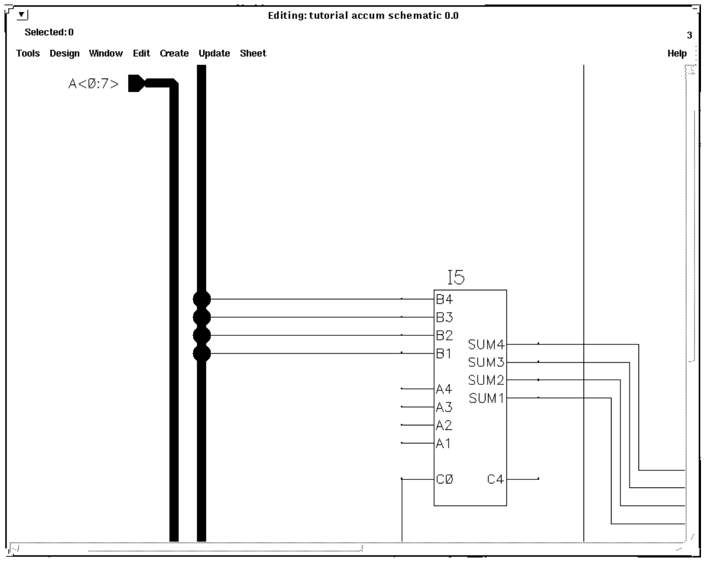
22.8 Position the second wire.

22.9 Click to place the second wire.

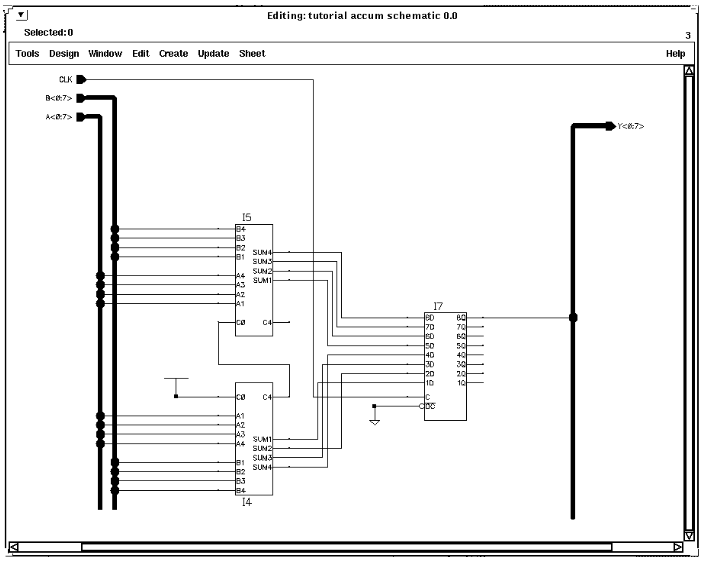


22.10 Position the remaining wires.

22.11 Click to place the remaining wires.



22.12 Similarly, copy and place the remaining input wires.



22.13 Press ***F3*** to redisplay the form.

22.14 Type ***7*** in the Rows field.

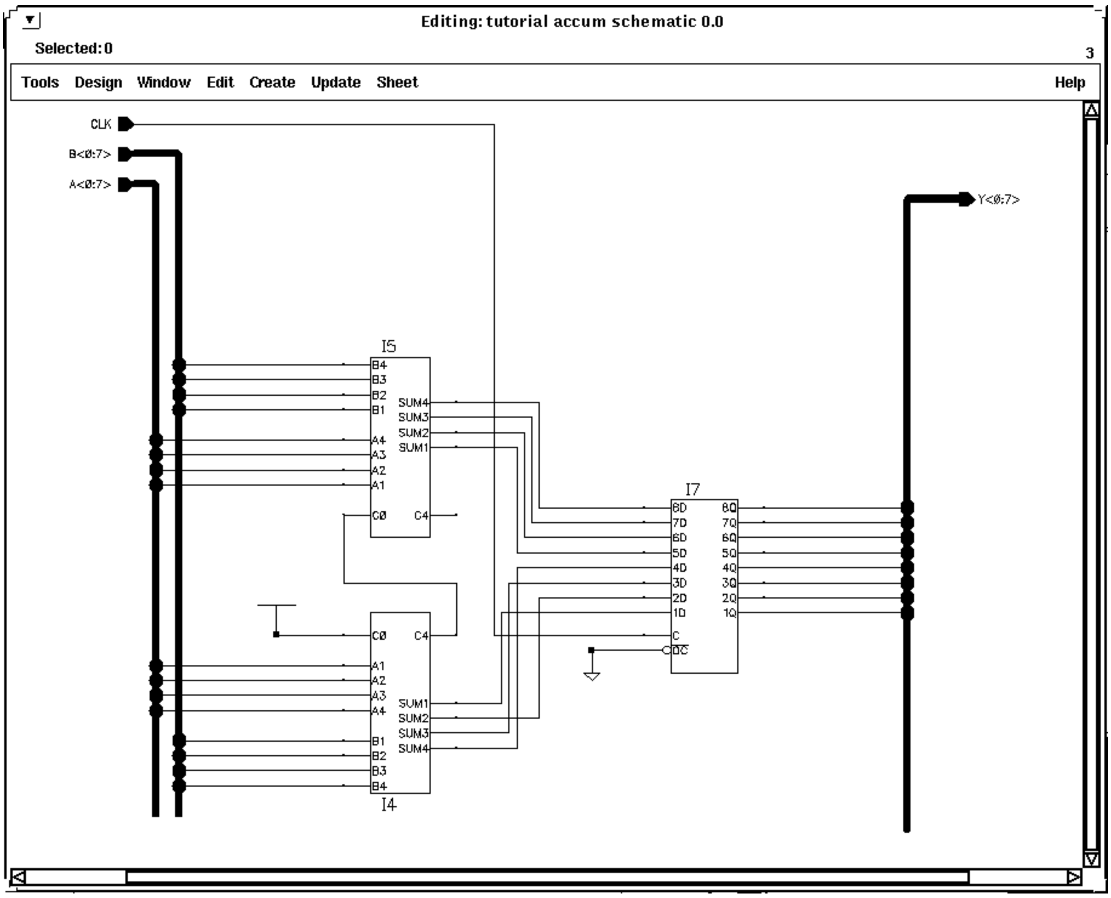
22.15 Click on the wire that connects the 8Q pin on the 373 instance to indicate that you want to copy it.

22.16 Position the second wire.

22.17 Click to place the second wire.

22.18 Position the remaining wires.

22.19 Click to place the remaining wires.



22.20 Press ***Esc*** to cancel the command.

23.0 Labelling the Wires

For labelling the wires you may need to Zoom In on the required areas.

23.1 Move the cursor on one of the wires.

23.2 Press the middle button to reveal the ***Wire menu***.

23.3 Slide to ***Create Label***.

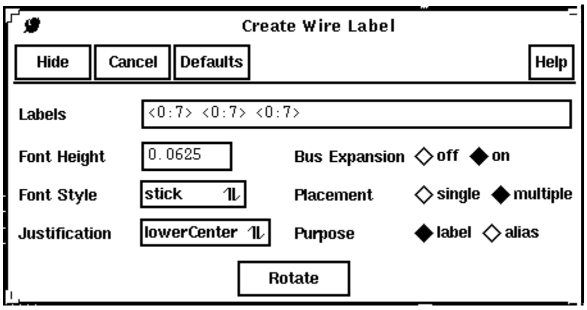
23.4 Release the mouse button.

The Create Label form appears. You need to create three 7-label arrays.

23.5 Type ***<0:7> <0:7> <0:7>*** in the Labels field.

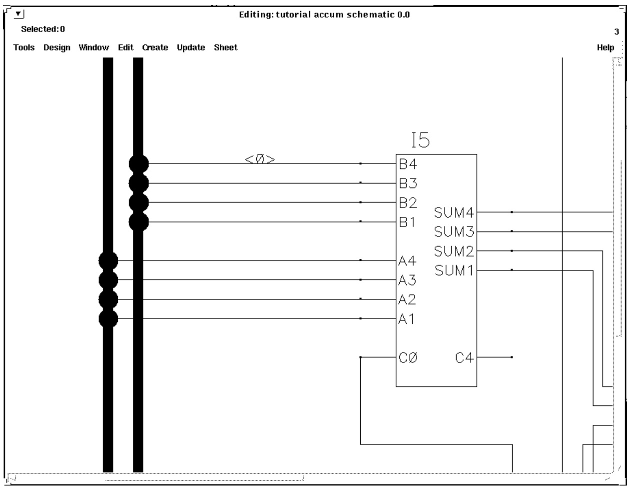
23.6 Click on ***Bus Expansion on***.

23.7 Click on ***Placement multiple***.



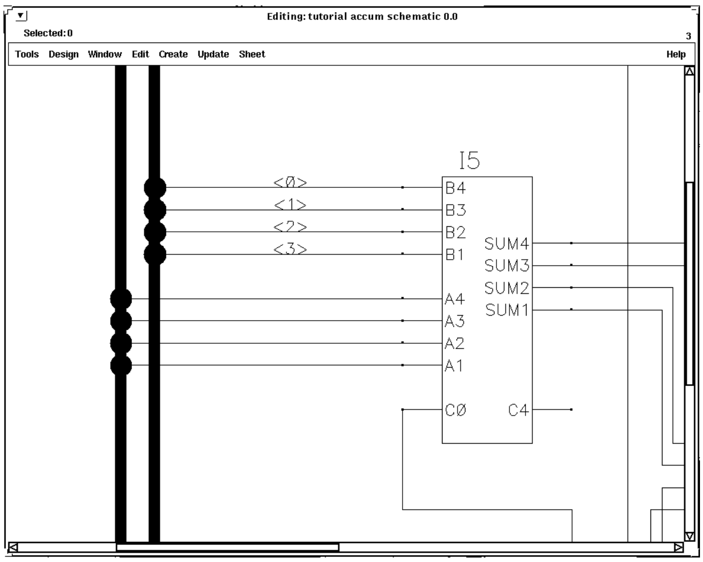
23.8 Click to place the ***<0>*** label above the wire connecting the B4 pin on the top 283 instance.

23.9 Click on the wire to glue the label to it.



23.10 Move the cursor to the ***B1*** pin so that the next three labels are positioned.

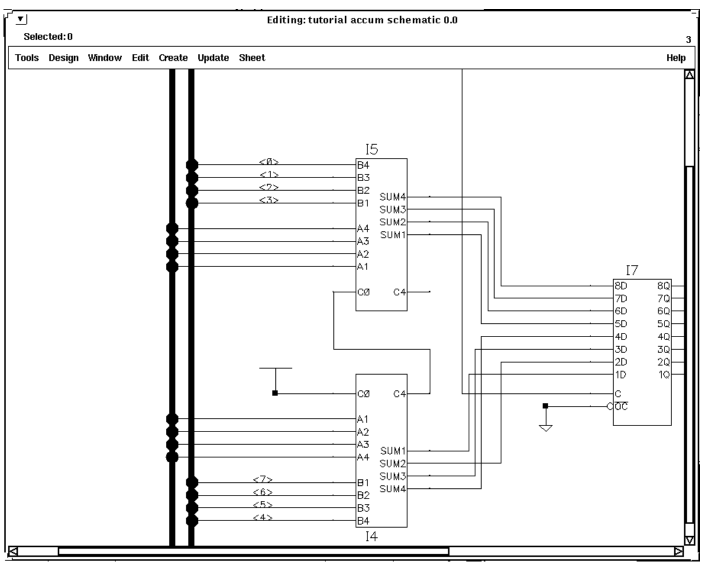
23.11 Click on the bottom wire to glue these three labels.



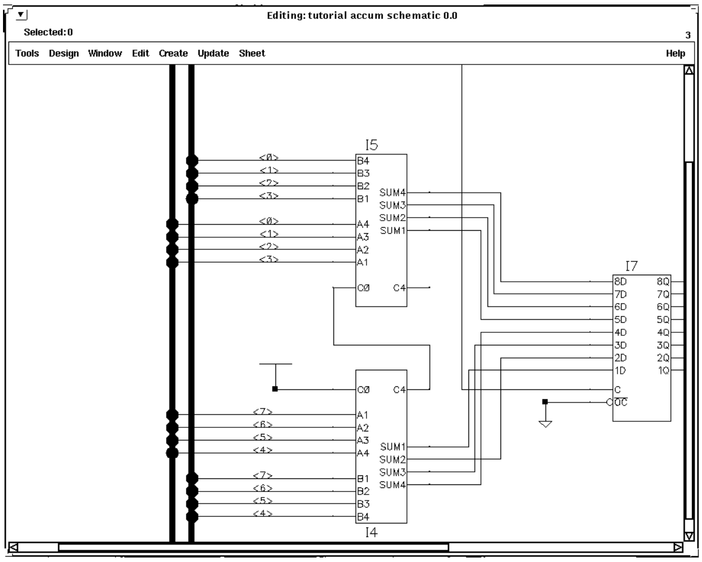
23.12 Click to position the ***<4>*** label above the wire connecting the ***B4*** pin on the bottom instance.

23.13 Click on the wire to glue the label.

23.14 Move the cursor to pin ***B1*** to display the remaining labels in the array, Click to glue the labels.



23.15 Repeat the exercise for the other input pins connected to the A bus.



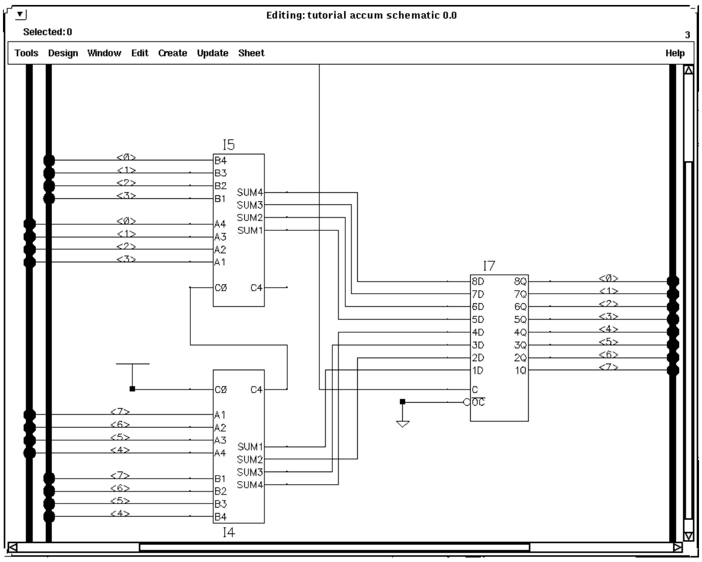
23.16 Move the cursor to the wire connecting the 8Q pin on the 373 instance.

23.17 Click to place the ***<0>*** label.

23.18 Click on the wire to glue the label.

23.19 Move the cursor down to the wire connecting the 1Q pin to display the remaining labels.

23.20 Click on the wire to glue these labels.



23.21 Press ***Esc*** to cancel the command.

23.22 Press ***f*** to redisplay the whole schematic.

24.0 Checking the Schematic

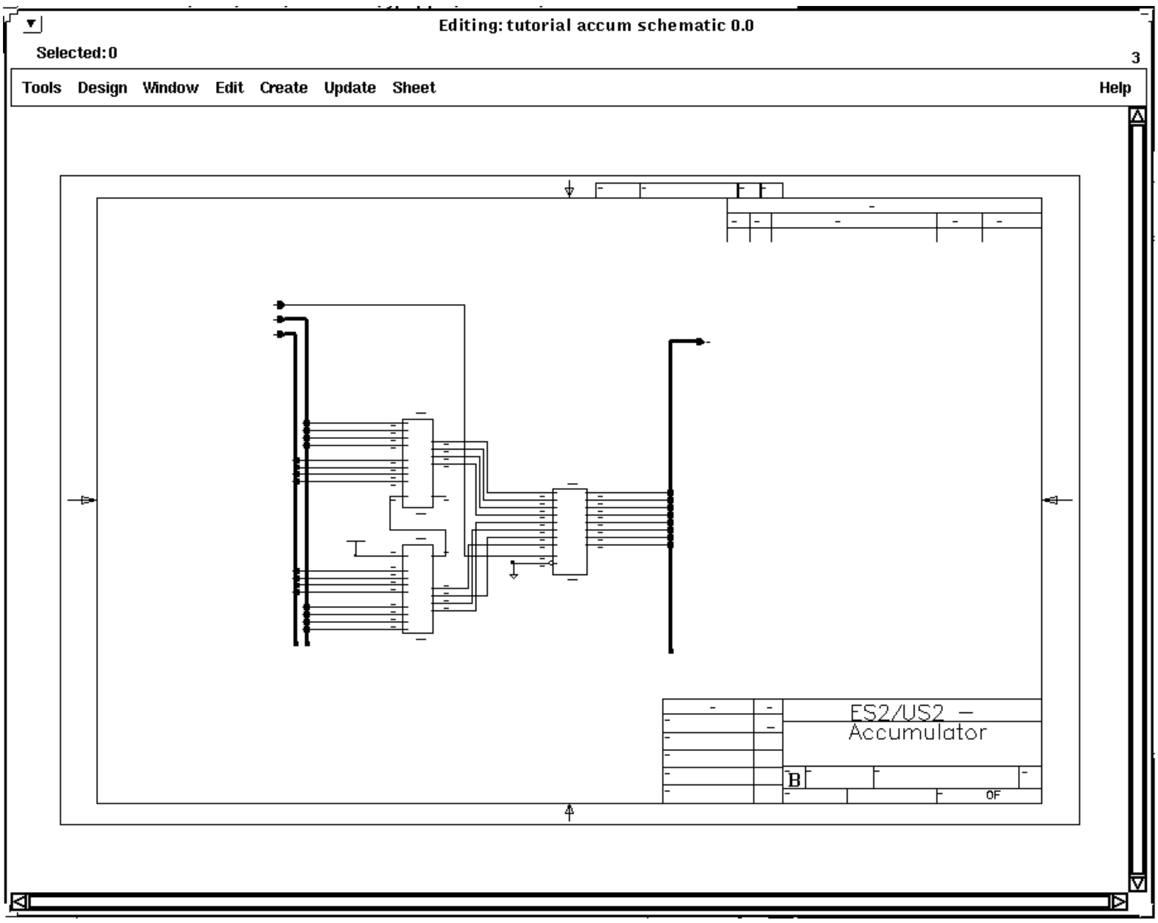
24.1 Click on ***Design->Check and Save*** on the schematic menu banner.

24.2 Click on ***close*** on the dialog box.

"accum schematic" saved'

25.0 Attaching a Border

To attach a border to the design follow the steps in section 11 described earlier.



25.1 Check and save the design.

26.0 Checking in the Design

26.1 Close the ***accum*** and ***top_level*** schematic window.

26.2 Open the ***Library Browser***.

26.3 Click on ***tutorial***.

26.4 Click on ***accum.***

26.5 Click on ***schematic.***

26.6 Hold down the middle mouse button on 0.0 to pop up the ***Version-Level*** menu

26.7 Choose the ***Checkin*** option.

26.8 Click on ***OK*** on the form.

The new version number ***(0.1)*** is displayed on the Library Browser.

26.9 Click on ***Commands->Close Browser*** to close the Library Browser.

26.10 Click on ***Open->Quit*** to exit DF2.