

Date of Last Revision: February 6, 2007

Table of Contents

1. Objective	3
2. <u>Setup & Preparation</u>	4
3. <u>Launching IC Studio</u>	5
4. <u>Creating a project</u>	6
1. Opening icstudio and assigning the project a name	6
2. Specifying Location Map	, 7
3. Specifying Process files and other settings	8
5. Creating a Library and Cells	9
1. Create a New Library	9
2. Capturing a cell	10
3. Design Architect	11
6. <u>Schematic Entry</u> .	12
1. Adding Components and Ports	12
2. Wiring the components	13
3. Adding Text / Changing Labels of Components	14
4. Sizing Transistors & Modifying other Properties	14
7. Checking & Saving the Schematic	16
8. Creating a Viewpoint	17
9. Printing the Schematic	18
10. <u>Creating a Symbol</u>	19

1. Objective

in the second se

2. Setup & Preparation

The set of directives listed below is applicable to users of the *Engineering Design Center* at Santa Clara University. If you are working in a different environment please check with your system administrator.

The steps below are necessary only for the first time to setup the Mentor Graphics environment by changing the settings in your .profile or .cshrc file. Add the following lines in your **.profile**:

setup mentor-2006.1 alias swd="export MGC WD=\'pwd\'"

Remember to execute **\$..profile**

If using C-shell add the following lines in your, cshree source /usr/local/scripts/setup.mentor-2006.1.csh alias swd="export MGC WD=\'pwd\'"

Remember to execute **\$** source .cshrc

3. Launching IC Studio

On the command line:

- To Create a directory to contain your projects type: • mkdir analogtut
- To change the current directory to Tutorial type: cd analogtut
- To open ICSTUDIO type: "icstudio&"

* To open ICSTODIO type. "icstudio&"	
This launches the ICStudio window shown below.	
— ICstudio – <no project=""></no>	•
<u>Eile Edit T</u> ools <u>H</u> elp	
Selected:	
Cell	
View	
Note: Reading user settings Note: ICstudio startup complete	
<u> C</u> Log	

4. Creating a project

To create a project the follow the three steps given below:

1. Opening icstudio and assigning the project a name

On the ICStudio Window

- Click **File > New > Project** to create a new project.
- Click Next in the New project pop-up window
- Enter the **Project name** (e.g analog_tutorial) and the **Project Location** i.e., the name of the directory you created (e.g. analogtut) to contain the project, and click **Next** in the **New project** pop-up window.

	ICs	tudio - «No Project»	•
<u> </u>	s <u>H</u> elp		
🗲 🖉 X 🖻	6 🛍 🗙 🖬 🕅 📢		
Selected:			
Libr	ary	Cell	
		New Duciest	
		New Project	
	Name and Locat	ion	
		Project name:	
	Introduction	analog tutorial	
	Name and Location		_
	Technology Settings	Project location:	
	Summary	/users/student/rwhite/analogtut	
			_
Note: Reading			
Note: IUstud: Note: Reading			
	🕐 <u>H</u> elp	< <u>Back N</u> ext > <u>C</u> ancel -	
🖉 Log			

2. Specifying Location Map

On the next window that appears

- Click the **Open Location Map Editor** button. The Location Map Editor appears.
- To Add the *design kit's standard cell libraries* to the location map

Click Edit Menu > Add Standard MGC Libraries pull down menu item.

Following libraries will be automatically added.

/opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/mgc_ic_commlib /opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/mgc_ic_comm_rff /opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/device_lib /opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/device_lib /opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/generic_lib /opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/sources_lib /opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/sources_lib /opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/mgc_ic_verilog /opt/mentor-2006.1/app/2006.1_sun4os5/icflow_home/mgc_icstd_lib/mgc_ic_macrolib

- Add the MGC design kit to the location map
- Click Edit Menu > Add MGC Design Kit.
- Specify MGC Design Kit path as /opt/mentor-2004.3/sol/adk2 5
- The Library List editor looks as in the figure below.
- Click **OK** on the Library List Editor.
- Click Next on the New Project pop-up window,

_	ICstudio -	- «No Project»
<u>F</u> ile	Edit Tools Help	
) 📂 (New Project
Select	ted:	Library List
		Introduction Name and Location Library List The Library List contains the physical locations of the ICstudio libraries on the file system. Introduction Name and Location Summary Open Library List Editor Image: Control of the Control of the Control of the Intervence Summary Open Library List Editor Note: You may modify the library List Library List Editor Note: You may modify the library List Editor
F	- Library List Edito	pr
	B ₂ B ₂ A → B ²⁺ Ø New Row Import Move Up Move Drive Del Row Del All	Man Vars Edit Menu
	Library Name Location	ck <u>Next</u> <u>Cancel</u>
	1 MGC_IC_GENERICtor-2006.1/app/2006.1_sun4os5/icflov	v_home/mgc_icstd_lib/generic_lib 🗃
	2 MGC_IC_DEVICE_Ltor-2006.1/app/2006.1_sun4os5/icflov	v_home/mgc_icstd_lib/device_lib 📴
No	3 MGC_IC_SOURCESor-2006.1/app/2006.1_sun4os5/icflow	/_home/mgc_icstd_lib/sources_lib 🚅
No No	4 MGC_IC_VERILOG2006.1/app/2006.1_sun4os5/icflow_hc	ome/mgc_icstd_lib/mgc_ic_verilog 😅
	5 MGC_IC_COMMLIB006.1/app/2006.1_sun4os5/ictiow_hom	ne/mgc_icstd_lib/mgc_ic_commilb
	7 MGC_IC_COMMLIB061/app/2006.1_sun4os5/icflow_home	e/mgc_icstd_lib/mgc_ic_comm_rf 📴
	8 MGC MACROLIB06.1/app/2006.1 sun4os5/icflow home	e/mgc_icstd_lib/mgc_ic_macrolib
	9 MGC_DESIGN_KIT //opt/mentor-2004.3/sol/adk2_5	
	Special Editing Keys: Escape = abort text edit, Enter = accept t	text change
	OF	< Cancel 🕢 Help
╎└		
δ	og	

3. Specifying Process files and other settings

- Click **Open Settings Editor** button. The **Project tab** of the **Preferences** dialog box appears.
- Load the process file (tsmc0XX).

Process files are present in /opt/mentor-2004.3/sol/adk2_5/technology/ic/process

• Load the following rules file to the project: tsmc0XX.rules

Rules files are present in /opt/mentor-2004.3/sol/adk2_5/technology/ic/process

- o DRC rules file: tsmc0XX.rules
- o LVS rules file: tsmc0XX.calibre.rules
- SDL rules file: tsmc0XX.accusim.rules
- PEX rules file: tsmc0XX.calibre.rules

NOTE: Select value for XX from 18, 25, and 35 depending on design requirement

- Fill in any other dialog box field to suit your project needs.
- Click **OK** on the Preferences dialog box.

	ICstudio -	<no project=""></no>	• 🗆
<u>F</u> ile	<u>E</u> dit <u>T</u> ools <u>H</u> elp		
] 🖻	🖉 X 🖻 🛍 X 🔛 🖗 📢	-	New Project
Sele	ected:	Technology Set	tinge
	Library		
		Introduction Name and Location Library List	You can edit the project-specific technology settings at this point.
		Technology Settings	S. Open Settings Euror
		Summary	Note: You may modify these settings later by using the Edit > Preferences menu.
	Preferences		
	General 🐣 Project		
		I=	
	IC Layout HDL Miscellaneous		< <u>Back Next > Cancel</u> -
	Angle mode: 45 degrees 🔻		
4	_ Technology Settings		
	Process file: s/mentor/2004.3/sol/adk2 5/technology/ic/process	s/tsmc035 🚘	
	DRC rules file: http://www.astronometers/technology/ic/process/tsmc	:035.rules 🚘	
	LVS rules file: 04.3/sol/adk2_5/technology/ic/process/tsmc035.ca	libre.rules 🚘	
	SDL rules file: 1.3/sol/adk2_5/technology/ic/process/tsmc035.acc	usim.rules 🚘	
	PEX rules file: 04.3/sol/adk2_5/technology/ic/process/tsmc035.ca	libre.rules 🚘	
	,		
	Cance	l 🕜 Help	
┢╴┖			

- Click Next on the New Project pop-up window.
- View the **Summary** to make sure all the information is correct.
- Click Finish.

5. Creating a Library and Cells

1. Create a New Library

- Click New > Library. This opens the Create Library dialog box.
- Enter the **name of the library** you want to create (e.g. analogtut).
- Click OK.

The library appears in the ICstudio library pane as well as in the location map.



2. Capturing a cell

To create a new Schematic cell:

- Select a library (e.g. analogtut) where you want a cell view to be created
- Click File > New > View. The Create New View dialog box opens.
- Enter the Cell Name (e.g. CommonSource). If the cell does not exist, it is created.
- Specify the View Type as Schematic and click Finish

ICstudio - Project analog_tutorial Elle Edit Iools Help Elle Amore Analogut Ibray Cell Ubray Cell Ubray Cell Ubray Cell Ubray Cell Ubray Cell Ubray Cell Umage Common Source View Type Ibray Name: analogtut Cell Name: Common Source View Type View Type Schematic View Name: Schematic Schematic
Elle Edit Iools Help Selected: analogtut Gell
Note: Reading user settings Note: Reading preferences from cur: Note: Reading preferences for this
Selected: analogtut Cell Gradie Status Cell Gradie Status Gradie Status Note: Reading user settings Note: Icstudio startup complete Note: Reading preferences from cur; Name: Schematic Note: Reading preferences from cur; Name: Schematic Note: Reading preferences from cur; Name: Schematic
Iiirary Cell Iiirary Create New View View Type View Type Iiirary Name: analogtut Cell Name: CommonSourcel View Type: Schematic View Name: Schematic View Name: Schematic View Name: Schematic View Name: Schematic
Image: Schematic Image: Note: Reading user settings Note: Reading preferences from curr Note: Reading preferences from tur Note: Reading preferences from tur Note: Reading preferences for this
Biological C_COMMLIB General Community Biological C_COMMLIB_RF General C_C_COMMLIB_RF Biological C_C_CENERIC_LIB Create New View Biological C_SOURCES_LIB View Type Biological C_MACROLIB View Type Library Name: analogtut Cell Name: CommonSource View Type: Schematic Options View Name: View Name: Schematic View Name: Schematic View Name: Schematic View Name: Schematic
AMGC_IC_COMMLIB_QS AMGC_IC_COMMLIB_RF AMGC_IC_DEVICE_LIB AMGC_IC_SOURCES_LIB AMGC_IC_VERILOG_LIB AMGC_MACROLIB AMGC_MACROLIB AMGC_MACROLIB AMGC_MACROLIB Create New View View Type Library Name: analogtut Cell Name: CommonSource View Type: Schematic Options View Name: Schematic View Name: Schematic Note: Reading preferences from cur: Note: Reading preferences from cur: Note: Reading preferences from this
Avide_C_C_OOMMLB_AP Avide_C_C_OOMMLB_AP Avide_C_C_OEVICE_LIB Avide_C_C_GENERIC_LIB Avide_C_C_SOURCES_LIB Avide_C_C_VERILOG_LIB Avide_C_MACROLIB Avide_C_C_MACROLIB Avide_C_C_MACROLIB Avide_C_MACROLIB A
Mode_C_GENERIC_LIB MGC_IC_SOURCES_LIB MGC_IC_VERILOG_LIB MGC_MACROLIB Library Name: analogtut Cell Name: CommonSource View Type: Schematic View Name: Schematic
MGC_IC_SOURCES_LIB MGC_IC_VERILOG_LIB MGC_MACROLIB Library Name: analogtut Cell Name: CommonSource View Type: Schematic View Type: Schematic View Name: Schematic View Name: Schematic
MGC_IC_VERILOG_LIB MGC_MACROLIB Library Name: analogtut Library Name: analogtut Cell Name: CommonSource View Type: Schematic View Name: Schematic View Name: Schematic View Name: Schematic View Name: Schematic
MGC_MACROLIB analogtut Library Name: analogtut Cell Name: CommonSource View Type: Schematic Options View Name: Schematic View Name: Schematic
Ibrary Name: analogtut Library Name: analogtut Cell Name: CommonSource View Type: Schematic Options View Name: Note: Reading preferences from cur: Note: Schematic
Note: Reading user settings Cell Name: CommonSource Note: ICstudio startup complete View Name: Schematic Note: Reading preferences from cur: View Name: Schematic Note: Reading preferences from tur: View Name: Schematic Note: Reading preferences from tur: View Name: Schematic
Note: Reading user settings View Type: Schematic Note: ICstudio startup complete View Name: Schematic Note: Reading preferences from cur: View Name: Schematic Note: Reading preferences for this View Name: Schematic
Note: Reading user settings Note: ICstudio startup complete Note: Reading preferences from cur: Note: Reading preferences for this
Note: Reading user settings View Name: Schematic Note: ICstudio startup complete View Name: Schematic Note: Opening project "/users/studio Image: Schematic Image: Schematic
Note: Reading user settings View Name: Schematic Note: ICstudio startup complete View Name: Schematic Note: Opening project "/users/stude Image: Schematic Note: Reading preferences for this Image: Schematic
Note: Reading preferences from cur: Note: Reading preferences from this
Note: Reading preferences from cur: Note: Reading preferences for this
Note: Opening project "/users/stude Note: Reading preferences for this
Note: Keading preferences for this
Help < Back Next > Finish Cancel
Total Total Total
دُلْتَ لَمَ اللَّهُ عَلَي ا

3. Design Architect

Design Architect (DA-IC) will open automatically where in you can capture the required cell design.

You will see an empty sheet in the Design Architect window. This sheet is where you will draw your schematic by placing the parts for your circuit and wiring them together. Standard devices and circuits are available in the device lib and MACRO LIB palettes which are accessed by clicking on the **Library** palette button.

Click on the **Show/Hide Palette** Icon indicated below to open Palette Menu. The palette menu will appear on right hand side of the window as shown in the figure below.

2	Add	<u>E</u> dit					_				_	_			срос	5130	V.)	 0,000	 .rog_		1.001			
2	-r 117		Select	Viev		ols j	<u>S</u> etup	<u>R</u> ep	port	<u>Windov</u>	v <u>H</u> el	p												
12 C	<u> ×</u> 🖌	1 🖪		173	٣		21 12	<u> </u>	1	* # 6	3													
	0	(W dae) (Co	Zo	om in	by 2	pernat	ic she	eet1) () ()								(0.320	6, 1.833	12)		Hotke	eys: On
•	-							Scl	nema	tic#1	analo	gtut /	Comm	onSou	rce /	Sche	matic				Q 1	r =		schematic e
																								Session
																								Library
																								Simulation
																				Dale	.#.0			Edit
																				Pale	эше			Draw
																								Text
																								Check & Sa
																								Select
																								By Propert
																								Unselect A
																								Edit
													5	7										Move
													2	[Doloto
																								Lindo
		Sho	w./Hi	de E) alet	te	1																	Flin
	-	-		uo .	aiot																			Rotate
																								Properties
																								Add
																								Instance
Ι.																								Wire
																								Bus/Bundle
																								Add Sourc
											M	essad	ie Are	a										Property
																								Name
	•																					•		Instance
e	: Userv	ware f	ile \$MG	с но	ME/sh	aredAn) kas/ic	studio	Juserv	/are/icst	td clier	it.ample	successf	fully load	ed.									
9	Addir	ng par	ameter '	lambda	" with	value	"0.3"				14/61a av						al a al							

6. Schematic Entry

Now let's start drawing the schematic for the amplifier.

1. Adding Components and Ports

Select the parts for your schematic of the amplifier by choosing from a library of components.

To place transistors on the sheet:

- In the palette menu shown alongside, click on Library.
- In the subsequent palette menu click on Device Lib.
- Select NMOS (4-pin) and place it on the sheet by clicking on the position you wish to place it. Similarly, place ideal resistors and ideal capacitors as shown.
- Click BACK to go to the IC Library palette and click on Generic Lib. Place Portin, Portout, VDD and Ground on the sheet.



schematic edit

Session Library

Simulation

Edit

Draw

Text

Check & Save

Select

By Property

Unselect All

Edit

Move

Сору

Delete

Undo

Properties Add Instance Wire Bus/Bundle Add Source Property Name Instance

•

۲

۰.

Flip

Rotate

2. Wiring the components

To wire up the components:

- Press function key **F3** to wire up the components according to the circuit diagram in Figure.
 - Pressing F3 activates the wiring command.
 - Clicking on **Cancel** in the prompt bar or pressing **Esc** on your keyboard disables the wire command.
- Alternatively, wires can be added by using the schematic edit palette or from menu bar by clicking ADD > WIRE



3. Adding Text / Changing Labels of Components

To change the value or text associated with any components,

- Point the cursor on that text/value and press SHIFT + F7 (Note You do not have to click/select the text or value. Just point the cursor on the value/text.)
- A display prompt bar appears at the bottom with the current value/text in the New Value box. Input the new name in this box and click OK.

For example:

To change the label **NET** of the **PORTIN** symbol to **VIN**,

- Point the cursor on the text NET near the **PORTIN** symbol and press **SHIFT** + **F7**.
- A display prompt bar appears at the bottom. Type **VIN** in the **New Value** box and click **OK**.

Similarly change the label of **PORTOUT** symbol from **NET** to **VOUT**.

Change the values for the capacitors and resistors by pointing to the value and pressing SHIFT + F7.

Note: You can also change sizes of transistors, instance names or any other text values associated with any component in the circuit. Alternatively, you can modify multiple properties of a device as explained in the next section.

4. Sizing Transistors & Modifying other Properties

To change multiple properties of one device (e.g. NMOS) at a time:

• Highlight NMOS device, click right mouse button, and select **Properties > Edit.**

In the box that appears, change following properties if needed:

INST:	M1
L:	5u (drawn length of device: 5 micron)
W:	5u (width of device: 5 micron)
ASIM_MODEL:	NCH (required for simulation)

Note: Click on "Apply" button every time you change any value.

If you want to prepare a layout of your circuit and run LVS check, you also need to modify following properties. (If you just want to do simulations, you may skip these two properties)

PHY_COMP : mn INSTPAR : \$strcat("w=", W, " l=", L)

Important: Change the type for INSTPAR property to "**expr**". Also, you need to type this string carefully, as it is. Specifically, do not forget the **spaces** in this string, as highlighted below for your reference.

```
$strcat("w=", <space>W,<space> "<space>l=",<space>L)
```

In case, there is no INSTPAR property (or any other required property) listed in the multiple properties table of a device, insert it in the last blank row of the table.

-		Edit Object Pro	pertie	s	· 🗆
L	: MGC_IC_DEVICE_LIB	C: nmos_4pin		V: Symbol	Replace
ļ	Name _v	Value	Туре	Visibility	
	PHY_COMP PS	\$mos	string string	hidden 🔺	
	RDC		string	visible	Remove
	TEMP		string	visible	
	W subtype	5u n	string string	visible hidden	
	Common 🗔 Evaluate Ex	pressions 🖂 Translate P	arameter	Names T Hide Non	-SPE Props.
N	lame: W	Value:	5u		
1	Type: string	🔹 Visible: 🔽 Us	age:		
ŀ	Help:				
_	OK Cancel	Apply Reset	Nex	t	Options

7. Checking & Saving the Schematic



Now the schematic should look as shown in the figure.

Click **Check & Save** from the **schematic** > **edit** palette or **in** the **menu bar** to check and save your sheet.

Note: If your sheet does not pass check, you cannot simulate the design. Check the log if errors are listed.

If your sheet passes all checks you will get the following message in the message pane:

Note: "CommonSource/schematic/sheet1" passed check Note: Registering schematic model with part \$analogtut/default.group/logic.views/CommonSource/part : Common Source Note: Version 1 of part \$analogtut/default.group/logic.views/CommonSource/part has been written Note: Version 1 of sheet \$analogtut/default.group/logic.views/CommonSource/schematic/sheet1 has been written

8. Creating a Viewpoint

To prepare for simulation in DA_IC, a Design Viewpoint needs to be created. The tool automatically creates the viewpoint when you enter into **simulation mode**.

- Click on simulation from the schematic edit palette on the RHS Or Click on on the LHS Vertical Icon Bar to enter the simulation mode.
- Click **OK** to accept default options and to create a viewpoint. You will use the viewpoint during layout of the circuit.
- Click on end simulation in the Schematic_sim palette or \checkmark on the LHS Vertical Icon Bar to end the simulation and get back in schematic edit mode.



9. Printing the Schematic

To print the schematic:

- Click File > Print
- Enter laser in the Printer name box. Do not modify any other settings.
- Click **OK**.

F		Design Architect-IC v2006.1_2.3 (p081906) - Project	analog_tutorial	• 🗆
File	Add <u>E</u>	dit Select View Tools Setup Report Window Help		
	· 🔁 🗹			
Se	4: 0	(W dae) (CommonSource schematic sheet1) (portin portin) ()	(-1.2633, 2.6938)	Hotkeys: On
` ≑	—	Print Sheet with HotPlot	Q 1 r	schematic edit
•	•	Destination Send to Printer Export to File		Session
×		Printer Name Taser		Library
2				Simulation
9		Working Directory /users/student/rwhite/analogtut/analog_tutorial.proj		Eart
1₽		Page Width (inches) 8.5 Page Length (inches) 11 Select Page Size		Text
译				Check & Save
l ∎ r°		Continuous Paper Roll Panel Name		Select
•+•		Output Format Color]. Ţ	By Property
°+ dbe		Gasic PostScript		Unselect All
		Raster PostScript OMonochrome		Edit
+ 6ª		Basic HP-RTL Date Basic Hp-RTL		Move
500 H		Dots Per Inch 300		. Пору
1		Rotation # Copies (1-99) 1		Delete
ET	1	Omit Page Numbers		Undo
		C Landscape Tot Social V 100		Filp Patata
1	•	Text Scale % Too Text Options		Properties
	•	Magnification (0=fill page) 0 Format Options		Add
	•			Instance
	•	OK Reset Cancel Help		Wire
				Bus/Bundle
				Add Source
	•			Property
	·			Name
	•			Instance
No	te : Release	Select button to complete MOVE		
No No	ite : Release ite : Release	Select button to complete MUVE Select button to complete MUVE		
Ľ,		//////		· · ·

0000

10. Creating a Symbol

Hierarchical design allows you to instantiate lower level cells (circuits) into upper level cells to create a tree structure. Since, at higher levels, we really don't need to see the detailed transistor-level description of the base cells, we create symbols for them. Also we will use this symbol to perform various simulations on the circuit in the simulation tutorial.

Make sure that the schematic is checked & saved before making the symbol.

To generate a Symbol Automatically:

- From the menu bar select **Tools > Generate Symbol**
- In the **Generate symbol** dialog box,
 - Click Choose shape.
- Notice that variety of shapes are available for the symbol and one can choose any of these shape that best describes the circuit.
 - Select **Buffer** and click **OK**.
 - Change **Min Height** to 3 so the symbol is large enough for the labels we have used.
 - Click the **OK** button to generate a symbol for the cell.
- The symbol is created automatically and displayed in a new window as shown below. Do not forget to save this symbol by selecting **Check & Save** from the **symbol_draw** palette on the right hand side. You can now use this symbol in other schematics. You can edit symbol by selecting different shapes from right hand palette.

The symbol generated for the amplifier is shown in the figure below.

