Procedures to install the Design Architect IC Tool

- Extract eldo.tar.gz and daic.tar.gz to any location. They occupy 1.4 GB and 1.1 GB respectively when extracted. It should correspondingly create folders with names "AMS_2005.3" and "icflow_home".
- > Copy the files 'mgc_location_map' and 'startdaic' to your home directory.
- > Open the file 'mgc_location_map' in a text editor. Insert the full path(no symbols or aliases) of the folder "icflow_home". There are 9 variables and you should set path for all of them. The commented lines should help you.

For example:

If 'icflow_home' directory's path was /mnt/home/xxxx/ then insert "/mnt/home/xxxx/" before what is already

The file before editing :

MGC_LOCATION_MAP_2

4

.

.

MGC IC sources_lib \$MGC_IC_SOURCES_LIB icflow_home/pkgs/utilities_ic.ixe/mgc_icstd_lib/sources_lib

MGC IC generic_lib \$MGC_IC_GENERIC_LIB icflow_home/pkgs/utilities_ic.ixe/mgc_icstd_lib/generic_lib

The file after editing :

MGC_LOCATION_MAP_2

MGC IC sources_lib \$MGC_IC_SOURCES_LIB /mnt/home/xxxx/icflow_home/pkgs/utilities_ic.ixe/mgc_icstd_lib/sources_lib

MGC IC generic_lib \$MGC_IC_GENERIC_LIB /mnt/home/xxxx/icflow_home/pkgs/utilities_ic.ixe/mgc_icstd_lib/generic_lib

> Open the file "startdaic" in a text editor. You have to set two environmental variables here. There is a commented instruction which variable to set.

Third line : Insert the path where the directory "icflow_home" is (extracting daic.tar.gz creates this folder.) After adding the path the third line looks like this

setenv MGC_HOME /mnt/home/xxxx/icflow_home

Thirteenth line : Insert the path where the "AMS_2005.3" folder is (extracting eldo.tar.gz creates this folder) After adding the path the thirteenth line look likes this

setenv anacad /mnt/home/xxxx/AMS_2005.3

Change your shell to TC Shell with the command 'tcsh' and invoke the tool with the command ./startdaic

Important: Changes to be made in the library file before simulation

Download the tsmc018.lib from the link <u>http://www.ee.iitm.ac.in/~nagendra/cadinfo.html.</u>Open it in a text editor, make the following change.

• Change the value of level to 53 in lines Nine and Fifty One.

If you use any other library like UMC library, it should be in .lib format. And you ensure that the level is 53.

Working around with DAIC

- Create a folder (say 'Mydesign') where you will save each project. Create sub folders(say 'Inverter') inside this folder ('Mydesign') for each design. The tool Design Architect IC creates further sub folders and files inside this work folder ('Inverter').
- Lets see an example of how to simulate an CMOS inverter. Create a folder named 'Inverter' inside the 'Mydesign' folder.
- Click File --> Open --> Schematic. Select browse and point to the folder 'Inverter'. Click Options and select 'New sheet'. Click 'OK' twice.
- A new blank sheet will be opened. The right corner button in the title bar of the sheet maximizes the window and lets you see the status bar. This will help you see the progress of your work and errors if any will be reported here.
- Select the 'Show/Hide Library palette' (penultimate button in the left hand side tool bar). A new panel will be opened in the right side. The 'devices library' has the

Transistors and other components like resistors, capacitors... The 'Generic Library' has the Vdd, ground and other symbols. The 'Sources Library' has various sources.

==											De	esign	Arch	itect-l	C v20	06.1_:	1.1
<u>M</u> GC	C <u>Fi</u> le	<u>E</u> dit	<u>A</u> dd	Selec	: <u>t S</u> et	tup	Miscellan		<u>R</u> eport	<u>Vi</u> ew	<u>H</u> elp						
🔁 l	ど 💾	G	i 🔒 🖲	29	<u>R</u>	₽₽	圁 📙	₩ R	*	Ē							
Sel:	0	(γ	V dae) (test	t sche	matic	sheet1) ()	()								
₩	-											Sc	hemat	tic#1 t	test sl	heet1	
	÷																
×																	
5																	
℃ ⊉+	•																
-+ ₩4	÷																
.Γ+°	÷																
F +															*		
dbe 14	Ŷ		Ŷ	•	*	C٢	owl	цiд	la Lih	ranu	Dal	otto	hut	ton	Č	Ů	Ť
₽	÷					51	1000/	пи	le Lib	rary	rai	elle	Dui	lon			
A ª+	÷				7	÷	٠	٠	٠	٠	٠	٠	٠	٠	X	٠	
5		_			*	*	*	*		*	*	*	*	•	*	*	•
₩ E																	
	*		*	*													
	Show/Hid	le Libr	ary Pal ◎	ette °													
	*																
	*		*	*	*	*	*	٠	٠	*	*	*	*	٠	*	*	*

• Select components and place it in the sheet. To rotate the component press the 'r' key. Select a component and press the key 'q' to get its properties window where you can change various attributes. Press 'w' to get the wire and connect the components.

 When using transistors, make sure you change the ASIM_MODEL parameter in the properties dialog box of each transistor to the model name mentioned in the library. By default ASIM_MODEL will be NCH/PCH. To know what to enter in the ASIM_MODEL field, open the library file in a text editor. The picture below shows the details for tsmc018.lib. Apply similar attributes to the PMOS transistor if necessary.

* T58F SPICE BSIM3 VEF * * SPICE 3f5 Level 8, St *			Level 8	Change it from 49 to 53	
* DATE: Oct 31/05					
* LOT: T58F	WAF	: 9005		Medal Name	
* Temperature_parameter	s=Default			Model Name =	
.MODEL CMOSN NMOS (+VERSION = 3.1 +XJ = 1E-7 +K1 = 0.5864999 +K3B = 0.0294061 +DVTOW = 0 +DVTO = 1.2064649 +UO = 273.8094484 +UC = 6.504826E-11 +AGS = 0.4449958 +KETA = -0.0164863 +RDSW = 123.3376355	TNOM NCH K2 WO DVT1W DVT1 UA VSAT BO A1 PRWG	= 27 = 2.3549E17 = 1.127266E-3 = 1E-7 = 0 = 0.4215486 = -1.40499E-9 = 1.355009E5 = 1.901075E-7 = 3.868769E-4 = 0.5	LEVEL TOX VTHO K3 NLX DVT2W DVT2 UB A0 B1 A2 PRWB	= 49 = 4.1E-9 = 0.3662473 = 1E-3 = 1.630684E-7 = 0 = 0.0197749 = 2.408323E-18 = 2 = 4.99995E-6 = 0.4640272 = -0.197728	

• After connecting various components click 'check and save schematic' button (Second button in the top tool bar). Errors if any will be shown in a separate window.

==	== Design Arc																								
<u>M</u> G	С	<u>Fi</u> le	в	<u>E</u> dit	E	<u>v</u> dd	S	elec <u>t</u>	. 1	<u>B</u> etu	qı	Mig	scell	aned	ous	<u>B</u>	epo	rt	⊻ie	w	<u>H</u> el	р			
≥	1		-	പ	æ	9 🔁	< ·	2	Ð	ĮĘ	<u>+</u>	►		4	R	~			ì						
Sel:	:	0 +		$\langle \cdot \rangle$	~ I ¢	dae)		test	scl	hem	natio	: sł	heet	1)	(d	c_v_	sou	irce	de	-v_	_sou	rce) ()	
▶.	-	-T	<														S	ch	ema	atio	:#1	te	st :	she	et
	٠	٠	*	~	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠	٠
**	*			*	~																				*
	°	* .	*	*	2		*	*	*	*	*	* .	* *	* *	*	*	* *	*	*	*	*	* *	*	• •	*
2	٠								•		•	•			•	•	~	•	•	~			*		*
+≯	٠						*	•	С	he	ec	k (& :	Sa	iv	e E	Зu	tte	on				100		٠
14	° ^							*								~				_	NS2 PCH		-İ		*
₀ Γ +°	÷																			-	PCH	761	Į		*
• +	٠								٠											Ļ	٠		1		٠
dbe	٠							• -												ł		•	Ţ		٠
\square	°.							਼੍ਰੈਆ	İ	-					1			-		Ť			Ť		*
Aª	。 。	• •	•	*	•	•	•	4	¥	Ň.	° •	•	• •	×2	,	_ -	• •	•	•	Ī	¢	*	Ī	• •	*
	٠							~	÷	/ m	-, s			-{	Ħ	\rightarrow	c *			ł	NG1 NCH	21	- 44 - 361		٠
經 12	*								ł					*	Ť	/_**	e= 17	*		Į_	-1	٠	Ì		*
	°	*	*	*	*	*	*	*	-	*	*	*	*	*	İ	*	*	*	*	*	*	L.	١.	*	÷
		*	*	*	*	*	*	*	*	*	*	*	•	*	Ţ		*	*	*	*	*	*	Ţ		*
	*														*								÷		٠
	٠																								٠
	*																								*

- Click the 'Simulation' button in the right side panel and click 'OK' in the dialog box that pops up. You enter simulation mode.
- On the right side panel, click the button session and then simulator/viewer. Select a folder where you want to store the files that are produced during simulation. Better choose the working folder 'Inverter.' Select Xelga as the waveform viewer.

<u>R</u> eport	<u>Vi</u> ew	<u>C</u> ommlib	<u>P</u> arasitics	<u>H</u> elp												
	Ì															
/ee07s	016/DAI	C/test/eldon	et)													
	/:st	neet1 : <i>I</i> hi	ome/msnl	nd/ee07s016/D	AlC/test/eldonet			schematic sim	-							
	110		omornopi		VDD			End Sim								
							-	Setup								
				Sei	tup Simulator/Viewer	×		Session								
	Simula	tor Invoke	d From Di					Lib/Temp/Inc								
l r					+			Analyses								
	/nom	e/mspha/e	eu/suib/i	mydesign/inver	ter Brows	e		Parameters								
-	Simula	tor:						Forces/ICs								
	• Eld				Enable marching waveforms with EZ Wave?			Safe Operating A								
	🔿 Eld	lo Interact	ive	Viewer: ● Xelga ○ EZwave		(Only valid with Eldo and Eldo interactive)			Options							
		MS						Multiple Runs								
		MS Intera .ch-TA	ctive		-	-	-	-				Wave Outputs				
H		ich-TA Int	eractive						 EZwave 	 EZwave 	 EZwave 	tive C EZwave	active CZwave			
		pice					Advanced Setup			Model Selector						
D		artspice						Execute								
Με	⊖ Otł	ner						Netlist								
				ОК	Reset Cancel Help			Run ELDO								
								Results								
					M= 1			ASCII Files								
								View Waves								
								Cross Probe								
								View Measure:								
								DCOP/TRAN								
						-		Utilities	-							
						•	•		-							

- In the right side panel, click the button 'session' and then 'netlister' in the pop menu. Enter the name of the nets where it should be ground. By default only the nets where ground label is attached becomes ground.
- In the right side panel, click the button session and then 'environment'. You can ask the simulator to display the waveforms directly after running simulation. In this case the information on nets which are saved will be plotted. The next point describes how to save various nets for waveforms.

msphd/ee07s016/DAIC/test/eldonet)										
Setup Simulation Environment	schematic sim									
Simulation Depository:	End Sim									
/home/msphd/ee07s016/mydesign/inverter/eldonet Browse	Setup									
	Session									
Command File: test_eldonet.cir Netlist File: test_eldonet.spi	Lib/Temp/Inc									
Output File: test_eldonet.chi	Analyses									
	Parameters									
Auto-Run Simulation Setup:	Forces/ICs									
Run Simulation Only Gun Simulation and Display Waveforms Automatic check design before netlist	Safe Operating A									
C Run Simulation and Display Waveforms Automatic check design before netlist O Netlist and Run Simulation	Options									
Netlist and Nun simulation Netlist, Run Simulation, and Display Waveforms Remove invalid waveform entries before simulation	Multiple Runs									
Annotated Property Text DCOP/Transient Display	Wave Outputs									
	Measurements									
Font: stroke Menu Font: stroke Menu	Model Selector									
Half height font	Execute									
Height: 0.125 Decimal places to display: 2	Netlist									
	Run ELDO									
Info Block Display Options?	Results									
Yes No	ASCII Files									
	View Waves									
	Cross Probe									
	View Measure:									
	DCOP/TRAN									
	Utilities									
OK Reset Cancel Help										

Before you save nets you have to choose the analysis. Here we want a DC analysis
with the input voltage being swept from 0 to Vdd. Click 'Analysis' button in the right side
panel and in the pop up box select DC simulation. Click 'Setup' button and select the
source for sweep and enter the sweep range.

• Select a Net where you want to results to be plotted. In the right panel click 'Wave outputs' button. Click Save selected. In the pop up menu select voltage. Click the advanced menu. Choose the Analysis type (here DC) and the result type (here magnitude) To plot the current, select a node and repeat the above steps.

		nced Options	MS2 PCH W=1.4u L=0.35u
V1	Analysis type: DC AC TRAN FOUR TSST FSST SSTAC FMODSST TMODSST FOURMODSST	<i>Display FFT Results:</i> Real Imaginary Mag Mag-DB Phase	Save Selected (Nets) × Save: • • Voltage • • Digital Signal Plot item(s)? OK Reset Cancel Help
	Harmonics: (Separated by space OK Reset	s) Cancel Help	

• To Add library files corresponding to the FET, Click 'Lib/Temp/Include' button in the right panel and click 'Add Path' Point to the library file. The library file should have the extension as '.lib' Here we add the tsmc018.lib file. There are some important changes to be made in the library file before adding them. Please see the note above this documentation.

	/ : sheet1	: /home/msphd/ee07s016/DAIC/test/eldonet		schemaut sim						
		VDD	^	End Sim						
				Setup						
				Session						
		MS2		Lib/Temp/Inc						
		Set Include Paths	×	Analyses						
	Entries 1 - 6	Entries 7 - 12		Parameters						
	Include Path	Include Path /home/msphd/ee07s016/Download Basket/lib/tsmc018.lib Browse								
				Safe Operating A						
DD	Include Path	Broy	wse	Options						
				Multiple Runs						
	Include Path	Broy	wse	Wave Outputs						
				Measurements						
	Include Path	Bro	wse	Model Selector						
/				Execute						
· _	Include Path	Bro	wse	Netlist						
				Run ELDO						
	Include Path	Bro	wse	Results						
				ASCII Files						
				View Waves						
		OK Reset Cancel Help		Cross Probe						
				View Measure:						
				DCOP/TRAN						
			_	Utilities						

• Click 'Run Eldo' button and wait for the results to be plotted. To make changes and simulate, Click 'End Simulation' button and repeat the steps.

