



Microelectronic Systems

Lab 6

Physical Design: standard-cells based layout

Generate a new directory cap6. Copy all the files from `/home/repository/ms/cap6/`
To set correctly the environment variables type:

```
prompt> setinnovus
```

6.1 Physical design of the Ripple Carry Adder

Your first exercise will be to work on a simple SUM structure organized as in figure 6.1 based on a registered 128 bit RCA (the same you used in a previous lab). You have the vhdl files, and, as Innovus starts from a verilog netlist, you have the verilog file as well **sum.v**.

You also have in this directory the synthesis constraint script file (sum-t.scr, read it and notice the command used and the period) and the resulting timing and power performance obtained AFTER the synthesis: read them and notice the requested period and frequency (sum_timeopt_t.rpt and sum_timeopt_pw.rpt). We will compare these values to those obtained after the layout phase.

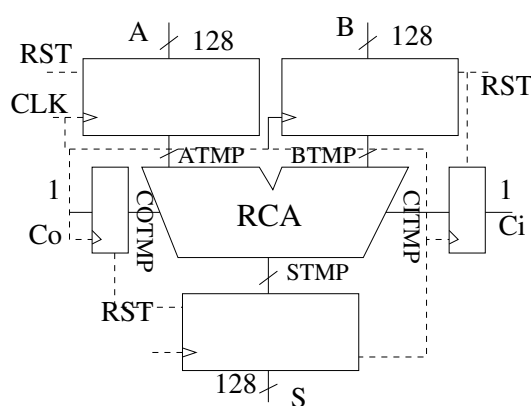


Figure 6.1:

We are now ready to start. In the shell in which you set the environment variables type:

prompt> **innovus**

Pay attention: this shell is now a dialog shell in which it is better if you do not write anything: just read the results each time you give a command in the GUI.

You will be helped step by step in the following. Anyway, you can also read Innovus manual by typing: **help_cds_innovus** in the command line. The online help (top right box) is also useful in case you want to know details on commands.

We are going to process the architecture physical design step by step:

Configuring \rightarrow Floorplanning \rightarrow Power planning and routing \rightarrow Cell placing \rightarrow Signal routing

Configuring The first step basically helps Innovus finding the path to the libraries used for the design and reading the synthesized version of the circuit.

From the top Innovus menu import the configuration file which sets the correct references to the libraries: **File** → **Import Design**; a new window opens: select in the menu below **Load** and select from the browse menu in the new windows the file **SUM.globals**. Notice the definition of the verilog file (.v) containing the post synthesis cell view, the reference to the layout view of the library of cells (file .lef) and the definition of vdd and gnd power supply names. Notice also the Default.view file included. This stores information on the MMMC (MultiMode MultiCorner) analysis, i.e. references to files and data definitions organized and combined in order to consider not only default conditions but best case or worst case conditions in terms of temperature and of process variations. Now click **OK** in the Design Import window: the design is imported with the correct cell reference and with the RTL verilog description.

A square with several lines appears. This image represents a rough structure where several cells will be placed in this sequence of lines.

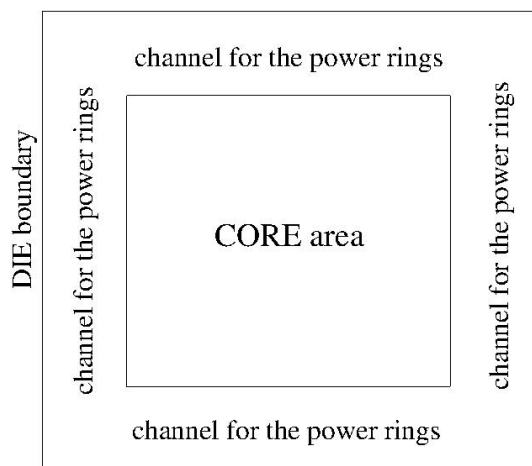


Figure 6.2: Area of the die dedicated to the core (the place for the cells) and to the power supply rings all around the core

Structuring the Floorplan At this step Innovus sets the area to be assigned to the cell ensemble and the area where the power supply ring will be routed as shown in figure 6.1.

From the Main window top menu select: **Floorplan** → **Specify Floorplan**. An option window opens, allowing you to define the **Core aspect ratio** (set to 1.0) and utilization (set to 0.6). In section *Core Margins by* select **Core to die boundary** and force 5 (μm) from the four sides of the core (unity is μm here). Click **OK** and see what happens. The cells height is already known at this point, as you can note by the grey horizontal lines. All the standard cells have the same height by construction. The width changes coherently with the transistors and interconnections area. At this point Innovus knows how many rows will be needed for the design.

Inserting power Rings The channel defined before will be filled with two metal rings for power and ground respectively. These metal stripes will be connected with the VDD and GND pads (if this is the final chip a wirebonding package is supposed here, if not, more probable, these rings will be connected to other rings of other blocks) and will distribute hierarchically power and ground signals to the whole chip. For this reason we will use for the rings an high metal layer. For avoiding congestion we will choose a different layer for the horizontal lines (M9) and for the vertical lines (M10).

From the Main window top menu select: **Power** → **Power Planning** → **Add Rings**. An option window opens, allowing you to define:

- the nets to be associated to power and ground: click on the browse button on the right end of the Net box, select and add both *vdd* and *gnd*
- the type of ring: in the *Ring Type* section leave the default configuration as “Around core boundary”
- the metal associated to the rings: in the *Ring Configuration* section select M9 for top and bottom lines and M10 for left and right lines; set all widths and spacings to 0.8 and select the option *Offset: Center in channel*.

Click **OK**. Now two rings have been designed: one for VDD and one for GND (the connection to the electrical generator will be done later). Note that in the corners there are vias for the connection between M9 and M10. If you click on one of the stripes you will receive information in the bottom window on the metal layer, on the stripe geometry and on its coordinates. What does this rectangle represent for the foundry that is going to produce your chip?

Inserting stripes. In this step vertical metal wires will be added connecting the ring from top to bottom, and will be still VDD and GND lines. This step could be avoided, but it helps in correctly distributing the power and ground signals to the chip center, so it is generally performed. The higher is the vertical stripes number, the better will be the distribution, but the more problematic will be congestion when the other signals will be routed.

From the Main window top menu select: **Power** → **Power Planning** → **Add Stripes**. A new window opens:

- in the *Set Configuration* section click on the browse button of the Net box and add both *vdd* and *gnd*; select M10 for routing leaving the vertical option for the direction and set 0.8 for both width and spacing

- in the *Set Pattern* section select **Set-to-set-distances** and choose a value of 20
- in the *First/Last Stripe* section select the **Relative from core or selected area** option and set the *Start* to 15, while leave *Stop* unconstrained.

Click **OK**. As you can see three sets of stripes have been routed. Each set is composed of one stripe connected to GND (click on it to see the properties) and one stripe used as a VDD line. Use the ruler (left menu) to check distances.

Standard Cell Power routing. This operation allows to place horizontal wires preparing the VDD and GND wires for the standard cells. Such wires will be connected to the ring and to the vertical stripes as well.

From the Main window top menu select: **Route** → **Special Route**, select *vdd* and *gnd* as nets and click **OK** with the default values.

Once the operation is concluded you can reckon the metal layer and connectivity by clicking the arrow in the left menu, sweeping on the metal stripes and reading in the bottom left window the metal lines attributes.

Placement. Now the cells will be placed. Up to this point the only thing known was the total area of the circuit and the number of rows to be used. After this point the layout of each cell has a unique position in one of the predefined rows.

Before starting the placement phase select **Place** → **Specify** → **Placement Blockage** and select layers from M1 to M8, such that the cells are placed out from the space occupied by the power and ground stripes, just to avoid congestion problems. Now click on **Place** → **Place Standard Cell** and then click **OK**. What happens?

You can select the cells to read (bottom left window) the name, and zoom in the view for recognizing the input/output pin names. If you click on a pin you will see its connectivity (virtual) to the other pins. In the die border you can see input and output pins (to be connected to the pads) and if you click on one of them you can see its connection to the cell pins.

What you see is only an abstract view of each cell, in which only its sizing, its orientation and the position and connectivity of its pins which will be used during the routing phase for connecting the cells among them.

Exploring placed design In the second line menu click on **Design Browser**. A new window opens useful to browse your circuit. Click, for example, on **Modules** → **maprca**: all the gates included in the RCA block are highlighted. It is also useful to select the “amoeba view” instead of the physical view. Now play with other sub-blocks...

Placing I/O Pins From the main menu click on **Edit** → **Pin Editor**. A new window opens allowing you to define where to place I/O pins along the edges of the DIE. In the *Pin Group* section you can notice all I/O signals defined in your top level module. Select the first one (A) and in the *Location* section select **Spread** and for *Spread Type* select: **Along Side** if the signal is a **bus**, **From center** if the signal is a **single wire**. Then, in the *Pin Attribute* section, force the *Side/Edge* option to **Top**, for example. Repeat the same steps for the other signals but changing the *Side/Edge* option. For example, you can place B on the left edge, S on the right and the other signals on the bottom. Now click **OK**. You will see all pins distributed along the edges of the DIE.

Post Clock-Tree-Synthesis (CTS) optimization Before running the routing we can try to optimize our design to achieve the required timing constraints. In the Main window select **ECO** → **Optimize Design**. In the *Design Stage* section choose **Post-CTS** and in *Optimization Type* check both **Setup** and **Hold**. Then, click **OK**.

Place filler. It is of help for technological reasons to complete the placement with filler cells. These will fill the holes to ensure continuity in N+ and P+ wells in each rows. From the top menu select **Place** → **Physical Cell** → **Add Filler**. A new window opens. Click **Select** and choose all the filler cells available and add them to the left list. These are possible fill cells among the placer will choose for filling the placement gaps. Now click **OK** and inspect what happens.

Routing. This phase is the last one: the connection among the cells will be performed using the available metal layers (the routing already visible is only a logical connection among cell pins, that have been used during the placement for taking into account the interconnection length).

Click on **Route** → **NanoRoute** → **Route** and **OK**. Note in the Innovus shell a few messages on the routing iteration steps for this optimization, together with some interesting data on the routing performed: the number of wires used for each layer, the total length routed in each layer, the number of via used. You still can analyze the metal layer used for the connections. The white boxes are violations which should be solved by a detailed operation that we will not perform.

Post routing optimization Note that at this point the design is complete (even if we do not have used real PADS all around the design). During this last step we can try to optimize our design to achieve the required timing constraints. Before starting the last optimization, you have to issue on Innovus command line (the shell where you launched Innovus from) the command `setAnalysisMode -analysisType onChipVariation`. Then, in the Main window select **ECO** → **Optimize Design**. In the **Design Stage** section choose **Post-Route** and in *Optimization type* check both **Setup** and **Hold**. Then, click **OK**.

Before going on, save your routed view: **File** → **Save Design**. Check that selected *Data Type* is **Innovus** and name the file with a significant name. Hereinafter you will be able to import your design at the routed level. You can plot the image from the main menu: **Tools** → **Screen Capture** → **Screen Dump** and you can see the result using **Tools** → **Screen Capture** → **Display Screen Dump** (select **All** as File Type).

We can now perform a few analyses for timing and integrity.

Parasitics For analyzing the time behavior Innovus uses the resistance and capacitance parasitic values for each metal wire. The extraction of such parasitics is the task of this step. The engine is able to compute the resistance and capacitance associated to each rectangle using its properties (technology and geometry information).

From the main menu select: **Timing** → **MMMC Browser**. Inspect the three possible RC Corners defined. We will use the standard.

From the main menu select: **Timing** → **ExtractRC** and select all the possible files to save. Moreover, select **standard** as *RC Corner* and click **OK**. Once done, look at the file SUM.spf (spf means standard parasitics format): what does it represent? Note the *.subckt* instruction at the top file and the two “net section” (file top) and “instance section” (file bottom). And what about the SUM.spf file? Analyze the information contained in it and search the net with the greater capacitance and the greater length.

Read the file “SUM.setload” and “SUM.setres” which will be used later: what do they force?

Delay From the main menu select: **Timing** → **Report Timing**. A new window opens: in *Design Stage* choose **Post-Route** and in *Analysis Type* choose **Setup**. Click **OK**. Repeat the same step choosing **Hold** in the *Analysis Type* section. In the directory named *timingReports* you find all the results produced by the timing analysis. Files *.slk* and *.tarpt* contain general and detailed information on the timing paths and violations. Violations are referred to the timing constraints defined in the file **SUM.sdc**, that are identical to those used during the synthesis, are already loaded with the initial configuration file. Remember that slack means “what is remaining between what you asked to have and what you actually have”. So, if the slack is positive you are ok (you spared some time), if it is negative you are violating the constraint you decided. Remember that if you have not a real clock in the design you have to perform this step and specify a virtual clock.

Now complete the timing analysis by selecting **Timing** → **Debug timing**. A new window appears. There you have an overview of the critical paths that violated the constraint (i.e. the value respected at the synthesis level). If you double-click on one (the first for example) a new window appears that allows you to press element by element (in the horizontal bar) and see the corresponding delay and the path in evidence in the layout window. Play with these two windows in order to understand the potentials. It is also interesting to compare the delay found at synthesis level with the delay at the physical design level... play and LEARN!!

Design analysis and verification Before ending the place and route phase we have to verify the connectivity and the design rules: **Verify** → **Verify Connectivity** and **OK**. Check the message produced by Innovus and verify there are no violations. Usually violations are caused by floating wires. To verify the design rules: **Verify** → **Verify Geometry** and **OK**. Check the message shown by Innovus and verify there are no violations. Usually violations are caused by wrong constraints on the geometric feature during the place and route design flow. As an example, if we require a narrow spacing between VDD and VSS layers when creating the power supply rings we may violate the design rules imposed by the technology we are employing. We can save area and gate count data as: **File** → **Report** → **Gate Count** and **OK**. Finally, we save i) the post place and route verilog netlist as **File** → **Save** → **Netlist** and **OK**; ii) the file with delay annotation (.sdf) as **Timing** → **Write SDF** and **OK**.

Further analyses YOU ARE LUCKY! (or maybe not...) due to problems with the new tool version and the new technology the electromigration and IR drop analysis will be skipped. You can, if you want, play looking at the manual at the section: Rail Analysis.

Summary of what is requested

A dump of the layout, a saved placed and routed file with the Innovus extension in the working directory, the extracted capacitance, resistance and spf file, the Delay report.

6.2 Physical design of your ADDER

Now you can play with the ADDER you designed. The exercise is to perform the previous steps for the ADDER you designed in the previous lab and analyze how the critical path changes. To correctly execute the physical design you have to perform again the synthesis with the correct wire model and the correct constraints. You can start from the **sum-t.scr** script provided and modify it accordingly to your design. Don't forget also to modify the constraint for the CLK signal. For the physical design you must create a new file **ADDER.globals** starting from the SUM.globals of the previous step: copy the SUM one, change the name to ADDER.globals and change the verilog file name. If the SDC file produced by the synthesis have a name different from SUM.sdc you should also update Default.view accordingly. The references to all the libraries remains the same.

Summary of what is requested

A dump of the layout, the saved Innovus design, the extracted capacitance, resistance and spf file, the Delay report

6.3 Physical design of your MULTIPLIER

Now you can play with the MULTIPLIER you designed. The exercise is to perform the physical design also for the multiplier you designed in the previous lab and analyze how the critical path changes. Of course this will be useful for the DLX project.

The steps are the same followed for the ADDER.

Summary of what is requested

A dump of the layout, the saved Innovus design, the extracted capacitance, resistance and spf file, the Delay report.