German-Russian Institute of Advanced Technologies

TU-Ilmenau (Germany) and KNTRU-KAI (Kazan, Russia)

Guidelines for laboratory work №4 of the subject

«**Computer systems**»

« Proteus Virtual System Modeling (VSM) »

Kazan 2014

Proteus Virtual System Modeling (VSM)

Proteus Virtual System Modeling (VSM) software offers the ability to co-simulate both high and low-level micro-controller code in the context of a mixed-mode SPICE circuit simulation. It combines mixed mode SPICE circuit simulation, animated components and microprocessor models to facilitate co-simulation of complete micro-controller based designs.

With VSM, it is possible to develop and test such designs before a physical prototype is constructed. The designer can interact with the design using on screen indicators such as LED and LCD displays and actuators such as switches and buttons. The simulation takes place in real time, e.g., a 1GMHz Pentium III can simulate a basic 8051 system clocking at over 12MHz. Proteus VSM also provides extensive debugging facilities including breakpoints, single stepping and variable display for both assembly code and high level language source.

Overview of the ISIS Editor

To start the ISIS program, click on the Start button and select Programs, Proteus 7 Professional and then the ISIS 7 Professional option (See Figure 4.1). The ISIS schematic editor will then load and run.

The ISIS editor consists of three main areas as shown in Figure 4.2 :

1. **The Editing Window** : acts as a window on the drawing - this is where you will place and wire-up components.

2. **Object Selector** : lists objects inserted into the Editing Window and allows you to select new objects to be inserted from the ISIS library.

3. **Overview Window** : In normal use, the Overview Window displays an overview of the entire drawing - the blue box shows the edge of the current sheet and the green box the area of the sheet currently displayed in the Editing Window.

However, when a new object is selected from the Object Selector the Overview Window is used to preview the selected object

Editing Modes

The ISIS provide several editing tools (or modes) to facilitate schematic editing. These tools can be selected from the left bar menu. Figure 4.3 list the editing tools and their corresponding icons for your reference during this experiment.

Zooming

There are several ways to zoom in and out of areas of the schematic:

– Point the mouse where you want to zoom in and out of and roll the middle mouse button (roll forwards to zoom in and backwards to zoom out).

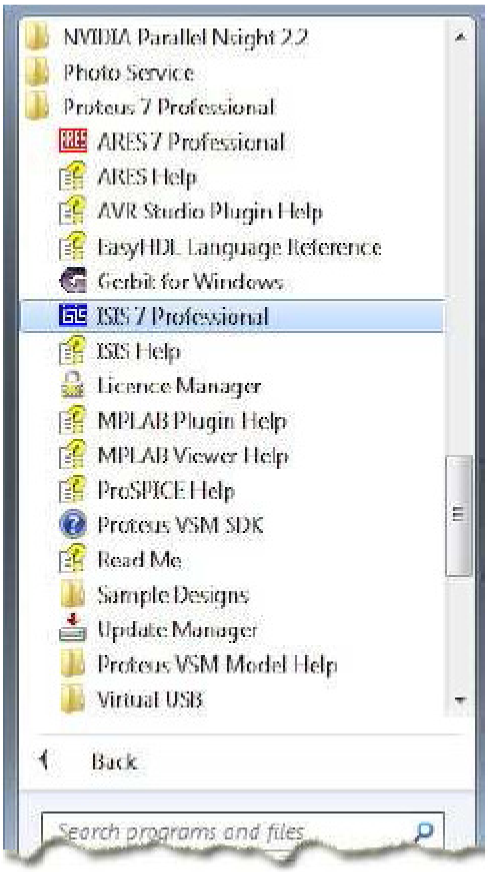


Figure 4.1 – Starting the ISIS editor.

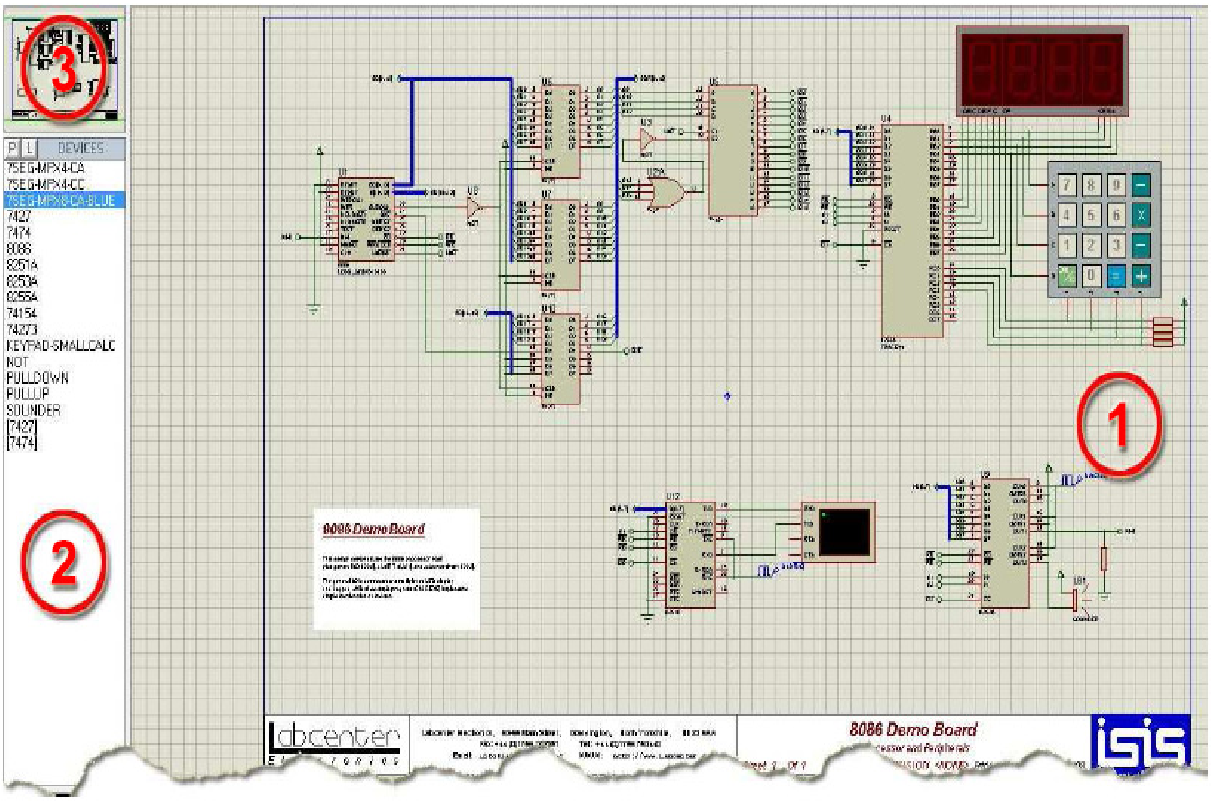
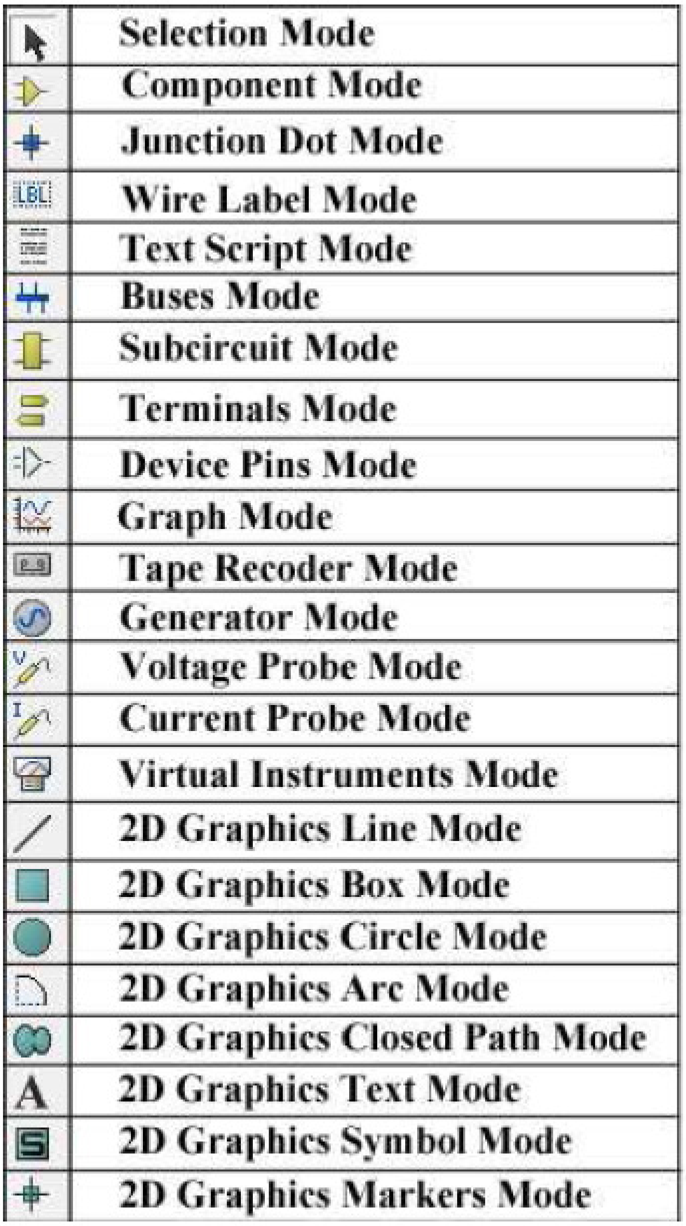


Figure 4.2 – ISIS schematic capture window. (1) Editing Window, (2) Object Selector, and (3) Overview Window.



	Selection Mode
	Component Mode
	Junction Dot Mode
	Wire Label Mode
	Text Script Mode
	Buses Mode
	Subcircuit Mode
	Terminals Mode
	Device Pins Mode
	Graph Mode
	Tape Recoder Mode
	Generator Mode
	Voltage Probe Mode
	Current Probe Mode
	Virtual Instruments Mode
	2D Graphics Line Mode
	2D Graphics Box Mode
	2D Graphics Circle Mode
	2D Graphics Arc Mode
	2D Graphics Closed Path Mode
	2D Graphics Text Mode
	2D Graphics Symbol Mode
	2D Graphics Markers Mode

Figure 4.3 – Editing modes.

Point the mouse where you want to zoom in or out of and and press the **F6** or **F7** keys respectively.

– Hold the **SHIFT** key down and drag out a box with the left mouse button around the area you want to zoom in to. We call this Shift Zoom

– Use the *Zoom In*, *Zoom Out*, *Zoom All* or *Zoom Area* icons on the toolbar (See Figure 4.4).

Figure 4.4 – Zoom icons.

Visual Aids of the Design

ISIS provides two main ways to help you see what is happening during the design process. Objects are encircled with a dashed line or twitched when the mouse is over them and mouse cursors will change according to function. Essentially, the object-twitching scheme tells you which object the mouse is over (the hot object) and the mouse cursor tells you what will happen when you left click the mouse on that object. A summary of cursors used, together with their actions, is provided in Figure 4.5.

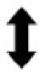
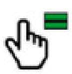
Cursor	Description
	Standard Cursor - Used in selection mode when not over a 'hot' object.
	Placement Cursor - Placement of an object will commence on a left click of the mouse.
	Placement Cursor - Placement of an object will commence on a left click of the mouse.
	Placement Cursor - Placement of an object will commence on a left click of the mouse.
	Placement Cursor - Placement of an object will commence on a left click of the mouse.
	Placement Cursor - Placement of an object will commence on a left click of the mouse.
	Placement Cursor - Placement of an object will commence on a left click of the mouse.
	Placement Cursor - Placement of an object will commence on a left click of the mouse.

Figure 4.5 – Cursor description.

Design Example

In this experiment we will design the binary counter which counts (0, 1, 2, 3, 4, 0, 1, 2, . . . , *etc*).

This counter can be implemented using a 4-bit binary counter with asynchronous reset input. The count is achieved by gating the outputs of the 4-bit binary counter ($Q3Q2Q1Q0$) through an "AND" gate and connecting it to the asynchronous reset input MR such that the counter is cleared when the count reaches 5 ($Q3Q2Q1Q0$ = 0101). Note that, the binary value 0101 can be distinguished from lower binary counts (0000, 0001, 0010, 0011, 0100) by the values of $Q2$ and $Q0$ together. Thus, as shown in Figure 4.6, the asynchronous reset input of the counter is connected to $Q2Q0$.

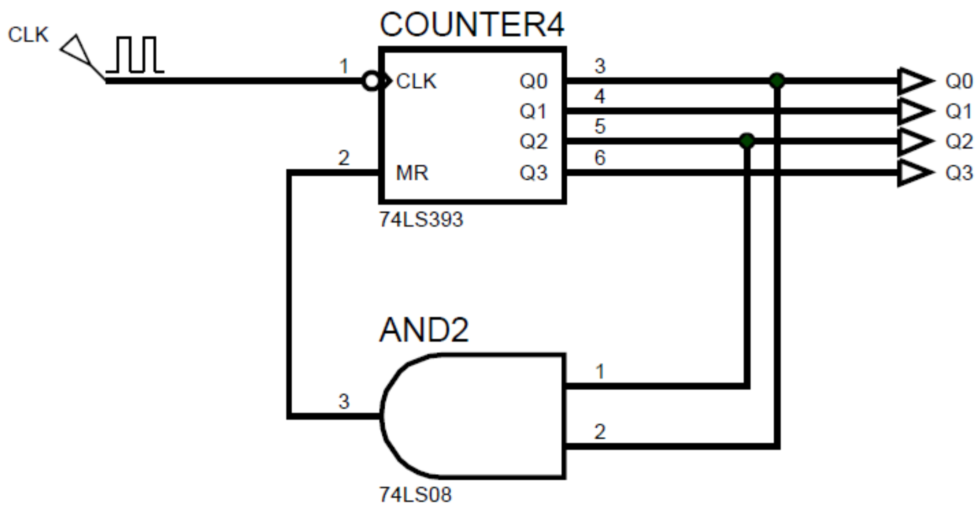


Figure 4.6 – The binary counter.

Schematic Entry of the Binary Counter

This section will guide through the design process of the binary counter shown in Figure 4.7.

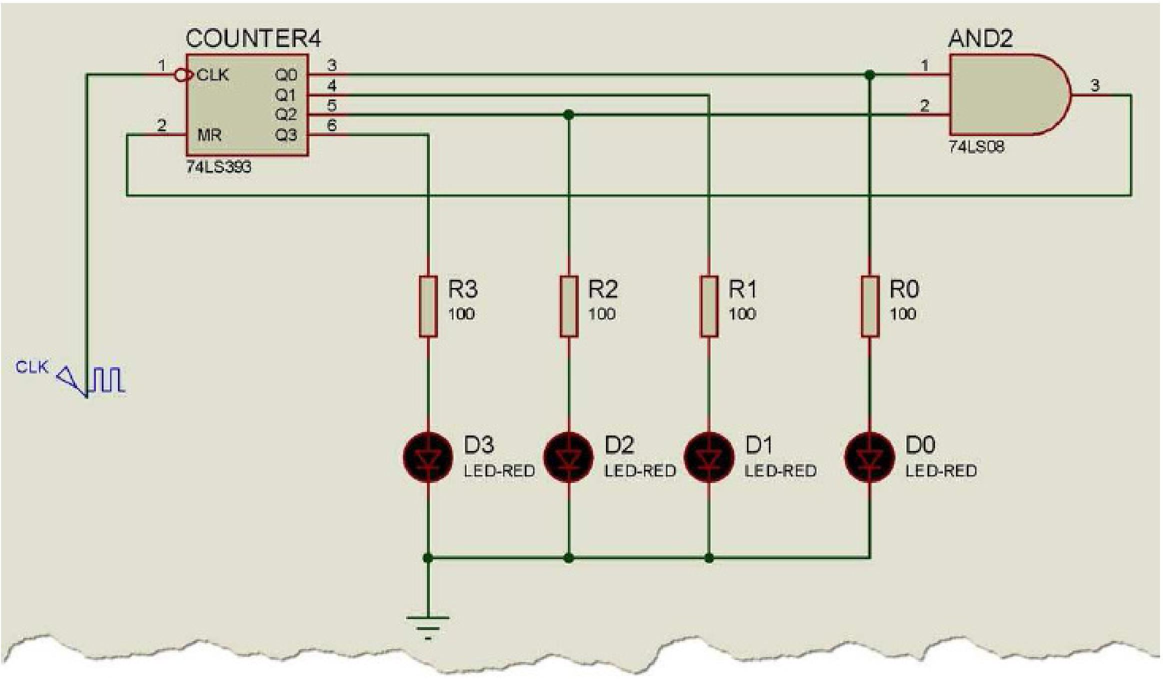


Figure 4.7 – Schematic of the binary counter.

The first thing we need to do is to get the components from the libraries that we need in our schematic.

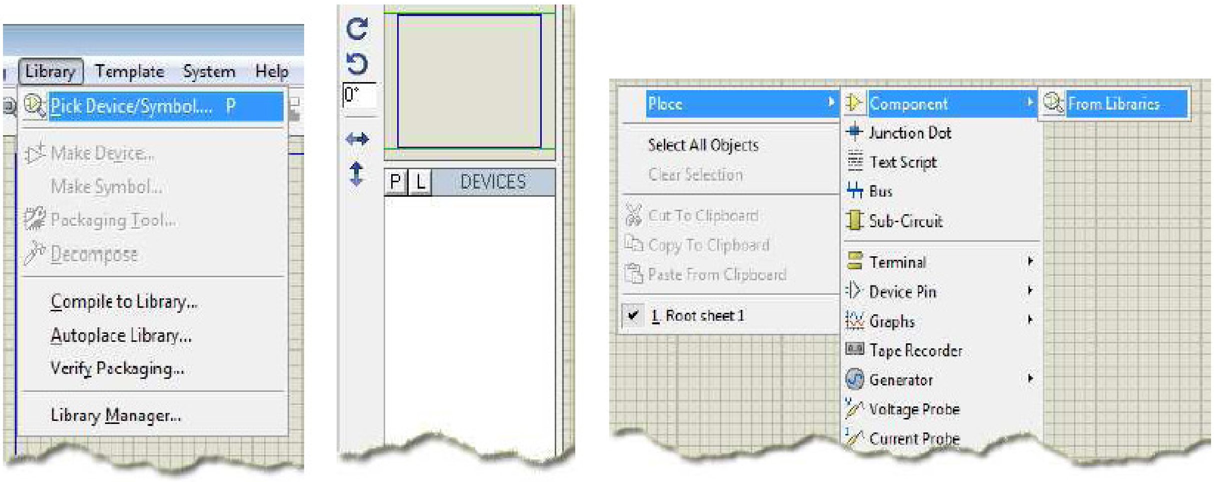
Selecting Components from the Library

You can select components from the library in one of three ways :

– Click on the *Library* drop-down menu in the ISIS menu bar, and then click on *Pick Device/Symbol* as shown in Figure 4.8(a).

– Click on the *P* button at the top left of the Object Selector as shown in Figure 4.8(b).

– Right click the mouse on an empty area of the schematic and select *Place →Component → From Libraries* from the resulting context menu as shown in Figure 4.8(c).



(a) (b) (c)

Figure 4.8 – Selecting Components. (a) library menu, (b) parts button, (c) place menu.

Either one of these three methods will cause the *Device Library Browser* dialogue form to appear (See Figure 4.9). For reference, the following is a list of all the components we will need for our design :

74LS393

74LS08

MINRES100R

LED-RED

There are several ways in which we can find and import components from the libraries into the schematic:

– You can enter a device name "74LS393" directly into into the *Keywords* field on the *Device Library Browser* dialogue form.

– You can search for a device type such as "resistor" in the library browser.

– You can search for a device by selecting a device category such as "TTL 74LS series" from the *Category List Box*.

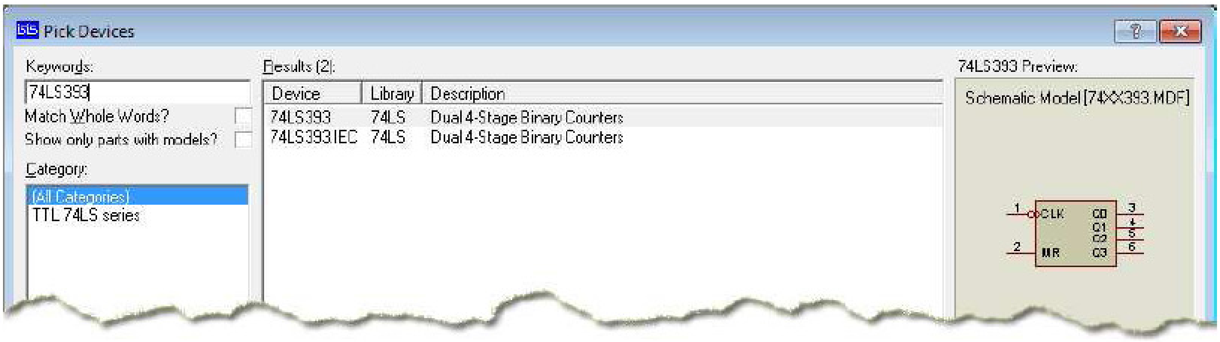


Figure 1.9 – Device library browser.

Placing Objects on the Schematic

Having selected the devices we need the next thing is to actually place them on the drawing area - the Editing Window - and wire them together.

To place the selected device on the drawing area, you can do the following :

– Click the "OK" button in the library browser to get back to the drawing area.

– Left click on the schematic to enter placement mode.

– Move the mouse to the desired location for the device.

– Left click the mouse again to "drop" the device and commit placement.

Often we need to move devices or blocks of circuitry after placement. The procedure for this should be familiar to most users ; we need to select the object(s) we want to move, left depress the mouse, drag to the new location and finally release the mouse to drop.

We can select an object in ISIS in several ways as detailed below :

– Choose the *Selection Mode* (See Figure 1.3 and then left click on the object.

– Right clicking the mouse on an object will both tag the object and present a context

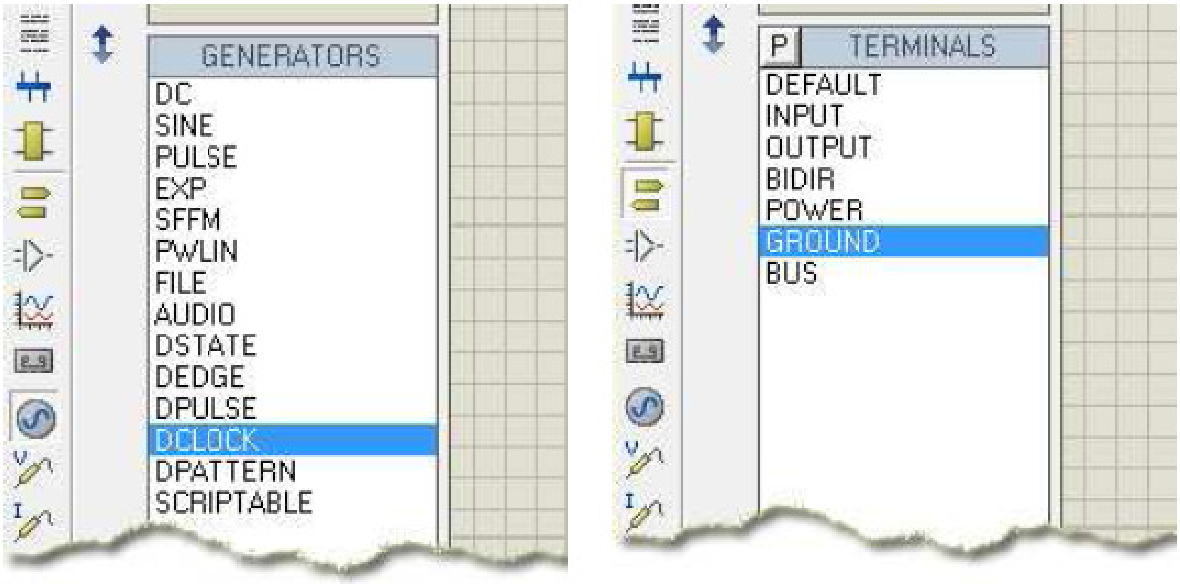
menu containing available actions on that object.

– Draw a "tagbox" around the object by depressing the left mouse button and dragging the mouse to form a box encompassing the object to be selected. This is the technique that should be used for moving multiple, connected objects or blocks of circuitry.

To place the clock generator, choose the *Generator Mode* from the left menu bar and then select **DCLOCK** from the *Generators* list box as shown in Figure 4.10(a). Similarly, you can place the ground terminal by choosing the *Terminals Mode* and then select **GROUND** from the *Terminals* list box as shown in Figure 4.10(b).

Changing Properties of a Device

When a component is placed on the drawing area, we can modify its properties either by double clicking the component object or by right clicking on the component and selecting "Edit Properties". The *Properties* window contains a number of characteristics (if available or applicable depending on the component type) such as : component name, type, value, data sheet, PCB layout, *etc*. As an example, consider the *Properties* window of the resistance "R3" in the modulo-5 counter schematic as shown in Figure 4.11. In this window, you can modify the resistance name, value, model type, or PCB package.



(a) (b)

Figure 4.10 – Placing components using *Editing Modes*. (a) clock generator, (b) ground terminal.

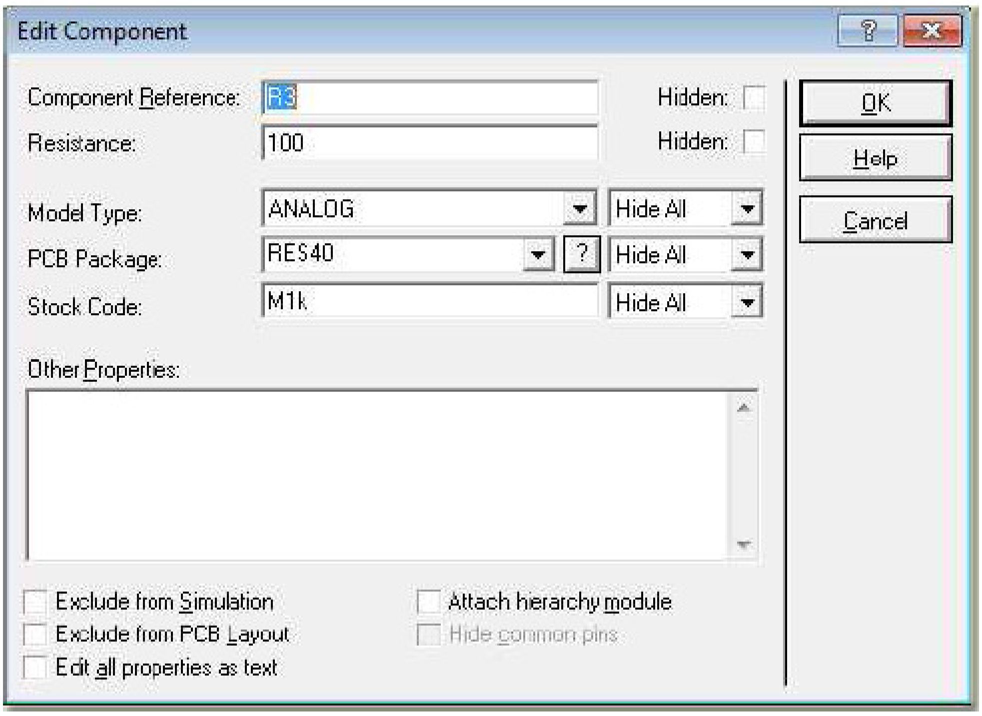


Figure 4.11 – Properties window for a resistance component.

Wiring Up

There are three main techniques used to make wiring a circuit :

1. **Modeless Wiring** : there is no "wiring mode" in ISIS - wires can be placed and edited at any time, without the need of entering a dedicated wiring mode prior to placement.

2. Follow-Me Wire Auto-routing : after starting to place a wire, the proposed route of the wire will follow the movement of the mouse orthogonally to the termination point of the wire.

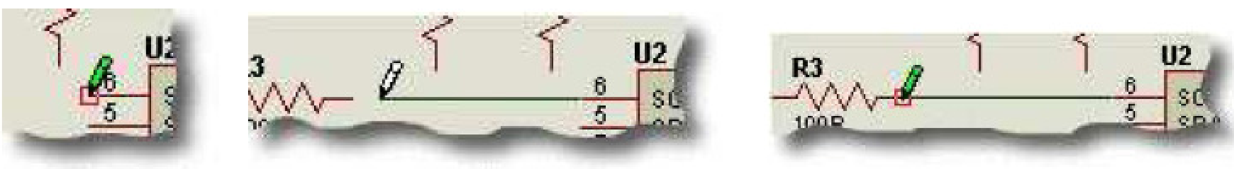
3. **Live Cursor Display** : the cursor will change as a visual indicator when wiring to show when a wire can be placed, when a wire can be terminated and when a wire is being placed.

The basic procedure for placing a wire between two pins is given below:

– Move the mouse over the first pin to be connected - the cursor will change to a green pen as shown in Figure 4.12 (a).

– Left click the mouse and then move it until it is over the second pin to be connected. The wire will follow the mouse and the cursor / pen is white during wiring as shown in Figure 4.12 (b).

– Left click the mouse again to commit the connection and place the wire as shown in Figure 4.12 (c).



(a) (b) (c)

Figure 4.12 – wring up.

The procedure for wiring onto an existing wire is almost identical but there are a couple of items to note :

– You cannot directly start a connection from an arbitrary point on a wire.

– When you terminate the connection on another wire a junction dot will be placed automatically to complete the connection as shown in Figure 4.13.

Figure 4.13 – Wiring onto an existing wire.

Making Connection with Terminals

In the case of large circuits, using direct connections between different components of the circuit usually results in many wire cluttering. This may result in connection errors and makes it difficult to understand, debug or modify the design. The solution to this problem is to use terminals for connection. Figure 4.14 shows how to wire up the counter using terminals. We can name the terminals in any fashion we liked but sensible names make the schematic more legible and easy to understand. Essentially what we are doing by labeling a terminal is making a connection to a terminal with the same name without placing a physical wire between the two objects.

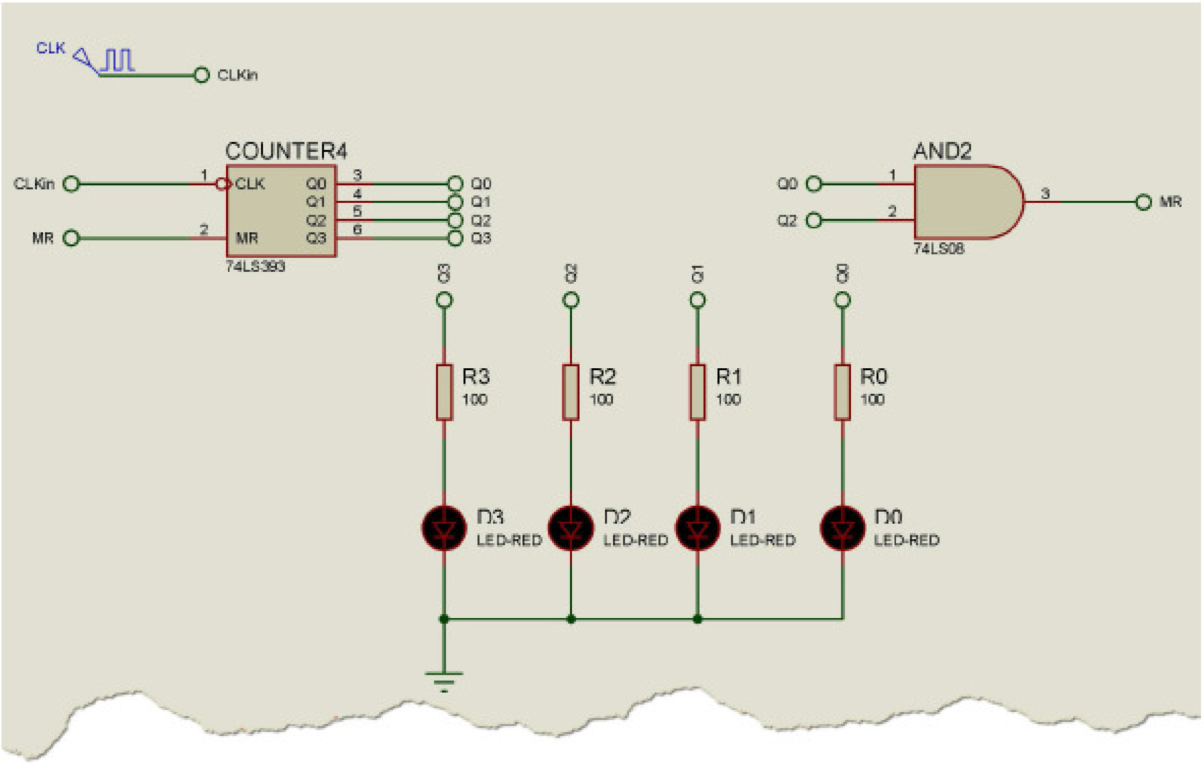


Figure 4.14 – Modified schematic of the modulo-5 counter using terminals for connection.

The *Power* and *Ground* terminals are the special type of terminals. Although there is no reason not to label them; an unlabelled power terminal is assigned to the *VCC* net and an unnamed ground terminal will be assigned to net *GND*.

You can insert a terminal into the drawing area by choosing *Terminal Mode* then selecting **DEFAULT** from the *Terminals* list box as shown in Figure 4.15. You change the orientation of the terminal using the rotate and mirror buttons from the left bar menu.

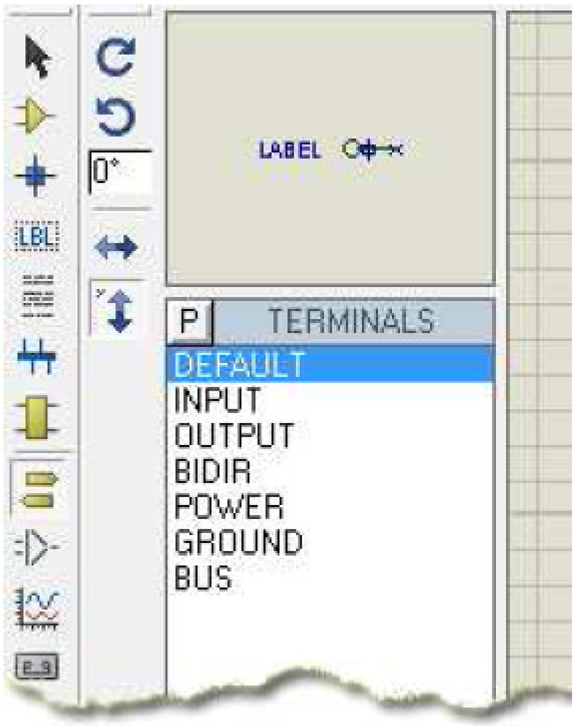


Figure 1.15– Terminals selection.

To change the terminal name, double click on the terminal point and enter an appropriate name in the "Edit Terminal Label Window" as shown Figure 4.16.

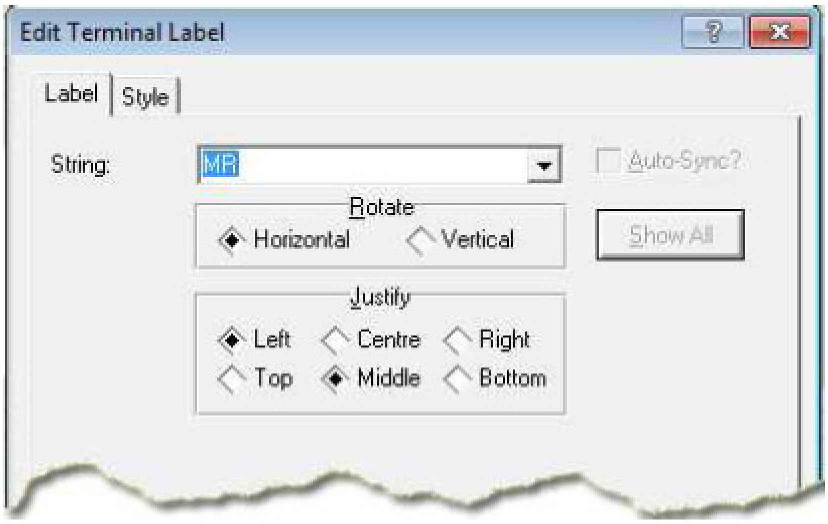


Figure 4.16 – Editing terminal label.

Power Connections

ISIS supports a powerful scheme for making power connections implicitly, thus vastly reducing the number of wires on the schematic. Almost all relevant components in ISIS have their power pins hidden (not visible on the schematic). It is important to remember that in such cases the name of the power and ground pins are set to **VCC** and **GND** by default.

To change these default values, you can use the component *Properties Window* as shown in Figure 4.17.

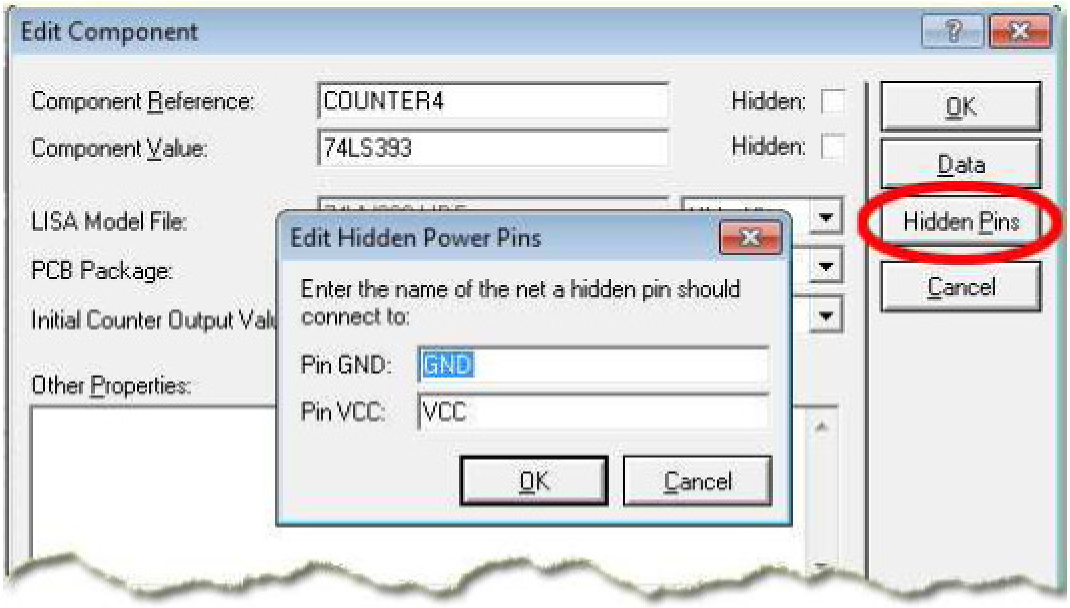


Figure 4.17 – Edit power connections default.

Component Labels and Annotation

You should see that all the components you have placed have both a unique *reference* and a *value*. The reference is set by a feature of ISIS called "*Real Time Annotation*" which can be found on the *Tools Menu* and is enabled by default. Basically, when enabled, this feature annotates components as you place them on the schematic, saving you the time and effort of doing this manually.

You have full control over the position and visibility of component labels - you can change the values, move the position or hide information that you feel is unnecessary. The discussion below details how to manipulate component labels on a per component basis.

If you zoom in on any resistor you have placed you will see that ISIS has labeled it with a unique Reference (*e.g. R*1) and also with a Value (*e.g.* 100). You can edit both these fields and their visibility via the *Edit Component* dialogue form. You can launch this dialogue form by double clicking the left mouse button over the resistor (or using one of the alternative methods discussed previously). From the resulting dialogue form you can edit both the component reference name and its value (in this case, the resistance). You can also, show or hide these two properties as highlighted in Figure 4.18.

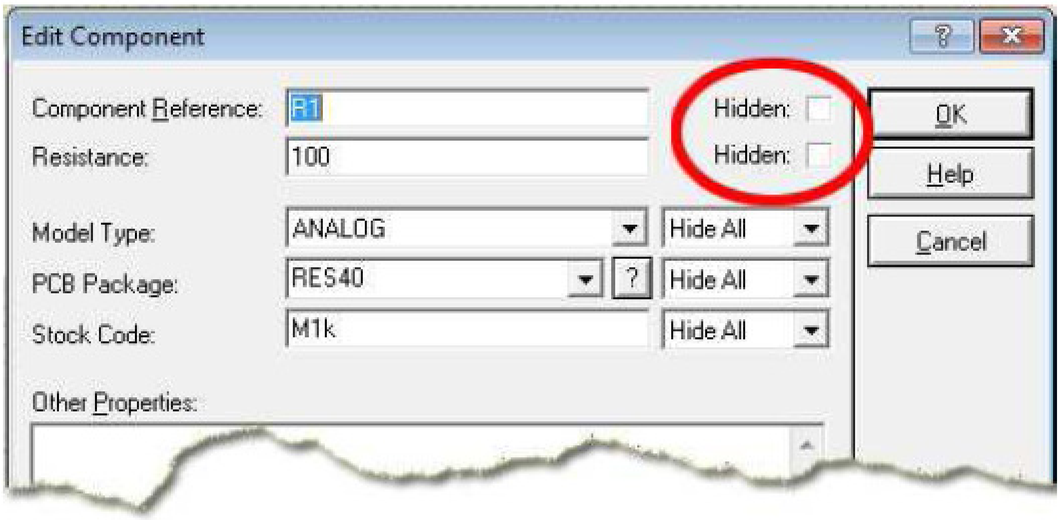


Figure 4.18 – Choose to hide or show references and values

Block Editing

ISIS editor allows you to copy, move, rotate or delete a group (block) of selected components. This action is called *block editing* which can be performed by first drag a box around the required group of components (*e.g.* devices and wires), and then applying the required editing from *Block Editing* menu as shown in Figure 4.19.

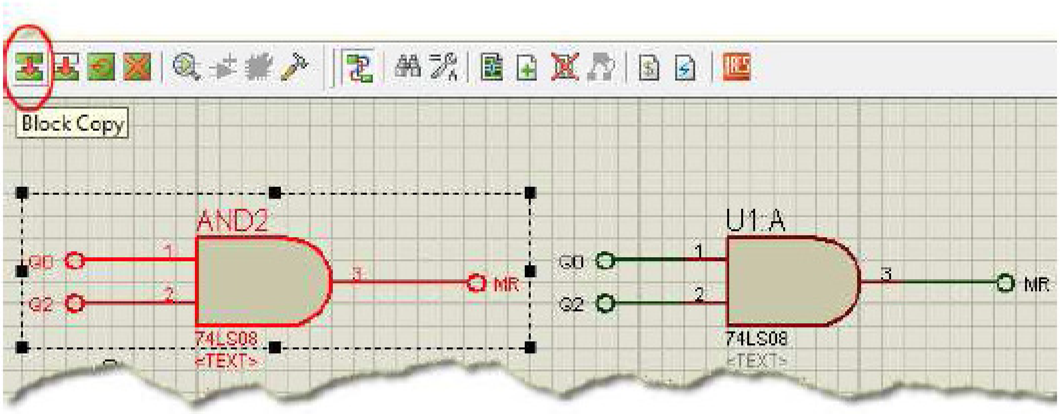


Figure 4.19 – Block editing.

Simulating the Counter

After completing the schematic editing of the counter, we can simulate design by clicking on the Play button at the bottom of the *Editing Widow* as shown in Figure 4.20.

If there are no errors in your design, then you will notice that the LEDs are blinking to reflect the count. You can check the *Simulation Log* by double clicking on the *Simulation Log Status* to right of the simulation buttons as shown in Figure 4.21. The *Simulation Log* window displays compilation status and errors.

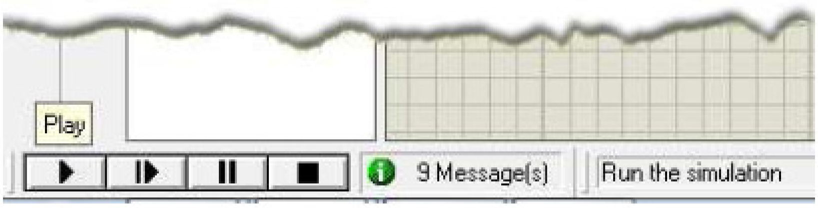


Figure 4.20 – Run simulation.

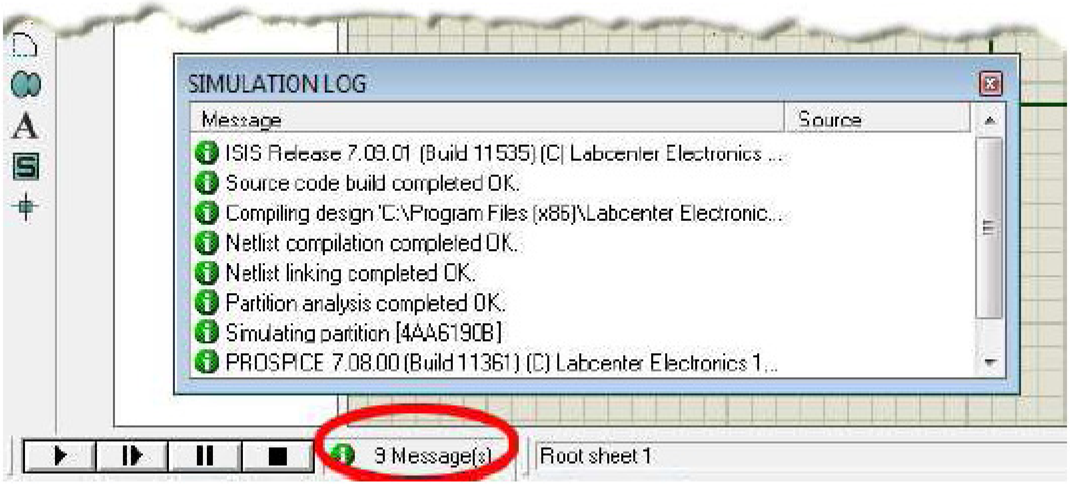


Figure 4.21 – Simulation log.

For demonstration purpose, let's change the power pin name from the default value **VCC** to another name say "V" as illustrated in Figure 4.22.

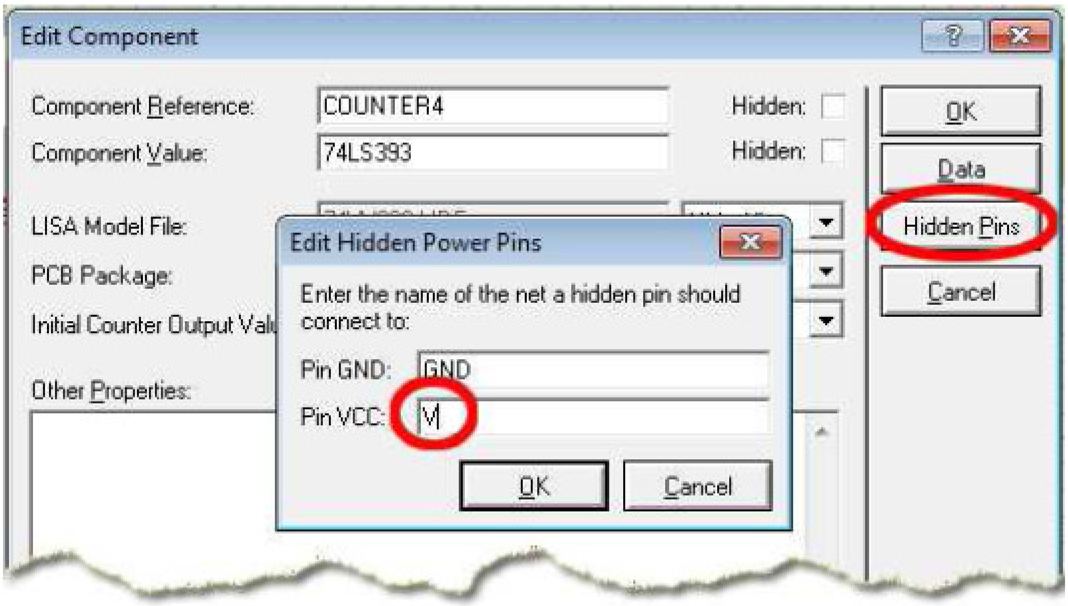


Figure 4.22 – Change power pin default.

When we run the simulation again, the compiler displays an error on the Simulation Log as shown in Figure 4.23.

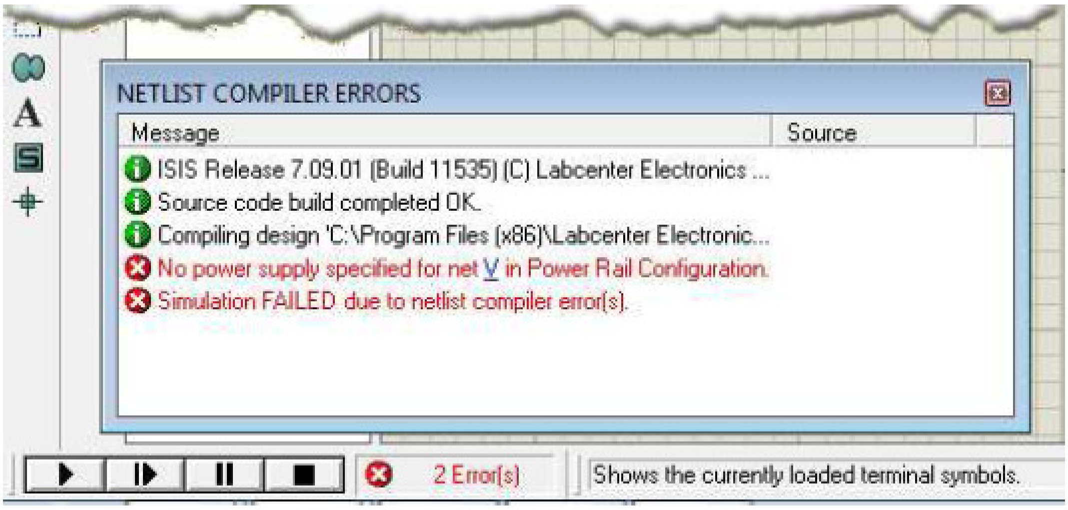


Figure 4.23 – Simulation errors.

TASKS

Exercise 4.1 List the binary counts for a counter and show how to gate the outputs of the 4-bit binary counter to implement this counter.

Exercise 4.2 Referring to Figure 4.7, place the following components into the Editing Window: one 4-bit binary counter (74LS393), four 100 resistors (MINRES100R), four light-emitting diodes (LED-RED), and one "AND" logic-gate (74LS08).

Exercise 4.3 Check the properties of the components you have placed in Exercise 4.2 and modify their names according to Figure 4.7.

Exercise 4.4 Referring to Figure 4.7, wire up the components you have placed in Exercise 4.2.

Exercise 4.5 Referring to Figure 4.14, modify your schematic for the counter replacing direct connections with terminals and labeling them appropriately.

CONTROL QUESTIONS

1. How to create a new project?
2. Why do we need the Editing modes?
3. Why do we need the Library?
4. Why do we need the Properties Window?
5. How many editing modes do you know?

REPORT FORM

German-Russian Institute of Advanced Technologies

TU-Ilmenau (Germany) and KNTRU-KAI (Kazan, Russia)

Laboratory work 4

«Proteus Virtual System Modeling (VSM)»

Student: ____________________

Teacher: ____________________

Kazan 2014