DOWNLOAD PDF ALTIUM DESIGNER TUTORIAL SCHEMATIC PCB DESIGN AND SIMULATION

Chapter 1 : Altium Designer 18 - Best PCB Design Software for Engineers

Contents â€¢ How to install Altium Designer â€¢ Understanding Altium Designer â€¢ Walk-through Tutorial -Schematic Capture -Mixed signal simulations.

The required Spice Prefix field will be set automatically. For the exceptions, you will need to set the Model Kind to General, the Model Sub-Kind to Generic Editor and enter the applicable netlist template format. You will also need to set the Spice Prefix accordingly. For more information on the level of support for PSpice models in Altium Designer, including netlist template entries and supported parameters, refer to the Support for PSpice Models in Altium Designer application note. This is a special descriptive language that allows digital devices to be simulated using an extended version of the event-driven XSpice. You will need to enter the netlist template specific to the digital device being modeled, on the Netlist Template tab of the dialog. Example of a linked Digital SimCode model. For more information on working with digital device models, refer to the Creating and Linking a Digital SimCode Model application note. This must be the name, as it appears in the model file. For an MDL file, the name must be that appearing in the. Consider a model for a diode with the following definition: This is the name that must be entered into the Model Name field. For a CKT file, the name must be that appearing in the. Consider a model for a fuse with the following definition: Use the optional Description field to enter a brief description of the model - for example its purpose - should you wish. Any - searches all valid model locations for a matching model. In Integrated Library - draws the model directly from the integrated library used to place the component instance. The integrated library must be available in a valid location. Valid model locations consist of: Model location information for the JAS33 diode example. Notice in the image above that the model has been located by using the Full Path option. When the search is across all valid model locations, the order of the search is Project Models - Installed Models - Project Search Paths. The search ceases as soon as a match is found - i. MODEL line of a. Typically the model file is named the same as the model itself, e. This is not a constraint however. Mapping the Ports Once the simulation model file has been linked to the schematic component, you need to ensure that the pins of the schematic component are correctly mapped to the pins of the model. This is carried out on the Port Map tab of the Sim Model dialog. Ensuring correct component pin-to-model port mapping. If no commenting is evident, then the pinout of the model will typically be that of the physical device itself. Consult the datasheet for the device in this case. For each schematic pin, simply use the available drop-down to change the associated Model Pin entry accordingly. If the device is multi-part, be sure to check the mapping for each part. When linking models using the Generic Editor you will need to add applicable parameters manually. For many model types it is possible to set one or more parameters at the component level. This is carried out on the Parameters tab of the Sim Model dialog. For a semiconductor capacitor, for example, specifying a value for the component-level Width parameter will override any value specified for the DEFW parameter in the associated model file. Many component-level parameters will have a default value assigned to them which, although not displayed, will be used if no specific value is set. If a parameter is specified at the component level for a digital device, that value will override the value specified for that parameter in the source SimCode definition. The image below shows the parameters available for our JAS33 diode device model. If you require any parameter values to be displayed on the schematic sheet, simply enable the corresponding Component Parameter option. Component-level parameters for the JAS33 diode example. Linking from an External Database Component placement with sim link direct from a database library. Placement is carried out from the Libraries panel which, after installing the database library, acts as a browser into your database shown above. With this method of linking the component symbol, model and parameter information for a component is stored as part of the record definition for that component in the external database. SchLib is simply an empty shell, with a defined symbol only. There are no linked models and no defined design parameters. When the component is placed, its parameter and model information is created on-the-fly, using the corresponding fields in the matched database record and in accordance with defined mapping. As part of the Database Library feature, you can add simulation model information to a component record in the external database. When the component is placed on the schematic, this information will be used to create the link to the referenced simulation model - filling-in the required and corresponding areas of the Sim Model dialog. This document should ideally be read prior to, or at least in conjunction with, the remainder of this document. Adding Sim Information to an External Database Table The following sections discuss each of the database fields that can be added to an external database table in order to define the simulation model link, which will be created upon component placement. If the field names are named exactly as indicated, the Database Field-to-Design Parameter mapping will be automatically set on the Field Mappings tab of the DBLib file. You can, of course, use your own field names in the database table, but you will then need to manually map these fields to the correct design parameters. Note that simulation information must be entered manually into the external database. We shall use our trusty JAS33 diode model to illustrate the linking. Only one simulation model link can be defined in an external database. Typically there will only ever be a single model linked to the component. Should you wish to set up multiple simulation model links, the other links will need to be defined and stored with the component in the source schematic library file. Sim Model Name Create this field in the database to specify the name of the model that you wish to use. After the component is placed, this information will appear in the Model Name field, on the Model Kind tab of the Sim Model dialog. In the database record for our example diode, we would enter JAS33 into this field. Sim Description Create this field in the database if you wish to provide a description for the linked model. This information is optional and does not affect the operation of the simulation model link. After the component is placed, this information will appear in the Description field, on the Model Kind tab of the Sim Model dialog. Sim File Create this field in the database if you wish to specify a particular model file in which to find the simulation model specified in the Sim Model Name field. There are a number of ways in which this field can be used: You can enter an absolute path to a model file e. The model specified in the Sim Model Name field will be searched for within this file and used if found. You can enter a relative path relative to the DBLib file to a model file e. You can enter the model filename only e. Search paths defined as part of the DBLib file will be used to locate the first model file that matches the specified name, and which contains a match for the model specified in the Sim Model Name field. You can leave the field blank. Search paths defined as part of the DBLib file will be used to locate the first model file containing a match for the model specified in the Sim Model Name field. Providing model location information. Sim Kind Create this field in the database to specify the parent category for the model being linked to. The text entered must be as it appears in the Model Kind field. For our example diode, we would enter General. Sim SubKind Create this field in the database to specify the type of model being linked to. The text entered must be as it appears in the Model Sub-Kind field. For our example diode, we would enter Diode. Sim Netlist Create this field in the database to enter the netlist template information, in accordance with the type of model being linked to. For all of the predefined model kinds and sub-kinds, this information can be sourced from the Netlist Template tab of the Sim Model dialog. After the component is placed, this information will appear on the Netlist Template tab of the Sim Model dialog. This field must be defined and not left blank, otherwise no entry for the model will be made in the simulation netlist, and the part will not simulate when placed from the database library. Example netlist specification for a linked digital device model 74LS For our example diode, we would enter D. For information on the Spice prefix required for a particular model type, refer to the relevant section for that model type in the SPICE3f5 Models. Sim Port Map Create this field in the database to specify the mapping of pins from the schematic component to the pins of the linked model. After the component is placed, this information will appear on the Port Map tab of the Sim Model dialog.

Chapter 2 : Altium Designer Documentation | Online Documentation for Altium Products

Altium Designer includes tools for all circuit design tasks: from schematic and HDL design capture, circuit simulation, signal integrity analysis, PCB design, and FPGA-based embedded system design and development. In addition, the Altium Designer environment can be customized to meet a wide variety of users' requirements.

See Also Welcome to the world of Altium Designer - a complete electronic product development environment. This tutorial will get you started with creating a PCB project based on an astable multivibrator design. If you are new to Altium Designer then you might like read the article The Altium Designer Environment for an explanation of the interface, information on how to use panels, and managing design documents. A project file, eg. Links to schematic sheets and a target output, eg. Once the project is compiled, design verification, synchronization and comparison can take place. Any changes to the original schematics or PCB, for example, are updated in the project when compiled. The process of creating a new project is the same for all project types. We will use the PCB project as an example. We will create the project file first and then create the blank schematic sheet to add the new empty project. Later in this tutorial we will create a blank PCB and add it to the project as well. To start the tutorial, create a new PCB project: If this panel is not displayed, select Files from the System button at the bottom right of the main design window. PrjPCB with no documents added. Rename the new project file with a. Navigate to a location where you would like to store the project on your hard disk, type the name Multivibrator. Creating a New Schematic Sheet Next we will add a new schematic sheet to the project. It is on this schematic we will capture the astable multivibrator circuit. Create a new schematic sheet by completing the following steps: A blank schematic sheet named Sheet1. SchDoc will open in the design window and an icon for this schematic will appear linked to the project in the Projects panel, under the Source Documents folder icon. Save the new schematic with a. Navigate to a location where you would like to store the schematic on your hard disk, type the name Multivibrator. SchDoc in the File Name field and click on Save. Since you have added a schematic to the project, the project file has changed too. Right-click on the project filename in the Projects panel, and select Save to save the project. When the blank schematic sheet opens you will notice that the workspace changes. The main toolbar includes a range of new buttons, new toolbars are visible, the menu bar includes new items and the Sheet panel is displayed. You are now in the Schematic Editor. You can customize many aspects of the workspace. For example, you can reposition the panels and toolbars or customize the menu and toolbar commands. Setting the Schematic Document Options Tip: In Altium Designer, you can activate any menu by pressing the menu accelerator key the underlined letter in the menu name. Subsequent menu items will also have accelerator keys that you can use to select that item. Additionally, many submenus, such as the DeSelect menu in the Edit menu , can be called directly. The first thing to do before you start drawing your circuit is to set up the appropriate document options. Complete the following steps. For this tutorial, the only change we need to make here is to set the sheet size to A4, this is done in the Standard Styles field of the Sheet Options tab of the dialog. Click OK to close the dialog and update the sheet size. Next we will set the general schematic preferences. T, P to open the schematic area of the Preferences dialog. This dialog allows you to set global preferences that will apply to all schematic sheets you work on. Open the Schematic - Default Primitives page of the dialog and enable the Permanent option on the right hand side of the dialog. Click OK to close the dialog. Note that Altium Designer has multilevel Undo, allowing you to undo many previous actions. The Undo stack size is user-configurable and limited only by the available memory on your computer, configure it in the Schematic - Graphical Editing page of the Preferences dialog. Drawing the Schematic Circuit for the multivibrator. You are now ready to begin capturing drawing the schematic. For this tutorial, we will use the circuit shown in the figure above. This circuit uses two 2N transistors configured as a self-running astable multivibrator. Locating the Component and Loading the Libraries To manage the thousands of schematic symbols included with Altium Designer, the Schematic Editor includes powerful library searching capabilities.

Although the components we require are in the default installed libraries, it is useful to know how to search through all libraries to find components. Work through the following steps to locate and add the libraries you will need for the tutorial circuit. First we will search for the transistors, both of which are type 2N If it is not visible, display the Libraries panel. The easiest way to do that is to click the System button down the bottom right of the application, then select Libraries from the menu that appears. Refer to the Working with Panels article to learn more about configuring and controlling panels. Ensure that the dialog options are set as follows: For the first Filter row, the Field is set to Name, the Operator set to contains, and the Value is The Scope is set to Search in Components, and Libraries on path. The Path is set to point to the installed Altium libraries, which will be something like C: Search installed or all available libraries for components. Click the Search button to begin the search. The Query Results are displayed in the Libraries panel as the search takes place. Click on the component name 2N found in the Miscellaneous Devices. IntLib library to select it. This library has symbols for the available simulation-ready BJT transistors. If you choose a component that is in a library that is not currently installed, you will be asked to Confirm the installation of that library before you can place a component from it. Since the Miscellaneous Devices library is already installed, the component is ready to place. Added libraries appear in the drop down list at the top of the Libraries panel, as you select a library in the list the components in that library are listed below. Use the component Filter in the panel to quickly locate a component within a library. Search results for components with the string in their name. Placing the Components on Your Schematic The first components we will place on the schematic are the two transistors, Q1 and Q2. Refer to the rough schematic sketch shown above for the general layout of the circuit. V, D to ensure your schematic sheet takes up the full window. Display the Libraries panel by clicking on its tab on the right of the workspace, if it is pop-out mode. Select the Miscellaneous Devices. IntLib library from the Libraries drop-down list at the top of the Libraries panel to make it the active library. Use the filter to quickly locate the component you need. Click on the 2N entry in the list to select it, then click the Place button. Alternatively, just double-click on the component name. The cursor will change to a cross hair and you will have an outlined version of the transistor "floating" on your cursor. You are now in part placement mode. If you move the cursor around, the transistor outline will move with it. Do NOT place the transistor yet. Before placing the part on the schematic we will edit its properties. While the transistor is still floating on the cursor, press the TAB key to open the Component Properties dialog. We will now set up the dialog options to appear as below. In the Properties section of the dialog, type in the Designator Q1. Since this is an integrated library each component has a symbol and at least 1 footprint, as well as simulation models for some of the components. Leave all other fields at their default values, and click OK to close the dialog. You are now ready to place the part. When you are in any editing or placement mode a cross hair cursor is active , moving the cursor to the edge of the document window will automatically pan the document. This can be done even when you are in the middle of placing an object Tip: Use the following keys to manipulate the part floating on the cursor: Move the cursor with the transistor symbol attached to position the transistor a little left of the middle of the sheet. Move the cursor and you will find that a copy of the transistor has been placed on the schematic sheet, but you are still in part placement mode with the part outline floating on the cursor. This feature of Altium Designer allows you to place multiple parts of the same type. This transistor is the same as the previous one, so there is no need to edit its attributes before we place it.

Chapter 3 : Linking a Simulation Model to a Schematic Component | Online Documentation for Altium Prod

How to design PCB with Altium designer? In this part, we will make PCB layout for the schematic of altium designer which I did in first part. All the components had unique reference designator.

We will create a new project file first and then add a new blank schematic sheet. If this panel is not displayed, click on the Files tab at the bottom of the workspace panels. The Projects panel displays. The new project file, PCB Project1. PrjPCB, is listed here with no documents added. Rename the new project file with a. Navigate to a location where you would like to store the project on your hard disk, type the name Filter. Next, we will create a schematic to add to the empty project file. This schematic will be for a Filter circuit. If you do not have the time to draw the schematic from scratch, open a similar project Filter. Creating a New Schematic Sheet Create a new schematic sheet by completing the following steps: A blank schematic sheet named Sheet1. SchDoc displays in the design window of the Schematic Editor and the schematic sheet is now listed under Source Documents beneath the project name in the Projects panel. Rename the new schematic file with a. Navigate to a location where you would like to store the schematic on your hard disk, type the name Filter. SchDoc in the File Name field and click on Save. Drawing the Schematic Now we can create the Filter circuit shown below in Figure 1. Before we can run a simulation, the schematic must contain components with SIM models attached, voltage sources to power the filter, an excitation source, a ground reference for the simulations and some net labels on the points of the circuit where we wish to view waveforms. In this section of the tutorial, we will locate the components required, set up their component properties and then wire the schematic. Click on the Libraries tab to display the Libraries workspace panel. This will open the Libraries Search dialog. Set the scope to Libraries on Path and make sure that the Path field contains the correct path to your libraries. If you accepted the default directories during installation, the path should be similar to C: Click on the folder icon to browse to the library folder, if necessary. Ensure that the Include Subdirectories option is enabled ticked. We want to search for all references to LF, so in the search text field at the top of the Libraries Search dialog, type LF Click on Search and the query results display in the Libraries panel as the search takes place. Placing a Simulation-ready Component The first component we will place on the schematic is the op amp, U1. For the general layout of the circuit, refer to the schematic drawing shown in Figure 1. Alternatively, double-click on the component name. The Confirm dialog will display if the library has not been installed. Click on Yes to install the library. An outlined version of the op amp appears "floating" on the cursor. You are now in part placement mode. Before placing the part on the schematic, first edit its properties. While the op amp is floating on the cursor, press the TAB key to open the Component Properties dialog for this component. In the Properties section of the dialog, set the value for the first component designator by typing U1 in the Designator field. Next, we will have a look at the SIM model that will be used when running the simulation. For this tutorial, we have used integrated libraries, which mean that the recommended models for circuit simulation are already included. Notice that the model file path name has been set and successfully found in the NSC Operational Amplifier. Click on the Model File tab to display the contents of the model file. If no model file has been found, an error message will appear in this tab. The Netlist Template shown by clicking on the Netlist Template tab will now be filled with data from the model file and can be viewed by clicking on the Netlist Preview tab. Click OK until all dialogs are closed. You are now ready to place the op amp on the schematic sheet. If you refer to the schematic diagram Figure 1 , you will notice that U1 is placed as a mirror of the symbol that is floating on the cursor. To flip the orientation of the op amp vertically before final placement, press the Y key. Position the component on the sheet and left-click or press ENTER to place it onto the schematic sheet. Exit part placement mode by right-clicking or pressing the ESC key. You must use the correct file extension for each model type. To find existing model file names in the Altium integrated libraries: Click on the Search button in the Libraries panel. The Libraries Search dialog appears. To search for all Simulation models available in the supplied libraries, type in the following query: You can also narrow down the search by adding more detail to the query, e. Search results will display in the Libraries panel. You may want to use a simulation model in your design other than the one already supplied with the component in its integrated library. If you want to use a simulation model that resides in another integrated library: Copy the model files from the output folder generated when you opened the integrated library into the folder that contains your project. You can then add this model and make modifications, if required, using the Sim Editor. Add the model file to the project by selecting the project name Filter. Choose the model file and click Open. Now we can add the model to the component in the schematic. This could also be done in the Schematic library for this component, if required. Double-click on the op amp U1 to open its Component Properties dialog. Delete the existing SIM model in the Models section by selecting it and clicking on Remove and confirming the deletion. The dialog name changes to reflect the Model Sub-Kind. Altium Designer stops searching for a model as soon as a match is found. In this example, the search will find the model file LFC. Whenever a model search does not find a match, an error will appear in the Model File tab. An interactive error will also appear in the Messages panel when you compile the project. The final step is to check the pin mapping of the new model to make sure that it matches the pin numbering of the schematic component. The way to work out the order of the pin numbers is to look at the Netlist Template tab. Note that the order for this model is 1, 2, 3, 4, 5. These correspond to the. SUBCKT header found in the Model File tab, even though these numbers may not match identically in other models as they do in this particular model file. The actual numbers in the subcircuit header are not important; what is important is the order in which the connections appear in the Spice netlist. These must match the order in the header of the. The netlist header describes the function of each pin. Use this information to link them to the appropriate schematic pin. When you have modified the pin mapping, click OK until all dialogs are closed. Now we will continue with setting up our Filter schematic for simulation. Setting Up Simulation-ready Resistors Next, we will place the two resistors. In the Libraries panel, make sure the Miscellaneous Devices. IntLib library is active. Set the filter by typing res1 in the filter field below the Library name. Click on Res1 in the components list to select it and then click the Place button. You will now have a resistor symbol floating on the cursor. In the Properties section of the dialog, set the value for the first component designator by typing R1 in the Designator field. Set up a parameter field for the resistor that will display on the schematic and be used by Altium Designer when running the circuit simulation later in this tutorial. The Value parameter can be used for any general component information but discrete components use it when simulating. If there is not a Value parameter already included in the component properties, click Add in the Parameters list section to display the Parameter Properties dialog. Enter the name Value and a value of k. This type of resistor does not require a model file and takes the value required by the Netlist Template from the Value parameter. Click OK to close the dialog and return to the Component Properties dialog. Now place resistor R2. The designator will automatically increment when you click to place it. Once you have placed the resistors, right-click or press ESC to exit part placement mode. Setting Up Simulation-ready Capacitors Now find and place the two capacitors.

Chapter 4 : Defining & running Circuit Simulation analyses - English documentation - The Altium Wiki

With Altium Designer, it won't take you more than an hour to get the hang of the schematic to PCB layout process. Previously, we focused on making a simple active amplifier using the TI LM op amp.

Modified by Jason Howie on Oct 2, Altium Designer combines a myriad of features and functionality, including: Advanced routing technology Support for cutting-edge rigid-flex board design Powerful data management tools ECAD Libraries containing over , ready-to-use components Powerful design reuse tools Real-time cost estimation and tracking Dynamic supply chain intelligence Native 3D visualizations and clearance checking Flexible release management tools All of this functionality is delivered through, and the entire design process performed within, a single Unified Design Environment - the only one of its kind, and engineered to deliver optimal productivity. The unified nature of Altium Designer allows for seamless movement of design data from one design realm to the next, but to begin with, the perceived steep learning curve can appear a formidable blockade to this productivity-enhancing landscape, and the wealth of features it contains. This, the core space for documentation specific to Altium Designer, provides all the information needed to quickly get you up and running with the software. The Altium Designer documentation is versioned. F1 mapping functionality, and other documentation links, are instilled with the smarts necessary to arrive at the correct documentation destination, for the version of the software you are actively designing with. This tutorial will take you from a blank schematic sheet all the way through to generating the files needed to fabricate the bare board for a simple 9-component circuit. The design you will be capturing and then designing a printed circuit board PCB for, is a simple astable multivibrator. The circuit - shown to the left - uses two general purpose NPN transistors, configured as a self-running astable multivibrator. Exploring Altium Designer Altium Designer includes all the editors and software engines needed to perform all aspects of the electronic product development process. All document editing, compiling and processing is performed within the Altium Designer environment. And providing further flexibility, this environment is fully customizable, allowing you to set up the workspace to suit the way you work. Coming from a different design tool? Scoot on over to the area of the documentation that looks at Interfacing to Other Design Tools. This covers not only updates to the core functionality or system resources , but also the ability to install, update, or remove additional functionality. The latter is made possible through the provision of optional Extensions. An extension is effectively an add-on to the software, providing extended features and functionality. A core set of features and functions are installed and handled transparently as part of the initial install, referred to as System Resources. In addition, a range of Optional Extensions are available - packets of functionality that are optionally installed or removed by the user as required. It is the extension concept that enables the installation to be handcrafted in accordance with design needs. This functionality could include a new importer or exporter, a tool for schematic symbol generation, or maybe support for mechanical CAD collaboration. In short, any targeted packages of functionality that extend and enhance the feature set of Altium Designer. Extensions are offered either free or paid subscribed , and can be from Altium itself, or from a Third Party. In addition, and with the Altium Developer extension , you can extend the functionality of Altium Designer yourself through use of the Altium Designer SDK Software Development Kit - creating your own extensions for the software. Managed Content Server A managed content server works in harmony with Altium Designer to provide an elegant answer to the question of handling design data with secured integrity. The server not only provides rock-solid, secure storage of data, but also enables re-release of data as distinctly separate revisions - essentially tracking design changes over time, without overwriting any previously released data. The server becomes both the source and destination of design elements, with each new design utilizing elements released to, and managed through, the server. And by designing only with elements from a managed content server, the integrity of those designs is inherently assured. Altium Designer Preferences Altium Designer provides a central location from where you can set up various preferences across different functional areas of the software. These are global system settings that apply across projects and relevant documents. Configuration of preferences is performed from within the Preferences dialog click on the control at the top-right of the workspace. Use the controls and options available on the loaded page to configure your preferences for that area of the software as required. This could be a mixture of satisfying company policy, and your preferred working environment. The Preferences dialog provides a number of useful tools to ensure your set of preferences is just as you require, including: Ability to import preferences defined in a previous instance, or version of the software. Ability to set the options and controls on the active child preferences page, or all pages, back to their defaults. And if you have a managed content server, you can formally release your Altium Designer Preferences into a target Item and revision thereof in that server. Once the preferences set has been released, and its lifecycle state set to a level that the organization views as ready for use at the design level, the preferences can be reused across installations of the software. They cover every aspect of the design - from routing widths, clearances, plane connection styles, routing via styles, and so on - and many of the rules can be monitored in real-time by the online Design Rule Checker DRC. Design rules target specific objects and are applied in a hierarchical fashion. Multiple rules of the same type can be set up. It may arise that a design object is covered by more than one rule with the same scope. In this instance, a contention exists. All contentions are resolved by a priority setting. The system goes through the rules from highest to lowest priority and picks the first one whose scope s match the object s being checked. With a well-defined set of design rules, you can successfully complete board designs with varying and often stringent design requirements. And as the PCB Editor is rules-driven, taking the time to set up the rules at the outset of the design will enable you to effectively get on with the job of designing, safe in the knowledge that the rules system is working hard to ensure that success. For an overview of the system used to verify adherence to defined rules, see Design Rule Checking. Project Compiler Violations Reference The process of compiling is integral to producing a valid netlist for a project. Connectivity awareness in your schematic diagram can be verified during compilation according to rules defined as part of the options for the design project - on the Error Reporting and Connection Matrix tabs respectively. This area of the Altium Designer documentation provides a comprehensive reference describing each of the possible electrical and drafting violations that can exist in source documents when compiling a project. By entering queries into this engine you can logically scope precisely those objects you require. A query is a string you enter using specific keywords and syntax, which will return the targeted objects. Queries are primarily defined in a Filter panel, but are also used to define scoping for PCB design rules. As you build your knowledge of the Query Language, and the functions, keywords and syntax available, you will be able to type expressions directly. However, until that level of confidence is built, the Query Helper can be a beneficial crutch on which to lean! The vastness of the Query Language may seem a little daunting to begin with, but over time you will learn to appreciate its power - building a set of favorite query expressions with which to target common sets of objects and committing them to muscle memory. And to quickly get up to speed, information is available for each of the query functions. Simply highlight or click inside any given keyword - in the Query Helper, a Filter panel, or the Full Query field of a PCB design rule - and press F1 to access its page within the documentation. For a look at how queries are used in the scoping of design rules, see Scoping Design Rules. Together with the core platform itself, these servers provide the resources of the software - its features and functionality. The resources are delivered in the form of commands, dialogs, panels, and the like. They are documented across the following sections of this documentation space:

Chapter 5 : PCB Design Tutorials & PCB Software Guides | PCBCart

Altium Designer Guide This guide is a beginner's guide to PCB design using Altium Designer and is FPGA schematic design.

We will start by creating a new project file and then add a blank schematic sheet. The Projects panel displays. The new project file, PCB Project1. PrjPCB, is listed, with no documents added. Navigate to a suitable location, type the name Filter. Next, we will create a new schematic sheet to add to the empty project file. This schematic will be for a Filter circuit. If you do not have the time to draw the schematic from scratch, you can download a similar project Filter. Creating a New Schematic Sheet Create a new schematic sheet by completing the following steps: A blank schematic sheet named Sheet1. SchDoc displays in the design window of the Schematic Editor and the schematic sheet is now listed under Source Documents beneath the project name in the Projects panel. Rename the new schematic file with a. Navigate to a location where you would like to store the schematic on your hard disk, type the name Filter. SchDoc in the File Name field and click on Save. Drawing the Schematic Now we can create the Filter circuit shown below. Before we can run a simulation, the schematic must contain components with SIM models attached, voltage sources to power the filter, an excitation source, a ground reference for the simulations and some net labels on the points of the circuit where we wish to view waveforms. In this section of the tutorial, we will locate the components required, set up their component properties and then wire the schematic. Click on the Libraries tab to display the Libraries workspace panel. This will open the Libraries Search dialog. Set the scope to Libraries on Path and make sure that the Path field contains the correct path to your libraries. Click on the folder icon to browse to the library folder, if necessary. Ensure that the Include Subdirectories option is enabled ticked. If a suitable component is not found in the installed libraries, you will need to download the library instead. IntLib library click to download. Once this has been done, repeat the search process to continue with the tutorial. Placing a Simulation-ready Component The first component we will place on the schematic is the op amp, U1. For the general layout of the circuit, refer to the schematic drawing shown above. Alternatively, double-click on the component name. The Confirm dialog shown above will display if the library has not been installed. Click on Yes to install the library. An outlined version of the op amp appears "floating" on the cursor. You are now in part placement mode. Before placing the part on the schematic, first edit its properties. In the Properties section of the dialog, set the value for the first component designator by typing U1 in the Designator field. Next, we will have a look at the SIM model that will be used when running the simulation. For this tutorial, we have used a component from an integrated library, which mean that the recommended models for circuit simulation are already included. Note that in the Model Location area of the dialog the Found In information shows that the model has been found in the integrated library. Click on the Model File tab at the bottom of the dialog to display the contents of the model file shown below. If no model file has been found, an error message will appear in this tab. Click on the Netlist Template tab to show the netlist template. Because the circuit is incomplete and has not been compiled, this detail will not be complete yet. What is shown in the schematic pin number from the component symbol, these are replaced by the name of the net connected to that pin once the circuit has been compiled. Click OK until all dialogs are closed. You are now ready to place the op amp on the schematic sheet. Position the op amp at a suitable location on the schematic sheet, and click to place. Tips on orienting a floating component are detailed below. Component placement tips while the component is floating on the cursor: As well as model files stored within integrated libraries, discrete model files can be used. Note that you must use the correct file extension for each model type. Save the file into the same folder as the other project documents. Add the model file to the project by selecting the project name Filter. Select the model file and click Open. Note that this is not required if the model is in the Projects folder, but must be done if the models are stored elsewhere, such as a central model storage folder. We can now add the model to the component in the schematic. This could also be done in the Schematic library for this component, if required. Double-click on the op amp U1 to open its Properties dialog. Note that the dialog name changes to reflect the Model Sub-Kind. Note that the file name is not used to search for the model, rather all. CKT files found in the valid locations are searched, for the. Altium Designer stops searching for a model as soon as a match is found. In this example, the search will find the model file LFC. Whenever a model search does not find a match, an error will appear in the Model File tab. An interactive error will also appear in the Messages panel when you compile the project. The final step is to check the pin mapping of the new model to make sure that it matches the pin numbering of the schematic component. Compare the detail shown in the Model File tab of the dialog, to the Schematic Pin and Model Pin details listed in the upper part of the dialog. Note that the numerical values used to identify the connections in the model file are not pin numbers, they are simply a unique numerical identifier for each connection in the model file. SUBCKT statement eg, 3 for the third connection , then set the Model Pin numerical value in the upper part of the dialog to match double click to edit. If the schematic symbol was a multi-part component, you would then repeat the process for each part in the package. When you have finished mapping the pins, click OK until all dialogs are closed. We will now continue with creating the simulation-ready Filter schematic. Adding the Simulation-ready Resistors Next, we will place the two resistors. In the Libraries panel, make the Miscellaneous Devices. Click on Res1 in the components list to select it, then click the Place Res1 button. You will now have a resistor symbol floating on the cursor. In the Properties section of the dialog, set the Designator to R1. If there is no Value parameter, add one and set its value to K. Confirm that the Visible checkbox for the Value parameter is cleared not enabled. This instructs Altium Designer to display the contents of the Value parameter in place of the Comment string. Confirm that the Visible checkbox for the Comment field is enabled. Now to check the SIM model in the Models list, down the lower right of the dialog. This type of resistor does not require a model file, and it takes the value required by the Netlist Template from the Value parameter. Click OK to close this dialog, ready to complete the placement of the resistor. Using the schematic image shown earlier on this page as a guide, position the resistor and left-click or press Enter to place the part. If required, press the Spacebar to rotate the component before placing it. Another resistor will appear floating on the cursor. Because you defined the designator of R1 during placement, the designator will automatically increment when you click to place R2, position and place it in a suitable location. Setting Up Simulation-ready Capacitors Now to find and place the two capacitors. There is a suitable capacitor in the Miscellaneous Devices. IntLib library, which should already be selected as the active library in the Libraries panel. Type cap in the filter field in the Libraries panel. Double-click on Cap in the components list to commence the placement process. In the Properties section of the dialog, set the Designator to C1. Enable the Visible checkbox for the Comment field. Position and place the capacitor in a suitable location. Right-click or press Esc to exit placement mode. Adding the Voltage Sources Now we can add the voltage sources needed to power the design when simulating. We will place the VDD power source first. To make the library available, click the Libraries button at the top of the Libraries panel, opening the Available Libraries dialog. Locate and select the Simulation Sources.

Chapter 6 : Altium Designer - Wikipedia

You can also perform a simulation directly from a SPICE netlist, allowing you to use the Altium Designer simulator in conjunction with other schematic capture tools. To do this: Include the netlist in an Altium Designer PCB project .PrjPCB).

It is developed and marketed by Altium Limited. Including a schematic, PCB module, and an auto-router and differential pair routing features, it supports track length tuning and 3D modeling. Altium Designer includes tools for all circuit design tasks: The DXP platform underlies Altium Designer, supporting each of the editors that you use to create your design. The Altium Designer environment consists of two main elements: There are a number of panels in Altium Designer, the default is that some are docked on the left side of the application, some are available in pop-out mode on the right side, some are floating, and others are hidden. Create a new project 1. Place schematic parts 1. Open your saved schematic file. Set them to any direction based on your needs. Click Tools bar, there are lots of tools for schematic modification. Update the PCB from the schematic 1. Back to the schematic, double click any component, in the Models, select Footprint and then Add. Then, the component view is like this. Then you can remove these displays according to your design requirement to make it look clear and neat. Board Shape and Layer 1. Circuit board design should be completed with a certain boundary so board shape must be redefined. Usually a PCB has some layers. In the process, we have to define the specific layer for components on custom PCB. Click Design and select Layer Stack Manager. If a component is on the bottom layer of a PCB, double click the component display and set its Layer in Component Properties. Then, the layer situation of this component will be shown in the PCB design graphic. Rules and Routing 1. Click Design, select Rule Click Auto Route, and then select All Auto routs is basically not the best way. Click Interactively Route Connections and you can set your routes as you wish. Mounting Holes and File Generation 1. Click the yellow hole on the top. Here, you can set the hole size, shape, net and so on. You can add a string to your PCB. Click Place and select String. You can edit the content of the string and set it placed at any layer of your PCB. Up to now, you have finished most of your PCB design. Of course, the function realization in this tutorial is within the basic range. They all depend on your practice and utilization. PCBCart is here to help! Got PCB design files ready for production? You may start from getting your PCB price.

Chapter 7 : Altium designer PCB designing tutorial step by step guide

Altium Designer Tutorial 1 for beginners: Schematic capture and PCB layout - Part1 Altium Designer Tutorial 1: Schematic capture and PCB layout - Part2 Robeson Design Recommended for you.

Altium designer PCB designing tutorial step by step guide: In this article, we will learn how to use Altium designer for PCB designing. I will teach you step by step how to create schematic with Altium designer? Altium designer is very popular PCB designing software in among industrial people. Due to its expensive price, it is not so much popular among students. As you can see in fig 2. Here I have quite a few options, I can either use the libraries tab on the left where I can choose the library and look for the components. I could use the place part dialogue which I can bring up either by clicking. By clicking at the place part or by using the keyboard shortcut P. As shown in figure 3. As shown in fig 4. If you are not sure about the name, you can change this to As shown in figure 5. And in a search for that usually you would have to make some components yourself. As shown in fig 6. And there are already existing some parts in the library. Show in fig 7. And next again I click Ok in previous window. Then that part will be generated as in fig 8. Going back by right click to previous window place part dialogue , now right click or you can press the Escape Both of them usually work the same way. Now I found the 3 5 5 5 chip and I am going to look for the resistors, capacitors, and place those as well. Fig 9 a and I will find headers in the misc connectors library header. Fig 9 b press ok and again does ok. And make P1 first. Then I can click the X or Y to flip it the right way. And those can be found in misc devices. After that I place these as before, right-click to cancel that and now finish up by placing the Power Ports. Fig 12 Text default, here we have VCC and to change that before placing, I can press tab and specify the name that I want for this. Then press Ok Now I place VCC10 and you will notice that the Red Cross appears in the junction and that indicates that electrical connection will be made. After placing VCC, now we place the grounds and finish off. Now I am going to place wire. Fig 14 Now I can set the component value. And now double click the value to change it. Then editing it directly. Last option is to specify the value and delete the capacitor and go into the place portal and use the capacitor from here. Before placing them, press tab here I specify the value and add the footprint.. Before I finish I have to give the components unique reference designator names. Fig 16 Now all the question marks and names here. How to design PCB with Altium designer? In this part, we will make PCB layout for the schematic of altium designer which I did in first part. All the components had unique reference designator. Add PCB in this project from where we took new project. Now save the PCB file in project directory. Now I can synchronize them. Then I go up to an icon named design and import changes. Just I move around just like you would in the schematic drag with the right button to pane drag with the middle mouse button to zoom in and out. And here see a collection of components. As shown in fig 1. The red rectangle which is a room and we see that this room as the name square wave circuit that corresponds to the schematic sheets named square vase wave circuit. This tells you that rooms are automatically generated for each schematic in your project. Now I resize the room and set it at the corner. The entire black area and put here room. Now I start placing the corners of board place where you can see the edge mode is in degree angle mode. So to toggle between different edge modes or corner much I should say you use shift space just do that a couple of times and you will get a degree corner. As sown in fig 2. Now I edit origin and I will set my origin somewhere smart like a corner of your board. Here you can add layers as you wish or plains. Now make sure the rules from designs. I can set the rules manually. Shown in fig 3. You check them with the specifications that your manufacturer gives you. Now I can move components around placing them in the arrangement that seems good for the routing. As sown in fig 4 a. Now join these as you can see in fig 4 b Now that pressing tab gives you a list of a collection of settings that you can specify the trace width but you can also specify how Altium Designer behaves routing. You can choose different modes. I recommend turning on push obstacles and turning on follow miles trail and also try Strong You must try different things to see how they work. Well now press OK. Now complete its junctions. As you can see in fig

6. Now have got my board wire. After choosing Gerber files, I can specify the resolution. Here I can choose the layers that I want to include because this is just one layer board so I choose the first to player copper the top solder mask. Here on the fabrication outputs and choose NC Drill Files and remaining options leave as default Now go to Files and import the Dell files. I would navigate to the project directory and to the projects output directory. After selecting drill files, Click Ok. They are located in the project outputs directory. As shown in fig 8. And the Drill files with the extension of. And that should be enough to get you started should also read the documentation and check your manufacturer.

Chapter 8 : Getting Started with Altium Designer - YouSpice

Set up a parameter field for the resistor that will display on the schematic and be used by Altium Designer when running the circuit simulation later in this tutorial. The Value parameter can be used for any general component information but discrete components use it when simulating.

Chapter 9 : Altium Designer PCB Design Tutorial | PCBCart

Linking the Model to a Schematic Component Once you have the required simulation model file, you will need to link it to the schematic component that you wish to place and use in your design. To illustrate by example the steps that need to be taken to link a model to a schematic component, we shall use a simple (and fictitious) diode model.