

# Vivado Design Suite Tutorial

## *Designing with IP*

UG939 (v2022.1) June 10, 2022

Xilinx is creating an environment where employees, customers, and partners feel welcome and included. To that end, we're removing non-inclusive language from our products and related collateral. We've launched an internal initiative to remove language that could exclude people or reinforce historical biases, including terms embedded in our software and IPs. You may still find examples of non-inclusive language in our older products as we work to make these changes and align with evolving industry standards. Follow this [link](#) for more information.



# Table of Contents

<b>Designing with IP Overview.....</b>	<b>4</b>
Navigating Content by Design Process.....	4
Introduction.....	4
Tutorial Description.....	5
Hardware and Software Requirements.....	5
Locating and Preparing the Tutorial Design Files.....	6
<b>Lab 1: Customizing the FIFO Generator.....</b>	<b>7</b>
Introduction.....	7
Step 1: Open the Project.....	7
Step 2: Customize the FIFO Generator.....	10
Step 3: Instantiate IP into the Design.....	16
Step 4: Synthesize the Design.....	19
Conclusion.....	21
<b>Lab 2: Creating and Managing Reusable IP.....</b>	<b>23</b>
Introduction.....	23
Step 1: Start a Manage IP session.....	23
Step 2: Customize the FIFO Generator.....	25
Step 3: Customize the Clocking Wizard.....	30
Step 4: Use Third-Party Simulators.....	31
Step 5: Add Additional IP.....	34
Step 6: Use Third-Party Synthesis Tools.....	35
Step 7: Create a Netlist Project.....	36
Conclusion.....	39
<b>Lab 3: Scripting the Project Mode.....</b>	<b>41</b>
Introduction.....	41
Step 1: Create a Project.....	41
Step 2: Add RTL Source Files.....	42
Step 3: Add XDC Constraints.....	43
Step 4: Add Existing IP.....	44

Step 5: Disable the IP XDC Files.....	45
Step 6: Upgrade an IP.....	46
Step 7: Setup Design Runs for IP.....	47
Step 8: Launch Synthesis and Implementation.....	48
Step 9: Run the Script.....	49
Conclusion.....	51
<b>Lab 4: Scripting the Non-Project Mode.....</b>	<b>52</b>
Introduction.....	52
Step 1: Read the Design Source Files.....	52
Step 2: Add Existing IP.....	53
Step 3: Disable XDC Files.....	55
Step 4: Upgrade IP.....	56
Step 5: Create DCP for IP.....	57
Step 6: Run Synthesis.....	58
Step 7: Run Implementation.....	59
Step 8: Run the Script.....	60
Conclusion.....	62
<b>Appendix A: Additional Resources and Legal Notices.....</b>	<b>64</b>
Xilinx Resources.....	64
Documentation Navigator and Design Hubs.....	64
References.....	64
Revision History.....	65
Please Read: Important Legal Notices.....	65

# Designing with IP Overview

---

## Navigating Content by Design Process

Xilinx® documentation is organized around a set of standard design processes to help you find relevant content for your current development task. This document covers the following design processes:

### Hardware, IP, and Platform Development

Creating the PL IP blocks for the hardware platform, creating PL kernels, subsystem functional simulation, and evaluating the Vivado® timing, resource use, and power closure. Also involves developing the hardware platform for system integration. Topics in this document that apply to this design process include:

- [Step 4: Add Existing IP](#)
- [Step 6: Upgrade an IP](#)

### System Integration and Validation

Designing a PCB through schematics and board layout. Also involves power, thermal, and signal integrity considerations. Topics in this document that apply to this design process include:

- [Structural Netlists for Simulation](#)
  - [Step 8: Launch Synthesis and Implementation](#)
- 

## Introduction



**IMPORTANT!** *This tutorial requires the use of the Kintex®-7 family of devices. You must update your Vivado® tools installation if you do not have this device family installed. See Vivado Design Suite User Guide: Release Notes, Installation, and Licensing (UG973) for more information on Adding Design Tools or Devices.*

---

The Vivado Design Suite provides multiple ways to use IP in a design. You can customize and add it from the IP catalog into a project. You can also create and reference a repository of customized IP in either a project or non-project based flow, with full scripting capabilities as well.

The Vivado Design Suite provides an IP-centric design flow that helps you quickly turn designs and algorithms into reusable IP.

The Vivado IP catalog is a unified IP repository that provides the framework for the IP-centric design flow. This catalog consolidates IP from all sources including: Xilinx® IP, IP obtained from third parties, and end-user designs targeted for reuse as IP into a single environment.

The Vivado IP packager is a unique design reuse feature based on the IP-XACT standard. The IP packager provides any Vivado user the ability to package a design at any stage of the design flow and deploy the core as system-level IP.



**VIDEO:** You can also learn more about the creating and using IP cores in the Vivado Design Suite QuickTake videos: [Configuring and Managing Custom IP](#) and [Customizing and Instantiating IP](#).

---

## Tutorial Description

This tutorial contains several labs as described in the following:

- **Lab 1:** Open a modified version of the Xilinx `wave_gen` example design that is missing a FIFO; locate and customize the IP in the catalog; and instantiate the IP into the design.
- **Lab 2:** Create and customize IP using the Manage IP flow. Create a project, include an IP from the IP catalog as the top-level source; customize and verify the IP. Optionally, use the customized IP as a black box in a third-party synthesis flow.
- **Lab 3:** Write and run a Tcl script using the Vivado® Design Suite to create a project, add IP, upgrade IP, disable IP sources, and generate output products including synthesized design checkpoints (DCP).
- **Lab 4:** Write and run a Non-Project Tcl script using the Vivado Design Suite to read in IP sources, upgrade IP, disable IP sources, and generate output products including a design checkpoint (DCP) file.

---

## Hardware and Software Requirements

This tutorial requires that the 2021.2 Vivado Design Suite software release or later is installed.

See the *Vivado Design Suite User Guide: Release Notes, Installation, and Licensing* ([UG973](#)) for a complete list and description of the system and software requirements.

---

# Locating and Preparing the Tutorial Design Files

There are separate project files and sources for each of the labs in this tutorial. You can find the design files for this tutorial on the Xilinx website: [Reference Design Files](#).

1. Download the zipped reference file from the Xilinx website.
2. Extract the zip file contents into any write-accessible location on your hard drive, or network location.

The extracted source directory is referred to as `<Extract_Dir>` throughout this tutorial.



---

**RECOMMENDED:** *You will modify the tutorial design data while working through this tutorial. Use a new copy of the `<Extract_Dir>` directory each time you start this tutorial.*

---

# Customizing the FIFO Generator

---

## Introduction

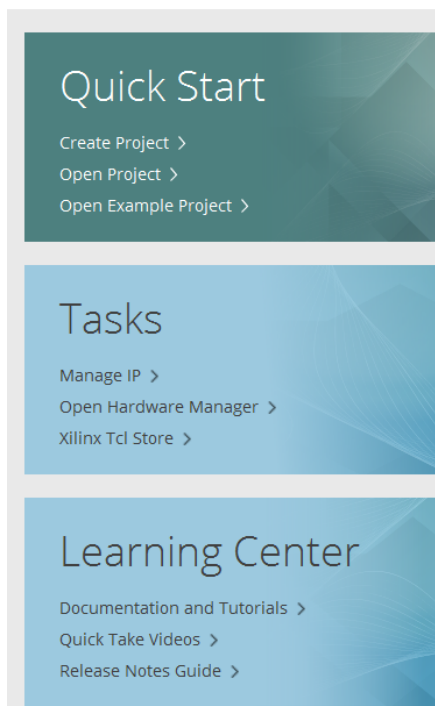
In this lab, you open a Vivado® project and customize the FIFO Generator IP core. You will generate the output products for the IP and instantiate it in the design RTL source. Finally, you will synthesize the project.

---

## Step 1: Open the Project

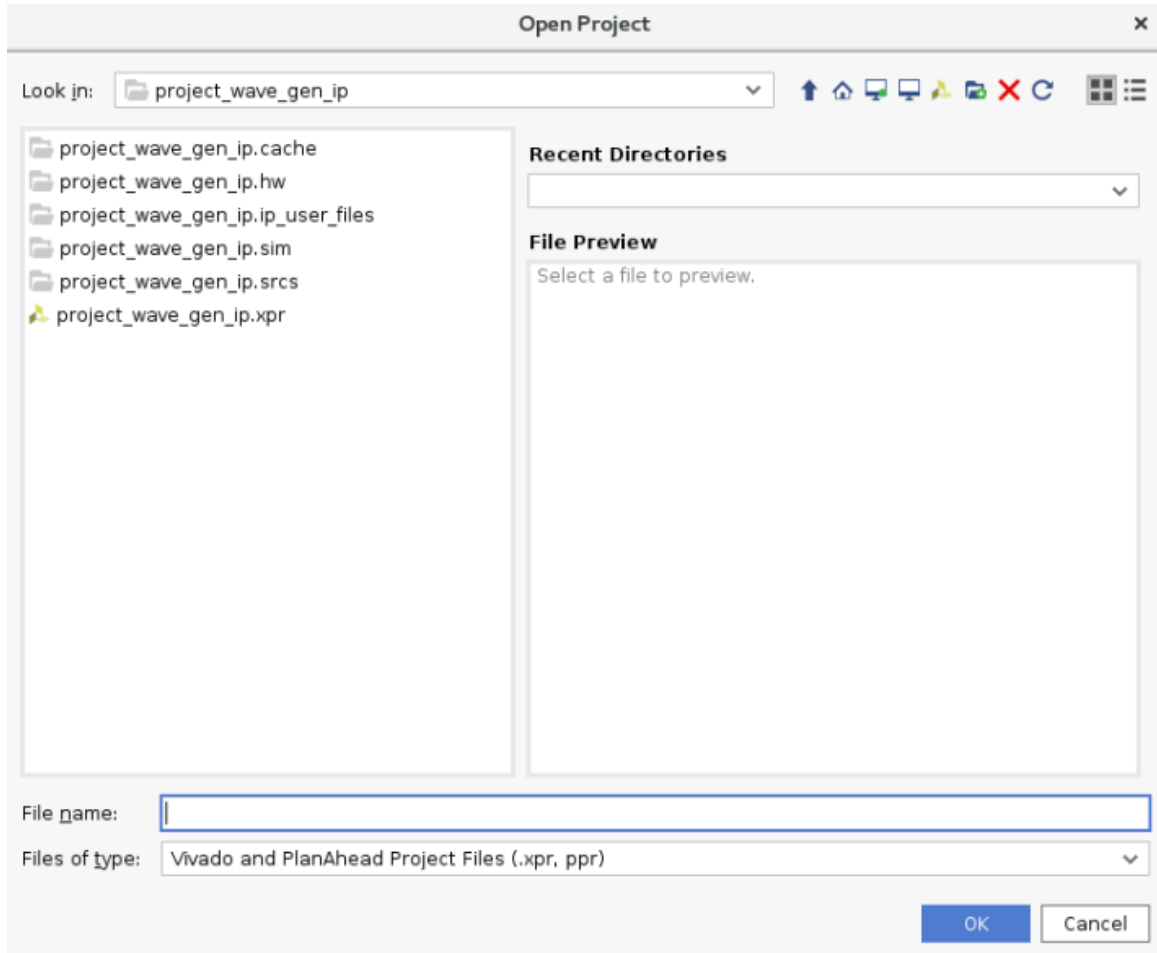
1. From the Getting Started page, select **Open Project** and browse to: `<extract_dir>/lab_1/project_wave_gen_ip`.
2. On Windows, select **Start** → **All Programs** → **Xilinx Design Tools** → **Vivado 2022.1** → **Vivado 2022.1** to launch the Vivado® Design Suite.
3. As an alternative, click the **Vivado 2022.1** Desktop icon to start the Vivado IDE.

The Vivado IDE Getting Started page, shown in the following figure, contains links to open or create projects and to view documentation.

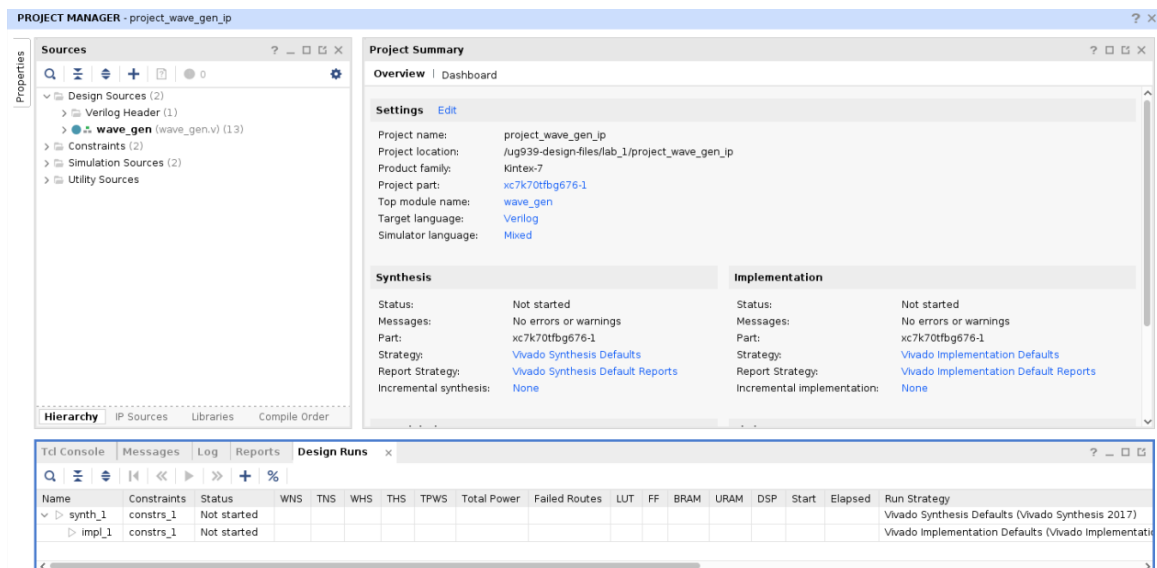


4. From the Open Project window, shown in the following figure, select **project\_wave\_gen\_ip.xpr**, and click **OK**.





The design loads and you see the Vivado IDE in the default layout view, with the Project Summary information as shown in the following figure.



Because this is an RTL project, you can run behavioral simulation, elaborate the design, launch synthesis and implementation, and generate a bitstream for the device. The Vivado IDE also offers a one-button flow to generate a bitstream, which will automatically launch synthesis and implementation. For more information, see the *Vivado Design Suite User Guide: Using the Vivado IDE* ([UG893](#)).

---

## Step 2: Customize the FIFO Generator

1. From the Flow Navigator, select **IP Catalog**.

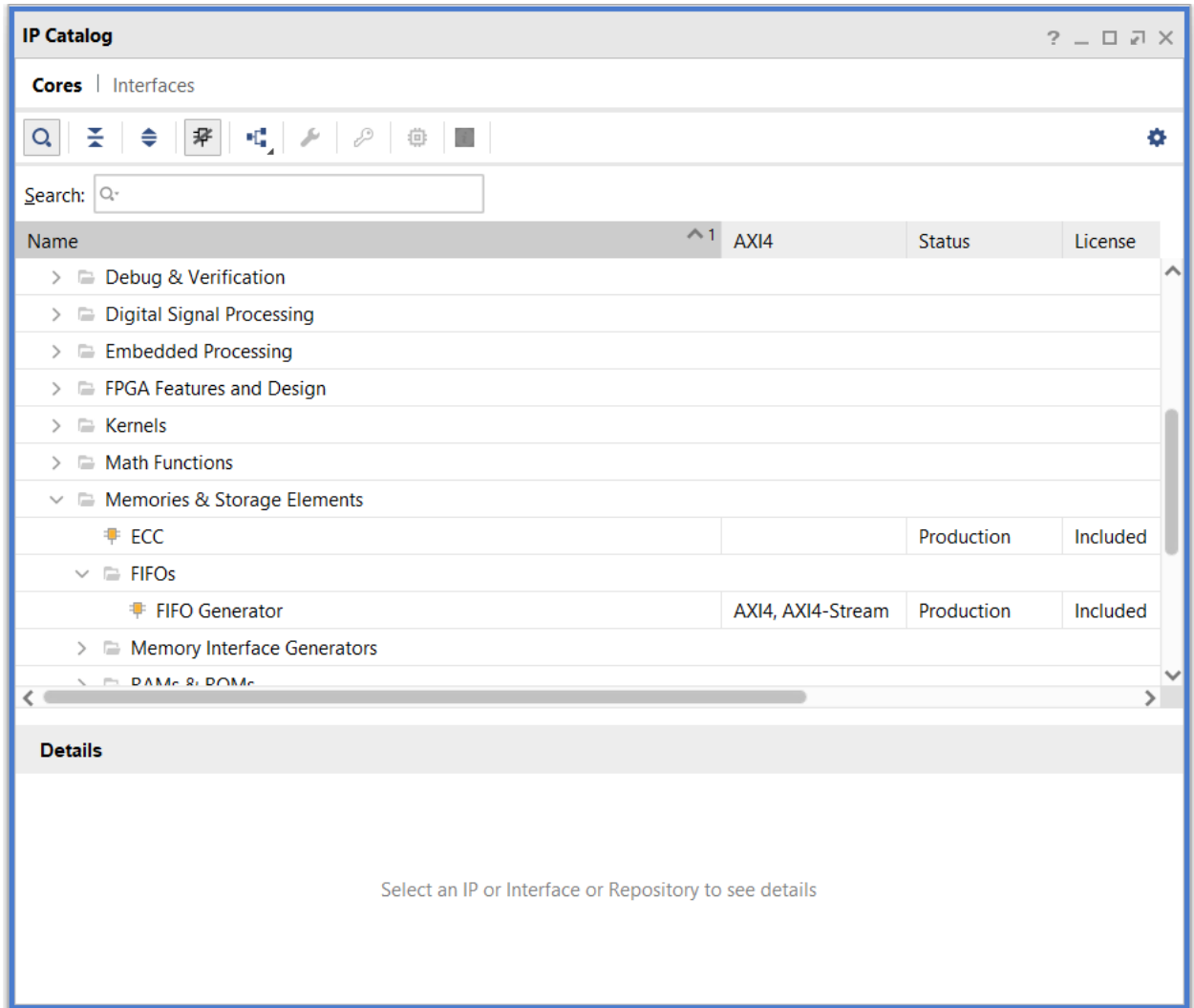
The Xilinx IP catalog displays in a new tab.

2. You can work with the IP catalog in a variety of ways. You can search using keywords in the search box or browse through the catalog in the various categories.

3. Type `fifo` in the search box.

The search results narrow the list of IP definitions displayed in the catalog.

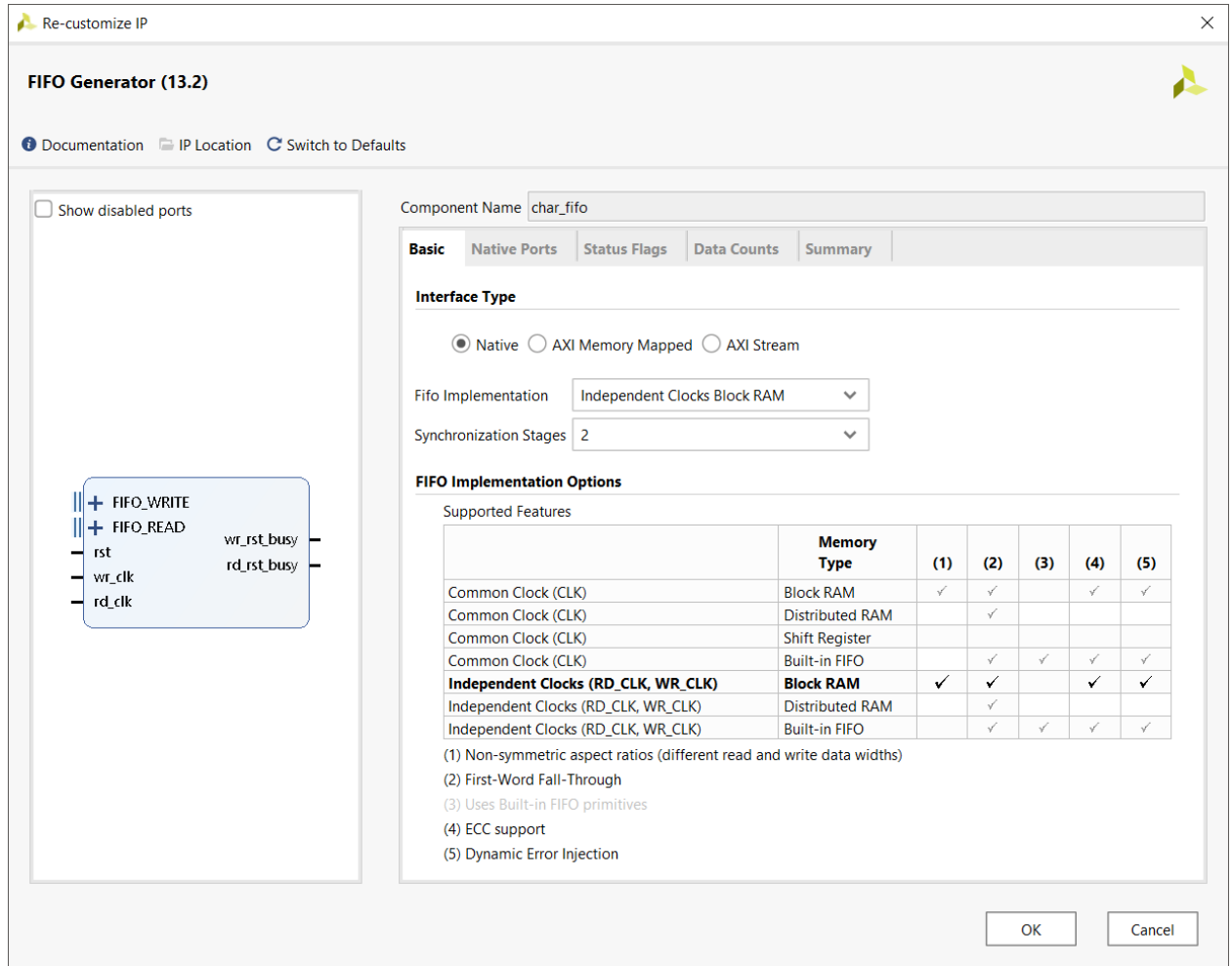
4. From the **Memories & Storage Elements** → **FIFOs** group select **FIFO Generator**, as shown in the following figure.



- Right-click and select **Customize IP**, or double-click the selected IP.

The FIFO Generator customization window opens, as shown in the following figure. There is a schematic symbol for the selected core displayed on the left. The schematic symbol changes as you customize the IP.

**Note:** This is not the original setting; it shows the change that you make to the customization.



6. Zoom into the schematic symbol using mouse strokes with the left mouse button, just like in the Device window.
7. If checked, disable the **Show Disabled Ports** checkbox to hide unused ports on the symbol.
8. Above the symbol, click the **Documentation** menu to examine the options for viewing available information.

The Documentation menu lets you open the PDF file datasheet for the IP, open the change log to review the revision history of the core, or open an Internet browser to navigate to the IP webpage, or view any Answer Records related to the IP.

The IP Location specifies the location to which to save the IP customization (XCI) file and any generated output products. By default, these are saved inside the project directory structure in the `project_wave_gen_ip/project_wave_gen.gen/ip` directory.

The Switch to Defaults option resets the configuration options back to the default settings.

9. In the Customize IP dialog box, change the following:
  - a. **Component Name** to `char_fifo` from the default name.

The Basic tab defines the interface type, memory type, and other implementation IP options.

- b. Default Interface Type of `Native`.
- c. Change **Fifo Implementation** to **Independent Clocks Block RAM**.

The customization window should now look like the preceding figure.

- 10. Select the **Native Ports** tab to set the Read Mode, Data Port Parameters, ECC and Output Register Options, and configure Initialization.

- a. Set **Read Mode** to **First Word Fall Through**.
- b. Set the **Write Width** to 8 bits.

Setting the Write Width automatically changes the Read Width to match when you click in the Read Width field.

- c. Click the **Read Width** field to automatically change it to 8 bits.

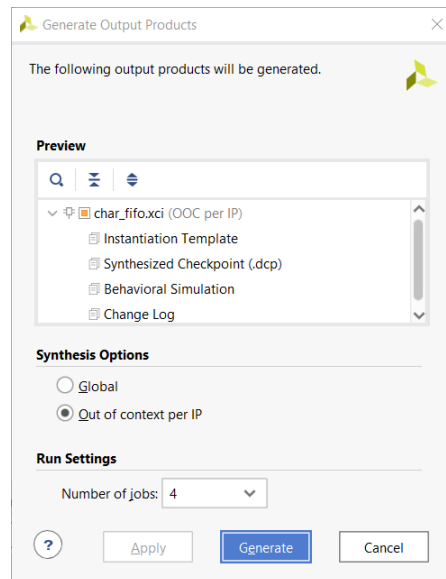
- 11. Leave everything else with the default settings on this tab.

- 12. Examine the fields of the Status Flags and Data Counts tabs.

These fields configure other options for the FIFO Generator. For this design, leave everything with the default settings.

- 13. Click **OK**.

The Generate Output Products dialog box opens for you to generate required files, as shown in the following figure.



- 14. Leave **Out of context per IP** selected, and click the **Generate** button.

- 15. Click **OK**.

By default, the Vivado IDE generates the Synthesized Checkpoint (DCP) file.

To create this DCP, You see that the Vivado tools have added an OOC module run for the `char_fifo` IP to the Design Runs window, and launched the run to synthesize the IP, as shown in the following figure.

Name	Constraints	Status	WNS	TNS	WHS	THS	TPWS	Total Power	Failed Routes	LUT	FF	BRAMs	URAM	DSP	Start	Elapsed	Run Strategy
synth_1 (active)	constrs_1	Not started															Vivado Synthe
impl_1	constrs_1	Not started															Vivado Implem
Out-of-Context Module Runs																	
char_fifo_synth_1	char_fifo	synth_design Complete!								82	1...	0.50	0	0	10/...	00:02:09	Vivado Synthes

The out-of-context (OOC) run creates the DCP for the specified IP customization, which does the following:

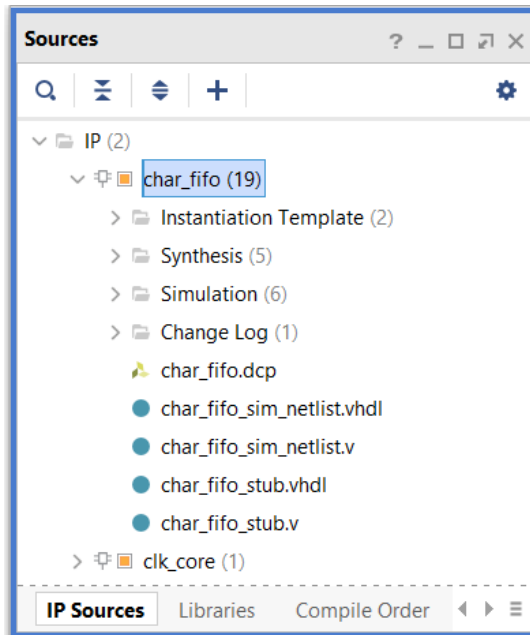
- Allows the IP to be complete with regard to synthesis.
- Ensures that the integrity of the core is preserved in the current design.
- Reduces synthesis time for the top-level design in future iterations of the design flow.

For more information on OOC runs, and the use of DCP files, see the *Vivado Design Suite User Guide: Designing with IP (UG896)*.



**IMPORTANT!** Generate the DCP file to reduce synthesis time for the top-level design. Do not change the default synthesis setting for the IP Design Runs.

The FIFO now appears in the Sources > IP Sources view, as shown in the following figure.



16. In the Sources window > IP Sources tab, examine the output products produced for the FIFO Generator customization. If you had not generated the output products, they would generate automatically when you launch the top-level synthesis run.

The FIFO IP is added to the design sources, but is not yet instantiated into your design. It is added at the same level as the top-level `wave_gen` module in the Hierarchy tab, and appears in the `Block Sources` folder in the Libraries and Compile Order tabs.

The FIFO Generator customization includes instantiation templates (this tutorial uses `char_fifo.v eo`), synthesis constraints, VHDL entity and architecture definition, Verilog simulation files, and the synthesized design checkpoint (DCP).

**Note:** A `char_fifo.v ho` template also exists, but will not be used in this design.

**Note:** Because the FIFO IP was originally defined in VHDL, the entity and architecture are added as VHDL source files. However, because Verilog is the target language for the current project, the instantiation template is a VEO file for instantiating the VHDL FIFO entity into the Verilog design. You can also customize and add IP to your design using Tcl commands.

17. Examine the Tcl Console in the Vivado IDE, as shown in the following figure and review the Tcl commands used to add the FIFO Generator core to your project.

```

Tcl Console
INFO: [IP_Flow 19-1686] Generating 'Synthesis' target for IP 'char_fifo'...
INFO: [IP_Flow 19-1686] Generating 'Simulation' target for IP 'char_fifo'...
INFO: [IP_Flow 19-1686] Generating 'Change Log' target for IP 'char_fifo'...
export_ip_user_files -of_objects [get_files c:/Tutorials/workspace/ug939-design-files/lab_1/project_wave_gen_ip/project_wave_gen_ip.srcs/sources_1/i
create_ip_run [get_files -of_objects [get_fileset sources_1] c:/Tutorials/workspace/ug939-design-files/lab_1/project_wave_gen_ip/project_wave_gen_ip
launch_runs -jobs 4 char_fifo_synth_1
[Mon Oct 15 15:03:30 2018] Launched char_fifo_synth_1...
Run output will be captured here: C:/Tutorials/workspace/ug939-design-files/lab_1/project_wave_gen_ip/project_wave_gen_ip.runs/char_fifo_synth_1/run
export_simulation -of_objects [get_files c:/Tutorials/workspace/ug939-design-files/lab_1/project_wave_gen_ip/project_wave_gen_ip.srcs/sources_1/ip/c

```

The following commands are related to the creation, customization, generation, and launch of the IP run:

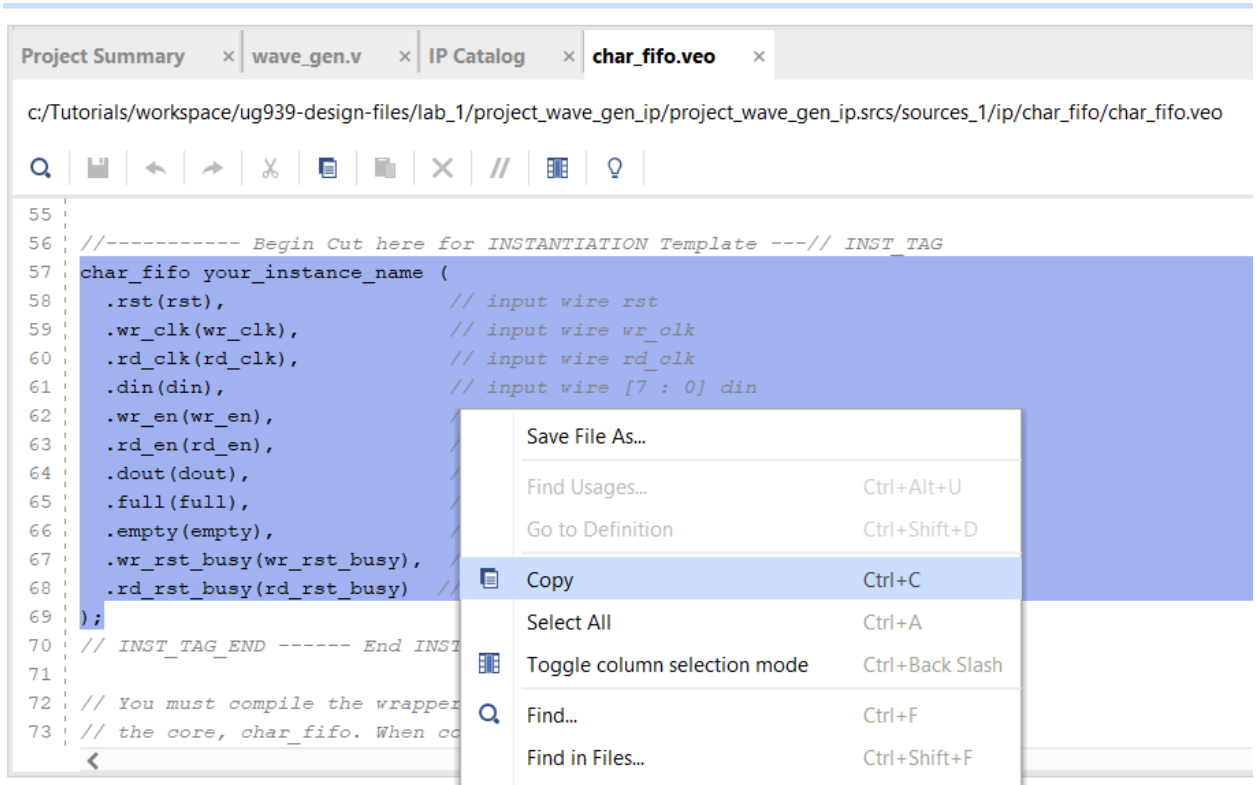
- The `create_ip` command adds the IP into the current project.
- The `set_property` command sets the various configuration options selected in the Customize IP dialog box. Scroll up in the screen to see this command.
- The `generate_target` command creates the specified output products for the customized IP. Scroll up in the screen to see this command.
- The `create_ip_run` command creates the Out-of-Context synthesis run for the customized IP.
- The `launch_runs` command runs synthesis on the IP customization to produce the synthesis DCP file, functional simulation netlists, and stub files for third-party synthesis tools to infer a black box for the IP.

See the *Vivado Design Suite Tcl Command Reference Guide* ([UG835](#)) for specific information on the different Tcl commands used in this tutorial.

## Step 3: Instantiate IP into the Design

You will now instantiate the IP customization into the design by copying and pasting the Verilog Instantiation Template into the appropriate Verilog source file in your project and modifying the signals.

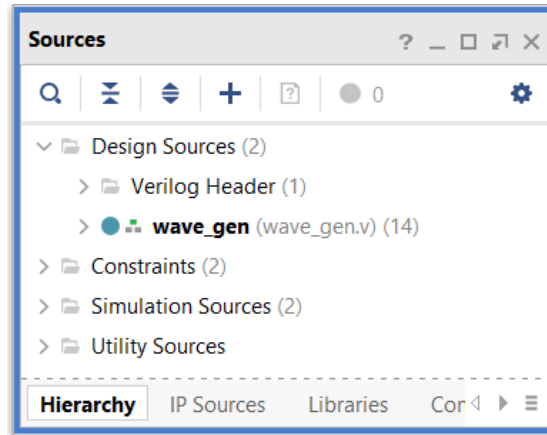
1. In the IP Sources tab of the Sources window, click the **Instantiation Template** to expand it and then double-click the **char\_fifo.veo** file to open the template in the Vivado® Text Editor.
2. Scroll down to line 57 of the template file, and select and copy the module instantiation text, as shown in the following figure.



Next, you paste the instantiation template into the appropriate RTL source file. In this case, you paste the module into the top-level of the design, in the `wave_gen.v` source file.

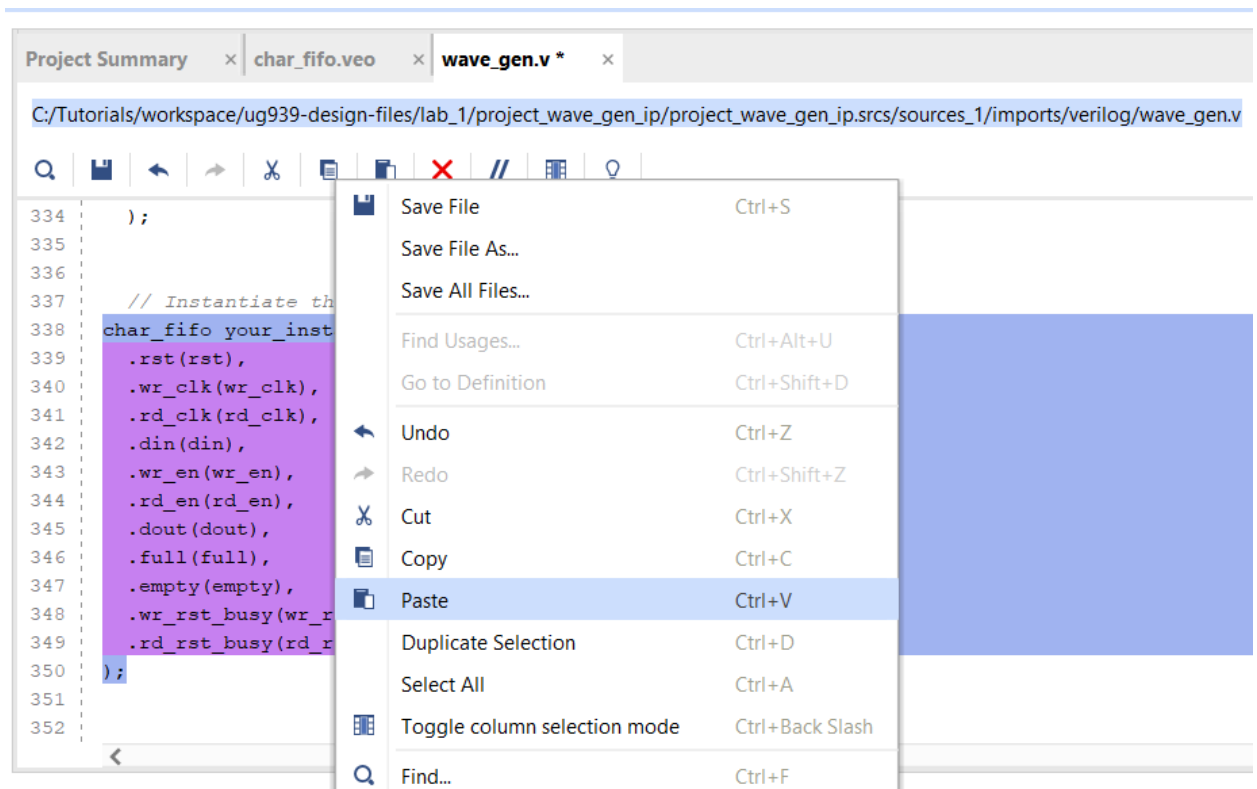
3. Open this file for editing from the Hierarchy tab of the Sources view, by double-clicking `wave_gen.v`. The following figure shows the file.





4. Go to line 337, which contains a comment stating the Character FIFO should be instantiated at this point.
5. Paste the template code into the file as shown in the following figure.

Because it is only a template for the module, you need to do some local editing to make the module work in your design.



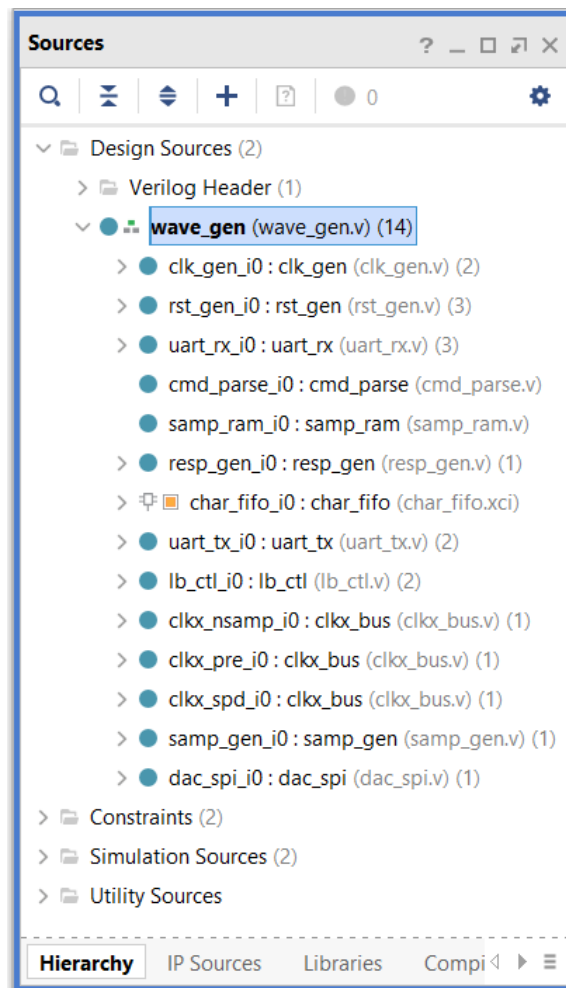
- a. In line 338 (approximately) above where you pasted the instantiation, change the module name from `your_instance_name` to `char_fifo_i0`.

- b. Change the wire names as follows, to connect the ports of the module into the design:

```
char_fifo char_fifo_i0 (
.rst(rst_i), // input wire rst
.wr_clk(clk_rx), // input wire wr_clk
.rd_clk(clk_tx), // input wire rd_clk
.din(char_fifo_din), // input wire [7 : 0] din
.wr_en(char_fifo_wr_en), // input wire wr_en
.rd_en(char_fifo_rd_en), // input wire rd_en
.dout(char_fifo_dout), // output wire [7 : 0] dout
.full(char_fifo_full), // output wire full
.empty(char_fifo_empty), // output wire empty
.wr_rst_busy(wr_rst_busy), // output wire wr_rst_busy
.rd_rst_busy(rd_rst_busy) // output wire rd_rst_busy
);
```

6. In the Text Editor menu, click the **Save File** button (📁) to save the changes to the `wave_gen.v` file.

Notice that the Hierarchy, Libraries, and Compile Order tabs update to indicate that the IP is instantiated into the design, as seen in the following figure.



## Step 4: Synthesize the Design

With the customized IP integrated into your design, you can now synthesize the design.

Xilinx IP in the IP catalog are delivered as RTL source. This provides the benefit of being able to perform behavioral simulation, which is faster than netlist-based simulation. However, synthesizing each IP along with the overall design, with every design iteration, can add significant synthesis time to the project development.

The design checkpoint (DCP) file delivered with the IP core output products, and generated by the out-of-context (OOC) synthesis run, eliminates the need to re-synthesize the core over multiple iterations.

The default behavior of the Vivado Design Suite is to generate the necessary output products, including the DCP file, when you create an IP customization.

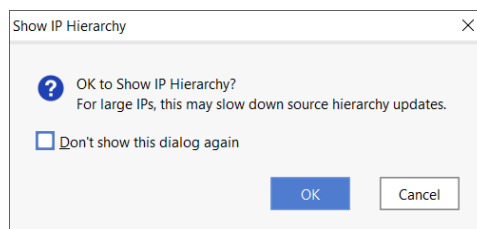
You can disable generation of the DCP file, as well as defer generating output products until later.

1. In the Hierarchy tab of the Sources window, look at the `clk_core` IP customization instantiated under the top-level of the design, `wave_gen/clk_gen_i0`.

Notice that the `clk_core` IP cannot be expanded with the '>' icon like you can expand the `char_fifo` IP. This indicates that no output products exist for the `clk_core` when it was instanced in the project.

2. In the Sources window > IP Sources view, click the Plus (>) icon next to the `char_fifo` IP.

The Vivado tool displays the Show IP Hierarchy dialog box, (following figure), warning you that expanding the hierarchy of very large IP cores can add significant delay to updating the hierarchy in the Sources window.



**Note:** If you do not get the same results, ensure that you are in the Source Window > IP Sources view.

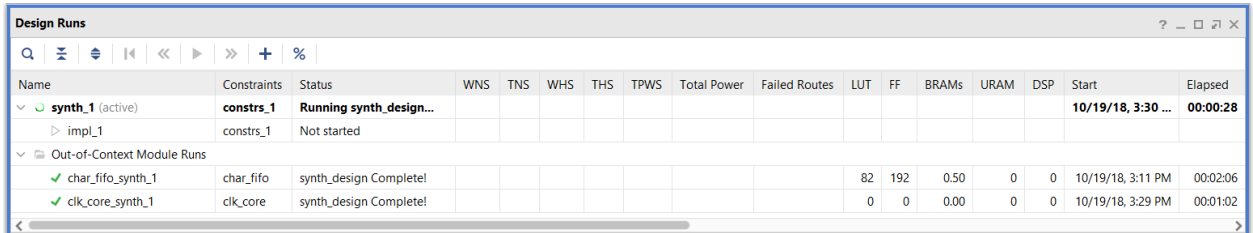
3. Click **Cancel** to close the Show IP Hierarchy dialog box.

If you skip generation of output products when the IP is customized, the Vivado® Design Suite automatically generates the required output products at the point in the design flow they become necessary, such as during synthesis or simulation.

By default, a DCP file for an IP core is created unless you disable this in the Generate Output Products dialog box. See the *Vivado Design Suite User Guide: Designing with IP (UG896)* for more information. If DCP file generation is disabled, the IP RTL synthesizes along with the top-level design.

- In the Flow Navigator, click the **Run Synthesis**  **Run Synthesis** button.

The Vivado tool automatically creates a new OOC module synthesis run for the `clk_core`, and launches that synthesis run as seen in the following figure.

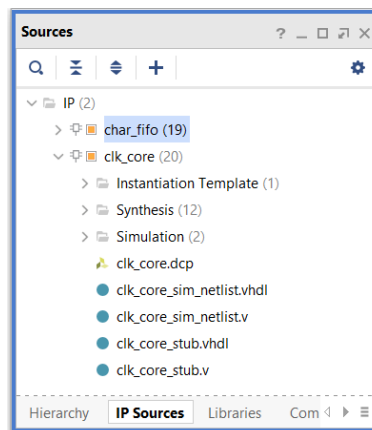


Name	Constraints	Status	WNS	TNS	WHS	THS	TPWS	Total Power	Failed Routes	LUT	FF	BRAMs	URAM	DSP	Start	Elapsed
synth_1 (active)	constrs_1	Running synth_design...													10/19/18, 3:30 ...	00:00:28
impl_1	constrs_1	Not started														
Out-of-Context Module Runs																
char_fifo_synth_1	char_fifo	synth_design Complete!								82	192	0.50	0	0	10/19/18, 3:11 PM	00:02:06
clk_core_synth_1	clk_core	synth_design Complete!								0	0	0.00	0	0	10/19/18, 3:29 PM	00:01:02

This synthesis run creates the DCP file for the IP customization.

- When the `clk_core` synthesis run is finished, click **OK**.

You can examine the contents of the IP Sources tab of the Sources window. You will see the output products generated by the Vivado tools for the IP, (following figure).




With the required output products for the `clk_core` created, the Vivado synthesis tool runs on the top-level of the design, (see the following figure).

When the top-level of the design is synthesizing, Vivado synthesis infers a black box for the FIFO generator IP (`char_fifo`) and the Clock generator IP (`clk_core`) in the design.

Name	Constraints	Status	WNS	TNS	WHS	THS	TPWS	Total Power	Failed Routes	LUT	FF	BRAMs	URAM	DSP	Start	Elapsed
✓ synth_1 (active)	constrs_1	synth_design Complete!								650	4...	0.50	0	0	10/19/18, 3:30 ...	00:02:08
> impl_1	constrs_1	Not started														
Out-of-Context Module Runs																
✓ char_fifo_synth_1	char_fifo	synth_design Complete!								82	192	0.50	0	0	10/19/18, 3:11 PM	00:02:06
✓ clk_core_synth_1	clk_core	synth_design Complete!								0	0	0.00	0	0	10/19/18, 3:29 PM	00:01:02

- When the Synthesis Completed dialog box opens, select the **View Reports** option, and click **OK**.

This opens the Reports window at the bottom of the Vivado IDE.

- Switch to the Reports window, and select **Synthesis** → **Synth Design** → **synthesis\_report**.
- In the Log window, click the **Find** button  to search for “blackbox.”
- Click **Find Next** until you come to the following section, which summarizes the black boxes found in the current design, then click **Next**. The following figure is a snippet of the "Report BlackBoxes" report.

```

+-----+-----+-----+
| BlackBox name | Instances |
+-----+-----+-----+
| 1 | char_fifo | 1 |
| 2 | clk_fifo | 1 |
+-----+-----+-----+

```

X23129-081919

- Review the report.

You can also review the project IP runs folder for the results of the out-of-context synthesis runs:

```
<Extract_Dir>/lab_1/project_wave_gen_ip/project_wave_gen_ip.runs
```

You can use the IP customizations created in other projects by adding the XCI file as a source. All the output products for the IP, including the DCP, are used automatically. If you change the part, you must update the IP and regenerate the output products.

## Conclusion

This concludes Lab 1. You have successfully created a FIFO Generator IP customization, and instanced it in a design. Close the project and exit the Vivado® tool, or continue and implement the design to explore further. In this Lab, you learned how to:

- Open the Vivado IDE.
- Select and customize an IP from the IP catalog.

- Instantiate the customized IP into an HDL design.
- Use some details of the output products required to support the IP in the design flow.

You can add and manage IP in a design interactively within the Vivado IDE, or using Tcl scripting.

# Creating and Managing Reusable IP

---

## Introduction

To simplify revision control, and to support the use of customized IP across multiple projects and designs, you can manage and store the customized IP in a repository, separate from any design projects in which they are used. The IP customization file (XCI), and the output products for synthesis, simulation, and other outputs, are contained together in a unique directory.

You can reference these IP customizations in new projects and designs, to simulate, synthesize, and implement the IP as part of the design. Having all the generated output products available also preserves that customized version of the IP for use in a future release of the Vivado® Design Suite, even if the IP is updated in the Xilinx® IP catalog. See the *Vivado Design Suite User Guide: Designing with IP* (UG896) for more information on managing IP.

In this lab, you create and verify an IP customization in an IP repository using the Manage IP flow in the Vivado Design Suite. This flow lets you browse the IP catalog to create and manage IP customizations for use in either a Project or a Non-Project design flow.

You can create a repository of the customized IP for use in your design(s), managed and maintained outside of a Vivado Project. A special IP project is created at the location specified for the Manage IP flow. This special project facilitates the creation of a synthesis design checkpoint (DCP) and structural simulation models for the IP.

When using an IP customization, in a project or non-project flow, all output products, including a DCP if present, are used in the design flow. The use of IP synthesis DCP file speeds synthesis of the top-level design because the IP have been pre-synthesized. In addition, a stub file is produced for use in third party synthesis tools to infer a black box for IP.

---

## Step 1: Start a Manage IP session

1. Change to the directory where the lab materials are stored:

```
cd <extract_dir>/lab_2
```

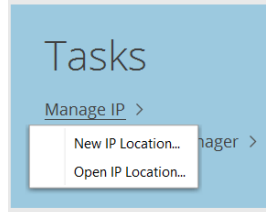
2. Launch the Vivado® IDE:

- a. On Linux, by typing `vivado`.
- b. On Windows, launch the Vivado Design Suite IDE by pressing **Start → All Programs → Xilinx Design Tools → Vivado 2022.1**.

As an alternative, click the Vivado 2022.1 Desktop icon to start the Vivado IDE.

The Vivado IDE Getting Started page displays with links to open or create projects, and to view documentation.

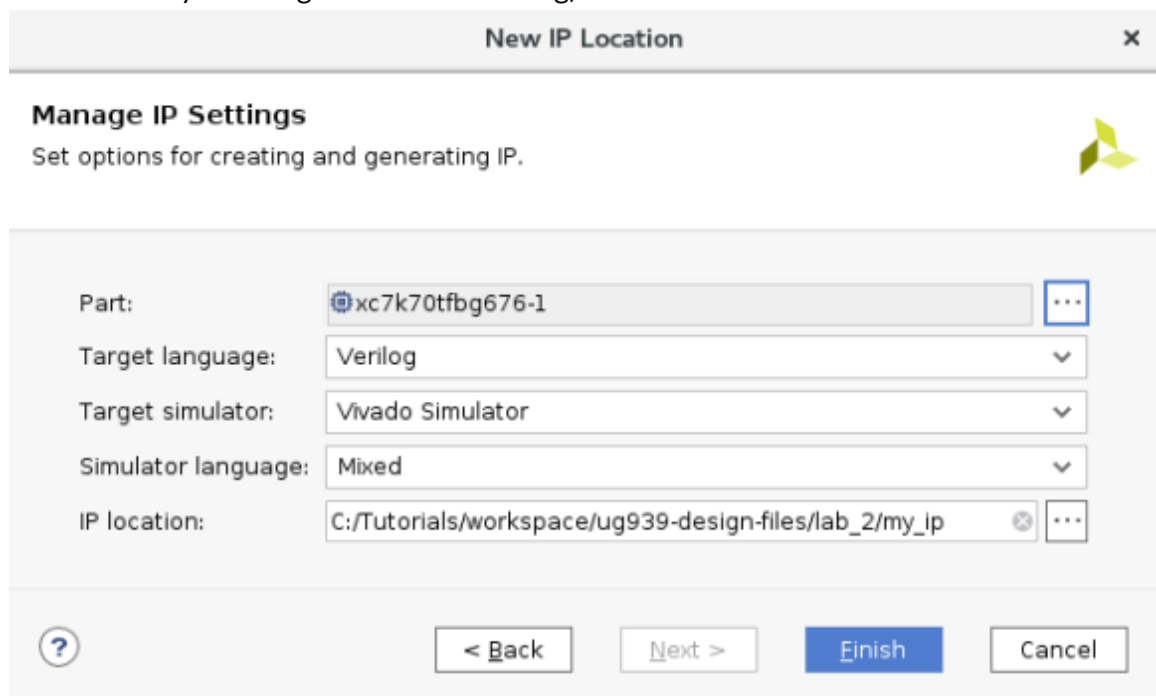
3. On the Getting Started page, click the **Manage IP** link, as shown in the following figure.



In the menu that displays, you can click one of the following:

- Click **New IP Location** to specify a New IP location for a new Manage IP project and generated output products.
- Click **Open IP Location** to open an existing IP location for prior Manage IP projects.


4. Select **New IP Location**.
5. Click **Next** to move to the Manage IP Settings dialog box shown in the following figure (this shows to what you change the default setting).





6. Configure the dialog box settings as follows:

- **Part:** xc7k70tfbg676-1

**Note:** Use the browse button to select the specified target part. 

- **Target Language:** Verilog
- **Target Simulator:** Vivado Simulator
- **Simulator Language:** Mixed
- **IP location:** <Extract\_Dir>/lab\_2/my\_ip

7. To create the folder click **OK** to create the dir.

8. Press **Finish**.

The IP catalog displays in an IP Project, which is a simple interface for the creation and management of IP customizations.

---

## Step 2: Customize the FIFO Generator

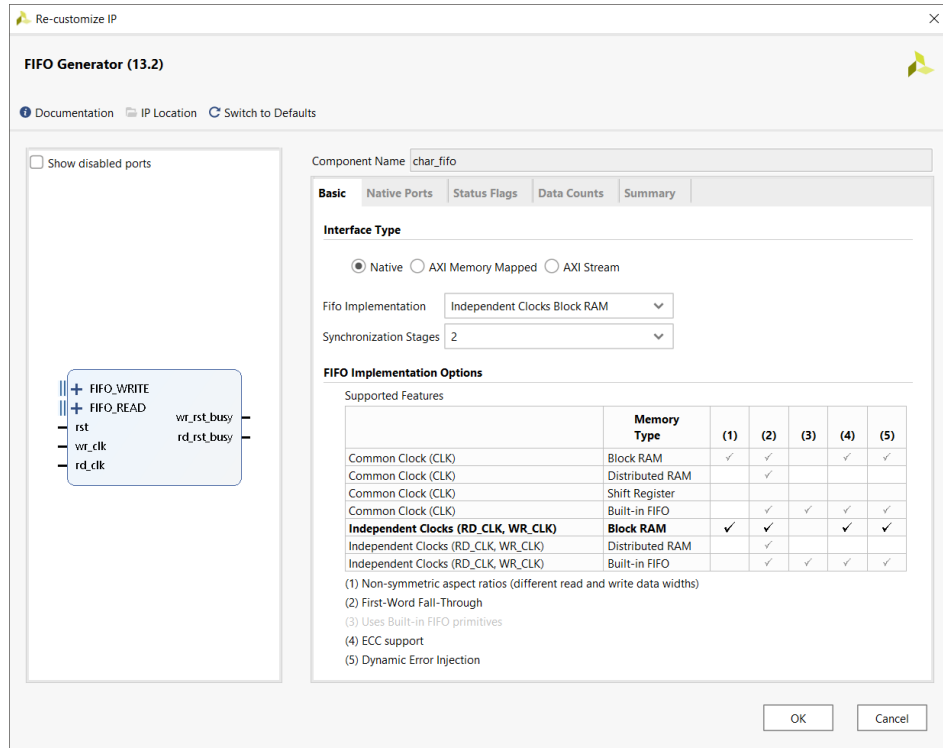
You can work with the IP catalog by searching with a keyword or browsing through the categories.



**TIP:** For a complete description of the Customize IP dialog box, and its use, see [Lab 1: Customizing the FIFO Generator](#). This Lab assumes that you have previously completed Lab 1, and are familiar with the concepts covered in that lab.

1. In the IP catalog search bar, type: `fifo`.
2. Double-click the **FIFO Generator** from the Memories & Storage Elements group.

The Customize IP dialog box opens, as shown in the following figure (this is not the original setting; it shows the change that you make to the customization).



3. At the top of the Customize IP dialog box, change the Component Name to `char_fifo`.
4. In the Basic tab:
  - Select the default Interface Type of `Native`.
  - From the Fifo Implementation menu, set `Independent Clocks Block RAM`.
5. In the Native Ports tab:
  - Set the Read Mode to `First Word Fall Through`.
  - Set the Write Width to be `8 bits`.
  - Click the **Read Width** field to adjust it automatically to `8 bits` as well.
6. Select the **Summary** tab.

The Summary page displays a summary of all the options selected as well as listing resources used for this configuration. The summary for the FIFO Generator core should look like the following figure. For this configuration you will see you are using one 18K BRAM.

Basic	Native Ports	Status Flags	Data Counts	Summary
<b>WARNING</b> : The use of Asynchronous Reset can lead to BRAM data corruption(AR 42571). It is recommended to Enable Safety Circuit				
Block RAM resource(s) (18K BRAMs): 1				
Block RAM resource(s) (36K BRAMs): 0				
Clocking Scheme		Independent Clocks		
Memory Type		Block RAM		
Model Generated		Behavioral Model		
Write Width		8		
Write Depth		1025		
Read Width		8		
Read Depth		1025		
Almost Full/Empty Flags		Not Selected/Not Selected		
Programmable Full/Empty Flags		Not Selected/Not Selected		
Data Count Outputs		Not Selected		
Handshaking		Not Selected		
Read Mode / Reset		First-word Fall-through / Asynchronous		
Read Latency (From Rising Edge of Read Clock)		0		

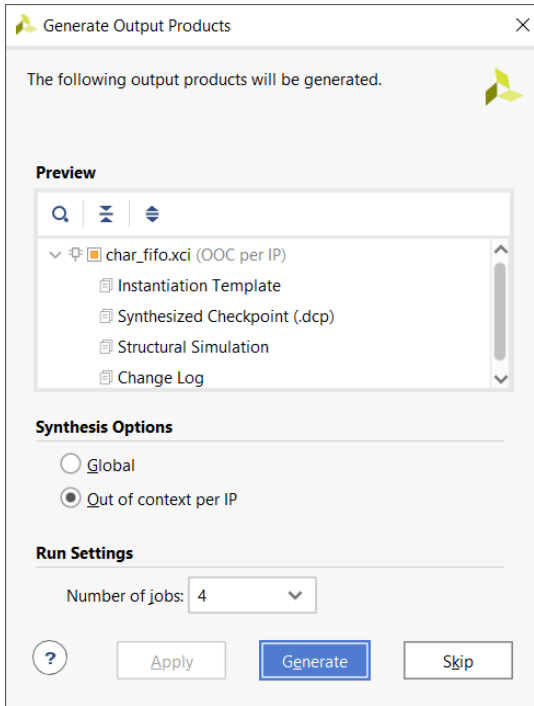
- Verify that the information is correct as shown, and click **OK** to generate the customized IP core for use in your design.

The Generate Output Products popup appears, as seen in the following figure. When creating a repository of customized IP using the Managed IP flow, generate all output products, including the design checkpoint (DCP), for each IP. A specific release of the Vivado tools only supports a single version of an IP core. You cannot re-customize or generate outputs for a prior version of IP in the Vivado Design Suite. Instead, you would need to update the IP to the latest version if you have not preserved the needed output products.



**IMPORTANT!** Only one version of an IP is supported in a given release of the Vivado tool. To use older versions of an IP, ALL output products must be available in your custom IP repository.

- Click the **Generate** button to create the output products, as shown in the following figure.



9. When prompted, click **OK**.

To reduce synthesis runtime for a design using one or more customized IP cores, the IP are pre-synthesized as a standalone module by default, with the netlist saved in a synthesized checkpoint file (DCP).

During synthesis of the overall design, the Vivado synthesis infers a black box for the IP.

During implementation, the netlists from the DCP files are linked into the design. This flow is scriptable in Tcl for both Project and Non-Project Mode.

To create the synthesized checkpoint file, the Vivado IDE creates and launches a corresponding out-of-context module run.

This run automatically launches, and when synthesis completes, the `char_fifo` directory contains the following additional files:

- `char_fifo.dcp`: The synthesis design checkpoint, which contains a netlist for the IP.
- `char_fifo_stub.v`: A Verilog port module for use with third-party synthesis tools to infer a black box for the IP. The stub contains directives for Synplify Pro to not insert IO buffers, this may require editing for other third-party synthesis tools.
- `char_fifo netlist files, sim_netlist.v, and sim_netlist.vhd`: A Verilog and VHDL netlist for functional simulation of the IP core.

If you are using a third-party synthesis tool for the design, a Verilog stub file with the port declarations, or a VHDL component declaration, is required to infer the black box.

The Vivado Design Suite automatically creates this file along with the synthesis design checkpoint (DCP) when generating the output products.

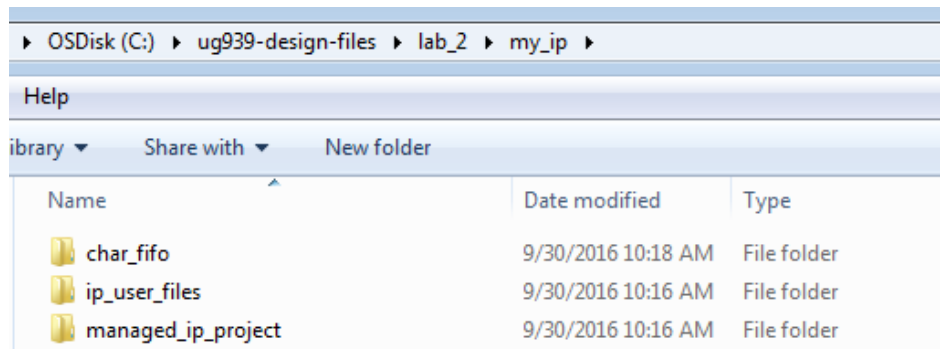
Customized IPs can be referenced using the XCI file from both Project and Non-Project Mode.

In a project-based design, Xilinx recommends that you do not copy sources into the local project structure, but rather reference them from your custom IP repository. For more information on Project Mode and Non-Project Mode, see the *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#)).

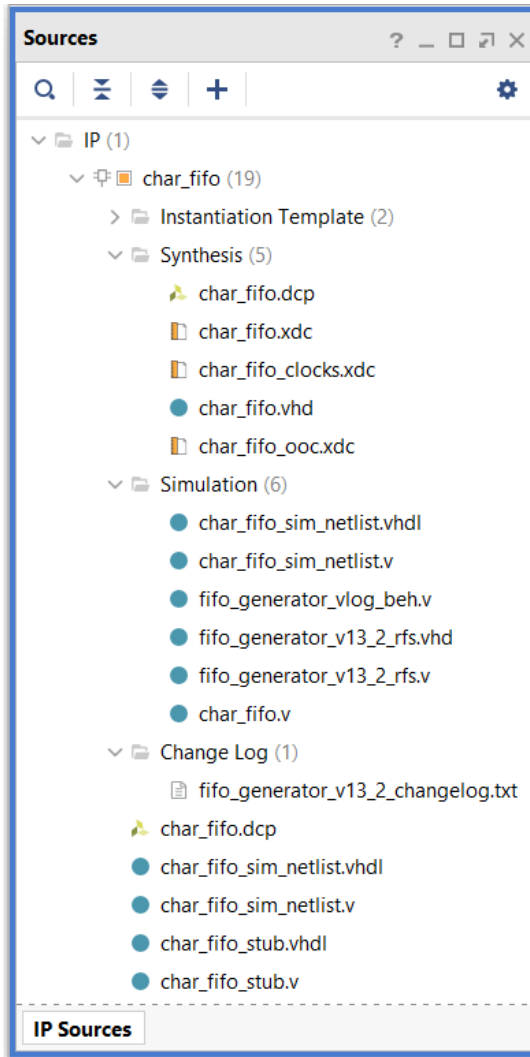
10. Examine the `<extract_dir>/lab_2/my_ip` location.

Notice that three directories are created, as shown in the following figure:

- The IP customization (`char_fifo`) containing the XCI file, which has all the customization information for the IP, and all the output products generated.
- The IP Project (`managed_ip_project`) which you created in this lab to customize the `char_fifo` IP and manage the output products, including the post synthesis DCP.
- The support files for IP (`ip_user_files`) which contains simulation scripts for all supported simulators, copies of the instantiation templates, stub files for use with third party synthesis tools, and static RTL files used for simulation. All these files with the exception of the simulation scripts are copied from the IP customization directory (in this case `char_fifo`). See the *Vivado Design Suite User Guide: Designing with IP* ([UG896](#)) for more details on the `ip_user_files` directory.



11. Examine the Sources window as shown in the following figure, to see the generated output products in the Manage IP project.



The generated output products for the `char_fifo` core include the instantiation template, synthesis files, simulation files, synthesized design checkpoint, functional simulation netlists as well as stub files for use with third-party synthesis tools.

**Note:** The FIFO customization created in this lab is the same as the one from Lab 1: Customizing the FIFO Generator. If you want to add existing IP into a Managed IP project, select **File** → **Add Existing IP**.

### Related Information

[Step 2: Customize the FIFO Generator](#)

---

## Step 3: Customize the Clocking Wizard

Now, you create a customization for the Clocking Wizard IP.

1. Follow the same processes you used for finding the FIFO Generator IP in the IP catalog to find and launch a Clocking Wizard IP customization.
2. Change the Component Name to `clk_core`.
3. In the Clocking Options tab, leave all the settings on the default, except for the Input Clock Information as shown in following figure.
  - **Primary Clock Input Frequency:** 200
  - **Source:** Differential clock capable pin

Input Clock	Port Name	Input Frequency(MHz)	Jitter Options	Input Jitter	Source	
<input checked="" type="checkbox"/> Primary	clk_in1	200.000	10.000 - 800.000	UI	0.010	Differential clock capable pin
<input type="checkbox"/> Secondary	clk_in2	100.000	120.000 - 240.000		0.010	Single ended clock capabl...

4. In the Output Clocks tab, change the Port Name, as well as Output Freq Requested as follows:
  - **clk\_out1:** clk\_rx, 200
  - **clk\_out2:** clk\_tx, 166.66667

You should end up with the configuration shown in the following figure.

Output Clock	Port Name	Output Freq (MHz)	Phase (degrees)	Duty Cycle (%)	Drives	Use Fine PS		
		Requested	Actual	Requested	Actual			
<input checked="" type="checkbox"/> clk_out1	clk_rx	200.000	200.000	0.000	50.000	50.0	BUFG	<input type="checkbox"/>
<input checked="" type="checkbox"/> clk_out2	clk_tx	166.667	166.667	0.000	50.000	50.0	BUFG	<input type="checkbox"/>

5. Click **OK** to bring up the Generate Output Product window.
6. Click **Generate**, and then click **OK**.
7. Vivado® creates and launches a design run for the Clocking Wizard IP as shown in the following figure.

Name	Constraints	Status	Progress	Incremental	Run Strategy	Report Strategy	Part	Description
Out-of-Context Module Runs								
char_fifo_synth_1	char_fifo	synth_design Complete!	100%	Off	Vivado Synthesis Defaults (Vivado Synthesis 2018)	Vivado Synthesis Default Reports (Vivado Synthesis 2018)	xc7k0tbg676-1	Vivado Synthesis Defaults
clk_core_synth_1	clk_core	synth_design Complete!	100%	Off	Vivado Synthesis Defaults (Vivado Synthesis 2018)	Vivado Synthesis Default Reports (Vivado Synthesis 2018)	xc7k0tbg676-1	Vivado Synthesis Defaults

8. Verify the output products and directory created for the Clocking Wizard IP customization (`clk_core`) you just created, just as you did with the customized FIFO Generator IP.

## Step 4: Use Third-Party Simulators

The purpose of the Managed IP project is to create and manage IP customizations; there is no support for directly simulating IP in a Managed IP project. Customized IP can be instantiated into a standard design project for simulation, in either Project or Non-Project Mode.

Xilinx® IP delivered in the Vivado® Design Suite are encrypted using industry standard IEEE P1735 encryption. Supported simulator use this encryption standard, and can run behavioral simulation.

The Vivado Design Suite includes the Vivado simulator, for mixed language simulation, as well as direct support for Mentor Graphics® Questa Advanced Simulator, ModelSim Simulator, Cadence® Incisive Enterprise Simulator (IES), Synopsys® Verilog Compiler Simulator (VCS), and Aldec®.

By default, Vivado creates a directory called `ip_user_files` in the same directory as the IP and Managed IP directories. This directory contains scripts for simulation with all supported simulators for all the IP created. You can use these scripts as a starting point for simulating with the IP. The scripts will reference all the files needed for simulating the IP.

You can use the following Tcl command to generate a Tcl script for the target simulator:

```
export_simulation -simulator <simulator>
```

See the *Vivado Design Suite User Guide: Logic Simulation (UG900)* for more information on using third-party simulators, including creating scripts for running simulations outside of the Vivado Design Suite.

## Get the Required Simulation Files

As an alternative to the `export_simulation` command, you can manually gather the required files to support third-party simulation.

**Note:** If there is a need to use the VIP (Verification IP) on IP, use the Tcl command: `set_property CONFIG.<AXI interface> [get_ips <ip>]` and additional simulation files will be generated. See *Vivado Design Suite User Guide: Designing with IP (UG896)* for more information.

1. To get all files that an IP delivers for simulation, use the `get_files` Tcl command. In the Tcl Console, type the following command:

```
get_files -compile_order sources -used_in simulation -of_objects  
[get_ips char_fifo]
```

Where:

- `-used_in`: Lets you specify files that are marked for use in simulation, or marked for use in synthesis.
- `-of_object`: Lets you extract files that are associated with the specified IP customization file.

This produces a list of file names, including the full path, required to simulate the IP.



In this case, the list includes:

```
.../char_fifo/fifo_generator_v13_2/simulation/fifo_generator_vlog_beh.v
.../char_fifo/fifo_generator_v13_2/hdl/fifo_generator_v13_2_rfs.vhd
.../char_fifo/fifo_generator_v13_2/hdl/fifo_generator_v13_2_rfs.v
.../char_fifo/sim/char_fifo.v
```

Each simulation file has a `LIBRARY` property that you can query. For VHDL files, the library associated with each file is required for simulation.

2. To extract the `LIBRARY` property, type the following Tcl command:

```
get_property LIBRARY [get_files char_fifo.v]
```

This returns the `xil_defaultlib` library.

3. Use the following Tcl script to print out each file used for simulation, including the path, and its associated library:

```
# Get the list of files required for simulation
set ip_files [get_files -compile_order sources -used_in simulation -
of_objects
[get_ips <IP name>]]
# For each of these files, get the library information
foreach file $ip_files {
puts "[get_property LIBRARY $file] $file"
}
```

In the preceding script, replace `<ip_name>` with the name of the customized IP to extract files from. In this case, you would use `char_fifo`.

## Structural Netlists for Simulation

Depending on which Simulator Language you select during the Manage IP project creation (see the following figure) the results of the `get_files` commands above differs.

With some IP, you might not be able to do behavioral simulation if you did not specify support for a Mixed simulator, and instead selected Verilog or VHDL.

You would need to run simulations using a structural netlist, which the Vivado Design Suite produces automatically when a synthesized design checkpoint is available.

If the IP can deliver behavioral simulation files based upon your selected simulator language, when generating the output products you see Behavioral Simulation listed as an output product.

However, if the IP does not deliver simulation files for the selected simulator language you see Structural Simulation as shown in the following figure.



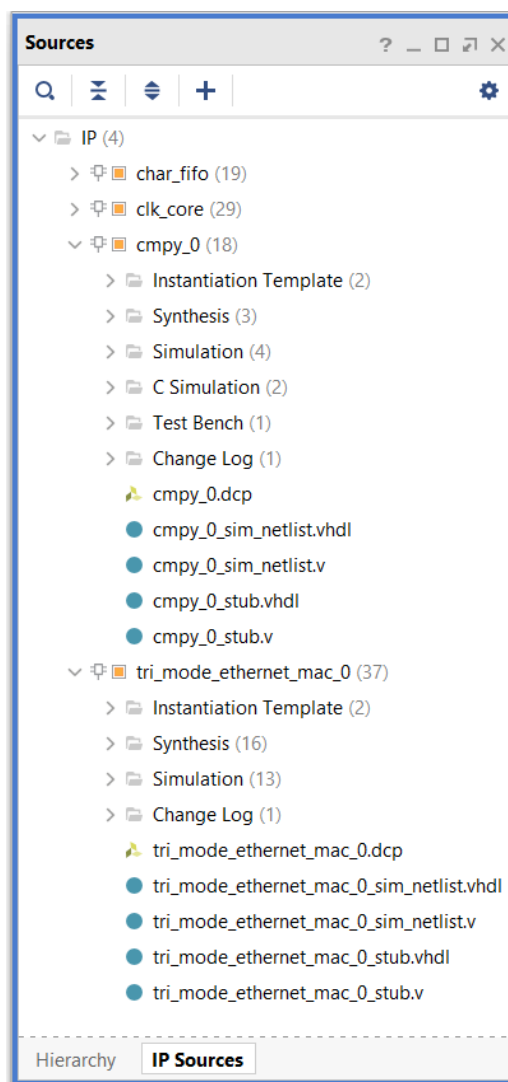
**TIP:** Structural simulation output products, for both Verilog and VHDL, are always created when a synthesized design checkpoint is produced. Querying the simulation files with `get_files` will vary depending on whether behavioral simulation was possible with the selected simulator language setting.

## Step 5: Add Additional IP

At this point in the tutorial, you can repeat some of the earlier steps to add some additional IP cores to the Managed IP project, and customize them as well. In the following figure, Tri-Mode Ethernet MAC and Complex Multiplier IP customizations have been added to the project.

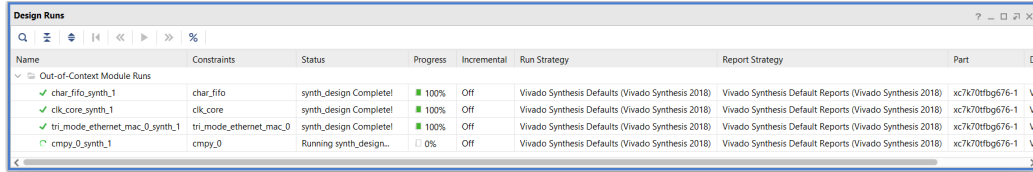
1. Explore the IP catalog and create customizations for a few additional IP cores.
2. Generate output products for the additional cores.

In the IP Sources tab of the Sources window, you see the various output products that have been generated for the additional cores.



When you create an IP customization, the Vivado® Design Suite adds Instantiation Templates to the project, even if you choose to skip generating other output products. The Instantiation Templates are the minimum output product.

By default, the Vivado Design Suite creates a synthesized design checkpoint for each IP customization added to the project, adding Out-of-Context module runs for each of the IP cores. You can view the progress of these synthesis runs in the Design Run tab, as shown in the following figure.



## Step 6: Use Third-Party Synthesis Tools

Xilinx® IP delivered with the Vivado® Design Suite supports synthesis using the Vivado synthesis tool only. The user logic can be synthesized using a supported third-party synthesis tools, such as Synopsys® Synplify Pro or Mentor Graphics® Precision.

The Vivado Design Suite generates a Verilog/VHDL stub file for each IP customization that can be used by third-party synthesis tools to infer a black box for the Vivado IP.

Vivado automatically creates the stub file if a synthesized DCP is generated for the IP, which is the default behavior of the tool. Because it is important that the third-party synthesis tool does not insert I/O buffers for ports that are connected to the Vivado IP, the `<ip_name>.stub.v` contains synthesis directives to prevent I/O buffer insertion.

This lab has you use a Manage IP project to create and customize two IP used in the sample design.

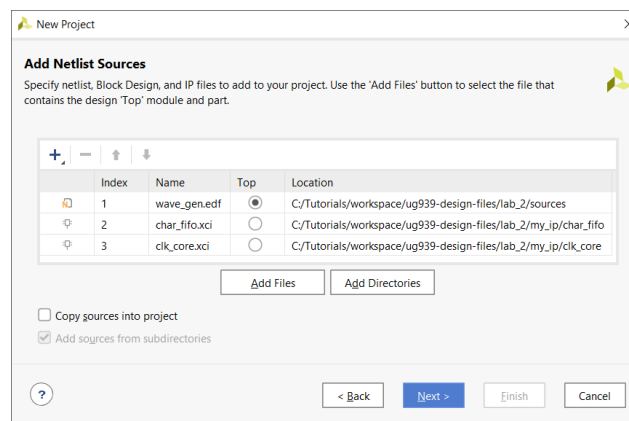
- To avoid requiring access to a third-party synthesis tool you are provided an EDIF produced using Synplify Pro.
- You are also provided two Xilinx Design Constraints (XDC) files for constraining the design during implementation.
- An optional stage is provided on how to perform simulations with the IP.
- You create a netlist project and combine the IP with the netlist produced by the third-party synthesis tool.
- A script is provided to demonstrate how you would use a non-project flow to bring the sources together for implementation.

## Step 7: Create a Netlist Project

The process to implement the design consists of bringing the netlist from the third-party synthesis tool into Vivado®, the two IP customizations created in the Manage IP project, and top-level constraints. The EDIF netlist and the top-level XDC files are provided for you.

1. If open, close the Manage IP project or launch Vivado.
2. From the Getting Started page of Vivado, select **Create Project** and then select **Next**.
3. In the Project Name dialog box, provide a project name and specify a location:
  - **Project name:** `wave_gen_netlist`
  - **Project location:** `<extract_dir>/lab_2/`
4. Leave the Create project subdirectory checked, and click **Next**.
5. Set the Project type to be a `Post-synthesis` project and leave the Do not specify sources at this time box unchecked.
6. In the Add Netlist Sources window, browse for and select the following files:
  - `<extract_dir>/lab_2/sources`, and select `wave_gen.edf`
  - `<extract_dir>/lab_2/my_ip/char_fifo` (or wherever you put the Manage IP project) and select `char_fifo.xci`
  - `<extract_dir>/lab_2/my_ip/clk_core` and select `clk_core.xci`

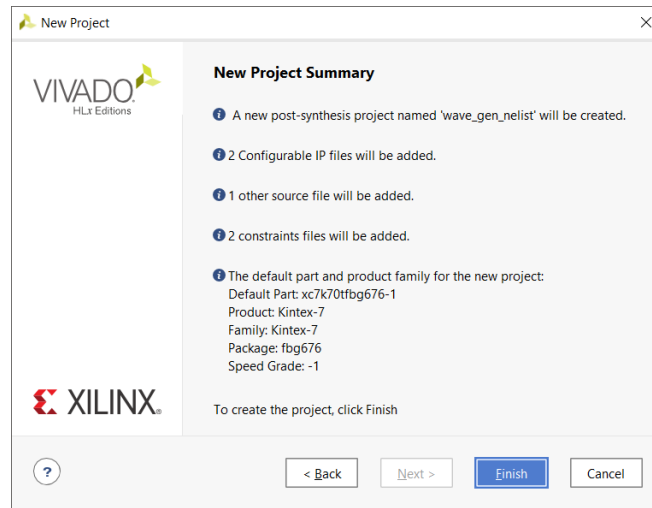
You can either remotely reference the source files or copy them into the netlist project.
7. Ensure that the `wave_gen.edf` is set as top, as shown in the following figure, then click **Next** to continue.



8. In the Add Constraints window, browse to the `<Extract_Dir>/lab_2/sources` area and select the `wave_gen_timing.xdc` and `wave_gen_pins.xdc` files.
  - a. Again, either reference them remotely or add them to your project.
  - b. Click **Next** when done selecting the two files.

- In the Default Part window, select the **xc7k70tfbg676-1** and click **Next**.

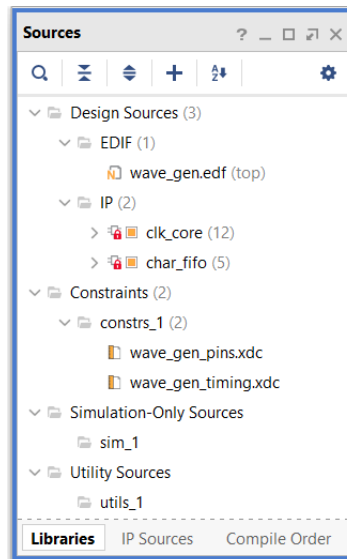
The New Project Summary window opens, as shown in the following figure.



- Click **Finish**.

The newly created netlist project opens.

- Expand the sources window as shown in the following figure, to see the netlist, IP, and constraints that comprise the project.

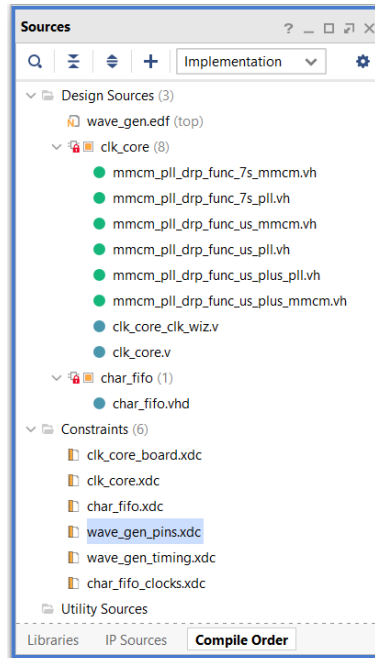


In a netlist project you can check the status of the IP that are present.

- Select **Report** → **Report IP Status**, and see that the IP are locked. This is because changing IP in a netlist design requires you to modify them from their original project or a Managed IP project.

Even if the IP is no longer current, because you have all the output products, you can proceed through implementation.

13. Select the **Compile Order** tab and select **Implementation** to see the order in which the design source files and constraints are processed (see the following figure).



14. In the Tcl Console, type: `report_compile_order`

You see three sections:

- **Source compile order for implementation:** Lists the EDIF and DCP sources that are used during implementation. The only thing used from the DCPs for the IP are the EDIF netlists.
- **Source compile order for simulation:** RTL sources for the IP set to be used during implementation. There are no top-level simulation files for this project so simulation cannot be performed at this time on the project.
- **Constraint evaluation order for implementation:** IP XDC output product files as well as the two top-level constraint files. The constraints in the IP DCP files are not used.

15. Select **Flow Navigator** → **Implementation** → **Run Implementation**.

The Launch Runs dialog box appears.

16. Click **OK**.

17. After implementation completes, select **Report Timing Summary** in the Flow Navigator to open the implemented design and report timing. View and familiarize yourself with the feature.



**IMPORTANT!** *The design does not meet Timing at this point of the flow.*

You could also script this flow.

See the `.jou` file for the script commands for a project flow. Continue to the next section for information on how to create a non-project based flow script.

## Create a Non-Project Third-Party Script

You can create a non-project script to read a top-level design netlist, the IP, the design constraint files, and then implement the design.

Modify this script accordingly to point to your source file locations.

```
# Set the target part to be used
set_part xc7k70tfbg676-1
# Read top-level EDIF netlist produced by the third party synthesis tool
read_edif ./sources/wave_gen.edf
# Read the two IP
# Read the IP and use any required output products generated (XDC, DCP,
etc.)
read_ip ./my_ip/char_fifo/char_fifo.xci
read_ip ./my_ip/clk_core/clk_core.xci
# read top-level constraints
read_xdc ./sources/wave_gen_timing.xdc
read_xdc ./sources/wave_gen_pins.xdc
# Link the netlists to build the in-memory design database
link_design -top wave_gen
# Implement the design in Non-Project Mode
opt_design
place_design
route_design
# write out an implemented design checkpoint
write_checkpoint -force wave_gen_post_route.dcp
```

All black boxes for that IP customization are replaced with the netlist from the synthesized DCP when the design is linked, and the design constraints are applied to each instance.

---

## Conclusion

This concludes Lab 2. You can continue examining the design, and create additional reports, or exit the Vivado® Design Suite.

In this lab, you learned to:

- Use the Manage IP flow to browse the IP catalog and create IP customizations to store in an IP repository for later reuse. It is a convenient method for creating custom IP to share between users and projects and to manage the IP under revision control.
- From the Manage IP Flow you can use the customized IP in a script (Project or Non-Project Mode), from an RTL project, or for inferring a black box for use with a third-party synthesis tool.
- For convenience, Vivado also generates structural netlist Verilog/VHDL files for simulation.
- Perform behavioral simulation with an IP, use the IP in a project, or generate scripts for the Vivado simulator or ModelSim Simulator/Quarta Advanced Simulator for standalone simulation.

- Use other third party simulators, such as Cadence® Incisive Enterprise Simulator (IES) or Synopsys VCS MX, and Aldec where you either query both the HDL files required for simulation, and the libraries with which the files are associated, or use the `export_simulation` Tcl command.



# Scripting the Project Mode

---

## Introduction

In this lab, you write a Project Mode Tcl script, creating a new project and adding source RTL and customized Xilinx® IP.

When working in Project Mode, the Vivado® IDE creates a directory structure on disk in order to manage design source files, run results, and track project status. A runs infrastructure manages the automated synthesis and implementation process and tracks the run status.

In [Lab 4: Scripting the Non-Project Mode](#), you explore creating a Tcl script to run the Vivado tools in Non-Project Mode, which does not rely on project files and managed source files. For more information on Project Mode and Non-Project Mode, see the *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#)).

The best way to become familiar with scripting the Project Mode design flow is by first running the design flow interactively in the Vivado IDE, and then referencing the journal file, `vivado.jou`, that the Vivado® Design Suite creates. By editing the journal file, you can create a Tcl script that encompasses the entire Project Mode design flow. In this lab, you will build the script by hand. For more information on writing and using Tcl scripts, see the *Vivado Design Suite User Guide: Using Tcl Scripting* ([UG894](#)).



**CAUTION!** When copying Tcl commands and script examples from this Tutorial document and pasting them into the Vivado Design Suite Tcl shell, or Tcl Console, the dash “-” character can sometimes be converted to an em-dash “–” which will result in errors when the commands are run.

---

---

## Step 1: Create a Project

1. Invoke a text editor of your choice, such as Emacs, VI, or Notepad; or launch the Text Editor from within the Vivado® IDE.
2. Save a new file called `project_run.tcl` in `<Extract_Dir>/lab_3`.

You start your script by creating a new project using the `create_project` command.

This results in a new project directory created on disk. However, you want to make sure that the project is created in the right location to find source files referenced by the script.

3. Add the following line to your Tcl script to change to the appropriate directory for this lab:

```
cd <extract_dir>/lab_3
```

Now you are ready to create your project.

4. Add the following Tcl command to your `project_run.tcl` script:

```
create_project -force -part xc7k70tfbg484-3 my_project my_project
```

This command creates a directory called `my_project`, and adds a project named `my_project` to that directory. The directory and project are created at the location where the script is run. You can specify a different directory name with the `-dir` option of the `create_project` command.

All the reports and log files, design runs, project state, and any source files that you import into the project are stored in the project directory..

The target Xilinx® part for this project is specified by the `-part xc7k70t` option. If `-part` is not specified, the default part for the Vivado release is used.



**TIP:** You can use the `set_property` command to change the part at a later time, for example:

```
set_property part xc7k325tfbg900-2 [current_project]
```

The `-force` option is technically not required, because the project directory should not exist prior to running this script. However, if the project directory does exist, the script will error out unless the `-force` option is specified, which will overwrite the existing directory and all its contents.

See the *Vivado Design Suite Tcl Command Reference Guide* ([UG835](#)), or at the Tcl prompt type `help <command_name>`, for more information on the `create_project` command, or any other Tcl command used in this tutorial.

---

## Step 2: Add RTL Source Files

For this script, you will be copying all the RTL source files into the local project directory.

Because all the HDL files that you need are located in `<extract_dir>/lab_3/sources`, you can add the entire directory directly.

Add the following two lines to your script:

```
add_files -scan_for_includes ../Lab_3_4_sources/HDL
import_files
```

The `-scan_for_includes` option scans the Verilog source files for any ``include` statements, and also adds these referenced files as source files to the project. By default, ``include` files are not added to the fileset.

The use of `../Lab_3_4_sources` provides a relative path for locating the source files from where the Tcl script is being run. Recall that the `project_run.tcl` script is being created in the `<extract_dir>/lab_3` directory, so the `../Lab_3_4_sources` folder is found in the directory above that.

The `import_files` command copies the files into the local project directory. When no files are specified, as is the case here, the Vivado® Design Suite imports files referenced in the source fileset for the current project.



---

**TIP:** You can reference source files from their original location by not importing the files by using the `import_files` command.

---

## Step 3: Add XDC Constraints

For this design, two XDC files that are required:

- `top_physical.xdc`
- `top_timing.xdc`

1. Add the following lines to your script to import the XDC files into your project:

```
import_files -fileset constrs_1 \  
{../Lab_3_4_sources/Constraints/top_timing.xdc \  
../Lab_3_4_sources/Constraints/top_physical.xdc}
```



---

**TIP:** The “\” character in the preceding text is used to split Tcl commands across multiple lines. This is useful for making scripts that are more readable and easier to edit.

---

By default, all XDC files are used in both synthesis and implementation. However, in this case, you assign the XDC files for use as follows:

- `top_timing.xdc` is used in both synthesis and implementation.
  - `top_physical.xdc` is used only in implementation.
2. To disable the use of `top_physical.xdc` during synthesis, add the following line to your script:

```
set_property used_in_synthesis false [get_files top_physical.xdc]
```

This disables the `used_in_synthesis` property on the specified XDC file.

The property for implementation is `used_in_implementation`, though you will leave that enabled (`true`).

## Step 4: Add Existing IP

★ **IMPORTANT!** *NGC format files are not supported in the Vivado® Design Suite for UltraScale™ devices. It is recommended that you regenerate the IP using the Vivado Design Suite IP customization tools with native output products. Alternatively, you can use the `NGC2EDIF` command to migrate the `NGC` file to EDIF format for importing. However, Xilinx® recommends using native Vivado® IP rather than XST-generated NGC format files going forward.*

You will also import IP cores into the project. There are four IP cores used in this design:

- **Accumulator:** Vivado 2022.1 version with all output products.
- **Block Memory Generator:** An outdated version of native Vivado Design Suite IP with no output products generated.
- **FIFO Generator:** Vivado 2022.1 version with all output products, including DCP.
- **Clock Wizard:** Vivado 2022.1 version with no output products.

All of these IP, are native Vivado cores. They have already been customized, and have a Xilinx Core Instance (XCI) file.

To import these IP cores into the project, add the following lines to your script:

```
import_ip -files { \  
  ../Lab_3_4_sources/IP/Accumulator/c_accum_0.xci \  
  ../Lab_3_4_sources/IP/blk_mem/blk_mem_gen_v7_3_0.xci \  
  ../Lab_3_4_sources/IP/clk_wiz/clk_wiz_0.xci \  
  ../Lab_3_4_sources/IP/char_fifo/char_fifo.xci}
```

When this line is processed, the Vivado Design Suite returns one warning messages:

```
WARNING: [IP_Flow 19-2162] IP 'blk_mem_gen_v7_3_0' is locked:  
* IP definition 'Block Memory Generator (7.3)' for IP 'blk_mem_gen_v7_3_0'  
has a newer major version in the IP Catalog.  
Please select 'Report IP Status' from the 'Tools/Report' menu or run Tcl  
command 'report_ip_status' for more information.
```

The `c_accum_0` is current and all output products are present so no warning messages display when importing.

The `blk_mem_gen_v7_3_0` core is not the latest version in the IP catalog. There are no output products to drive synthesis or simulation, so it will have to be upgraded to the latest version. You will upgrade this IP in a subsequent step. If no upgrade path is available, you must recreate the IP.

For the `clk_wiz_0`, no output products were found with the XCI customization (`.xci`) file, but the IP is the current version in the IP catalog. You will manually generate the output products for this IP in the next step.

The `char_fifo` version is current and all output products are present so no warning messages display when importing.



**RECOMMENDED:** When there is a major version change to an IP core in the catalog, changes to the IP such as parameter or port name changes, may make upgrading the IP to the latest version require changes to the design.

## Step 5: Disable the IP XDC Files

For this design, you disable the XDC files that are included with the Clock Wizard IP, so that you can apply the top-level timing constraints to the `clk_wiz_0` module. This IP has not had the output products generated, so you will first generate the synthesis targets, which includes the XDC files.

In Project Mode, you are not required to generate output products manually. The Vivado® tools generate output products from IP automatically as needed in the design flow, including the generation of a Vivado synthesis DCP. However, because you are changing a property on the XDC files delivered with the `clk_wiz` IP, you must manually generate the output products to create the constraints files or files to change the property.

Also, because you are going to disable the use of an IP XDC file and provide the constraints during synthesis of the top-level design, you must disable the generation of the `clk_wiz` synthesis DCP (or netlist) as well.

1. To disable the automatic generation of a synthesis design checkpoint (DCP) file add the following to your script:

```
set_property generate_synth_checkpoint false [get_files clk_wiz_0.xci]
```

2. Now when the design is synthesized, an out-of-context module (OOC) synthesis run is not automatically created and launched for the `clk_wiz_0` IP. Instead, the `clk_wiz_0` IP is synthesized as part of the top-level design.

3. Add the `generate_target` command to your Tcl script:

```
generate_target all [get_files clk_wiz_0.xci]
```

Multiple output products can be generated by passing a list to the command, such as `{synthesis instantiation_template}`, or you can generate all the available output products by specifying `all`.



**TIP:** To find out what output products an IP supports, use either the `report_property` command on the IP, or `get_property` to get the `KNOWN_TARGETS` property from the IP. For example (do not add these to your script):

```
report_property [get_ips clk_wiz_0]
```

```
get_property KNOWN_TARGETS [get_ips clk_wiz_0]
```

4. To disable the XDC constraints delivered with the Clock wizard, you need the names of the files. You can query the XDC files(s) that are delivered with an IP, by using the `get_files` command with the `-of_objects` and `-filter` options.
5. To capture the XDC file names of the Clock Wizard IP in a Tcl variable, add the following lines to your script:

```
set clk_wiz_xdc [get_files -of_objects [get_files \  
clk_wiz_0.xci] -filter {FILE_TYPE == XDC}]
```

This returns the names of the XDC files that are delivered with the Clock wizard.

6. To disable the XDC files, add this line to your script as well:

```
set_property is_enabled false [get_files $clk_wiz_xdc]
```

The XDC files delivered with `clk_wiz` IP are disabled when you run your script.



**TIP:** To check what XDC files are evaluated in a project, and in what order, you can use the `report_compile_order` command with the `-constraints` option.

## Step 6: Upgrade an IP

As mentioned earlier, the block memory generator IP in the design has a newer version available in the IP catalog. The IP is *locked* as a result, because it cannot be re-customized from the IP catalog unless you upgrade it to the latest version. When adding the XCI to a project, this warning opens:

```
WARNING: [IP_Flow 19-2162] IP 'blk_mem_gen_v7_3_0' is locked:  
* IP definition 'Block Memory Generator (7.3)' for IP 'blk_mem_gen_v7_3_0'  
has a newer major version in the IP Catalog.  
Please select 'Report IP Status' from the 'Tools/Report' menu or run Tcl  
command 'report_ip_status' for more information.
```

In an interactive session, this message can be helpful, but in a batch mode script this would not be seen until after synthesis or implementation fails. To anticipate and prevent this, you can use your script to:

- Determine if an IP is locked.
- Check for a newer version of the IP in the catalog.

- Upgrade an IP if it is locked, and a new version is available.

The following sequence shows you how to do this.

1. Check to see if the IP is locked, and store the state in a Tcl variable. Add the following line to your Tcl script:

```
set locked [get_property IS_LOCKED [get_ips blk_mem_gen_v7_3_0]]
```

2. Next, check to see whether there is an upgrade available in the IP catalog, and store that information in a Tcl variable as well. Add the following line to your Tcl script:

```
set upgrade [get_property UPGRADE_VERSIONS [get_ips blk_mem_gen_v7_3_0]]
```

This returns the VLVN (Vendor/Library/Name/Version) identifier of the upgrade, if there is one available. If there is no upgrade, the variable will contain an empty string (""). In the case of the `blk_mem_gen_v7_3_0` IP, there is an upgrade available.

3. Check the stored Tcl variables, `$locked` and `$upgrade`, to see if the IP is locked, and if there is an upgrade version for the IP. If so, you can upgrade the IP.
4. Add the following lines to your Tcl script:

```
if {$upgrade != "" && $locked} {  
  upgrade_ip [get_ips blk_mem_gen_v7_3_0]}
```

This results in an upgrade to the block memory generator IP from the current version in the design to the latest version in the IP catalog.

---

## Step 7: Setup Design Runs for IP

Now that you upgraded all the IP to the latest version, you can optionally setup and launch out-of-context synthesis runs for the IP. You already configured the Clocking wizard IP to not use the out-of-context flow in Step 5.

The FIFO Generator IP and the Accumulator IP already had all the output products, including the DCP, generated when it was imported. However, the Block Memory Generator IP has not had any output products generated.

In a project flow, if the IP is current then all the output products, including the DCP, are generated automatically. However, you can also script this step. One reason to manually create and launch design runs for the IP cores in a project is to have the out-of-context synthesis processes run concurrently. By default, they are launched serially.

## Create and Launch a Design Run for the Block Memory Generator IP

For a synthesis design checkpoint (DCP) to be generated, an IP Design Run must first be created.

1. Add the following to your Tcl script:

```
create_ip_run [get_ips blk_mem_gen_v7_3_0]
```

A new run is created for the IP, with the name `<ip_name>_synth_1`. The IP design run is launched using the `launch_runs` command.

2. Add the following to your script:

```
launch_runs [get_runs blk_mem_gen_v7_3_0_synth_1]
```

The run is now launched and when completed a synthesis design checkpoint is added to the project. If you do not launch the IP run, it launches automatically when synthesizing the top-level.

If you have multiple IP runs that you created, those runs are launched serially. To launch in parallel you need to use the `launch_runs` command.

During synthesis of the top level logic, a black box is inferred for the IP. During implementation the DCP is opened and the netlist is read and constraints are applied.

When launching the top level synthesis run it waits automatically for any IP runs to complete, so there is no need to use `wait_on_run` commands for each IP run.

**Note:** If you wanted to have a number of IP generate DCPs in parallel, you can either:

- Create all the runs first and then launch all the runs.
- Create and launch IP runs one after another.

---

## Step 8: Launch Synthesis and Implementation

The project is now ready for synthesis and implementation. The Vivado® Design Suite automatically generates the necessary output products for the various IP in your project, as needed. You do not need to manually generate the output products unless you want to make changes to the generated output products prior to using them in synthesis, simulation, or implementation.

In the Project Mode, the Vivado Design Suite manages the details of synthesis and implementation runs, using run strategies and maintains the state of the design. Therefore, you will use the `launch_runs` command to run synthesis and implementation in project-based designs.



1. Add the following line to your Tcl script:

```
launch_runs synth_1
```

By default, a synthesis run called `synth_1` is created for every project. You can also manually create new runs using the `create_run` command, and configure run properties using the `set_property` command. See the *Vivado Design Suite User Guide: Design Flows Overview (UG892)* for more information on creating and configuring runs.

After the synthesis run has completed, you can launch an implementation run. However, because the implementation run is dependent on the completion of the synthesis run, you must use the `wait_on_run` command to hold your Tcl script until synthesis is complete.

2. Add these two lines to your script:

```
wait_on_run synth_1  
launch_runs impl_1 -to_step write_bitstream
```

When the synthesis run, `synth_1`, completes, the implementation run, `impl_1`, begins. Implementation is a multi-step process that begins with netlist optimization, runs through placement and routing, and can even include generating the bitstream for the Xilinx FPGA.

The `-to_step` option that you added to your Tcl script indicates that implementation should include generating the bitstream for the device. By default, implementation does not include that step. See the *Vivado Design Suite User Guide: Implementation (UG904)* for more information.



**TIP:** Alternatively, you can use the `write_bitstream` command; this requires that you open the implementation run first using the `open_run` command.

Just as implementation needed to wait on synthesis to complete, wait for your Tcl script to allow implementation to complete before generating any reports, or exiting.

3. Add the `wait_on_run` command to your Tcl script, to wait for the implementation run to complete:

```
wait_on_run impl_1
```

The script will wait until the implementation run completes before continuing.

---

## Step 9: Run the Script

You are now ready to run the script. Your script should be similar to the following:

```
#Step 1: Create Project  
cd C:/ug939-design-files/lab_3  
create_project -force -part xc7k70tfbg484-3 my_project my_project  
#Step 2: Adding RTL Files  
add_files -scan_for_includes ../Lab_3_4_sources/HDL  
import_files
```

```

#Step 3: Adding XDC Files
import_files -fileset constrs_1 \
{../Lab_3_4_sources/Constraints/top_timing.xdc \
../Lab_3_4_sources/Constraints/top_physical.xdc}
set_property used_in_synthesis false [get_files top_physical.xdc]
#Step 4: Importing IP
import_ip { \
../Lab_3_4_sources/IP/Accumulator/c_accum_0.xci \
../Lab_3_4_sources/IP/blk_mem/blk_mem_gen_v7_3_0.xci \
../Lab_3_4_sources/IP/clk_wiz/clk_wiz_0.xci \
../Lab_3_4_sources/IP/char_fifo/char_fifo.xci}
#Step 5: Disable XDC
set_property generate_synth_checkpoint false [get_files clk_wiz_0.xci]
generate_target all [get_files clk_wiz_0.xci]
set clk_wiz_xdc [get_files -of_objects [get_files \
clk_wiz_0.xci] -filter {FILE_TYPE == XDC}]
set_property is_enabled false [get_files $clk_wiz_xdc]
#Step 6: Upgrade IP
set locked [get_property IS_LOCKED [get_ips blk_mem_gen_v7_3_0]]
set upgrade [get_property UPGRADE_VERSIONS [get_ips blk_mem_gen_v7_3_0]]
if {$upgrade != "" && $locked} {
upgrade_ip [get_ips blk_mem_gen_v7_3_0]}
#Step 7: Launch IP run
create_ip_run [get_ips blk_mem_gen_v7_3_0]
launch_runs [get_runs blk_mem_gen_v7_3_0_synth_1]
#Step 8: Launching Synthesis and Implementation launch_runs synth_1
launch_runs synth_1
wait_on_run synth_1
launch_runs -to_step write_bitstream impl_1
wait_on_run impl_1
  
```

## Source the Tcl Script

You can run the `project_run.tcl` script in Vivado® Design Suite batch mode or Tcl mode.

- Batch mode will run the sourced script, and then automatically exit the tool after the script has finished processing.
- Tcl mode will run the sourced script, and return to the Tcl command prompt when finished.
  - On Linux:

Change to the directory where the lab materials are stored:

```
cd <Extract_Dir>/lab_3
```

Launch the Vivado Design Suite Tcl shell, and source a Tcl script to create the tutorial design:

```
vivado -mode tcl -source project_run.tcl
```

- On Windows:

Launch the Vivado Design Suite Tcl shell:

```
Start > All Programs > Xilinx Design Tools > Vivado 2020.2 > Vivado
2020.2 Tcl Shell
```

Your Vivado Design Suite installation might be called something different from Xilinx Design Tools on the Start menu.

1. In the Tcl shell, change to the directory where the lab materials are stored:

```
Vivado% cd <extract_dir>/lab_3
```

2. Source the Tcl script to create the design:

```
Vivado% source project_run.tcl
```



**IMPORTANT!** *If your Tcl script has an error in it, the script will halt execution at the point of the error. You will need to fix the error, and re-source the Tcl script as needed. If you are running in Tcl mode, you may need to close the current project with `close_project`, or exit the Vivado tool with `exit` to source the Tcl script again.*

After the script executes, a project directory structure is created, default log and reports are generated, a synthesized netlist is produced, design is fully implemented, and a bitstream is created. You can use the Vivado IDE to examine the design, the various reports, and to perform analysis on the design.

3. To open the GUI from the Tcl prompt type:

```
start_gui
```

---

## Conclusion

Creating a design project does not require the use of the Vivado® IDE. The benefits of the Project Mode, such as automatic report generation, design runs infrastructure, and managed source files, are available from a Tcl script. The result of the script is a design project, which you can open in the Vivado IDE for interactive timing analysis and design exploration. Specific areas covered in this lab are:

- Creating a project and adding HDL sources.
- Adding IP sources to a project.
- Generating IP Output Products.
- Disabling IP Output Products, such as an XDC file.
- Disabling the creation of a Synthesis Design Checkpoint (DCP) file.
- Creating and launching IP design runs.
- Querying the properties of the IP.
- Updating an IP to the latest version.

# Scripting the Non-Project Mode

---

## Introduction

In [Lab 3: Scripting the Project Mode](#), you created a Tcl script to run the Vivado Design Suite in Project Mode. In this lab, you will create a Tcl script to run the Vivado tools in Non-Project Mode. Many of the commands used in this lab are the same commands used in Lab 3. The main difference between Project Mode and Non-Project Mode is that there is no project file or directory structure created on disk. Instead, Vivado Design Suite manages the design directly in an in-memory database.

In Non-Project Mode, you do not have a project file to add source file references to, or a project directory to manage source files. In Non-Project Mode, you read source files into the Vivado Design Suite to create the in-memory design. In addition, there is no design runs infrastructure to store run strategies and results. Instead, you directly call the various commands to run the different stages of synthesis and implementation. Unlike Project Mode, you must manually write out design checkpoints, netlists, and reports. These items are not created automatically for you in Non-Project Mode. For more information on Project Mode and Non-Project Mode, refer to the *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#)).

When working with IP in Non-Project Mode you must manually generate output products, including synthesis Design Checkpoints if desired.



**CAUTION!** When copying Tcl commands and script examples from this Tutorial document and pasting them into the Vivado Design Suite Tcl shell, or Tcl Console, the dash “-” character can sometimes be converted to an em-dash “–” which results in errors when the commands are run.

---

## Step 1: Read the Design Source Files

1. Invoke a text editor of your choice, such as Emacs, VI, or Notepad; or launch the Text Editor from within the Vivado® IDE.
2. Save a new file called `non_project_run.tcl` in `<Extract_Dir>/lab_4`.

In Lab 4, you started your project by creating a project; here you will begin by creating an in-memory design, and reading source files. However, you want to first make sure that the Tcl script is in the right location to find source files referenced by the script.

3. Add the following line to your Tcl script to change to the appropriate directory for this lab:

```
cd <extract_dir>4/lab_4
```

4. You can replace the `<extract_dir>` variable with the actual path to your tutorial data. For example: `C:/ug939-design-files/lab_4`.

A default target part is used unless you specify another. This target part is used for all IP that are added to the design, otherwise the default part for the Vivado® Design Suite is used. Additionally, setting the target part removes the need to specify a part when synthesizing the top-level design, or any IP for out-of-context synthesis.

5. Add the following to your script to set the part to be used:

```
set_part xc7k70tfbg484-3
```



**RECOMMENDED:** *In Non-Project Mode, there is no project part unless you create an in-memory project. If you do not create an in-memory project, the IP output products are generated using the default part of the Vivado Design Suite release. This default part might not be the intended target part specified by the `synth_design` command, and can result in mismatched synthesis results between the IP and the top-level design in Non-Project Mode designs. To see how a target part is specified for a customized IP see Lab 2: Creating and Managing Reusable IP for more information on managing IP customizations.*

Now you are ready to read the source files for the design. In Project Mode, you use commands such as `add_files` and `import_files` to add source files to the project.

In Non-Project Mode, you can use `add_files`, which calls the appropriate lower-level command, but it is more typical to directly read the file type. This is similar to an ASIC tool flow. For this lab, you are working with Verilog source files and will use `read_verilog` to read them.

6. Add the following line to your script to read all the Verilog source for this project:

```
read_verilog [glob ../Lab_3_4_sources/HDL/*.v]
```



**TIP:** *The `glob` command is a built-in Tcl command that creates a list out of the specified objects. Alternatively, you can make a Tcl list to pass to `read_verilog`, or use a separate `read_verilog` command for each file.*

## Step 2: Add Existing IP

1. Read the following IP cores into the design:
  - **Accumulator:** Version native IP with all output products, including a DCP.

- **FIFO Generator:** Version native IP with all output products, including a DCP.
- **Clock Wizard:** Version native IP with no output products.
- **Block memory generator:** An outdated version of a native Vivado Design Suite IP with no output products generated.

All of these IP have already been customized, and have Xilinx Core Instance (XCI) files.

The FIFO Generator IP already has all required output products available, and can be read and used directly from its current location. Whenever you create an IP customization, you should always generate all available output products.

2. To read the FIFO Generator IP and the Accumulator IP, including all the output products that are present, add the following lines to your script:

```
read_ip ../Lab_3_4_sources/IP/Accumulator/c_accum_0.xci
read_ip ../Lab_3_4_sources/IP/char_fifo/char_fifo.xci
```

3. Add the following to your Tcl script to create a local directory, with sub-directories for the block memory and the clock wizard IP:

```
file mkdir IP/blk_mem
file mkdir IP/clk_wiz
```

4. Add the following to your Tcl script to copy the needed XCI files from the current IP repository into the local directory:

```
file copy -force ../Lab_3_4_sources/IP/blk_mem/
blk_mem_gen_v7_3_0.xci ./IP/blk_mem
file copy -force ../Lab_3_4_sources/IP/clk_wiz/clk_wiz_0.xci ./IP/clk_wiz
```

The `-force` option causes the file to be overwritten if it already exists.




---

**IMPORTANT!** Without the `-force` option, an error could be returned and the script will quit.

---

For the clocking wizard IP, you need to generate the output products before you can synthesize the design. In Non-Project Mode, the Vivado® Design Suite does not automatically generate output products, and errors will be encountered if you do not do this prior to launching synthesis.

Generating output products results in files and directories being written to the location the IP XCI files are read from. In a managed IP repository, these locations may be read-only or under revision control. In this case, you would copy the needed XCI files to a local directory before reading them into the in-memory design.




---

**IMPORTANT!** Have each XCI file stored in its own directory. When output products are generated, they are written to the same directory as the XCI file. If IP files are written to the same directory, it is possible that output products from different IPs could overwrite each other.

---

5. Add these lines to your Tcl script to read in the needed XCI files:

```
read_ip ./IP/blk_mem/blk_mem_gen_v7_3_0.xci
read_ip ./IP/clk_wiz/clk_wiz_0.xci
```

The specified XCI files are read into the in-memory design.

Unlike in Lab 3, the warnings related to locked IP do not display when the IP are processed into the design using the `read_ip` command. In Project Mode, the Vivado Design Suite performs checks as an IP is added to a project, resulting in the warning messages seen in Lab 3, [Step 4: Add Existing IP](#). In Non-Project Mode, the checks are only performed when the IP are processed during synthesis.

---

## Step 3: Disable XDC Files

As in Project Mode, if an IP delivers XDC constraints, they are automatically processed and added to the in-memory design. For this design, you will disable the XDC files that are included with the Clock Wizard IP as you have constraints in the top-level XDC file that you will apply instead. However, the Clock Wizard IP has not had the output products generated, so you will first need to generate the output products, which include the XDC files.

When generating the output products for an included XCI file IP, you must decide whether to use an out-of-context flow, including the creation of a synthesis Design Checkpoint (DCP), or to let the IP be synthesized as part of the top-level design.

Because, for the Clocking wizard, you want to disable the use of an XDC; consequently you need to also configure the IP to not use a DCP.

1. Add the following to your script to configure the IP to not use a Synthesis Design Checkpoint:

```
set_property generate_synth_checkpoint false [get_files clk_wiz_0.xci]
```

Now when synthesis is done in the top-level, the RTL output products are used and a DCP is not expected. Unlike in a Project Flow, output products are not automatically generated as needed, and need to be manually created.

2. Add the `generate_target` command to your Tcl script:

```
generate_target all [get_ips clk_wiz_0]
```

Because you copied the XCI file from the source IP repository into a local directory, the output products are written to the local directory.



**TIP:** To find out what output products an IP supports, use either the `report_property` command on the IP, or `get_property` to get the `KNOWN_TARGETS` property from the IP. For example, do not add these to your script:

```
report_property [get_ips clk_wiz_0]  
get_property KNOWN_TARGETS [get_ips clk_wiz_0]
```

Multiple output products can be generated by passing a list to the command, such as `{synthesis instantiation_template}`.

To disable the XDC files delivered with the Clock Wizard, you need the names of the files. You can query the XDC file(s) that are delivered with an IP, by using the `get_files` command with the `-of_objects` and `-filter` options.

3. To capture the XDC file names of the Clock Wizard IP in a Tcl variable, add the following lines to your script:

```
set clk_wiz_xdc [get_files -of_objects \  
[get_files clk_wiz_0.xci] -filter {FILE_TYPE == XDC}]
```

This returns the names of the XDC files delivered with the Clock Wizard.

4. To disable the XDC files, add this line to your script as well:

```
set_property is_enabled false [get_files $clk_wiz_xdc]
```

The XDC files delivered with `clk_wiz` IP are disabled when you run your script.

To check what XDC files are evaluated, and in what order, you can use the `report_compile_order` command with the `-constraints` option.

---

## Step 4: Upgrade IP

If you attempt to run synthesis at this time, you will see this warning for the Block Memory Generator IP:

```
WARNING: [IP_Flow 19-2162] IP 'blk_mem_gen_v7_3_0' is locked:  
* IP definition 'Block Memory Generator (7.3)' for IP 'blk_mem_gen_v7_3_0'  
has a newer major version in the IP Catalog.  
Please select 'Report IP Status' from the 'Tools/Report' menu or run Tcl  
command 'report_ip_status' for more information..
```

The Block Memory Generator is locked because it is not the most recent version of the IP. It also does not have any output products and so must be upgraded before output products can be generated.

The Block Memory Generator has updated versions in the Xilinx® IP catalog. In an interactive design session, these messages can be helpful; but in a batch mode Tcl script these messages are not seen until after synthesis or implementation fails.

To anticipate and prevent this, you can use your script to:

- Determine if an IP is locked.
- Check for a newer version of the IP in the catalog.
- Upgrade an IP if it is locked, and a new version is available.
- Generate the output products for the IP.

Do this for the `blk_mem_gen_v7_3_0` IP using following sequence:



1. First, check to see if the IP is locked, and store the state in a Tcl variable. Add the following line to your Tcl script:

```
set locked [get_property IS_LOCKED [get_ips blk_mem_gen_v7_3_0]]
```

2. Next, you will check to see if there is an upgrade available in the IP catalog, and store that information in a Tcl variable as well. Add the following line to your Tcl script:

```
set upgrade [get_property UPGRADE_VERSIONS [get_ips blk_mem_gen_v7_3_0]]
```

This returns the VLNV (Vendor/Library/Name/Version) identifier of the upgrade, if there is one available.

If there is no upgrade, the variable contains an empty string (""). In the case of the `blk_mem_gen_v7_3_0` IP, there is an upgrade available.

3. Now you can check the stored Tcl variables, `$locked` and `$upgrade`, to see if the IP is locked, and if there is an upgrade version for the IP. If so, you can upgrade the IP. Add the following lines to your Tcl script:

```
if {$locked && $upgrade != ""} {  
  upgrade_ip [get_ips blk_mem_gen_v7_3_0]}
```

This results in an upgrade to the block memory generator IP from the current version in the design to the latest version in the IP catalog.

4. Now that the IP is current, add the following to your script:

```
generate_target all [get_ips blk_mem_gen_v7_3_0]
```

5. Similarly, you can check if other IP's like Accumulator or FIFO Generator are locked. This tutorial uses Vivado 2022.1 native IP with output products therefore these IP's need not be updated.

Since the IP's are in the current state, add following to your script.

```
generate_target all [get_ips c_accum_0]  
generate_target all [get_ips char_fifo]
```

---

## Step 5: Create DCP for IP

The default flow for Vivado® is to use a Synthesis Design Checkpoint for IP. Typically when referencing IP in a Non-Project Flow you would have created the IP customizations using a Manage IP project. At that point, you would have created the output products and decided to disable DCP use or generated the DCP. This is the case with the FIFO Generator IP (`char_fifo`); it has all the output products generated including a DCP. The other three IP all consist of just an XCI or XCO file and thus you need to configure the synthesis option. Either create a DCP for the IP or configure it to be synthesized with the top-level logic.

In Step 3, you configured the Clocking Wizard to not use a DCP by setting a property on the IP XCI file. At this point, you will generate a DCP for the Block Memory Generator IP.

Add the following lines to your script to create the DCP for the Block Memory Generator IP, FIFO Generator IP, and Accumulator IP:

```
synth_ip [get_ips blk_mem_gen_v7_3_0]  
synth_ip [get_ips c_accum_0]  
synth_ip [get_ips char_fifo]
```

This results in a DCP file being created and stored in the directory containing the IP XCI file.

---

## Step 6: Run Synthesis

For this design, there are two XDC files that are required: `top_timing.xdc` and `top_physical.xdc`. One of the XDC files is used in both synthesis and implementation (`top_timing.xdc`) while the other is used only during implementation (`top_physical.xdc`).

At this point in your Tcl script, you want to read the XDC file, using `read_xdc`, which is used in both synthesis and implementation.

1. Add the following to your Tcl script:

```
read_xdc ../Lab_3_4_sources/Constraints/top_timing.xdc
```

The design is now ready for synthesis.

In Non-Project Mode, unlike Project Mode, there are no design runs to launch, and no runs infrastructure managing the strategies used and the state of the design. You will manually launch the various stages of synthesis and implementation.

2. For synthesis, you use the `synth_design` command. Add the following to your Tcl script:

```
synth_design -top sys_integration_top
```

Because you created an in-memory project and set the target part, defining the target part is not needed here; however, you must provide the top-level module name with the `synth_design` command.

The various Verilog files read into the in-memory design in Step 1: Reading Design Source Files do not reference other files via an ``include` statement. If they did, you would need to define the ``include` search directories with the `-include_dirs` option.

After synthesis, you should generate a design checkpoint to save the results. This way you can restore the synthesized design at any time without running synthesis again.

3. Add the following `write_checkpoint` command to your Tcl script:

```
write_checkpoint -force post_synth.dcp
```

The `-force` option is used to overwrite the checkpoint file if it already exists.

You can also generate any needed reports at this point, such as a post-synthesis timing summary report.

4. Add the following line to your Tcl script:

```
report_timing_summary -file timing_syn.rpt
```

This command creates a comprehensive timing report for the design and writes the report to a file.

---

## Step 7: Run Implementation

With synthesis completed, you are now ready to script implementation. There are many steps to implementation, in both Project Mode and Non-Project Mode. However, in Project Mode, you select a design run strategy that controls all of the various steps, and launch that run. In Non-Project Mode, without a design run, you must determine your implementation strategy by manually running each step of implementation, and selecting the Tcl command options to use at each step. You can also choose to skip some steps, such as logic optimization, power optimization, and physical synthesis.

In this lab, you run the following steps:

- Logic optimization: `opt_design`
- Placement: `place_design`
- Physical synthesis: `phys_opt_design`
- Routing: `route_design`
- Bitstream generation: `write_bitstream`

For a complete description of each of these steps, see *Vivado Design Suite User Guide: Implementation* ([UG904](#)).

Between each of these steps, you can generate reports, and write checkpoints to save the design in different stages of implementation.

Before launching implementation, you must read the design constraints that are only used in implementation. The XDC file, `top_physical.xdc`, contains physical constraints that are used in implementation, but do not apply to synthesis.

In this case, these constraints could have been read into the in-memory design prior to synthesis, because synthesis ignores them; however, this file could also contain different timing constraints, not to be used in synthesis, that must be read in after synthesis and just prior to implementation.

1. Add the following line to your Tcl script:

```
read_xdc ../Lab_3_4_sources/Constraints/top_physical.xdc
```

2. Add optimization and placement commands to your Tcl script:

```
opt_design  
place_design  
write_checkpoint -force post_place.dcp  
report_timing -file timing_place.rpt
```

After placement completes, your script writes a post-placement checkpoint and create a custom timing report, which provides a detailed timing report for the single worst timing path in the design.

3. Add physical synthesis and routing commands to your Tcl script:

```
phys_opt_design  
route_design  
write_checkpoint -force post_route.dcp  
report_timing_summary -file timing_summary
```

After routing completes, your script writes a post-routing design checkpoint and creates a timing summary report.

4. Finally, write out a bitstream by adding the following:

```
write_bitstream -force sys_integration_top.bit
```

This is the complete Non-Project Mode design flow for implementing a design from RTL source files, including designing with IP, through bitstream generation.

---

## Step 8: Run the Script

You are now ready to run the Tcl script. Your script should be similar to the following:

```
#Step 1: Reading RTL  
cd C:/ug939-design-files/lab_4  
set_part xc7k70t-fbg484-3  
read_verilog [glob ../Lab_3_4_sources/HDL/*.v]  
#Step 2: Adding Existing IP  
read_ip ../Lab_3_4_sources/IP/char_fifo/char_fifo.xci  
read_ip ../Lab_3_4_sources/IP/Accumulator/c_accum_0.xci  
file mkdir IP/blk_mem  
file mkdir IP/clk_wiz  
file copy -force ../Lab_3_4_sources/IP/blk_mem/blk_mem_gen_v7_3_0.xci ./IP/  
blk_mem  
file copy -force ../Lab_3_4_sources/IP/clk_wiz/clk_wiz_0.xci ./IP/clk_wiz  
read_ip ./IP/blk_mem/blk_mem_gen_v7_3_0.xci
```

```

read_ip ./IP/clk_wiz/clk_wiz_0.xci
#Step 3: Disable DCP and XDC
set_property generate_synth_checkpoint false [get_files clk_wiz_0.xci]
generate_target all [get_ips clk_wiz_0]
set clk_wiz_xdc [get_files -of_objects [get_files \
clk_wiz_0.xci] -filter {FILE_TYPE == XDC}]
set_property is_enabled false [get_files $clk_wiz_xdc]
#Step 4: Upgrade IP
set locked [get_property IS_LOCKED [get_ips blk_mem_gen_v7_3_0]]
set upgrade [get_property UPGRADE_VERSIONS [get_ips blk_mem_gen_v7_3_0]]
if {$upgrade != "" && $locked} {
upgrade_ip [get_ips blk_mem_gen_v7_3_0]}
generate_target all [get_ips blk_mem_gen_v7_3_0]
generate_target all [get_ips c_accum_0]
generate_target all [get_ips char_fifo]
#Step 5: Creating DCP for IP
synth_ip [get_ips blk_mem_gen_v7_3_0]
synth_ip [get_ips c_accum_0]
synth_ip [get_ips char_fifo]
#Step 6: Running Synthesis
read_xdc ../Lab_3_4_sources/Constraints/top_timing.xdc
synth_design -top sys_integration_top
write_checkpoint -force post_synth.dcp
report_timing_summary -file timing_syn.rpt
#Step 7: Running Implementation
read_xdc ../Lab_3_4_sources/Constraints/top_physical.xdc
opt_design
place_design
write_checkpoint -force post_place.dcp
report_timing -file timing_place.rpt
phys_opt_design
route_design
write_checkpoint -force post_route.dcp
report_timing_summary -file timing_summary
write_bitstream -force sys_integration_top.bit
  
```

## Source the Tcl Script

You can run the `non_project_run.tcl` script in Vivado® Design Suite batch mode or Tcl mode.

- Batch mode runs the sourced script, and then automatically exit the tool after the script has finished processing.
  - Tcl mode runs the sourced script, and return to the Tcl command prompt when finished.
1. Change to the directory where the lab materials are stored:
    - On Linux: `cd <Extract_Dir>/lab_4`
  2. Launch the Vivado® Design Suite Tcl shell, and source a Tcl script to create the tutorial design:
    - On Linux: `vivado -mode tcl -source non_project_run.tcl`
    - On Windows, click **Start** → **All Programs** → **Xilinx Design Tools** → **Vivado 2021.2** → **Vivado 2021.2 Tcl Shell** to launch the Vivado Design Suite Tcl shell.

### 3. In the Tcl shell:

- Change to the directory where the lab materials are stored:

```
Vivado% cd <Extract_Dir>/lab_4
```

- Source the Tcl script to create the design:

```
Vivado% source non_project_run.tcl
```

After the sourced script has completed, the Tcl shell displays the `Vivado%` prompt.



**IMPORTANT!** If your Tcl script has an error in it, the script will halt execution at the point of the error. You will need to fix the error, and re-source the Tcl script as needed. If you are running in Tcl mode, you may need to close the current project with `close_project`, or exit the Vivado tool with `exit` to source the Tcl script again.

```
Tcl Console x Messages Log Reports Design Runs ? _ □ ↻
[?] [?] [?] [?] [?] [?] [?]
INFO: [IP_Flow 19-234] Refreshing IP repositories
INFO: [IP_Flow 19-1704] No user IP repositories specified
INFO: [IP_Flow 19-2313] Loaded Vivado IP repository 'C:/Xilinx_0324/Vivado/2017.1/data/ip'.
cd c:/xilinx/ug939-design-files/lab_4/solutions/
source non_project_run.tcl
```

Running the script results in the creation of a directory called IP. Output products for the various IPs used in the design are written to this directory. Reports, design checkpoints, and a bitstream for the design are also written to disk.

- ### 4. You can open the design in the Vivado IDE to perform further analysis. To open the Vivado IDE from the Tcl prompt type: `start_gui`.

## Conclusion

Using the Non-Project Mode gives you the greatest control over the Vivado® Design Suite, and gives you access to advanced features that may not be available in Project Mode. However, Non-Project Mode also requires manually managing source files, updating the design when source files have changed, and manually planning and running synthesis and implementation strategies. Specific areas covered in this lab are:

- Reading in Verilog source files and reading IP sources.
- Generating required IP output products for synthesis and implementation, and disabling them as needed.
- Querying the upgradability of an IP, and updating to a newer version when appropriate.
- Creating synthesis design checkpoints (DCPs) for IP.
- Manually running synthesis and individual steps of implementation.

- Generating custom reports.

# Additional Resources and Legal Notices

---

## Xilinx Resources

For support resources such as Answers, Documentation, Downloads, and Forums, see [Xilinx Support](#).

---

## Documentation Navigator and Design Hubs

Xilinx® Documentation Navigator (DocNav) provides access to Xilinx documents, videos, and support resources, which you can filter and search to find information. To open DocNav:

- From the Vivado® IDE, select **Help** → **Documentation and Tutorials**.
- On Windows, select **Start** → **All Programs** → **Xilinx Design Tools** → **DocNav**.
- At the Linux command prompt, enter `docnav`.

Xilinx Design Hubs provide links to documentation organized by design tasks and other topics, which you can use to learn key concepts and address frequently asked questions. To access the Design Hubs:

- In DocNav, click the **Design Hubs View** tab.
- On the Xilinx website, see the [Design Hubs](#) page.

**Note:** For more information on DocNav, see the [Documentation Navigator](#) page on the Xilinx website.

---

## References

These documents provide supplemental material useful with this guide:



1. *Vivado Design Suite User Guide: Release Notes, Installation, and Licensing* ([UG973](#))
2. *Vivado Design Suite User Guide: Using the Vivado IDE* ([UG893](#))
3. *Vivado Design Suite User Guide: Designing with IP* ([UG896](#))
4. *Vivado Design Suite Tcl Command Reference Guide* ([UG835](#))
5. *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#))
6. *Vivado Design Suite User Guide: Logic Simulation* ([UG900](#))
7. *Vivado Design Suite User Guide: Using Tcl Scripting* ([UG894](#))
8. *Vivado Design Suite User Guide: Implementation* ([UG904](#))

## Revision History

The following table shows the revision history for this document.

Section	Revision Summary
<b>06/10/2022 Version 2022.1</b>	
General updates	Validated for Xilinx release 2022.1.

## Please Read: Important Legal Notices

The information disclosed to you hereunder (the "Materials") is provided solely for the selection and use of Xilinx products. To the maximum extent permitted by applicable law: (1) Materials are made available "AS IS" and with all faults, Xilinx hereby DISCLAIMS ALL WARRANTIES AND CONDITIONS, EXPRESS, IMPLIED, OR STATUTORY, INCLUDING BUT NOT LIMITED TO WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT, OR FITNESS FOR ANY PARTICULAR PURPOSE; and (2) Xilinx shall not be liable (whether in contract or tort, including negligence, or under any other theory of liability) for any loss or damage of any kind or nature related to, arising under, or in connection with, the Materials (including your use of the Materials), including for any direct, indirect, special, incidental, or consequential loss or damage (including loss of data, profits, goodwill, or any type of loss or damage suffered as a result of any action brought by a third party) even if such damage or loss was reasonably foreseeable or Xilinx had been advised of the possibility of the same. Xilinx assumes no obligation to correct any errors contained in the Materials or to notify you of updates to the Materials or to product specifications. You may not reproduce, modify, distribute, or publicly display the Materials without prior written consent. Certain products are subject to the terms and conditions of Xilinx's limited warranty, please refer to Xilinx's Terms of Sale which can be viewed at <https://>

[www.xilinx.com/legal.htm#tos](http://www.xilinx.com/legal.htm#tos); IP cores may be subject to warranty and support terms contained in a license issued to you by Xilinx. Xilinx products are not designed or intended to be fail-safe or for use in any application requiring fail-safe performance; you assume sole risk and liability for use of Xilinx products in such critical applications, please refer to Xilinx's Terms of Sale which can be viewed at <https://www.xilinx.com/legal.htm#tos>.

### **AUTOMOTIVE APPLICATIONS DISCLAIMER**

AUTOMOTIVE PRODUCTS (IDENTIFIED AS "XA" IN THE PART NUMBER) ARE NOT WARRANTED FOR USE IN THE DEPLOYMENT OF AIRBAGS OR FOR USE IN APPLICATIONS THAT AFFECT CONTROL OF A VEHICLE ("SAFETY APPLICATION") UNLESS THERE IS A SAFETY CONCEPT OR REDUNDANCY FEATURE CONSISTENT WITH THE ISO 26262 AUTOMOTIVE SAFETY STANDARD ("SAFETY DESIGN"). CUSTOMER SHALL, PRIOR TO USING OR DISTRIBUTING ANY SYSTEMS THAT INCORPORATE PRODUCTS, THOROUGHLY TEST SUCH SYSTEMS FOR SAFETY PURPOSES. USE OF PRODUCTS IN A SAFETY APPLICATION WITHOUT A SAFETY DESIGN IS FULLY AT THE RISK OF CUSTOMER, SUBJECT ONLY TO APPLICABLE LAWS AND REGULATIONS GOVERNING LIMITATIONS ON PRODUCT LIABILITY.

### **Copyright**

© Copyright 2013-2022 Xilinx, Inc. Xilinx, the Xilinx logo, Alveo, Artix, Kintex, Kria, Spartan, Versal, Vitis, Virtex, Vivado, Zynq, and other designated brands included herein are trademarks of Xilinx in the United States and other countries. All other trademarks are the property of their respective owners.