



HAL
open science

Installing Embedded Linux on ZedBoard

Clément Foucher

► **To cite this version:**

| Clément Foucher. Installing Embedded Linux on ZedBoard. 2015. hal-01232886v2

HAL Id: hal-01232886

<https://hal.science/hal-01232886v2>

Preprint submitted on 21 Mar 2017

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Installing Embedded Linux on ZedBoard

Clément Foucher ([homepage](#))
Clement.Foucher@laas.fr

LAAS-CNRS
Laboratoire d'analyse et d'architecture des systèmes

Version 1.1

This work is licensed under the Creative Commons
Attribution-ShareAlike 4.0 International License.

To view a copy of this license,
visit <http://creativecommons.org/licenses/by-sa/4.0/>.

February 23, 2017

Contents

1	Before starting	5
1.1	Document purpose	5
1.2	Disclaimer	6
1.3	Tools revisions and OS	6
1.4	Administrator privileges	6
1.5	Conventions and directories	6
1.6	Projects	7
1.7	Scripts and logs	7
1.8	Environment packages and libraries	7
1.8.1	Fedora 22 Workstation	7
1.8.2	Ubuntu 16.04 LTS	8
2	Additional technical information	9
2.1	Downloading required sources	9
2.2	ZedBoard boot process	9
2.3	Reinitializing the SD card to factory state	10
2.3.1	Restore partition scheme	10
2.3.2	Restore files	10
3	Preparing the environment	11
3.1	Configuring the scripts	11
3.2	Creating the project	11
4	Hardware layout and low-level software	12
4.1	Creating a base hardware design	12
4.1.1	Default configuration using script	12
4.1.2	Manual procedure	12
4.2	Generating the device tree	13
4.2.1	Downloading the device tree generator	13
4.2.2	Generating the device tree	13
4.3	Generating the first stage boot loader	14
4.4	Generating the bootloader	14
4.4.1	Default configuration using script	14
4.4.2	Manual procedure	14
4.5	Generating the Binary file	15
5	Generating Linux	17
5.1	Generating the kernel	17
5.1.1	Default configuration using script	17
5.1.2	Custom configuration using script	17
5.1.3	Manual procedure	18
5.2	Generating the device tree blob	18

5.2.1	Default configuration using script	19
5.2.2	Manual procedure	19
5.3	Generating the file system	19
5.3.1	Default configuration using script	19
5.3.2	Custom configuration using script	19
5.3.3	Manual procedure	20
6	Preparing the board	21
6.1	Partitioning the SD card	21
6.2	Copying the file system on the card	21
6.2.1	Using the script	21
6.2.2	Manually	22
6.3	Copying the system files	22
7	Working on the board	23
7.1	Hardware configuration	23
7.2	On the first boot	23
7.2.1	SSH configuration	23
7.3	Using Xilinx SDK to create an application	24

Document revisions

Revision number	Date	Changes
1.0	2015/09/21	Initial release.
1.0.1	2015/10/02	Minor spelling corrections; Fixed missing sudo in mount commands.
1.0.2	2015/10/06	Corrected BuildRoot options; Updated provided BuildRoot config file.
1.0.3	2015/11/26	Corrected emptiness check on SD card in copy script; Added libraries required for Ubuntu.
1.1	2017/02/23	Updated tools version to Ubuntu 16.04 and Vivado 2016.4; Reorganized scripts to mutualize common content; Added section for how to create a Linux application using XSDK; Updated zedboard_oob_design.zip download address; Minor spelling corrections.

Chapter 1

Before starting

Please read this chapter carefully before starting, as it contains valuable information that you'll require all along this document.

1.1 Document purpose

This document is a tutorial describing how to build an Embedded Linux system for use on a ZedBoard development board. Following this procedure, you'll obtain an Embedded Linux running on a persistent file system, which you can use as a base for your further developments. This tutorial describes every needed step, from scratch to a running system, and provides scripts to automate most of the steps.

The configuration depicted here will be the very minimum. It is only intended to show you the global approach, letting you build you own personal system matching your needs once you've understood how to do. The only addition we do to the bare minimum system is to enable an Ethernet connection, in order to allow remote control of the board using SSH.

All needed tools are open-source and can be downloaded using the scripts provided with this tutorial, except for Xilinx Vivado which requires a license.

You should have obtained this document inside an archive containing the scripts and other files used in the procedure. If you don't have this archive, you can download it at this address: <https://homepages.laas.fr/cfoucher/drupal/zedboard-development>.

Each section in this tutorial presents how to build one specific file. Most file generation procedures will be presented using three different ways. For each step, choose the way that correspond the most to you.

These sections use the following names:

Default configuration using script

This first way uses dedicated scripts to run the default configuration without any user interaction. This is the fastest way to build the target, but you'll be dependent on parameters I chose for you.

Custom configuration using script

If any customization is available for a file generation, I provide this second way of generating it. It is still scripted, but opens a customization window which let you select your preferred configuration. This is the fastest way to choose your personal configuration, while still hiding the details on how things are done.

Manual procedure

Finally, this third way will present all required details to perform the full procedure manually. This is how you should proceed to learn the generation steps and re-use them outside this tutorial.

1.2 Disclaimer

The procedure depicted in this document is intended to help you build an Embedded Linux system whatever your knowledge of the Linux system is.

It should be relatively safe, but some steps require advanced manipulations on your host system, ZedBoard platform and related peripherals, including the SD card provided with ZedBoard.

The author or its institution cannot be held responsible for any harm caused to your host system, ZedBoard platform or any element manipulated by the following tutorial.

Moreover, the author can't be held responsible for English misspelling that is probably present in this document ;-). Please inform me of any error using my e-mail on the front page.

1.3 Tools revisions and OS

The procedure described in this document has been conducted using Xilinx Vivado 2016.4 on Fedora 22 Workstation 64 bits and Ubuntu 16.04 LTS 64 bits. Some steps of the procedure are closely related to tools version, so I cannot guarantee this tutorial will work using a different version of Vivado, a different Linux distribution, a different Fedora or Ubuntu version, or any other version of a tool used in this tutorial.

If your system does not run a native Linux OS, you can install for free a Linux virtual machine matching the above specifications.

1.4 Administrator privileges

Some of the manipulations described in this document require root privileges. To obtain them, use the `sudo` command in order to get administrator privileges for the current command.

These commands require you entering your password to work, and the current user must be registered in the `sudo` privileges list. If you're not familiar with the `sudo` command, you'll find all needed information about it online or by typing `man sudo` in a console.

1.5 Conventions and directories

The *host* system is the system used to generate the files, while the *target* system is the Embedded Linux system that will be generated.

Texts beginning by `$` are shell commands to be typed in the console, without the initial `$`. Other texts written using `monospaced` characters are to be typed as it.

Paths are displayed in `monospaced blue`. Paths ending with a `/` are directories (e.g., `${BASEDIR}/scripts/`), while the others are files (e.g., `${BASEDIR}/scripts/initialize_project.sh`).

In the current tutorial, we use the syntax `${DIRECTORY}` to refer to specific paths. Notably, `${BASEDIR}/` is the base working directory provided in the archive coming with this tutorial. If you obtained this file from the complete archive, the current file is located in `${BASEDIR}/doc/`. Please make sure the `${BASEDIR}/` path does not contain any space.

These kinds of paths are to be replaced by the user when doing the manipulation using matching directory. E.g., if the base directory is `/home/user/ZedBoard/`, user has to understand occurrences of `${BASEDIR}/scripts/` as `/home/user/ZedBoard/scripts/`. This can be achieved by manually replacing the variable in the command line, or by defining the variable content before typing the command. Using the previous example, defining the `${BASEDIR}` variable value would be achieved by typing the following command:

```
$ export BASEDIR="/home/user/ZedBoard/"
```

If you choose to use manual procedures, you're not required to work in the `#{BASEDIR}/` tree. Thus, these sections rather refer to another directory called `#{CUSTOMDIR}/` which can be any directory you want.

1.6 Projects

A project is a set of both hardware design and software environment generation. Projects are located in `#{BASEDIR}/projects/`. The base folder of project `#{project_name}` will be `#{BASEDIR}/projects/#{project_name}/`, and will be referred to as `#{PROJECT_ROOT}/`.

Variables `#{project_name}` and `#{PROJECT_ROOT}` can be set the same way as variable `#{BASEDIR}` (see Section 1.5).

1.7 Scripts and logs

Most of the manipulations depicted in this document provide scripts to automate the procedure. These scripts should work whatever the directory they are called from, but if anything fails, you should try `cd` to `#{BASEDIR}/scripts/` directory and use the `./<script_name>.sh` syntax.

These scripts usually create a log file in `#{PROJECT_ROOT}/logs/<script_name>.log`. You can check for this file after a script run for more information about errors that might have occurred.

1.8 Environment packages and libraries

First, Vivado is required, and the installation must provide SDK, which is facultative in the installation process, so make sure it is installed. 2016.4 version is used in this tutorial. As Vivado only supports 64-bit OS versions, the procedure depicted here assumes you run in such an environment.

Moreover, some libraries and tools are required on the host system to execute this procedure. The following sections list the required packages.

1.8.1 Fedora 22 Workstation

Here are the main packages required for a bare Fedora 22 Workstation installation, but some other may be required depending on your configuration.

- gcc
- gcc-c++
- git
- qt4-devel
- flex
- bison
- patch
- ncurses-devel
- openssl-devel
- parted
- glibc.i686

Before starting, you can use the following command in order to ensure all needed libraries are available on your system:

```
$ sudo dnf install gcc gcc-c++ git qt4-devel flex bison patch ncurses-devel \  
$ openssl-devel gparted glibc.i686
```

1.8.2 Ubuntu 16.04 LTS

Here are the main packages required for a bare Ubuntu 14.04 LTS installation, but some other may be required depending on your configuration.

- gcc
- g++
- git
- qt4-dev-tools
- flex
- bison
- patch
- libncurses5-dev
- libssl-dev
- gparted
- device-tree-compiler
- glibc:i386

Before starting, you can use the following command in order to ensure all needed libraries are available on your system:

```
$ sudo apt-get install gcc g++ git qt4-dev-tools flex bison patch libncurses5-dev \  
$ libssl-dev gparted device-tree-compiler  
$ sudo dpkg -i -force-architecture i386 glibc
```

Chapter 2

Additional technical information

The information depicted in this chapter is optional. You may refer to this section for specific needs.

2.1 Downloading required sources

Apart from the libraries required by the host, some sources are used in this procedure which are not provided in the archive (E.g. Linux kernel, boot loader generator, etc.). When a script requires one of these, it will ask the user for download on the first run.

If you just want to download some or all sources in order to use them manually or for any other reason, a dedicated script is available which can be used as follows:

```
$ ${BASEDIR}/scripts/download_sources.sh
```

And follow the instructions from there.

Note that this script will not download already existing sources. If for any reason you want to force source re-download, please delete or move existing source before launching this script.

2.2 ZedBoard boot process

To boot the ZedBoard on Linux using the SD card, you need the following elements:

- A first stage boot loader, in charge of early loading,
- A bitstream representing the FPGA fabric configuration,
- A boot loader, in charge of loading the Linux kernel,
- A Linux kernel,
- A device tree blob,
- A file system.

When you use the SD card to boot the ZedBoard, the boot process is as follows:

It begins with the First Stage Boot Loader (FSBL) which is in charge of the early boot process. The FSBL first reconfigures the FPGA fabric of the Zynq, and then launches the boot loader. Then the boot loader boots the Linux kernel.

From there, the Linux kernel searches for a file system. Usually, the file system is placed in a RAMdisk. But as the RAMdisk is loaded in RAM, its content disappears when board is turned off. This tutorial uses a persistent file system which is stored on the SD card.

Moreover, the Linux kernel needs the device tree blob to be aware of the hardware configuration surrounding the processor core.

As the FSBL, the bitstream and the boot loader are packaged together within a binary file, the following files are needed on the SD card:

A first partition containing:

- The binary file,
- The Linux kernel,
- The device tree blob.

And a second partition, containing the Linux file system.

2.3 Reinitializing the SD card to factory state

This section is only to be followed if you need to restore the SD card filesystem to its original state. It is not needed as part of this procedure, but it can be used to revert your SD card to its original state, as we modify its partition scheme during this procedure.

2.3.1 Restore partition scheme

Launch GParted. In the jumplist, select the entry matching your SD card (E.g., `/dev/sdc/`). On each existing partitions, right-click, and select **Unmount**. When all partitions are unmounted, select menu **Device** **>> Create Partition Table...**, make sure MS-DOS type is selected and click on **Apply**.

Right-click on the empty space, select **New**, choose a *fat32* file system and select **Add**. Select **Apply All Operations** and validate using **Apply**. When finished, exit GParted.

2.3.2 Restore files

The card partition scheme is now ready; all you need now is to put back the original files on it. These files are contained in the archive provided at this address: https://reference.digilentinc.com/_media/zedboard/zedboard_oob_design.zip. Download it, expand it, and copy the content of folder `sd_image/` back to the card.

Your card is now ready as new.

Chapter 3

Preparing the environment

3.1 Configuring the scripts

First, file `${BASEDIR}/scripts/user-config/environment.sh` must be edited. This file contains configuration variables that are used by other scripts. Each variable purpose is described in the file, and must be set according to your host system configuration. Some may require a little bit of Linux knowledge, such as mount directories, so be careful when you set them.

Please take some time to ensure variables in this files have a correct value, or some scripts used in this document may not work.

3.2 Creating the project

First, choose a name for the project. This name must not contain any space character. Choosing a name ensures you may have various projects at the same time, with different configurations.

This name will have to be used in replacement of all `${project_name}` occurrences in this document.

To begin, we must create a placeholder for the project. To do so, open a shell and type:

```
$ ${BASEDIR}/scripts/initialize_project.sh ${project_name}
```

This will create the root folder of your project, as well as some sub-folders and files that will be used in the following steps.

Chapter 4

Hardware layout and low-level software

In this chapter, we will generate the lower part of the system: the hardware design and the low-level software running before Linux takes on.

4.1 Creating a base hardware design

The hardware layout depends on what you want to do with your design. Only you can know what you want or don't want to instantiate on the reconfigurable fabric of your Zynq device.

Thus, we provide here a guide to build a simple design for the sole purpose of this tutorial, containing the very minimum.

For further information about creating a design using Vivado, please refer to Xilinx user guides.

4.1.1 Default configuration using script

Open a shell and type:

```
$ ${BASEDIR}/scripts/generate_bitstream.sh ${project_name}
```

When done, the bitstream should be located in `${PROJECT_ROOT}/output/hardware_design/bitstream.bit`.

4.1.2 Manual procedure

First, open Vivado and select `Create New Project`.

Enter project name `hardware_design` and browse to `${CUSTOMDIR}/`. Make sure `Create project subdirectory` is checked, and click `Next`.

Select `RTL Project`, check `Do not specify sources at this time`, and click `Next`. Select `Boards`, highlight the ZedBoard in the boards list, and click `Next` then `Finish`.

Once the project is loaded, click `Create Block Design` under the *IP Integrator* section of the left menu bar, and click `OK` in the pop-up window.

You now have a blank design. Hit the `Add IP` button, and select `ZYNQ7 Processing System` from the list. Double-click on the processing system block, and click on `Presets` and select `ZedBoard`. Close the window using the `OK` button.

At this point, a green banner should be displayed, click on `Run Block Automation`. Select `OK` on the pop-up window.

Finally, we need to connect the system clock. Click on the wire going out the `FCLK_CLK0` out pin, and drag a connection to the `M_AXI_GPO_ACLK` input pin.

Save the design, and close the block design.

Right-click the `design_1` item under *Design sources* of the *Project Manager* tab, and select `Generate Output Products...` and hit `Generate`. Wait until generation is over.

Right-click again on `design_1_i - design_1`, then select `Create HDL Wrapper...`. In the pop-up window, select `Let Vivado manage wrapper and auto-update` and hit `OK`.

Finally, generate the bitstream by clicking on `Generate Bitstream` under the *Program and Debug* section of the left menu bar. If a dialog warns you about missing synthesis and implementation, select `Yes`.

This procedure will probably last for a couple minutes. You can follow the completion state in the *Design Runs* tab of the bottom section.

When generation is over, `write_bitstream Complete!` should be displayed, and a pop-up window may appear depending on your settings. If so, just hit `Cancel`.

Hardware generation is over, and the bitstream file is located at `${CUSTOMDIR}/hardware_design/hardware_design.runs/impl_1/design_1_wrapper.bit`. In order to prepare the next steps, select `File` `>>` `Export` `>>` `Export Hardware` and validate using `OK`.

We now need to generate the device tree, so you may not want to close Vivado immediately.

4.2 Generating the device tree

4.2.1 Downloading the device tree generator

The first time you reach this step, you need to download some sources. Other projects can use the same files, so if you already did it, there is no need to repeat this step.

We will use the generic script `${BASEDIR}/scripts/download_sources.sh` to download the device tree generator. Launch the script using the following command:

```
$ ${BASEDIR}/scripts/download_sources.sh
```

From there, make sure you answer yes (“y”) when asked for device tree generator download. If you’re in a hurry, you may want to skip the other downloads (“n”), as they will be automatically performed when required.

4.2.2 Generating the device tree

Due to scripting issues with SDK, I do not provide a script for this part, so here is the manual way of doing things.

First, you need to open the previously created Vivado project. If you used the script to generate the hardware design, the project is located in `${PROJECT_ROOT}/hardware_design/`.

In Vivado GUI, launch XSDK by selecting `File` `>>` `Launch SDK`, and validating.

You’re now in the Xilinx Software Development Kit (XSDK) environment. We need to tell it where to find the device tree generator. Select `Xilinx Tools` `>>` `Repositories`.

Then, select the `New...` button, either in the “local” area for this project only or in “global” if you want to re-use it later. Browse to `${BASEDIR}/resources/device_tree/device_tree-generator/`, and then select `Open`.

Now that the tool is known by the environment, select `File` `>>` `New` `>>` `Board Support Package`. In the **Board Support Package OS** section, select `device_tree` and validate using `Finish`.

In the settings window which opens automatically select `Overview` `>>` `device_tree`. We need to provide the `bootargs` variable, which represent the Linux kernel boot arguments. Set the following value:

```
console=ttyPS0,115200 root=/dev/mmcblk0p2 rw rootfstype=ext4 earlyprintk rootwait
```

Moreover, if you plan on using an Ethernet connection, you may add the **ip** argument to automatically connect at startup. If your board will be plugged on a DHCP-managed network, add the following to the bootargs list:

```
ip=::::eth0:dhcp
```

You can also setup a static IP address using the **ip** boot argument, but it will be very dependent on your network configuration, so it is recommended to get some insight online about how this argument is to be used.

Once done, validate using **OK** and wait for generation.

The output file is then `${PROJECT_ROOT}/hardware_design/hardware_design.sdk/device_tree_bsp_0/system.dts`. Copy this file to the `${PROJECT_ROOT}/output/device_tree/` folder. Copy as well files `zynq-7000.dtsi` and `skeleton.dtsi`, which are needed as they are referenced in the device tree file.

You're done with the device tree, but you may not want to exit XSDK as the FSBL generation will continue with it.

4.3 Generating the first stage boot loader

Due to scripting issues with SDK, I do not provide a script for this part, so here is the manual way of doing things.

In XSDK, select **File** **>>** **New** **>>** **Application Project**. Set project name *FSBL*, make sure *OS Platform* is **standalone** and *Language* is **C**, and then click **Next**. Select template **Zynq FSBL** and then **Finish**.

Right-click on the **FSBL** project in the **Project explorer** section, and select **Build Configurations** **>** **Set Active** **>>** **Release**. Right-click again and select **Clean Project**. If auto-build is disabled, you want to continue by clicking on **Build Project**.

You can follow the generation process in the **Console** tab at the bottom of the screen. Once generation is over, the output file is `${PROJECT_ROOT}/hardware_design/hardware_design.sdk/FSBL/Release/FSBL.elf`. Copy this file to the `${PROJECT_ROOT}/output/first_stage_boot_loader/` folder.

You may not want to exit XSDK, as the step after the next (which is quite fast) will require it.

4.4 Generating the bootloader

4.4.1 Default configuration using script

If the board has to be used on an Ethernet network, you'll need to define a MAC address for the board. This address will be asked as part of the following script. If you have no use of the network, just answer "n" when asked for MAC address.

Open a shell and type:

```
$ ${BASEDIR}/scripts/generate_boot_loader.sh ${project_name}
```

When done, the boot loader should be located in `${PROJECT_ROOT}/output/boot_loader/u-boot.elf`.

4.4.2 Manual procedure

The procedure consists in downloading U-Boot, configuring it, and generating the boot loader.

First, download U-Boot using the following commands:

```
$ git clone https://github.com/Xilinx/u-boot-xlnx.git ${CUSTOMDIR}/boot_loader
$ git -C ${CUSTOMDIR}/boot_loader checkout xilinx-v2016.4
```

We then need to edit the default configuration to match our needs. Open file `${CUSTOMDIR}/boot_loader/include/configs/zynq-common.h`. All lines numbers in the following are given for the specific Vivado version supported in this guide, and may differ if using a different version.

First, we don't use a ramdisk image. Search for section beginning with `sdboot=` (line 259). We must remove the line `"load mmc 0 ${ramdisk_load_address} ${ramdisk_image} && " \` (line 264). Then, edit the next line to remove the `${ramdisk_load_address}` part, replacing it by a `'-'` character. The line should now look like this:

```
"bootm ${kernel_load_address} - ${devicetree_load_address}; " \
```

We also need to edit `fdt_high` value, from `x20000000` to `x19000000` (line 221).

Then, if the board has to be used on an Ethernet network, you'll need to define a MAC address for the board. To do so, edit the line `ethaddr=00:0a:35:00:01:22` (line 206) to introduce the MAC address desired for the board, and save the file. If you have no use of the network, just ignore this step.

The following steps require your cross-compilation environment to be set. If you have a standard Xilinx configuration, you can type the following in the console:

```
$ export PATH=/opt/Xilinx/SDK/2016.4/gnu/arm/lin/bin:$PATH
$ export CROSS_COMPILE=arm-xilinx-linux-gnueabi-
```

Then configure U-boot using the following commands:

```
$ make -C ${CUSTOMDIR}/boot_loader distclean
$ make -C ${CUSTOMDIR}/boot_loader zynq_zed_config
```

Finally, generate the boot loader image:

```
$ make -C ${CUSTOMDIR}/boot_loader
```

If everything went well, the output file containing the boot loader is `${CUSTOMDIR}/boot_loader/u-boot`. It is recommended copying this file in some other place, using name `u-boot.elf`. Indeed, the missing extension is needed in the following step.

4.5 Generating the Binary file

Due to scripting issues with SDK, I do not provide a script for this part, so here is the manual way of doing things.

In XSDK, select `Xilinx Tools` `>>` `Create Boot Image`. Make sure target architecture is Zynq, and check `Create a new bif file...`.

In field `Output BIF file path:`, click on `Browse` and navigate to `${PROJECT_ROOT}/binary_generation/boot.bif`. This `.bif` file can be re-used to generate the binary file if you update any of the files used for the binary generation.

In `Output path` field, type `${PROJECT_ROOT}/output/bin/boot.bin`.

If the `Boot image partitions` list already contains elements, clean it by selecting them one by one and clicking `Delete`. In `Boot image partition`, click on `Add`, add file `${PROJECT_ROOT}/output/`

`first_stage_boot_loader/FSBL.elf` and make sure *bootloader* is the partition type. Validate using .

Click on again, and select the bitstream. If you followed this tutorial for hardware generation, it should be available at `${PROJECT_ROOT}/output/hardware_design/bitstream.bit`. If not, the bitstream should be in your Vivado folder, under the `<project>.runs/impl_1/` folder. Make sure the partition type is *datafile*, and validate.

Select a third time, and then choose `${PROJECT_ROOT}/output/bootloader/u-boot.elf`, with a *datafile* partition type.

Finally, select .

The file `${PROJECT_ROOT}/output/bin/boot.bin` has been created.

You may now exit XSDK and Vivado.

Chapter 5

Generating Linux

This chapter concerns the Linux kernel and file system generation. This is the software part that can be customized and tailored to your needs.

5.1 Generating the kernel

The kernel generation requires the boot loader. Please make sure you already generated the boot loader before running into this step (see Section 4.4).

5.1.1 Default configuration using script

Open a shell and type:

```
$ ${BASEDIR}/scripts/generate_linux_kernel_default.sh ${project_name}
```

When done, the kernel image is located in `${PROJECT_ROOT}/output/linux_kernel/uImage`.

5.1.2 Custom configuration using script

Open a shell and type:

```
$ ${BASEDIR}/scripts/generate_linux_kernel_custom.sh ${project_name}
```

After source download (if needed) and copy, a configuration window opens.

From there, set the configuration as you want. I personally set the following configuration to make sure system is very light by removing elements I don't need, but adapt depending on your needs:

General setup	→ Initial RAM filesystem and RAM disk (initramfs/initrd) support	→ NO	
Device drivers	→ Block devices	→ RAM block device support	→ NO
—	→ Network device support	→ Wireless lan	→ NO
—	→ Sound card support	→ NO	
Networking support	→ CAN bus subsystem support	→ NO	
—	→ Wireless	→ NO	
File systems	→ Network file system	→ NO	

Save, exit, and wait for process completion. When done, the kernel image is located in `${PROJECT_ROOT}/output/linux_kernel/uImage`.

5.1.3 Manual procedure

The procedure first consists in getting the latest kernel version, and regressing to the correct branch, as follows:

```
$ git clone https://github.com/Xilinx/linux-xlnx.git ${CUSTOMDIR}/linux_kernel
$ git -C ${CUSTOMDIR}/linux_kernel checkout xilinx-v2016.4
```

The following steps require your cross-compilation environment to be set. If you have a standard Xilinx configuration, you can type the following in the console:

```
$ export PATH=/opt/Xilinx/SDK/2016.4/gnu/arm/lin/bin:$PATH
$ export CROSS_COMPILE=arm-xilinx-linux-gnueabi-
```

Finally, configure the kernel with ZedBoard defaults and launch the configuration GUI:

```
$ make -C ${CUSTOMDIR}/linux_kernel distclean
$ make -C ${CUSTOMDIR}/linux_kernel ARCH=arm xilinx_zynq_defconfig
$ make -C ${CUSTOMDIR}/linux_kernel ARCH=arm xconfig
```

From there, set the configuration as you want (for a configuration example, see Section 5.1.2), save then exit the GUI.

The final make requires using some tools generated by the boot loader make procedure. Use the following to make the system know where these tools are:

```
$ export PATH=${CUSTOMDIR}/boot_loader/tools:$PATH
```

Finally, build the image using the following command:

```
$ make -C ${CUSTOMDIR}/linux_kernel ARCH=arm UIIMAGE_LOADADDR=0x8000 uImage
```

If everything went well, the output file containing the kernel is `${CUSTOMDIR}/linux_kernel/arch/arm/boot/uImage`.

5.2 Generating the device tree blob

You must now generate the device tree blob, which will be used by the kernel to know the system map. This part of the procedure requires that you already generated the device tree (see Section 4.2.2) and the Linux kernel (Section 5.1).

5.2.1 Default configuration using script

Type the following in a shell:

```
$ ${BASEDIR}/scripts/generate_device_tree_blob.sh ${project_name}
```

The output file is then `${PROJECT_ROOT}/output/dtb/devicetree.dtb`.

5.2.2 Manual procedure

We assume here the device tree is located at `${CUSTOMDIR}/system.dts` along with other required files (see Section 4.2.2). Open a shell, and type

```
$ ${CUSTOMDIR}/linux_kernel/scripts/dtc/dtc -O dtb -I dts \  
$ -o ${CUSTOMDIR}/devicetree.dtb ${CUSTOMDIR}/system.dts
```

Output file is then `${CUSTOMDIR}/devicetree.dtb`.

5.3 Generating the file system

5.3.1 Default configuration using script

Open a shell and type:

```
$ ${BASEDIR}/scripts/generate_file_system_default.sh ${project_name}
```

When done, the generated file system is located in `${PROJECT_ROOT}/output/file_system/rootfs.ext4`.

5.3.2 Custom configuration using script

Open a shell and type:

```
$ ${BASEDIR}/scripts/generate_file_system_custom.sh ${project_name}
```

After source download (if needed) and copy, a configuration window opens.

From there, set the following base configuration:

Target options	→ Target Architecture	→ ARM (little endian)
—	→ Target Architecture Variant	→ cortex-A9
—	→ Enable NEON SIMD extension support	→ YES
—	→ Floating point strategy	→ NEON
Toolchain	→ C library	→ glibc
—	→ Kernel Headers	→ 4.6
System configuration	→ System hostname: → <i>(set the name you want)</i>	
—	→ Enable root login with password	→ Root password → <i>(choose a password)</i>
—	→ Run a getty (login prompt) after boot	→ TTY port → ttyPS0

Filesystem images	→ ext2/3/4 root filesystem	→ YES
—	—	→ ext2/3/4 variant → ext4
—	→ tar the root filesystem	→ NO

I personally add the following configuration for my needs, but you should adapt it depending on your needs. Notably, if you require communicating with the board using network, you may want to enable OpenSSH:

Toolchain	→ Enable C++ support	→ YES
Target packages	→ Networking applications → openssl	→ YES

Save and close the window, and wait for process completion. When done, the generated file system is located in `${PROJECT_ROOT}/output/file_system/rootfs.ext4`.

5.3.3 Manual procedure

The procedure consists in downloading Buildroot, configuring it, and generating the file system.

First, download Buildroot using the following commands:

```
$ git clone git://git.buildroot.net/buildroot ${CUSTOMDIR}/file_system
$ git -C ${CUSTOMDIR}/file_system checkout 2016.11.2
```

The following steps require your cross-compilation environment to be set. If you have a standard Xilinx configuration, you can type the following in the console:

```
$ export PATH=/opt/Xilinx/SDK/2016.4/gnu/arm/lin/bin:$PATH
$ export CROSS_COMPILE=arm-xilinx-linux-gnueabi-
```

Then, launch the configuration tool using the following command:

```
$ make -C ${CUSTOMDIR}/file_system xconfig
```

Set the default configuration as indicated in 5.3.2, and add/remove what you need. Save, close, and type:

```
$ make -C ${CUSTOMDIR}/file_system
```

If everything went well, the output file containing the file system is `${CUSTOMDIR}/file_system/output/images/rootfs.ext4`.

Chapter 6

Preparing the board

SAVE ALL DATA CONTAINED ON THE SD CARD BEFORE PROCEEDING. THE OPERATION DEPICTED HERE WILL CAUSE ALL DATA ON THE SD CARD TO BE LOST.

For the following, you must first know what is the driver file representing your SD card. This is highly configuration-dependent so we cannot provide a default value. If you don't know the value for your computer, you can plug the card in and use a disk utility to check your local disks and identify your reader.

6.1 Partitioning the SD card

If your SD card never has never been partitioned, you'll need to prepare it first. If you already partitioned the card as part of this tutorial, you can skip this section.

As a scripted way of partitioning the card could damage your computer if wrong values are provided for drive driver, we rather provide here a manual way of doing it.

Anyway, beware that you select the right drive in the following procedure!

Launch GParted, and select the entry matching your SD card from the jumplist. First unmount all partitions on your card (right click then **Unmount**). Then, select menu **Device** **>** **Create Partition Table...**, make sure MS-DOS type is selected and select *Apply*.

Right-click on the empty space, select **New**, choose a *fat32* file system, a size of 1024 MiB and select **Add**. Right-click again on the empty space, select **New**, choose an *ext4* file system, enter *linux_fs* as the label and select **Add**.

Select **Apply All Operations** and validate using **Apply**. When done, exit GParted, remove the SD card from reader, and then plug it in again before next step.

6.2 Copying the file system on the card

6.2.1 Using the script

To use the script, first make sure the `$SDMOUNTDIR` variable in the `/${BASEDIR}/scripts/user-config/environment.sh` is correctly set, representing your SD card mount point.

Open a shell and type:

```
$ sudo ${BASEDIR}/scripts/copy_file_system_to_memory_card.sh ${project_name}
```

6.2.2 Manually

In the following, we assume the mounting point of the Linux partition of your SD card is `${SDLINUXMOUNT}/`, and that directory `/mnt/` is empty and can be used for temporary mount. Moreover, we assume the generated file system is `${CUSTOMDIR}/rootfs.ext4`.

We first need to temporarily mount the generated file system as follows:

```
$ sudo mount -t ext4 -o loop ${CUSTOMDIR}/rootfs.ext4 /mnt
```

Then we do the copy:

```
$ sudo cp -rf /mnt/* ${SDLINUXMOUNT}
```

Finally, we unmount the temporary mount point as follows:

```
$ sudo umount -l /mnt
```

6.3 Copying the system files

Finally, open the 1 GiB fat32 partition of the SD card, and copy the following files onto it: `${PROJECT_ROOT}/output/bin/boot.bin`, `${PROJECT_ROOT}/output/dtb/devicetree.dtb` and `${PROJECT_ROOT}/output/linux_kernel/uImage`.

You can now eject the SD card.

Chapter 7

Working on the board

7.1 Hardware configuration

Before booting, the board jumpers must be in the following position:

- JP7: SIG \Leftrightarrow GND
- JP8: SIG \Leftrightarrow GND
- JP9: 3V3 \Leftrightarrow SIG
- JP10: 3V3 \Leftrightarrow SIG
- JP11: SIG \Leftrightarrow GND

7.2 On the first boot

Plug the SD card onto the ZedBoard. Connect the MiniUSB cable to the USB UART port, and plug it to your computer. Open a terminal on your computer (115200 bauds, 8 bits, no parity), then start the board.

If you obtain a prompt indicating *zynq-uboot>*, type in *boot* and validate. If the prompt asks you for a login, you're already booted.

Login using account **root**, then enter the password you choose. If you used the default configuration script to generate the file system, the default password is **root**.

7.2.1 SSH configuration

If you defined no root password, the first thing to do is to set one as OpenSSH requires one. Type:

```
$ passwd
```

Then enter a password twice.

We then require a few tricks to allow SSH connection as root. If you only require to connect as a standard user, the following is not required.

To allow root connection using SSH, type:

```
$ vi /etc/ssh/sshd_config
```


Navigate down using the arrows until you find the following line:

```
#PermitRootLogin prohibit-password
```

Press *inser* to switch to edition mode, and change the line for:

```
PermitRootLogin yes
```

Then press *escape* and type `:wq` then *return* to save file.

If you defined no password when generating the file system, but added it manually using the `passwd` command, this is not over! When connecting using OpenSSH, you'll always be told your password to be expired. To solve that, do the following:

```
$ vi /etc/shadow
```

The first line should look like that: `root:[...]:0:0:[...]` We need to change the first “0” to anything else, e.g. to a “10”, to look like that: `root:[...]:10:0:[...]` Press *inser*, move to the “0”, and press “1” to change “0” to “10”. Press *escape* then type `:wq` then *return*.

Your Linux is now OK for a SSH connection.

To make sure modifications are written to the SD card, and depending on your needs, type:

```
$ reboot
```

Or:

```
$ poweroff
```

Remember to always use the `poweroff` command before turning off the board, as Linux uses a buffer which must be flushed before powering off.

You're now ready to use an Embedded Linux environment on your ZedBoard.

7.3 Using Xilinx SDK to create an application

If required, you can use Xilinx SDK to create an application for your platform using cross-compilation. To do so, re-open th SDK project you used to generate the software files (see Section 4.2.2 for how to open it).

Select `File >> New >> Application Project`, enter a name, set OS Platform to *linux*, and click `Next`. Choose either an empty application or a Hello World project.

It seems there can be an incorrect configuration of the tool used to generate the executable. To check it, right-click on the application project you created, and select `C/C++ Build Settings`. Then, select *ARM v7 Linux gcc assembler*, and check that the gcc utility name is *arm-xilinx-linux-gnueabi-gcc*. If not, change it to match that name, and do the same for items *ARM v7 Linux gcc compiler* and *ARM v7 Linux gcc linker*. Also change *ARM v7 Linux Print Size* command to *arm-xilinx-linux-gnueabi-size*.

When done, try a very simple application such as the Hello World sample by downloading it to the target, e.g. using *sftp* if you configured a SSH connection, or by manually copying it to the file system by plugging the SD card to the host computer.

That's all, folks!