



IO2-A2: DUAL EDUCATIONAL PACK



3D2ACT

3D2ACT:

FOSTERING INDUSTRY 4.0 AND 3D TECHNOLOGIES
THROUGH SOCIAL ENTREPRENEURSHIP: AN INNOVATIVE
PROGRAMME FOR A SUSTAINABLE FUTURE

**Author (s): NATIONAL CENTER FOR SCIENTIFIC RESEARCH "DEMOKRITOS" &
UNIVERSITY OF CRETE**

The European Commission's support for the production of this publication does not constitute an endorsement of the contents, which reflect the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

IO2-A2: DUAL EDUCATIONAL PACK

PROJECT INFORMATION

PROJECT ACRONYM:

3D2ACT

PROJECT TITLE:

FOSTERING INDUSTRY 4.0 AND 3D TECHNOLOGIES THROUGH SOCIAL
ENTREPRENEURSHIP: AN INNOVATIVE PROGRAMME FOR A SUSTAINABLE FUTURE

PROJECT NUMBER:

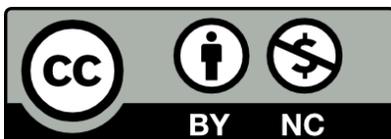
2020-1-EL01-KA202-078957

WEBSITE:

<https://3d2act.eu/>

CONSORTIUM: PARTNER LIST

- NATIONAL CENTER FOR SCIENTIFIC RESEARCH "DEMOKRITOS" (GREECE)
- EUROPEAN DIGITAL LEARNING NETWORK (Italy)
- POLITEKNIKA IKASTEGIA TXORIERRI S.COOP (Spain)
- A & A EMPHASYS INTERACTIVE SOLUTIONS Ltd (Cyprus)
- STICHTING INCUBATOR (Netherlands)
- REGIONAL DIRECTORATE EDUCATION OF CRETE (Greece)
- UNIVERSITY OF CRETE (Greece)



Attribution-NonCommercial
4.0 International ([CC BY-NC 4.0](https://creativecommons.org/licenses/by-nc/4.0/))



LESSON PLAN 1.3.2

TEACHING MODULE 1.3.2	
Chapter 1.3	3D Printer Basics and Preparation for the First Print
Equipment (if needed)	PC with Internet connection, 3D Printer, 10g of Filament
Duration	1.5 Hours
Short description	In this Worksheet students will learn how to perform their first 3D Print
Learning Outcomes	What is the Preheat process and why it is needed
	What is and how to perform Bed Leveling
	Why Bed Leveling is important for the success of a 3D print
	How to perform basic troubleshooting based on the First Layer
	How to deal with the most common 3D printing problems
	Imagination, Problem Solving, Critical Thinking, Adaptability, Experimentation, Improvisation, Patience
Activities	
Activity 1	Exercise 1.3.2.1
Aim of the Activity	Familiarize students with the steps they need to take in order to load a Filament to the 3D Printer
Duration	15 Minutes
Type of Activity	Worksheet
Teaching Objectives	How to preheat the Hot End of the Printer, Preparation of the Filament, Loading of the Filament
Resources	Worksheet 1.3.2 /Introduction, Exercise 1
Activity 2	Exercise 1.3.2.2



Aim of the activity	To familiarize students with the process of Bed Leveling and the reasons why this step is really important
Duration	15 Minutes
Type of Activity	Worksheet
Teaching Objectives	Understand the reasons Bed Leveling is an important step of the 3D Printing process, Measure and adjust the distance between the Nozzle and the Building Surface
Resources	Worksheet 1.3.2 / Exercise 2
Activity 3	
Activity 3	Exercise 1.3.1.3
Aim of the Activity	Students make their first 3D printing job
Duration	60 Minutes
Type of Activity	Worksheet
Teaching Objectives	Make the necessary settings in the Slicing Software, export the G-Code file and load it to the printer, Troubleshooting solutions for fixing the most common problems and failures
Resources	Worksheet 1.3.2 / Exercise 3, Whistle.stl
Further Reading	
	https://www.simplify3d.com/support/print-quality-troubleshooting/poor-bridging/ https://www.simplify3d.com/support/print-quality-troubleshooting/not-sticking-to-the-bed/ https://all3dp.com/2/3d-printing-for-beginners-all-you-need-to-know-to-get-started/ https://all3dp.com/2/3d-printer-bed-leveling-step-by-step-tutorial/ https://www.dobot.cc/resource/10-common-3d-printing-problems-troubleshooting.html



Activity Worksheet 1.3.2 (Teacher Version)

Level 1 (Novice Level: Basic Competences)

Chapter 1.3: 3D Printer Basics and Preparation for the First Print

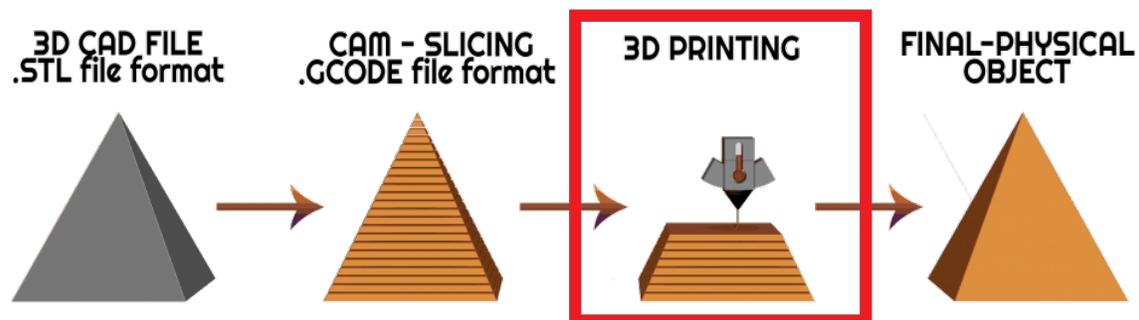
Activity Worksheet 1.3.2

In this Activity worksheet we will learn how to prepare the 3D printer and how to perform our first print. More specifically, we are going to learn:

- How to load the Filament on the printer
- What Bed Leveling is and why it is important
- How to properly Level the Bed
- How to identify the necessary corrections to be made based on the printing of the First Layer
- How to deal with the most common 3D printing problems.

Introduction

Once we save/export the instructions from the Slicer (i.e. the G-code file), we are ready to transfer these instructions to the printer (usually through a SD card) and finally begin the process of 3D printing.



However, before starting the 3D print job, a couple of things need to be done on our 3D printer:

- Loading the Filament
- Bed Leveling



Exercise 1: Loading the Filament

The **extruder needs to be ready** to extrude filament before printing begins. The loading process begins by **heating the hot end (nozzle) to the filament's molten temperature** (anything above 185 °C, depending on the filament), and then loading the filament into the heated extruder.

In order to heat the hot end you should enable the **Preheat Process** from the printer's menu. Where exactly to find this option depends on the specific 3D Printer's menu. In all cases, you can enable different Preheat Process options **based on the material** you want to load. The preheat temperatures are usually 10-15 degrees lower than the actual printing temperatures.



In the picture above, you can see examples of Preheat options for different materials. The first number states the required **hot end temperature**, while the second number states the required **Bed temperature**.

After enabling the Preheat option, you should wait for the printer's hot end to reach the required temperature and after that, you are ready to load the Filament.

Note that you can keep track of the hot end's temperature, in real time, on the Printer's display. It usually can be found at the bottom of the screen.



Before loading the filament, it is important to trim the filament's end and form a sharp tip (a side cutter works the best). Then you can load it to the extruder and push it until you feel that the filament has **reached the hot end** and a small amount of plastic has been extruded.

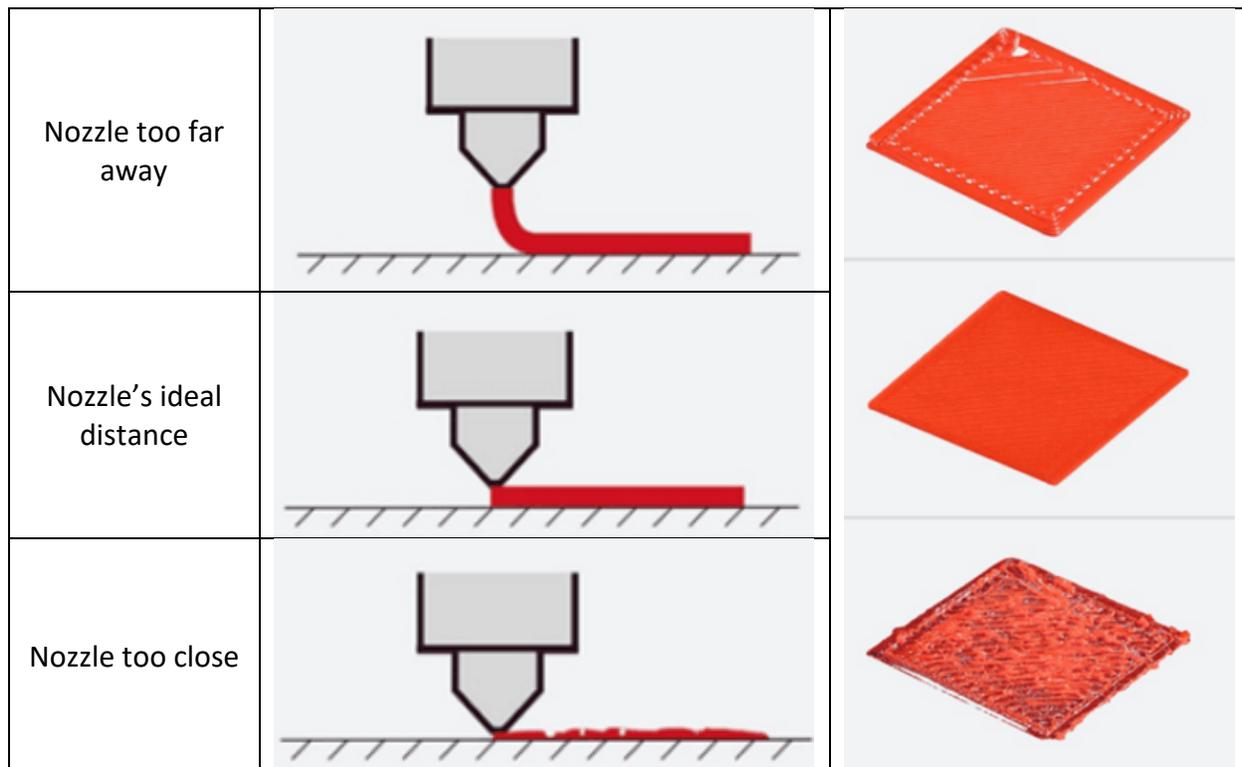


Note that the filament loading process may vary depending on the 3D printer you have available. Always refer to the printer manual for specific instructions for the process. It is also useful to watch some videos on the internet with instructions and tips for your printer. For example you can search “How to load filament on *Model name*”.

After the filament is loaded and you can cooldown the printer by selecting the cool down option or just shut down the printer from the power supply.

Exercise 2: Bed Leveling

In order for the printer to deposit filament and build the object successfully, **the build platform must be level**. Depending on the machine, leveling can be either **manual or automatic**. Bed leveling is **very important** because, if the print bed is too far away from the nozzle, **the first and the most important layer will not stick to the bed**, causing the print to **fail**.



Before we start the leveling process, we have to be sure that both the **Nozzle** and the **Bed** surface are **clean**. In order to clean the Nozzle, the only thing you need to do is to use a dry cotton cloth to wipe any material that has been “stuck” around the nozzle from an earlier print job. To clean the Bed surface, in case it is a removable one, cleaning it in the sink with ordinary dish soap and lukewarm water works quite well. For a non-removable build plate, you can use isopropyl alcohol on a paper towel or cloth. After that, dry it with a soft cloth. Cleaning the bed surface (preferably



before every print) is important because when we touch the build plate with our bare hands, some oil remains on it and can make it difficult for the model to stick properly.

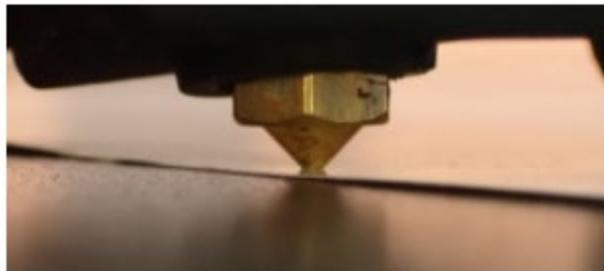
Now we are ready to level the Bed surface by following the steps below:

1. Position your Nozzle

To do this, simply reset your print head position by selecting the **“Home Option”** on your printer Menu. This will send the nozzle to location (0, 0, 0) in the build volume's space, as determined by the machine. The third 0 is significant because it specifies the Z coordinate or height, which is where your machine will print the first layer. Then, **turn off (disable) the stepper motors** so that you can freely move (by hand) the printhead in the XY plane. Again, your printer's Menu should include this choice. **Usually these options are under the “Preparation” menu of the printer. In order to be sure where to find these options read the manual of your printer.**

2. Measure and Adjust

The 3D printer bed should be calibrated so that it is **as level as possible** in all **four corners** and in the **middle**. In theory, the nozzle should only make contact with the build plate at these points, which is why we aim to sandwich something thin between them - like a piece of paper. Note: For more accurate results, instead of a piece of paper, you can use a metallic feeler gauge with a 0.2mm thickness.



The "gap" is correct and consistent at all five spots if you reach the **point right before experiencing resistance** while sliding the paper around. Place your paper **between the nozzle tip and the print bed**, then move the printhead to a corner of the bed. Adjust the **closest leveling screw** to tighten the spacing if there is no resistance when moving the paper back and forth between the nozzle and the bed. Adjust the screw so you can **feel a tiny drag** from the nozzle and 3D printer bed while sliding your paper.

Carry on in the same manner with the remaining corners. When the corners are adjusted correctly, move the print head to the center of the build plate and verify that the same resistance is experienced.



Finally, double-check each corner and the center, as your changes may have had an impact on other areas. If this is the case, continue the process until all five points have been leveled and you are ready to go. Usually you need check every point at least 2 or 3 times to make sure that everything is level.

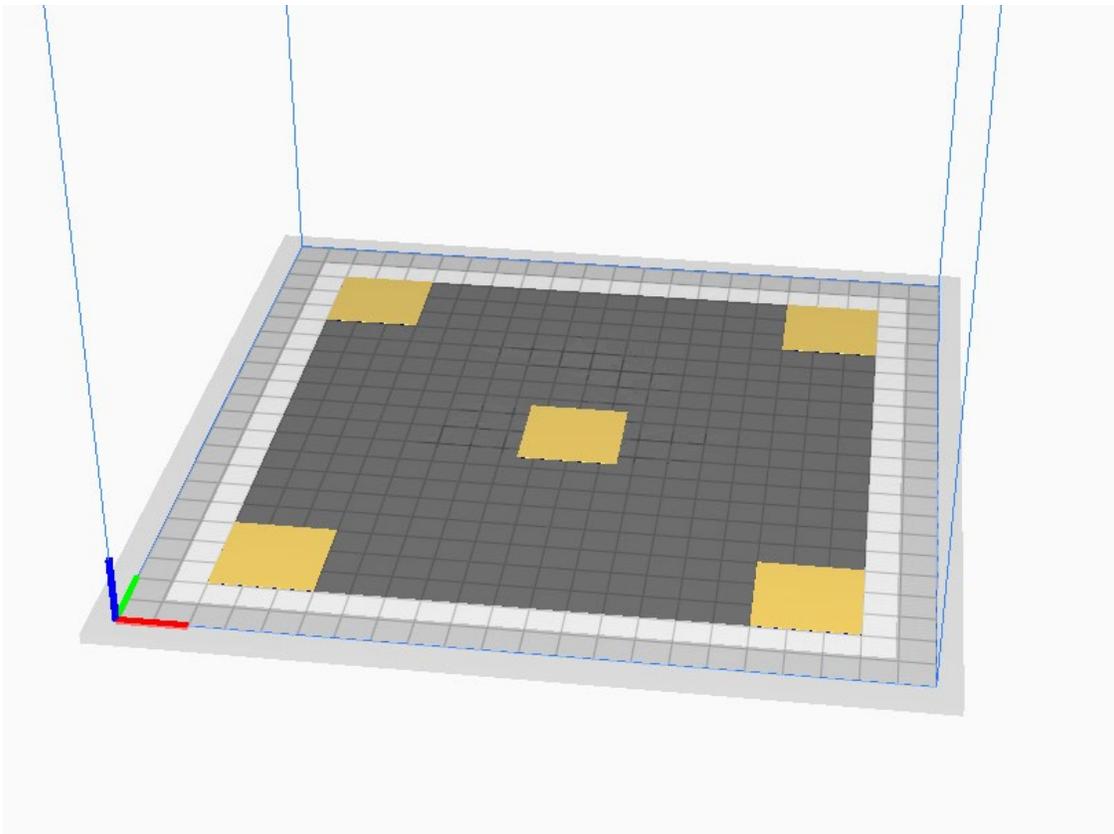
Note that bed leveling is a process that you need to do once in a while. Most of the problems in 3D printing are caused due to not having a level print surface, which is why the next step is really important.

3. Inspect the First Layer

When you start a new 3D print job you should keep an eye on the first layer in order to verify successful bed leveling. If successful, the first layer should look more or less the same over the whole surface. If not, you may need to cancel the print and further adjust your level.

Note that bed leveling is something that you don't have to perform regularly. Usually one time, during the first setup of the printer, is enough for a long time. You have to repeat bed leveling only when there is a problem and you have to repeat it. However, for educational reasons, we recommend even if we know that the Bed is Leveled, to have the process performed by the students.

Note: There are several bed leveling test prints that you can perform in order to verify how level is your build plate. One of these tests (bed leveling squares) is shown below. Most slicers (e.g. Cura) offer such test prints or, alternatively, you can easily find such tests online.

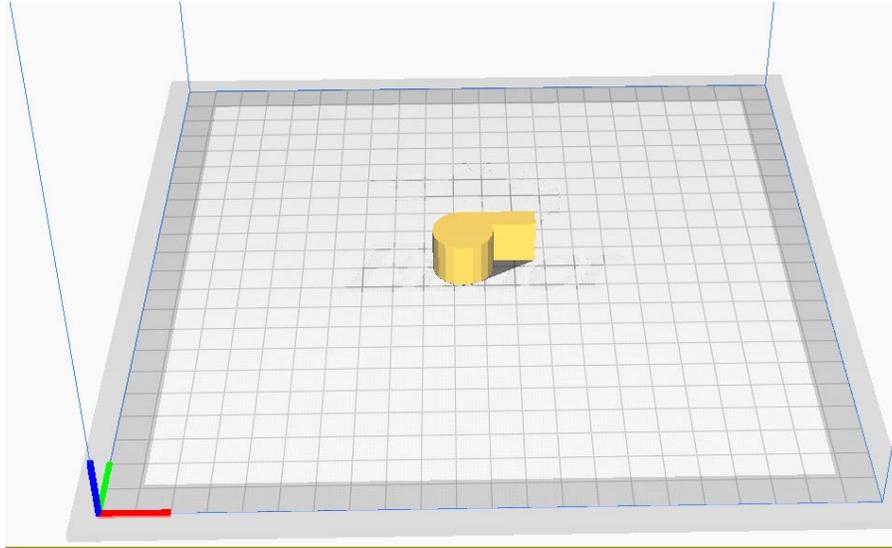


Exercise 3: The First Print

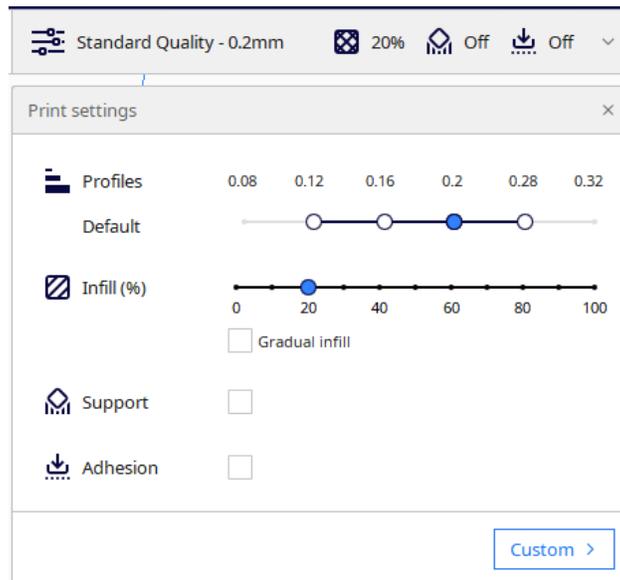
Now our 3D printer is ready to start our first print. In this section we will print, as an example, the whistle that we designed in chapter 1.2.

Step 1: Open Ultimaker Cura software and import the Whistle model (i.e. the .stl file). Make all the adjustments that are needed in order to slice it. Select the orientation on the build plate, the printing profile, decide whether to use support structures or not etc. Finally export the G-code file into an SD card (with the “Save to Removable Drive” or “Save to Disk” button).

The most suitable orientation for this model can be seen in the picture below.



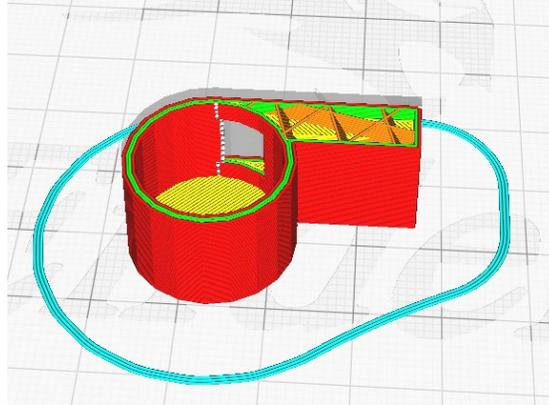
Recommended Settings:



Since this is a relatively simple print, **Standard Quality** (Layer Height 0.2mm) is the best option. The print quality will be fine and the Printing time will be rather short.

This print does not have to be very robust, so an **Infill of 20%** or even lower is enough.

It may seem at first glance that the use of **Support structures** is necessary, but in this case not only is it **not necessary**, but it will also be **destructive to the operation** of the whistle.

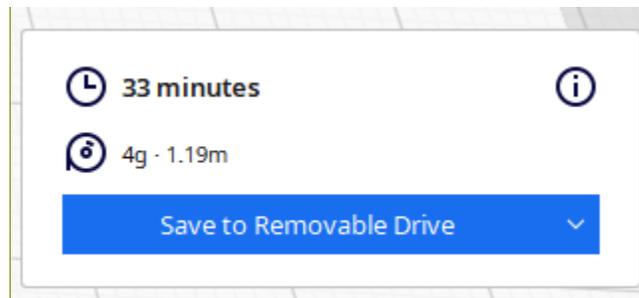


During the printing process of the upper section of the model a process called Bridging takes place. Bridging in 3D printing is an **extrusion of material that horizontally links two raised points**.

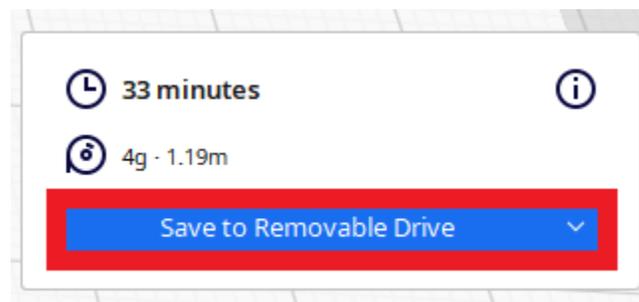
For larger bridges, you may need to add support structures, but short bridges can typically be printed without any supports to save material and print time. When you are bridging between two points, the plastic will be extruded across the gap and then quickly cooled to create a solid connection.

Finally, no Adhesion support is needed (i.e. no Brim or Raft is needed but is recommended to use a Skirt).

With these settings, the printing time is just above 30 minutes.



Step 2: Load the SD card to the 3D printer and start the printing process by selecting “Start Print” on the Printers menu. The print will start after the nozzle and the bed have reached the target temperatures, which may take a couple of minutes.





Step 3: Inspect the first layer in order to be sure that everything is working correctly. In case something goes wrong with the print of the first layer you should cancel the print and start troubleshooting in order to fix the problem.

Common problems with the First Layer

1. Filament does not stick to the build surface or comes out like “Spaghetti”.



That is typically a sign that the nozzle is too far from the build surface. You have to make sure that the build surface is leveled, so you need to level the Bed as described in Exercise 2 and/or clean the Bed.

If the surface is leveled but the first layer does not stick there are a couple more things you can do:

- Lower the Z-Axis Offset. In the printer’s menu you can lower the z height by changing it to a negative number (usually -0.01 is enough, so you can start from there and move to an even smaller number if necessary).
- Consider lowering the print speed of the First Layer. Sometimes it is a good idea to lower the print speed for just the first layer in order to give the filament more time to stick on the bed. It is a good practice to lower the speed, manually, as far as 50% till the first layer is printed and then switch it back to 100%.
- Consider lowering or even disabling the cooling fan. Plastic tends to shrink as it cools from a warm temperature to a cool temperature. Because of this fact, the plastic will tend to separate from the build platform as it cools. This is an important fact to keep in mind as you print your first layer. If you notice that the layer seems to stick initially, but later separates from the print bed as it cools, it is possible that your cooling settings are responsible. Through the printer’s menu you can disable the cooling fan till the first



couple of layers are printed and turn it back on again. Note that changing the fan speed or e.g. disabling the cooling fan for only the first layer is also possible through your slicer (e.g. see Print settings->Cooling in Cura)

2. No filament is leaving the nozzle in some places of the building surface.

The first thing you need to check is if the filament is loaded correctly. If that is not the case then your nozzle might be too close to the bed and thereby giving the melted filament little room to escape. The two most common solutions to this problem are:

- Raise the Z-Axis Offset - This method involves you raising the nozzle a little bit. With this setting, you can raise your nozzle a little higher from the print bed by giving the Z-axis setting a positive value. However, putting a negative value in the Z-axis offset setting will help you fix sticking issues. Therefore, you need to ensure that when setting a value, it isn't too high so your prints can stick to the bed.
- Level the Bed - This is an alternate solution you can do to fix this problem. It is a more complex fix though, given the fact that you'll have to level and recalibrate the bed as described in Exercise 2.

There is also the possibility that your nozzle is (partially) clogged. Unclogging or changing the nozzle is an advanced procedure and will be discussed in the troubleshooting chapter.

References

<https://www.simplify3d.com/support/print-quality-troubleshooting/poor-bridging/>

<https://www.simplify3d.com/support/print-quality-troubleshooting/not-sticking-to-the-bed/>

<https://all3dp.com/2/3d-printing-for-beginners-all-you-need-to-know-to-get-started/>

<https://all3dp.com/2/3d-printer-bed-leveling-step-by-step-tutorial/>

<https://www.dobot.cc/resource/10-common-3d-printing-problems-troubleshooting.html>



Activity Worksheet 1.3.2 (Student Version)

Level 1 (Novice Level: Basic Competences)

Chapter 1.3: 3D Printer Basics and Preparation for the First Print

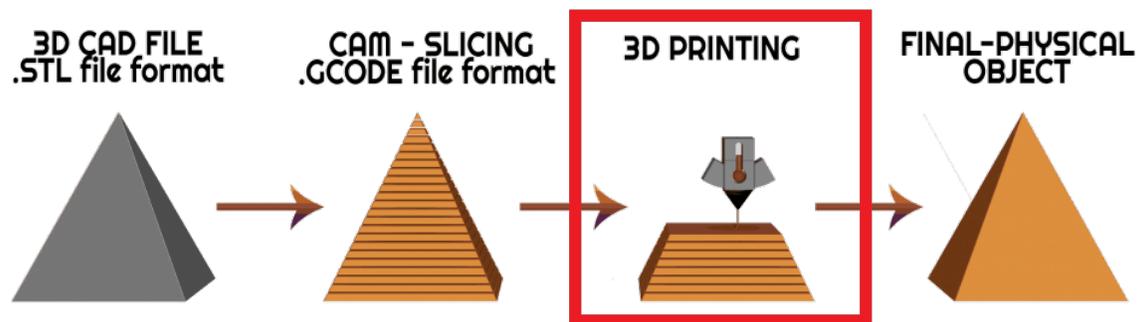
Activity Worksheet 1.3.2

In this Activity worksheet we will learn how to prepare the 3D printer and how to perform our first print. More specifically, we are going to learn:

- How to load the Filament on the printer
- What Bed Leveling is and why it is important
- How to properly Level the Bed
- How to identify the necessary corrections to be made based on the printing of the First Layer
- How to deal with the most common 3D printing problems.

Introduction

Once we save/export the instructions from the Slicer (i.e. the G-code file), we are ready to transfer these instructions to the printer (usually through a SD card) and finally begin the process of 3D printing.



However, before starting the 3D print job, a couple of things need to be done on our 3D printer:

- Loading the Filament
- Bed Leveling



Exercise 1: Loading the Filament

The **extruder needs to be ready** to extrude filament before printing begins. The loading process begins by **heating the hot end (nozzle) to the filament's molten temperature** (anything above 185 °C, depending on the filament), and then loading the filament into the heated extruder.

In order to heat the hot end you should enable the **Preheat Process** from the printer's menu. Where exactly to find this option depends on the specific 3D Printer's menu. In all cases, you can enable different Preheat Process options **based on the material** you want to load. The preheat temperatures are usually 10-15 degrees lower than the actual printing temperatures.



In the picture above, you can see examples of Preheat options for different materials. The first number states the required **hot end temperature**, while the second number states the required **Bed temperature**.

After enabling the Preheat option, you should wait for the printer's hot end to reach the required temperature and after that, you are ready to load the Filament.

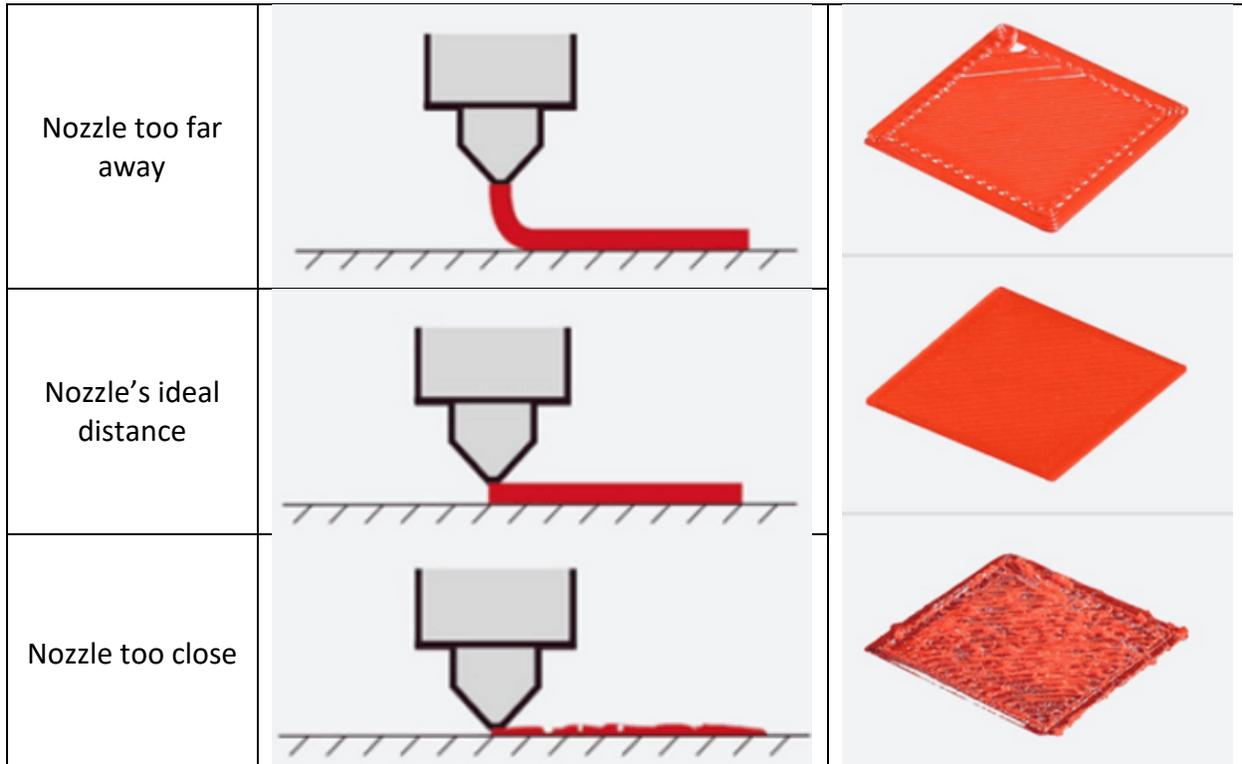
Before loading the filament, it is important to trim the filament's end and form a sharp tip (a side cutter works the best). Then you can load it to the extruder and push it until you feel that the filament has **reached the hot end** and a small amount of plastic has been extruded.



After the filament is loaded and you can cool down the printer by selecting the cool down option or just shut down the printer from the power supply.

Exercise 2: Bed Leveling

In order for the printer to deposit filament and build the object successfully, **the build platform must be level**. Depending on the machine, leveling can be either **manual or automatic**. Bed leveling is **very important** because, if the print bed is too far away from the nozzle, **the first and the most important layer will not stick to the bed**, causing the print to **fail**.



Before we start the leveling process, we have to be sure that both the **Nozzle** and the **Bed** surface are **clean**. In order to clean the Nozzle, the only thing you need to do is to use a dry cotton cloth to wipe any material that has been “stuck” around the nozzle from an earlier print job. To clean the Bed surface, in case it is a removable one, cleaning it in the sink with ordinary dish soap and lukewarm water works quite well. For a non-removable build plate, you can use isopropyl alcohol on a paper towel or cloth. After that, dry it with a soft cloth. Cleaning the bed surface (preferably before every print) is important because when we touch the build plate with our bare hands, some oil remains on it and can make it difficult for the model to stick properly.

Now we are ready to level the Bed surface by following the steps below:

1. Position your Nozzle

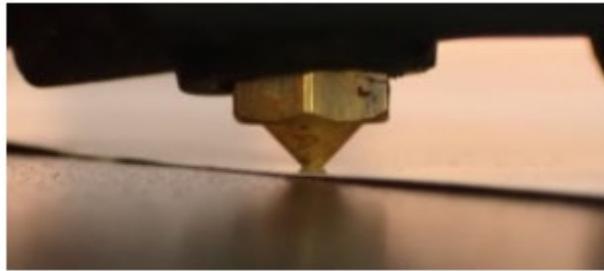
To do this, simply reset your print head position by selecting the “**Home Option**” on your printer Menu. This will send the nozzle to location (0, 0, 0) in the build volume's space, as



determined by the machine. The third 0 is significant because it specifies the Z coordinate or height, which is where your machine will print the first layer. Then, **turn off (disable) the stepper motors** so that you can freely move (by hand) the printhead in the XY plane. Again, your printer's Menu should include this choice.

2. Measure and Adjust

The 3D printer bed should be calibrated so that it is **as level as possible** in all **four corners** and in the **middle**. In theory, the nozzle should only make contact with the build plate at these points, which is why we aim to sandwich something thin between them - like a piece of paper. Note: For more accurate results, instead of a piece of paper, you can use a metallic feeler gauge with a 0.2mm thickness.



The "gap" is correct and consistent at all five spots if you reach the **point right before experiencing resistance** while sliding the paper around. Place your paper **between the nozzle tip and the print bed**, then move the printhead to a corner of the bed. Adjust the **closest leveling screw** to tighten the spacing if there is no resistance when moving the paper back and forth between the nozzle and the bed. Adjust the screw so you can **feel a tiny drag** from the nozzle and 3D printer bed while sliding your paper.

Carry on in the same manner with the remaining corners. When the corners are adjusted correctly, move the print head to the center of the build plate and verify that the same resistance is experienced.

Finally, double-check each corner and the center, as your changes may have had an impact on other areas. If this is the case, continue the process until all five points have been leveled and you are ready to go. Usually you need check every point at least 2 or 3 times to make sure that everything is level.

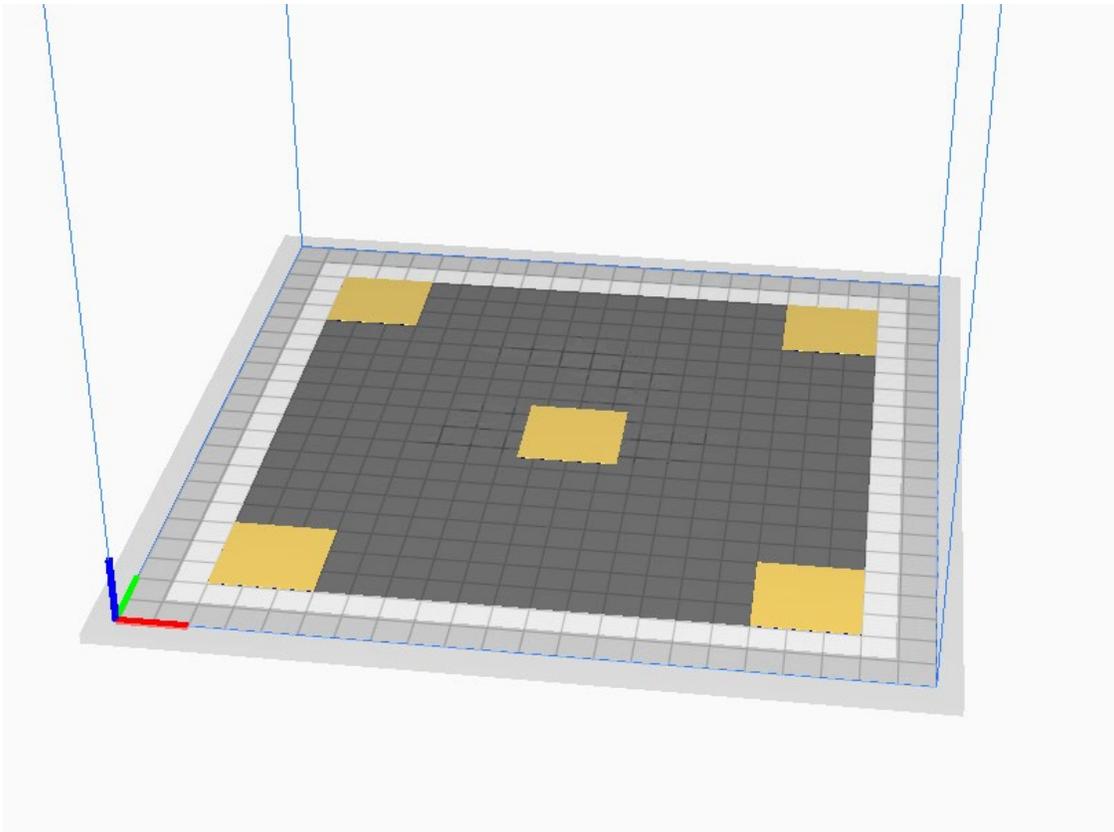
Note that bed leveling is a process that you need to do once in a while. Most of the problems in 3D printing are caused due to not having a level print surface, which is why the next step is really important.

3. Inspect the First Layer



When you start a new 3D print job you should keep an eye on the first layer in order to verify successful bed leveling. If successful, the first layer should look more or less the same over the whole surface. If not, you may need to cancel the print and further adjust your level.

Note: There are several bed leveling test prints that you can perform in order to verify how level is your build plate. One of these tests (bed leveling squares) is shown below. Most slicers (e.g. Cura) offer such test prints or, alternatively, you can easily find such tests online.



Exercise 3: The First Print

Now our 3D printer is ready to start our first print. In this section we will print, as an example, the whistle that we designed in chapter 1.2.

Step 1: Open Ultimaker Cura software and import the Whistle model (i.e. the .stl file). Make all the adjustments that are needed in order to slice it. Select the orientation on the build plate, the printing profile, decide whether to use support structures or not etc. Finally export the G-code file into an SD card (with the “Save to Removable Drive” or “Save to Disk” button).



Step 2: Load the SD card to the 3D printer and start the printing process by selecting “Start Print” on the Printers menu. The print will start after the nozzle and the bed have reached the target temperatures, which may take a couple of minutes.

Step 3: Inspect the first layer in order to be sure that everything is working correctly. In case something goes wrong with the print of the first layer you should cancel the print and start troubleshooting in order to fix the problem.

Common problems with the First Layer

1. Filament does not stick to the build surface or comes out like “Spaghetti”.



That is typically a sign that the nozzle is too far from the build surface. You have to make sure that the build surface is leveled, so you need to level the Bed as described in Exercise 2 and/or clean the Bed.

If the surface is leveled but the first layer does not stick there are a couple more things you can do:

- Lower the Z-Axis Offset. In the printer’s menu you can lower the z height by changing it to a negative number (usually -0.01 is enough, so you can start from there and move to an even smaller number if necessary).
- Consider lowering the print speed of the First Layer. Sometimes it is a good idea to lower the print speed for just the first layer in order to give the filament more time to stick on the bed. It is a good practice to lower the speed, manually, as far as 50% till the first layer is printed and then switch it back to 100%.
- Consider lowering or even disabling the cooling fan. Plastic tends to shrink as it cools from a warm temperature to a cool temperature. Because of this fact, the plastic will



tend to separate from the build platform as it cools. This is an important fact to keep in mind as you print your first layer. If you notice that the layer seems to stick initially, but later separates from the print bed as it cools, it is possible that your cooling settings are responsible. Through the printer's menu you can disable the cooling fan till the first couple of layers are printed and turn it back on again. Note that changing the fan speed or e.g. disabling the cooling fan for only the first layer is also possible through your slicer (e.g. see Print settings->Cooling in Cura)

2. No filament is leaving the nozzle in some places of the building surface.

The first thing you need to check is if the filament is loaded correctly. If that is not the case then your nozzle might be too close to the bed and thereby giving the melted filament little room to escape. The two most common solutions to this problem are:

- Raise the Z-Axis Offset - This method involves you raising the nozzle a little bit. With this setting, you can raise your nozzle a little higher from the print bed by giving the Z-axis setting a positive value. However, putting a negative value in the Z-axis offset setting will help you fix sticking issues. Therefore, you need to ensure that when setting a value, it isn't too high so your prints can stick to the bed.
- Level the Bed - This is an alternate solution you can do to fix this problem. It is a more complex fix though, given the fact that you'll have to level and recalibrate the bed as described in Exercise 2.

There is also the possibility that your nozzle is (partially) clogged. Unclogging or changing the nozzle is an advanced procedure and will be discussed in the troubleshooting chapter.