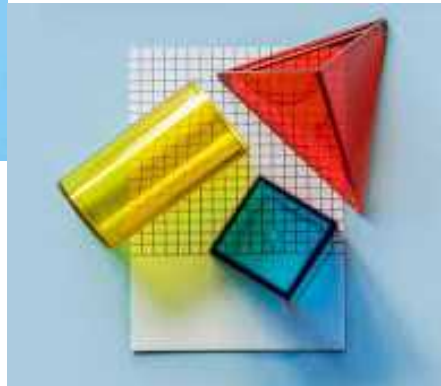




Project code:  
2019-1-EL01-KA201-062914

Erasmus+ Call: 2019 - KA2 -



Erasmus+

This project is funded by the European Union.



3D printing technology aims students understanding maths and recycling procedure

*Curricula 1: 3D Printing Technology and Application  
"Common 3D Printing Problems"*

Output 3 (O3) - 3D Printing and Maths

# Description

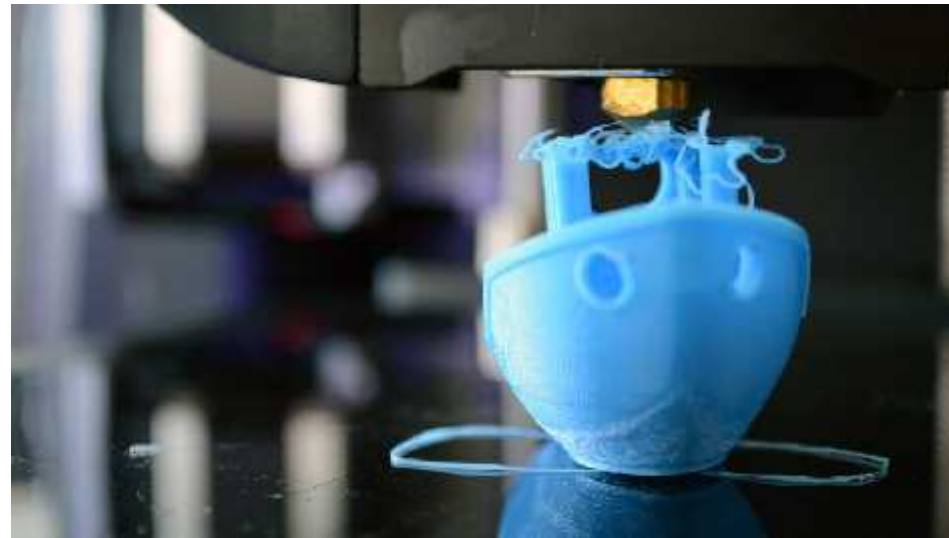


In this session, the most common 3D printing errors and issues will be presented. The session aims in explaining the more usual sources of those issues (whether they were caused by the model's design, by the slicer, by the 3D printer's settings or by a third party-factor) and providing the best possible solutions. Furthermore, an overall sum-up of the 3D printing procedures, issues and solutions will be given in the form of FAQ.

# Common 3D Printing Issues



- Nozzle too close to print bed
- The printer is working but nothing is printing
- Over-extrusion
- Incomplete and messy infill
- Warping
- Messy first layer
- Elephant's foot
- Print looks deformed and melted
- Snapped or stripped filament
- Getting cracks in tall objects
- Blocked nozzle
- Print head misses the bed
- Print Doesn't Stick to Print Bed
- Infill is Visible from the Outside
- Layers Don't Line Up Well
- Pits and Hollows in Top Layer (Pillowing)
- Print Has Lost Dimensional Accuracy



# OVER-EXTRUSION

- The Problem - Print Looks Droopy and Stringy

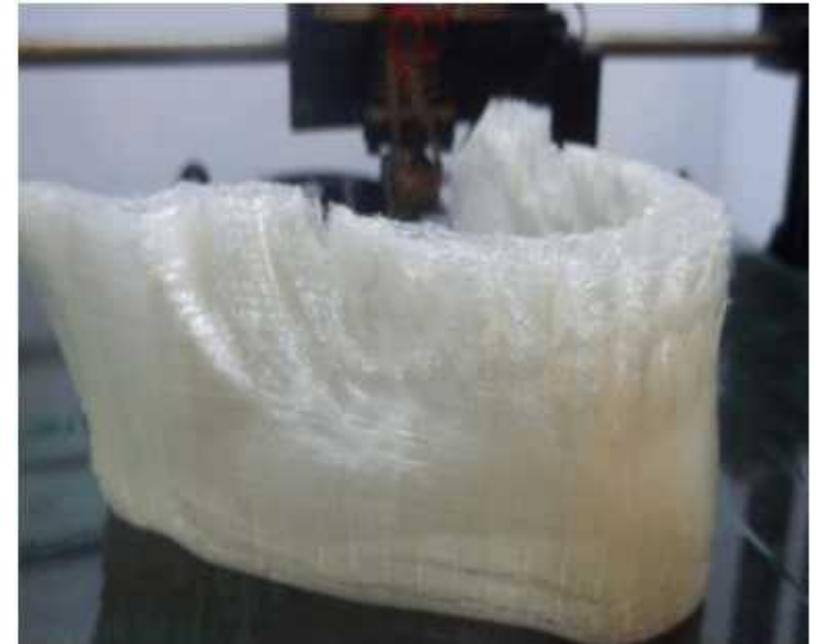
The printer is taking in more material than it requires, therefore, it produces more material than what's needed. Printed models will have excess materials on them.

- The Cause

Basically, the Flow setting or Extrusion multiplier setting in the slicing software is higher than normal.

- The Solution

To troubleshoot this 3D printing problem, check the Extrusion multiplier setting of the slicer software and. Ensure that you have selected the right value. If all seems to be okay, go to the Flow setting and decrease it.





# INCOMPLETE AND MESSY INFILL

- The Problem

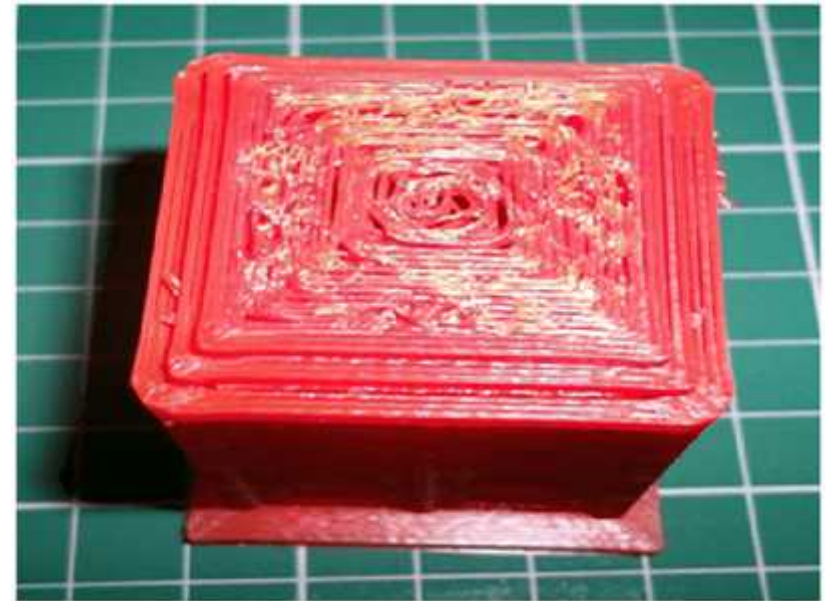
The print's internal structure is either broken or missing.

- The Cause

There are a lot of reasons why the internal structure of the model can be broken or missing. Although the most common is having incorrect settings in the slicing software, a slightly blocked nozzle can also bring up this problem.

- The Solution

Take a look at the fill density - Open your slicing software and take a look at the infill density. The best value is 20%; if the value is less than this, then you are bound to encounter problems. If you are running large prints though, you might want to increase this value to ensure that the model you are about to print gets enough support.



# MESSY FIRST LAYER

- The Problem

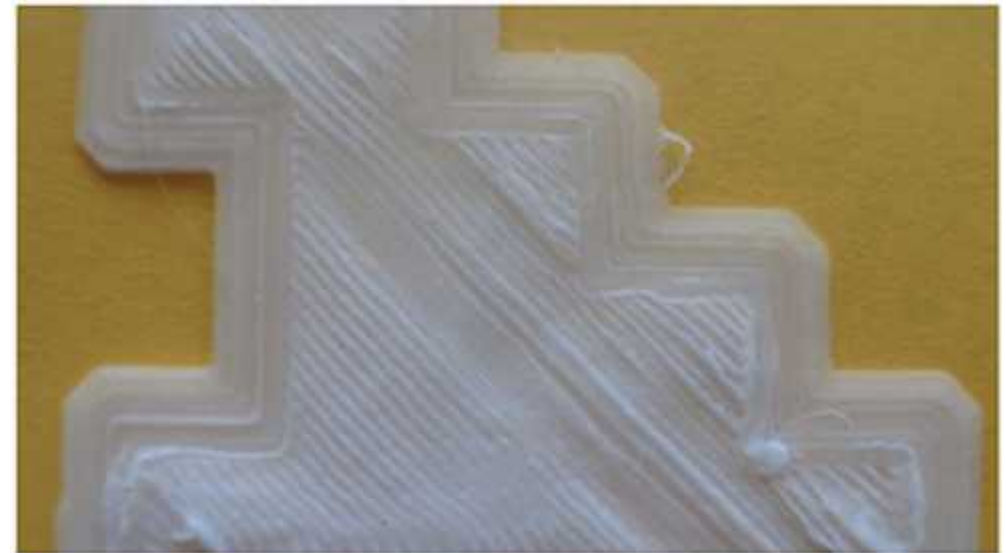
A print's first layers are often problematic. The problems are usually a non-sticky print, or the bottom shell has an erroneous look due to unwanted lines. Also, instead of getting a fine detail on the print's bottom, there is found a blurry, congealed design that does not look like the surface design.

- The Cause

The blurry and undefined detail of a print simply implies that the print bed temperature is too high. Unwanted lines happen as a result of the nozzle and the bed being too far apart, while blobs are occurring if the nozzle is way too close to the bed. Additionally, a non-sticky print is as a result of a bed that hasn't been properly levelled.

- The Solution

Reduce the bed temperature. Do this by bringing the temperature down five degrees at a time; continue reducing the temperature until you get the desired adhesion result without losing any detail.



# ELEPHANT'S FOOT

- The Problem

Elephant's foot is a 3D printing term that refers to the outward bulge of a model's base. Simply put, it is when the print bows or curves out at the bottom.

- The Cause

This usually happens when the weight of the model is pressing down on its base before it cools back to solid.

- The Solution

Another 3D printing troubleshooting tip you can use is to ensure that the base layers get enough coolness to be able to support the top structure. You need to ensure that the cooling is just the right amount as too much cooling will cause the base layers to get warped. You might find this part tricky but the best way to go about this 3D printing troubleshooting process is to lower the print platform's temperature by five-degree intervals to about plus or minus twenty degrees of the temperature recommended. If you have your Bottom/Top thickness set to 0.6mm, you can start the fan at a height that's slightly lower than that.



# SNAPPED FILAMENT



- The Problem

Nothing is coming out of the nozzle but when you take a look at the filament spool, you can see it's full and the feed tube also appears to have some filament in it. This problem is common to Bowden feed printers; it is not usually found in direct feeds. This is because the filament is usually hidden, therefore, you cannot see them immediately.

- The Cause

This problem can be caused by different factors but the most common one is old filament. While it is true that most ABS and PLA filaments have a long lifespan, they can become brittle if kept in bad weather conditions. And once they become bad, no amount of adjustment can make them right when they are fed to the printer.

- The Solution

Try making use of another filament. This would let you know if the brittle filament is actually the cause of the problem.

Check your printer's temperature and flow rate. This simply means that if the problem persists, check to see if the hot end is at the right temperature and getting hot like it's supposed to. You should also check to see if the filament's flow rate is not higher than a hundred percent.







# NOZZLE IS TOO CLOSE TO THE PRINT BED

- The Problem

Despite the fact that the filament is loaded correctly and nothing seems to be wrong with the print head, no filament is being deposited on the print bed.

- The Cause

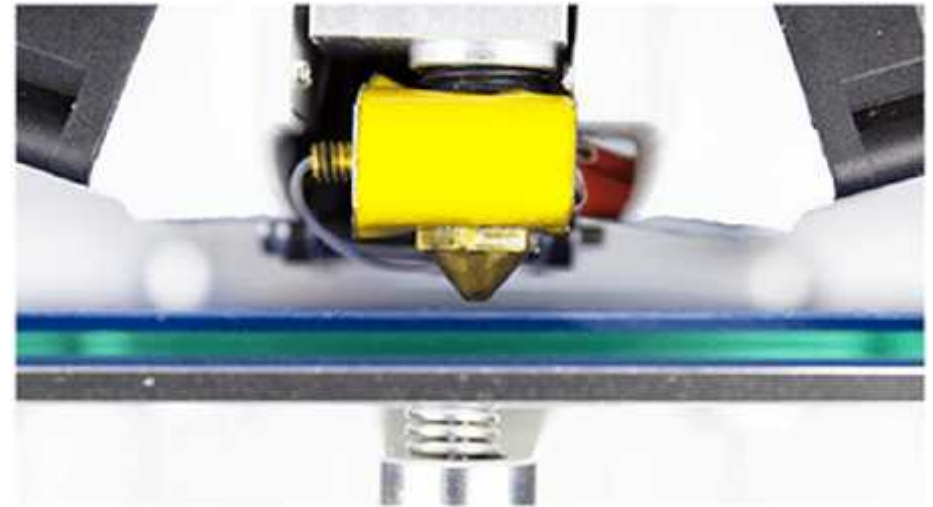
The nozzle and the print bed may be too close to each other. You may have inadvertently turned your print bed a little away from your nozzle opening, thereby giving the melted filament little room to escape. The best case scenario with this problem is that your print will possibly miss its first few layers and the chances of it not sticking when the filament does not extrude will be high. As for the worst case scenario, your printer's hot end will have a backup of a melted filament which will definitely increase the chances of a blockage.

- The Solution

There are two main ways to do this 3D printing troubleshooting:

**The Z-Axis Offset** - This method involves you raising the nozzle a little bit. Most 3D printers have a Z-axis offset setting in their system setting. With this setting, you can raise your nozzle a little high 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, that is, if your prints are not sticking to the print bed. Therefore, you need to ensure that when setting a value, it isn't too high so your prints can stick to the bed.

**Lower the Bed** - This is an alternate solution as not all 3D printers allow the lowering of the print bed. But if your printer allows it, you can do it to fix this problem. It is a more upsetting fix though, given the fact that you'll have to level and recalibrate the bed.



# THE PRINTER IS WORKING BUT NOTHING IS PRINTING



- The Problem - Out of Filament

You have correctly configured and set the model in the slicing software but yet nothing seems to be happening; no printing is actually taking place. You have repeatedly sent a print to the printer but all you're getting in return is a filament spit coming out of the nozzle. Or maybe the model is about to be printed and out of nowhere the filament extrusion stops working but the nozzle continues its run, printing nothing.

- The Cause

While this problem is obvious on machines that have their filament reel in full views, like the PRUSA i3, there are machines that are not designed with exposed filament reels, such as the MOOZ, Ultimaker, and Robox, which will make it a little difficult to immediately detect the problem. These type of 3D printers either have their filaments encased, or hidden at the back of the printer.

- The Solution

The 3D printing troubleshooting process here is quite easy. No matter the type of 3D printer you are using, all you'll have to do is remove the remaining filament and load in new material. Check the filament reel, if there's no material at all, load in another reel. It's as simple as that.



# BLOCKED NOZZLE (1/2)



- The Problem

You initiate a print job but whatever you try, nothing comes out of the nozzle. Extracting the filament and reinserting doesn't work.

- The cause

A small piece of filament has been left behind in the nozzle after changing spools, often because the filament has snapped off at the end. When the new filament is loaded, the piece of old filament that is left in the nozzle doesn't allow the new filament to be pushed through.

A little printer maintenance can go a long way to reducing the chance of problems like a blocked nozzle affecting your extrusions. In fact you'll often find that before a clog even appears, there is old carbonized filament sitting inside your nozzle. It can and will sit there for weeks or even months without you realising, but there will be small signs in the quality of your prints.

The effects are often overlooked; such as small nicks in the outer walls, small flecks of dark filament or small changes in print quality between models. These defects are often simply put down to the slight variants we come to expect from 3D printers, but really there could be something a little more sinister going on. A cleaning method known as the Atomic Pull or Cold Pull (which we detail below) can clear this up.

You'll commonly see this if you frequently switch from a PLA to ABS, for example. A small amount of PLA is left in the nozzle, and it is heated beyond its normal melting point. That can mean it will carbonise and burn.

Likewise, switch between ABS and Nylon and again you'll witness something similar. It's not uncommon to see a wisp of smoke appear briefly as the new filament is fed through.

# BLOCKED NOZZLE (2/2)

- Solution

Heat the nozzle and clean with a needle

Remove the feeder tube and try pushing the filament through with another piece of filament

Dismantle the hot end and see if you can extract the filament blockage

Try an Atomic Pull (Cold Pull)



# 3D Printing Mistakes



Youtube. Maker's Muse. "5 3D Printing Mistakes you WILL make - and how to avoid them! 3D Printing 101". 2017



# FAQ for 3D Printing (1/7)



- **How long will it take to print?**

Printing usually takes hours, not minutes, and can vary depending on the level of detail of the .stl file and other specifications. Larger, more precise models can take significantly longer to print than the same model reduced in size and precision.

- **How long does a 3D print take?**

That depends on several factors:

The size of your model

The material you are using

The layer height (smaller layers mean more layers to print)

The complexity of your model

Whether the object needs supports

The most common **slicers** give to you the estimated time of print the model, but since external factors affect the job the print could use more time or less.



## FAQ for 3D Printing (2/7)

- **What is an .stl file?**

A stereolithography (stl) file is a format that provides 3D printers the blueprint to print a part. The .stl file is created as an export option in many 3D software applications such as 3D Studio Max, Maya, Google Sketchup, and Blender.

- **What is infill in 3D printing?**

In 3D printing, infill refers to the structure printed inside the object. The size, shape, pattern, and percentage depends heavily on the design requirements. Higher infill percentage translates to physically stronger 3D printed samples.

# FAQ for 3D Printing (3/7)



- **What are supports?**

Your 3D models probably need some help to print properly, as these printers don't do well with big overhangs -- for example, an arm sticking out from a figure.

Your 3D printer software can usually automatically calculate and add supports, meaning little stands that hold up all those sticking-out parts of the model. After the print is done, clip the supports off with micro cutters and file down any nubs or rough edges with hobby files.



# FAQ for 3D Printing (4/7)



- **How big an object can I 3D print?**

It depends on the specific 3D printer you are using. Each has a maximum build volume

Typical maximum size for personal 3D printers: 150-300mm per side

Large prints typically take a VERY LONG TIME to complete, often hours or even days

Large 3D models can be split into pieces, printed and assembled into a larger object if they don't fit into a specific 3D printer

Industrial 3D printers can handle larger sizes - but they are very expensive

# FAQ for 3D Printing (5/7)



- **Which materials can be 3d printed? Can I print in <insert random material>?**

Personal 3D printers typically support PLA and ABS plastics or similar plastics

PLA is biodegradable and made from corn, but is brittle

ABS is made from petroleum and is strong

Other experimental materials are possible, such as mixing wood, nylon or metal particles into PLA plastic and certain plastics such as nylon, HIPS and others

Certain new professional 3D printers offer higher temperature build chambers, where more exotic high-temp materials such as PEEK, ULTEM, PAEK can be used

Food paste and other squishy substances can be 3D printed with specialized 3D printers equipped with syringes

Industrial 3D printers can print in sand, ceramics or metals, but are very expensive

# FAQ for 3D Printing (6/7)



## What software do I need to use for a 3D printer?

Each 3D printer includes software to drive the printer itself. Types vary by manufacturer, but there are some commonly used open source systems that work on most machines, as well as a couple of pay-for third party tools

If you're designing your own 3D Models, you need a 3D CAD (computer aided design) program, like Autodesk 123D, SolidWorks or any of many others. Note that some are extremely expensive to purchase, others are available at no charge

CAD programs frequently specialize in a type of design: mechanical parts, artistic shapes, architecture, etc. There are dozens of 3D CAD programs; choose one that best meets your budget, skill level and design goal

# FAQ for 3D Printing (7/7)



- **What is G-Code?**

G-code (gcode) is the code that the printer will use to print your 3D model. gcode can be in a file as commands instructing where the printer needs to move and extrude (X, Y, and Z) that the printer uses to build your model.

- **What is the bed auto leveling feature?**

A sensor measures the distance between the nozzle and the bed, on 3 or more points (up to 32), assuring you the perfect bed leveling and adhesion.



Thank you!!

[privasi.aegean.gr](http://privasi.aegean.gr)

