

3D4ELDERLY

PROJECT NUMBER: 2020-1-LT01-KA204-077896

IO1A1 - Transfer of knowledge about the basics of 3D printing concepts to the caregivers' staff working with people with Alzheimer and elderly people with dementia.

CONSORTIUM OF PARTNERS:



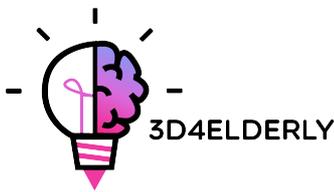
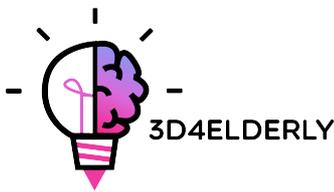


Table of Contents

1. Introduction to additive manufacturing	4
1.1. What is rapid prototyping?	4
1.2. What is additive manufacturing?	6
1.3. How does 3D printing work?	7
2. 3D printing main technologies	8
2.1. Fused Deposition Modeling (FDM)	9
2.1.1. FDM: Process, materials, application areas	9
2.1.2. FDM: Strengths and Weaknesses	10
2.2. Stereolithography (SLA)	10
2.2.1. SLA: Process, materials, application areas	10
2.2.2. SLA: Strengths and Weaknesses	11
2.3. Selective Laser Sintering (SLS)	11
2.3.1. SLS: Process, materials, application areas	11
2.3.2. SLS: Strengths and Weaknesses	12
2.4. Photopolymer Jetting (POLYJET)	13
2.4.1. POLYJET: Process, materials, application areas	13
2.4.2. POLYJET: Strengths and Weaknesses	13
2.5. Selective Laser Melting (SLM)	14
2.5.1. SLM: Process, materials, application areas	14
2.5.2. SLM: Strengths and Weaknesses	15
2.6. Electron Beam Melting (EBM)	15
2.6.1. EBM: Process, materials, application areas	15
2.6.2. EBM: Strengths and Weaknesses	16
2.7. Electron Binder Jetting	16
2.7.1. Electron Binder Jetting: Process, materials, application areas	16
2.7.2. Electron Binder Jetting: Strengths and Weaknesses	16
3. Producing pieces with 3D printing. Process	17
3.1. Obtaining the digital model	18
3.2. Exporting to STL file	20



3.3. <i>Obtaining the G-Code</i>	20
3.4. <i>3D printing</i>	25
3.5. <i>Extracting pieces</i>	25
4. <i>3D printing materials</i>	27
5. <i>Limitations</i>	27
6. <i>Practical uses of 3D printing. Example</i>	28
	31



1. Introduction to additive manufacturing

Additive Manufacturing or 3D printing (commonly called) is a process that creates a physical object from a digital design. There are different 3D printing technologies and materials you can print with, but all are based on the same principle: a digital model is turned into a solid three-dimensional physical object by adding material layer by layer.

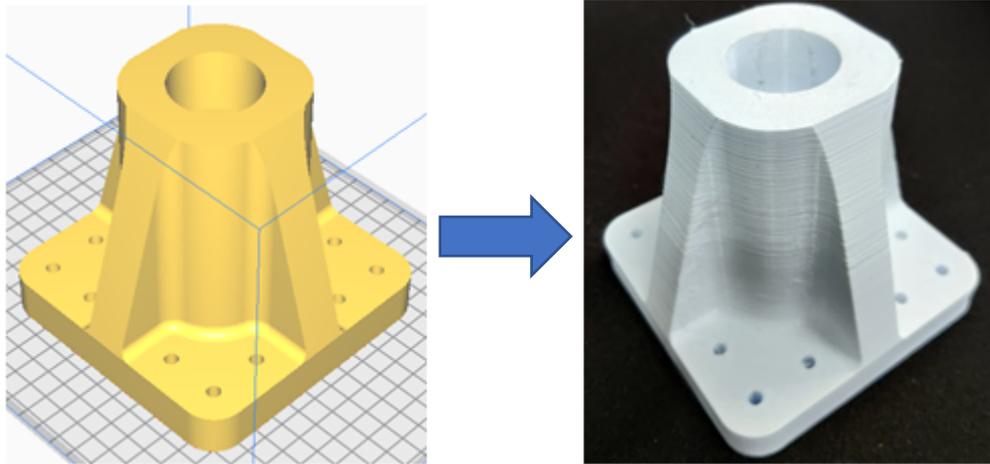


Figure 1. From digital model to physical object. Source: CETEM.

1.1. What is rapid prototyping?

Rapid prototyping is an automated process that quickly builds physical prototypes from 3D CAD files composed of surface quality or solid models.

Any manufacturing process can be classified as either subtractive, formative or additive. Every manufacturing process either falls completely into one of these categories or is a hybrid process falling into more than one. In the manufacturing arena, productivity is achieved by guiding a product from concept to market quickly and inexpensively. Rapid prototyping technology aids this process.

It is important not to confuse rapid prototyping with 3D printing or with additive manufacturing, because the concepts are used interchangeably and wrongly many times. We can say that additive manufacturing is one of the technologies with which we can produce a rapid prototyping product.

It is convenient to underline that every technology and every process has a starting point in common: computer aided design (CAD).



Figure 2. Silicone mould and pieces created by rapid prototyping techniques. Source: CETEM.

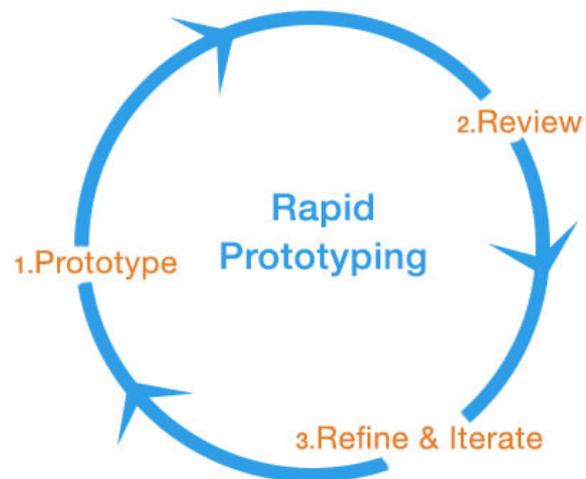
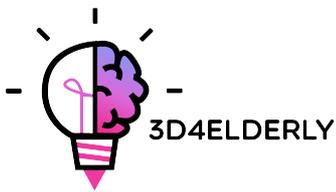


Figure 3. The concept of rapid prototyping. Source: <https://www.linkedin.com/pulse/rapid-prototyping-tool-mindset-future-business-fiona-triaca/>



1.2. What is additive manufacturing?

It is important to point out from the beginning that Additive Manufacturing does not constitute a single technology but a set of manufacturing processes, very different from each other, that share three common characteristics:

1. They are manufacturing processes by addition of material to construct a solid three-dimensional object.
2. The object is constructed by superimposing successive layers of material.
3. The object is made from a digital 3D model.

They are called ADDITIVE Manufacturing processes to differentiate them from conventional processes. Together with these, they are part of the set of processes available to the Industry.

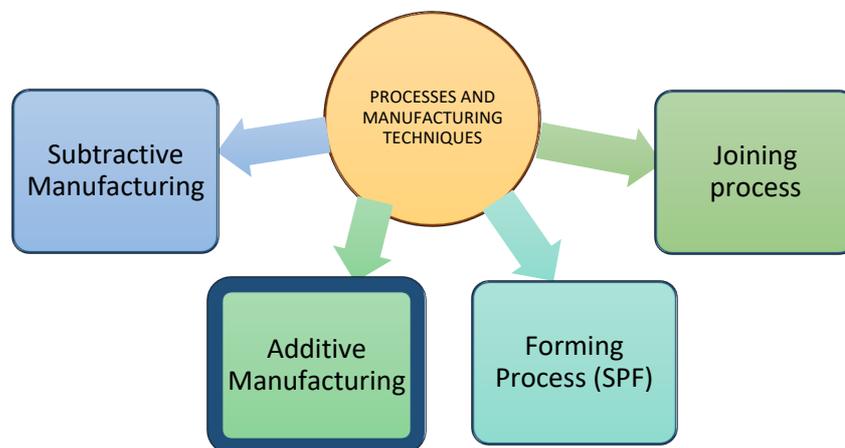


Figure 4. Processes and manufacturing techniques. Source: CETEM.

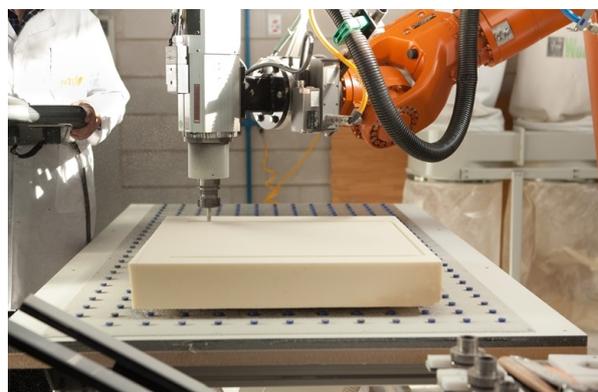


Figure 5. Subtractive manufacturing example. Source: CETEM.



Figure 6. Welding. Joining process example. Source: <http://www.minaprem.com/joining/welding/introduction/definition-of-welding/>

1.3. How does 3D printing work?

It all starts with making or obtaining a virtual design of the object you want to create. This virtual design can be made in a CAD (Computer Aided Design) file using a 3D modeling program (for the creation of a totally new object) or with the use of a 3D scanner (to copy an existing object). A 3D scanner makes a 3D digital copy of an object. There are also lots of online file repositories where you can download existing 3D files that will help get you started.

The 3D printing process turns an object into many, tiny little slices, then builds it from the bottom up, slice by slice. The layers then build up to form a solid object.

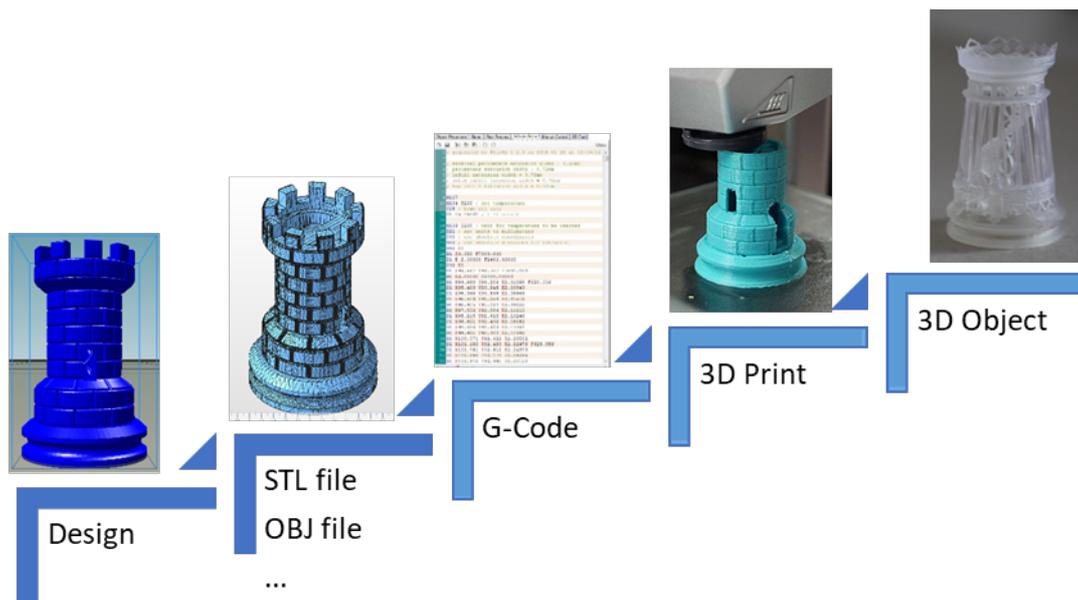
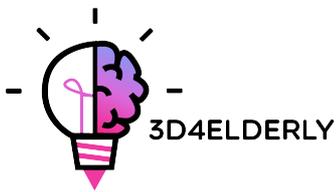


Figure 7. 3D printing process. Source: Karlsruher Institut für Technologie.



2. 3D printing main technologies

The following are considered as the main 3D printing technologies available in the market:

- Fused Deposition Modeling (FDM).
- Stereolithography (SLA).
- Selective Laser Sintering (SLS).
- Photopolymer Jetting (POLYJET).
- Selective Laser Melting (SLM).
- Electron Beam Melting (EBM).
- Electron Binder Jetting.
- Continuous Fibre Fabrication.
- Matterial Jetting.

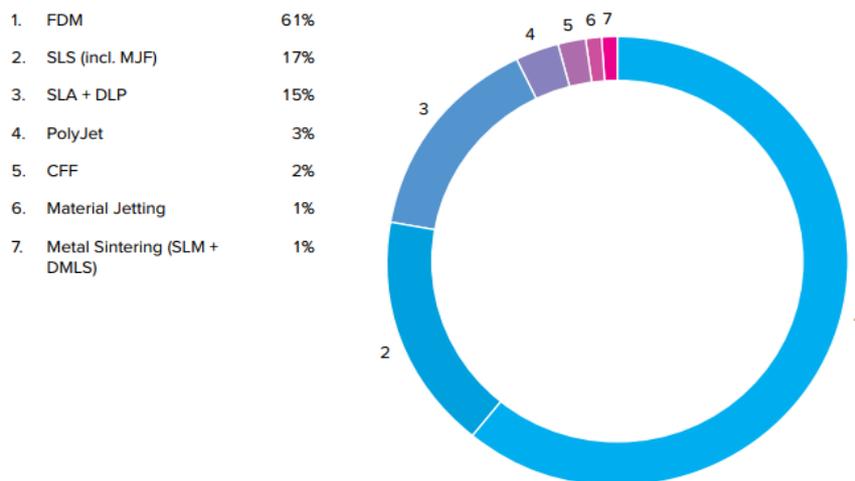
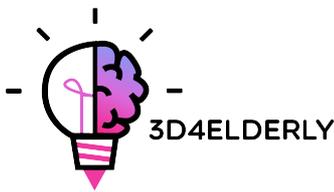


Figure 8. Most used technologies according to a report. (2017). Source: <https://f.3dhubs.com/yZgXoWzB88BhMHwG9fo3mV.pdf>

Now they are presented briefly, and also explained regarding the process, available materials, application areas and also strengths and weaknesses.



2.1. Fused Deposition Modeling (FDM)

2.1.1. FDM: Process, materials, application areas

Home printers typically work with plastic filament. The technology behind this is often referred to Fused Deposition Modeling (FDM) and it is a 3D printing technology that works by extruding a thermoplastic polymer through a heated nozzle which gets deposited on a building stage. FDM is also considered to be a form of additive manufacturing, which at the same time is a “process of joining materials to make objects from 3D model data, usually layer upon layer”.

The mere process involves a plastic filament which is fed by a spool to the nozzle where the material is liquefied and “drawn” on the platform. As soon as it touches the build stage, the filament hardens while being gradually deposited, following a certain structure, to create the final 3D print. When a layer is drawn, the platform lowers by one layer thickness so that the printer can start working on the next layer.

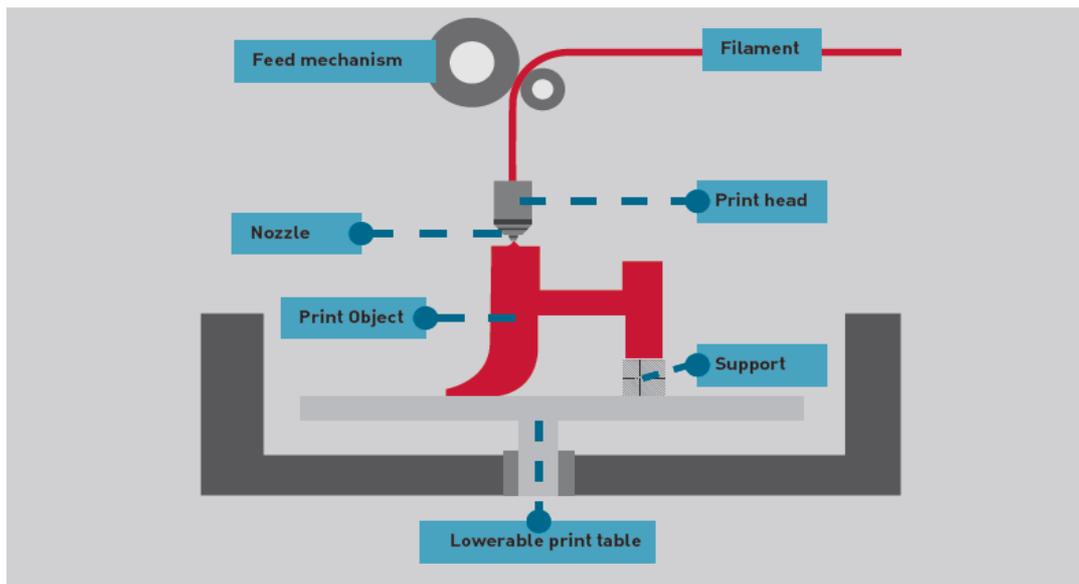
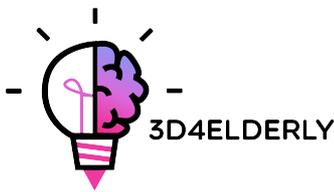


Figure 9. Scheme of FDM technology. Source: <https://i.materialise.com/blog/3d-printing-technologies-and-materials/>

There are many different materials which can be used with FDM. The most used are PLA (Polylactic Acid), ABS (Acrylonitrile Butadiene Styrene) and Nylon (Polyamide).

FDM is often used in non-functional prototypes to produce concept parts, functional models, prototypes in general, manufacturing tooling and modeling, and end use parts.



2.1.2. FDM: Strengths and Weaknesses

Those who want to engage with Fused Deposition Modeling have an advantage from the start; in fact, FDM printing machines are among the cheapest and most affordable especially for those who want to use it in a domestic environment.

FDM is a very clean technology, usually simple-to-use and office-friendly. The technology can also produce complex geometries and cavities that would otherwise be quite problematic.

As for accuracy, the 3D prints do not reach the same level of accuracy than other technologies, such as SLA or SLS. That said, the result is fairly qualitative.

2.2. Stereolithography (SLA)

2.2.1. SLA: Process, materials, application areas

Stereolithography (SLA) is a light-based process that builds individual layers of a model with liquid polymer, hardened by a laser beam. It is the oldest technology in the history of 3D printing, but it is still very much used nowadays.

After each layer, the resin tank peels away to release the hardened material. The building platform then moves up from 25 to 200 microns, depending on the chosen layer height, to prepare for the process of solidifying the next layer.

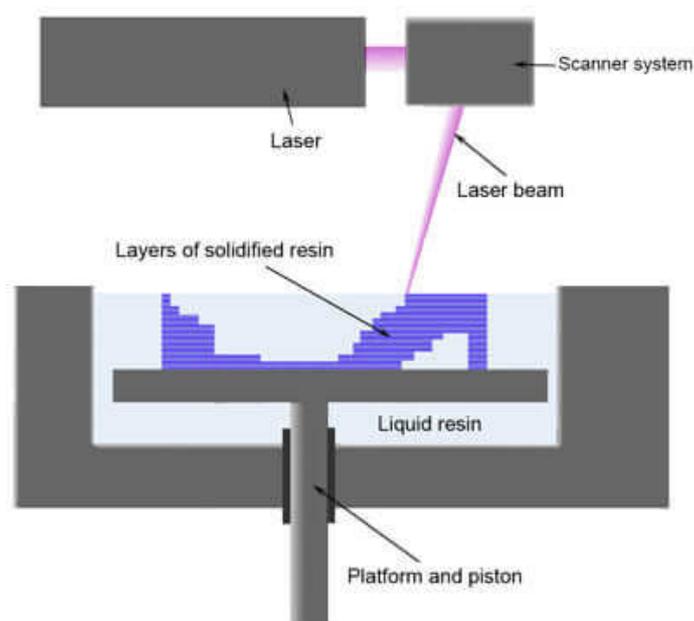
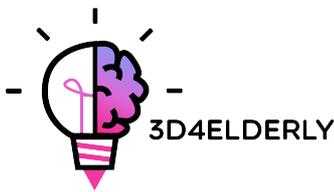


Figure 10. Scheme of SLA technology. Source: <https://all3dp.com/1/best-resin-dlp-sla-3d-printer-kit-stereolithography/>



When it comes to materials, Stereolithography offers a variety of choices which can be summarized in three categories. Within the first, paintable resin can be used. Transparent resin and prime grey are two other kinds of resin material that can be used as well.

Models made in Stereolithography are typically used as high-quality show models. Design and engineering departments use the models as visual prototypes or for presentations, but we have noticed that it works really well for character and toy models. Stereolithography can be used in many areas. Nowadays, the automotive, aerospace, medical and consumer goods fields are the most popular ones.

2.2.2. SLA: Strengths and Weaknesses

Among all the 3D printing technologies, Stereolithography is one of the best; its strength relies mainly on the high resolution of the 3D prints. It allows to print objects with very complex geometries while at the same time preserving the quality and the details.

However, costs are one of the major weaknesses in the case of Stereolithography. Cons when it comes to Stereolithography also involve the fact that liquid resins are generally irritating and toxic.

2.3. Selective Laser Sintering (SLS)

2.3.1. SLS: Process, materials, application areas

Selective Laser Sintering (SLS) is a technique that uses laser as power source to form solid 3D objects. The main difference between SLS and SLA is that it uses powdered material in the vat instead of liquid resin.

Selective Laser Sintering does not require the use of support structures, therefore cutting the number of materials that one needs to provide for the print. The object is, in fact, printed while being constantly surrounded by not sintered powder.

The mere process involves a laser which is used to selectively sinter a layer of granules, consequently binding together the material to create a solid form. At the end of the process, the object can very hot and therefore it will be left to cool off before being removed from the machine.

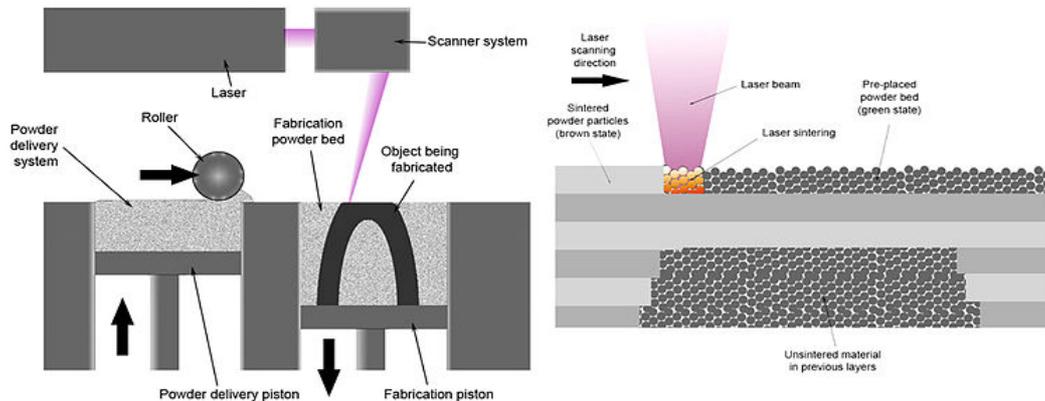


Figure 11. Scheme of SLS technology. Source: https://en.wikipedia.org/wiki/Selective_laser_sintering#Technology

The use of SLS technology makes it possible to involve a variety of materials which range from nylon, glass and ceramics, to aluminium, silver and even steel. However, some of them, like ceramics, are not laser sintered. A binder, in this case, is used to glue parts together and this is usually known as “Powder & Binder-based 3D Printing”.

Areas of application of SLS technology range from the automotive sector to the consumer goods sector. More specifically, it can be used in the case of product development and rapid prototyping in a wide range of commercial industries, as well as limited-run manufacturing of end-use parts. In the aerospace industry, for example, SLS is involved to build prototypes for aircraft components.

2.3.2. SLS: Strengths and Weaknesses

One of the very first pros that Selective Laser Sintering presents is the fact that it makes no use of support structures, as it is fully self-supporting. The second advantage in this case is that SLS can handle a high complexity of geometry. Some products are that complex that without this technology it would be complicated to produce them.

However, usually SLS prints present a certain surface porosity hence why, just like in Fused Deposition Modeling, post-processing is required.

2.4. Photopolymer Jetting (POLYJET)

2.4.1. POLYJET: Process, materials, application areas

Photopolymer Jetting is a 3D printing technology which makes use of inkjet print heads which work by jetting liquid photopolymers onto a build platform. Once the material reaches the platform, it is immediately cured by UV lamps and it hardens instantly, allowing the machine to keep building the top layers.

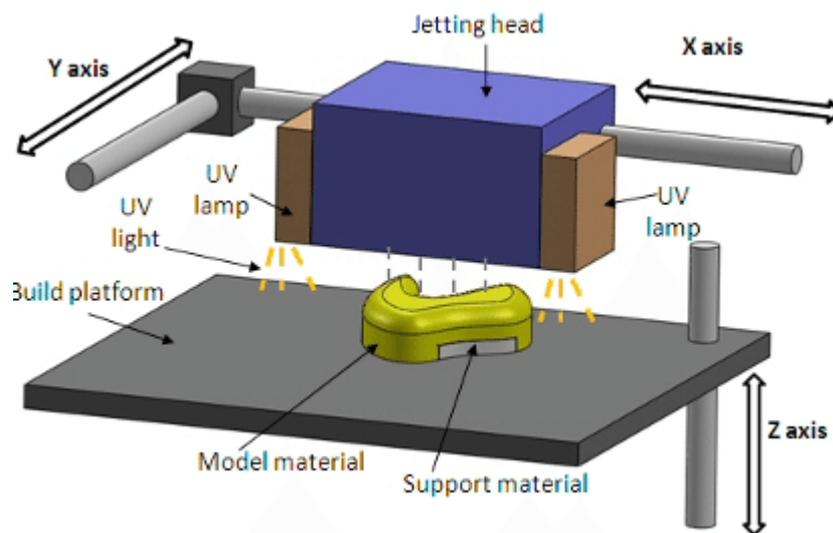
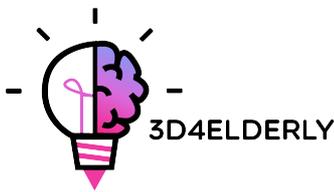


Figure 12. Scheme of POLYJET technology. Source: https://www.researchgate.net/figure/Schematic-representation-of-PolyJet-printing-process_fig1_318112255

The application areas include prototypes, which can be built in multiple materials with multiple colours and good surface finish. Hence why, photopolymer jetting parts are often used for visual and form and fit testing. Casting patterns are also another area, alongside with tools for injection molding of very small series and prototypes.

2.4.2. POLYJET: Strengths and Weaknesses

The positive side of this technology is that more materials can be used at the same time, which allows to have a final print with more than one colour. Through mixing these different proportions of materials, the print will be functionally graded and built-in order, for example, to create different shore hardness at different locations of the part. One of the strengths of this technology is also that it can achieve good accuracy and surface finishes.



However, some of the cons in this case include the fact that the usage is very much limited to photopolymer materials. Moreover, the pricing of the technology, which includes both the purchase of a machine and the relative materials, is normally expensive and the technology is overall rated as quite complex.

2.5. Selective Laser Melting (SLM)

2.5.1. SLM: Process, materials, application areas

Selective Laser Melting is a 3D printing technology that works thanks to the use of a laser which selects and distributes the material, which comes in the form of powder, onto a building platform. The powder is melted and distributed layer by layer. By repeating the process of coating powder and melting where needed, the parts are consequently built in the powder bed.

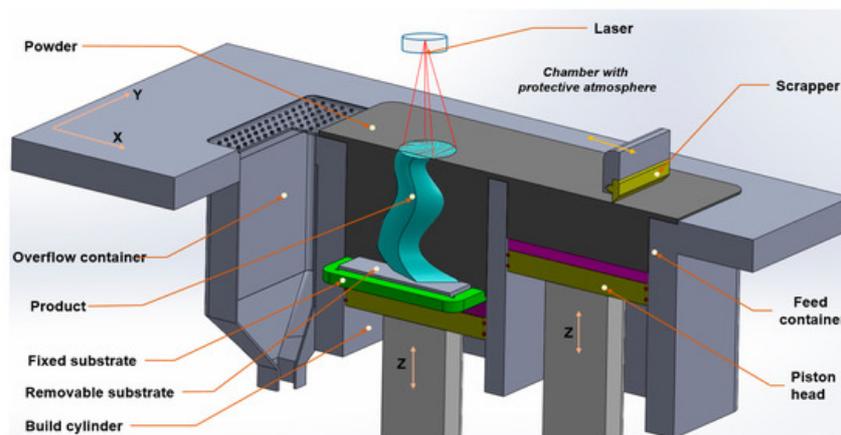
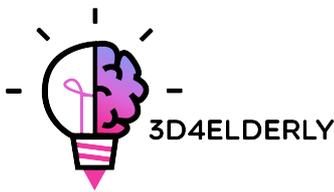


Figure 13. Scheme of SLM technology. Source https://www.mdpi.com/applsci/applsci-09-03031/article_deploy/html/images/applsci-09-03031-g001-550.jpg

Application areas for SLM include prototypes produced in standard metals for form, fit and functional testing, support parts, small series parts and tools for injection. SLM can be used to create dental products and anatomic parts, as well as mechanical components, for example lightweight components used to build aircrafts.



2.5.2. SLM: Strengths and Weaknesses

The positive side of Selective Laser Melting is that it can work with standard metals which can have a high density up to 99% and with good mechanical properties.

The negative side of this technology is that its process is rated as slow. More importantly, the pricing is not affordable to everyone which makes SLM not cost-effective.

2.6. Electron Beam Melting (EBM)

2.6.1. EBM: Process, materials, application areas

Just like SLM, Electron Beam Melting is an additive manufacturing 3D printing technology which can produce metal parts. The mere process consists in a thin layer of metal powder which is selectively melted by an electron beam. The parts are built up layer by layer in the powder bed. The difference with SLM is exactly that it uses a beam instead of a laser. The machine distributes a layer of metal powder onto a building platform, while being melted by the electron beam. After that, the building platform is lowered and the next later of metal powder will be coated on the top of the previous layers.

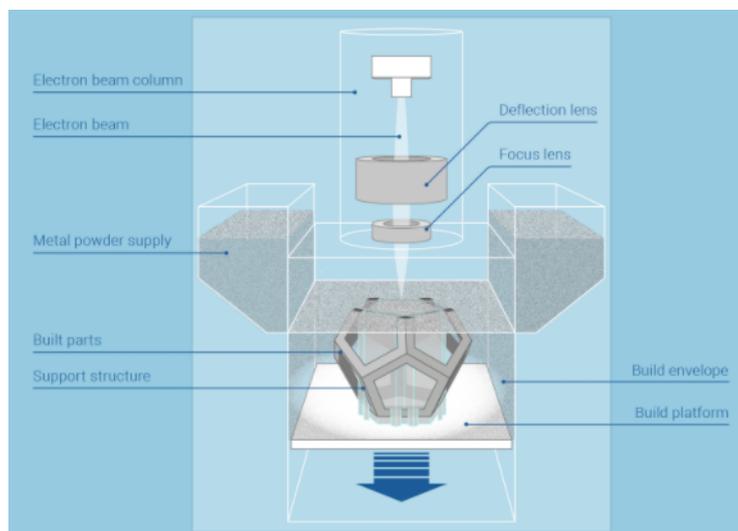
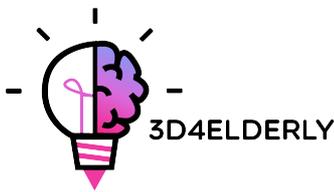


Figure 14. Scheme of EBM technology. Source: <https://www.additively.com/en/learn-about/electron-beam-melting>

Application areas include mainly the dental, the medical and the automotive sectors. More specifically, EBM is used to create medical implants and bone tissue models.



2.6.2. EBM: Strengths and Weaknesses

The advantage for using EBM is that it can use standard metals up to 99% of density. Compared to laser melting, EBM produces less thermal stress in parts and therefore requires less support structure. Moreover, the process is considered faster than in SLM. Accuracy is also rated fair generally.

Cons include mainly the pricing for both machines and materials, can become quite expensive, on the same level as SLM. Post-process steps generally require the removal of the build envelope, of the powder, thermal processing, the removal of supports and post-machining structures and finally surface finish.

2.7. Electron Binder Jetting

2.7.1. Electron Binder Jetting: Process, materials, application areas

Just like in POLYJET, Electron Binder Jetting is a 3D printing technology which works using inkjet print heads which apply a liquid bonding agent onto thin layers of powder. The jetting machine works in a way that it distributes a layer on the build platform, which at the same time will be lowered once the next layer of powder is to be laid out. The process keeps repeating itself until the parts are completely built in the powder bed.

Application areas of BJ technology include the architecture and the mechanical sectors. A specific example of a product created with BJ are pots and more general home furniture.

2.7.2. Electron Binder Jetting: Strengths and Weaknesses

One of the positive sides of using BJ technology is that there is no need for support structures. The build parts lie in the bed of not bonded powder. The entire build volume can therefore be filled with several parts, including stacking, and pyramiding of parts.

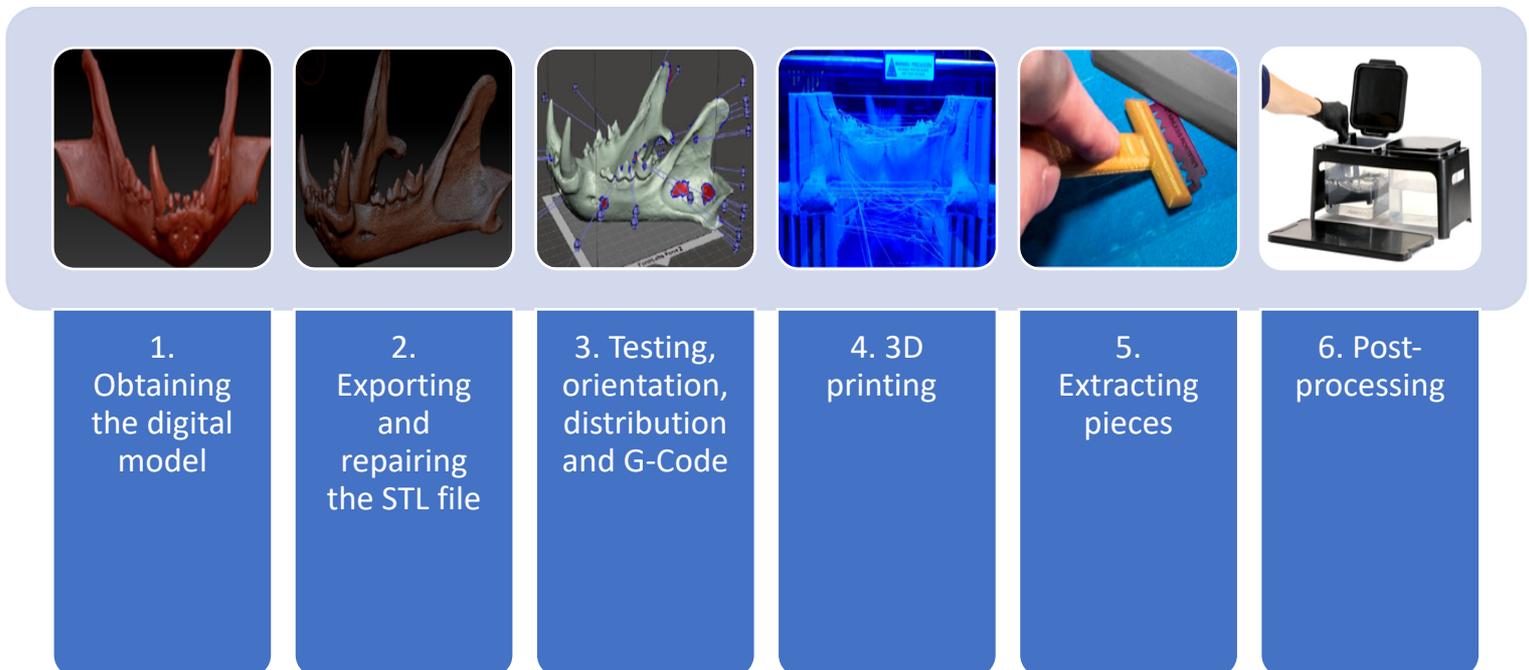
The process is fast, with a very low complexity and relatively low costs because powder particles are glued together, compared to EBM where the material is melted under protective gas environment. Certain machines can also work with colours and therefore they can produce coloured parts. The negative side is that the result presents fragile parts with limited mechanical properties.

3. Producing pieces with 3D printing. Process

In this chapter, all the necessary processes and steps for, starting with a digital design, obtaining a real printed piece, are going to be described.

It is important to mention that there is not just a single valid process for printing three dimensional pieces. What is explained in this guide are a certain number of steps that should be adapted to the type of piece, selected technology, type of machine, and even to the used software. Furthermore, the process that is described hereunder is mostly intended for fused deposition modelling (FDM) 3D printers.

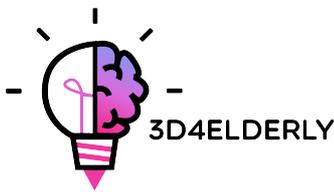
The production process, in general, is the following:



It is important that the tips given in each of the steps of the process are indicative and they have general term. This means that the tips should not be followed very closely. In the production process for 3D printing, experience, piece features, used machine, etc. have a lot of weight. It is for sure that someone with little or without experience will print a lot of 3D pieces with failures, before he finds the key.

Also, it is important to notice that mandatory steps are the following:

- Obtaining the digital model.
- Exporting to STL file.
- Obtaining the G-Code.



- 3D printing.
- Extracting pieces.

Now, each of these steps of the production process are explained in more detail and deeper.

3.1. Obtaining the digital model

There are several possibilities for obtaining the 3D model or digital model that is wanted to print. There are three possibilities:

- **Using a CAD software for designing the model.** There are a lot of available CAD software for modelling, and there is not a best option; it will depend on the user and their abilities with the software.

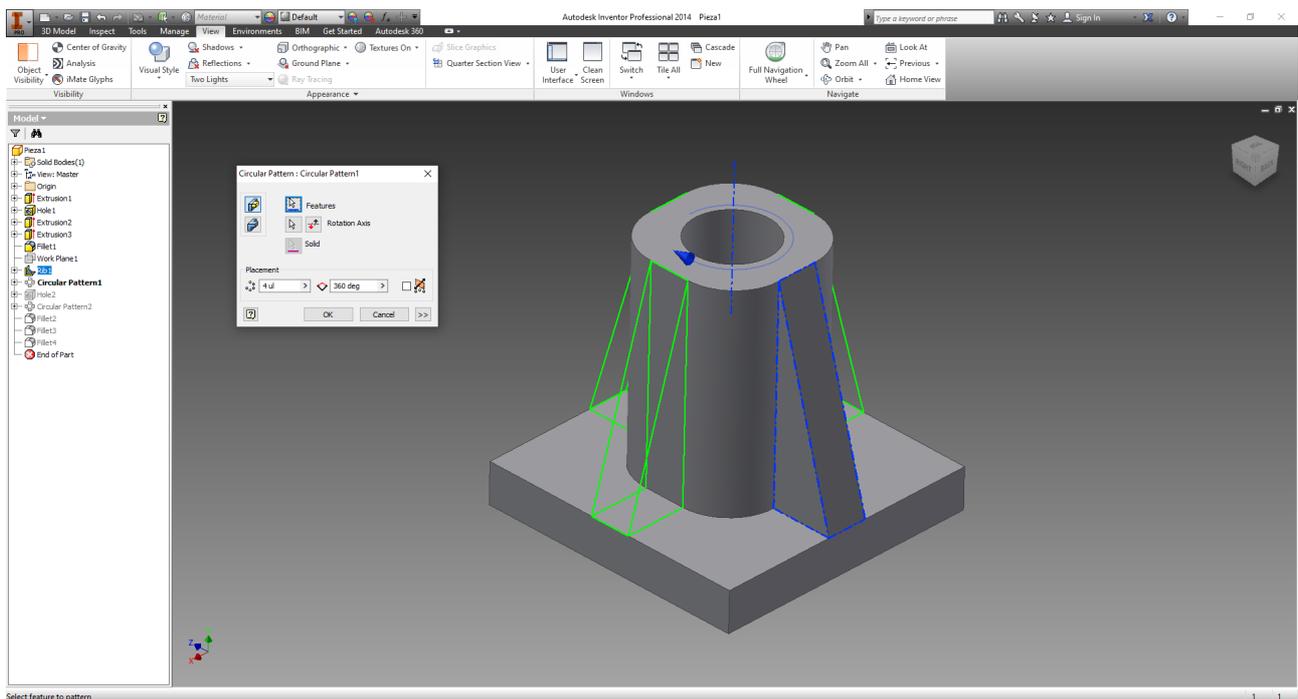


Figure 15. Modelling using Autodesk Inventor 2014. Source: CETEM

- **To obtain the geometry by Reverse Engineering and 3D scanners:** Here, a 3D scanner is used to digitally obtain the geometry of a real object. This is not a simple process and some ability and experience are required. In addition, there are several types of 3D scanners, and they are usually expensive.

Reverse Engineering is the process for copying, improving, or customizing real objects, or also for incorporating complex surfaces to a 3D modelled piece.

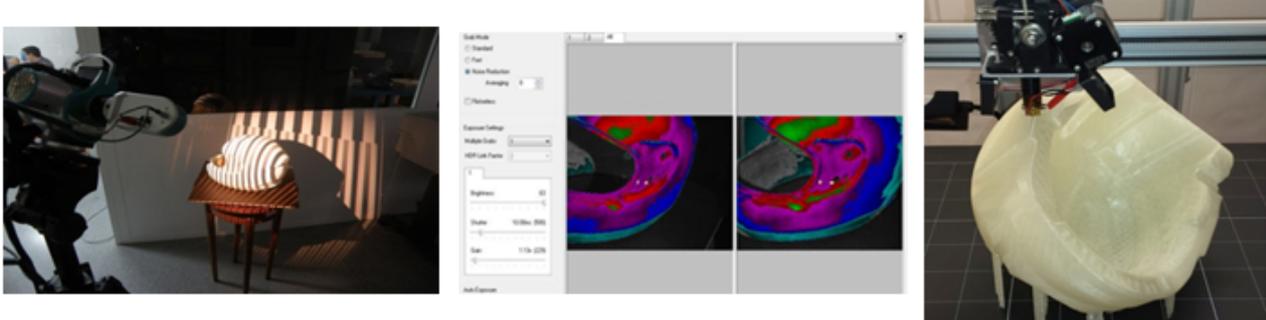


Figure 16. Reverse Engineering of a helmet. From left to right: 3D Scanning, redesigning and 3D printing. Source: CETEM.

- Downloading the model from repositories:** If you do not have knowledge in computer-aided 3D design, or you do not have the necessary equipment (or software, or even knowledge) for applying a process of reverse engineering, to download the model from a repository is the best option. There are specific repositories for 3D printing, with files almost already prepared to be printed, or general digital model repositories, with all kinds of CAD files. Best 2021 repositories can be found here: <https://all3dp.com/1/free-stl-files-3d-printer-models-3d-print-files-stl-download/>

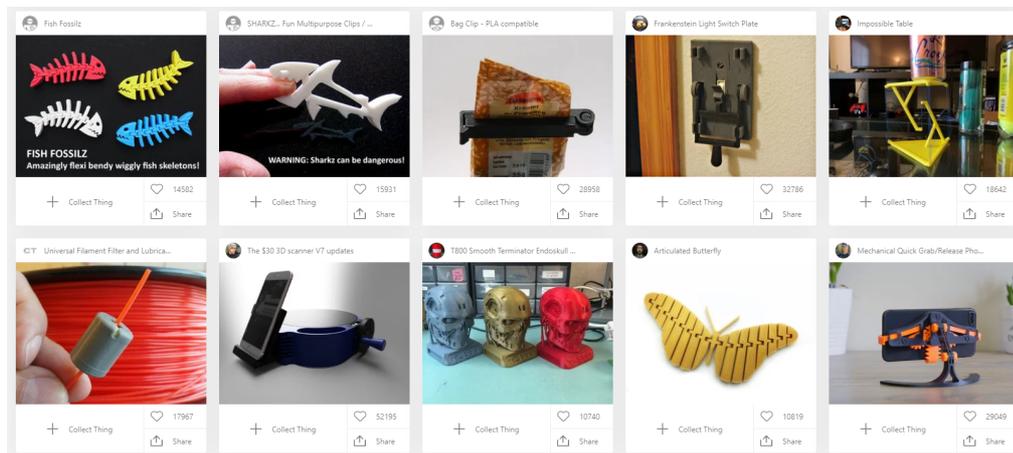
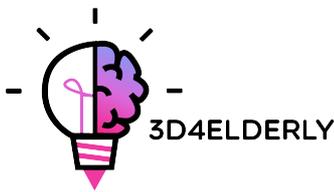


Figure 17. Example of a repository: Thingiverse. Source: Thingiverse.



3.2. Exporting to STL file

When you work designing and printing 3D models, a wide range of formats or types of files are available. Some of them are thought for designing or scanning, but others are associated to 3D printing, such as: STL, OBJ, PLY or FBX, among others.

When the piece is designed and modelled, a format conversion to the ". stl" file is needed. If the piece has been downloaded from a repository, this conversion is frequently already done.

STL stands for "Standard Triangle Language". In simple terms, this format uses linked triangles for recreating the solid model surface. Depending on the model complexity, more or less triangles (and with more or less size) will be necessary for recreating it, and the greater or the less big the size of the file also will be.

Normally, exporting a CAD design to STL format is as simple as going to the used software menu and clicking on "Save as..." or in "Export" and choosing STL.

3.3. Obtaining the G-Code

This stage of the production process in 3D printing is about the preparation of the pieces or digital models (previously exported to STL) for being 3D printed.

It is about performing the next process, orderly:

- To analyze the piece or model: thickness, holes, stability, angles, triangle mesh, etc.
(Not always necessary)
- To design the support structures or to automatically generate them.
- To choose the infill of the model, both percentage and shape.
- To locate the piece on the printing surface (or printing bed) and to select the most appropriate orientation.
- To generate the machine code or G-Code.

Regarding the **support structures**, there are necessary to beat gravity and to print overhang parts (or with internal gaps). Support structures are usually done by the same material as the piece, and they are usually necessary from 45 degrees (for FDM printers):

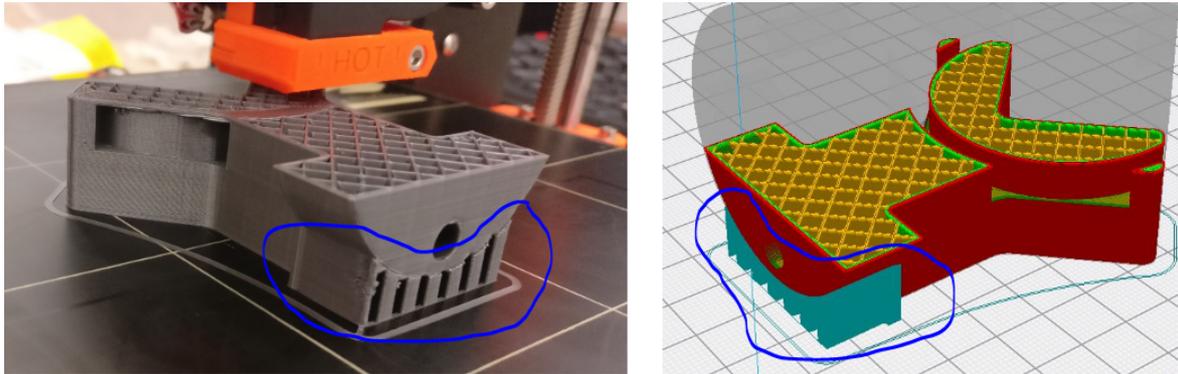


Figure 18. Supports structures in the printed piece (left) and in the software for generating the G-Code (right). Source: CETEM.

As support structures are thought just for holding the first layers of the model that are cantilevered or "floating", support structures are built lightly and using less material than for the piece itself. In addition, they will not mark so much the piece when they are removed. Most of the available software, either analysis software or either the own software of the printer machine, allow two options: making a design of the support structures, or automatically calculating and inserting these structures. If you do not have enough experience in 3D printing it is recommended let the program automatically calculates the supports.

Regarding the **model infill**, we are talking about the structure printed inside the object. It means, if we think in the example of a cube, the six external walls will be printed in a solid way, with a certain thickness, but the internal part of the cube will not be solid; you will have to choose the infill percentage, and even the geometrical shape of the infill. With some visual examples it can be better understood:

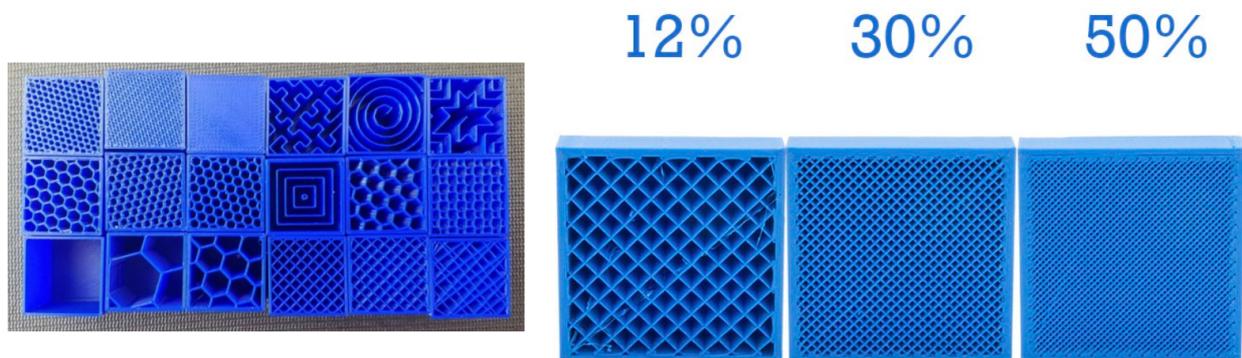
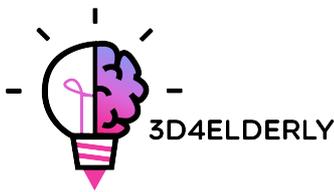


Figure 19. Different infill shapes and infill percentages. Source: <https://all3dp.com/2/infill-3d-printing-what-it-means-and-how-to-use-it/> Source: https://filament2print.com/gb/blog/71_importance-infill-3d-printing.html

Percentage and shape will be chosen according to the required strength and resistance for the piece, also according to total weight of the piece, used material, printing time, etc. The



greater the infill percentage, the stronger the printed piece will be, but the longer will take to be printed. A percentage about 15-30% usually is enough.

Regarding positioning and orientation of the piece, it is about deciding the position and orientation of the piece on the printing surface or printing bed. It will be done in the printing software.

There are different criteria to decide both position and orientation. One of the most used criteria is to use the minimum amount of material and spending less printing time. This is usually achieved by minimizing overhang parts. Consequently, less support structures will be printed, and the piece will be built in less time. However, sometimes quality to achieve is more important, so that orientations that are not optimal in terms of material and printing time, can be chosen.

Some general tips are:

- To center the pieces in the printing surface.
- When printing several pieces at once, place them together, with gaps between them about 10 -15 mm.
- When printing curved surfaces, piece should be positioned so that surfaces are on XY plane (horizontal).
- If the piece has an inner hole, or a through hole, it would be adequate to put this hole with its axis perpendicular to the printing bed.
- In a general way, the top surface of a printed piece will have the best finishing.
- A very long and plane section printed on the horizontal plane or XY plane may suffer deformation. Sometimes, to print these pieces so their longest section is perpendicular to the building plate is convenient.

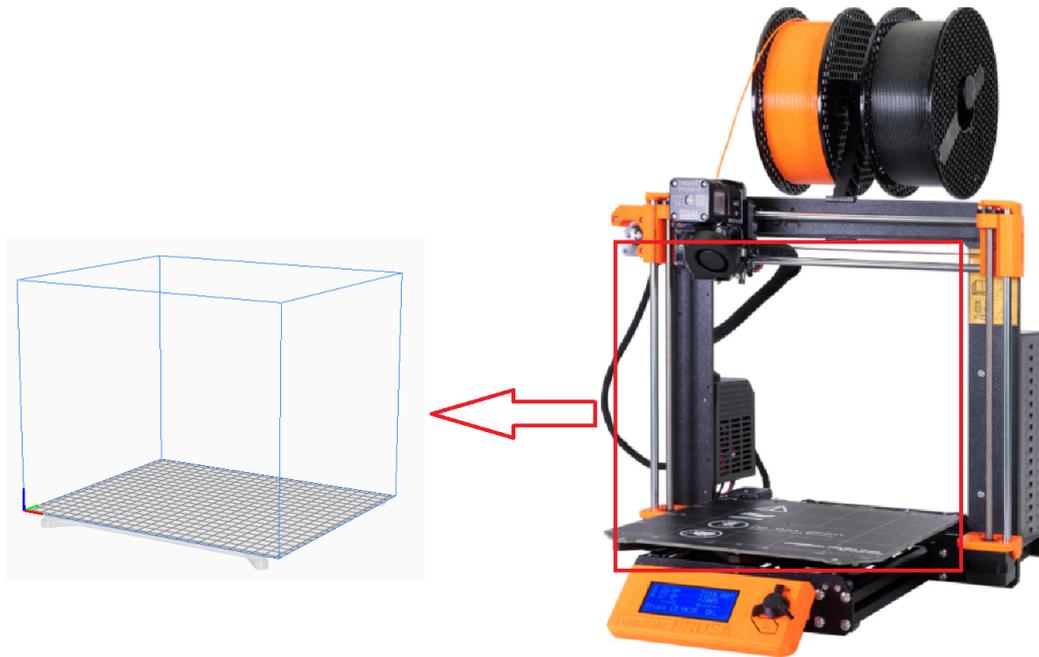


Figure 20. Printing bed. Source: CETEM.

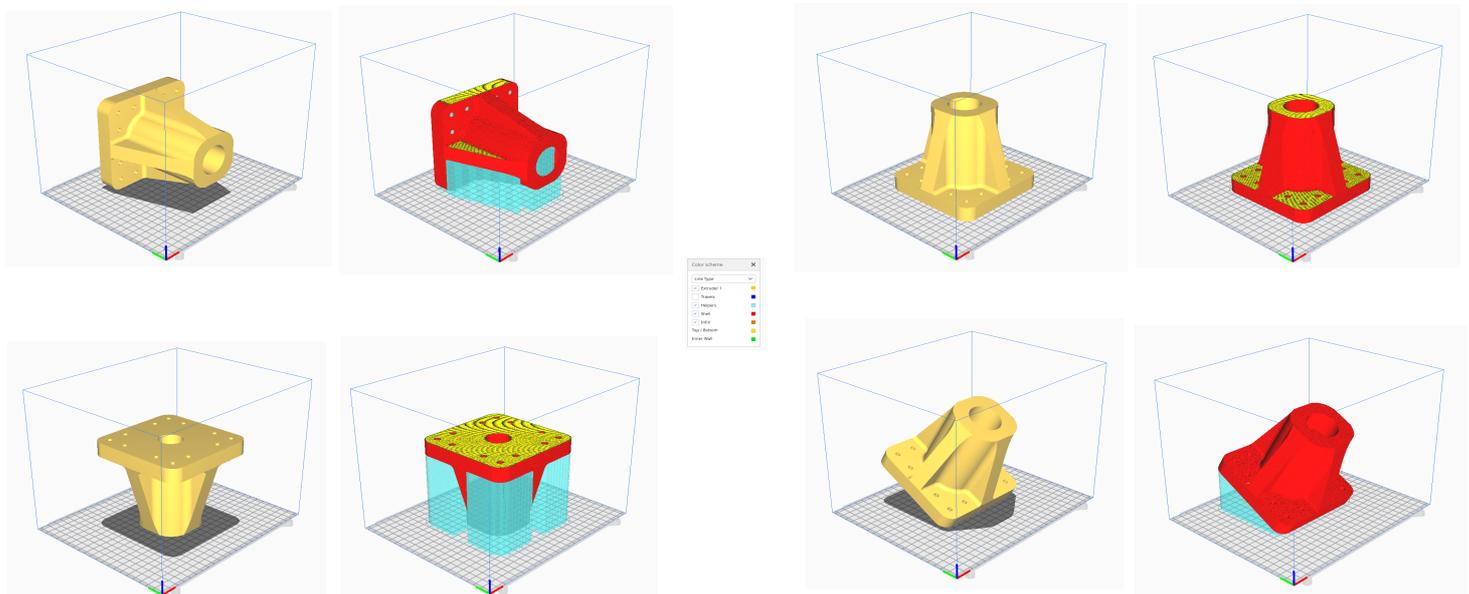
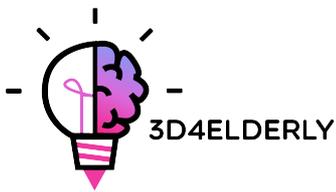


Figure 21. Different ways of positioning and orienting the same piece. In blue: supports structures. Source: CETEM.

Regarding generating the G-Code, this is the last step before going to our 3D printing machine. This code is the translation of the piece and all the selected parameters into instructions the machine can understand to printing our models.

Before getting this code, it is necessary to select some more parameters such as layer height. Also known as layer thickness. This parameter has also a lot of importance and it will



have a great impact on the final quality of the surface of the piece. A higher layer height will result on a great resolution or quality, but also will result on a greater printing time. Typical values for this parameter, regarding FDM technology, are: 0.05 mm (very fine and accurate, but it will take more time to be printed), 0.1 mm, 0.15 mm, ... and up to 0.4 or even 0.6 mm.



Figure 22. Visual differences according to different layer heights. Source: http://wiki.ikaslab.org/images/thumb/8/8c/Altura_de_capa.jpg/700px-Altura_de_capa.jpg

Depending on the chosen program or software, it will be necessary to choose parameters, such as wall thickness, printing speeds, bed and nozzle temperatures (the depend on the selected material), etc. But these are parameters to edit by users with experience, and it is advisable to let them default.

So, at this point, the G-Code must be generated. The program cuts the model into horizontal slices or layers and generates all the instructions for the machine, also calculating the amount of material and the estimated printing time.

In the figure below it is possible to see some chosen parameters, the model, the printing time, etc., and the option of "Save to file", which will generate the G-Code.

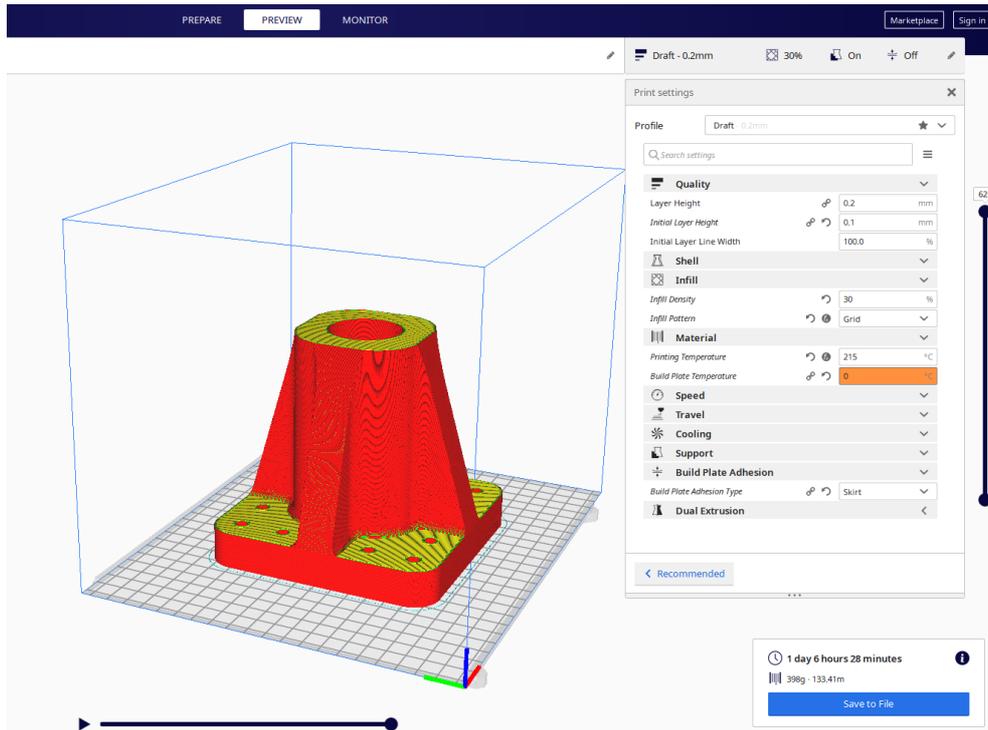


Figure 23. G-Code generation with Cura. Source: CETEM.

3.4. 3D printing

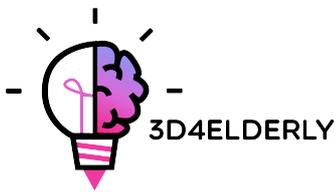
Now the G-Code has been obtained the process of 3D printing can be implemented. There are a few things to consider and to check before printing.

For FDM printers, the temperatures of the bed or printing platform and of the extruder must be checked (the software or the printer do that automatically when you start to print). To use lacquer or varnish is also recommendable in order to ease the extraction of the piece. It is advisable to read the instructions to also know how to load the filament on the printer for example, and for further information.

Therefore, the G-Code is sent to the machine, and the machine starts to work.

3.5. Extracting pieces

When the printer has already finished, it is the moment to implement the process of extracting the piece/s. Once again, depending on the machine and mainly, on the technology and used materials, the extraction processes will vary.



For fused deposition modelling machines (FDM), pieces often are removed by hand, or with some tool like a blade. It is also possible to have flexible platforms that ease a lot the extraction. There are even other methods such as using a solvent or applying cold or with dental floss.

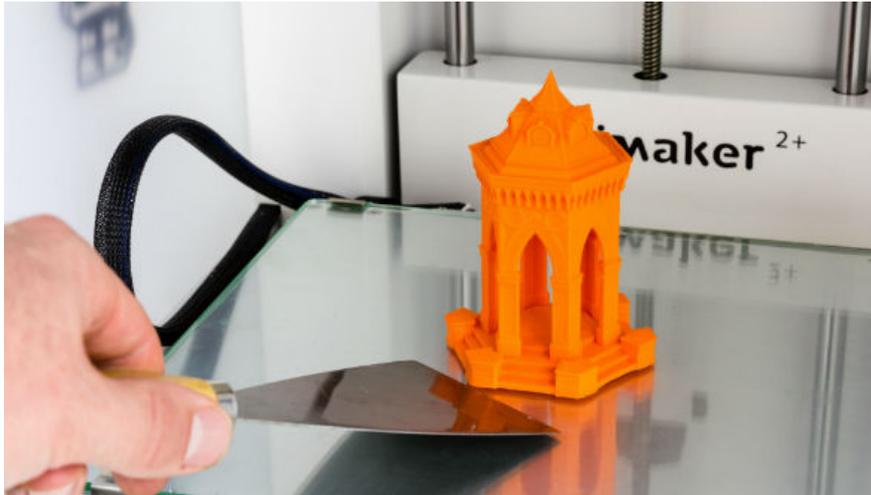
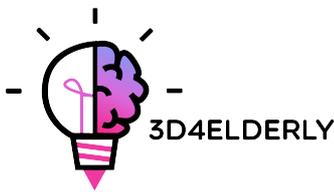


Figure 24. Removing a piece from a FDM 3D printing machine. Source: <https://lacountylibrary.org/removing-3d-prints/>

Now it is time to think if our piece requires some post-processing. The basic finishing process is removing mechanically the support structures (if the printed piece has) and sanding a little bit in order to improve the surface quality.



Figure 25. Removing supports structures from a FDM piece with pliers and sanding the surface. Source: CETEM.



4. 3D printing materials

When it comes to 3D printing, materials are often one of the most important choice. Today, the 3D printing market offers a variety of choice for what concerns materials. From polymers and metals, to ceramics and composites, many are the materials that have been created, each of them with its own advantage and disadvantages.

Each technology (SLS, SLA, FDM, POLYJET, etc.) has their own materials, but for the purpose of this guide, FDM materials are going to be briefly explained here below.

So, for what concerns Fused Deposition Modeling (FDM), some of the most popular materials are:

- PLA: Easy to print, very accurate, low melting point, rigid. Good for most things, not for warm/hot regions.
- ABS: Tends to warp, strong and slightly flexible. Used for many mechanical parts.
- PETG: Easy to print & accurate, tends to string, good layer adhesion.
- TPU: Flexible polymer; ok to print, kind of like stiff rubber. Best used with direct-drive machines.
- Nylon: Strong and flexible; special filament for printers exists.

5. Limitations

Now some limitations will be specified, focusing always on Fused deposition modelling technology (FDM):

- Non-profitable mass production: Additive manufacturing and 3D printing are designed for very short series of pieces (around 1 - 10 units). For longer series, production with these technologies is not profitable.
- Poor surface finish and low print speed compared to other 3D printing technologies.
- Small print size for desktop FDM printers. The typical size of these printers is 20x20x20 cm.
- Requires support structures to print parts with lower angles of 45 degrees.
- In general, holes smaller than 2 mm in diameter cannot be printed.
- The layers determine in which direction an object can absorb which forces. Tensile forces perpendicular to the direction of pressure can help to separate the layers more quickly. When designing objects to be printed using Fused Deposition Modelling, these properties must be considered.

6. Practical uses of 3D printing. Example

In this chapter it is intended to show, by a real example, all the steps to follow for, starting from a model or digital file, getting a real printed piece.

We started with the digital file: a 3D file resulting from the scanning of the lower part of a tiger jaw. The objective: to make a 3D printing copy of this piece. It was necessary to repair the scanned file. This is something very usual, because 3D scanners not always get the whole geometry of the real model. But this is not the objective of this guide.

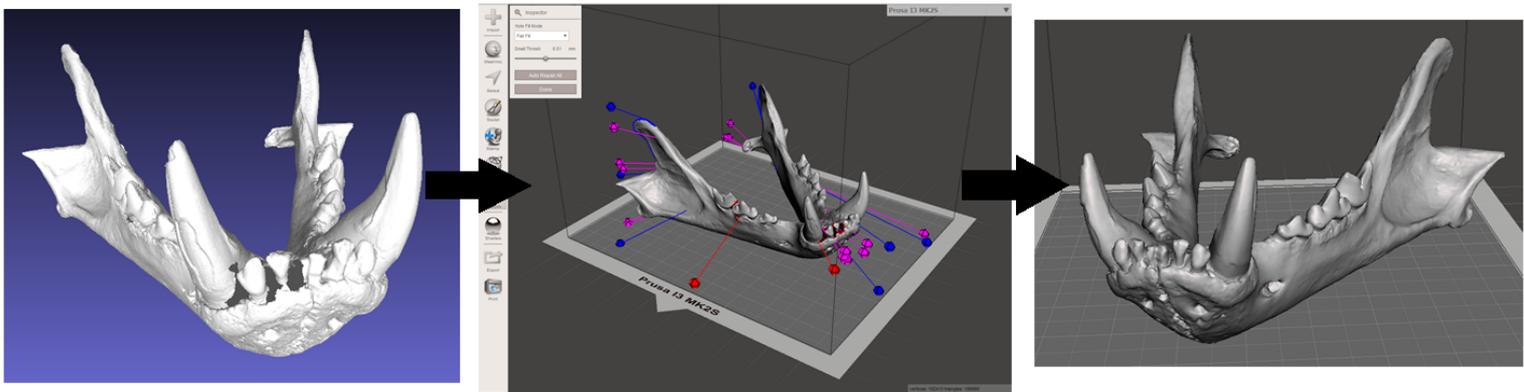


Figure 26. Repairing the 3D file. Source: CETEM.

So now, the repaired and fixed model must be exported or saved again in STL format.

The next step is to load the file in a software that allows to do, at least, the following things:

- Choosing an orientation in the printing bed. Rotating and moving the model.
- Scaling the model.
- Choosing the quality or layer height.
- Choosing the infill. Density among other issue.
- Generating the needed support structures.

So, in Cura the repaired model is imported. To configure the software to show the printing bed of the machine now is convenient. Cura has a very big data base of commercial 3D printers. If the used machine is not in that data base, it is also possible to create it from scratch.

This model is going to be scaled to 60%. 0.2 mm. is the chosen layer quality. A grid infill of 15% density is also chosen. Automatically support structures will be generated everywhere, and to get a good adhesion for the first layer, "first layer speed" will be set to 20 mm/s. working material is also chosen: PLA. Finally, the G-Code is saved in the device to insert in the printer by clicking on "Save to File".

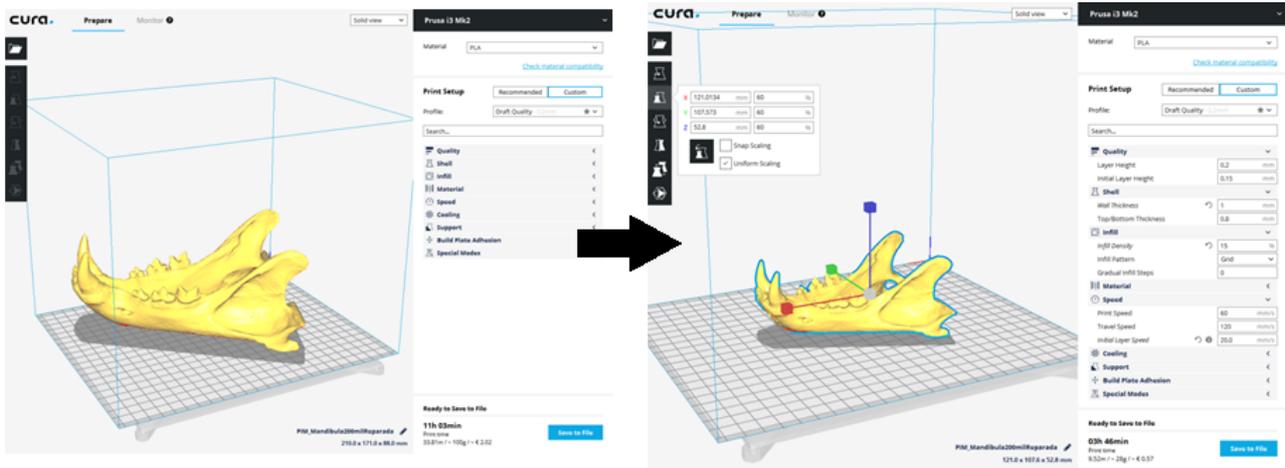


Figure 27. Importing, scaling and selecting parameters of the model, in Cura. Source: CETEM.

At this point, it is possible to go to the next step of the process: 3D printing itself. It is needed to go to the machine and, by the appropriated means, to launch the print job. The machine will be left working, and it is recommended to see it from time to time to see if everything goes correctly. It is important to say that it is highly recommended to check if the first printed layer is correctly stick to the bed or platform.

The next step is, obviously, to extract the printed piece. A spatula with rounded edges is used, in order not to scratch and not to damage the printing platform.

Once the piece has been extracted of the platform, it is time for the post-processing step, consisting of manually and mechanically removing the support structures, and sanding those parts of the piece that have marks (from support structures, for example). The printing bed should be correctly cleaned for the next print job.

As mentioned, to sand the marks of the support structures, among others, is very convenient.

If the result at any point of the printing process, or at the end of the printing process is not good, or some errors and shrinkages are seen, the following guide can be very helpful, to configure some printing parameters or parameters of the printer: <https://www.simplify3d.com/support/print-quality-troubleshooting/>

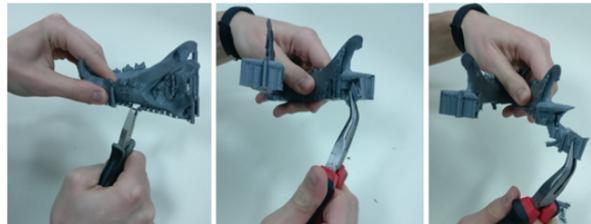
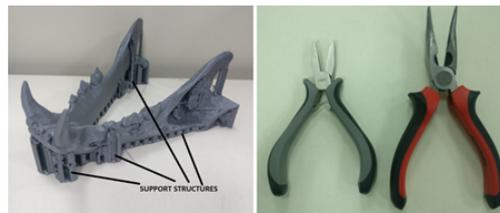
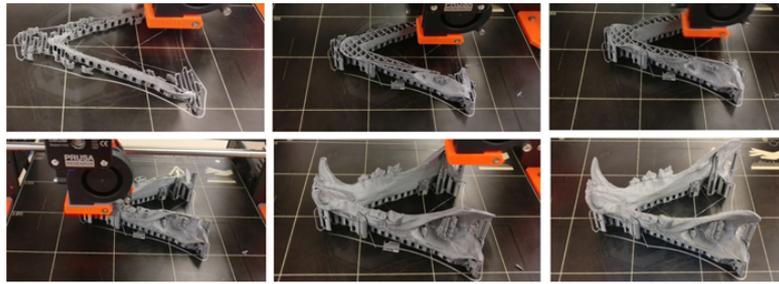
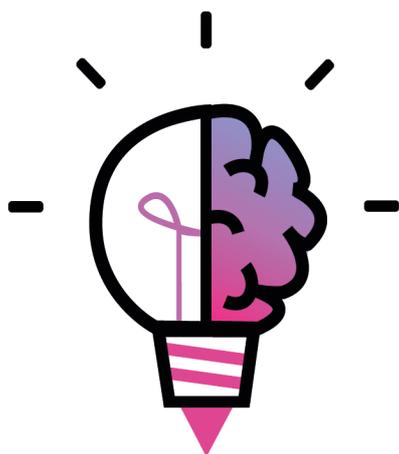


Figure 28. 3D Printing process, extracting piece, and removing supports and final piece. Source: CETEM.



3D4ELDERLY

CONSORTIUM OF PARTNERS:

