



ORTHOPUS

CUSTOMISABLE SKINS

Hello and welcome to this inlays 3D impression tutorial. Inlays are customisable parts of the ORTHOPUS Supporter, a robotic assistant for arm mobility, developed by **ORTHOPUS**.

This document will help you to print inlays by yourself. You just have to follow the steps of the tutorial: you'll find CURA print settings and all the useful details to do it.

ENJOY!

GENERAL INFORMATIONS

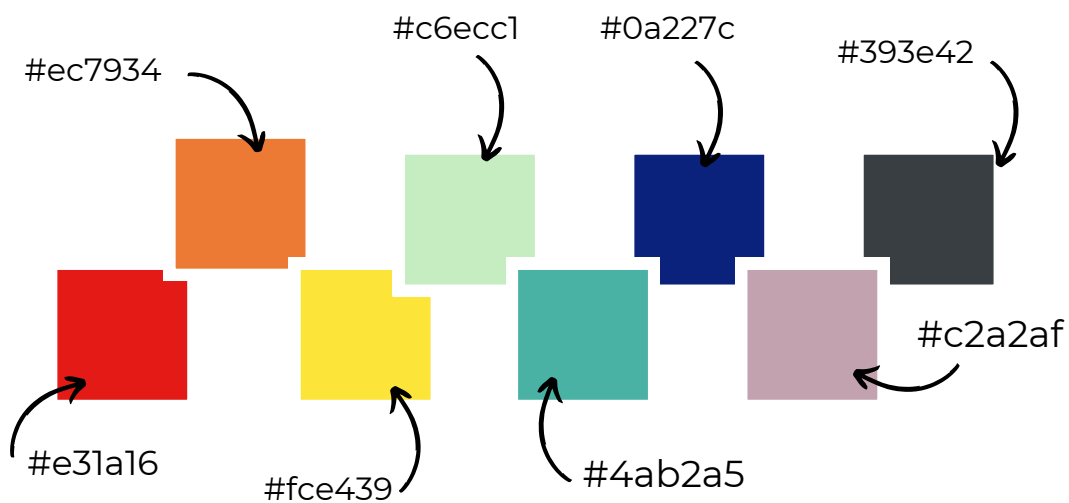
The printer we use is an Ultimaker **extended 2+** with a nozzle of **0.4mm**.

The filament used is **PLA** in 2.85mm diameter, from Francofil.

*
Making cost : **10 euros**
Printing time : **12h**
Manufacturing time : **2h**

ORTHOPUS' COLORS

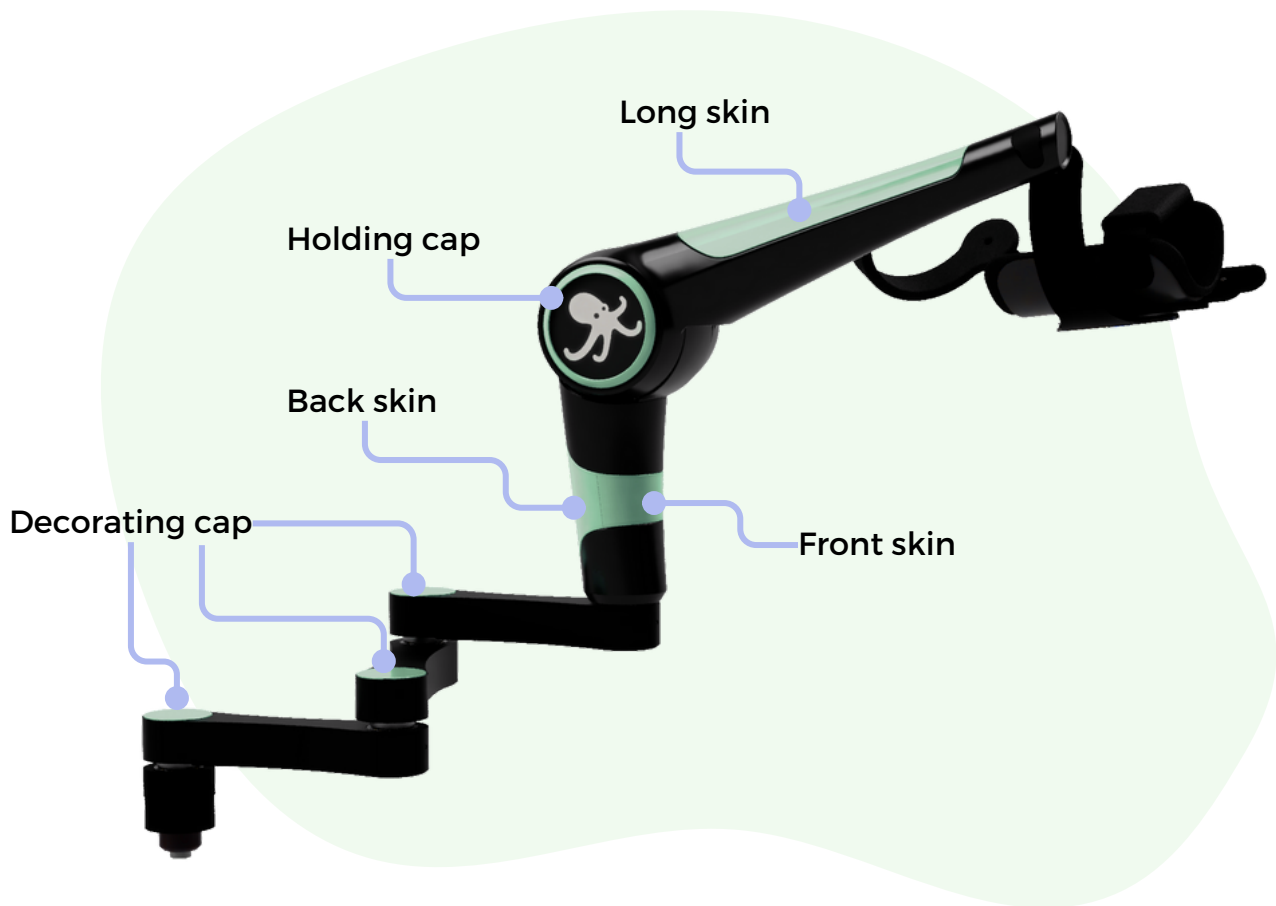
The ORTHOPUS colors range looks like this :



But you can choose your own color in PLA on different websites!

CUSTOMISABLE PARTS

of the ORTHOPUS SUPPORTER :



Files' names	Description	Quantity	Launch in print	Links
M-0204-22312-L2_HoldingCap	Holding Cap	2	2	
M-0204-1517-decorating_cap	Decorating cap	3	1	
M-0204-2233-L2_Skin	Long Skin	1	1	
M-0204-2214-L1_FrontSkin	Front Skin	1	1	
M-0204-2213-L1_BackSkin	Back skin	1	1	

Summary

0. Materials required

1. Printing files

1.1. Long skin

1.2. Holding cap

1.3. Front skin

1.4. Back skin

1.5. Decorating cap

2. Completion

2.1. Remove supports

2.2. Smoothing pieces

2.3. Installing magnets

3. Further informations

1. Printing files

1.1 Long skin

Print settings:

Nozzle temperature : 200-220° (depends of the filament)

Nozzle diameter : 0.4mm

About the piece :

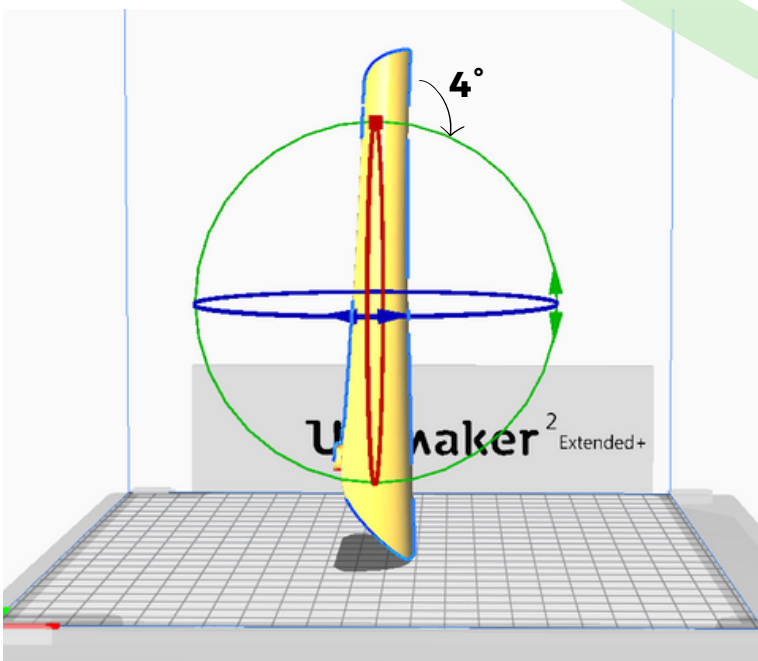
Printing time : 6h06

Weight of the matter : 46g

Filament's size used : 5.83m

The piece is printed vertically inclined 4° in back

.



Cura settings :

Layer height : 0.15

Side thickness : 1.05 (3 coats)

Bottom/top thickness : 0.8

Top/bottom design : concentric

Filling density : 25%

Print speed : 50mm/s

Moving speed : 90mm/s

Support : Yes

Support density : 15%

Platform grip : borders

Border width : 6mm

1. Printing files

1.2. Holding cap

Print settings :

The printing has to be done twice to have your set of two holding caps.

Nozzle temperature : 200-220° (depends of the filament)

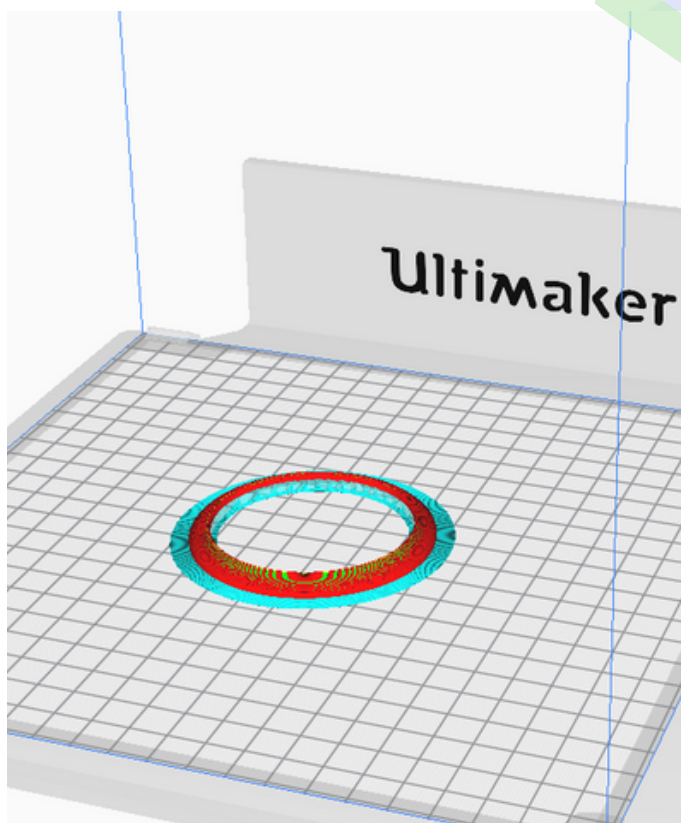
Nozzle diameter : 0.4mm

About the piece :

Printing time : 50min

Weight of the matter : 4g

Filament's size used : 0.55m



Cura settings :

Layer height : 0.15

Side thickness : 0.7mm (2 coats)

Top/bottom design : concentric

Filling density : 40%

Print speed : 55mm/s

Moving speed : 100mm/s

Support : Yes

Support design : zigzag

Support density : 10%

Z distance : 0.2

X/Y distance : 0.35

Platform grip : none

1. Printing files

1.3. Front skin

Prints settings

Nozzle temperature : 200-220° (depends of the filament)

Nozzle diameter : 0.4mm

About the piece :

Printing time : 1h

Weight of the matter : 5g

Filament's size used : 0.69m

Cura settings :

Layer height : 0.15

Side thickness : 0.7mm (2 coats)

Top/bottom design : concentric

Filling density : 20%

Filling design : cubic

Print speed : 55mm/s

Moving speed : 100mm/s

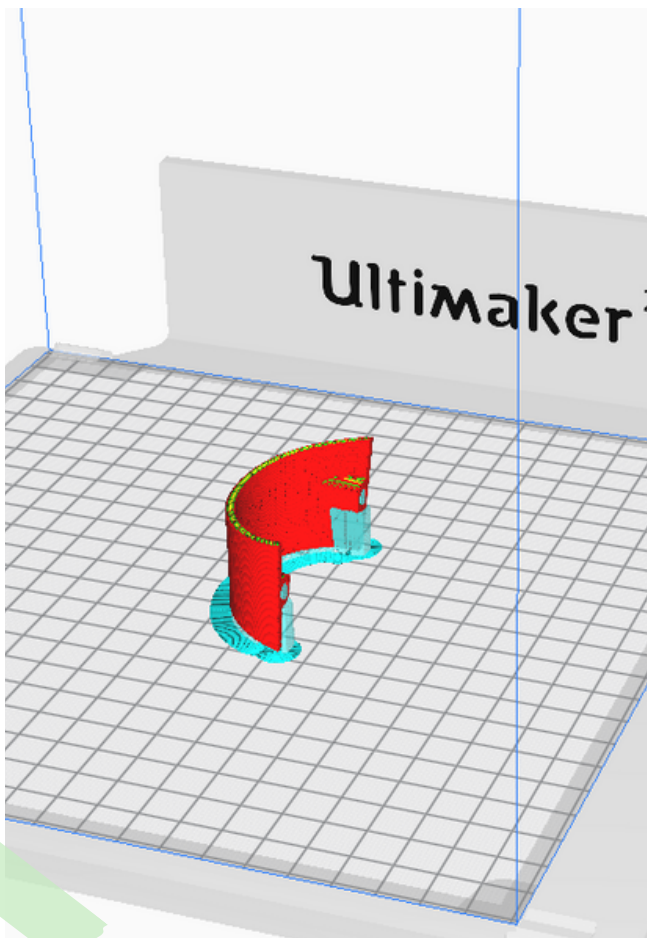
Support : Yes

Support design : zigzag

Support density : 10%

Platform grip : border

Bond thickness : 5mm



1. Printing files

1.3. Back skin

Prints settings

Nozzle temperature : 200-220° (depends of the filament)

Nozzle diameter : 0.4mm

About the piece :

Printing time : 2h30

Weight of the matter : 16g

Filament's size used : 1.98m

Cura settings

Layer height : 0.15

Side thickness : 0.7mm (2 coats)

Top/bottom design : concentric

Filling density : 20%

Filling design : cubic

Print speed : 55mm/s

Moving speed : 100mm/s

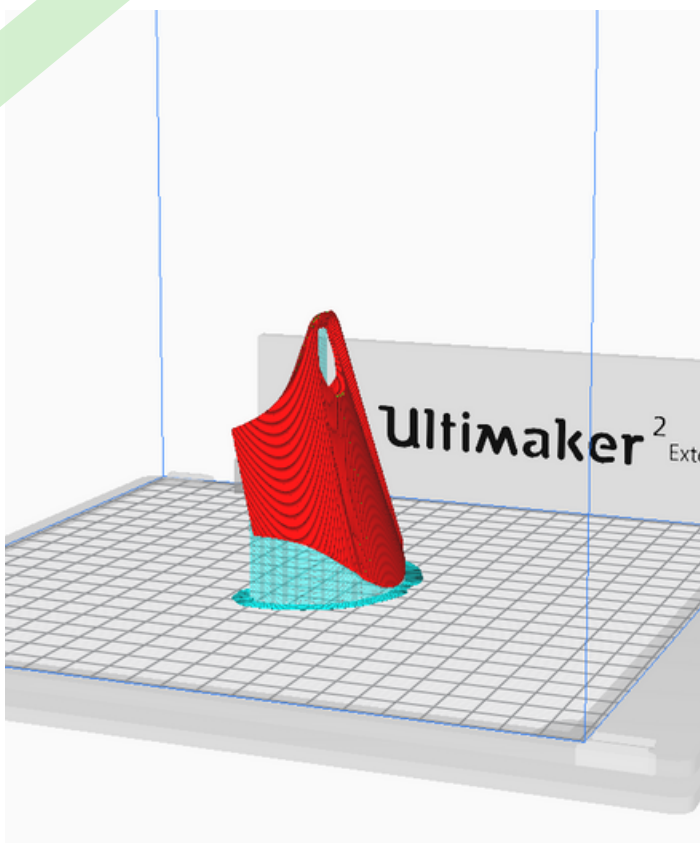
Support : Yes

Support design : zigzag

Support density : 10%

Platform grip : border

Bond thickness : 5mm



1. Printing files

1.3. Decorating cap

There are 3 decorating caps in the same file. To print one time in a textured plate.

Prints settings

Nozzle temperature : 200-220° (depends of the filament)

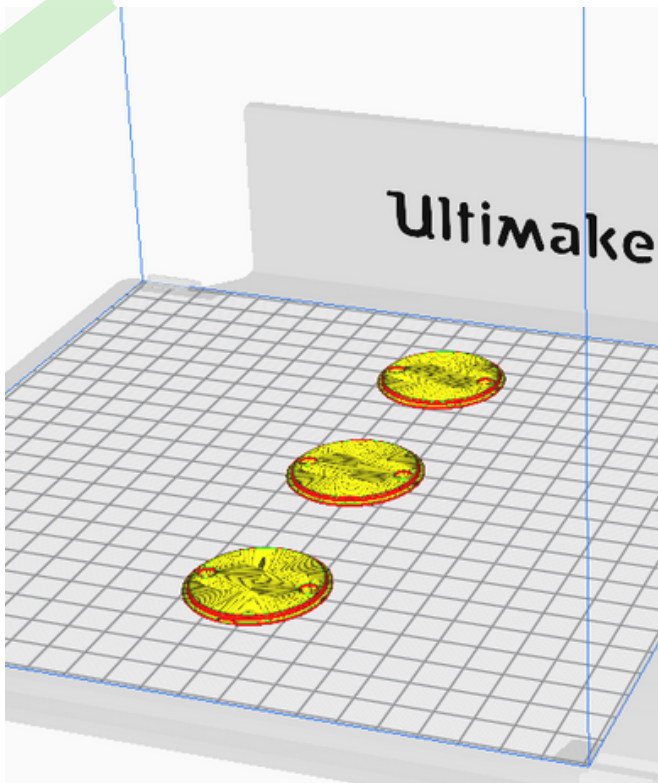
Nozzle diameter : 0.4mm

About the piece :

Printing time : 1h53

Weight of the matter : 9 g

Filament's size used : 1.10m



Cura settings

Layer height : 0.15

Side thickness : 0.7mm (2 coats)

Top/bottom design : concentric

Filling density : 20%

Filling design : cubic

Print speed : 55mm/s

Support : No

2. Smoothing pieces

2.1. Remove supports

Material required :

- 3D printed pieces
- A cutter
- A file
- A plier



2. Completion

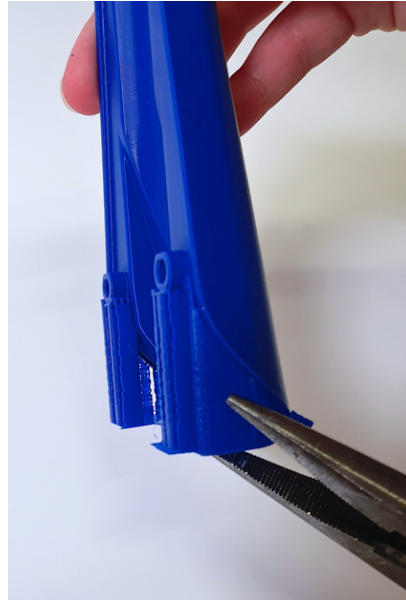
2.1. Remove supports



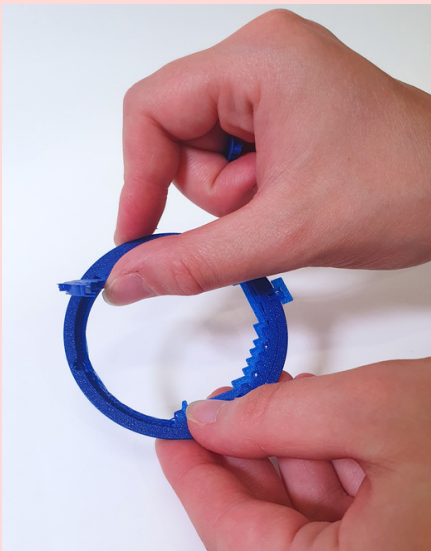
1) Use a cutter or a scalpel to remove the matter around the 3D print piece



2) Remove the fine support of inlays thanks to scalpel



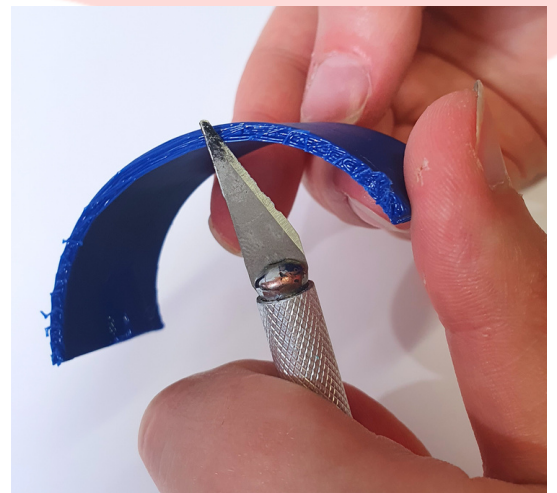
3) Take a plier and remove supports. It's easy to do. You can also use a screwdriver or a cutter.



4) Manually, remove the support under the holding caps.

5) Thanks to the scalpel, remove the excess of supports by scratching pieces.

Repeat this operation on every printed piece.



2. Completion

2.2. Smoothing pieces

Material required :

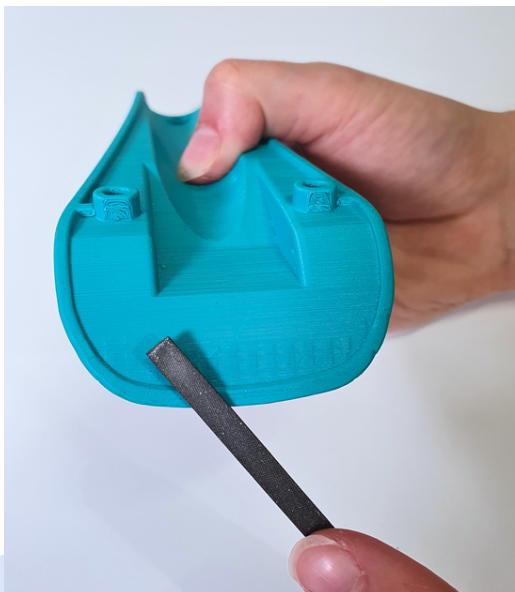
- Set of skins without supports
- A plate file
- A scalpel
- Sandpaper
- A lighter



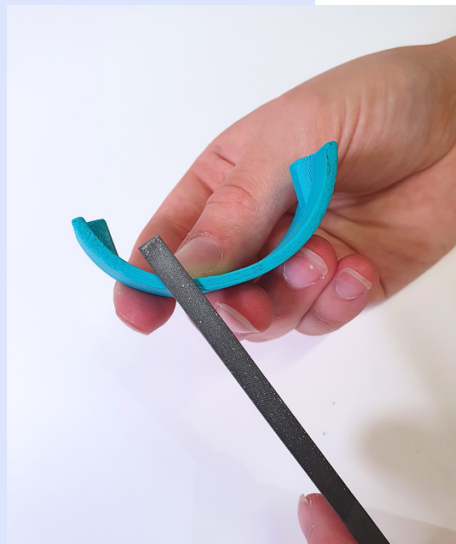
2. Completion

2.2. Smoothing pieces

1) Thanks to a plate file, sand the parts where you removed supports



Repeat the operation on every oriented piece.

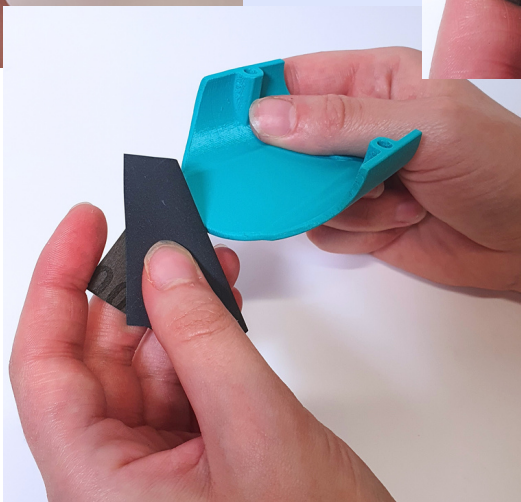
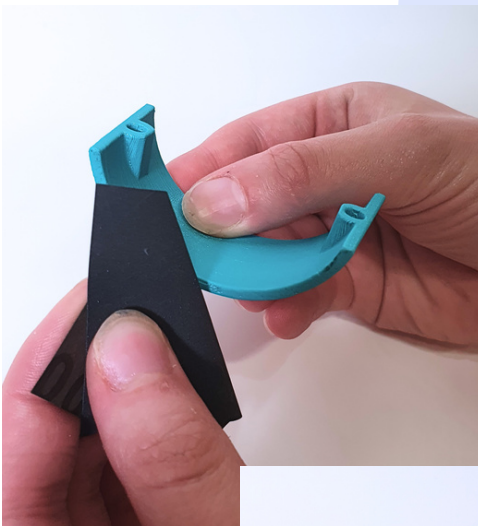


Be careful at the corners and where the magnets will be. Take care too to the buttons borders.



Repeat this operation on the long skin and on the holding caps

2) For fine-tuning, use really fine sandpaper (grain size:100) on every printed pieces.



3) Heat up the piece on its length, where the support was, thanks to the lighter.



Don't do it too long, or the plastic will melt !



4) Pass a finger on this part to smooth it.

5) Do the same with the other pieces.



2. Completion

2.3. Installing magnets



Material required :




- **Magnets :**
 - 6 magnets $\text{Ø}4 \times 1 \text{ mm}$ with a strength of 530g
 - 7 magnets $\text{Ø}4 \times 3 \text{ mm}$ with a strength of 530g
 - 6 magnets $\text{Ø}4 \times 3 \text{ mm}$ with a strength of 530g
- Super glue (ex: Loctite Super glue-3)
- A toothpick

2. Completion

2.3. Installing magnets



Notches positions

-  $\varnothing 4 \times 1\text{mm} \times 6$ magnets
-  $\varnothing 4 \times 2\text{mm} \times 6$ magnets
-  $\varnothing 4 \times 3\text{mm} \times 7$ magnets

2. Completion

2.3. Installing magnets



1) Place your magnets in front of those already placed on the ORTHOPUS Supporter. It will help you to know the orientation of the magnetization.

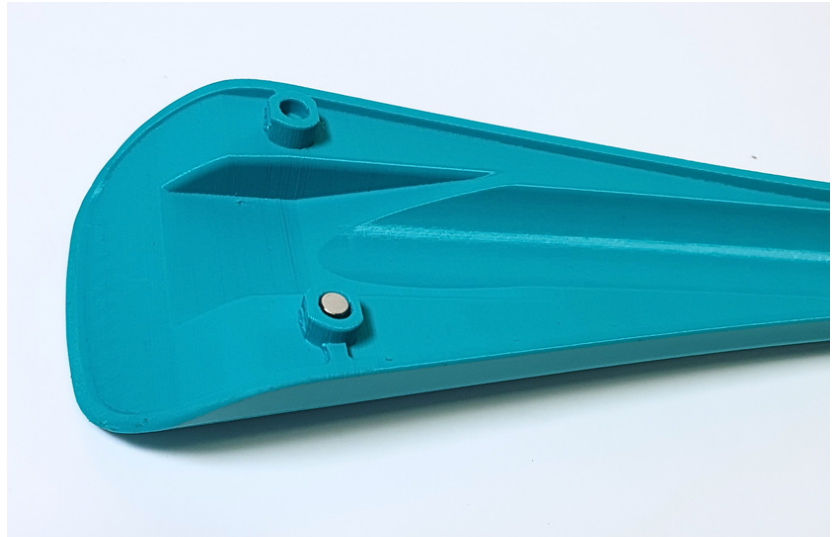
2) Using a marker, label the visible side of the magnet. The labeled side will be the side that adheres to the printed inlays.



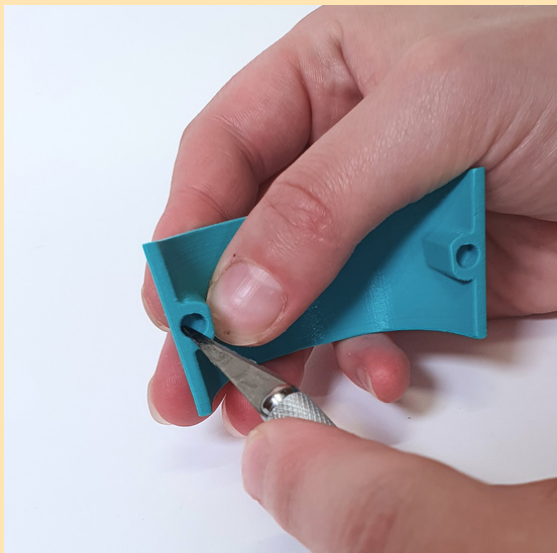
Do this step on all the ORTHOPUS Supporter



3) Check if your magnets fit well in the inlays notches.



There should be a little gap between the notch diameter and the magnet diameter. The height of the magnet should not exceed the notch of the printed inlay. .



If the magnets don't fit in the notches, you can resize the notch with a scalpel.



4) Put liquid glue on the marked side with the toothpick



1.5) Put liquid glue in the inlay notch with the toothpick and put your magnet on it.



Repeat steps 4 and 5 for the other magnets.

Repeat the past operations for all the other inlays magnets, **except the two holding caps.**



2. Completion

2.3. Installing magnets

Assembling holding caps



1) Remove the films on the plexiglass.



Where can I find plexiglass?

You can cut two 61.5mm diameter circles on a plexiglass leaf. You can also reuse the ones on your old custom logo disc.

2) Place your plexiglass in the holding caps, as deep as you can.

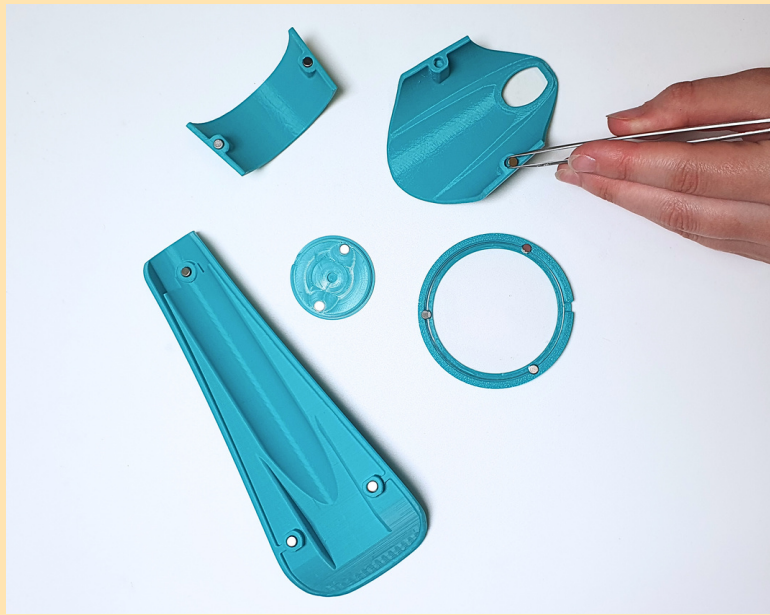


3) Put glue on the marked side of your $\text{Ø}4 \times 2\text{mm}$ magnet. Put glue in the holding caps notch. It's not easy to do: don't put glue on the plexiglass.



4) Place your magnet. Marked side has to be in contact with the inlay. A side of the magnet is in contact with the PLA, and the other is in contact with the plexiglass.

Repeat these steps with the other notches of the holding caps. There are 6 magnets to stick on these pieces.



After completing the task, allow it to **dry for a minimum of 30 minutes**

Now, you can place your inlays on
the ORTHOPUS Supporter



3. Further informations

The tutorial of custom skins for the ORTHOPUS Supporter ended here.

But you can continue to custom inlays with colors, paintings, or by smoothing outside part of the skins. You can print with wood or colored filament, you can even add fur on it!

There are some of our ORTHOPUS Supporter's customs :



If you want to change the image in the custom logo disc, follow this tutorial:
<https://orthopus.com/wp-content/uploads/2023/03/Changer-image-sur-l'ORTHOPUS-Supporter.pdf>

Share your creations with us on this mail : usercare@orthopus.com or with [#myorthopussupporter](https://twitter.com/myorthopussupporter) on social medias.

Visit our website : <https://orthopus.com/>

ORTHOPUS Team

