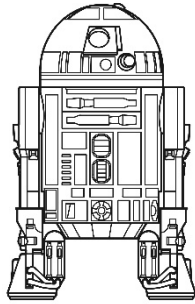


# Printed Droid Print Guide for Mr. Baddeleys R2-D2

v1.2 2020/2 by Nitewing

[www.printed-droid.com](http://www.printed-droid.com)



This print guide is based on my own personal experience. It is therefore quite possible that some settings do not apply to your setup. I am therefore not responsible for any misprints. The information in this guide is provided as-is with the hope that it may be useful. You agree to use this information entirely at your own risk.

Creative Common License v4.0 BY NC SA

This document is licensed under a **Creative Commons Attribution-Non CommercialShareAlike 4.0 International License**.

In Human readable format:

**You are free to:**

**Share** — copy and redistribute the material in any medium or format.

**Adapt** — remix, transform, and build upon the material.

- The licensor cannot revoke these freedoms as long as you follow the license terms.

**Under the following terms:**

**Attribution** — You must give appropriate credit, provide a link to the license, and indicate if changes were made. You may do so in any reasonable manner, but not in any way that suggests the licensor endorses you or your use.

**Non Commercial** — You may not use the material for commercial purposes.

**ShareAlike** — If you remix, transform, or build upon the material, you must distribute your contributions under the same license as the original.

- **No additional restrictions** — You may not apply legal terms or technological measures that legally restrict others from doing anything the license permits.

Additional details are located at: <http://creativecommons.org/licenses/by-nc-sa/4.0/legalcode>

All finished image rights, belong to respective image property owners.

## General Information:

It is absolutely possible to print the droid with a print bed of 200x200mm. However, it may be necessary to split some printed parts. The most important print files exist as a split version.

It is recommended to use a print space of at least 300x300mm, that allows the printing of most files without splitting them. To print the dome and the body rings in one part, a print space of 500x500mm is required.

In my opinion it is more economical and much faster to use 2 printers with 300x300 than one printer with 500x500. Also keep in mind that a misprint from a split dome file is not as bad as the misprint of the whole dome at 80%.

It will take between 20 and 25 rolls of filament to print the Droid. This depends on how reliably your printer can create acceptable parts.

I recommend that you print this frame in ABS (please note that a temperature controlled chamber is recommended).

PLA+ should work also. PLA can work for the non structural parts.

CAUTION: PETG can be too flexible so keep that in mind.

## Printer Calibration:

Make sure your printer is perfect calibrated for printing large parts with accurate X, Y, and Z dimensions (using large calibration objects), with a level bed and proper extrusion. You should get zero warp or lifting on these parts if your printer is set up correctly and your bed is leveled properly. If you are getting lift or warp the parts will not fit correctly – get that problem fixed before continuing.

**Before starting to print the Droid make some test prints of larger cubes.**

**For example:**

**Print a cube 100x100x100mm and check if all sides have the same length. An accurate dialed in printer is essential for a good endresult!**

**Use only one printer for printing - unless you know the printed parts from different printers have exactly the same size!**

## Controlling Warping

Very big 3D printed parts are always prone to warping from various causes, including shrinkage of the printed plastic, and temperature changes in the environment over the long print time. The body rings and dome parts are less prone to warping than the feet parts, but warp can still happen.

Suggestions for controlling warp:

Use a filament that shrinks less, like PLA+. If you want to use a filament like ABS that shrinks a lot then you may need a printer with a heated, temperature controlled print chamber.

Control drafts that cause the part to cool unevenly:

- Relocate the printer to a location with a consistent temperature away from breezes from air conditioning.
- Enclose the printer (or you could even cover it with a plastic bag). This will also retain some heat so the printed part will shrink somewhat less while it is printing. Do not tightly enclose a printer if you are printing PLA/PLA+, you really do not want to raise the chamber temperature for PLA/PLA+.

Warp in the First Inch: Some printers have an odd problem where the print will warp within the first inch, and adding support structure there does not help. If you have this problem then you may benefit from the helper discs. - Add .5mm to 1mm thick helper disks. You can stretch the helper disks even taller if you have warp farther up. Also reduce the temperature of the heated bed!

If you have problems achieving an acceptable print – ask for help! There are probably many people that have your same model printer and have already overcome the same issue.

## Check for good layer bonding:

- If you see any splits between the layers, then the part is not structurally solid.

It is not a problem with the dome or the panels. There you can fix it with filler or something.

But with structural parts like the shoulders, the drive system or the legs, the problem should be fixed before you continue

Some common causes of poor layer bonding are:

- Layer height too high. Try reducing layer height below .2 mm, and the individual extrusions will be pressed more tightly against each other in order to achieve the same extrusion width this will result in stronger layer bonds.

- Incorrect extrusion temperature.
- Inconsistent extrusion due to printing too fast, extrusion temperature too low, clogged extruder gear, etc.
- The room is too cold, or excess drafts causing excess shrinkage and warping during the print.

See the section above: Controlling Warp

## Joining the pieces:

**Before gluing the sections together always make a Dry Fit!**

**Use proper protection** when working with these! Eye protection and respiratory protection, as well as gloves, are to be used when necessary. Read the manual of the products you are working with! And protect your furniture with old newspaper etc. Acetone etc. is flammable so be carefull and use it in well ventilated areas or outside. Keep it out of reach of children!

### Preparations:

**For most glues, it is advisable to prepare the surface: sand it to increase the surface area, remove grease from fingerprints etc. Follow the manual!**

	Superglue	Acetone	Plumbers Cement	2k Epoxy	Polyurethane Glue	Hotglue
ABS	✓	✓	✓	✓	✓	✓
PLA / PLA+	✓	✓	✓	✓	✓	✓
PETG	✓			✓	✓	✓

### Superglue:

For most jobs, cyanoacrylate, or superglue, is the best option for gluing 3D printed parts together. It is easy to use and a quickly curing glue. You can get excellent results, a strong bond, and an almost invisible seam. Since it cures in a matter of seconds, think twice before using it and make sure you're prepared! Do your best to align parts properly before it cures, and hold it tightly in place for about one minute. Leave it for an additional couple of minutes to cure completely.

Most 3D printed materials will successfully bond with superglue, except for flexible material. When cured, cyanoacrylates form a thin rigid layer, which can be easily broken if applied to elastic material.

### Acetone:

Acetone (paint thinner) can be used for bonding 3D printed parts made of ABS and PLA. Any material soluble in acetone is a good candidate. This type of bonding gives very strong bond, and invisible seam.

In order to bond two pieces with acetone, apply thin layer of acetone on both surfaces being bonded. Be careful not to use too much acetone, because there is a risk of damaging 3D printed parts (especially if the parts are thin). The acetone will dilute a thin layer on the surface, and when you stick two pieces together, these layers will "mix". During this drying time, you may need to fix the pieces being glued together with some tape or clamps. After a couple of minutes or hours, depending on the size of the parts and amount of acetone used, the acetone will evaporate and a strong compact plastic layer will be formed. This may be the best way to bond two pieces made of ABS or PLA. It is very strong and, if done properly, it's invisible.

### Plumbers Cement:

Plumbers cement is almost the same like Acetone, but it's colored so the seam would be visible.

### 2K Epoxy:

Epoxy is another great way to join two pieces together. It will work on most filaments.

The nice thing about epoxy is that you can use it both as a glue and as a filler. If there is a void that has to be filled or seems to be hidden, use epoxy.

Compared to other methods, epoxy requires more work to get a good bond. It usually comes packed in two separate containers that have to be mixed prior to bonding. One has epoxy resin, while the other one contains a hardener. These components have to be mixed in a certain ratio (check the user manual for mixing, it's different for every type of epoxy). When thoroughly mixed, apply it to a surfaces being bonded. Attach all the pieces you want to bond and fix it properly. Depending on the mixing ratio and type of epoxy, curing can take from couple of minutes to couple of hours.

When cured, epoxy can be sanded and painted. **Sand outside and use a face mask!**

For small parts that align easy I recommend 5min epoxy. For Body Rings and filling or smoothening surfaces use 45min epoxy! Epoxy gets hot during hardening so keep that in mind. 5min epoxy will get much hotter than 45min epoxy and thin parts could deform! So use a slow hardening epoxy whenever possible!

#### Polyurethane Glue:

PU (polyurethane) glue is very easy to use and creates a pretty strong bond. The disadvantage is that it needs thickness of at least couple of millimeters for proper bonding. That makes it hard to hide bonding seams.

**So PU glue is an option, but not the best choice!**

#### Hot Glue:

Hot glue is an easy alternative to gluing 3D printed parts together. It's important to note that while the bond is pretty strong, applied hot glue will be visible. Hot glue requires at least 2-3 mm in thickness, so you should consider this when designing parts for 3D printing. So it's recommended to use it for parts glued inside etc.

## Welding / Soldering / 3D Pen

You even can combine 2 parts with welding or soldering. It's mostly recommended for PLA. This method also works for most other filament types, but is **not** advisable for ABS and other plastics that emit fumes without wearing respiratory protection!

- Take the parts and make sure on both sides is a cavity that can be filled.
- Take a soldering iron and set it to around 200°C.
- Take a length of PLA filament.
- Melt the filament with the soldering iron and use it as solder when combining the two pieces. Make sure that at least some filament gets into the cavities and sticks there - it can help to stick the soldering iron into the goop in to force it to merge with the infill/walls
- As the PLA cools and hardens, the joint is usually tougher than the actual layer boundaries.

#### 3D Pen

Instead of using a soldering iron you might also use a 3D printing with the same filament you're connecting.

#### Inserts

On a different note, a soldering iron is also a very good solution to make inserts into PLA - heat up the metal insert (like a nut) and press it into an undersized hole, and it will mold the plastic around itself into a perfect fit without any glue.

## Print settings:

As mentioned at the beginning it is possible to print most of the droid in PLA / PLA+. Structurally stressed areas should / must be printed in ABS or PETG.

**I made an Print list file that can be found here:**

[https://docs.google.com/spreadsheets/d/1ULDnJ6An2dtJzS48Dhf\\_XtPFmkV7JypzcvKuDSr6IJs/edit?usp=sharing](https://docs.google.com/spreadsheets/d/1ULDnJ6An2dtJzS48Dhf_XtPFmkV7JypzcvKuDSr6IJs/edit?usp=sharing)

**This covers infill etc. in detail**

#### General Print settings:

**Basic Rule: Outer Walls are important for strength and later sanding. The layer height defines the details. Outer walls are more important than infill! The nozzle diameter affects the overall level of detail almost exclusively in the horizontal plane (parallel to the print surface) and the print time**

#### Infill:

A general infill ~35% should be fine for most parts. Panels, Doors or the Dome would even be fine with much less infill. But stressed parts like the drive frame, shoulders and ankles need an infill of ~50%

Be aware that there are 2D, 3D and also 3D Flex infills. I use Cura and recommend in Cura the cubic or subcubic infill.

#### Layer Height:

0.2 - 0.3 (depending on the needed detail compared to the needed print time)

Some detailed parts should be printed with .10 or better 0.12mm height  
Most printers make better results with a multiple of 0.04  
So the layer heights for best result would be 0.08, 0.12, 0.16, 0.20, 0.24, etc...

#### **Perimeters:**

Perimeters are very important for structural integrity. While parts such as the holoprojector can be printed with 2 outer walls, at least 3 are generally recommended.

Because with 3 perimeters with a width of 0.4mm there is not much space left later when grinding.

But with structural parts I would actually recommend 4-6 perimeters.  
 $5 \times 0.4\text{mm} = 2\text{mm}$  thick outer wall

#### **The Nozzle and Layer height & Perimeters:**

In general, the nozzle diameter must also be observed.

With a 0.4mm nozzle I need 5 walls to get to 2mm, with a 0.5mm nozzle only 4.

You can also print 0.5mm thick lines with a 0.4mm nozzle without problems, you just have to set it correctly in the slicing software.

I would not recommend more than 0.8mm nozzles without changing the hotend. With 1.75mm thick filament and a 0.8mm nozzle, it is close to the limit of what a hotend can melt.

A Volcano Hotend is therefore recommended for nozzles over 0.8mm.

Therefore I recommend nozzles between 0.4mm and 0.6mm.

#### **The nozzle affects the print time.**

**A larger-diameter nozzle lays down wider perimeters**, which means that it uses fewer perimeters than a smaller-diameter nozzle to print a wall of the same thickness.

#### **The nozzles also affect the maximum layer height.**

The layer height should not exceed 80% of the nozzle diameter. If you are using the standard 0.4mm nozzle, the maximum layer height should be 0.32 mm. However, with a 0.6mm nozzle, it's possible to achieve up to a 0.48 mm layer height.

#### **Effect on mechanical properties:**

Another advantage of using larger nozzles is an increased toughness of printed objects. An impact resistance test revealed that the objects printed with the 0.6mm nozzle **absorbed up to 25.6 % energy** more than those printed with a 0.4mm nozzle.

## **Let's start printing**

The R2-D2 has different iterations and they are "compatible" as long as you use all parts from a certain "module". For example: The V2 Dome is one Module, MK3 Body is a module, V2 legs is a module.

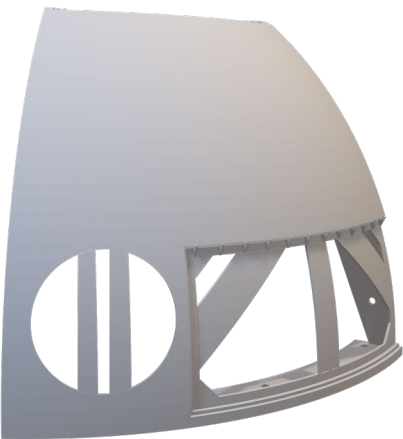
But using the MK3 Dome with V2 Body and MK3 legs would also work.

If in doubt ask in the group.

I printed the V2 Dome with MK3 Body. But there are other possible combinations.

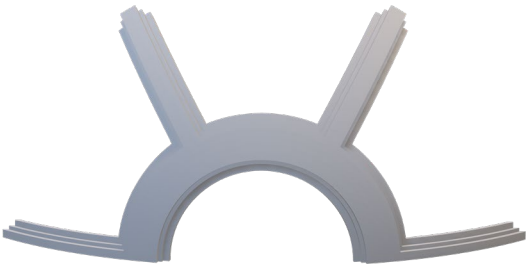
Print all pieces flat or the pictures show the recommended print orientation. If not mentioned explicit, then no support needed!

V2 Dome:  
I printed the 200mm cut version



File Name: 200-Dome1 – 6 (6 parts in total)  
print standing

Minimum Bed Size:	200
Layer height:	0.2-0.3
Infill:	15%
Outer walls:	4
Recommended Filament:	PLA+ or stronger
Module:	V2 Dome



File Name: 200-UpperPieA & B (2 parts in total)  
Print it like this

Minimum Bed Size:	200
Layer height:	0.2
Infill:	15%
Outer walls:	3
Recommended Filament:	PLA+ or stronger
Module:	V2 Dome



File Name: Lowering1-7 (7 parts in total)  
Print flat

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	20%
Outer walls:	3-5
Recommended Filament:	PLA+ or stronger
Module:	V2 Dome



File Name: 200-Panel1-16 (16 parts in total)  
Print flat on the long side

Minimum Bed Size:	200
Layer height:	0.2
Infill:	15%
Outer walls:	3
Recommended Filament:	PLA+ or stronger
Module:	V2 Dome



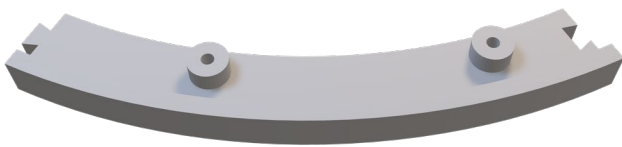
**File Name:** 200-Pie1-6 (6 parts in total)  
*Print flat on short side*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2
<b>Infill:</b>	15%
<b>Outer walls:</b>	3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome



**File Name:** Topdome  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12
<b>Infill:</b>	15%
<b>Outer walls:</b>	3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome



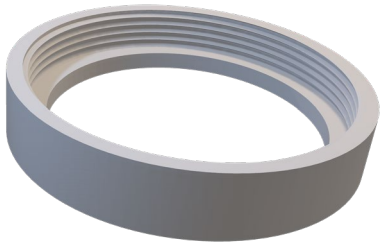
**File Name:** Uppering1-8 (7 parts in total)  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	20%
<b>Outer walls:</b>	3-5
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome



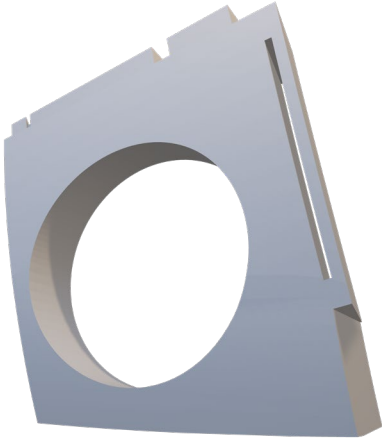
**File Name:** FrontPSI & RearPSI (2 parts in total)  
*Print flat maybe with a little support*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 (because of the thread)
<b>Infill:</b>	10-15% (Honeycomb)
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger <b>CLEAR Filament</b>
<b>Module:</b>	V2 Dome



**File Name:** FrontPSIRing & RearPSIRing  
(2 parts in total)  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 (because of the thread)
<b>Infill:</b>	15%-20%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome



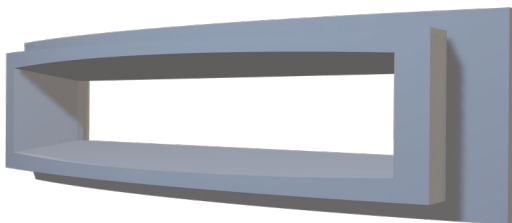
**File Name:** RadarEye  
*Print standing flat like this*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.25
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome



**File Name:** RadarLense  
*Print flat*

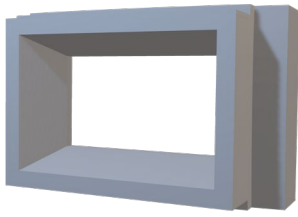
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.08 – 0.12 (because shape)
<b>Infill:</b>	~15
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome



**File Name:** LargeLogicLight  
*Print flat like this with support*

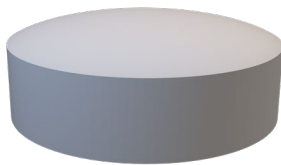
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.25
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome





**File Name:** *SmallLogicLight (needed 2x)*  
*Print flat like this with support*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.25
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome



**File Name:** *Button (needed 2x)*  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.25
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome

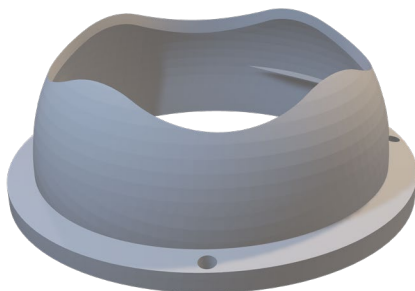
The HoloProjectors are needed 3x I will only cover the settings once so just transfer it to the other parts.  
 If you want moving HoloProjectors look at Thingiverse (will add in the next version)

The HoloMain is always the same but the surrounding is different for the Pie Holo.



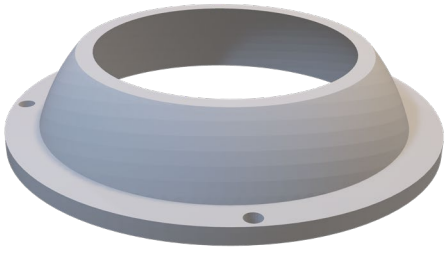
**File Name:** *HoloMain (needed 3x)*  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12 - 0.20 (because of the details)
<b>Infill:</b>	~15%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome

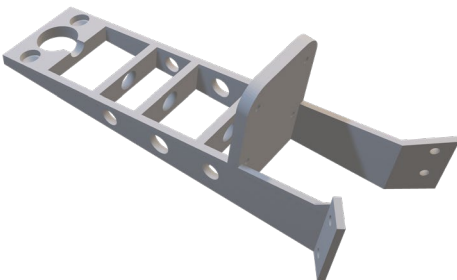
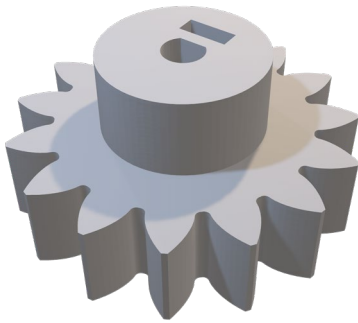
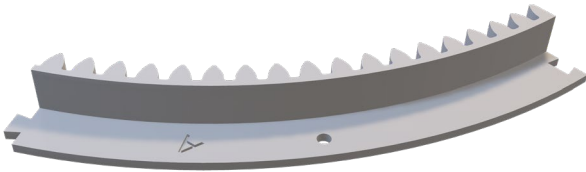


**File Name:** *HoloFrameA (needed 2x)*  
*+ Pie Version*  
*Print flat like this with support*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12 - 0.20 (because of the details)
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome



**Dome gear drive:**  
I printed the cut version



Please note that you might need a different motor frame

**File Name:** HoloFrameB (needed 2x)  
+ Pie Version  
*Print flat like this with support*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12 - 0.20 (because of the details)
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	V2 Dome

**File Name:** 379 Ring - 360 inner depth 15 22mm  
holeA – F (9 parts in total)  
*Print flat*

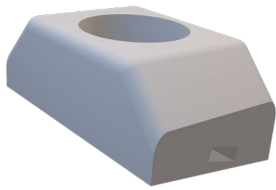
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4-5 (because of mechanical stress)
<b>Recommended Filament:</b>	PLA+ or stronger <b>PETG or ABS recommended</b>
<b>Module:</b>	MK3 Body

**File Name:** DriveGear  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4-5 (because of mechanical stress)
<b>Recommended Filament:</b>	PLA+ or stronger <b>PETG or ABS recommended</b>
<b>Module:</b>	MK3 Body

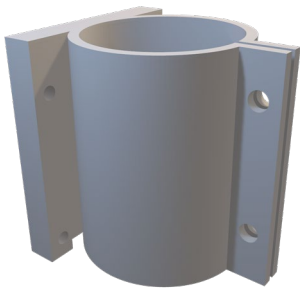
**File Name:** MotorFrameA-24mm  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4-5
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** MotorFrameA-24mm  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4-5
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



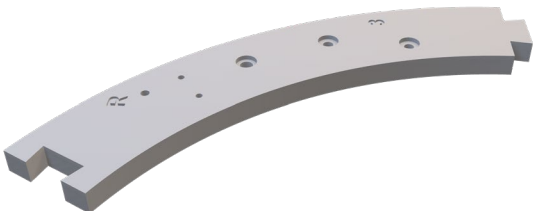
**File Name:** Motorbracket - 37mm  
*Print standing like this*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	25 - 35%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body

You might need a different Motor Bracket

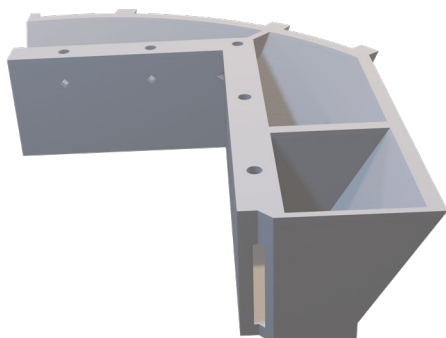
## MK3 Body:

I printed the cut version



**File Name:** MB-Uppering1 (Top Ring)  
(6 parts in total)  
*Print flat*

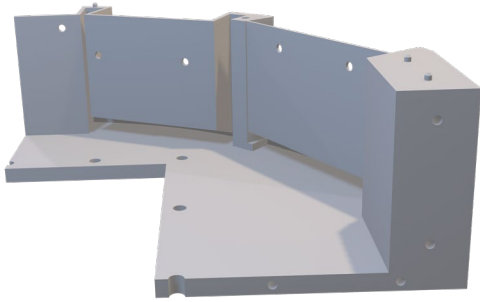
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4-5
<b>Recommended Filament:</b>	PLA+ or stronger <b>PETG or ABS recommended</b>
<b>Module:</b>	MK3 Body



**File Name:** Skirt\_a - b (Cut Version Large)  
(4 parts in total)  
*Print flat*

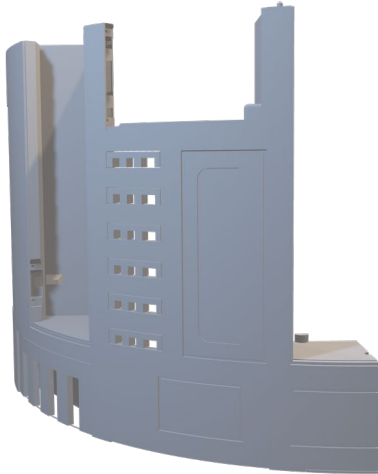
<b>Minimum Bed Size:</b>	300
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	15 - 20% (absolutely no stressed part)
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body

The “cut version small” has same settings but will fit onto a 200mm bed. But has 6 parts. Same settings for the singleprint skirt.



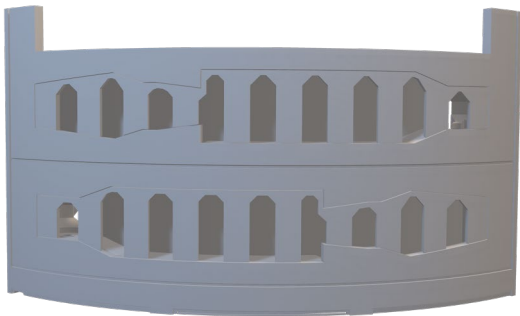
**File Name:** Mainbody 1a – 1d (4 parts in total)  
*Print flat*

<b>Minimum Bed Size:</b>	300
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	25 - 35%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** Mainbody 2a – 2e (5 parts in total)  
*Print flat*

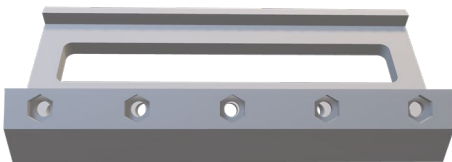
<b>Minimum Bed Size:</b>	300
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	25 - 35%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** Mainbody 3a – 3f (6 parts in total)  
*Print flat*

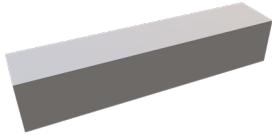
<b>Minimum Bed Size:</b>	300
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	25 - 35%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body

This pictured part needs 0.2 layer height due to the built in support. All other Mainbody 3 files can go with 0.3mm if you want



**File Name:** Nutholder :D  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	25 - 35%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



This small peg holds the nuts in the skirt

**File Name:** SkirtPeg (12x needed)  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 - 0.30
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body

## Utility Arms:



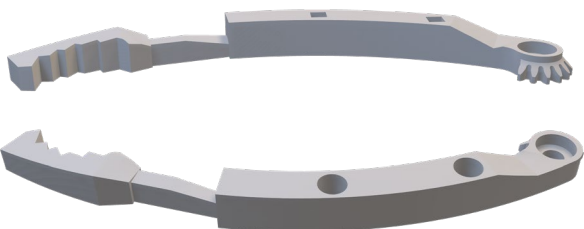
**File Name:** LowerDataPort  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.08 – 0.12
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2 - 3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



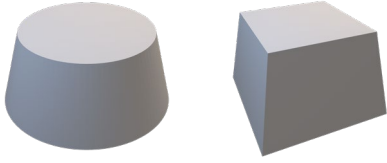
**File Name:** UpperDataPort  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.08 – 0.12
<b>Infill:</b>	~15%
<b>Outer walls:</b>	2 - 3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



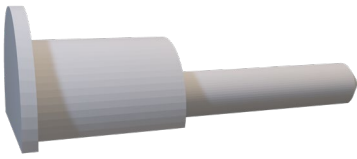
**File Name:** UtilityArmA & B (2x needed)  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	20%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** UtilityPlugA & B (4x needed)  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	20%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



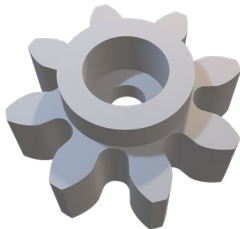
**File Name:** UpperUtilityPin  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	70%
<b>Outer walls:</b>	5
<b>Recommended Filament:</b>	PLA+ or stronger ABS or PETG recommended
<b>Module:</b>	MK3 Body



**File Name:** UtilityLowerPin  
*Print flat*

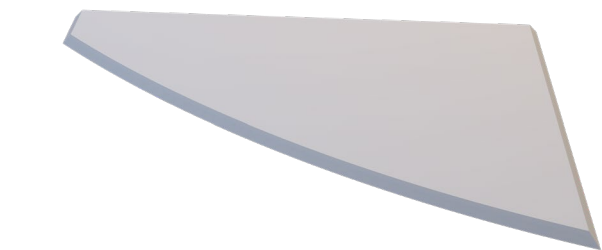
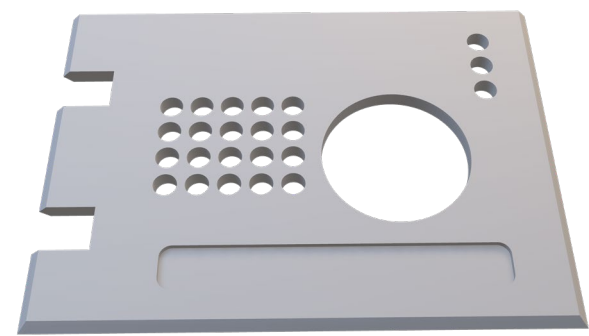
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	70%
<b>Outer walls:</b>	5
<b>Recommended Filament:</b>	PLA+ or stronger ABS or PETG recommended
<b>Module:</b>	MK3 Body



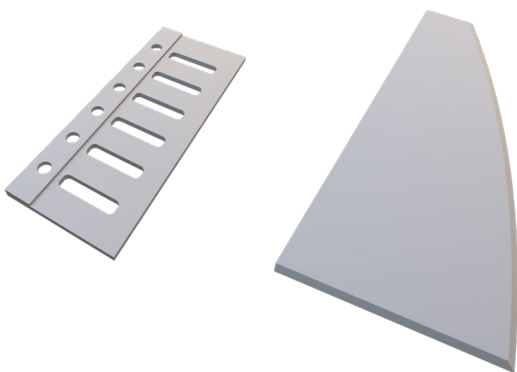
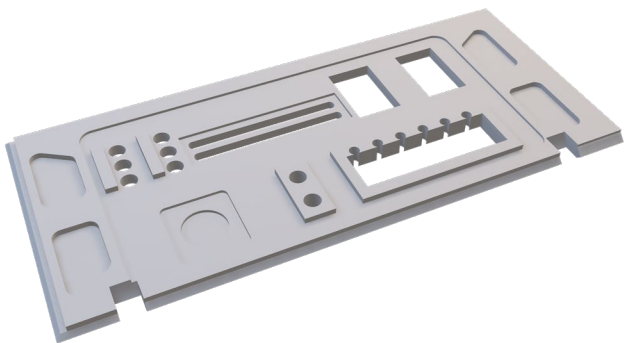
**File Name:** UtilityArmServoGear (2x needed)  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	70%
<b>Outer walls:</b>	5
<b>Recommended Filament:</b>	PLA+ or stronger ABS or PETG recommended
<b>Module:</b>	MK3 Body

Charge Panel:



Data Panel:



File Name: ChargePortBack  
Print flat

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	2 - 3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body

File Name: ChargePortTop  
Print flat

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	2 - 3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body

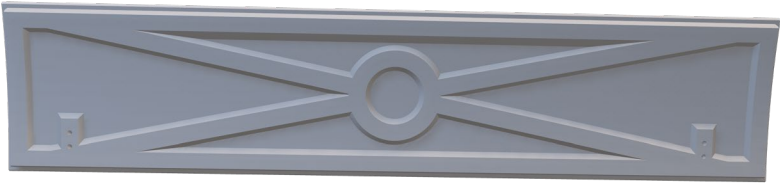
File Name: DataPanelBack  
Print flat

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	2 - 3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body

File Name: DatapanelFront & DatapanelTop  
Print flat

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	2 - 3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body

Doors:



File Name: BreadpanDoorA & B  
Print standing like this (with support)

Minimum Bed Size:	300
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	2 - 3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



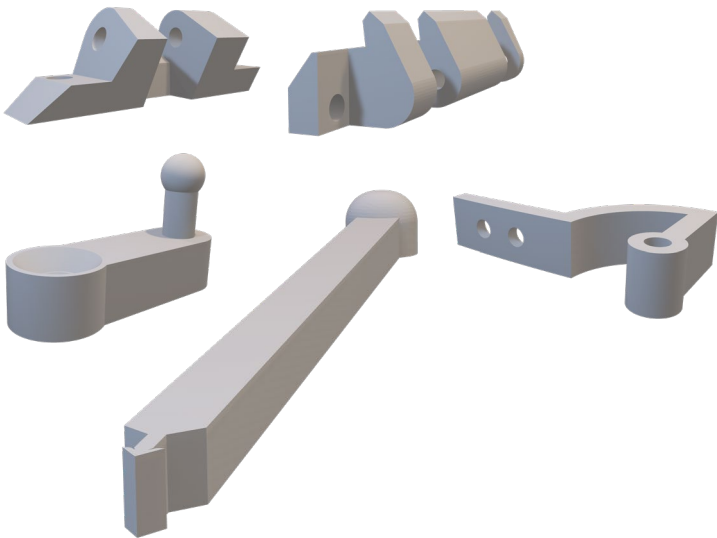
File Name: ChargePanel  
Print standing like this (with support)

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	2 - 3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



File Name: DataPortDoor  
Print standing like this (with support)

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	2 - 3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body

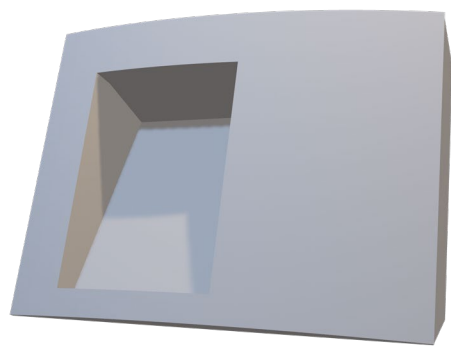


File Name: Hinges (all hinge files)  
Print flat

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	40%
Outer walls:	3 - 4
Recommended Filament:	PLA+ or stronger ABS or PETG recommended
Module:	MK3 Body

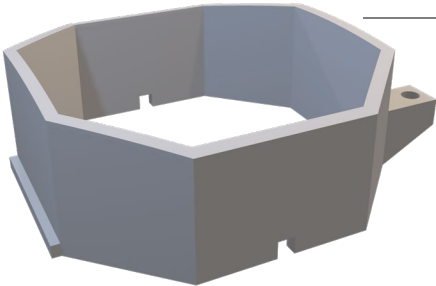
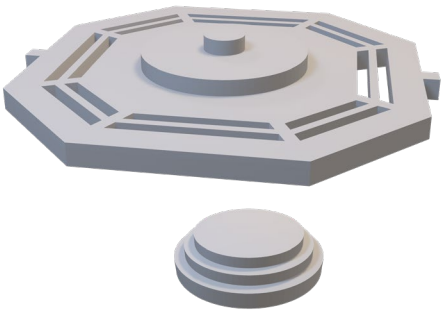


Greebles:



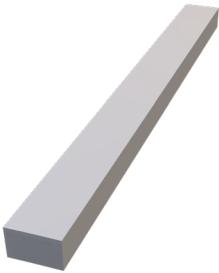
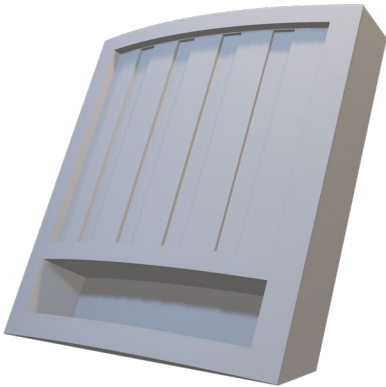
File Name: CoinRT (Front & 2x Rear)  
Print standing

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	3 - 4
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



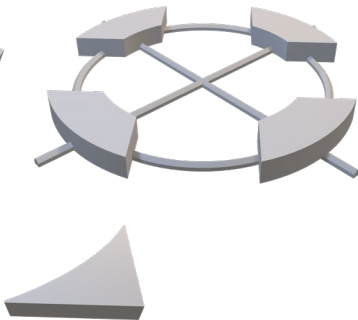
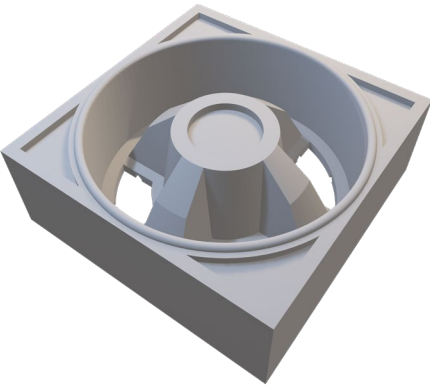
File Name: Hexport (needed 2x)  
(3 parts in total)  
Print flat

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	~15%
Outer walls:	3 - 4
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



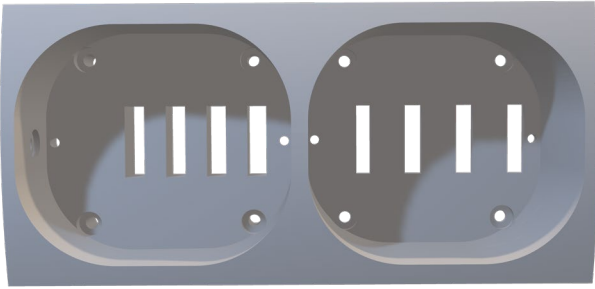
File Name: PocketVent  
(Fingers flat 4x)  
Print standing

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	~15%
Outer walls:	3 - 4
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



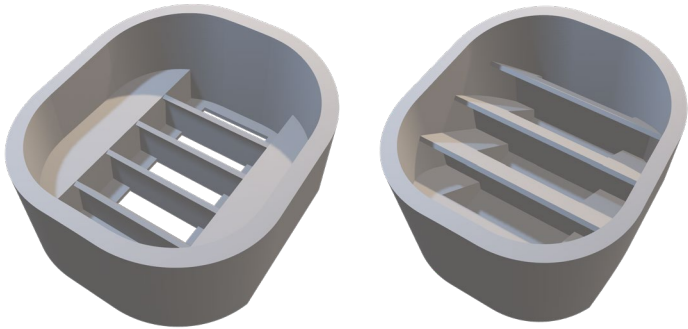
File Name: Power Coupler (2x)  
Small detail triangles 4x  
Print flat

Minimum Bed Size:	200
Layer height:	0.12 – 0.3
Infill:	~15%
Outer walls:	3 - 4
Recommended Filament:	PLA+ or stronger Resin for Details
Module:	MK3 Body



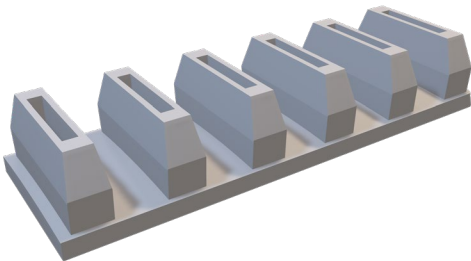
**File Name:** CenterVentPanel  
*Print standing on the long side*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	~15%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



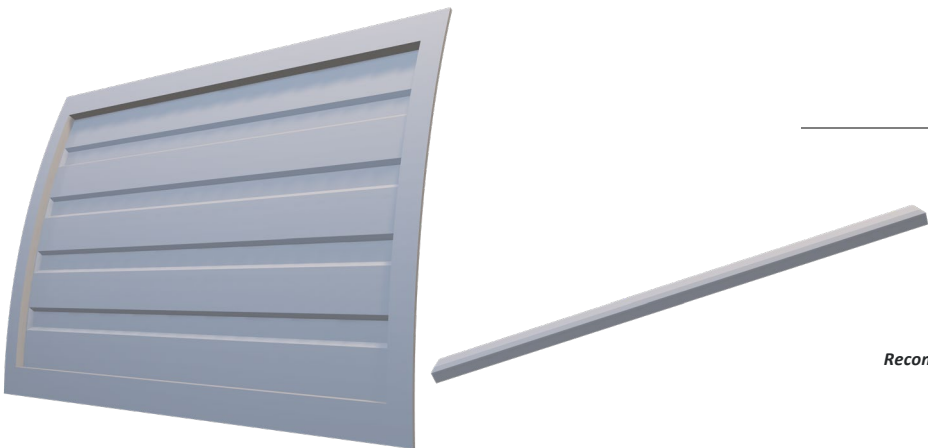
**File Name:** Lower- & UpperFrontVent  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	~15%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** Coinslots  
*Print flat*

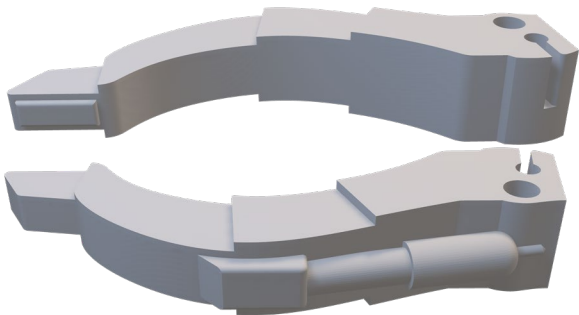
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12 – 0.2
<b>Infill:</b>	~15%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger Resin
<b>Module:</b>	MK3 Body



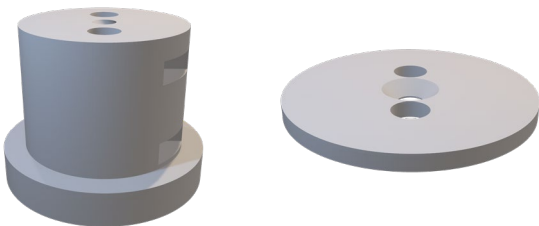
**File Name:** Sidevent (2x)  
*Print standing (with support)*  
*Fingers flat 4x*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12 – 0.25
<b>Infill:</b>	~15%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body

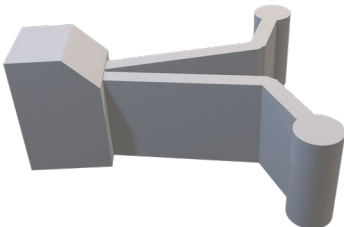
Gripper Arm (Eebel):



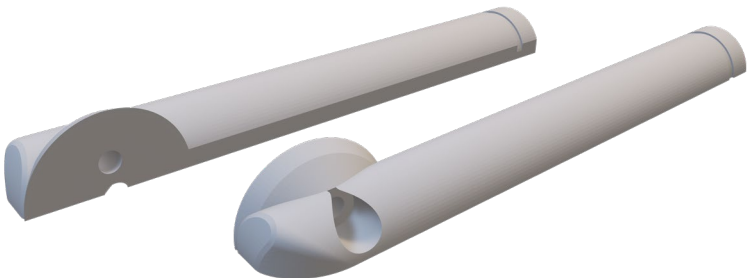
File Name: <i>Grip1 &amp; 2</i> <i>Print flat</i>	
Minimum Bed Size:	200
Layer height:	0.08 – 0.2
Infill:	~15%
Outer walls:	3 - 4
Recommended Filament:	PLA+ or stronger Resin
Module:	MK3 Body



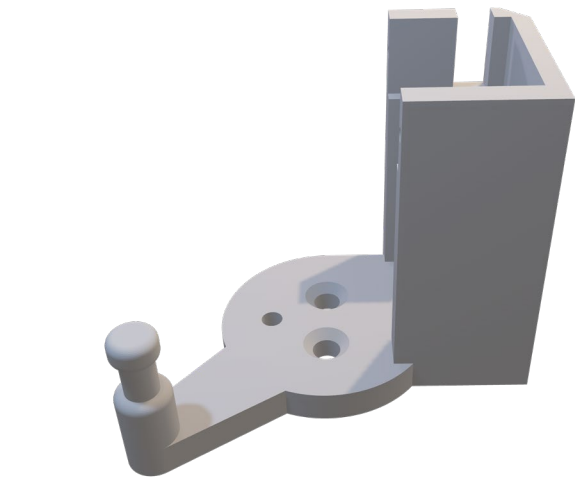
File Name: <i>GripperArmHub &amp; GripperSpacer</i> <i>Print flat</i>	
Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	~20%
Outer walls:	3 - 4
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



File Name: <i>Hinge</i> <i>Print flat</i>	
Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	~20%
Outer walls:	3 - 4
Recommended Filament:	TPU
Module:	MK3 Body

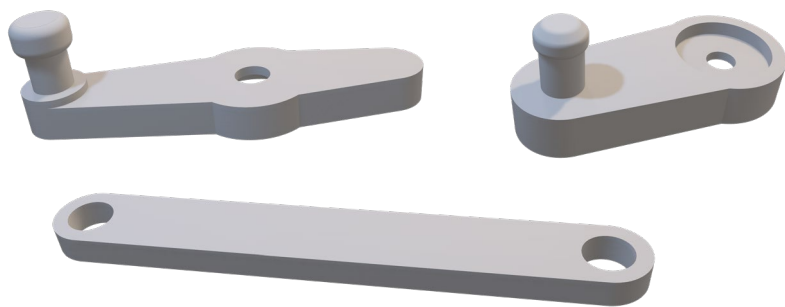


File Name: <i>MainArmA &amp; B</i> <i>Print flat</i>	
Minimum Bed Size:	200
Layer height:	0.08 – 0.2
Infill:	~20%
Outer walls:	3 - 4
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



**File Name:** ServoMountCap  
*Print flat/standing*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	~20%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** ServoArms & Bar  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	~20%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body

## Interface Arm (Eebel):



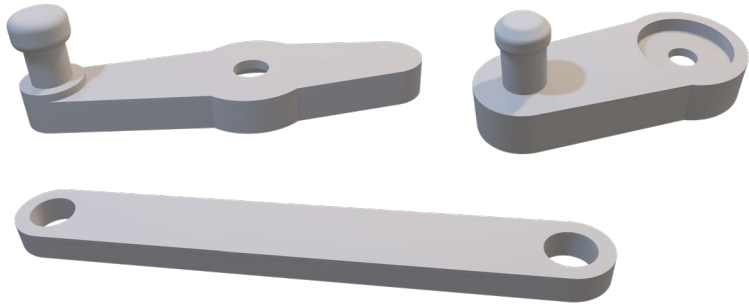
**File Name:** InterfaceHeadA & B  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.08 – 0.2
<b>Infill:</b>	~20%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger <b>Resin</b>
<b>Module:</b>	MK3 Body



**File Name:** IntMainarmC  
*Print standing*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.08 – 0.2
<b>Infill:</b>	~20%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger <b>Resin</b>
<b>Module:</b>	MK3 Body



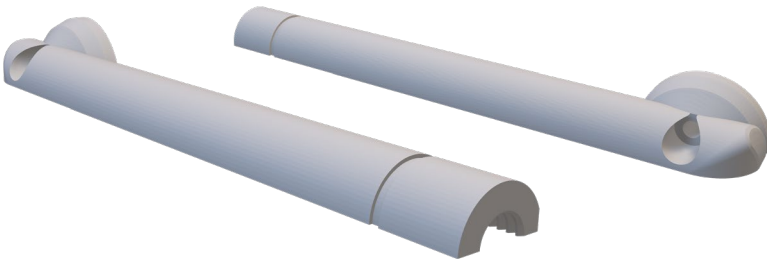
**File Name:** ServoArms & Bar  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	~20%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** IntRodA & B  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.08 – 0.2
<b>Infill:</b>	~20%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



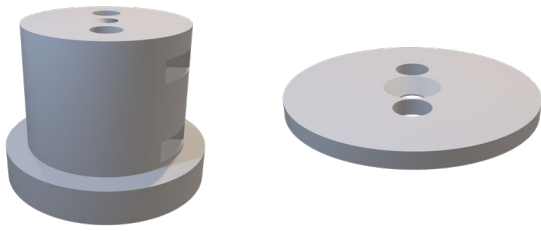
**File Name:** IntMainArma & B  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.08 – 0.2
<b>Infill:</b>	~20%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



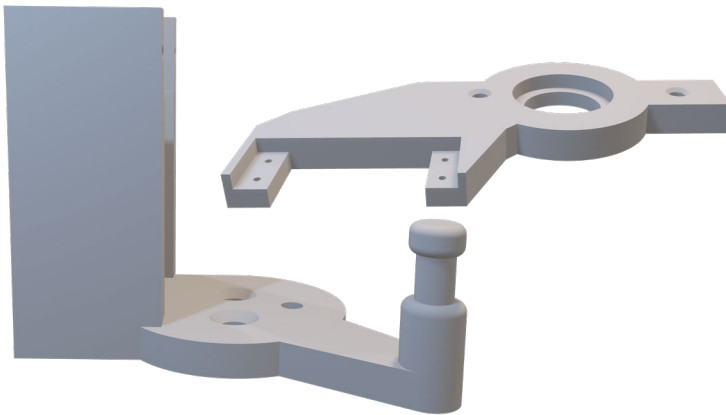
**File Name:** InterfaceInsert  
*Print standing*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	~20%
<b>Outer walls:</b>	3-4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** IArmHub & IArmSpacer  
*Print flat*

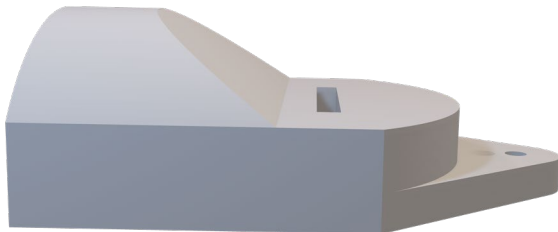
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	~20%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** IArmSevoMountCap & ServoBase  
*Print flat/standing*

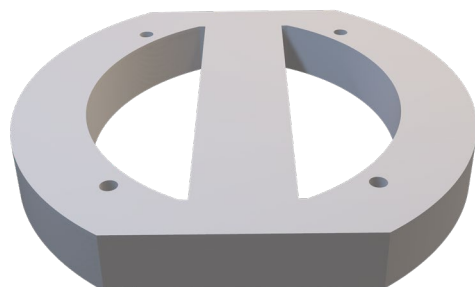
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	~20%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body

## Legs & Feet: Center Leg/Ankle:



**File Name:** AnkleA & B  
*Print flat*

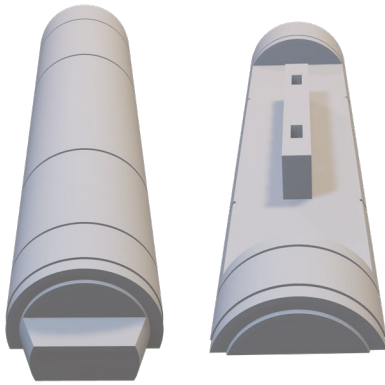
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	25 - 40%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body



**File Name:** AnkleA & B  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	25 - 40%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body

Only needed for 2-3-2 Version. But recommended to get a little more clearance or for later (2-3-2)



**File Name:** AnkCylinderA & B  
*Print flat*

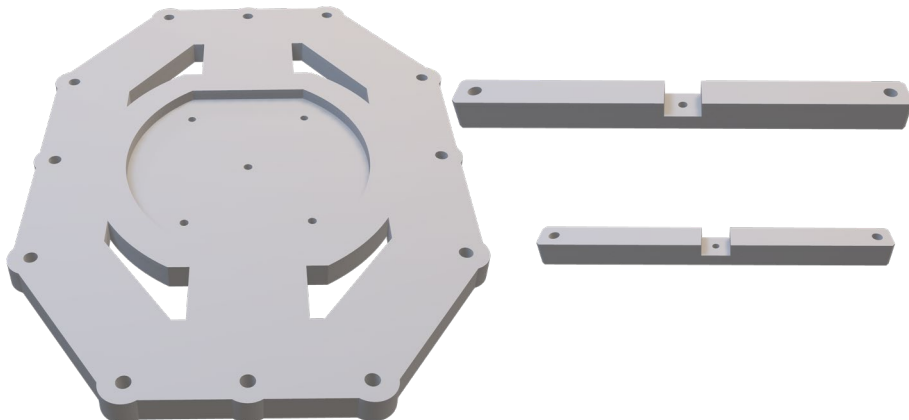
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12 – 0.25
<b>Infill:</b>	10-15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** AnkleA & B  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	25 - 40%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body

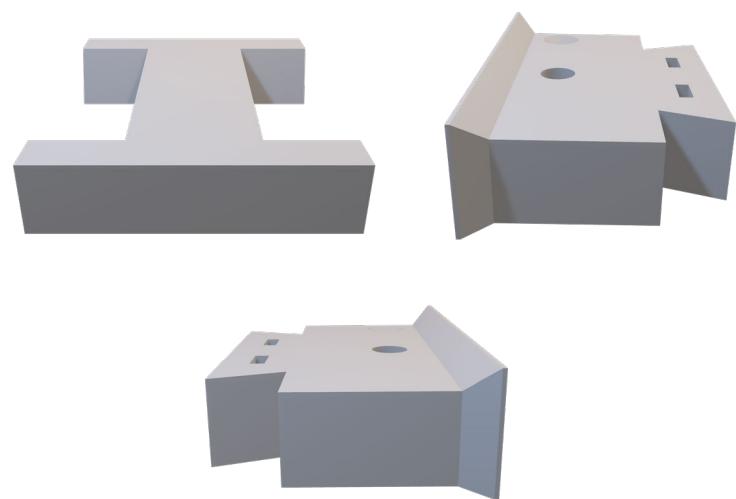
## Center Leg / Ankle Support Frame:



**File Name:** Ankle Support + 2 Bars  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body

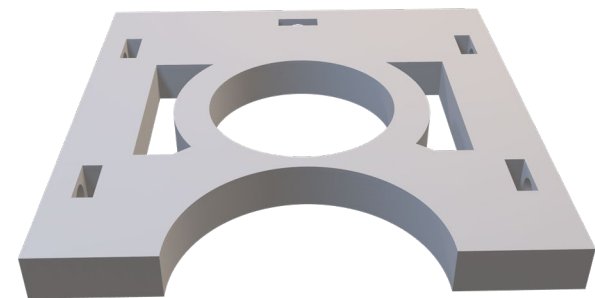
Center Leg / Foot Bracket:



File Name: Foot Bracket (3 parts total)  
Print flat

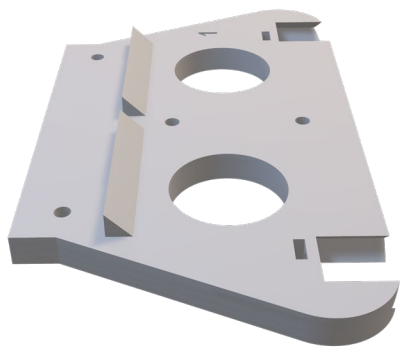
Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	30 - 40%
Outer walls:	4 - 5
Recommended Filament:	ABS or Petg
Module:	MK3 Body

Center Leg / Omniframe:



File Name: OmniframeCenter  
Print flat

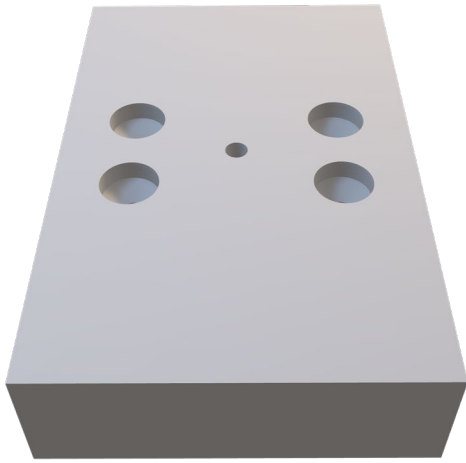
Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	30 - 40%
Outer walls:	4 - 5
Recommended Filament:	ABS or Petg
Module:	MK3 Body



File Name: OmniframeSide (2x)  
Print flat

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	30 - 40%
Outer walls:	4 - 5
Recommended Filament:	ABS or Petg
Module:	MK3 Body





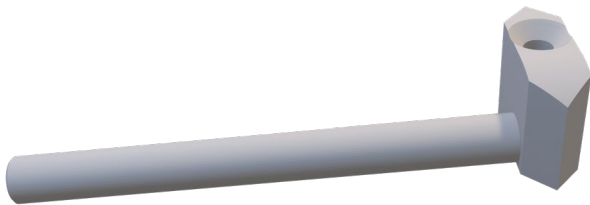
**File Name:** OmniframeTop  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	ABS or Petg
<b>Module:</b>	MK3 Body



**File Name:** OmniRod2 (2x)  
*Print standing*

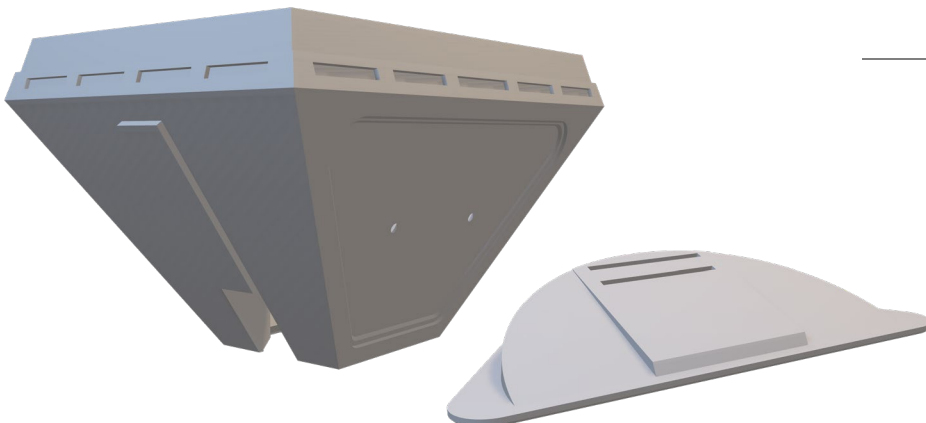
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	ABS or Petg
<b>Module:</b>	MK3 Body



**File Name:** OmniRod1 (2x)  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	30 - 40%
<b>Outer walls:</b>	4 - 5
<b>Recommended Filament:</b>	ABS or Petg
<b>Module:</b>	MK3 Body

## Center Leg / Skin:



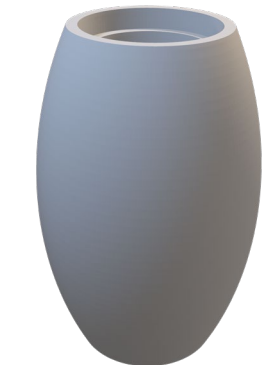
**File Name:** CentreFootSkin + 2 Sideskins  
*Print flat (support might be good)*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.25
<b>Infill:</b>	15 - 25%
<b>Outer walls:</b>	3 - 4
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body

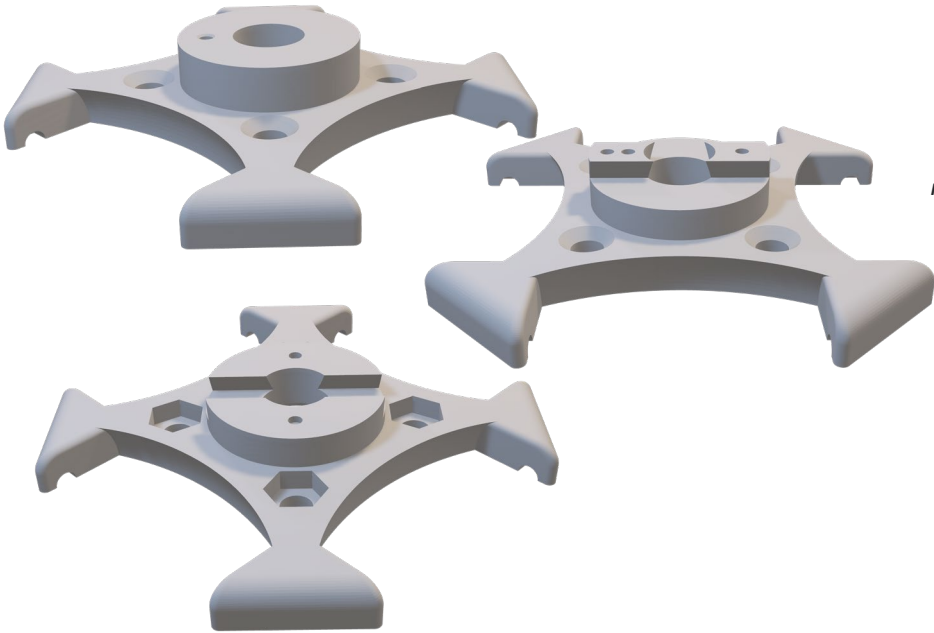
Center Leg / Omniwheels:



Please note to put the “flat” side down



Please note to put the “flat” side up



File Name: A2WheelCore (24x)  
Print standing

Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	30 - 40%
Outer walls:	4 - 5
Recommended Filament:	ABS or Petg
Module:	MK3 Body

File Name: A2WheelTyreFlex (24x)  
Print standing

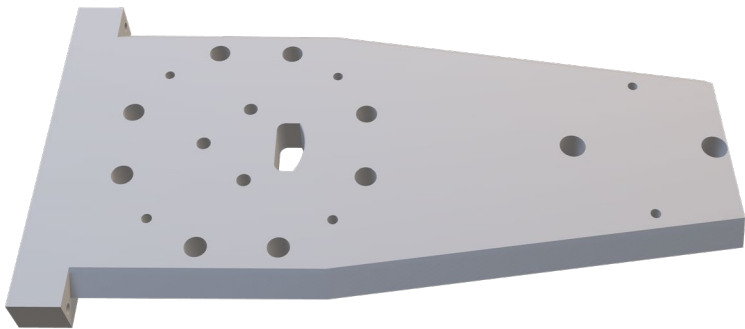
Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	50%
Outer walls:	5
Recommended Filament:	TPU
Module:	MK3 Body

File Name: CentreOmniFR1 - 6  
Print flat

Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	40%
Outer walls:	4-5
Recommended Filament:	ABS or Petg
Module:	MK3 Body

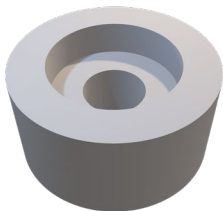
Shoulders:

The shoulders are some of the most stressed parts. They need to be solid!



File Name: AShoulderModuleFixed + B  
Print flat

Minimum Bed Size:	310
Layer height:	0.2 – 0.3
Infill:	40%-50%
Outer walls:	5 - 7
Recommended Filament:	PLA+ or stronger ABS or Petg recommended
Module:	MK3 Body



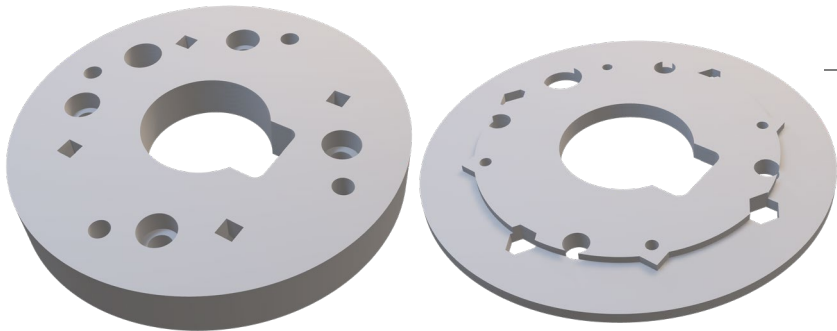
File Name: HubPinCap (4x for each shoulder)  
Print flat

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	40%-50%
Outer walls:	4 - 5
Recommended Filament:	PLA+ or stronger ABS or Petg recommended
Module:	MK3 Body



File Name: HubPin (4x for each shoulder)  
Print flat

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	40%-50%
Outer walls:	4 - 5
Recommended Filament:	PLA+ or stronger ABS or Petg recommended
Module:	MK3 Body

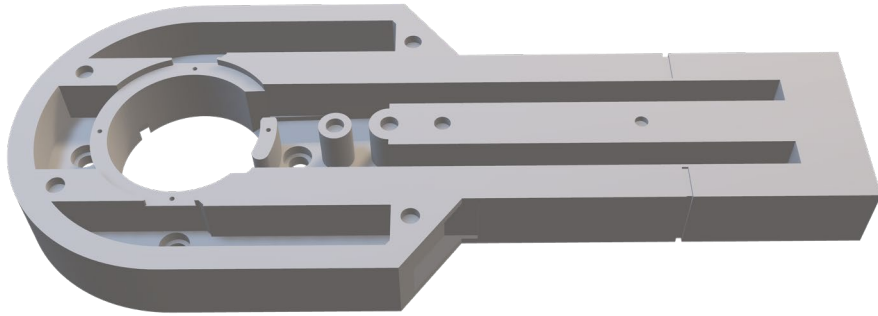


File Name: AHubfixed & AHubinsert + 2xBHub  
(4 in total)  
Print flat

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	40%-50%
Outer walls:	4 - 5
Recommended Filament:	PLA+ or stronger ABS or Petg recommended
Module:	MK3 Body

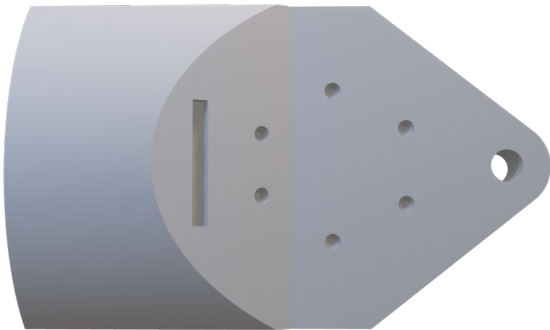
## Mainlegs:

The mainlegs are some of the most stressed parts. They need to be solid! (not the Greebles / details)



**File Name:** LegA & B  
*Print flat*

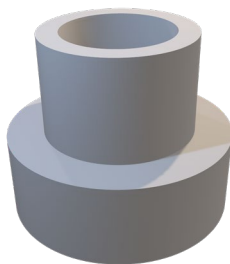
<b>Minimum Bed Size:</b>	<b>500</b>
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	<b>40%-50%</b>
<b>Outer walls:</b>	<b>4 - 7</b>
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body



**File Name:** AnkleA & B  
*Print standing like this*

<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	<b>40%-50%</b>
<b>Outer walls:</b>	<b>4 - 7</b>
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body

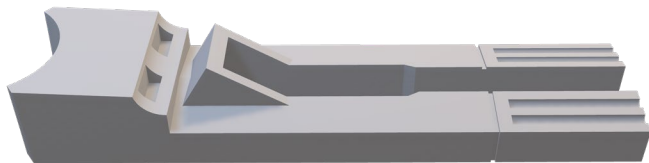
Please note, there are Standard and long versions.



**File Name:** 8mmLegPlug (4x for each leg)  
*Print standing like this*

<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	<b>40%-50%</b>
<b>Outer walls:</b>	<b>4 - 7</b>
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b> <b>MAYBE TPU as Shock absorber</b>
<b>Module:</b>	MK3 Body

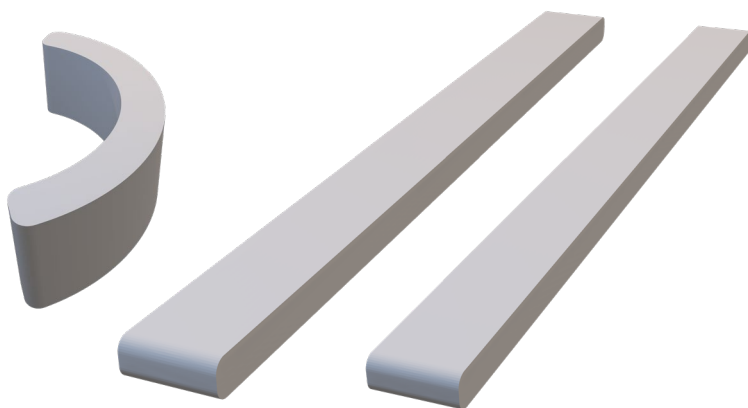
## Mainlegs / Greebles / Boosters:



Please use the long version if you printed long legs

**File Name:** ABooster & B  
*Print flat*

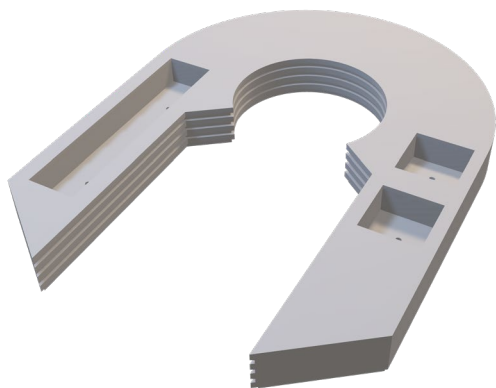
<b>Minimum Bed Size:</b>	<b>300</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>~15%</b>
<b>Outer walls:</b>	<b>3</b>
<b>Recommended Filament:</b>	<b>PLA+ or stronger</b>
<b>Module:</b>	<b>MK3 Body</b>



**File Name:** ABoosterPiece A – C (2x each)  
*Print flat*

<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>~15%</b>
<b>Outer walls:</b>	<b>3</b>
<b>Recommended Filament:</b>	<b>PLA+ or stronger</b>
<b>Module:</b>	<b>MK3 Body</b>

## Mainlegs / Greebles / Horseshoes :



**File Name:** AHorseshoe & B  
*Print flat*

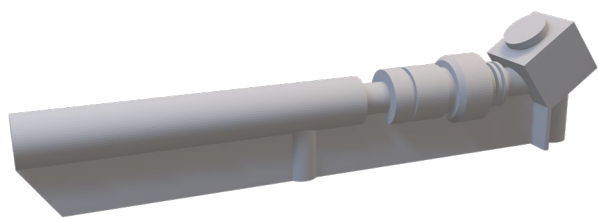
<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>~15%</b>
<b>Outer walls:</b>	<b>3</b>
<b>Recommended Filament:</b>	<b>PLA+ or stronger</b>
<b>Module:</b>	<b>MK3 Body</b>



**File Name:** Horseshoeinsert1-3 (2x each)  
*Print flat*

<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>~15%</b>
<b>Outer walls:</b>	<b>3</b>
<b>Recommended Filament:</b>	<b>PLA+ or stronger</b>
<b>Module:</b>	<b>MK3 Body</b>

Mainlegs / Greebles / Legstrut:

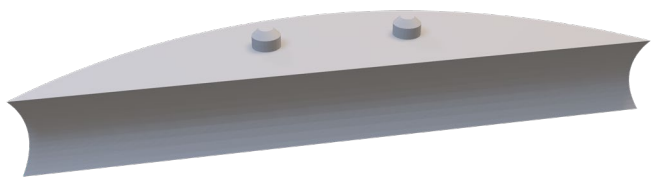


Print long version if you use long legs. There's also a split version.

File Name: Legstrut (2x)  
Print like this

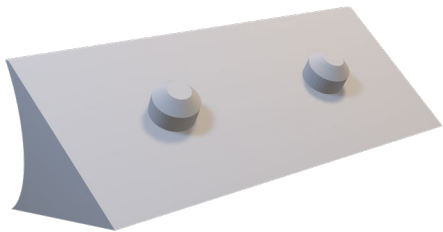
Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	~15%
Outer walls:	3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body

Mainlegs / Greebles:



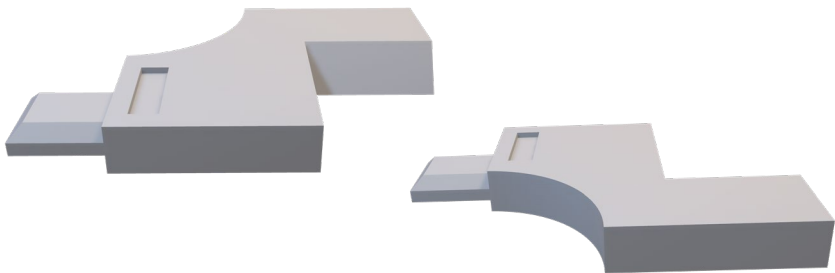
File Name: AnkleCylHolder (2x)  
Print like this

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



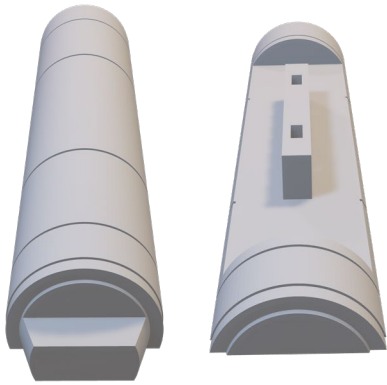
File Name: AnkleCylWedge (2x)  
Print like this

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



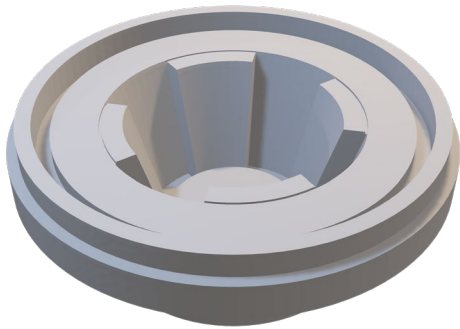
File Name: AnkleDet & B  
Print like this

Minimum Bed Size:	200
Layer height:	0.12 – 0.25
Infill:	~15%
Outer walls:	3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



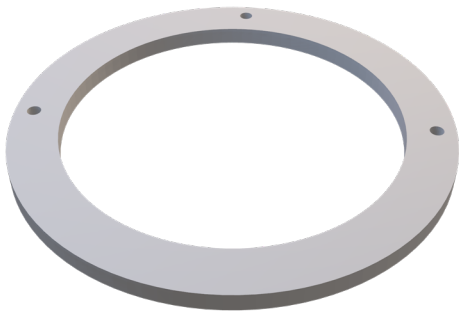
**File Name:** Cylinder1 & 2 (each 2x)  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12 – 0.25
<b>Infill:</b>	10-15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** LCentreHub & R  
*Print like this*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.12 – 0.25
<b>Infill:</b>	10-15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



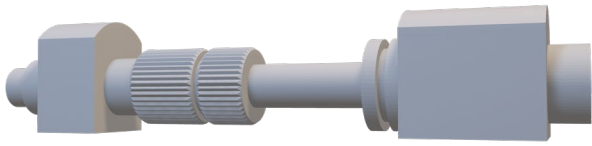
**File Name:** LRingHub & R  
*Print flat*

<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	10-15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



**File Name:** Anklebracelet (2x)  
*Print flat*

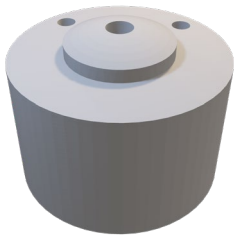
<b>Minimum Bed Size:</b>	200
<b>Layer height:</b>	0.2 – 0.3
<b>Infill:</b>	10-15%
<b>Outer walls:</b>	2-3
<b>Recommended Filament:</b>	PLA+ or stronger
<b>Module:</b>	MK3 Body



There’s also a splitted version

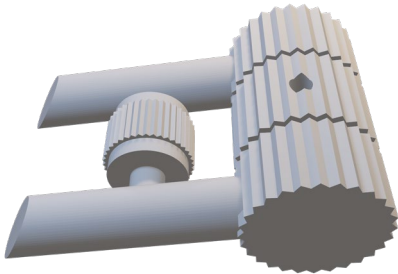
File Name: Shou\_HydrFull  
 Print flat (with support)

Minimum Bed Size:	200
Layer height:	0.12 – 0.2
Infill:	10-15%
Outer walls:	2-3
Recommended Filament:	PLA+ or stronger Resin
Module:	MK3 Body



File Name: ShoulderButton (4x)  
 Print flat

Minimum Bed Size:	200
Layer height:	0.12 – 0.2
Infill:	10-15%
Outer walls:	2-3
Recommended Filament:	PLA+ or stronger Resin
Module:	MK3 Body



There’s also a splitted version

File Name: ShoulderStabiliseFull  
 Print flat with support

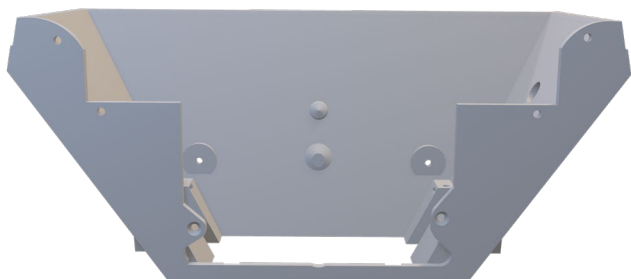
Minimum Bed Size:	200
Layer height:	0.12 – 0.2
Infill:	10-15%
Outer walls:	2-3
Recommended Filament:	PLA+ or stronger Resin
Module:	MK3 Body



## DriveUnits & OuterFeet:

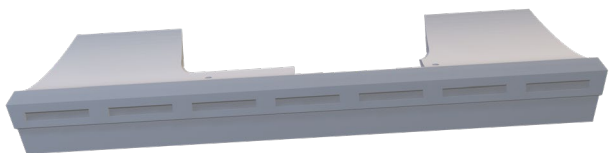
I only cover Drive A here. Same instructions for Drive B!

## Shells:



File Name: *AShell1*  
[Print like this](#)

Minimum Bed Size:	450
Layer height:	0.2 – 0.3
Infill:	10-15%
Outer walls:	2-3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



File Name: *AShell2*  
[Print like this](#)

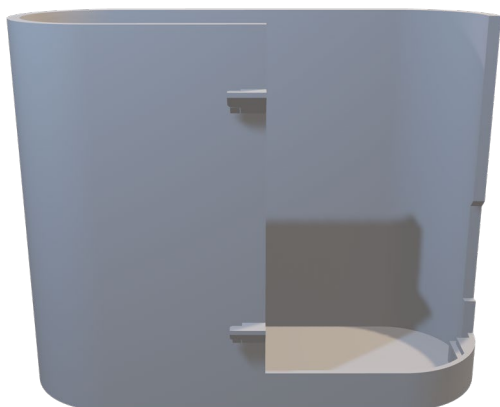
Minimum Bed Size:	350
Layer height:	0.2 – 0.3
Infill:	10-15%
Outer walls:	2-3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



File Name: *AShell3*  
[Print like this](#)

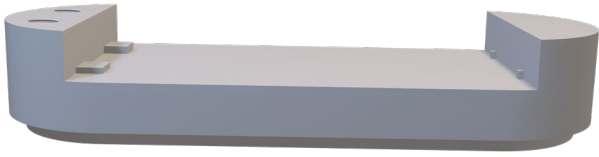
Minimum Bed Size:	300
Layer height:	0.2 – 0.3
Infill:	10-15%
Outer walls:	2-3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body

## Battery Box:



File Name: *ABatteryBoxMain*  
[Print like this](#)

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	10-15%
Outer walls:	2-3
Recommended Filament:	PLA+ or stronger
Module:	MK3 Body



**File Name:** ABatteryBoxMain  
*Print like this (with support)*

<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>10-15%</b>
<b>Outer walls:</b>	<b>2-3</b>
<b>Recommended Filament:</b>	<b>PLA+ or stronger</b>
<b>Module:</b>	<b>MK3 Body</b>

## Frame:

**The frame is a stressed part. It needs to be stable**

**File Name:** AMotorFrameB  
*Print like this (with support!!!)*

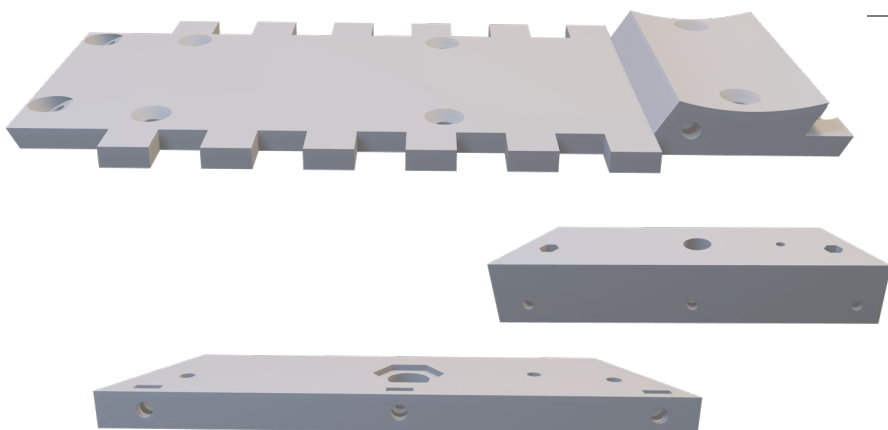
<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>30 - 40%</b>
<b>Outer walls:</b>	<b>4 - 6</b>
<b>Recommended Filament:</b>	<b>PLA+ or stronger</b> <b>ABS or Petg recommended</b>
<b>Module:</b>	<b>MK3 Body</b>

**File Name:** ASideFrameA  
*Print like this*

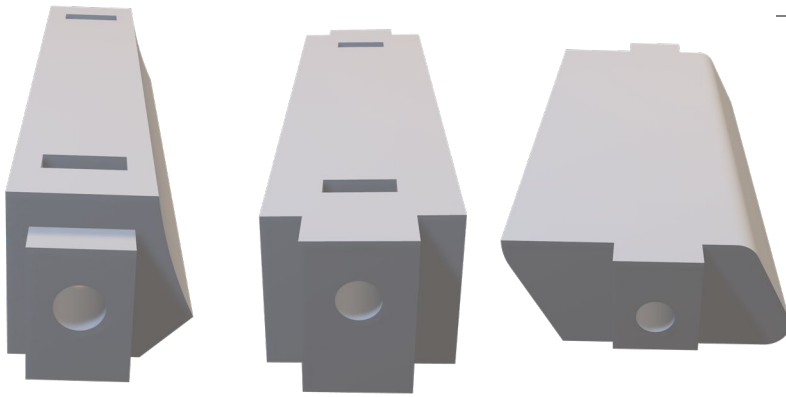
<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>30 - 40%</b>
<b>Outer walls:</b>	<b>4 - 6</b>
<b>Recommended Filament:</b>	<b>PLA+ or stronger</b> <b>ABS or Petg recommended</b>
<b>Module:</b>	<b>MK3 Body</b>

**File Name:** ATopFrameA – C (3 parts in total)  
*Print like this*

<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>30 - 40%</b>
<b>Outer walls:</b>	<b>4 - 6</b>
<b>Recommended Filament:</b>	<b>PLA+ or stronger</b> <b>ABS or Petg recommended</b>
<b>Module:</b>	<b>MK3 Body</b>

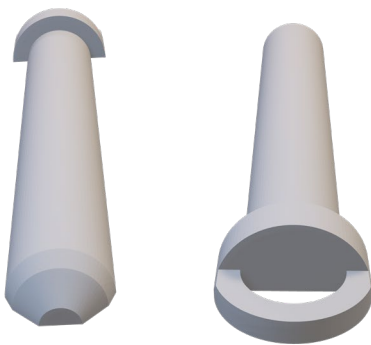


**File Name:** APost1 – 3 (3 parts in total)  
*Print like this*



<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>30 - 40%</b>
<b>Outer walls:</b>	<b>4 - 6</b>
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body

**File Name:** ALegPinA & B (2 parts in total)  
*Print like this*



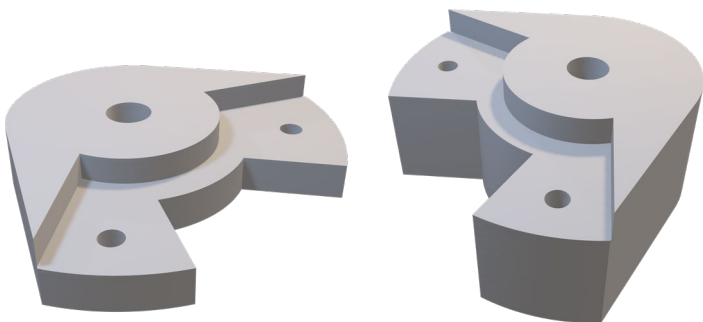
<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>30 - 40%</b>
<b>Outer walls:</b>	<b>4 - 6</b>
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body

**File Name:** AMotorBracketCover & Spacer  
*Print like this*



<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>30 - 40%</b>
<b>Outer walls:</b>	<b>4 - 6</b>
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body

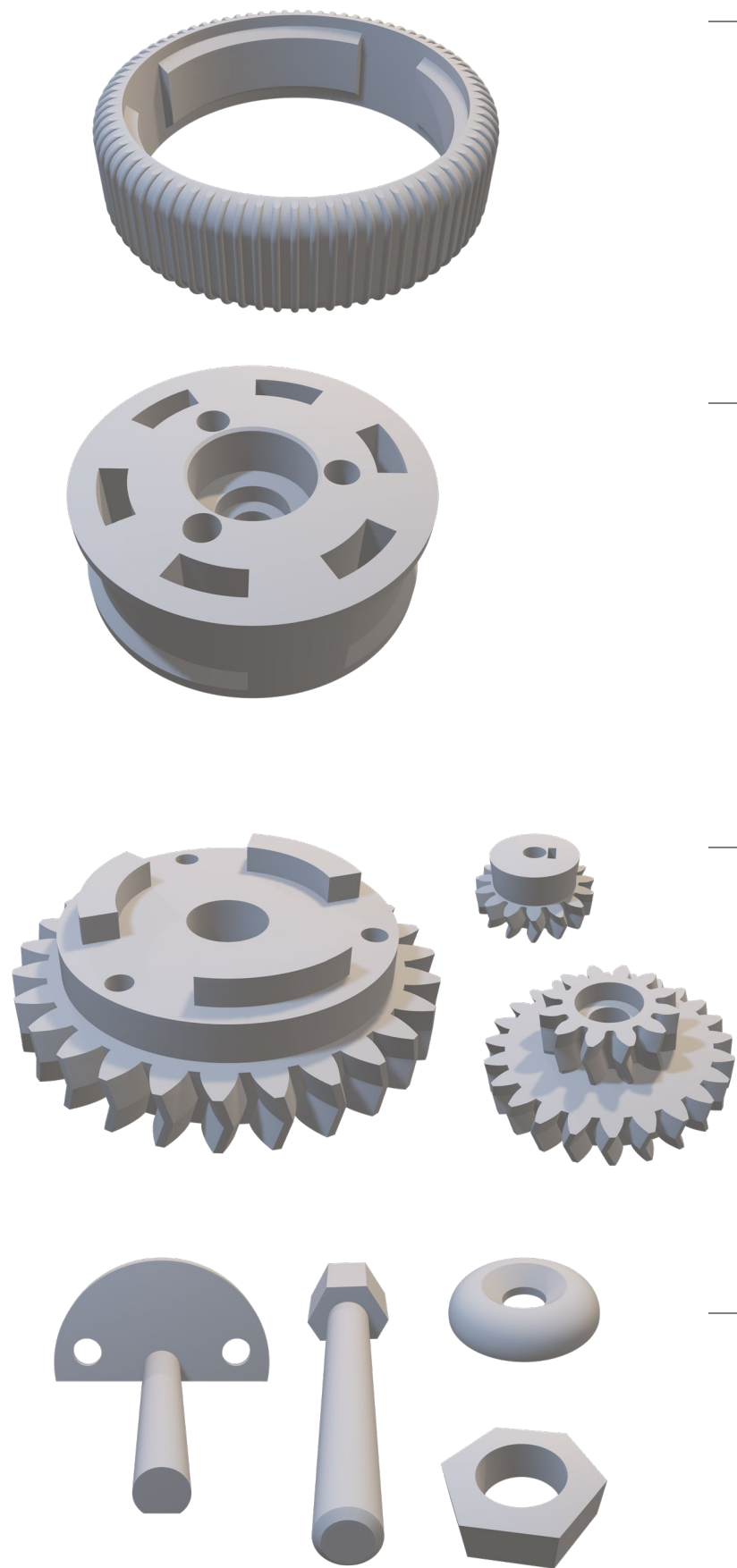
**File Name:** ARearOmniA & B (2 parts in total)  
*Print like this*



<b>Minimum Bed Size:</b>	<b>200</b>
<b>Layer height:</b>	<b>0.2 – 0.3</b>
<b>Infill:</b>	<b>30 - 40%</b>
<b>Outer walls:</b>	<b>4 - 6</b>
<b>Recommended Filament:</b>	PLA+ or stronger <b>ABS or Petg recommended</b>
<b>Module:</b>	MK3 Body

Drive Chain:

The drive chain is a stressed part. It needs to be stable



File Name: ATyre-Flex  
Print like this

Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	40 - 50%
Outer walls:	5 - 6
Recommended Filament:	TPU
Module:	MK3 Body

File Name: AWheel  
Print like this

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	30 - 40%
Outer walls:	5 - 7
Recommended Filament:	PLA+ or stronger ABS or Petg recommended
Module:	MK3 Body

File Name: Gears (3 in total)  
Print like this

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	30 - 40%
Outer walls:	5 - 7
Recommended Filament:	PLA+ or stronger ABS or Petg recommended
Module:	MK3 Body

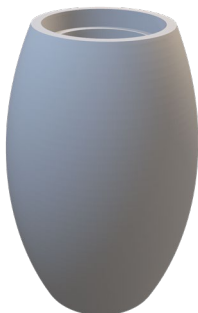
File Name: Pins, Nut, Wheelcap  
(4 in total)  
Print like this

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	30 - 40%
Outer walls:	5 - 7
Recommended Filament:	PLA+ or stronger ABS or Petg recommended
Module:	MK3 Body

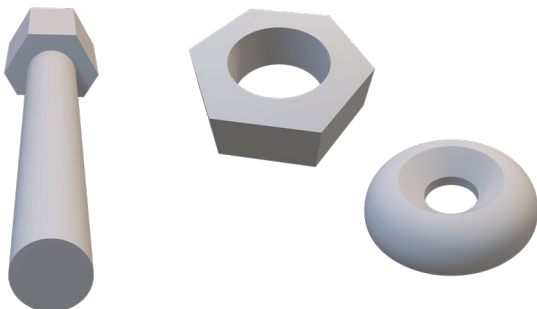
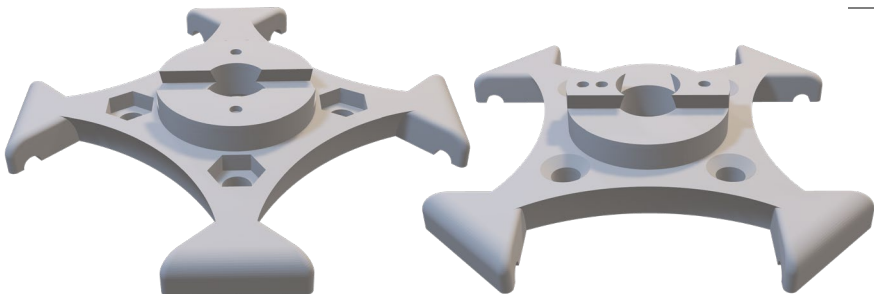
Omniwheels:



Please note to put the “flat” side down



Please note to put the “flat” side up



File Name: A2WheelCore (8x)  
Print standing

Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	30 - 40%
Outer walls:	4 - 5
Recommended Filament:	ABS or Petg
Module:	MK3 Body

File Name: A2WheelTyreFlex (8x)  
Print standing

Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	50%
Outer walls:	5
Recommended Filament:	TPU
Module:	MK3 Body

File Name: CentreOmniFrameA – D (4 in total)  
Print flat

Minimum Bed Size:	200
Layer height:	0.2 – 0.25
Infill:	40%
Outer walls:	4-5
Recommended Filament:	ABS or Petg
Module:	MK3 Body

File Name: Pin, Nut, Cap  
(3 in total)  
Print like this

Minimum Bed Size:	200
Layer height:	0.2 – 0.3
Infill:	30 - 40%
Outer walls:	5 - 7
Recommended Filament:	PLA+ or stronger ABS or Petg recommended
Module:	MK3 Body