CAMERADACTYL Brancopan Slicer Print Profiles

The Brancopan has many pieces, and will require multiple print profiles to properly slice all of your STL files into Gcode that your printer can read. I made a more in depth youtube video in realtime, slicing all of the necessary files to build the camera. If you are new to 3D printing, I suggest you watch that, but if you already understand slicing for 3D printing, this document should be much faster to use and get started with your prints.

These slicing profiles are suggestions. Feel free to tweak or modify or change them to your preference, and based on your machine and experience.

For the purpose of this document I am using Repetier Host, with the built in Slic3r slicer. The following images are screen shots of all relevant print profile settings for 5 print profiles that you will need to slice all of the Brancopan STL files. Each Brancopan file is labeled with a suffix (NB, NG, S1, S2, or S3) which correspond to the print slicer profile that I suggest that you print the files with.

There is a secondary suffix 'color' denoting pieces that can be any color without additional thought about opacity and light leakage/seepage/glowing. These pieces can be printed in colors other than black. You might find that you can print other files than these in colors other than black, but you are going to want to check that they are either light tight/ fully opaque, or apply a light sealing finish, such as black paint. If you are going to paint your parts anyway, I might suggest printing in black, and painting the outside your color of choice, rather than printing in color and painting the inside black. Experiment if this is your thing, and let me know how it goes!

The profiles are as follows:

NB : Non Supported Body Pieces

Plater | **Print Settings** ⊗

NB

Layers and perimeters
Infill
Skirt and brim
Support material
Speed
Multiple extruders
Advanced
Output options
Notes
Shortcuts

Layer height

Layer height:	0.3	mm
First layer height:	0.35	mm or %
Use adaptive slicing:	☐	
Adaptive quality:	——————●——— 75	%
Match horizontal surfaces:	☐	

Vertical shells

Perimeters:	4	(minimum)
Spiral vase:	☐	

Horizontal shells

Solid layers:	Top: 4	Bottom: 4

Quality (slower slicing)

Extra perimeters if needed:	☑
Avoid crossing perimeters:	☑
Detect thin walls:	☑
Detect bridging perimeters:	☑

Advanced

Seam position:	Aligned
External perimeters first:	☐

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

Plater | **Print Settings** ⊗

NB

Layers and perimeters
Infill
Skirt and brim
Support material
Speed
Multiple extruders
Advanced
Output options
Notes
Shortcuts

Infill

Fill density:	30	%
Fill pattern:	Rectilinear	
External infill pattern:	Top: Concentric	Bottom: Concentric

Reducing printing time

Combine infill every:	1	layers
Only infill where needed:	☐	

Advanced

Fill gaps:	☑	
Solid infill every:	0	layers
Fill angle:	45	°
Solid infill threshold area:	70	mm²
Only retract when crossing perimeters:	☑	
Infill before perimeters:	☐	

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

NB

Skirt

- Layers and perimeters
- Infill
- **Skirt and brim**
- Support material
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Loops (minimum):   1

Distance from object: 6   mm

Skirt height: 1   layers

Minimum extrusion length: 0   mm

Brim

Exterior brim width: 0   mm

Interior brim width: 0   mm

Brim connections width: 0   mm

Plater | Print Settings ⊗

NB

- Layers and perimeters
- Infill
- Skirt and brim
- **Support material**
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Support material

Generate support material:	☐	
Overhang threshold:	60%	° (or %)
Max layer count for supports:	0	layers
Enforce support for the first:	0	layers

Raft

Raft layers:	0	layers

Options for support material and raft

Contact Z distance:	0.2 (detachable) ▾	mm
Pattern:	pillars ▾	
Pattern spacing:	2.5	mm
Pattern angle:	0	°
Interface layers:	3	layers
Interface pattern spacing:	0	mm
Support on build plate only:	☐	
Don't support bridges:	☑	

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

Plater | Print Settings ⊗

NB

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- **Speed**
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Speed for print moves

Perimeters:	60 ▾	mm/s
↳ small:	15 ▾	mm/s or %
↳ external:	50% ▾	mm/s or %
Infill:	80 ▾	mm/s
↳ solid:	20 ▾	mm/s or %
↳ top solid:	15 ▾	mm/s or %
↳ gaps:	20 ▾	mm/s or %
Bridges:	60 ▾	mm/s
Support material:	60 ▾	mm/s
↳ interface:	100% ▾	mm/s or %

Speed for non-print moves

Travel:	130	mm/s

Modifiers

First layer speed:	30	mm/s or %

Acceleration control (advanced)

Perimeters:	0	mm/s²
Infill:	0	mm/s²
Bridge:	0	mm/s²
First layer:	0	mm/s²
Default:	0	mm/s²

Autospeed (advanced)

Max print speed:	80	mm/s
Max volumetric speed:	0	mm³/s

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

NG: Non Supported Gears

Plater   **Print Settings** ⊗

NG

- 🖼 Layers and perimeters
- ❌ Infill
- 📁 Skirt and brim
- 🗂 Support material
- ⏱ Speed
- 🏺 Multiple extruders
- 🔧 Advanced
- ➡ Output options
- 📒 Notes
- 🔑 Shortcuts

Layer height

Layer height:	0.3	mm
First layer height:	0.35	mm or %
Use adaptive slicing:	☐	
Adaptive quality:	──────●────── 75	%
Match horizontal surfaces:	☐	

Vertical shells

Perimeters:	6 ⌄	(minimum)
Spiral vase:	☐	

Horizontal shells

Solid layers: Top: 6 ⌄   Bottom: 6 ⌄

Quality (slower slicing)

Extra perimeters if needed:	☑
Avoid crossing perimeters:	☐
Detect thin walls:	☑
Detect bridging perimeters:	☑

Advanced

Seam position:	Aligned ⌄
External perimeters first:	☐

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

Plater   **Print Settings** ⊗

NG

- 🖼 Layers and perimeters
- ❌ Infill
- 📁 Skirt and brim
- 🗂 Support material
- ⏱ Speed
- 🏺 Multiple extruders
- 🔧 Advanced
- ➡ Output options
- 📒 Notes
- 🔑 Shortcuts

Infill

Fill density:	90 ⌄ %
Fill pattern:	Rectilinear ⌄
External infill pattern:	Top: Concentric ⌄ Bottom: Concentric ⌄

Reducing printing time

Combine infill every:	1 ⌄	layers
Only infill where needed:	☐	

Advanced

Fill gaps:	☑	
Solid infill every:	0 ⌄	layers
Fill angle:	45	°
Solid infill threshold area:	70	mm²
Only retract when crossing perimeters:	☑	
Infill before perimeters:	☐	

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

NG

Skirt

Loops (minimum):	1	
Distance from object:	6	mm
Skirt height:	1	layers
Minimum extrusion length:	0	mm

Brim

Exterior brim width:	0	mm
Interior brim width:	0	mm
Brim connections width:	0	mm

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Window 1: Print Settings — Support material

Plater | **Print Settings** ⊗

NG

- Layers and perimeters
- Infill
- Skirt and brim
- **Support material**
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Support material

Generate support material:	☐	
Overhang threshold:	10%	° (or %)
Max layer count for supports:	0	layers
Enforce support for the first:	0	layers

Raft

Raft layers:	0	layers

Options for support material and raft

Contact Z distance:	0.2 (detachable)	mm
Pattern:	honeycomb	
Pattern spacing:	4	mm
Pattern angle:	0	°
Interface layers:	3	layers
Interface pattern spacing:	0	mm
Support on build plate only:	☐	
Don't support bridges:	☐	

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

Window 2: Print Settings — Speed

Plater | **Print Settings** ⊗

NG

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- **Speed**
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Speed for print moves

Perimeters:	60	mm/s
↳ small:	15	mm/s or %
↳ external:	50%	mm/s or %
Infill:	80	mm/s
↳ solid:	20	mm/s or %
↳ top solid:	15	mm/s or %
↳ gaps:	20	mm/s or %
Bridges:	60	mm/s
Support material:	60	mm/s
↳ interface:	100%	mm/s or %

Speed for non-print moves

Travel:	130	mm/s

Modifiers

First layer speed:	30	mm/s or %

Acceleration control (advanced)

Perimeters:	0	mm/s²
Infill:	0	mm/s²
Bridge:	0	mm/s²
First layer:	0	mm/s²
Default:	0	mm/s²

Autospeed (advanced)

Max print speed:	80	mm/s
Max volumetric speed:	0	mm³/s

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

S1: Supported Pieces Profile 1

Plater **Print Settings** ⊗

S1

- 🖼 **Layers and perimeters**
- ❌ Infill
- 📁 Skirt and brim
- 🗄 Support material
- ⏱ Speed
- 🔻 Multiple extruders
- 🔧 Advanced
- ➡ Output options
- 🗒 Notes
- 🔧 Shortcuts

Layer height

Layer height:	0.3	mm
First layer height:	0.35	mm or %
Use adaptive slicing:	☑	
Adaptive quality:	————●——— 75	%
Match horizontal surfaces:	☐	

Vertical shells

Perimeters:	5	(minimum)
Spiral vase:	☐	

Horizontal shells

Solid layers:	Top: 6		Bottom: 6	

Quality (slower slicing)

Extra perimeters if needed:	☑
Avoid crossing perimeters:	☐
Detect thin walls:	☑
Detect bridging perimeters:	☑

Advanced

Seam position:	Aligned ▾
External perimeters first:	☐

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

Plater **Print Settings** ⊗

S1

- 🖼 Layers and perimeters
- ❌ **Infill**
- 📁 Skirt and brim
- 🗄 Support material
- ⏱ Speed
- 🔻 Multiple extruders
- 🔧 Advanced
- ➡ Output options
- 🗒 Notes
- 🔧 Shortcuts

Infill

Fill density:	30 ▾	%		
Fill pattern:	Rectilinear ▾			
External infill pattern:	Top: Concentric ▾		Bottom: Concentric ▾	

Reducing printing time

Combine infill every:	1	layers
Only infill where needed:	☐	

Advanced

Fill gaps:	☑	
Solid infill every:	0	layers
Fill angle:	45	°
Solid infill threshold area:	70	mm²
Only retract when crossing perimeters:	☑	
Infill before perimeters:	☐	

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

S1

- Layers and perimeters
- Infill
- **Skirt and brim**
- Support material
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Skirt

Loops (minimum):	1	
Distance from object:	6	mm
Skirt height:	1	layers
Minimum extrusion length:	0	mm

Brim

Exterior brim width:	0	mm
Interior brim width:	0	mm
Brim connections width:	0	mm

Print Settings — Support material

Plater	Print Settings	

S1

- Layers and perimeters
- Infill
- Skirt and brim
- **Support material**
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Support material

Generate support material:	☑	
Overhang threshold:	15	° (or %)
Max layer count for supports:	0	layers
Enforce support for the first:	0	layers

Raft

Raft layers:	0	layers

Options for support material and raft

Contact Z distance:	0.2 (detachable)	mm
Pattern:	honeycomb	
Pattern spacing:	5	mm
Pattern angle:	0	°
Interface layers:	3	layers
Interface pattern spacing:	0	mm
Support on build plate only:	☐	
Don't support bridges:	☑	

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

Print Settings — Speed

Plater	Print Settings	

S1

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- **Speed**
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Speed for print moves

Perimeters:	60	mm/s
↳ small:	15	mm/s or %
↳ external:	50%	mm/s or %
Infill:	80	mm/s
↳ solid:	20	mm/s or %
↳ top solid:	15	mm/s or %
↳ gaps:	20	mm/s or %
Bridges:	60	mm/s
Support material:	60	mm/s
↳ interface:	100%	mm/s or %

Speed for non-print moves

Travel:	130	mm/s

Modifiers

First layer speed:	30	mm/s or %

Acceleration control (advanced)

Perimeters:	0	mm/s²
Infill:	0	mm/s²
Bridge:	0	mm/s²
First layer:	0	mm/s²
Default:	0	mm/s²

Autospeed (advanced)

Max print speed:	80	mm/s
Max volumetric speed:	0	mm³/s

Version 1.3.0 - Remember to check for updates at http://slic3r.org/

S2: Supported Pieces Profile 2

Print Settings — Layers and perimeters

Tab bar: Plater | **Print Settings** ⊗

Profile: S2

Sidebar:
- **Layers and perimeters**
- Infill
- Skirt and brim
- Support material
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Layer height

Setting	Value	Unit
Layer height:	0.3	mm
First layer height:	0.35	mm or %
Use adaptive slicing:	☐	
Adaptive quality:	[slider] 75	%
Match horizontal surfaces:	☐	

Vertical shells

Setting	Value	
Perimeters:	6	(minimum)
Spiral vase:	☐	

Horizontal shells

Solid layers: Top: 6   Bottom: 6

Quality (slower slicing)

- Extra perimeters if needed: ☑
- Avoid crossing perimeters: ☐
- Detect thin walls: ☑
- Detect bridging perimeters: ☑

Advanced

Seam position: Aligned

External perimeters first: ☐

Print Settings — Infill

Tab bar: Plater | **Print Settings** ⊗

Profile: S2

Sidebar:
- Layers and perimeters
- **Infill**
- Skirt and brim
- Support material
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Infill

Setting	Value	
Fill density:	90	%
Fill pattern:	Rectilinear	
External infill pattern:	Top: Concentric	Bottom: Concentric

Reducing printing time

Setting	Value	
Combine infill every:	1	layers
Only infill where needed:	☐	

Advanced

Setting	Value	Unit
Fill gaps:	☑	
Solid infill every:	0	layers
Fill angle:	45	°
Solid infill threshold area:	70	mm²
Only retract when crossing perimeters:	☑	
Infill before perimeters:	☐	

S2

- Layers and perimeters
- Infill
- **Skirt and brim**
- Support material
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Skirt

Loops (minimum):	1	
Distance from object:	6	mm
Skirt height:	1	layers
Minimum extrusion length:	0	mm

Brim

Exterior brim width:	0	mm
Interior brim width:	0	mm
Brim connections width:	0	mm

Plater | **Print Settings** ⊗

S2

- Layers and perimeters
- Infill
- Skirt and brim
- **Support material**
- Speed
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Support material

Generate support material:	☑
Overhang threshold:	10% ° (or %)
Max layer count for supports:	0 layers
Enforce support for the first:	0 layers

Raft

Raft layers:	0 layers

Options for support material and raft

Contact Z distance:	0.2 (detachable) mm
Pattern:	honeycomb
Pattern spacing:	4 mm
Pattern angle:	0 °
Interface layers:	3 layers
Interface pattern spacing:	0 mm
Support on build plate only:	☑
Don't support bridges:	☐

Plater | **Print Settings** ⊗

S2

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- **Speed**
- Multiple extruders
- Advanced
- Output options
- Notes
- Shortcuts

Speed for print moves

Perimeters:	60	mm/s
↳ small:	15	mm/s or %
↳ external:	50%	mm/s or %
Infill:	80	mm/s
↳ solid:	20	mm/s or %
↳ top solid:	15	mm/s or %
↳ gaps:	20	mm/s or %
Bridges:	60	mm/s
Support material:	60	mm/s
↳ interface:	100%	mm/s or %

Speed for non-print moves

Travel:	130	mm/s

Modifiers

First layer speed:	30	mm/s or %

Acceleration control (advanced)

Perimeters:	0	mm/s²
Infill:	0	mm/s²
Bridge:	0	mm/s²
First layer:	0	mm/s²
Default:	0	mm/s²

Autospeed (advanced)

Max print speed:	80	mm/s
Max volumetric speed:	0	mm³/s

S3: Supported Pieces Profile 3