These are basic settings that were tested in **Cura 4.8.0** slicer.

The test models were printed on **Ultimaker 2**, **Creality Ender 3**, **Creality CR-10S Pro V2**, **Anycubic I3 Mega**, **Anycubic I3 MegaS** 3D printers with **PLA** and **PETG filaments**.

__Disclaimer__: The following printing settings are a __recommendation, not an obligation__. The parameters can vary depending on the peculiarities of your 3D printer, the material you use and especially the particular assembly part at hand. Each part that any model comprises often needs preliminary review and you are free to tweak the settings the way you find suitable.

<u>Note</u>:
- You can ***scale up*** the model (*<u>downscaling is not recommended!</u>*);
- All connectors should be printed at ***100% Infill***;
- For all parts of locks ("**ge_lock**" in "Source files") you need to change "Brim" type to "***Skirt***" in <u>Build Plate Adhesion</u> section.

Quality
Layer Height: 0.12 mm (*<u>you can also set Layer Height at 0.16 or 0.2mm for 0.4mm nozzles</u>*)
Initial Layer Height: 0.2 mm (*<u>carefully level the print bed and keep your Initial Layer Height the same as the main Layer Height</u>*)
Line Width: 0.4 mm
      Wall Line Width: 0.4 mm
            Outer Wall Line Width: 0.4 mm
            Inner Wall(s) Line Width: 0.4 mm
      Top/Bottom Line Width: 0.4 mm
      Infill Line Width: 0.4 mm
      Skirt/Brim Line Width: 0.4 mm
      Support Line Width: 0.4 mm
Initial Layer Line Width: 100%

Shell
Wall Thickness: 0.8 mm
   Wall Line Count: 2
Outer Wall Wipe Distance: 0.3 mm
Top Surface Skin Layers: 0
Top/Bottom Thickness: 0.6 mm
      Top Thickness: 0.6 mm
         Top Layers: 5
      Bottom Thickness: 0.6 mm
         Bottom Layers: 5
         Initial Bottom Layers: 5
Top/Bottom Pattern: Lines
Bottom Pattern Initial Layer: Lines
Top/Bottom Line Directions: [ ]

Outer Wall Inset: 0 mm
Optimize Wall Printing Order: Check
Compensate Wall Overlaps: Check
       Compensate Inner Wall Overlaps: Check
Minimum Wall Flow: 0%
Fill Gaps Between Walls: Everywhere
Filter Out Tiny Gaps: Check
Horizontal Expansion: 0 mm
Initial Layer Horizontal Expansion: 0 mm
Hole horizontal expansion: 0
Z Seam Alignment: User Specified
Z Seam Position: Back
       Z Seam X: *Average length of your printer's plate (e.g."150" if your plate is 300mm on the X-axis)*
       Z Seam Y: *A value higher than the length of your plate on the Y-axis* (e.g. 700)
Seam Corner Preference: Hide Seam
Extra Skin Wall Count: 1
Skin Overlap Percentage: 10%
Skin Overlap 0.04 mm

Infill
Infill Density: 20% (*for all smaller parts and for all parts of connectors use 100% Infill*)
Infill Pattern: Triangles
Connect Infill Lines: Check
Infill Line Directions: [ ]
Infill X Offset: 0 mm
Infill Y Offset: 0 mm
Infill Line Multiplier: 1
Extra Infill Wall Count: 0
Infill Overlap Percentage: 10-20%
       Infill Overlap: 0.04 mm
Skin Overlap Percentage: 5%
       Skin Overlap: 0.02 mm
Infill Wipe Distance: 0 mm
Infill Layer Thickness: 0.24 mm
Gradual Infill Steps: 0
Infill Before Walls: Check
Minimum Infill Area: 0 mm2
Skin Removal Width: 0.8 mm
       Top Skin Removal Width: 0.8 mm
       Bottom Skin Removal Width: 0.8 mm
Skin Expand Distance: 0.8
       Top Skin Expand Distance: 0.8
       Bottom Skin Expand Distance: 0.8

Maximum Skin Angle for Expansion: 90˚
Minimum Skin Width for Expansion: 0.0
Skin Edge Support Thickness: 0
Skin Edge Support Layers: 0

Material
Initial Layer Flow: 100%
Printing Temperature: See your filament settings
Initial Printing Temperature: Your filament settings
Final Printing Temperature: Your filament settings
Build Plate Temperature: Your filament settings
Build Plate Temperature Initial Layer: Your filament settings + 5°
Flow: 100% (*Important! If you face difficulty printing the model, you may need to adjust the Flow parameter. You may research the topic using the Internet or seek assistance at our Customer Support Team at support@gambody.com*)

Speed

You can increase the printing Speed by 20% when you print simple objects. For small/thin parts you need to decrease the Speed by 25% - 50%.

Print Speed: 50 mm/s
        Infill Speed: 50 mm/s
        Wall Speed: 25 mm/s
                Outer Wall Speed:25 mm/s
                Inner Wall Speed: 50 mm/s
        Top/Bottom Speed: 25mm/s
        Support Speed: 25 mm/s
                Support Infill Speed: 45 mm/s
                Support Interface Speed: 25 mm/s
                        Support Roof Speed: 25 mm/s
                        Support Floor Speed: 25 mm/s
Travel Speed: 80 mm/s
Initial Layer Speed: 80 mm/s
        Initial Layer Print Speed: 20 mm/s
        Initial Layer Travel Speed: 80 mm/s
Skirt/Brim Speed: 20 mm/s
Z Hop Speed: 5 mm/s
Number of Slower Layers: 2
Enable Acceleration Control: Check

When printing simple objects, you need to set all Acceleration parameters at 500 mm/s. For small/thin parts you need to decrease the Acceleration by 50% - 70%.

Travel
Enable Retraction: Check
Retraction Distance: 4-8 mm, 1-3 mm for Direct Extruder (*This is the most important retraction parameter. You can find your optimal value of Retraction Distance by printing any test object, e.g. bridges, towers etc.*)
Retraction Speed: 25mm/s
      Retraction Retract Speed: 25 mm/s
      Retraction Prime Speed: 25 mm/s
Retraction Extra Prime Amount: 0 mm3
Retraction Minimum Travel: 1.5 mm
Maximum Retraction Count: 100
Minimum Extrusion Distance Window: 6,5 - 10 mm
Limit Support Retractions: Check
Combing Mode: All
Max Comb Distance With No Retract: 30 mm
Retract Before Outer Wall: Check
Avoid Printed Parts When Travelling: Check
Avoid Supports When Travelling: Check
Travel Avoid Distance: 1 mm
Layer Start X: 0.0 mm
Layer Start Y: 0.0 mm
Z Hop When Retracted: Check
Z Hop Height: 0,3 mm

Cooling
Enable Print Cooling: Check
Fan Speed: 100%
      Regular Fan Speed: 100%
      Maximum Fan Speed: 100%
Regular/Maximum Fan Speed Threshold: 10 s
Initial Fan Speed: 0%
Regular Fan Speed at Height: 0.36 mm
      Regular Fan Speed at Layer: 3
Minimum Layer Time: 10 s
Minimum Speed: 10 mm/s

Support
Generate Support: Check
Support Structure: Normal (*you can try using Tree Support Structure if you have difficulty printing any particular assembly part*)
Support Placement: Everywhere
Support Overhang Angle: 60° (*this parameter can range from 30° to 70° depending on the part at hand*)

Support Pattern: Zig Zag
Support Wall Line Count: 1 (*stronger support that might be more difficult to remove*) 0 (*less strong support but is easier to remove*)
Support Density: 15%

      Support Line Distance: 2.6667 mm

      Initial layer support line distance: 2.667 mm

Support Z Distance: 0.12 mm

      Support Top Distance: 0.12 mm

      Support Bottom Distance: 0.12 mm

Support X/Y Distance: 0.8-1 mm
Support Distance Priority: Z overrides X/Y
Support Stair Step Height: 0.3 mm
Support Stair Step Maximum Width: 5.0 mm
Support Stair Step Minimum Slope Angle: 10°
Support Join Distance: 2.0 mm
Support Horizontal Expansion: 0.2 mm
Support Infill Layer Thickness: 0.2 mm
Gradual Support Infill Steps: 0
Minimum Support Area: 2 mm
Enable Support Interface: Check (*generates additional "pillow" on the support structure that leads to a more even surface, but can be difficult to remove in hard-to-reach areas*)

      Enable Support Roof: Check

      Enable Support Floor: Check

Support Interface Thickness: 0.8 mm

      Support Roof Thickness: 0.8 mm

      Support Floor Thickness: 0.8 mm

Support Interface Resolution 0.2 mm
Support Interface Density: 50-100%

      Support Roof Density: 50-100%

         Support Roof Line Distance: 0.8 mm

      Support Floor Density: 50-100%

         Support Floor line Distance: 0.4 mm

Support Interface Pattern: Grid

      Support Roof Pattern: Grid (*this parameter should differ from Bottom Pattern Initial Layer in "Shell" section*)

      Support Floor Pattern: Grid

Minimum Support Interface Area: 10mm

      Minimum Support Roof Area: 10 mm

      Minimum Support Floor Area: 10 mm

Support Interface Horizontal Expansion: 0.0 mm

      Support Roof Horizontal Expansion: 0.0 mm

      Support Floor Horizontal Expansion: 0.0 mm

Fan Speed Override: Check
Supported Skin Fan Speed: 100%

Use Towers: Check
Tower Diameter: 4 mm
Minimum Diameter: 3.0 mm
Tower Roof Angle: 65°

Build Plate Adhesion
Build Plate Adhesion Type: Skirt/Brim (*For unsteady parts, and those parts that may come unstuck use "Brim". For bigger assembly parts that have large adhesion area and for all parts of locks and claws that you want to come out clean use "Skirt"*)
Skirt/Brim Minimum Length: 250 mm
Brim Width: 8.0 mm
    Brim Line Count: 10
Brim Only on Outside: Check

Mesh Fixes
Union Overlapping Volumes: Check
Merged Meshes Overlap: 0.15 mm

Special Modes
Print Sequence: All at Once
Surface Mode: Normal

Experimental
Slicing Tolerance: Middle
Maximum Resolution: 0.01 mm
Flow rate compensation max extrusion offset: 0 mm
Flow rate compensation factor: 100%