Settings in Repetier-Host to slice for the Malyan M180
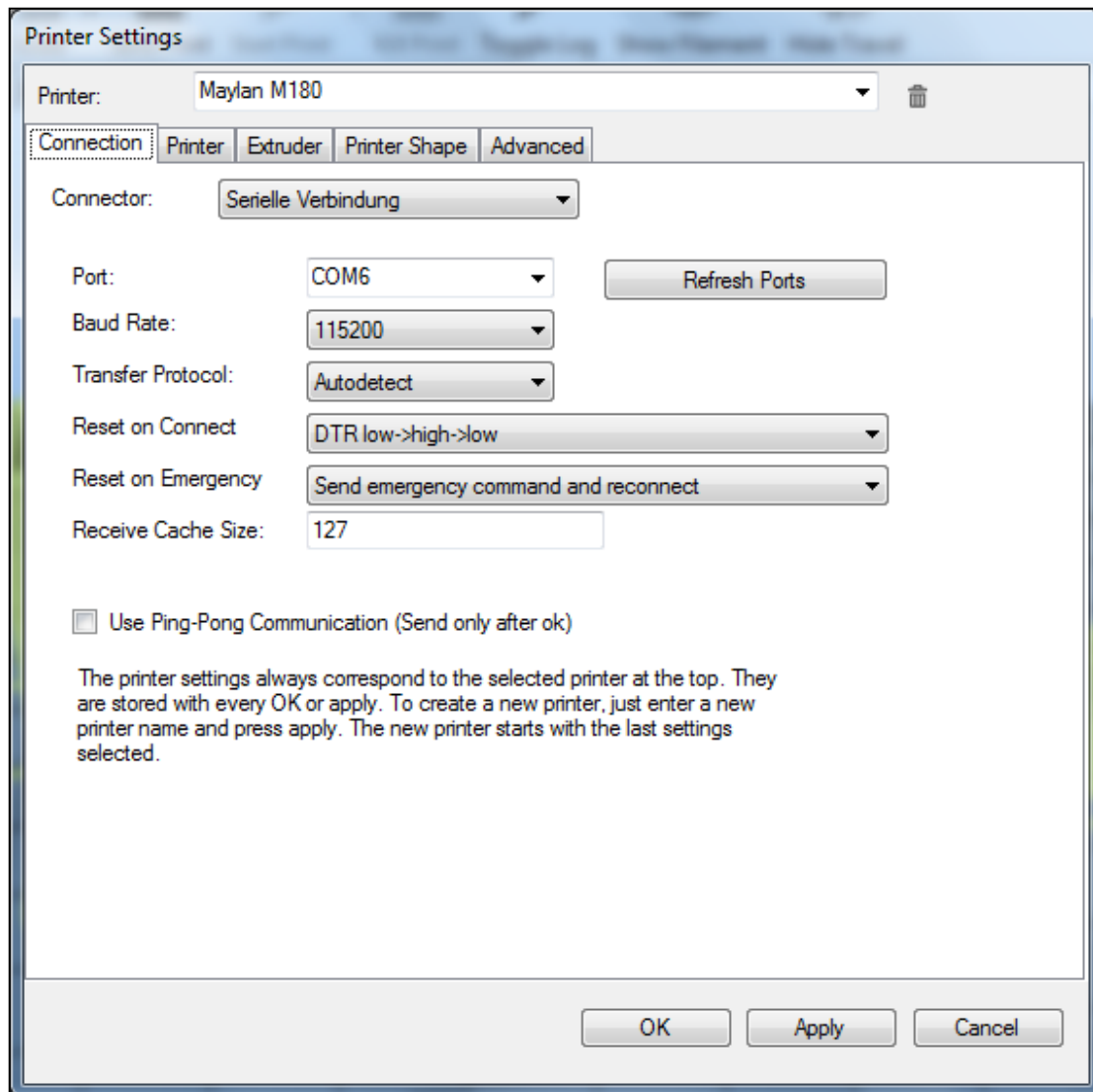
By D. Herrendoerfer.

Repetier-Host settings:

In connection settings, choose any available port, but don't ever connect to the Malyan 3D printer directly.

Some sane settings for speeds and features (but won't be used much)

Printer Settings

Printer: Maylan M180

Connection | Printer | Extruder | Printer Shape | Advanced

Travel Feed Rate:	3000	[mm/min]
Z-Axis Feed Rate:	120	[mm/min]
Manual Extrusion Speed:	2	20 [mm/s]
Manual Retraction Speed:	30	[mm/s]
Default Extruder Temperature:	185	℃
Default Heated Bed Temperature:	0	℃

☑ Check Extruder & Bed Temperature
☐ Remove temperature requests from Log
Check every 3 seconds.

Park Position: X: 0   Y: 0   Z min: 0   [mm]

☑ Send ETA to printer display   ☐ Go to Park Position after Job/Kill
☑ Disable Extruder after Job/Kill   ☑ Disable Heated Bed after Job/Kill
☑ Disable Motors after Job/Kill   ☐ Printer has SD card

Add to comp. Printing Time   8   [%]

Invert Direction in Controls for   ☐ X-Axis   ☐ Y-Axis   ☐ Z-Axis

OK | Apply | Cancel

Settings for the extruders, these are important to have correctly.

Printer shape settings, also very important.

Printer Settings

Printer: Maylan M180

| Connection | Printer | Extruder | Printer Shape | Advanced |

Printer Type: Classic Printer

Home X: Max   Home Y: Max   Home Z: Min

X Min: -105   X Max: 105   Bed Left: -105

Y Min: -75   Y Max: 75   Bed Front: -75

Print Area Width: 210   mm

Print Area Depth: 150   mm

Print Area Height: 150   mm

The min and max values define the possible range of extruder coordinates. These coordinates can be negative and outside the print bed. Bed left/front define the coordinates where the printbed itself starts. By changing the min/max values you can even move the origin in the center of the print bed, if supported by firmware.

E

OK   Apply   Cancel

Cura Slicer settings.

cura_m180_settings.rcp:

```
[print]
name = cura-m180-settings
defaultQuality = NMWgWloHpH9161JgopwO3tiCRTfi6IL0
travelSpeedMin = 80
travelSpeedMax = 150
firstLayerSpeedMin = 30
firstLayerSpeedMax = 30
infillSpeedMin = 60
infillSpeedMax = 100
infillType = 0
innerPerimeterMin = 40
innerPerimeterMax = 80
outerPerimeterMin = 30
outerPerimeterMax = 60
printSpeedMin = 40
printSpeedMax = 60
nozzleDiameter = 0
gcodeFlavour = 0
retractionCombing = True
multiVolumeOverlap = 0
cutOffObjectBottom = 0
shellThickness = 0.8
infillTopBottomThickness = 0.6
infillDensityPercent = 20
supportType = 0
platformAdhesion = 0
dualExtrusion = 0
wipeAndPrimeTower = True
oozeShield = True
enableRetraction = True
retractionSpeed = 40
retractionDistance = 4
multiExtruderSwitchAmount = 16
startGCode = """
;start.gcode for Malyan M180, dual head
(**** set temperatures ****)
M109 S{BED} T0   (set bed temperature)
M104 S{TEMP0} T0 (set extruder0 temperature)
M104 S{TEMP1} T1 (set extruder1 temperature)
(**** set defaults ****)
M103 (RPM off)
M73 P0 (enable build progress)
G21 (set units to mm)
G90 (set positioning to absolute)
(**** begin homing ****)
G162 X Y F2500 (home XY axes maximum
G161 Z F1100 (home Z axis minimum)
G92 Z-5 (set Z to -5)
```

```
G1 Z0.0 (move Z to "0")
G161 Z F100 (home Z axis minimum)
M132 X Y Z A B (Recall stored home offsets for XYZAB axis)
(**** move to waiting pos and haet up ****)
G1 X104 Y-74 Z20 F3300.0 (move to waiting position)
G130 X0 Y0 A0 B0 (Lower stepper Vrefs while heating)
M108 T0 (Select tool 0)
M6 T0 (wait for toolhead, and HBP to reach temperature)
M6 T1 (wait for toolhead, and HBP to reach temperature)
G130 X127 Y127 A127 B127 (Set Stepper motor Vref to defaults)
M108 R3.0 T0
G0 X104 Y-74 (Position Nozzle)
M108 R4.0     (Set Extruder Speed)
M101          (Start Extruder)
G4 P1500      (Create Anchor)
"""
endGCode = """
;end.gcode for Malyan M180
M73 P100 ( End  build progress )
G0 Z150 ( Send Z axis to bottom of machine )
;M18 ( Disable steppers )
M109 S0 T0 ( Cool down the build platform )
M104 S0 T0 ( Cool down the Right Extruder )
M104 S0 T1 ( Cool down the Left Extruder )
G162 X Y F2500 ( Home XY endstops )
;M18 ( Disable stepper motors )
"""
preSwitchExtruderCode =
postSwitchExtruderCode =
wipeAndPrimeTowerVolume = 15
retractionMinimumTravel = 1.5
minimumExtrusionBeforeRetract = 0.02
zhop = 0
skirtLineCount = 3
skirtDistance = 3
skirtMinimumLength = 150
solidInfillTop = True
solidInfillBottom = True
infillOverlapPercent = 15
supportStructureType = 0
supportOverhang = 60
supportFillPercent = 15
supportDistanceXY = 0.7
supportDistanceZ = 0.15
supportExtruder = 1
spiralizeOuterContour = False
brimWidth = 10
raftExtraMargin = 5
raftLineSpacing = 1
raftBaseThickness = 0.3
raftBaseWidth = 0.7
raftInterfaceThickness = 0.2
```

```
raftInterfaceLineWidth = 0.2
raftAirGap = 0
raftAirGapLayer0 = 0
raftSurfaceLayer = 1
fixCombineEverythingA = True
fixCombineEverythingB = False
fixKeepOpenFaces = False
fixExtensiveStitching = False
fanFullAtHeight = 0.5
minimumSpeed = 10
coolHeadLift = False
numQuality = 2
[quality_0]
name = 0.2 mm
signature = bKX8jW8x15ajVUlmKXEmH65Co1PEfZwC
layerHeight = 0.2
firstLayerHeight = 0.3
[quality_1]
name = 0.15 mm
signature = NMWgWloHpH9161JgopwO3tiCRTfi6IL0
layerHeight = 0.15
firstLayerHeight = 0.2
```

Fixing G-Code glitches

If you want to slice and print dual color, you need a small script to fix up the generated G-Code to work with the Malyan. (Since firmware is Sailfish vs. Repetier) The script fixes the tool select codes.

M180_sanify.py:

```python
#!/usr/bin/python


import sys, getopt


def main(argv):
  inputfile = ''
  outputfile = ''
  count = 0
  try:
    opts, args = getopt.getopt(argv,"hi:o:",["ifile=","ofile="])
  except getopt.GetoptError:
    print 'test.py -i <inputfile> -o <outputfile>'
    sys.exit(2)
  for opt, arg in opts:
    if opt == '-h':
      print 'm180sanify.py -i <inputfile> -o <outputfile>'
      sys.exit()
    elif opt in ("-i", "--ifile"):
      inputfile = arg
    elif opt in ("-o", "--ofile"):
      outputfile = arg

  if inputfile == '' or outputfile == '':
   sys.exit()
```

```python
    print 'Input file is ', inputfile
    print 'Output file is ', outputfile
    f1 = open(inputfile, 'r')
    f2 = open(outputfile, 'w')
    for line in f1:
        if line in ("T0\n", "T1\n"):
            f2.write(line.replace('T0', 'M108 T0').replace('T1', 'M108 T1'))
            count += 1
        else:
            f2.write(line)
    f1.close()
    f2.close()
    print 'Replaced ',count,' tool change calls.'

if __name__ == "__main__":
    main(sys.argv[1:])
```

Slicing using Repetier-Host (Workflow)

When Repetier-Host is installed and configured, and the slicer config is complete, you can use it to generate g-code for the Malyan M180 or any other 3D printer.

The objects to print are converted first into g-code, a machine-understandable language that uses standard references to movements and measures, and that is almost completely machine-agnostic. This code is then transferred into a binary machine code that is suited to your printer model, and which includes some printer specific assumptions. These binary files will only work with the printer they were made for. Still they can be saved away so prints can be made over, and over again from the same file, using the same materials.
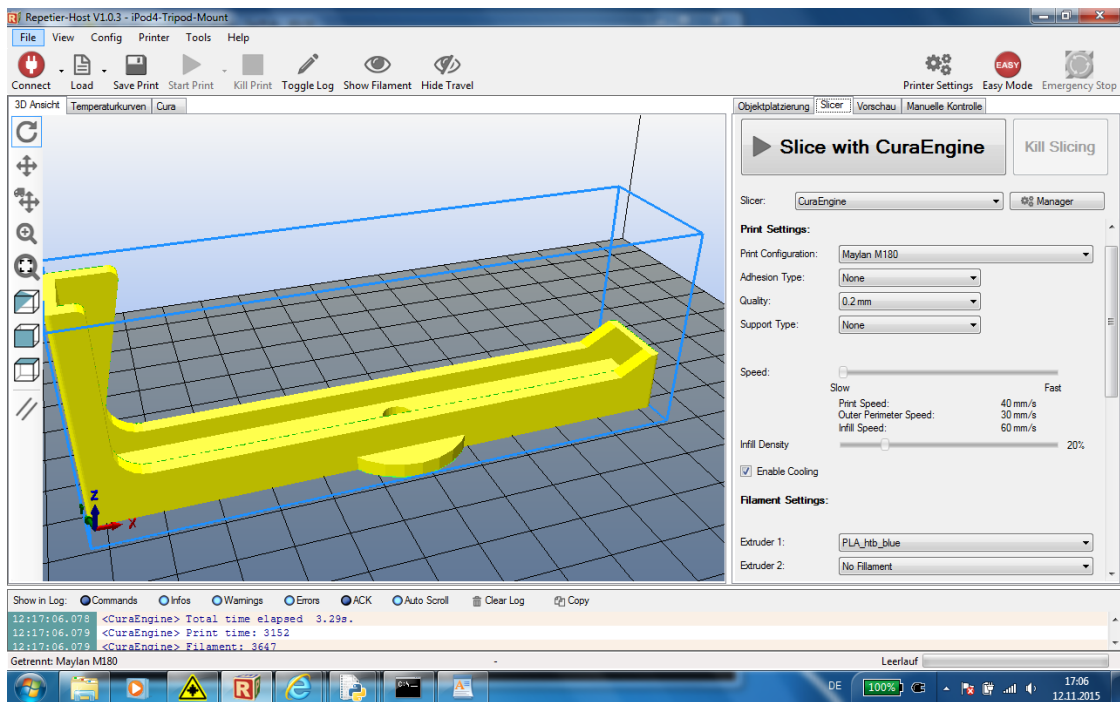

Step 1: Loading and placing the object:
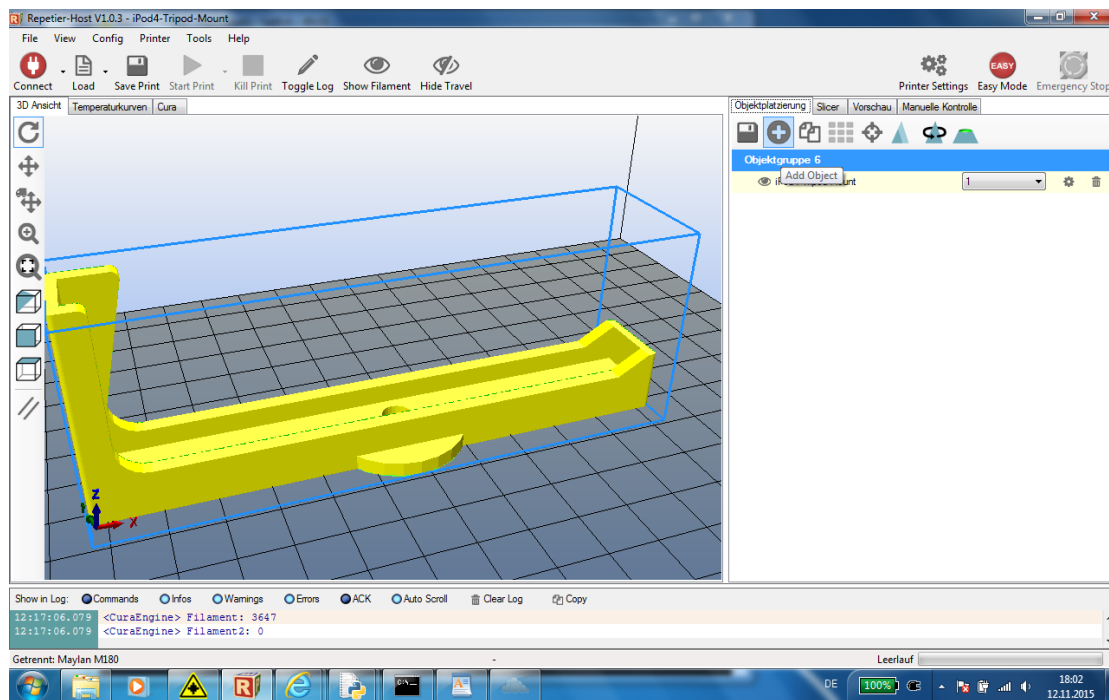
Load and place the object, make sure it is manifold, and oriented correctly.

You are free to place as many objects to the build platform as you can fit, everything will be sliced to one build file.

Step 2: Configure the slicer and slice the object(s):
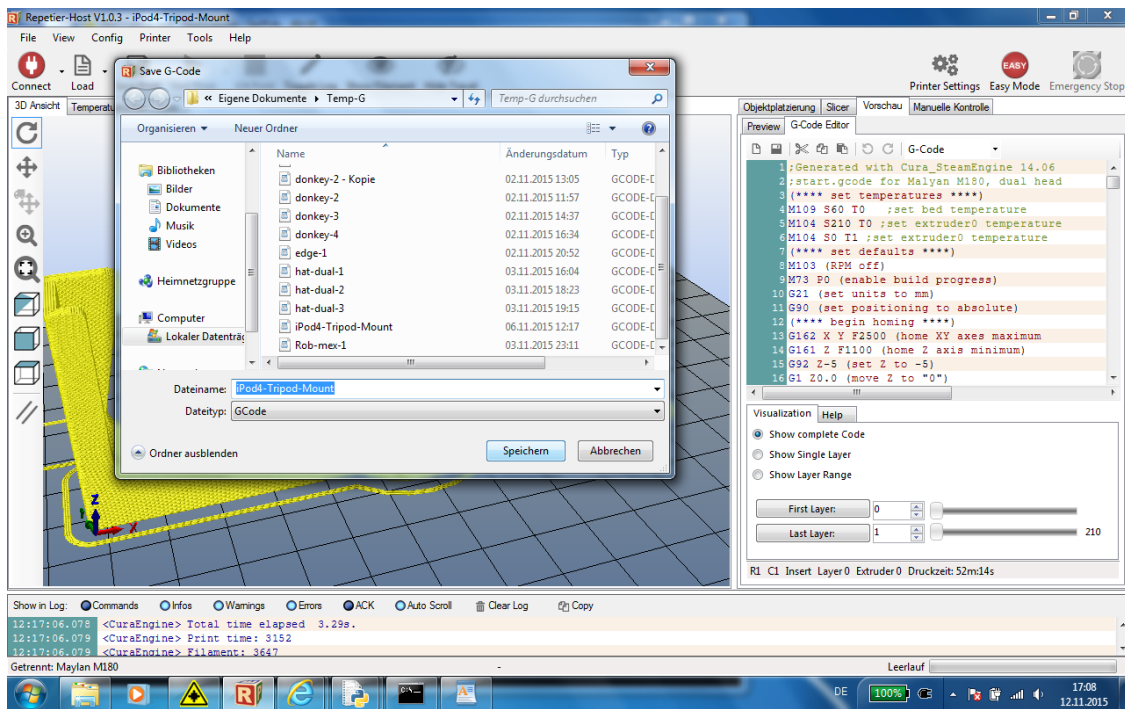

Select your options for the slice operation to the object, select a filament setting.
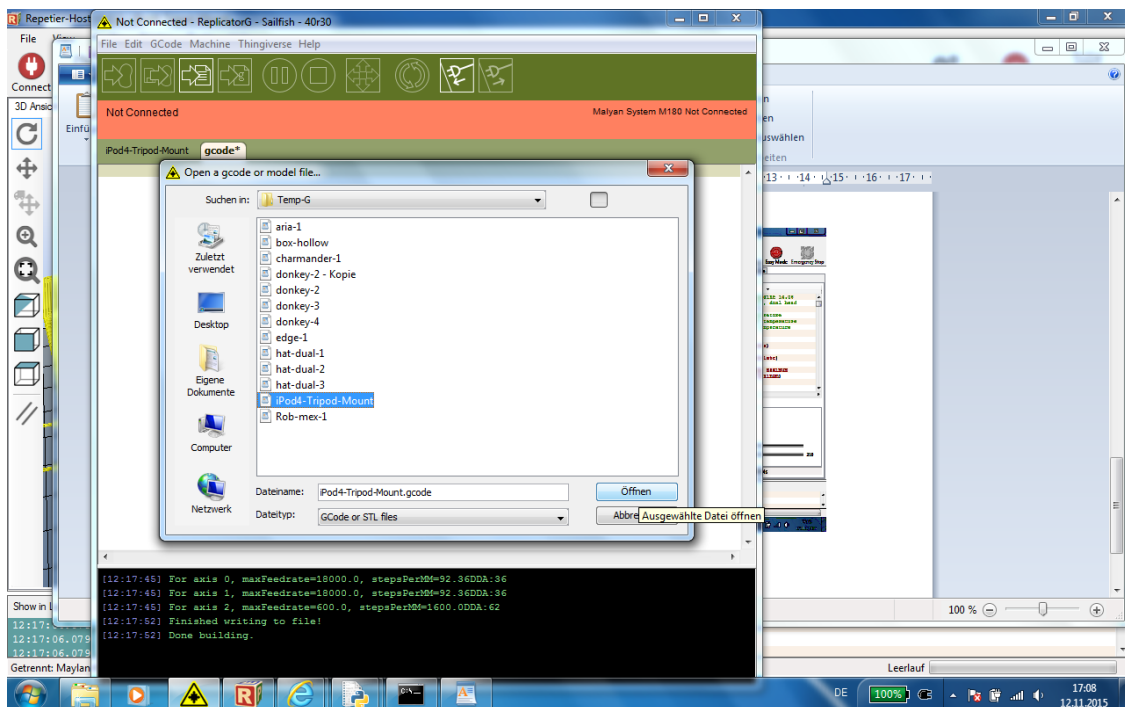
Step 3: Save the slicer output g-code to file.
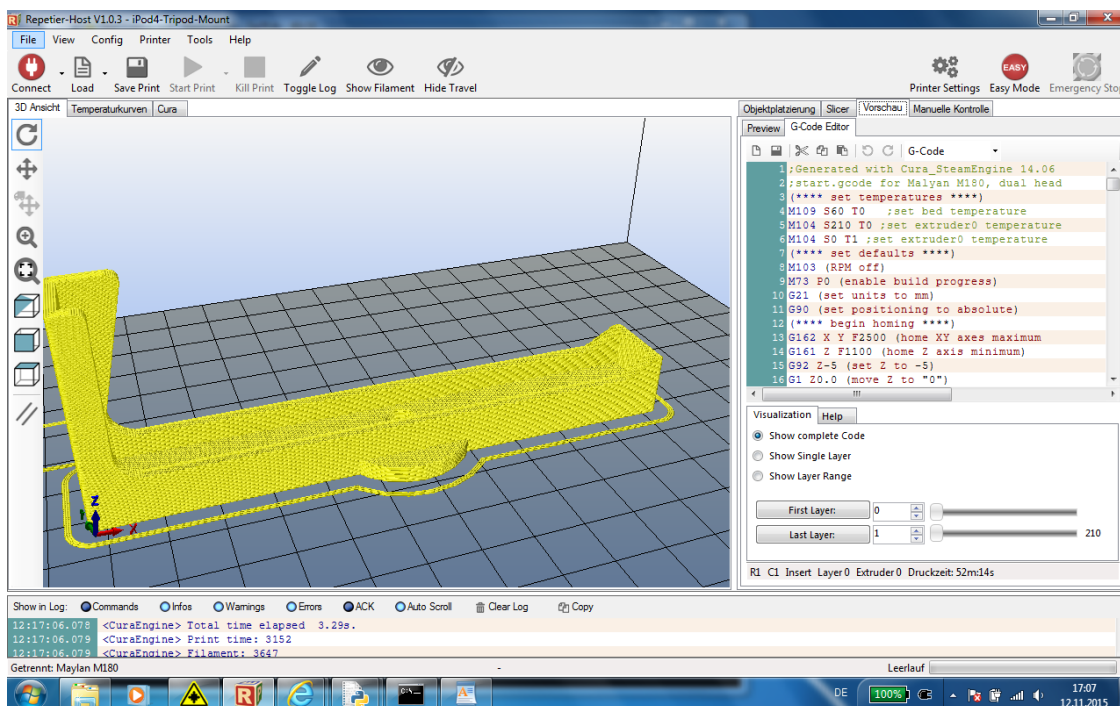
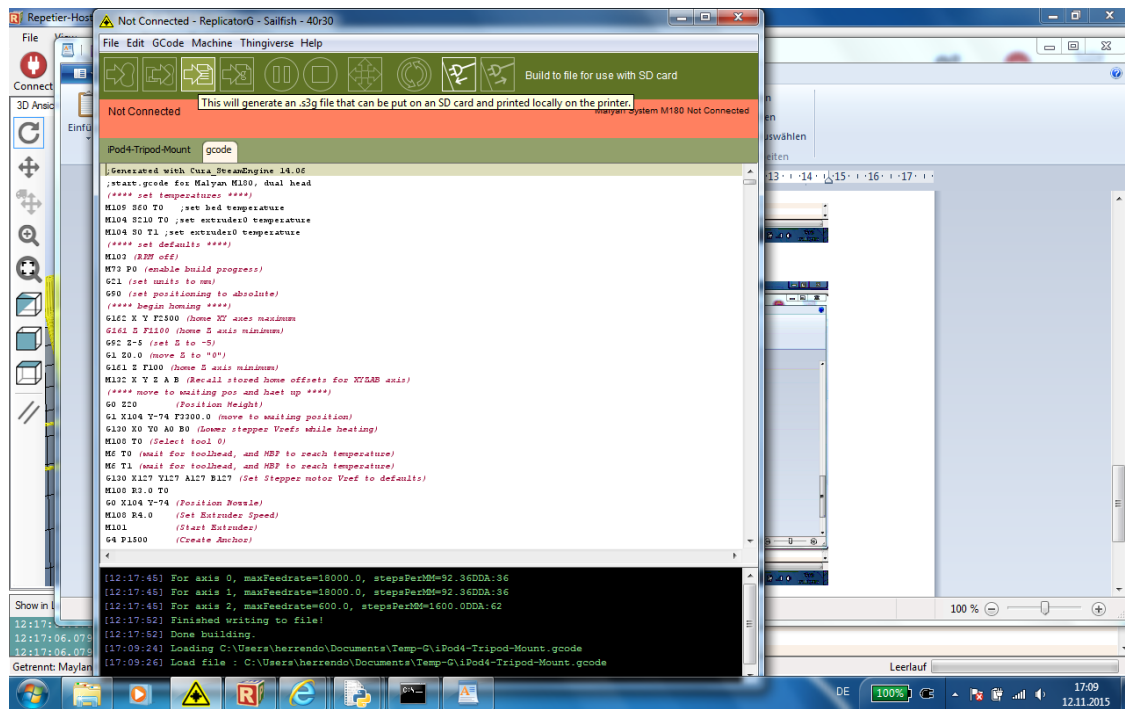Click the save icon in the g-code editor window.

Select a file name to save the g-code to and click save. (*For multi filament see remark 1)

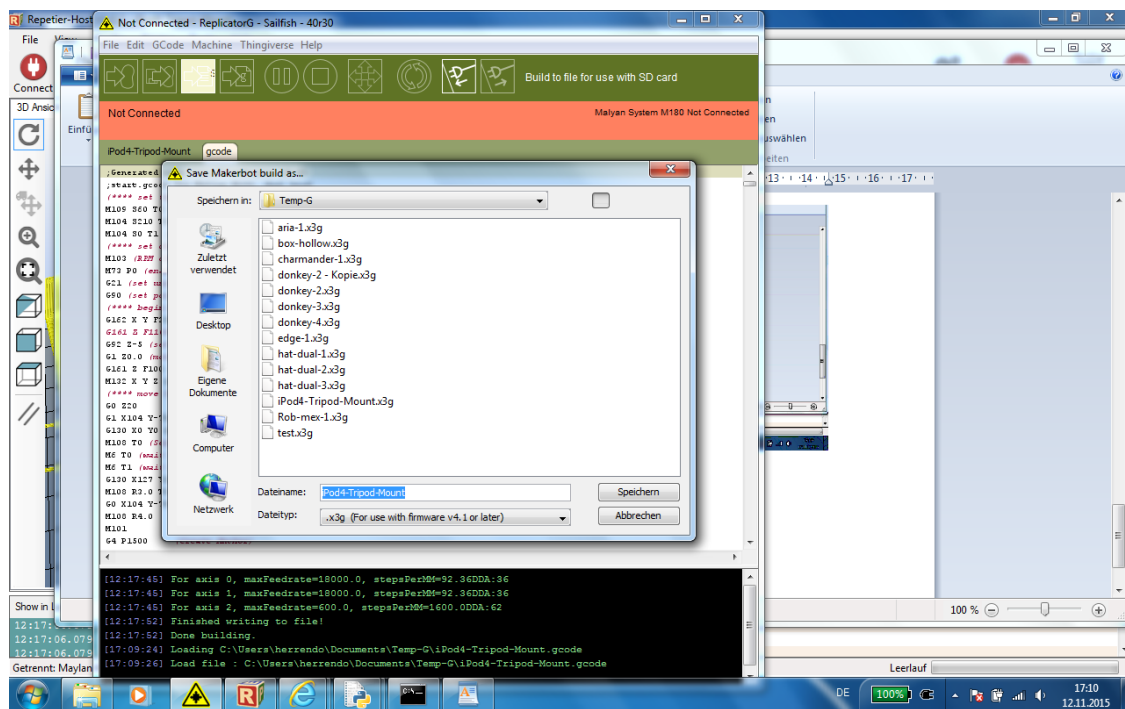Step 4: Import the generated g-code into ReplicatorG

In RelicatorG click open file and load the generated g-code file.

Step 5: Generate a binary (x3g) file from the g-code

In ReplicatorG click 'Build to file for use with SD card'

Enter a name for the generated x3g file, click save, generation of the file may take some time.

Step 6: 3D-Print the object:

Put the generated .x3g file into the root directory of a SD-card and print using the print from SD-card function of the printer.

Remark 1: If you intend to print with more than one filament using this method, it is required to do a manual transcode step after saving the g-code to a file. The step modifies the code to change the extruder. Any lines containing the single statement 'T0' must be replaced by 'M108 T0' , and all lines containing only 'T1' should be replaced with 'M108 T1'. All other references to T0 and T1 must remain unchanged. The included tool sanify.py will do this for you.