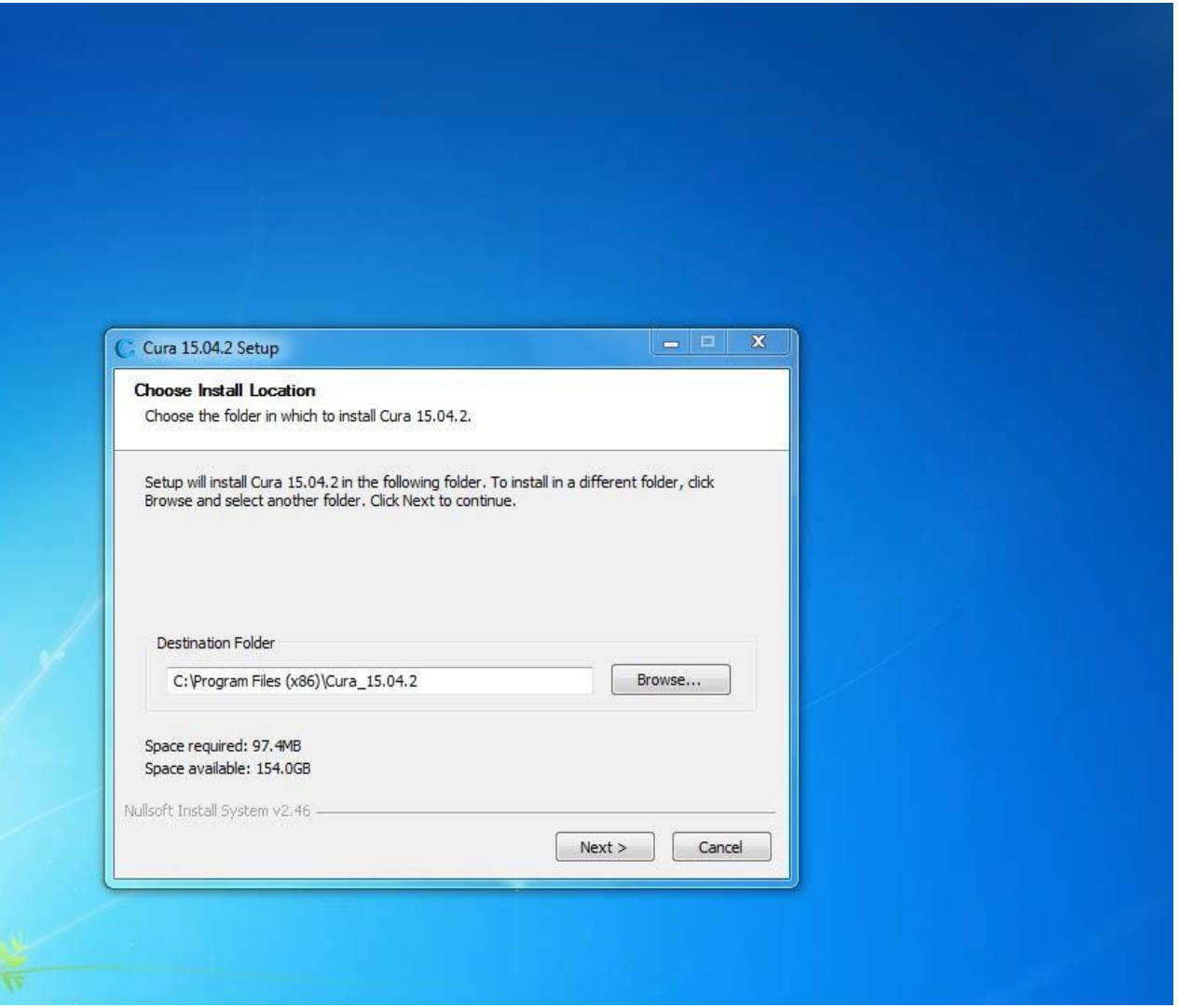
Slicer"CURA"

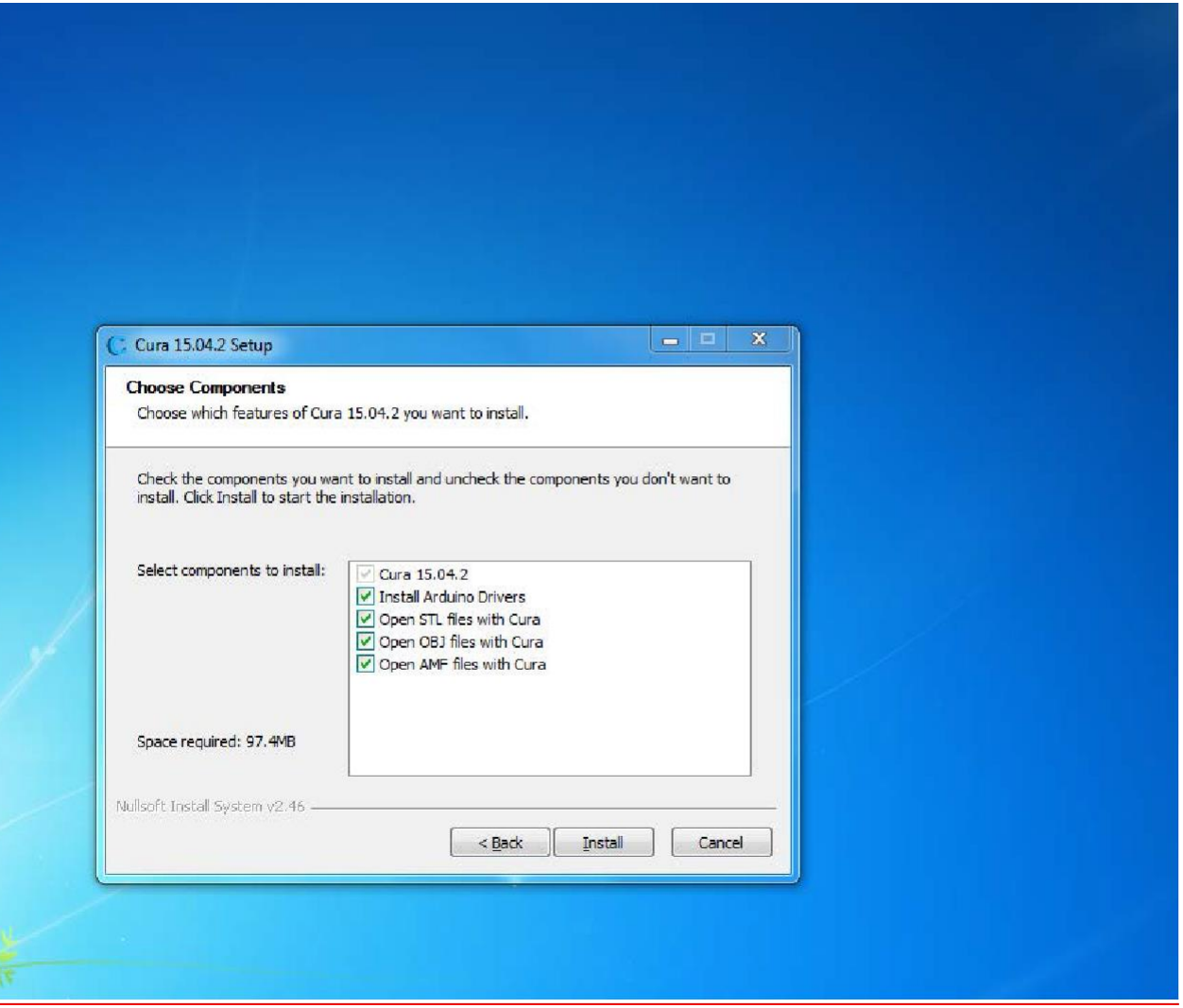
User Instructions for Olivetti 3D DESK

Cura Install - 1



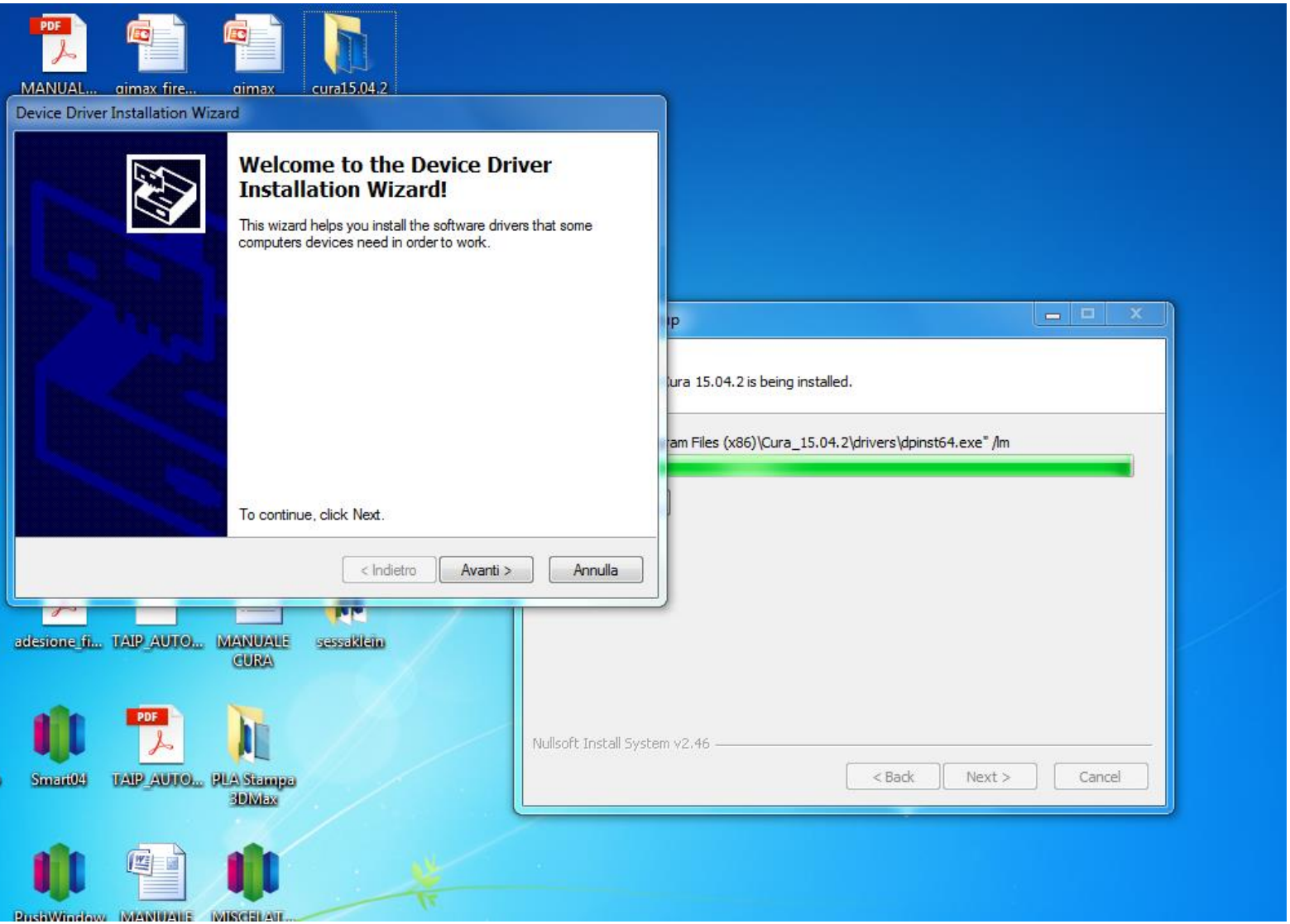
Click “Next”

Cura Install - 2



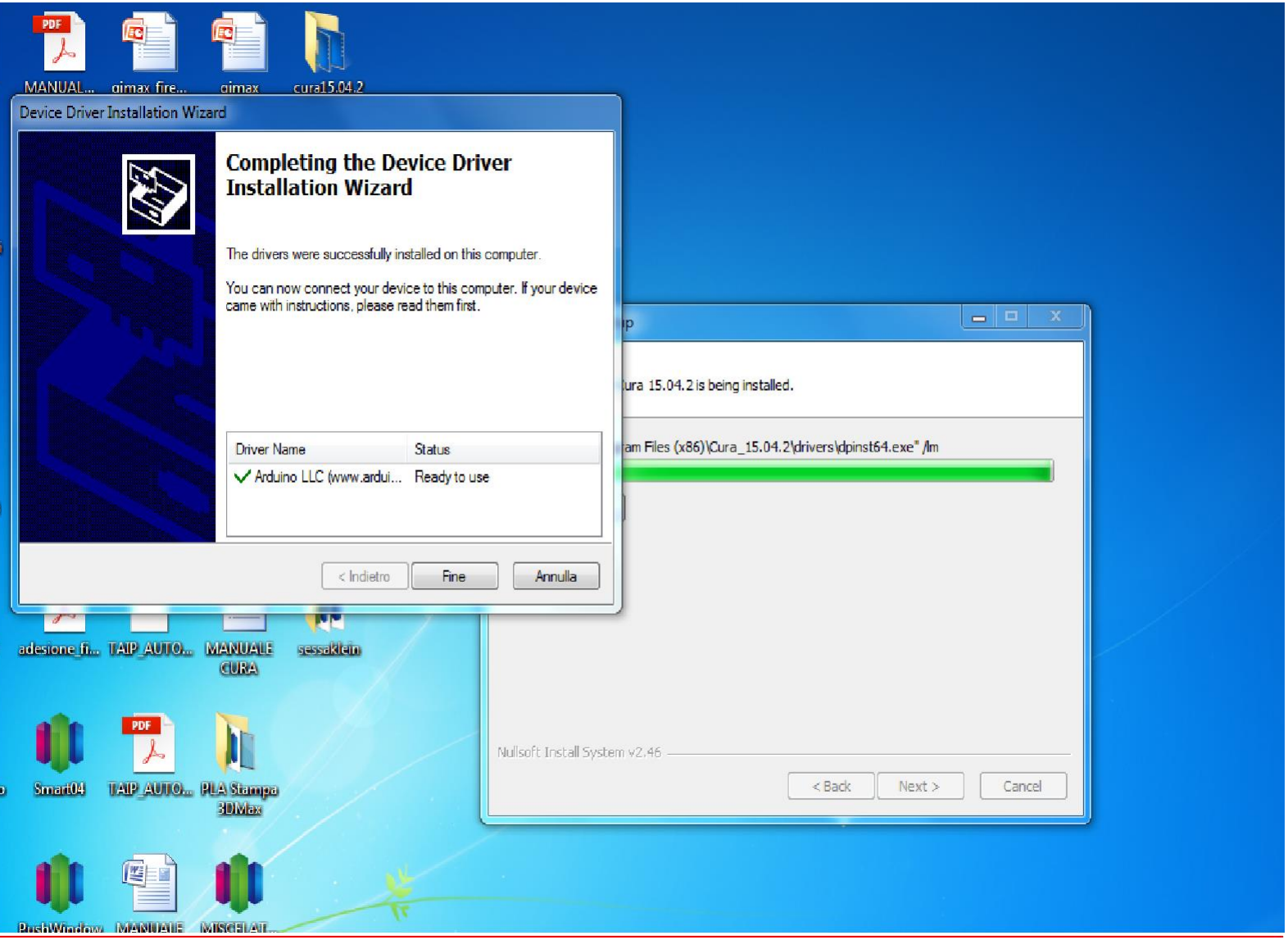
Check all components. Then click "Install"

Cura Install - 3



Click “Next”

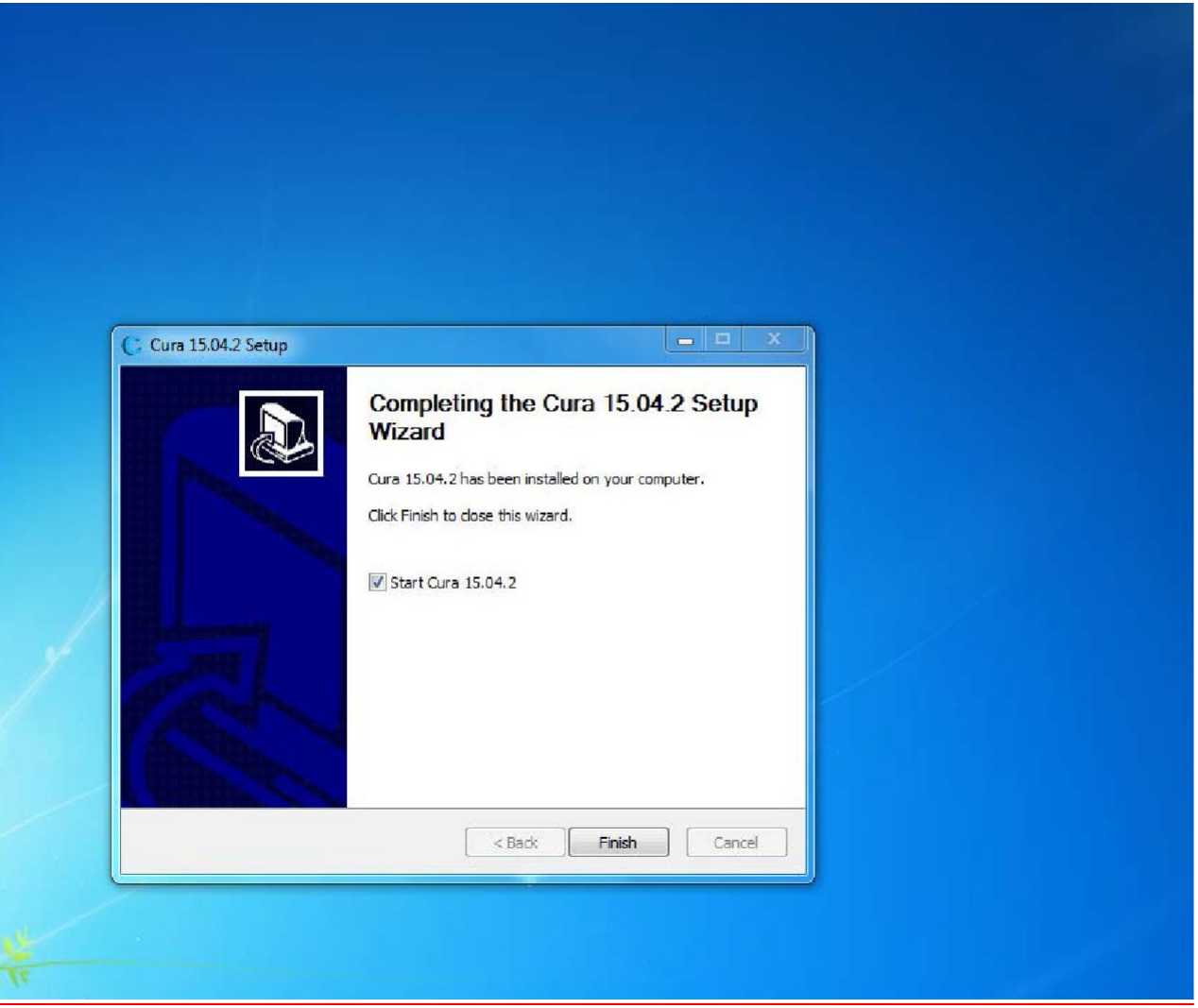
Cura Install - 4



The Wizard installs the Arduino drivers, to enable the PC to recognize the printer.

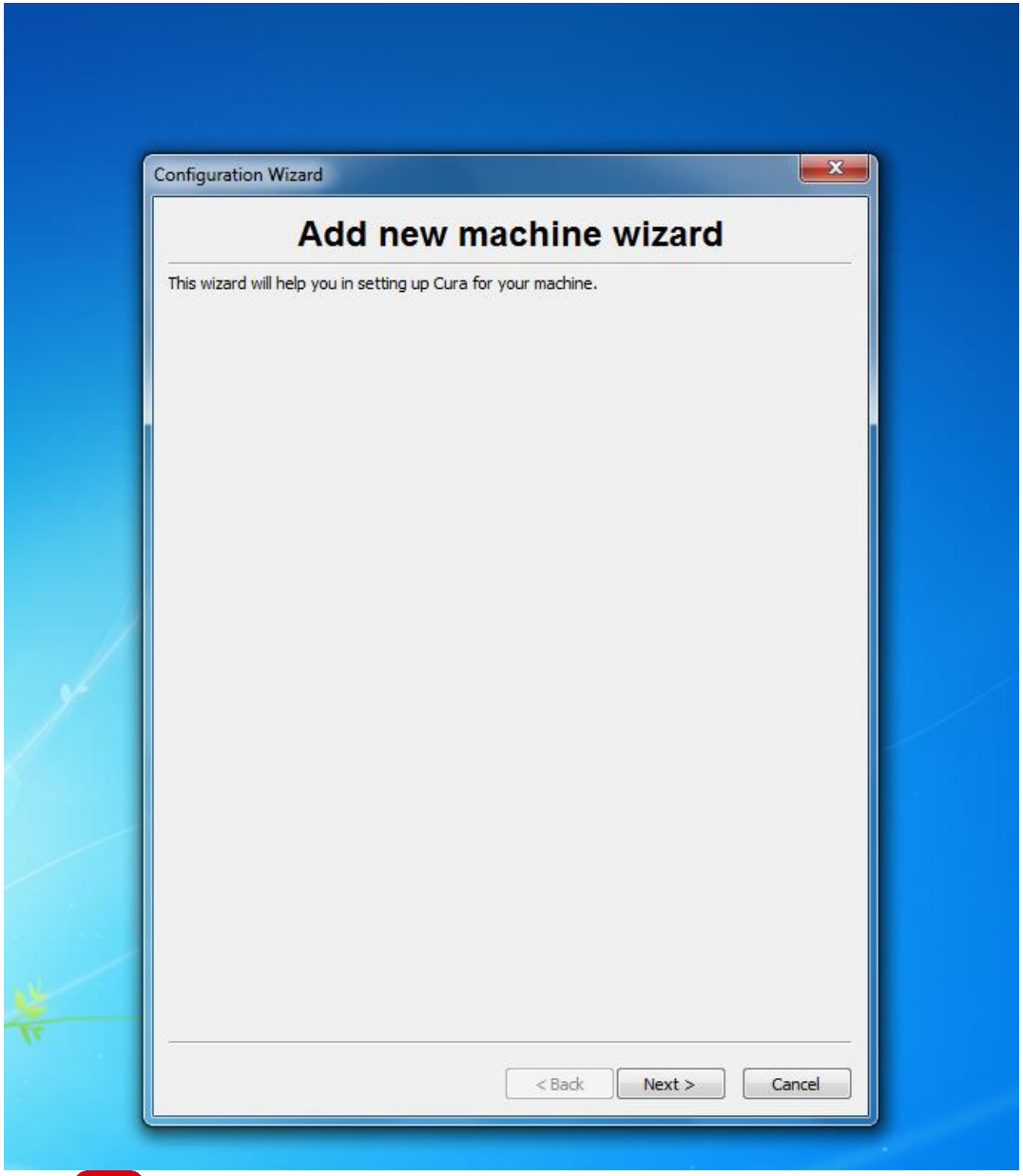
Then click “Finish”

Cura Install - 5



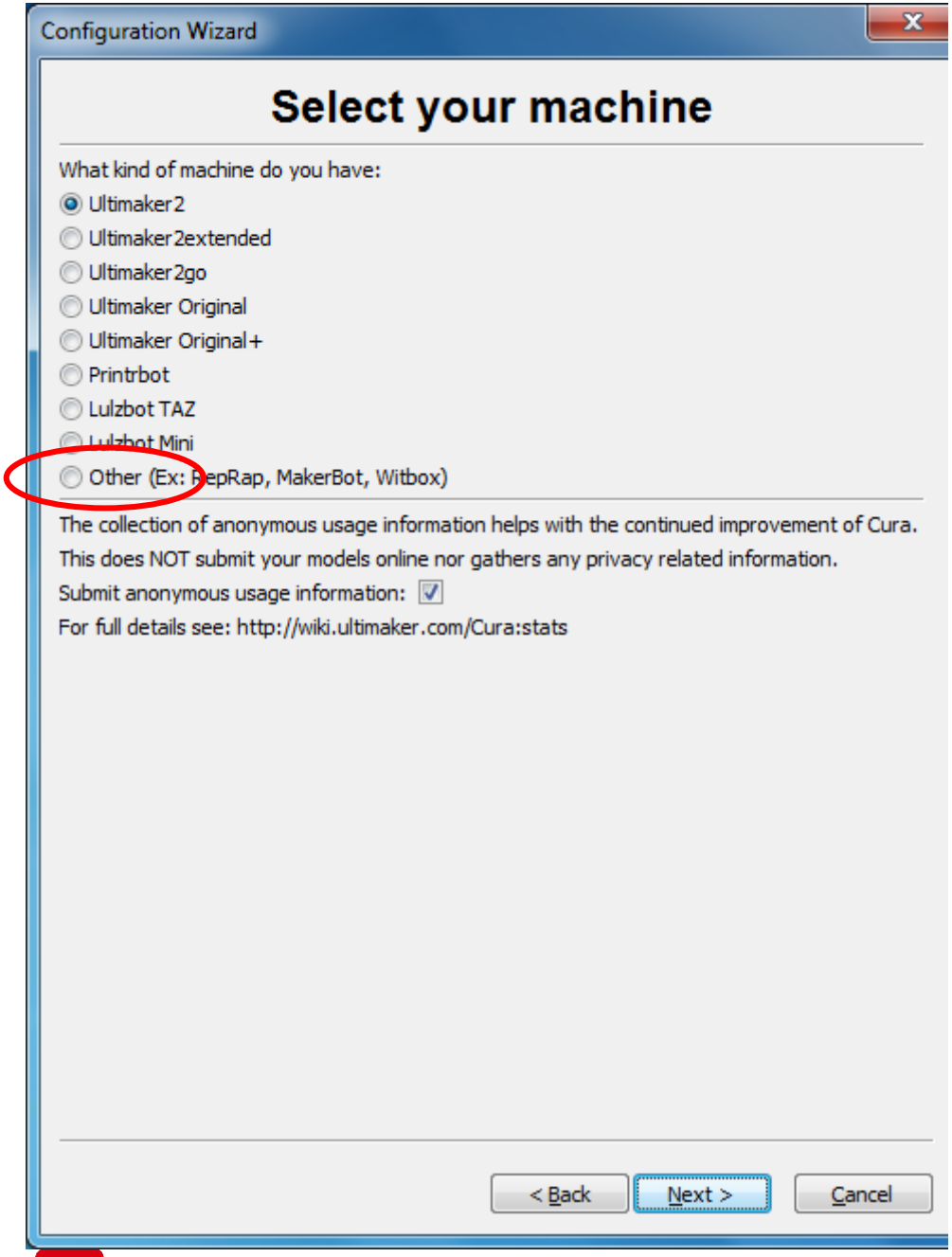
At the end of Setup, click “Finish”, keeping the flag on “Start Cura”, as shown in the figure.

Printer Setup- 1



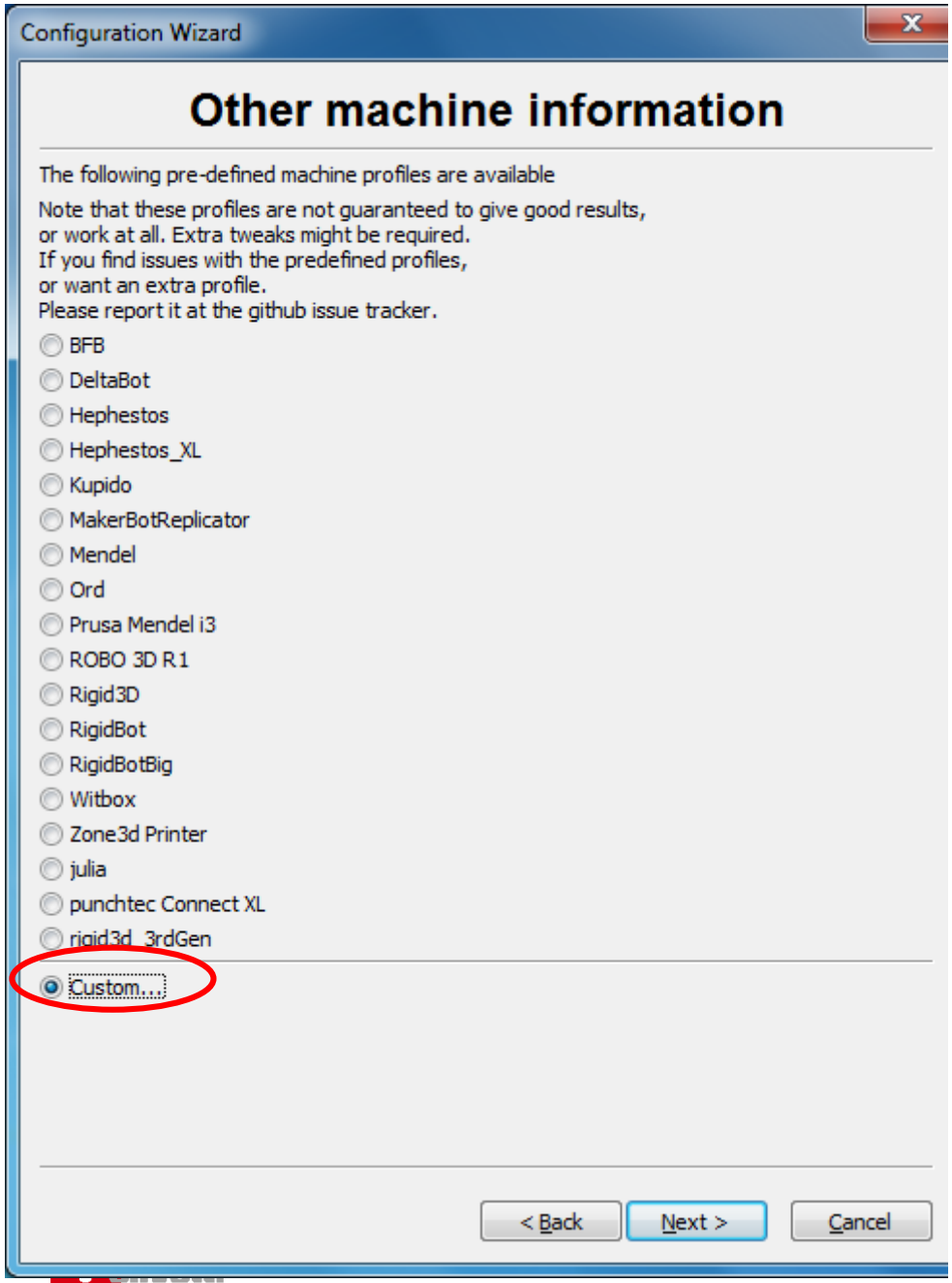
The Configuration Wizard starts automatically.
Click “Next”

Printer Setup - 2



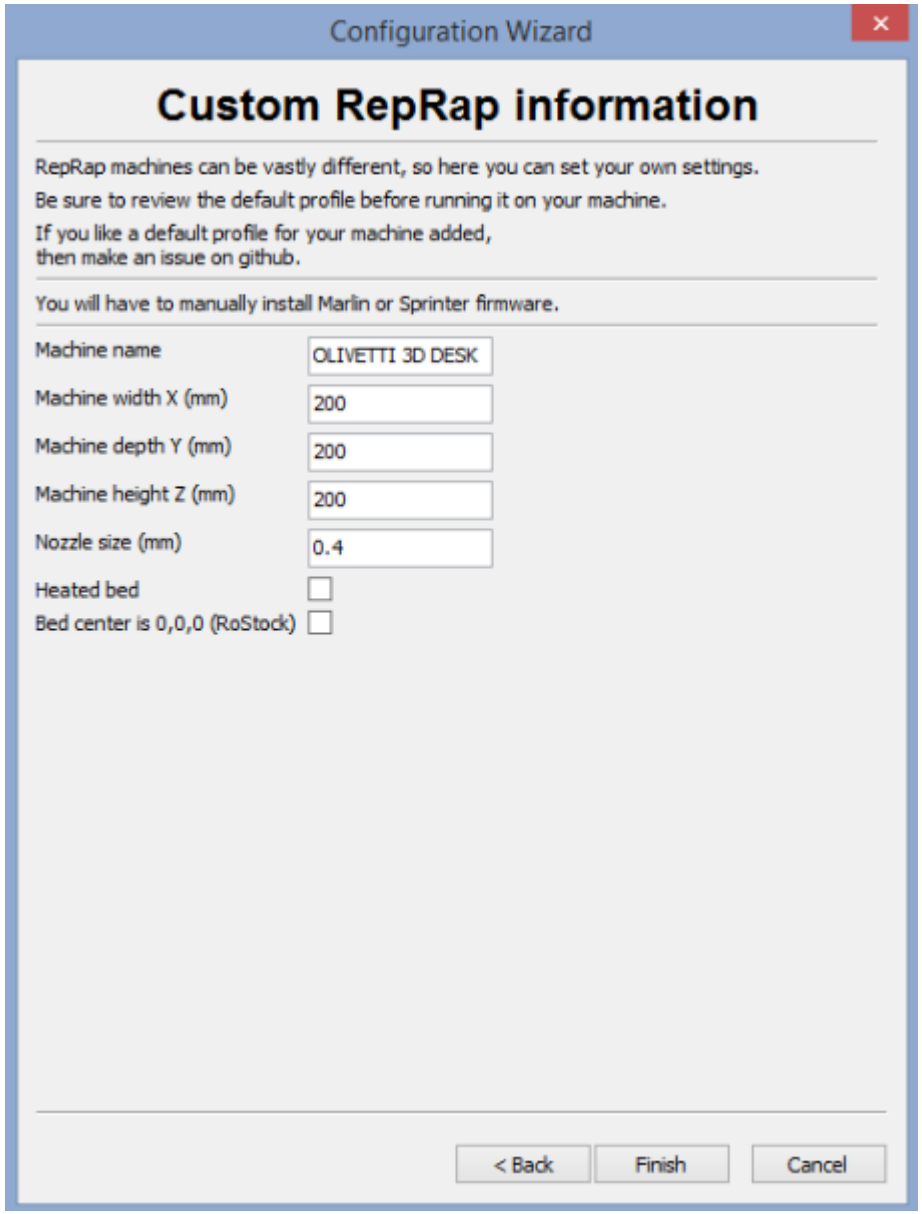
Select “Other”. Then click “Next”

Printer Setup - 3



Select “Custom”. Then click “Next”

Printer Setup - 4



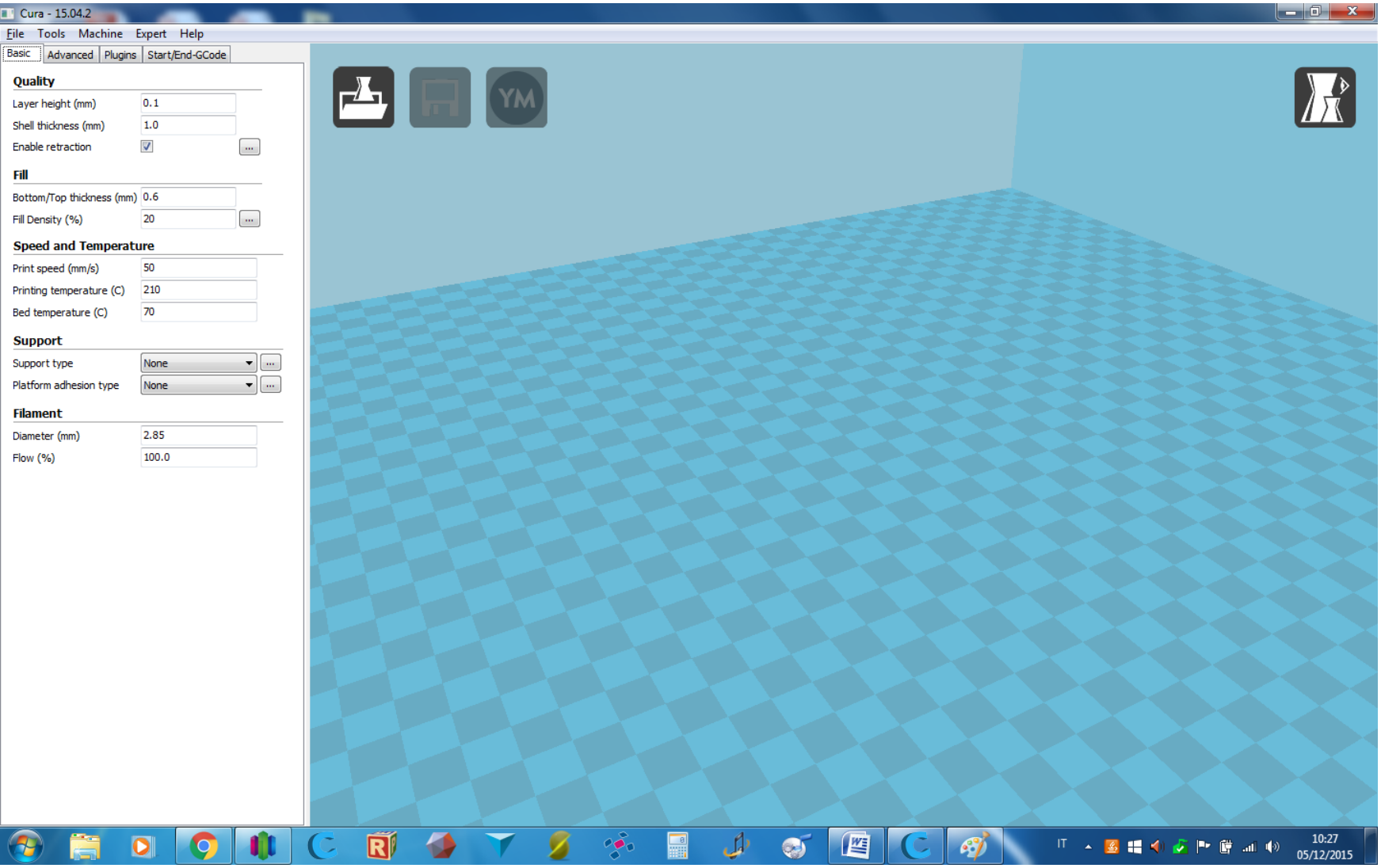
Enter the Olivetti 3D DESK data as shown in the Figure:

- **Width: 200**
- **Depth: 200**
- **Height: 200**
- **Nozzle size: 0,4**

Unflag “Heated Bed”

Click “Finish” to end the Printer Setup

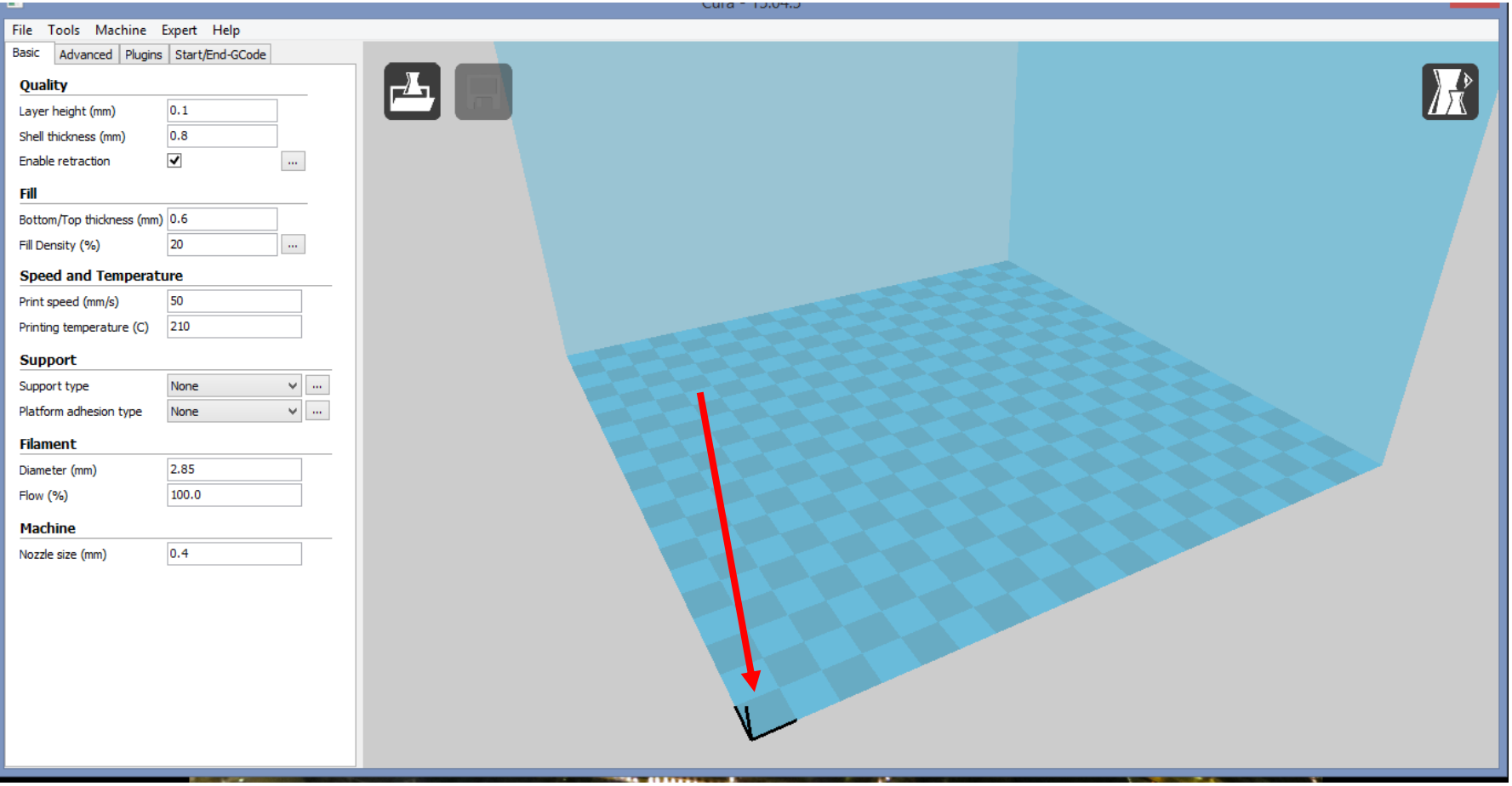
Software Setup - 1



In the first screen you can see a small robot on the printing bed.

Right-click to delete this or any other object

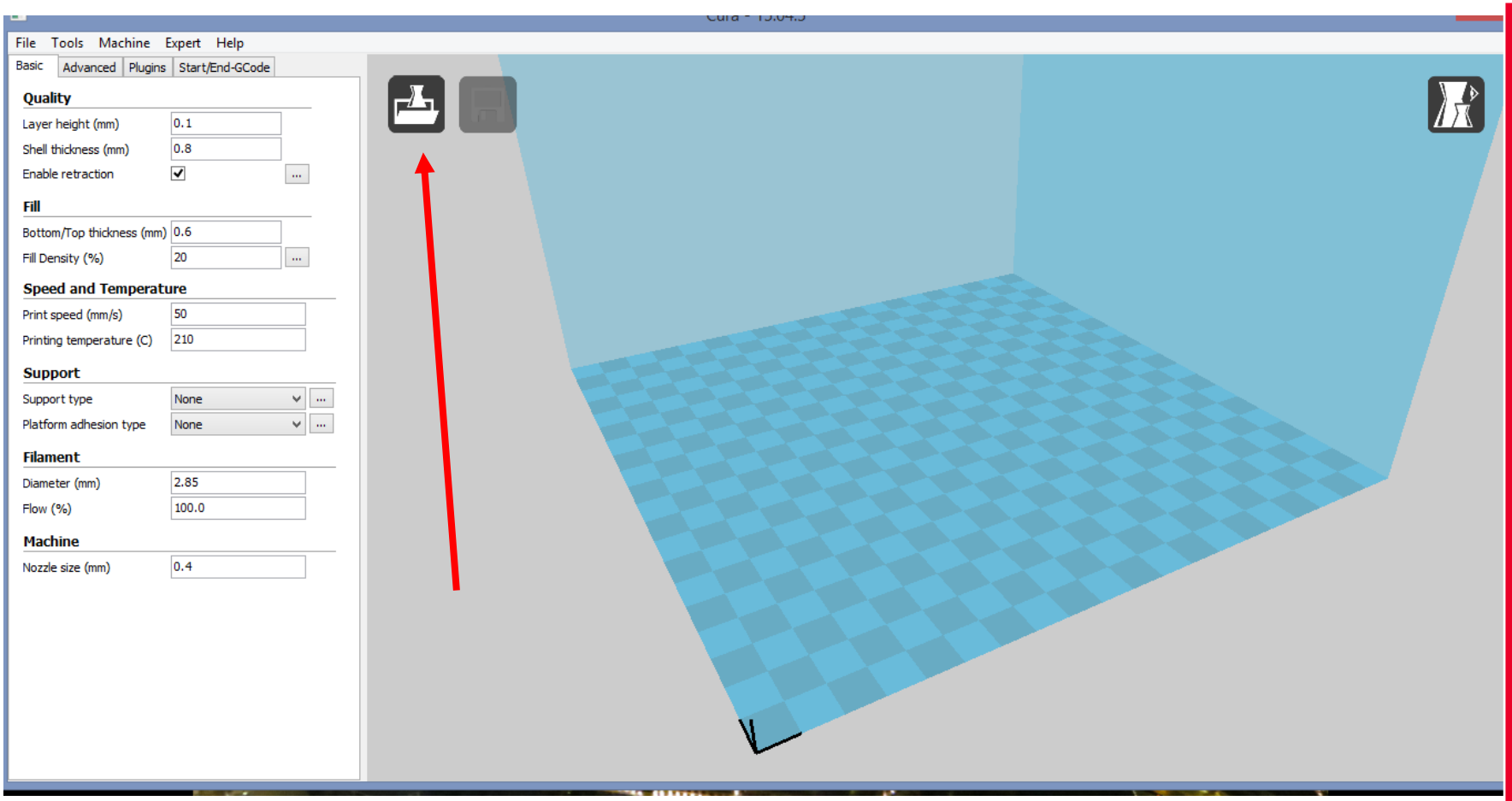
Software Setup - 2



The black triangle at the lower left corresponds to the zero point of the 3D printer.

At the printing start, the axes will move to the zero point.

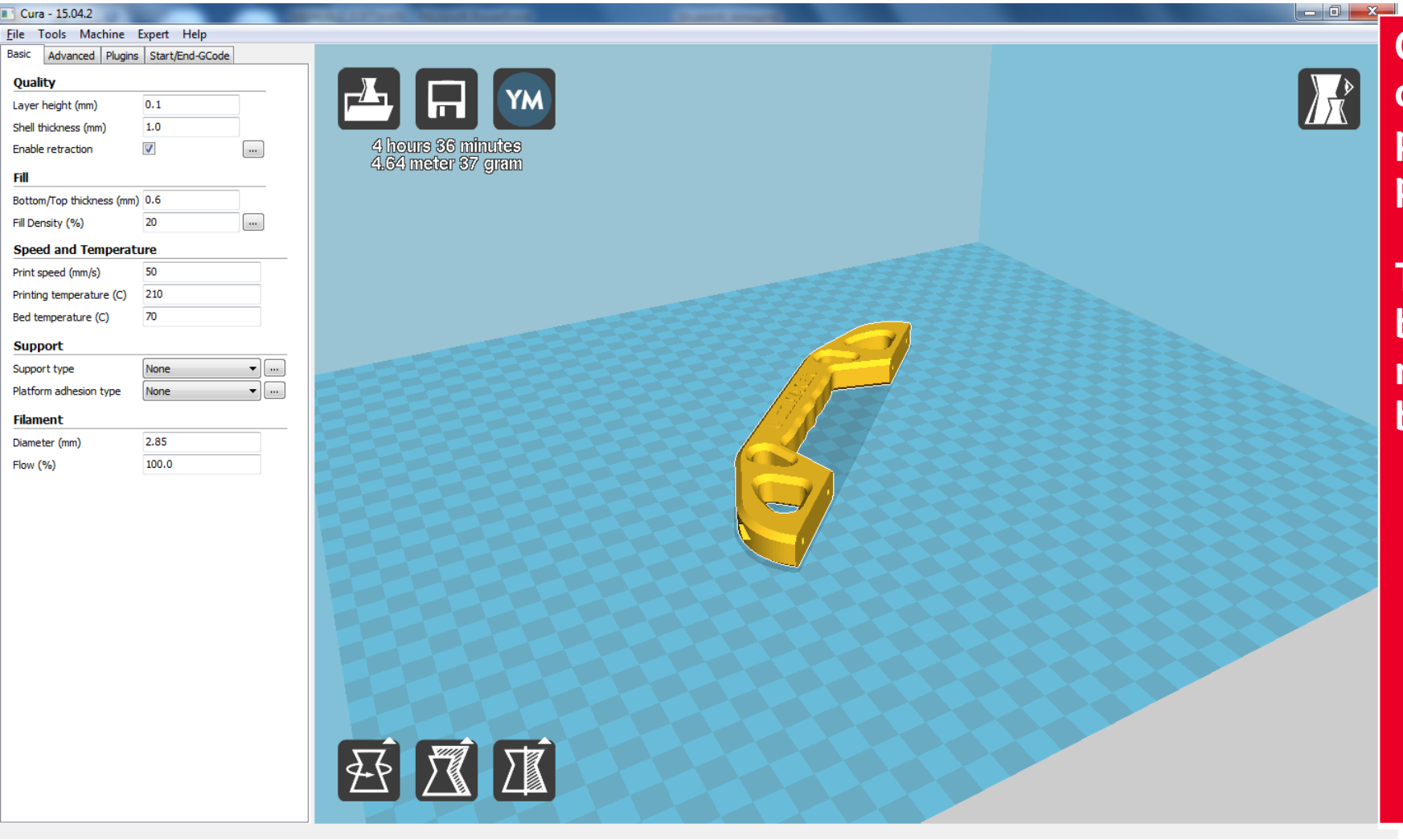
Choose and prepare the object - 1



Click on "Load" to select the object file and place it on the printing bed.

The compatible file formats are:
STL
OBJ
AMF

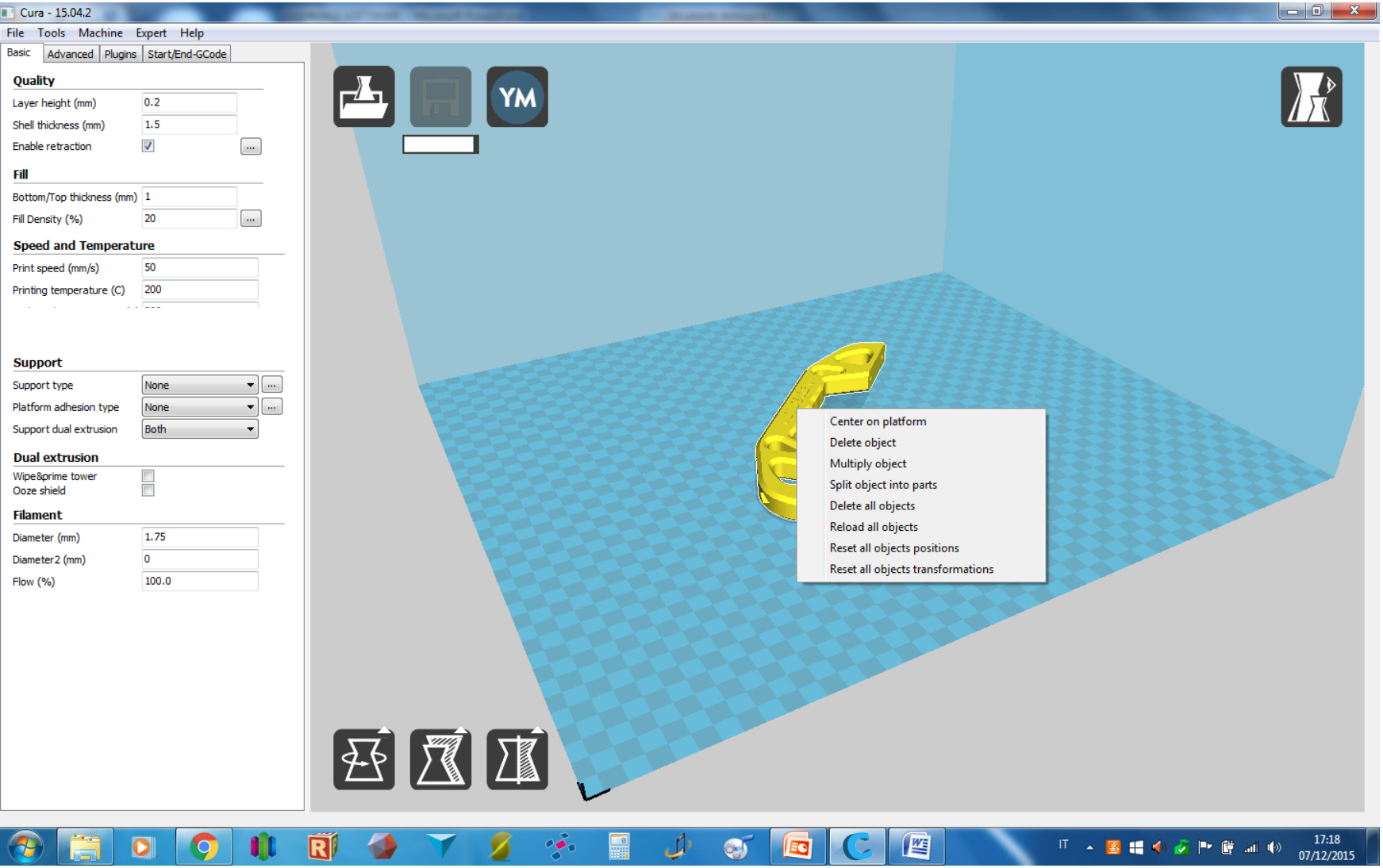
Choose and prepare the object - 2



Once imported the object, you need to position it and set the printing parameters.

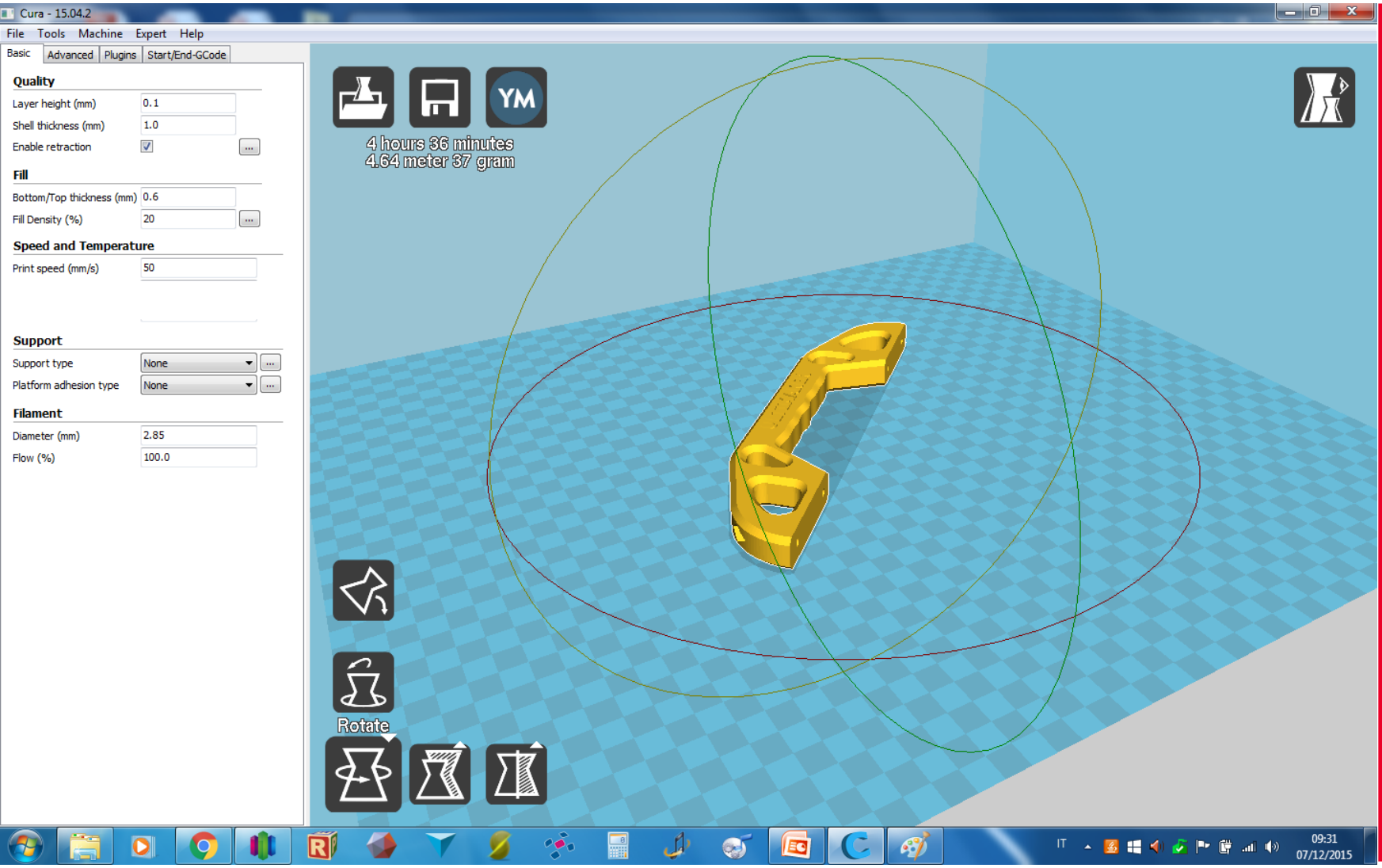
The object is placed by default in the middle of the printing bed.

Choose and prepare the object - 3



Right-click to center, delete, multiply..... the object.

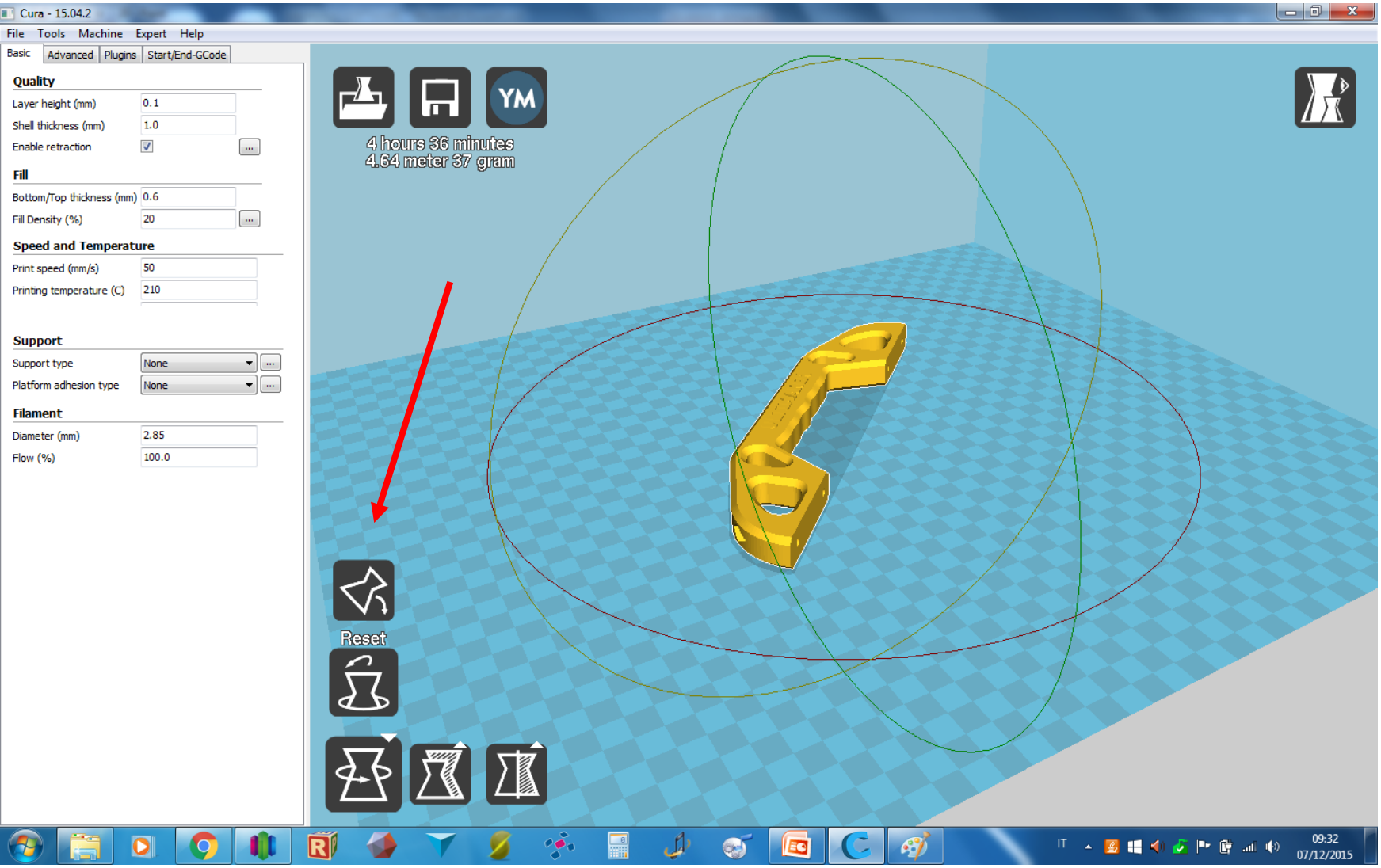
Choose and prepare the object - 4



Clicking on the object, three icons appear at the bottom left.

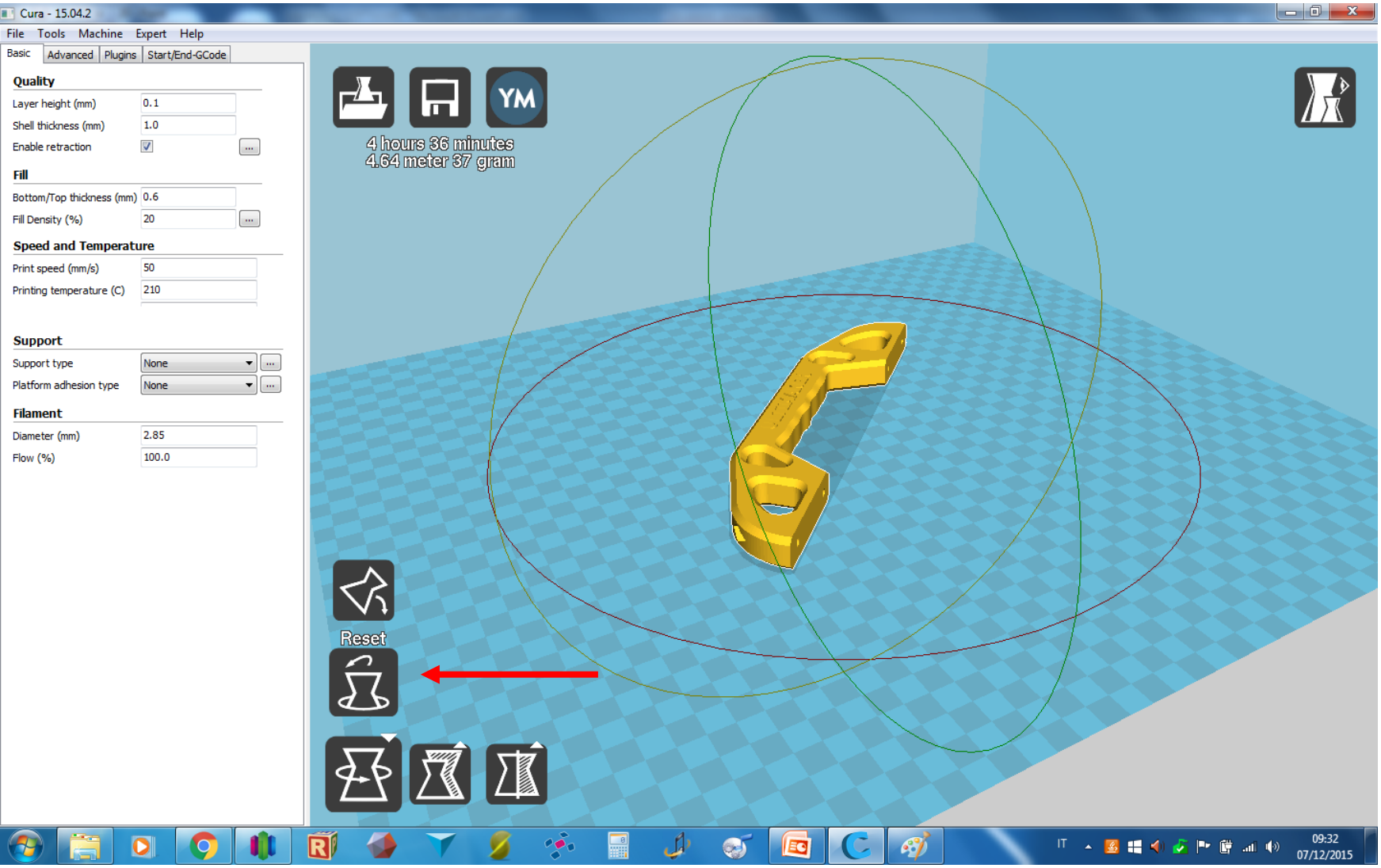
The icon 'Rotate' enables to rotate the object on the printing bed by using the mouse. The rotation is of 15°
Hold down the shift key to rotate of 1°

Choose and prepare the object - 5



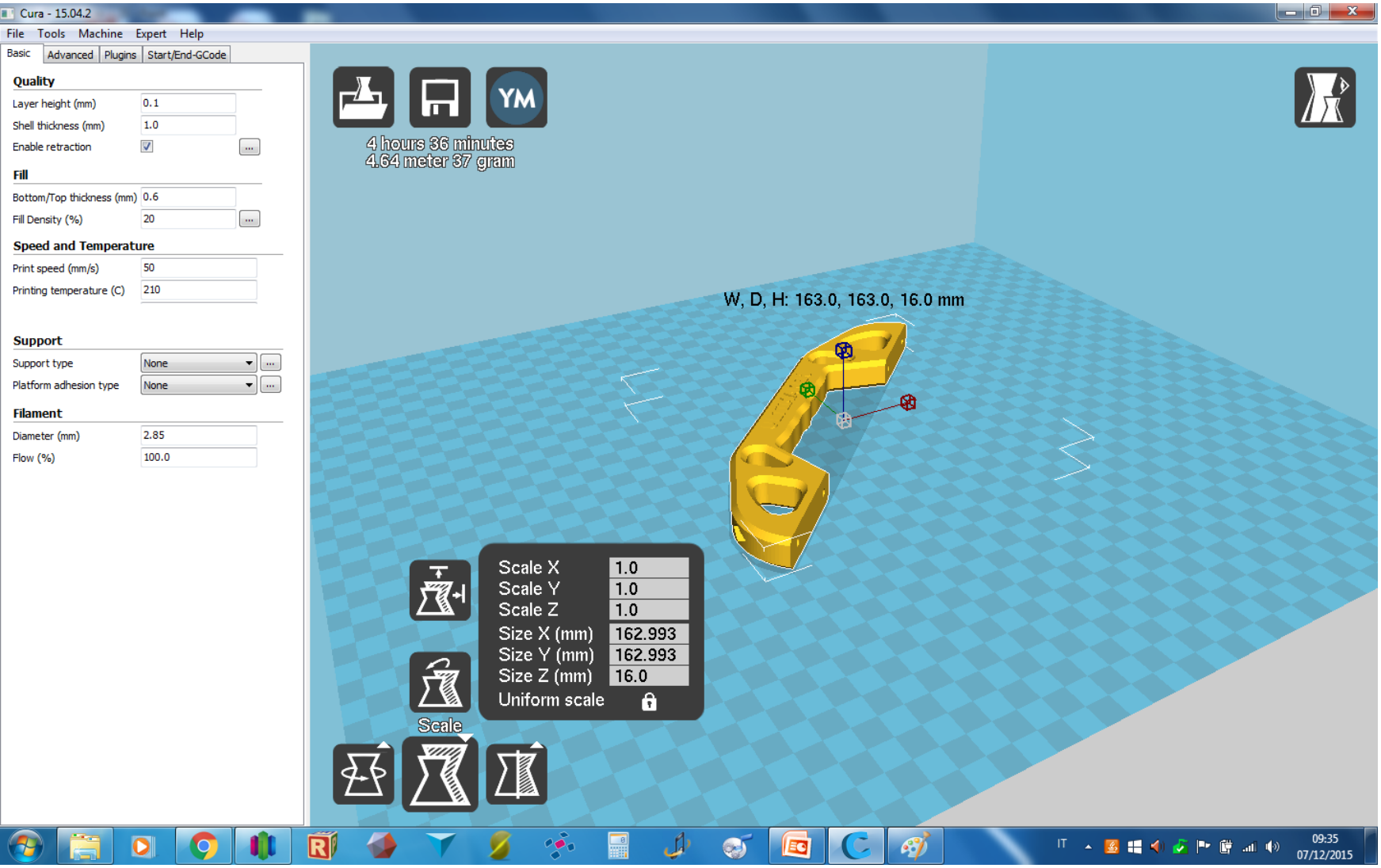
The "Lay Flat" option allows to change the object position and choose the side of the object that stands on the printing bed.

Choose and prepare the object - 6



The "Reset" option allows to reset changes and start over.

Choose and prepare the object - 7

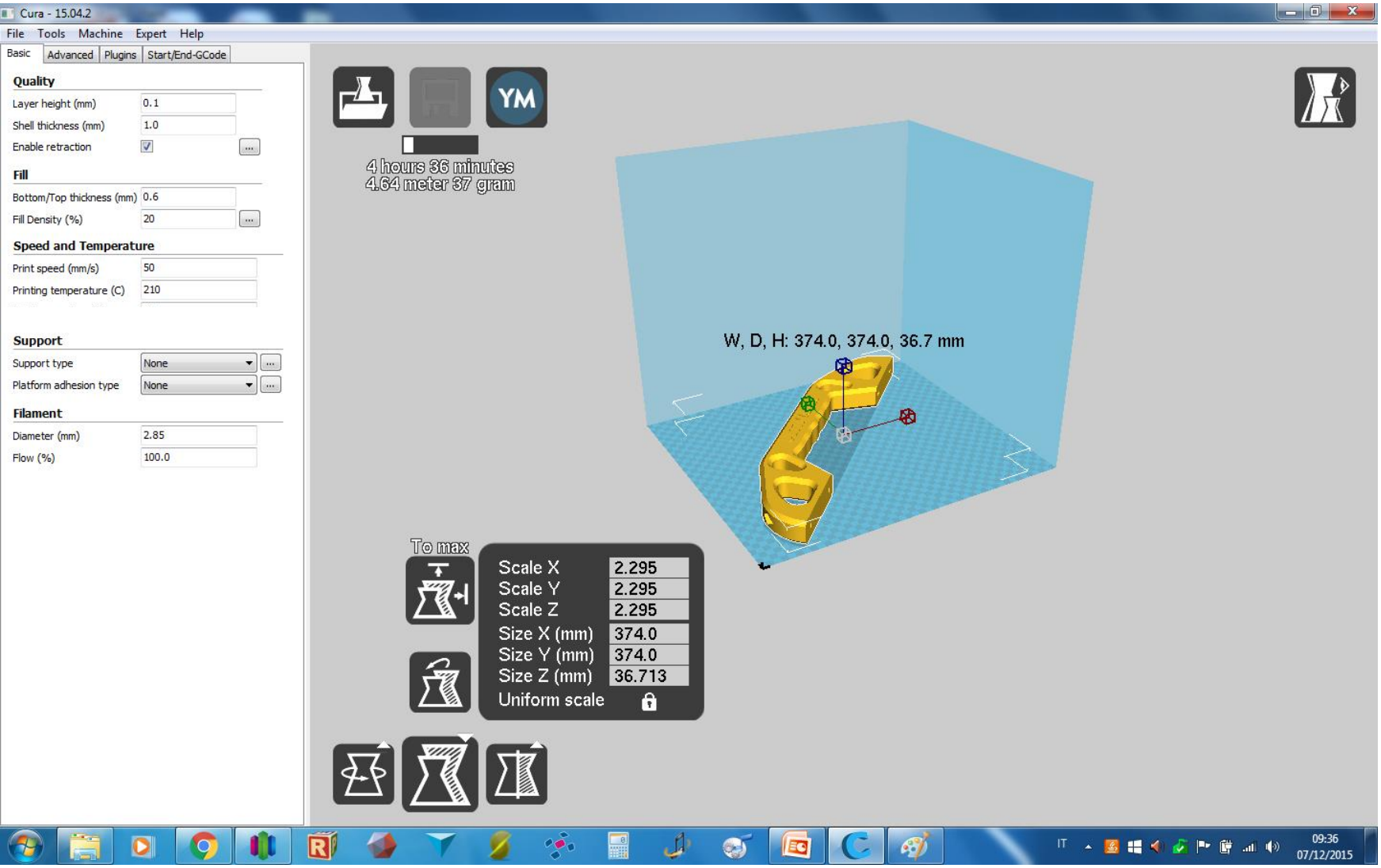


The "Scale" icon allows to modify the object size.

Scaling can be uniform or single axis.

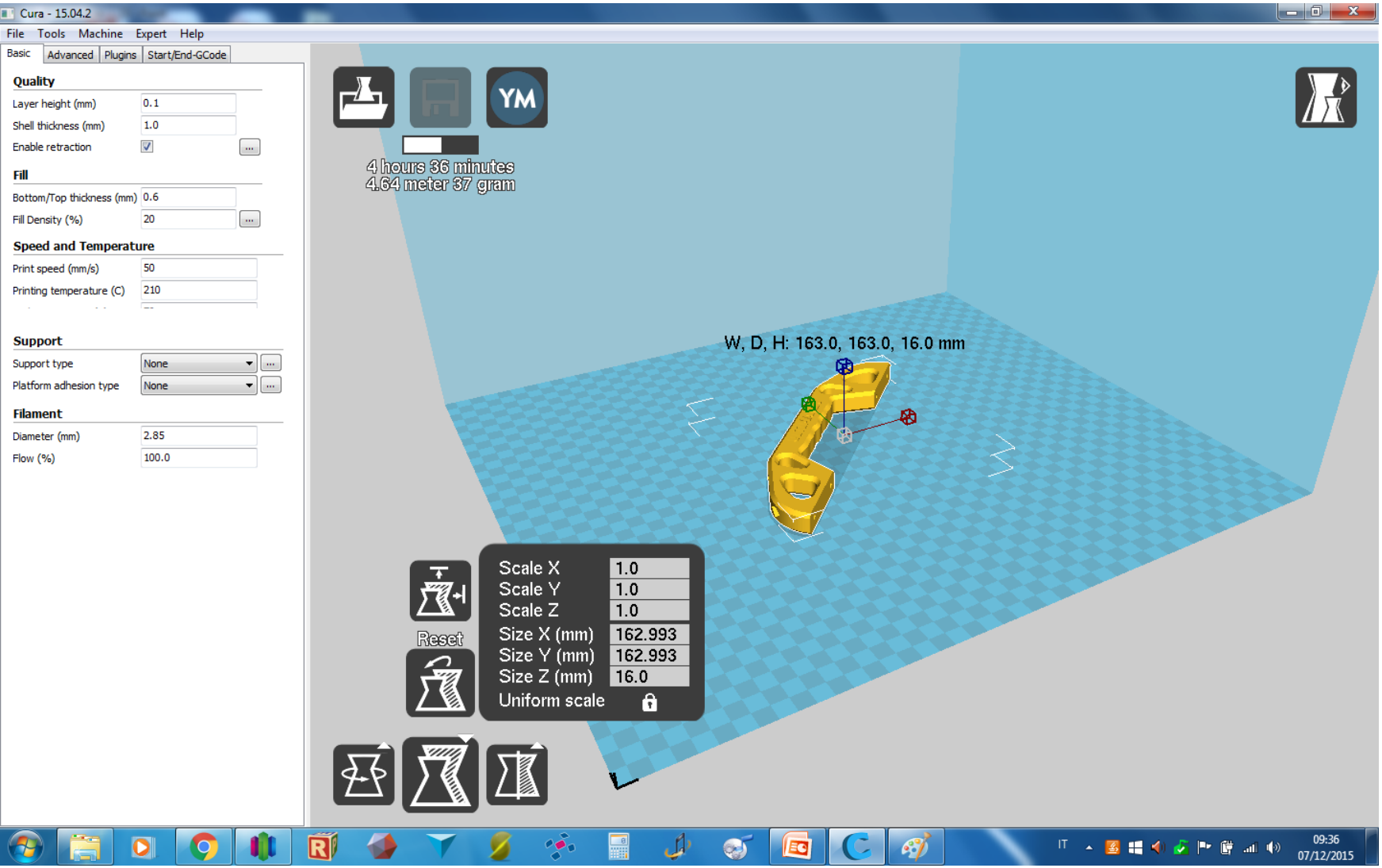
WARNING: excessively increasing the scale can lower the resolution and deteriorate print quality.

Choose and prepare the object - 8



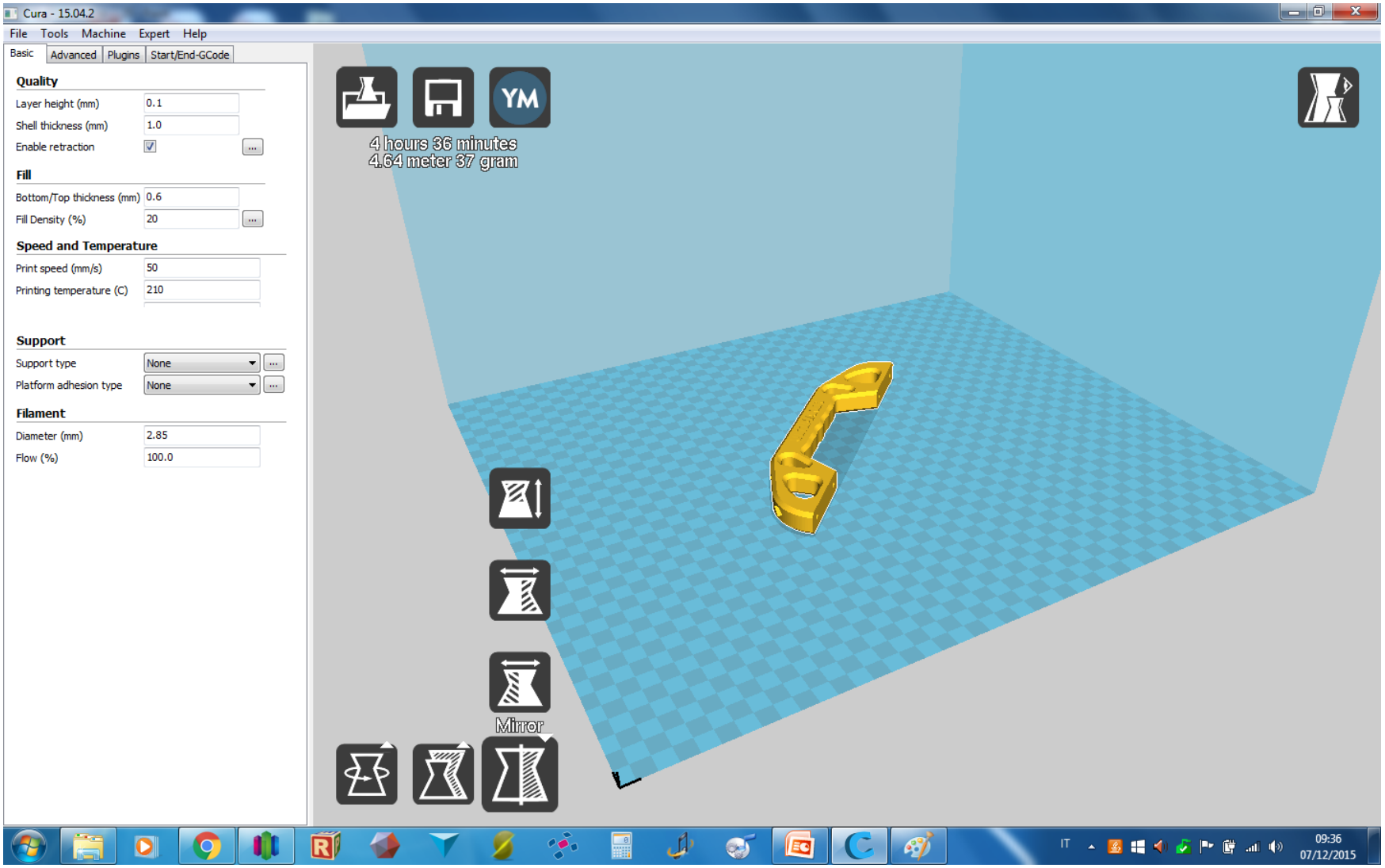
The "To Max" option allows to increase the object size up to the maximum print size.

Choose and prepare the object - 9



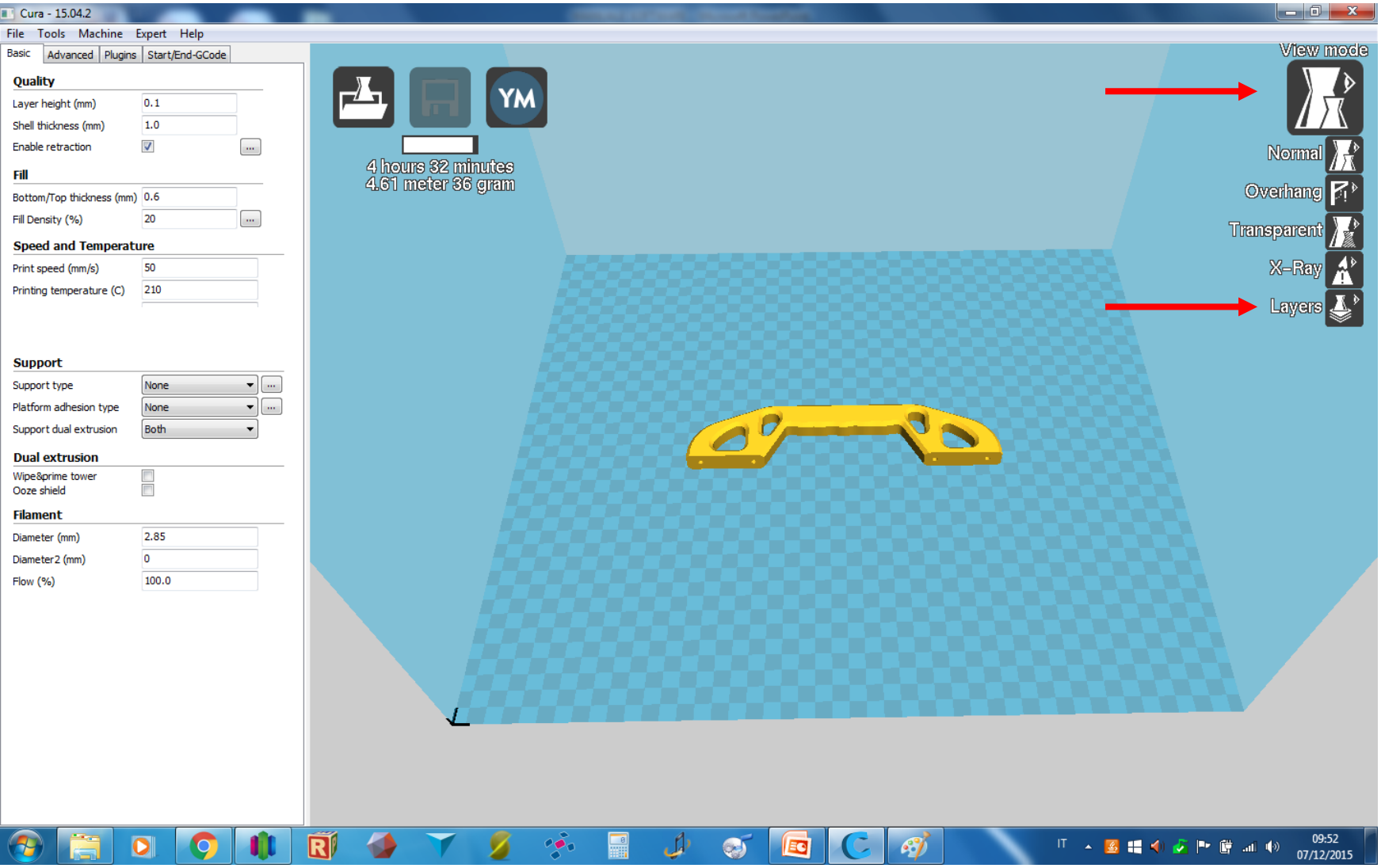
The "Reset" option allows to reset changes and start over.

Choose and prepare the object - 10



The "Mirror" option allows to *mirror* the object in different positions.

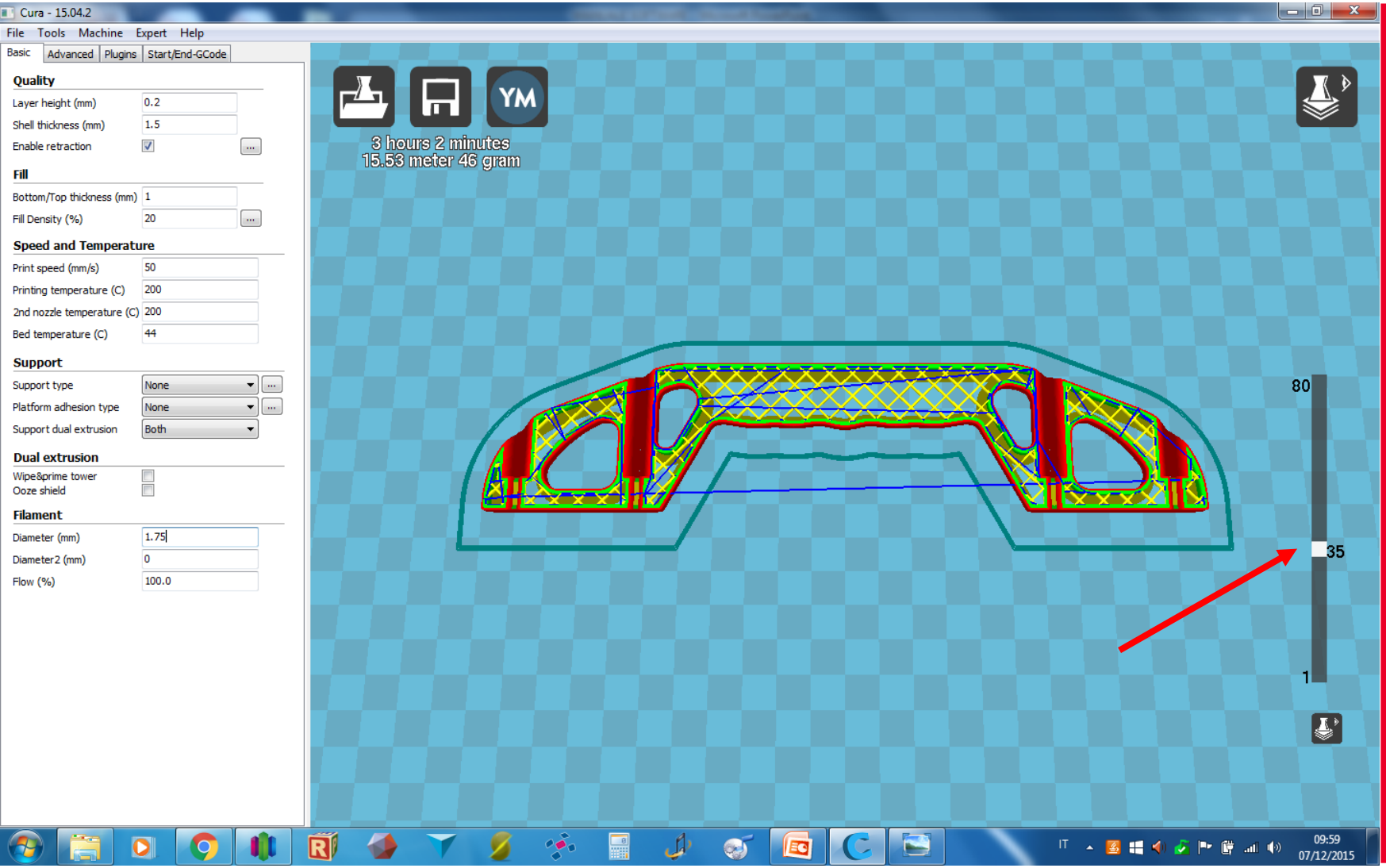
View Mode - 1



Click on "View Mode" icon to open the related Menu.

Select "Layers" to access the mode, that allows to view the object growth and to set all the printing parameters

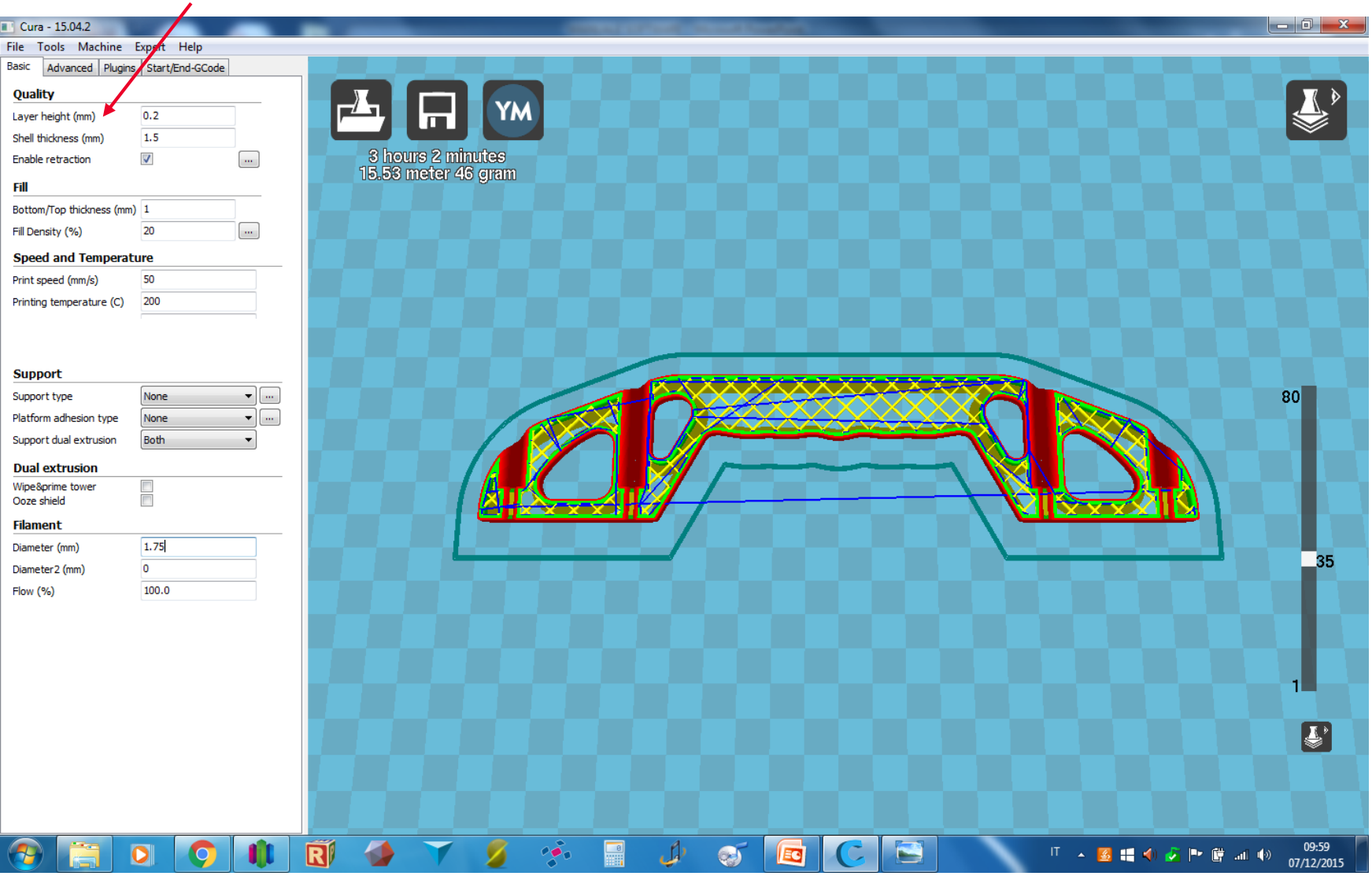
View Mode - 2



It's advisable to work on Layers mode to enter the printing parameters, as they are displayed in an optimal way.

Use the shift key and the arrow keys on the numeric string to see each layer and therefore progressive growth of the object.

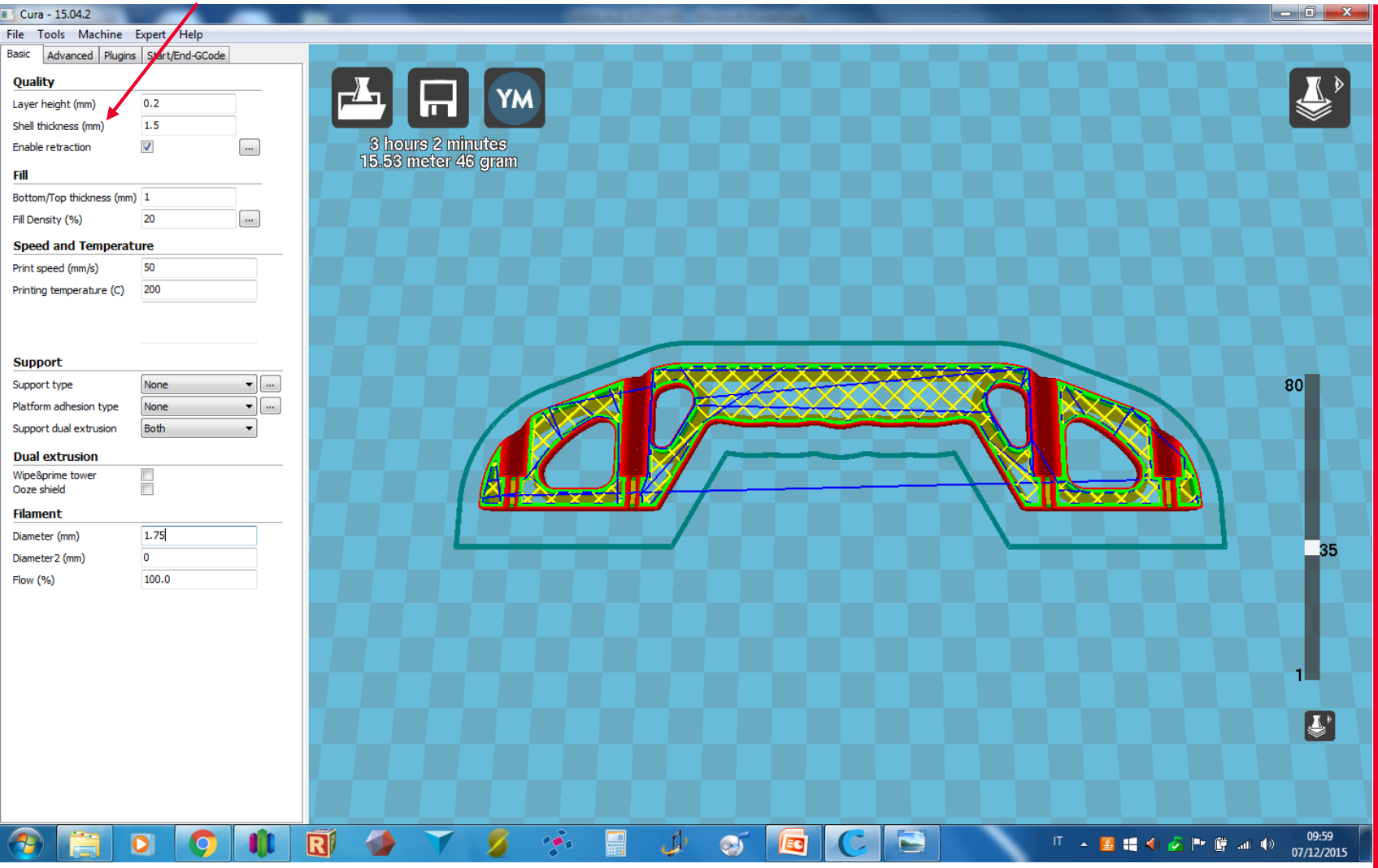
Print Setup - Basic/Quality section - 1



Layer height corresponds to the height of the layers that make up the object.

Thinner layers enable higher resolution, but require a longer printing time.

Print Setup - Basic/Quality section - 2



Shell thickness

This parameter sets the object perimeter thickness.

The shell setting is a function of the nozzle diameter.

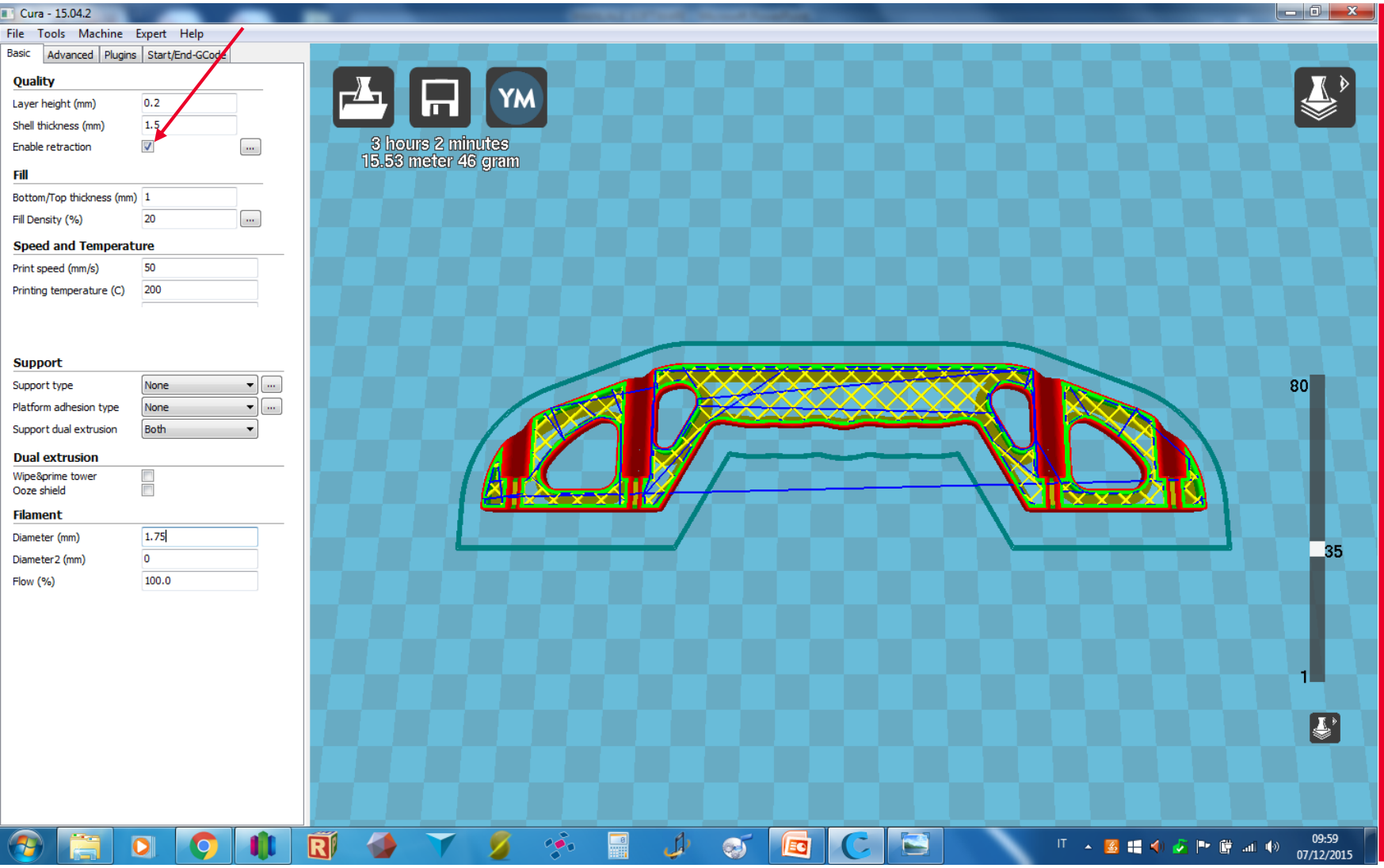
With a 0,4 mm nozzle diameter:

- 1 perimeter → 0,4
- 2 perimeters → 0,8
- 3 perimeters → 1,2

So the entered data must be a multiple of the nozzle diameter.

The outer perimeter is shown in red; the internal perimeters in green.

Print Setup - Basic/Quality section - 3

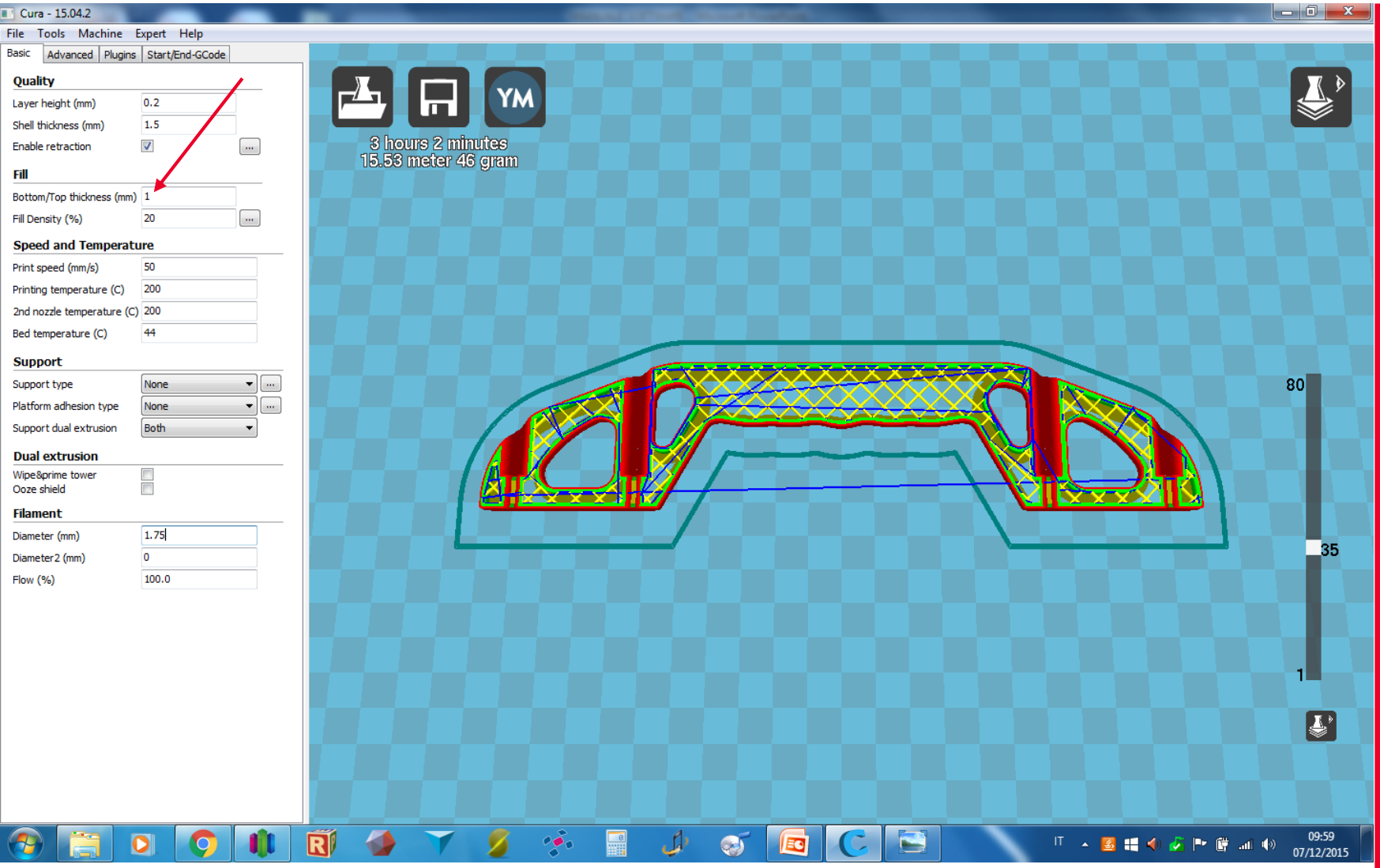


Enable retraction

It enables the material retraction.

<u>This parameter must be flagged</u> because it allows the control of material retraction during the printing.

Print Setup - Basic/Fill section - 1



Bottom/Top thickness

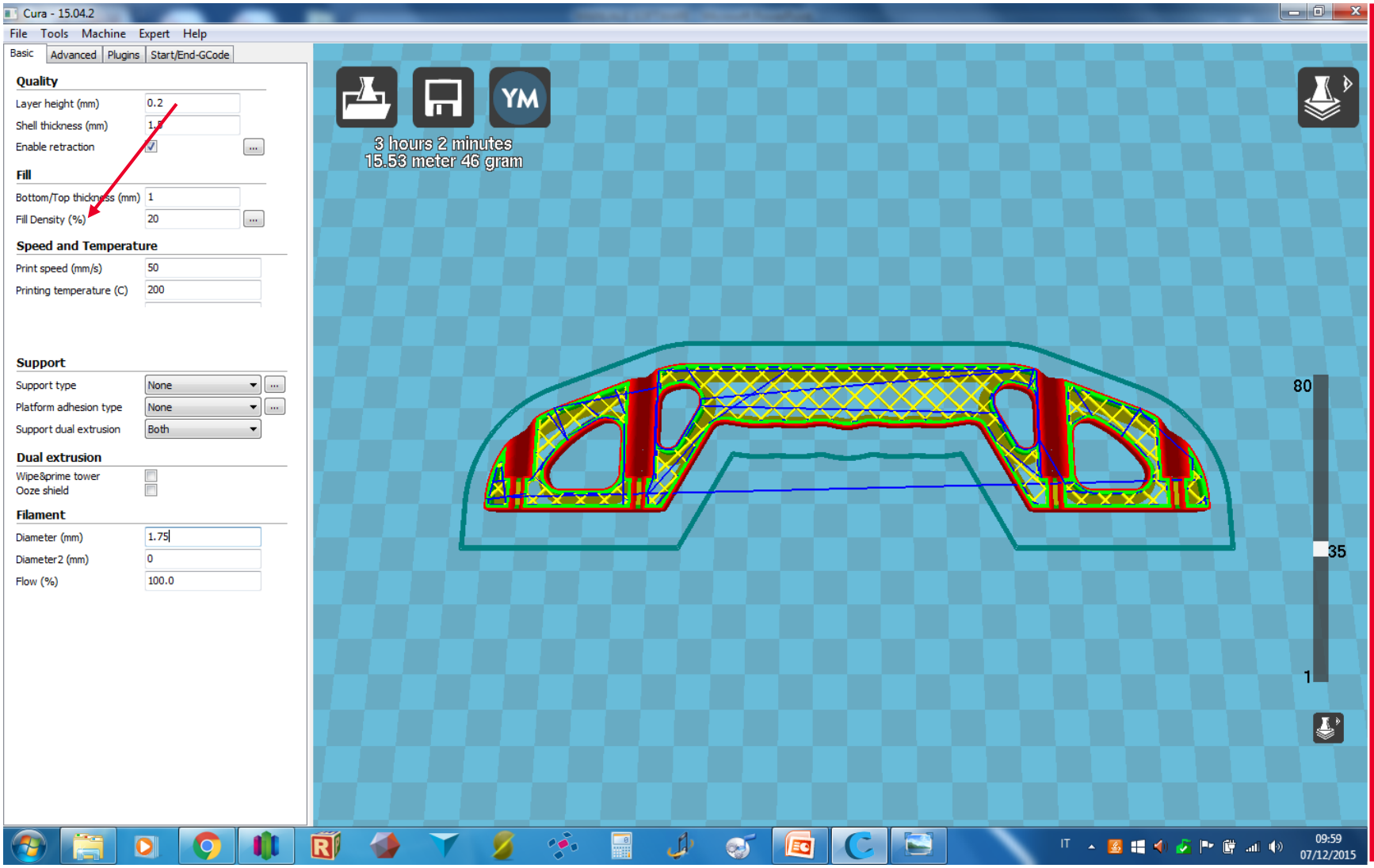
This parameter is related to bottom and top faces of the object.

This parameter has to be set as a function of the layer thickness. With layer at 0.2 mm:

- 1 lower and upper layer: 0.2
- 2 lower and upper layers: 0.4
- 3 lower and upper layers: 0.6

Then the entered data must be a multiple of the layer thickness.

Print Setup - Basic/Fill section - 2



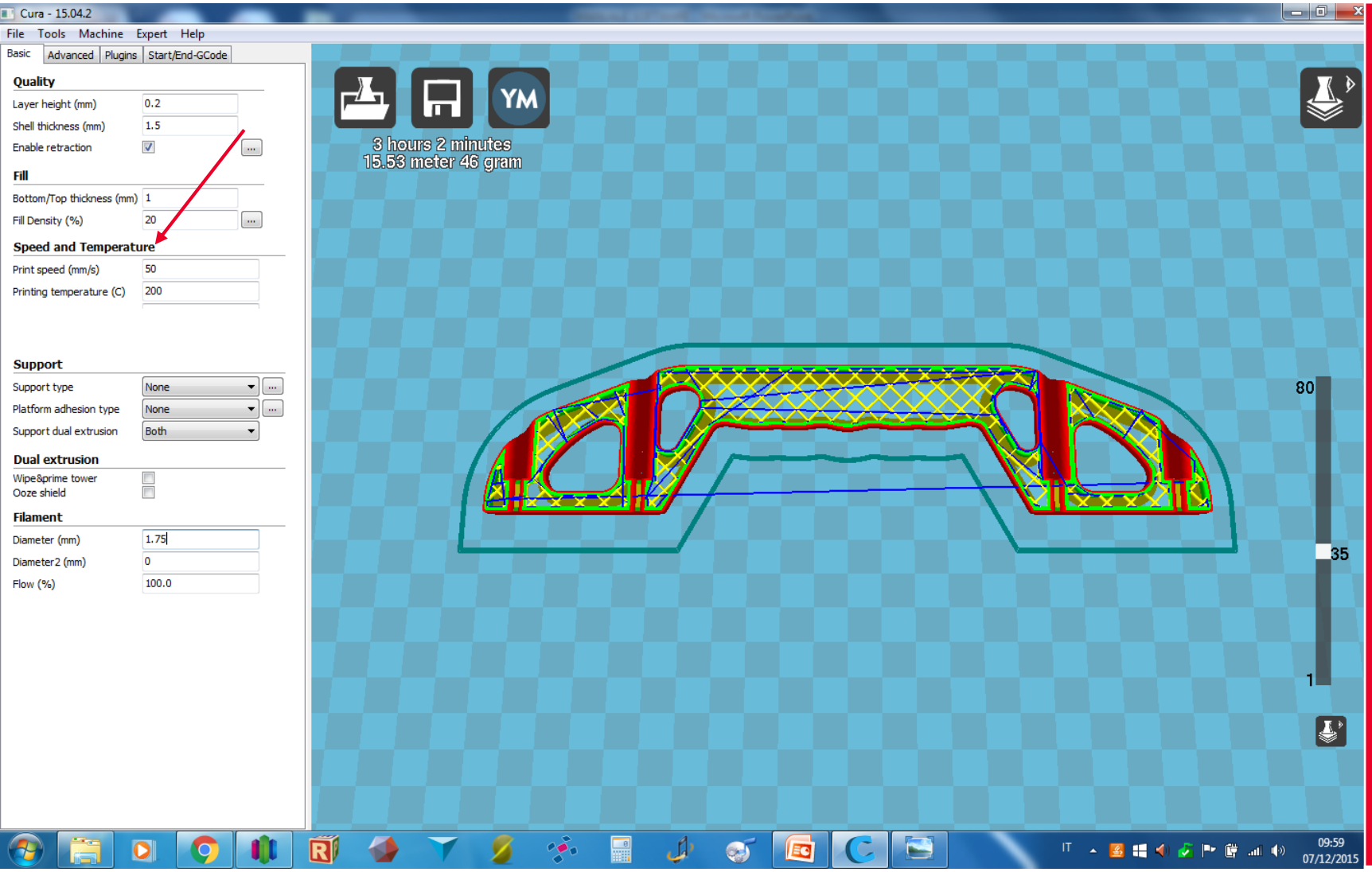
Fill density

It corresponds to the fill percentage of the internal parts of the object.

With the increase of filling percentage, the consistency of the object enhances.

- **hollow object: fill density 0%**
- **partially filled object: Fill density 1%/99%**
- **Totally filled object: Fill density 100%**

Print Setup - Basic/Speed and Temperature section



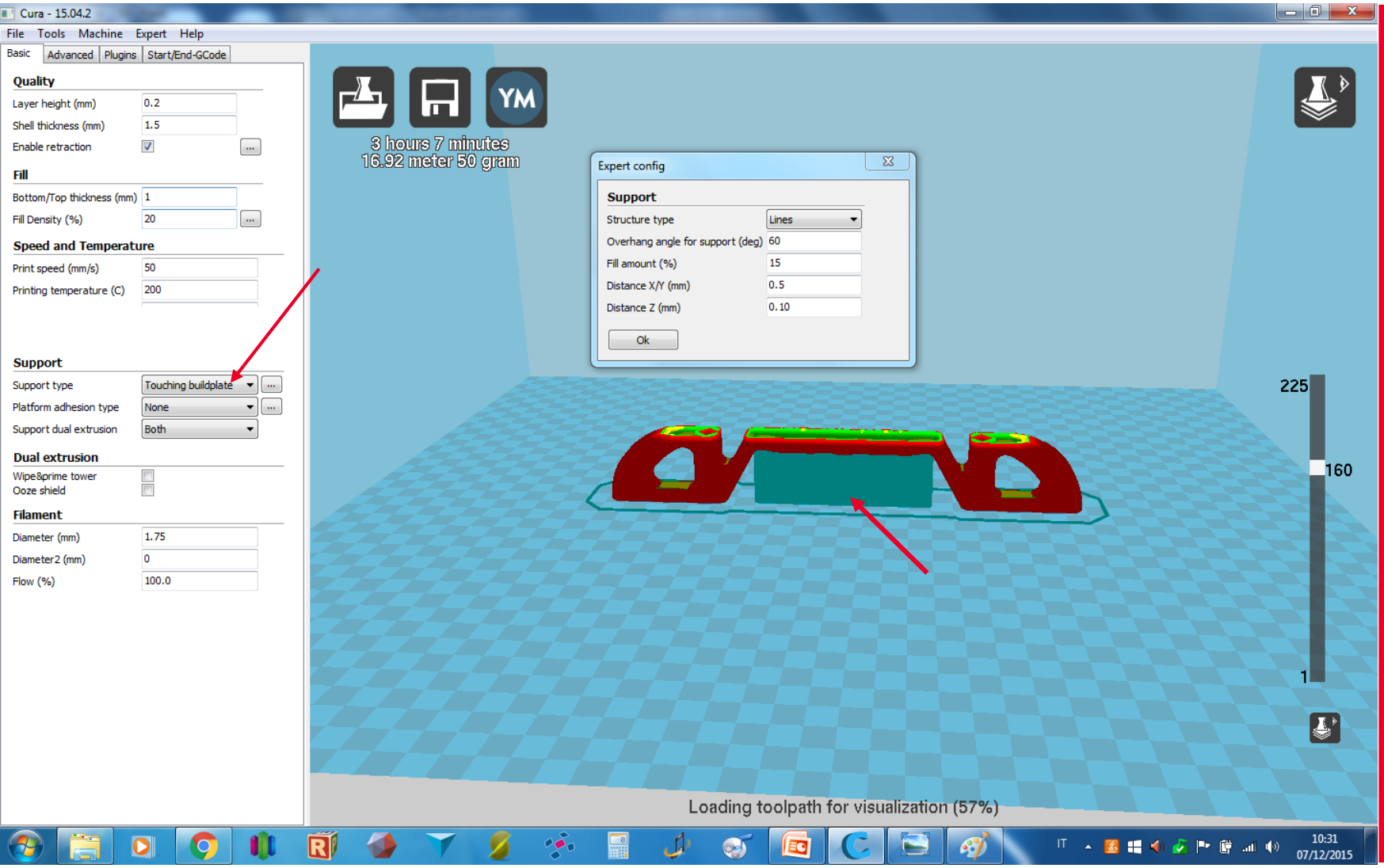
Print Speed

Basic print speed (mm/sec) to be set for printing.

Printing Temperature

Nozzle temperature, to be set according to the material characteristics.

Print Setup - Basic/Support section - 1



Support Type

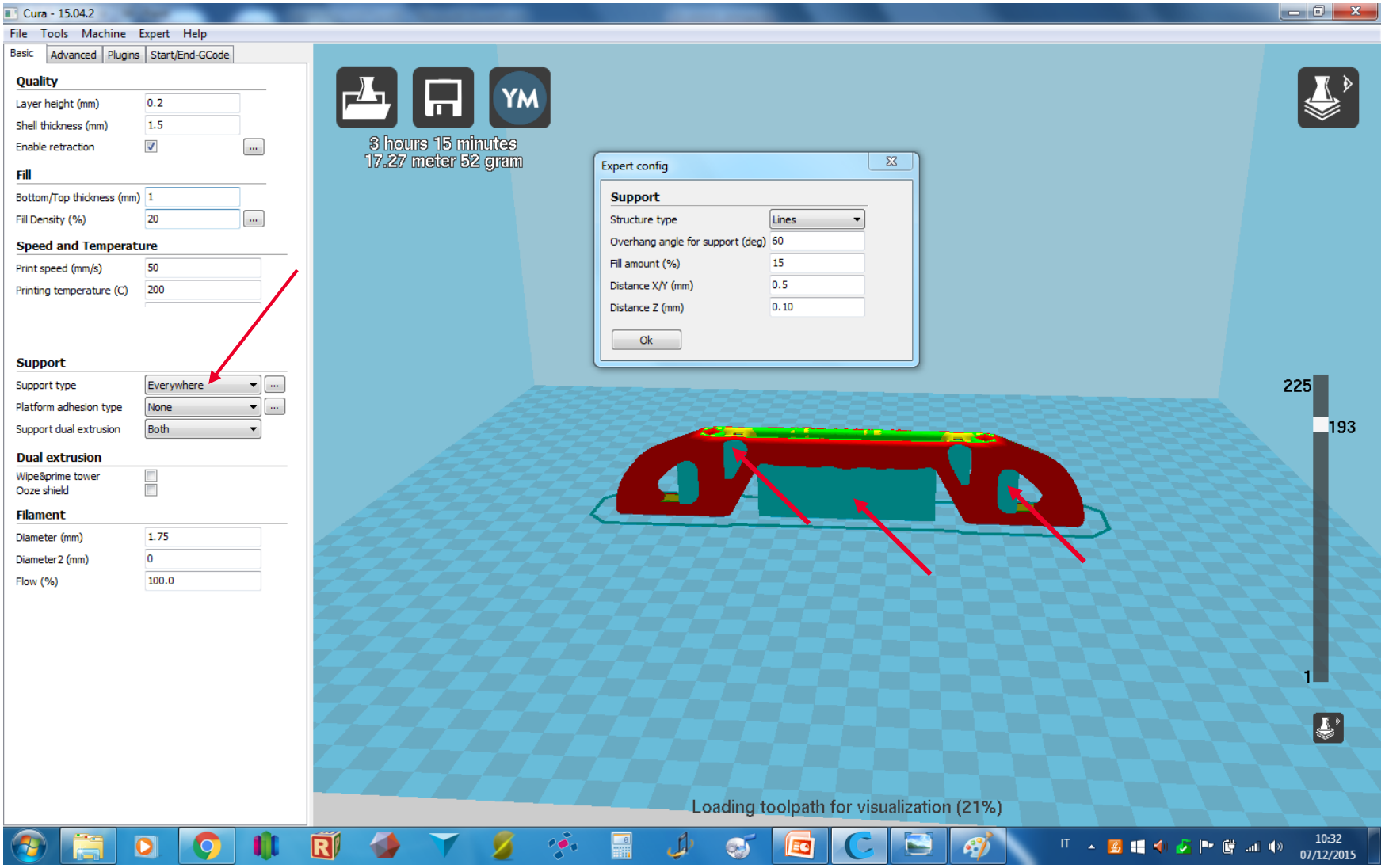
When the object needs a support because the shape has some significant overhang, you can select two options:

OPTION 1

Touching Buildplate

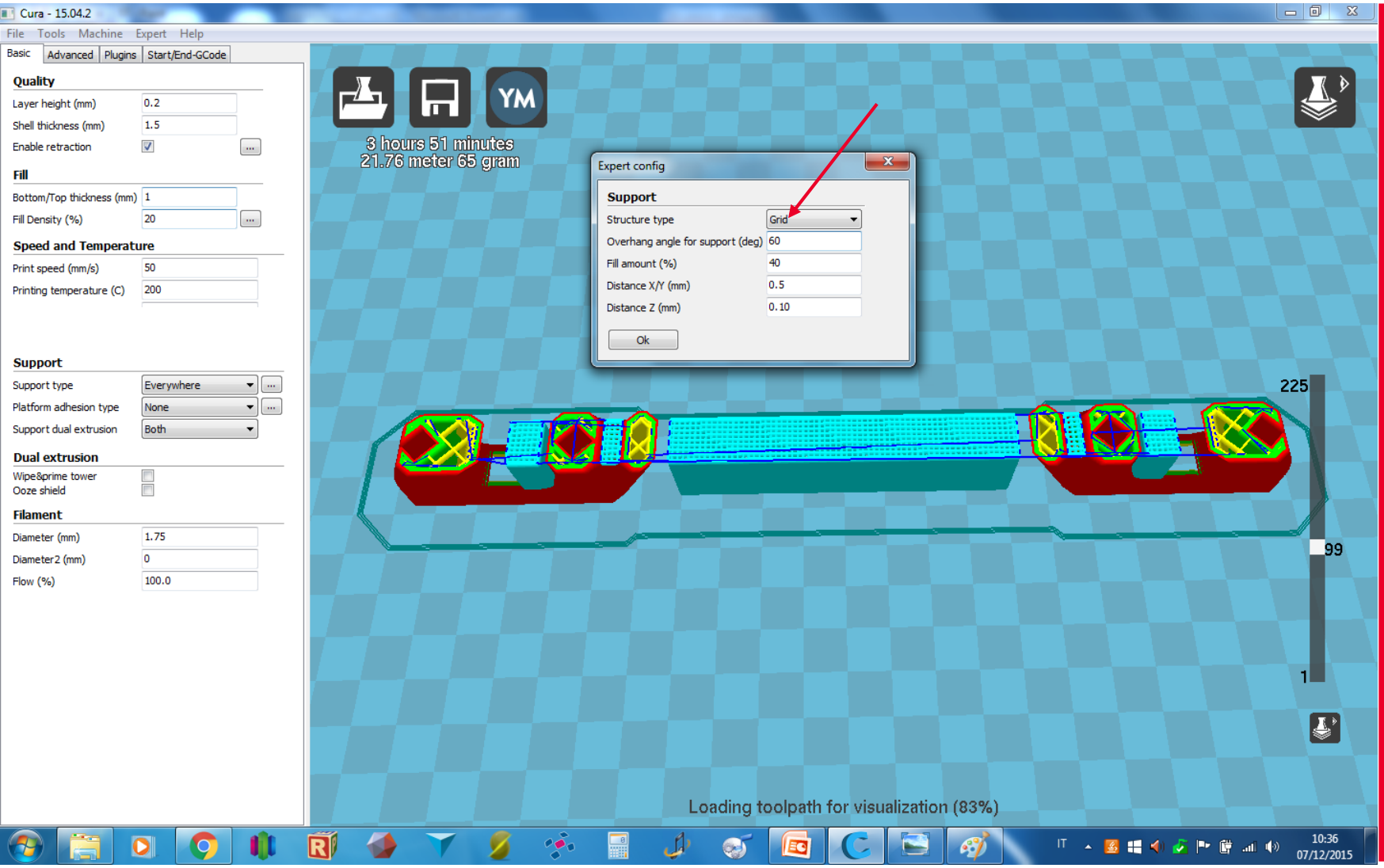
The software creates the support only where it is in contact with the printing bed

Print Setup - Basic/Support section - 2



OPTION 2:
***Everywhere*:**
It creates the support wherever there is any overhang

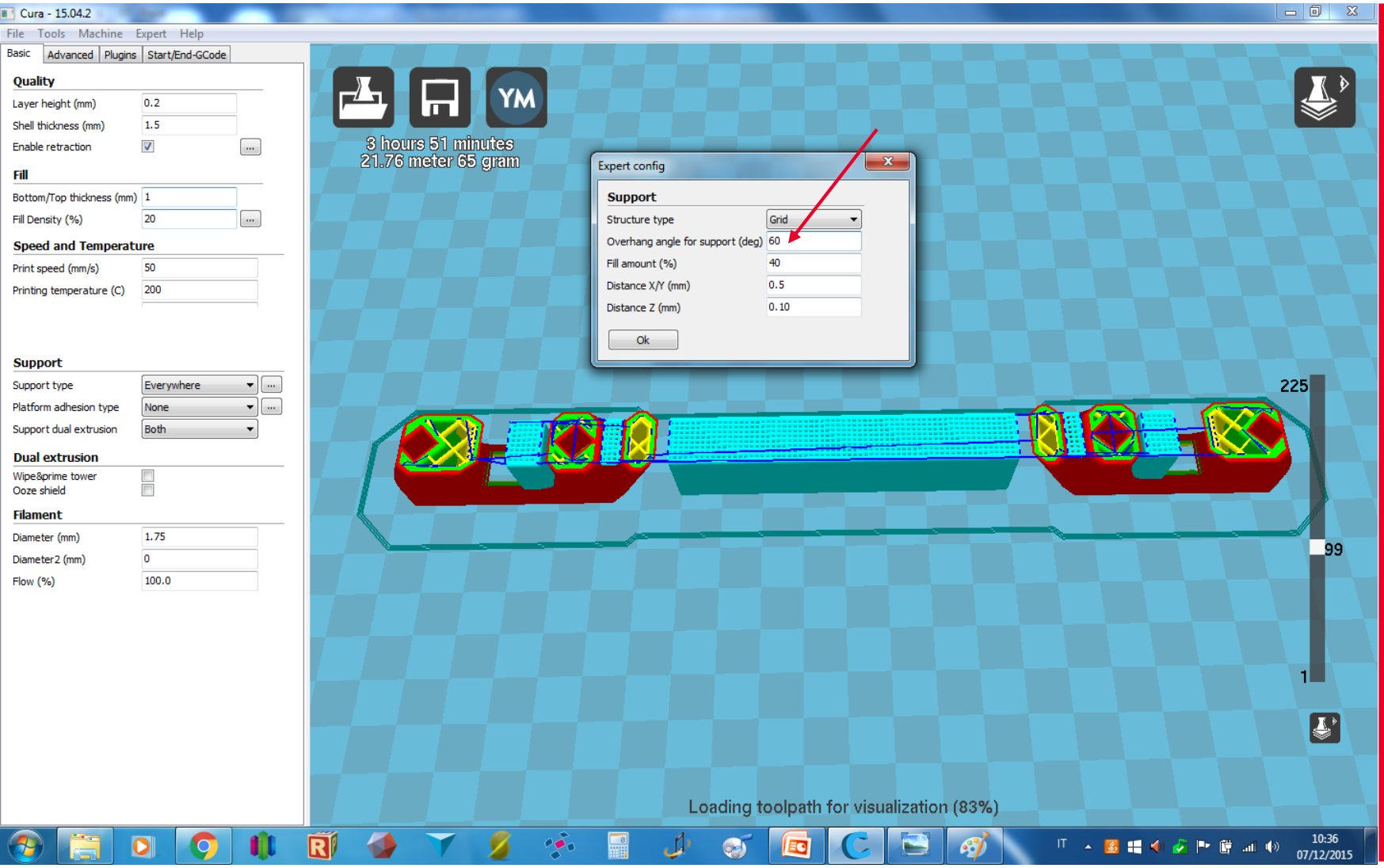
Print Setup - Basic/Support section - 3



At the line "Structure type" you can set whether the support should be linear (Lines) or grid (Grid).

It is advisable to choose "Lines", to facilitate the support material removal.

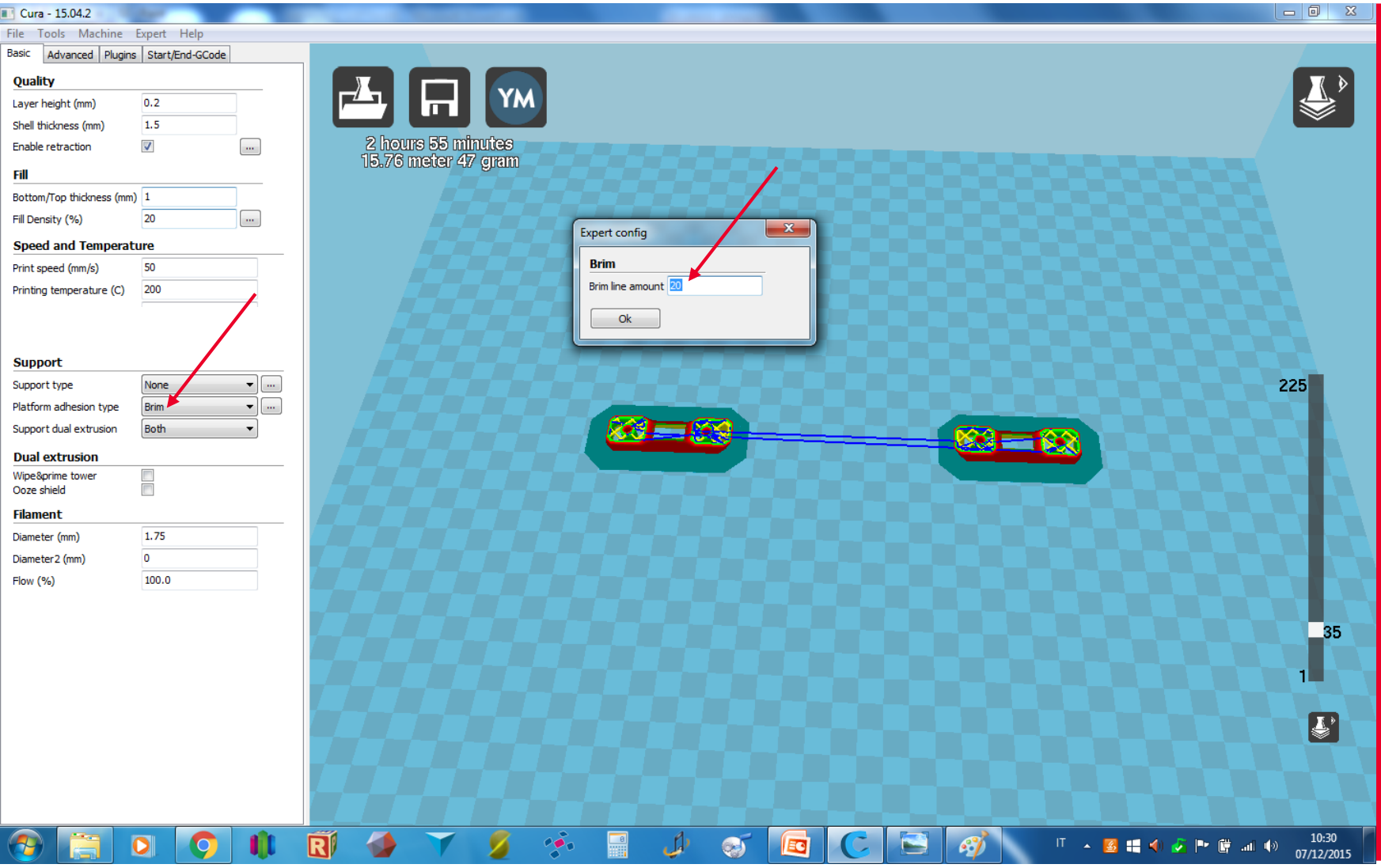
Print Setup - Basic/Support section - 4



At the line "Overhang angle for support", you can set the overhang angle from which the support will be generated.

It's advisable to keep this value at 60 °

Print Setup - Basic/Support section - 5

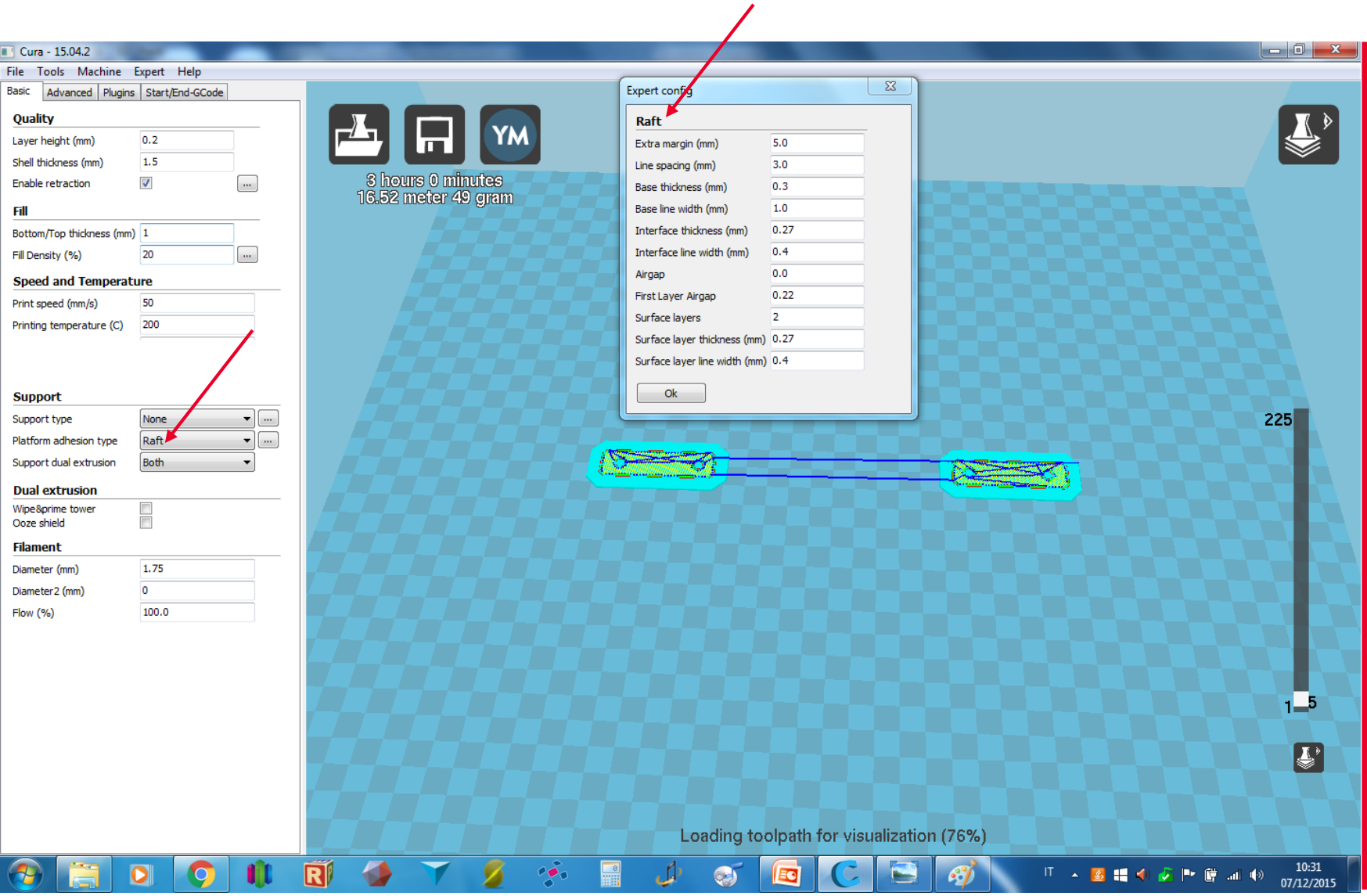


Platform Adesion Type

Two options:

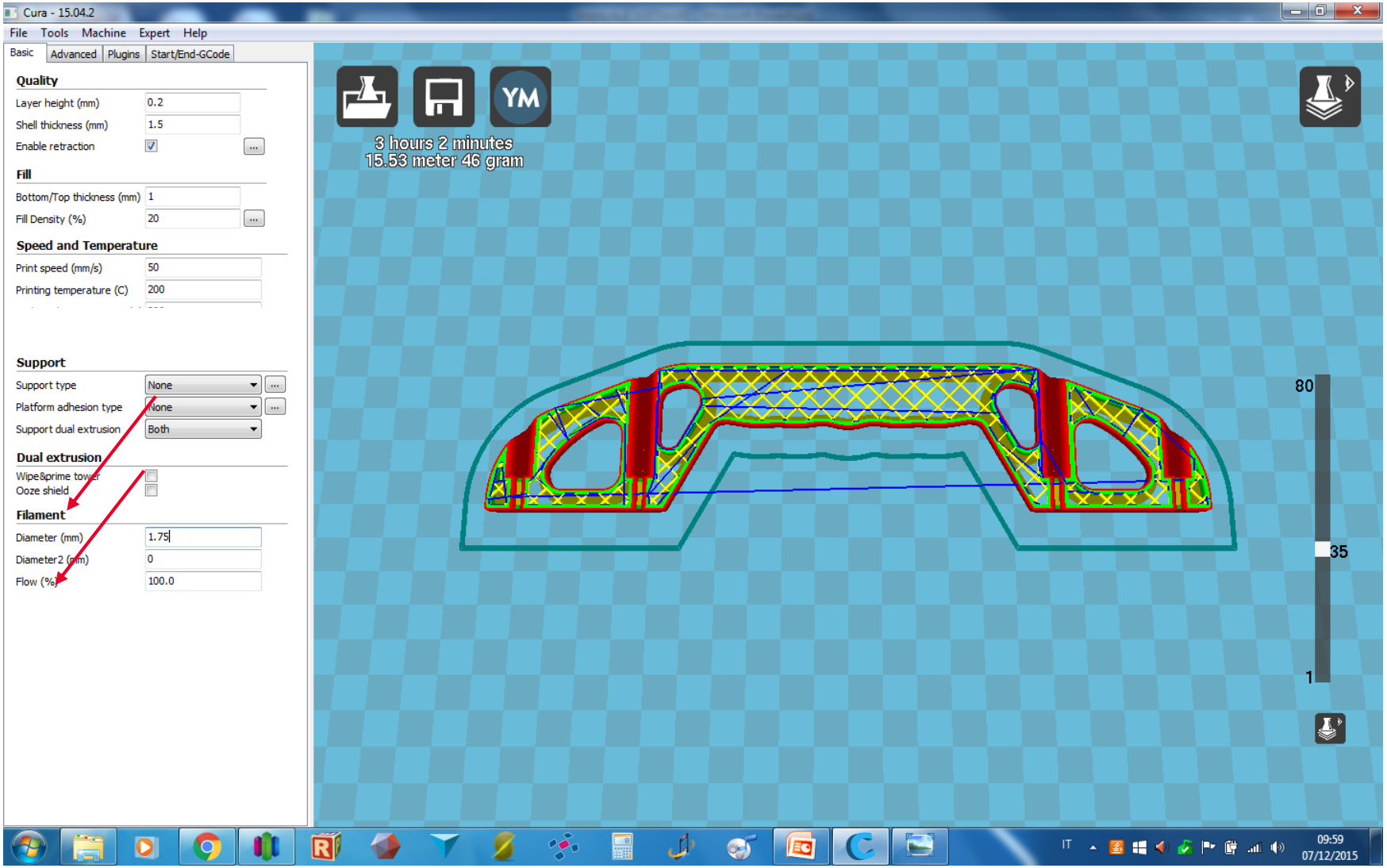
BRIM OPTION
This option allows to create a further basis (as shown in the figure), which will ensure better adhesion to the printing table. Opening the next dropdown you can define the width of the Brim (“Brim line amount”).

Print Setup - Basic/Support section - 6



RAFT OPTION
This function creates a basis of some layers under the object, which improve the adhesion to the printing bed.
This feature is especially used for objects with very small bases.
The config list allows to set the Raft, managing the specific parameters.

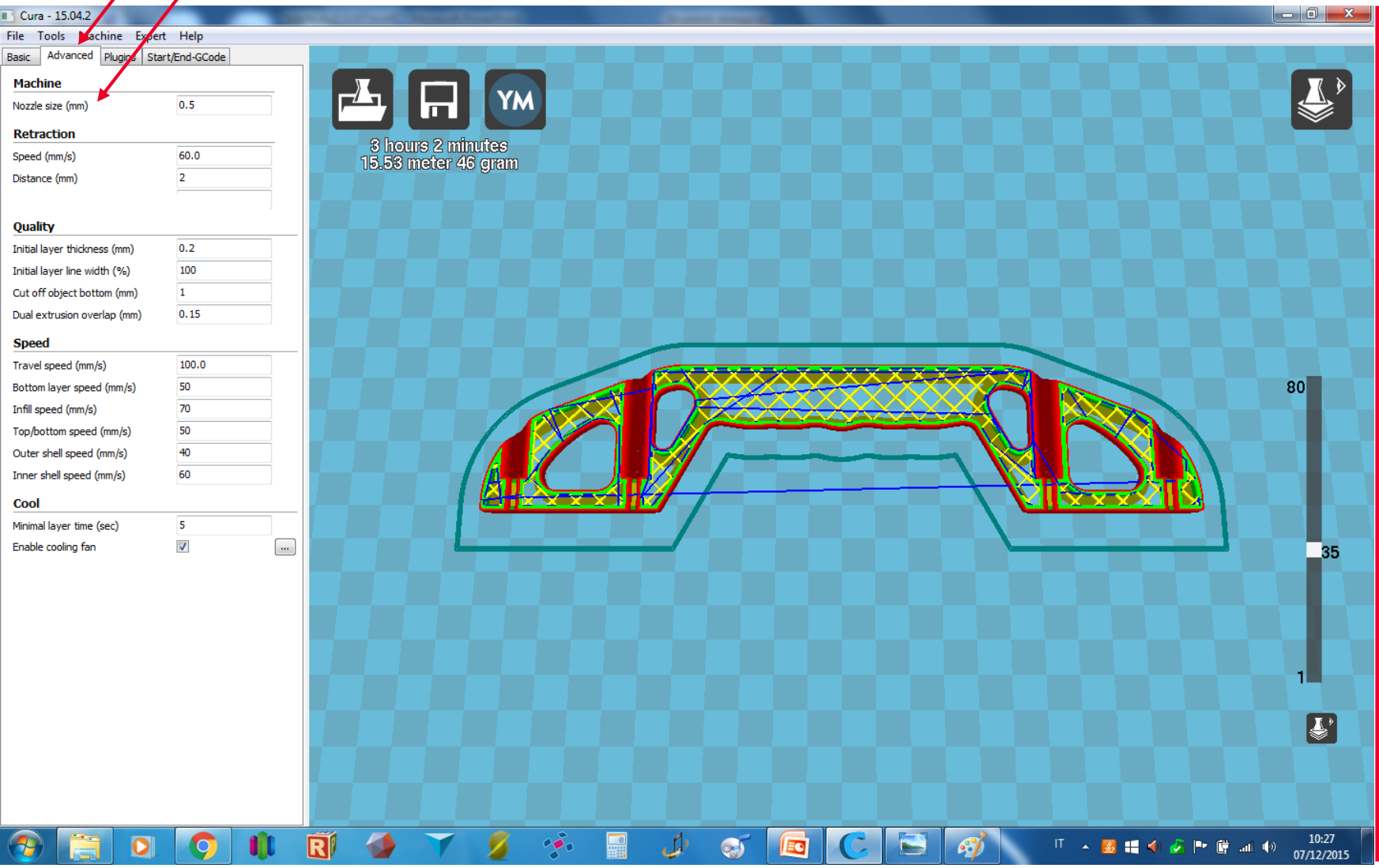
Print Setup - Basic/Filament section



Diameter
It indicates the filament diameter. The Olivetti 3D DESK works only with 1,75 mm filaments.

Flow
It's the material flow to be set for printing.
For not-experted users, it is advisable to keep the default value of 100%

Print Setup - Advanced/Machine section

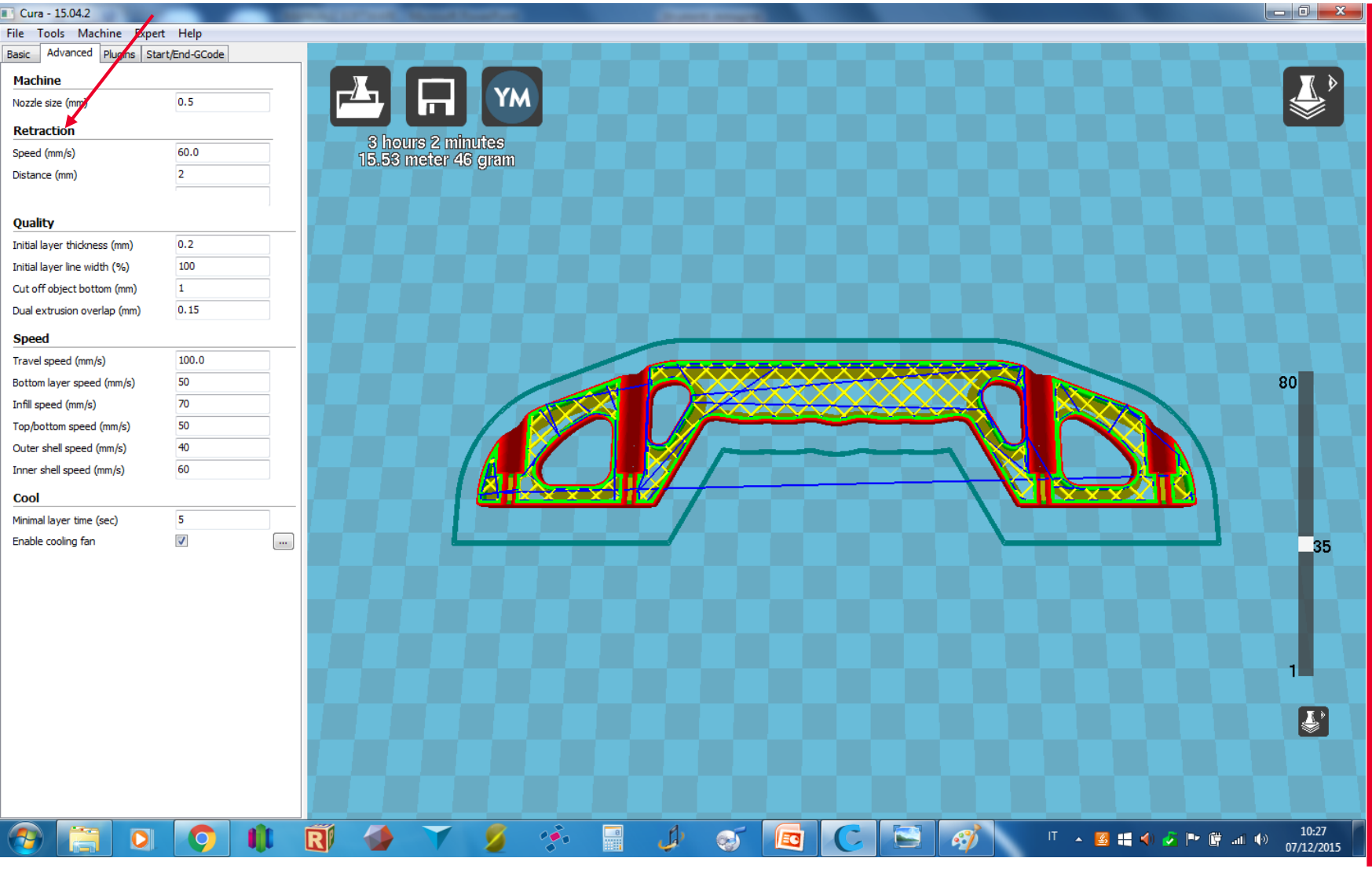


Select the ADVANCED Menu

Nozzle Size (nozzle hole diameter): enter the millimeter value of the mounted nozzle diameter.

On Olivetti 3D DESK it is mounted by default a nozzle of 0.4 mm.

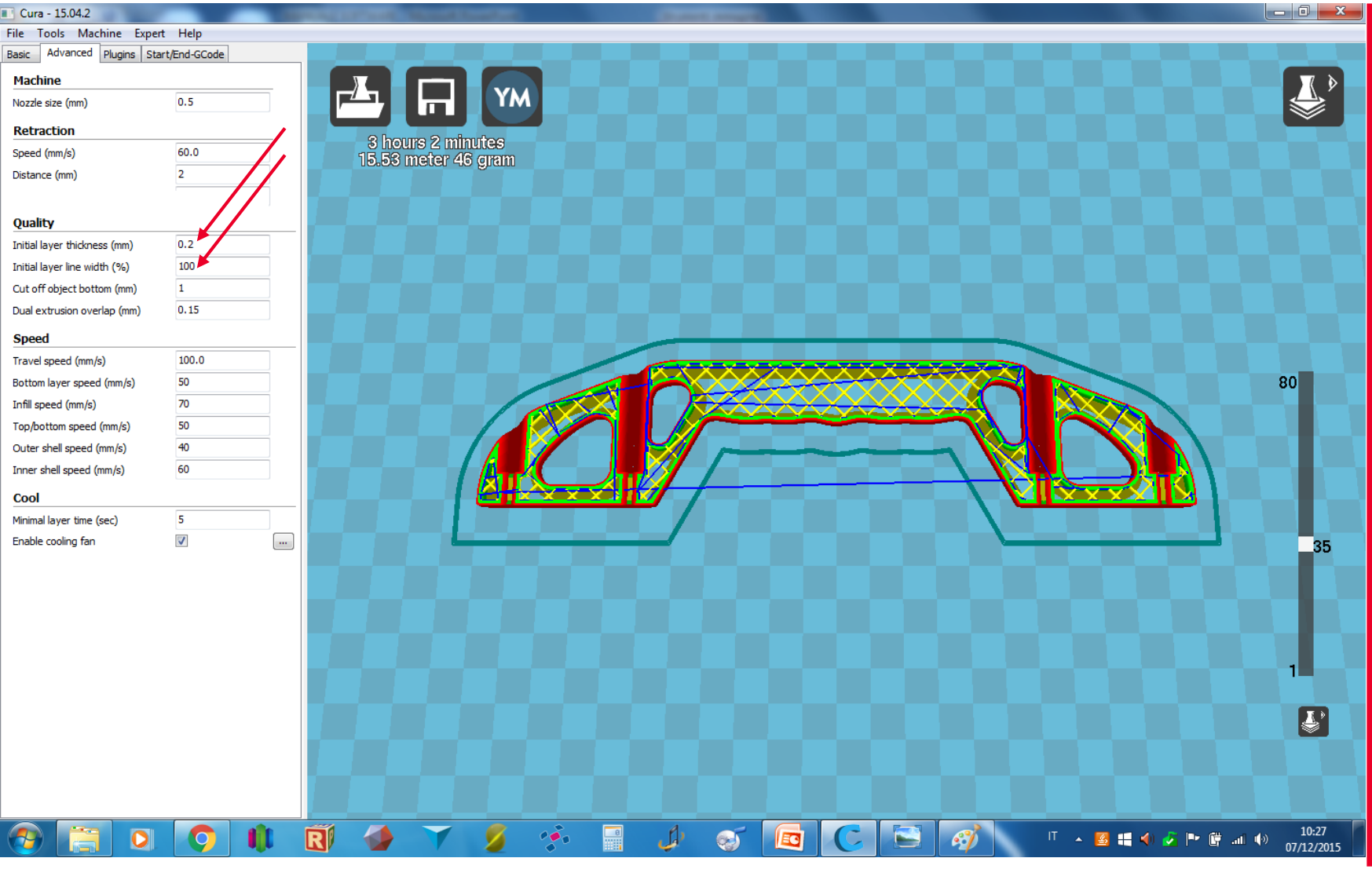
Print Setup - Advanced/Retraction section



Retraction

If the Retraction feature was flagged in the Basic section (see the slide n. 27), now it's possible to set the retraction Speed and Distance (ie quantity of retracted material).

Print Setup - Advanced/Quality section - 1



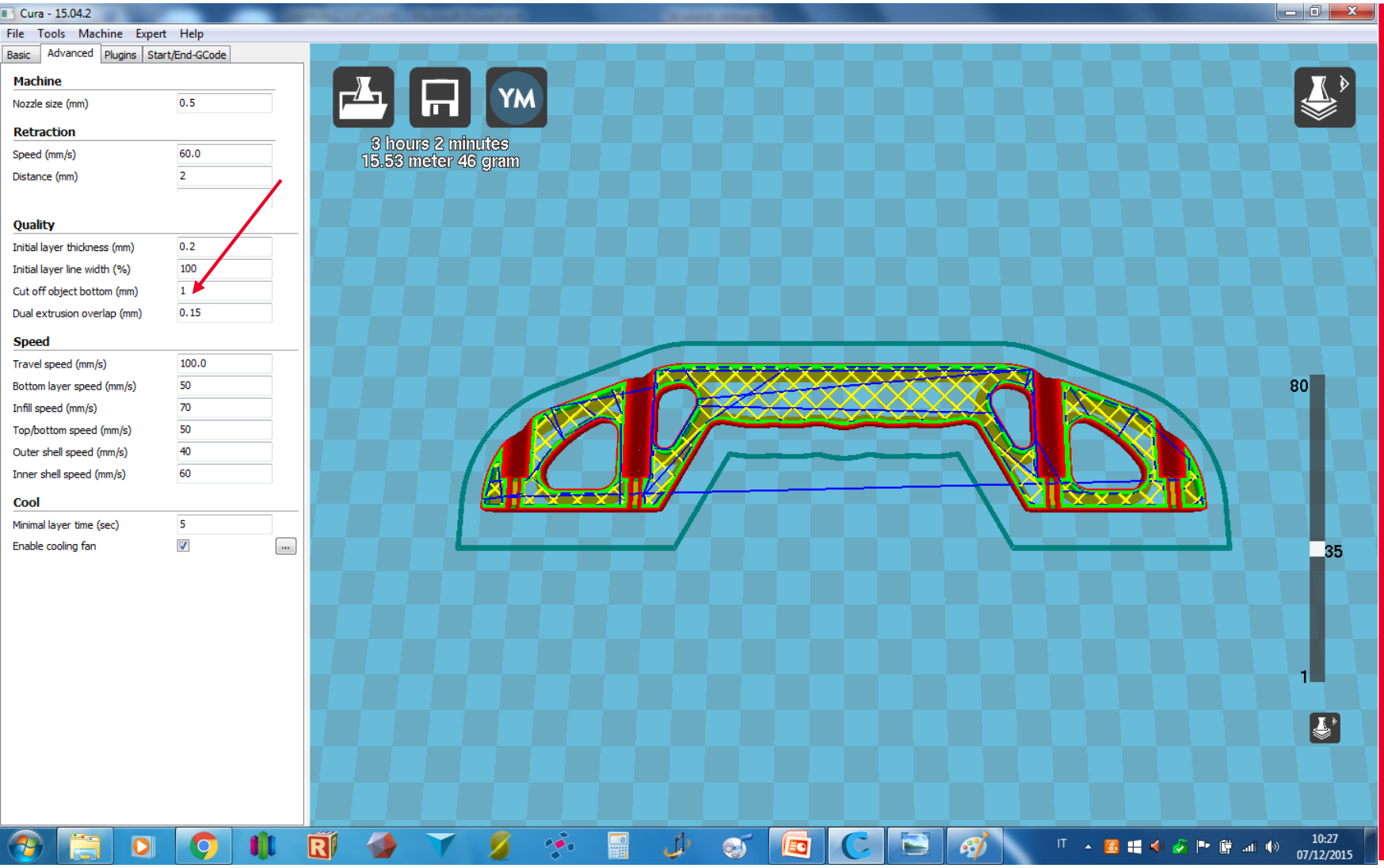
Initial Layer Thickness

It means the first layer height. It's advisable to keep the value at 0,2 mm.

Initial layer line width

This item indicates the line width for first layer. It's advisable to keep the value at 100%

Print Setup - Advanced/Quality section - 2

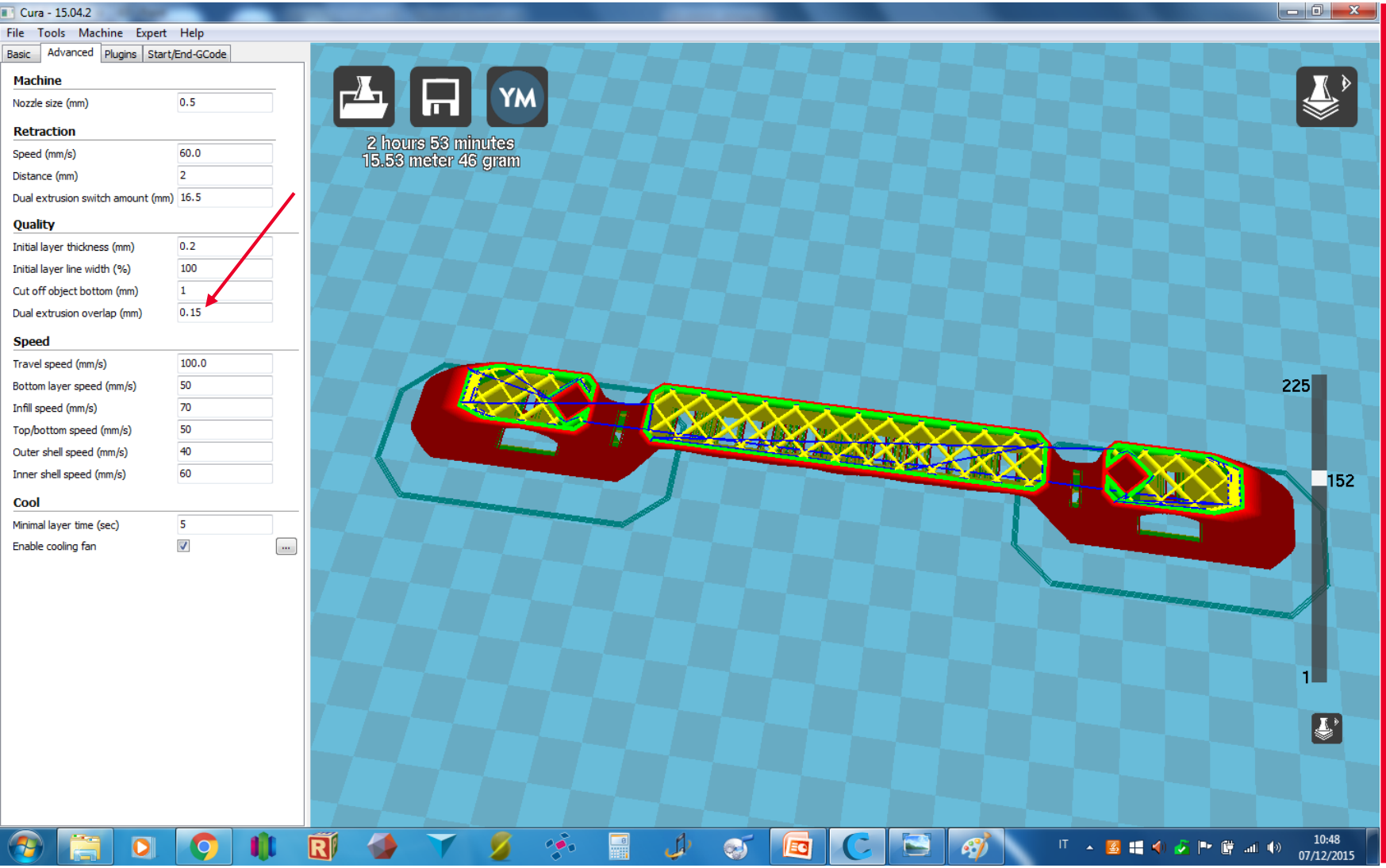


Cut Off object

It allows to cut the object, starting from the printing bed.

This feature is useful especially to check the print quality of the top section of the object.
Cutting off the object, printing starts directly at the height set.

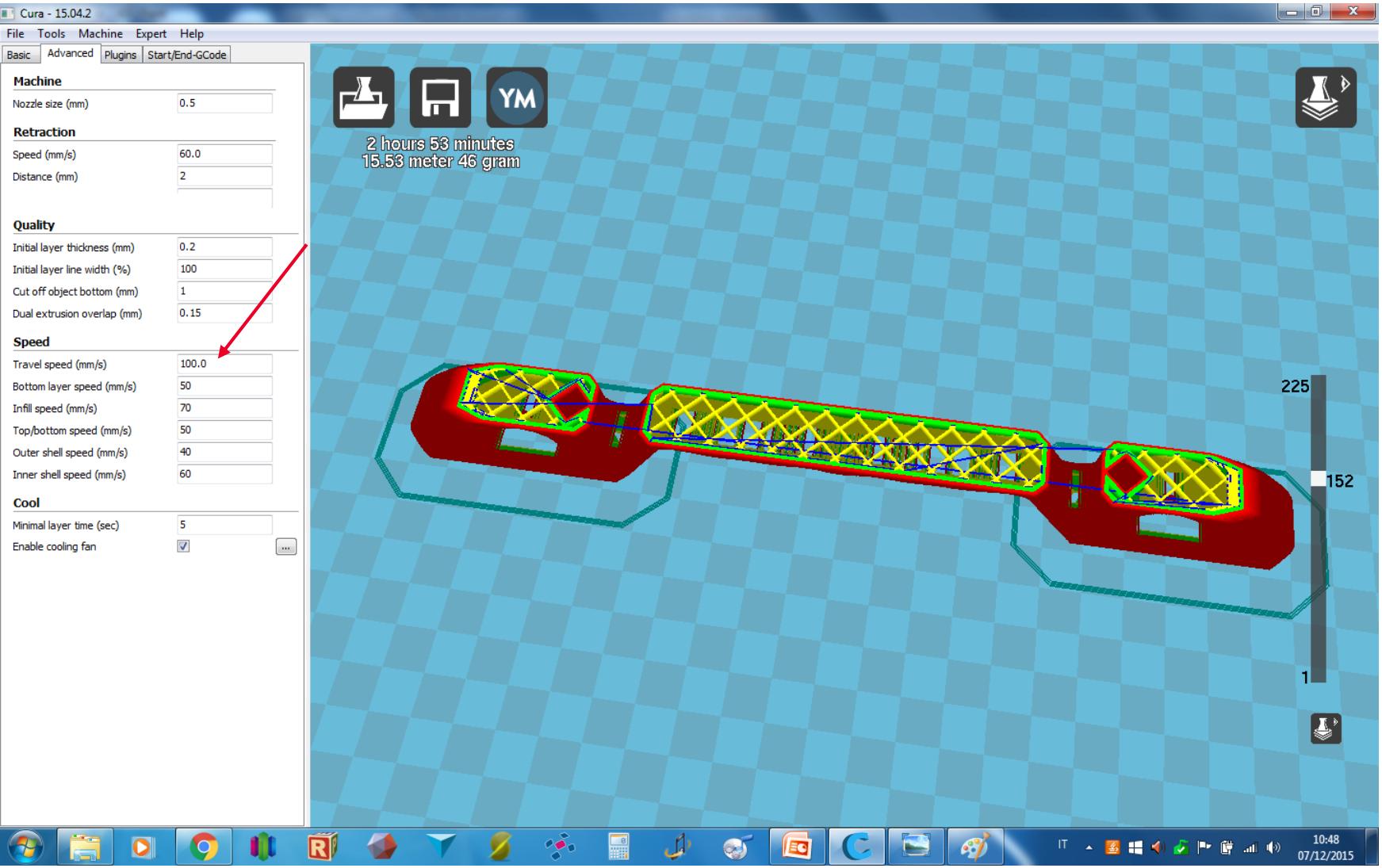
Print Setup - Advanced/Quality section - 3



Dual extrusion overlap

This feature can be used only with dual extruder printers

Print Setup - Advanced/Speed section - 1

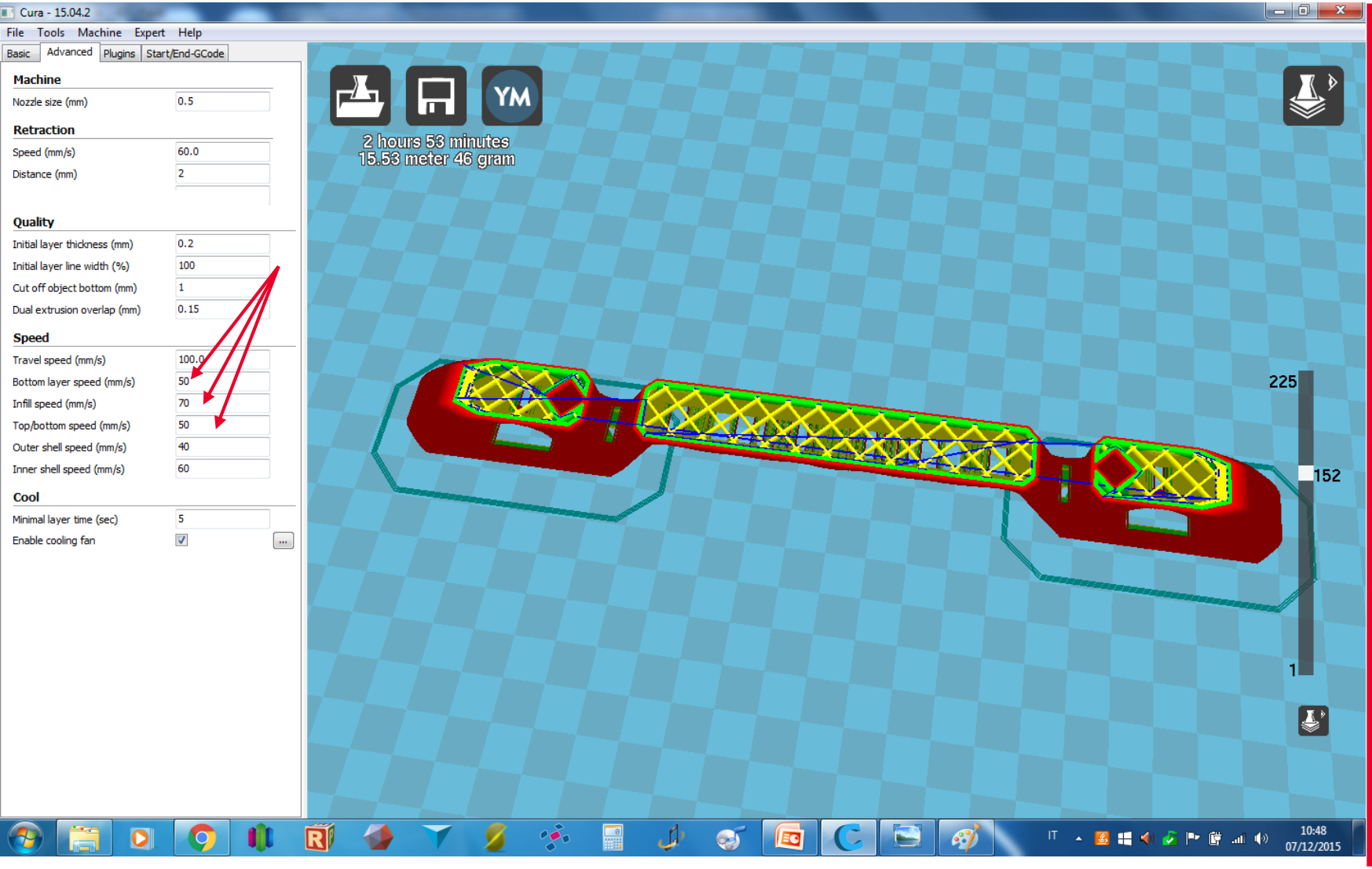


This section allows to set the different printing speed (in mm/sec) for each activity.

Travel Speed

It means the speed of the extruder during the non-printing movements (travels).
These movements are highlighted in blue.

Print Setup - Advanced/Speed section - 2



Bottom Layer Speed

Printing speed for the first full layer

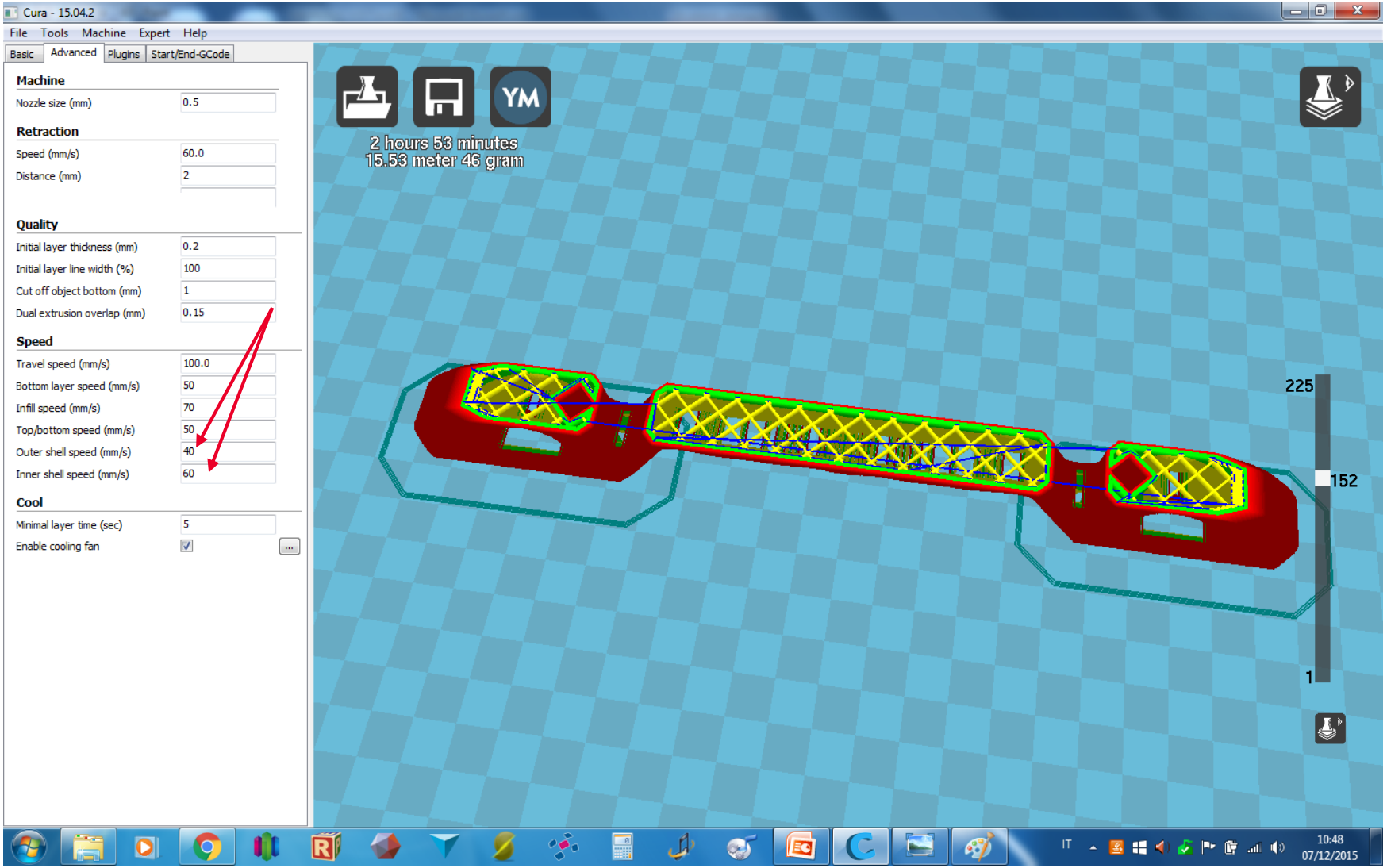
Infill Speed

Printing speed for intermediate layers

Top/Bottom Speed

Printing speed for bottom and top full layers.

Print Setup - Advanced/Speed section - 3



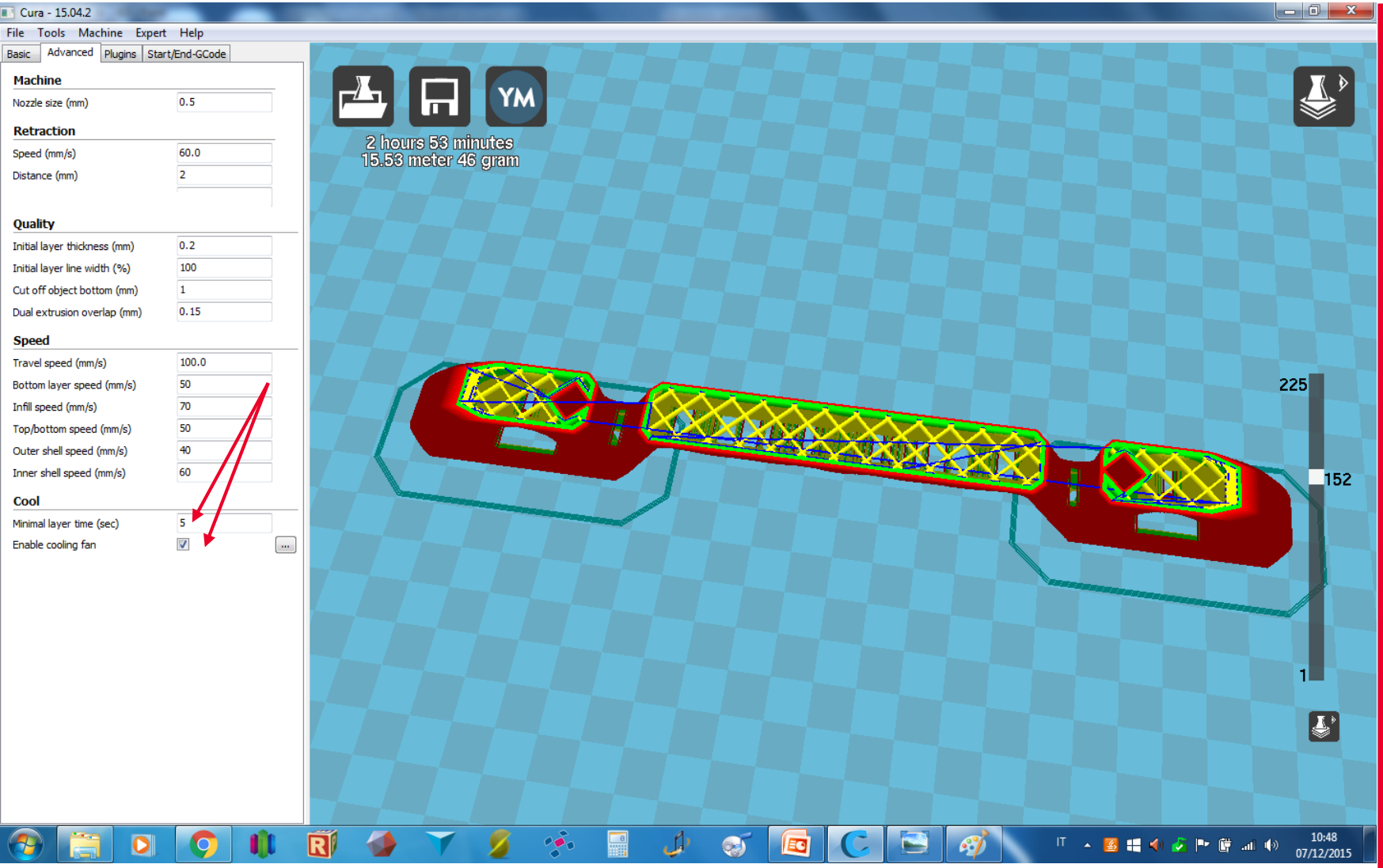
Outer Shell Speed

Printing speed for outer shell (highlighted in red)

Inner Shell Speed

Printing speed for inner shell (highlighted in green)

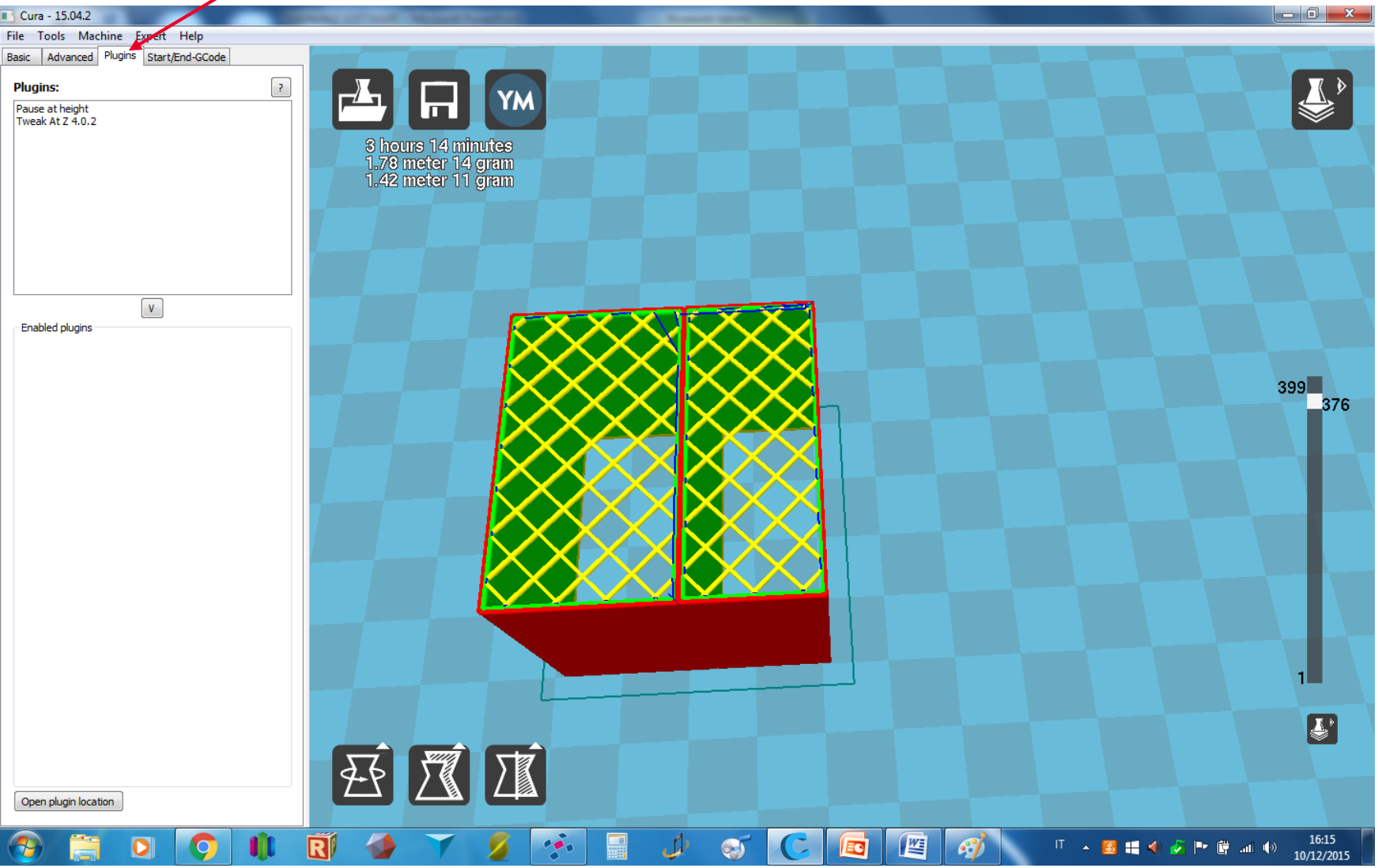
Print Setup – Advanced/Cool section



Cool Minimal layer time
It means the minimum time at which you want to print the single layer.
This feature is useful especially for very small pieces/very small parts, when the material cooling can be critical.

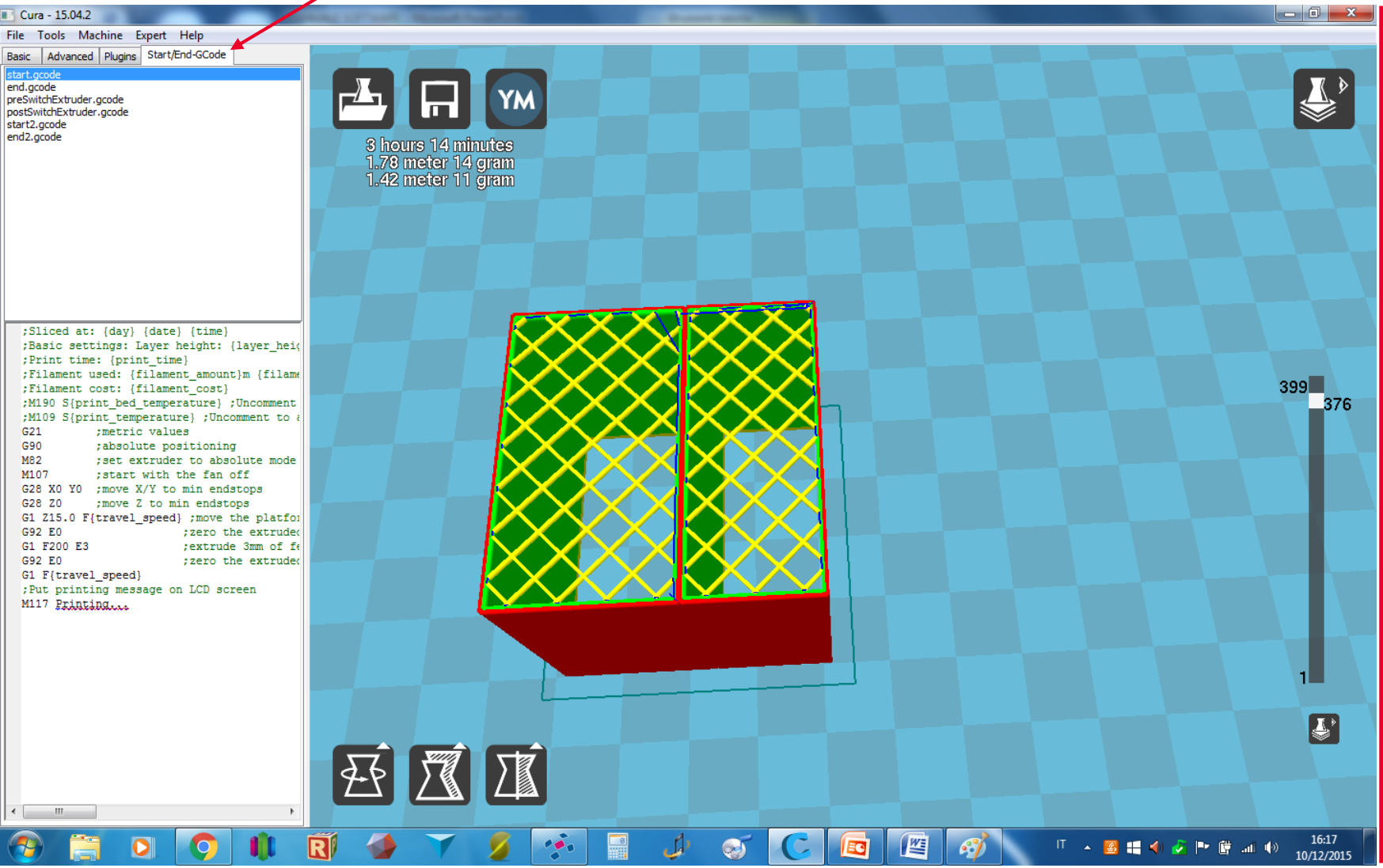
Enable cooling fan
This item must be flagged

Plugins section



It's advisable to NOT use these plugins

Start/End G-Code Section - 1



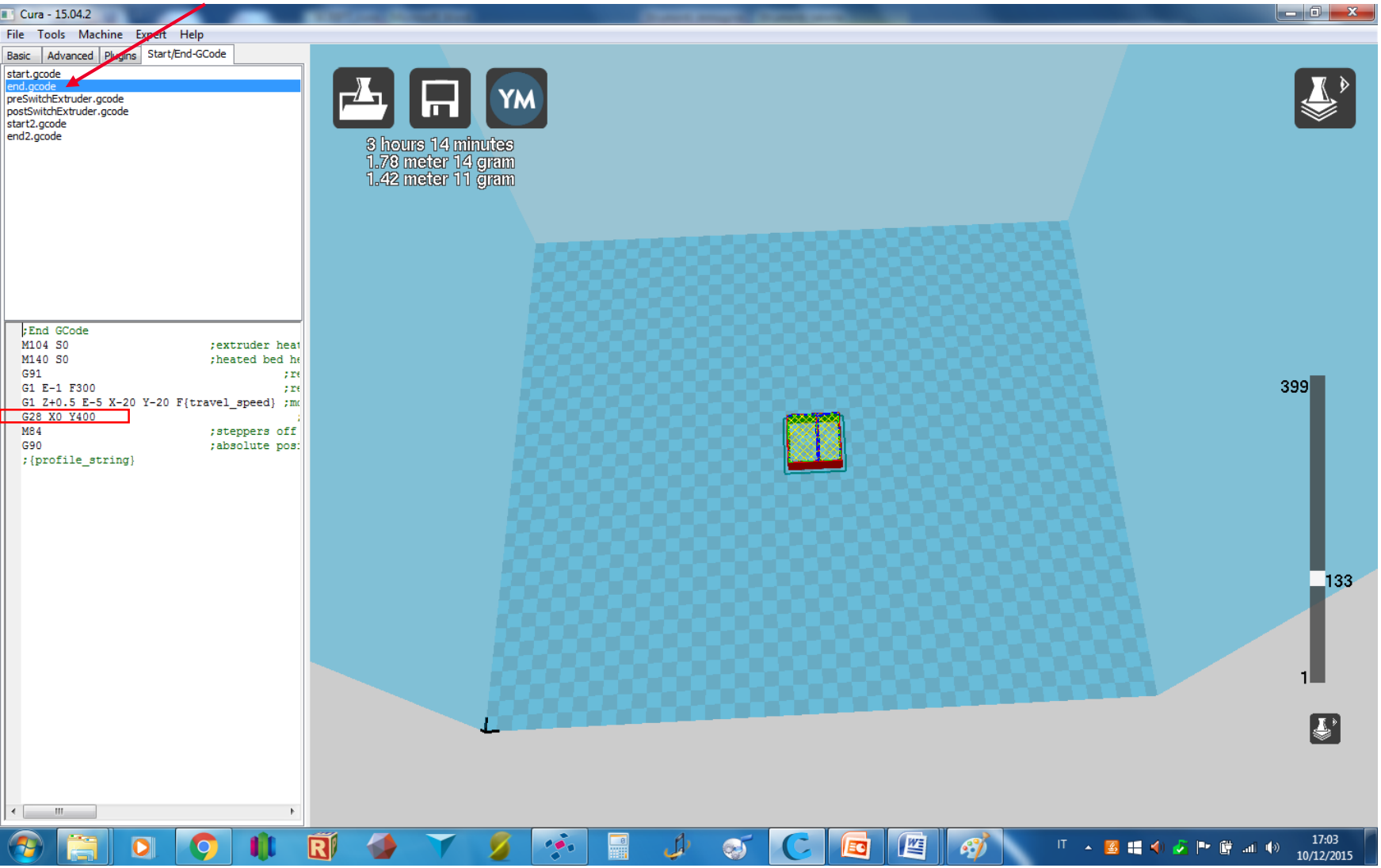
Start/End G-Code

These are strings of machine-code to implement specific printing features.

Some specific strings was created/modified for 3D DESK.

Next slides show the specific items where changes have been made

Start/End G-Code Section - 2

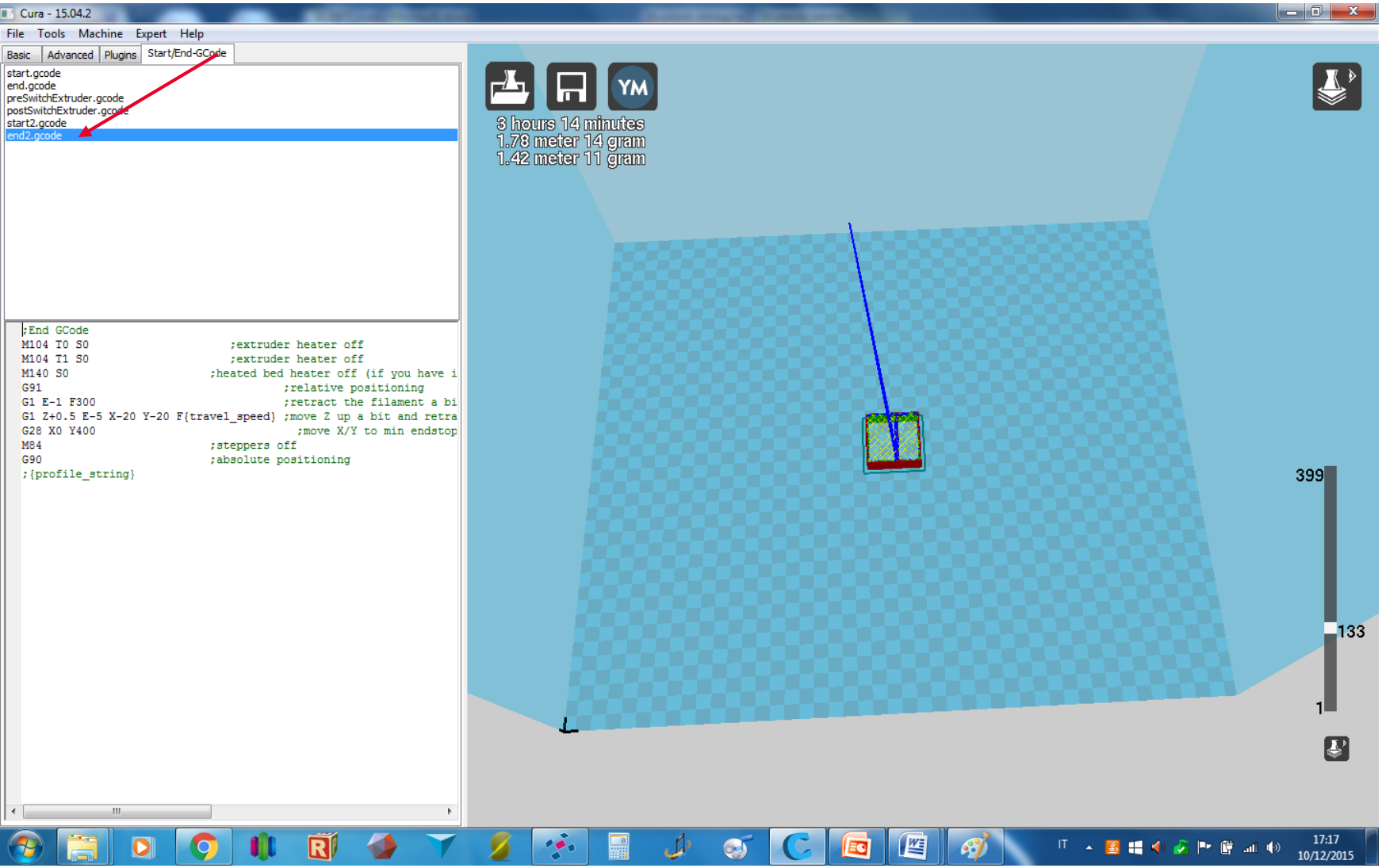


End g.code

Modified string.

The inserted value is
G28 X0 Y400

Start/End G-Code Section - 3



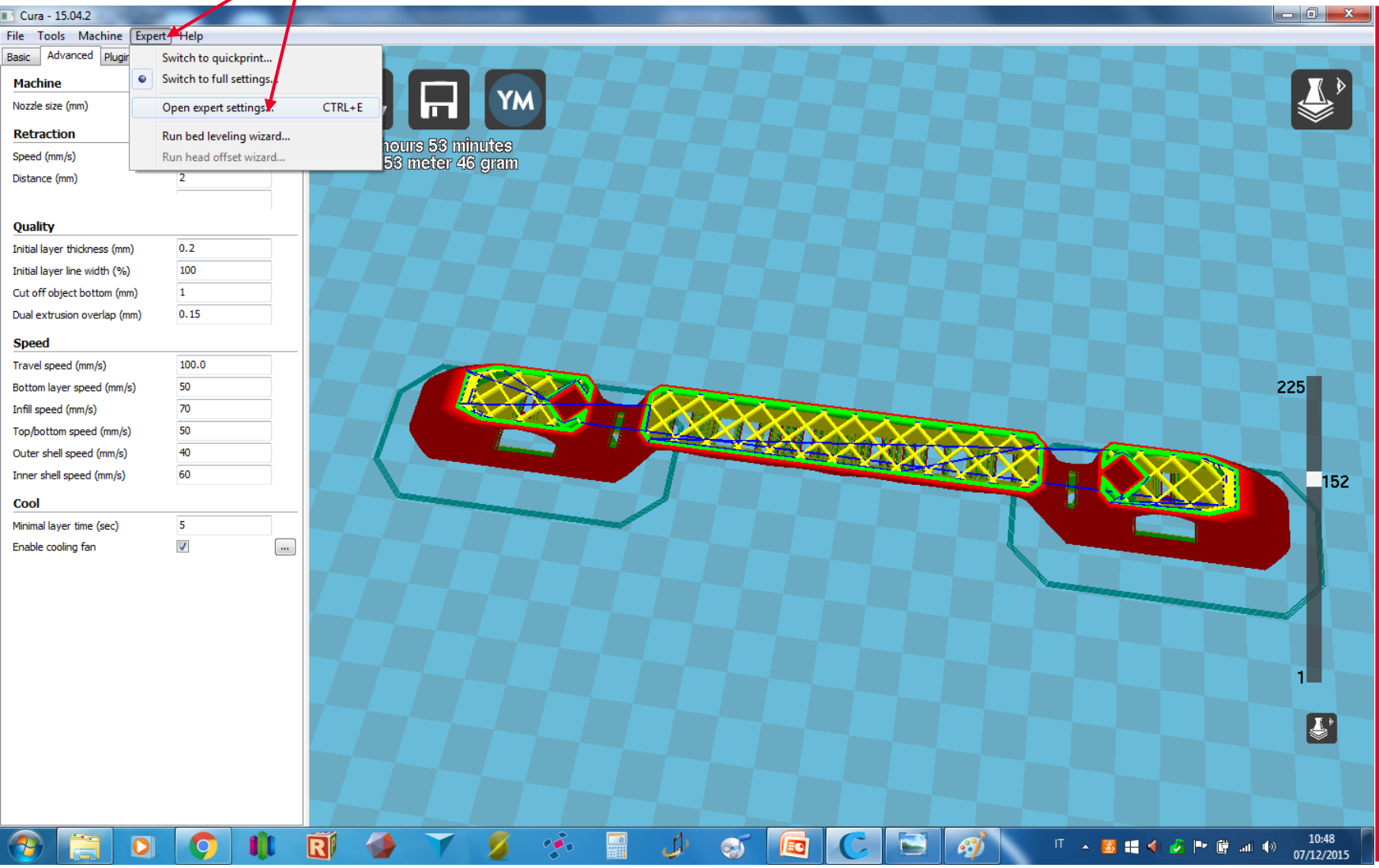
End2

Modified string

At the printing end, it moves the extruder in a position ideal to facilitate the object removal.

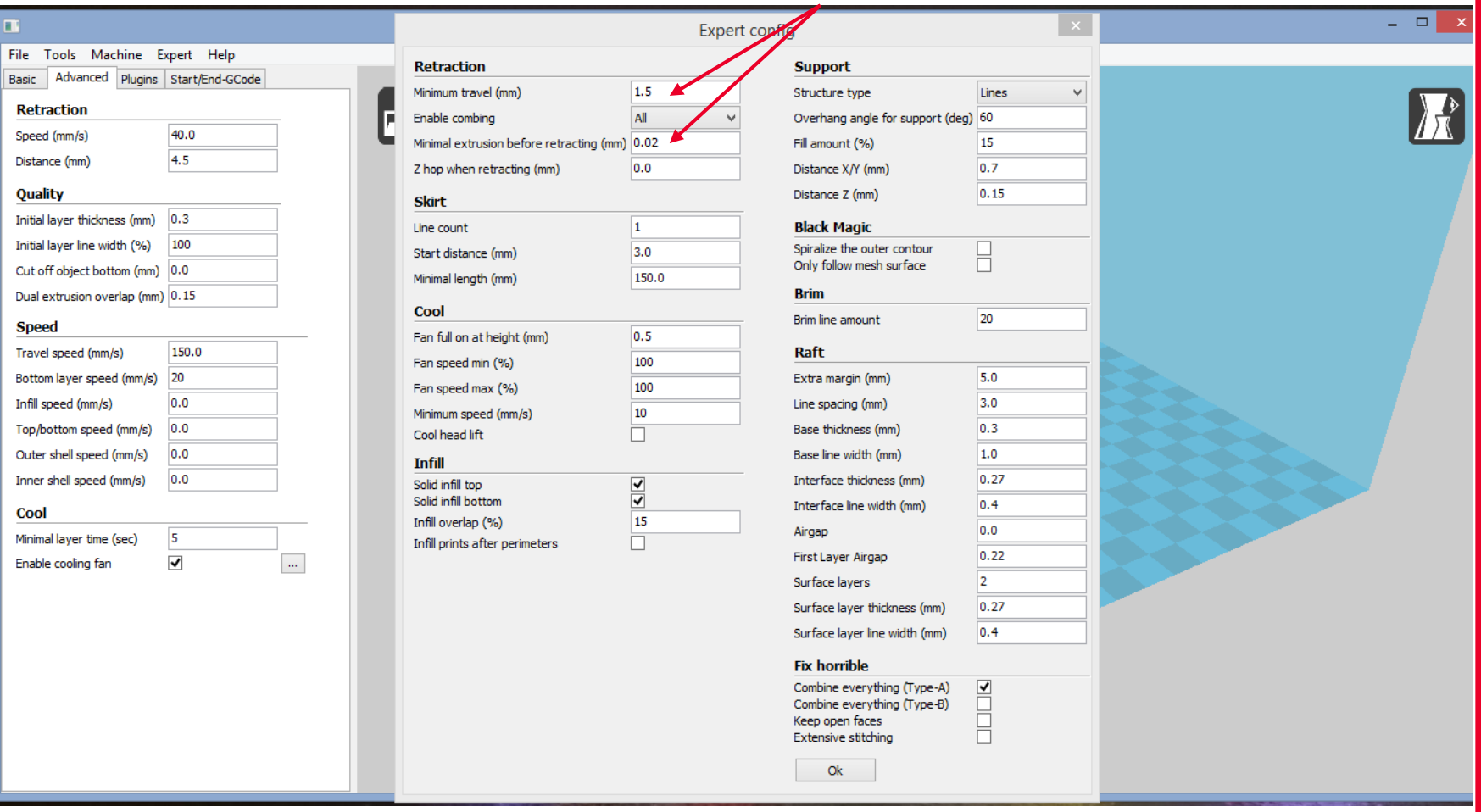
G28 X0 Y400

EXPERT section - 1



From the Expert Menu, select Open expert setting

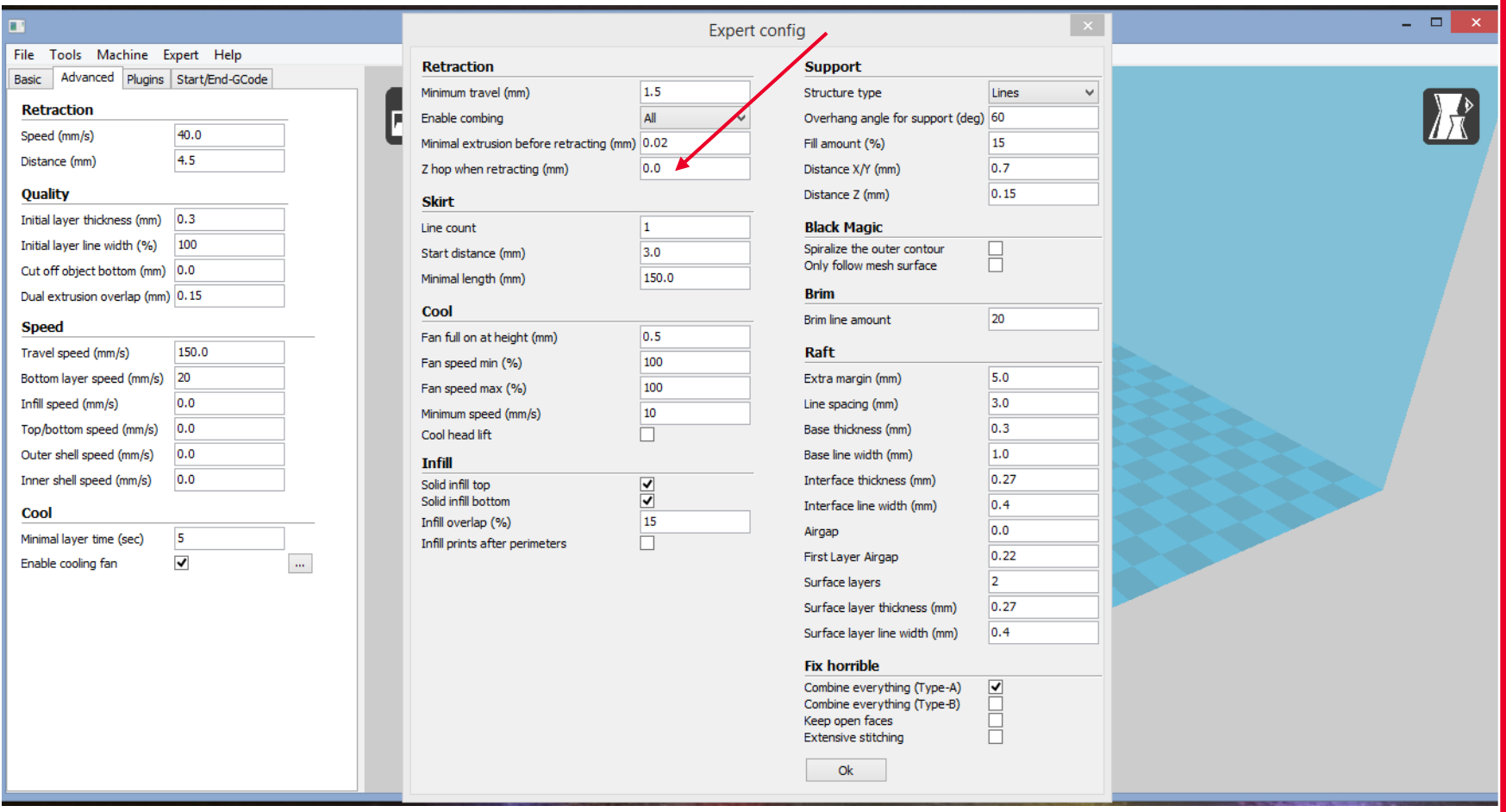
EXPERT/Retraction section - 1



Minimum travel
Minimum non-printing extruder movement (travel) for which a retraction is activated.

Minimal extrusion before retracting
Minimum estrusion which has to be made before a subsequent retraction.

EXPERT/Retraction section - 2

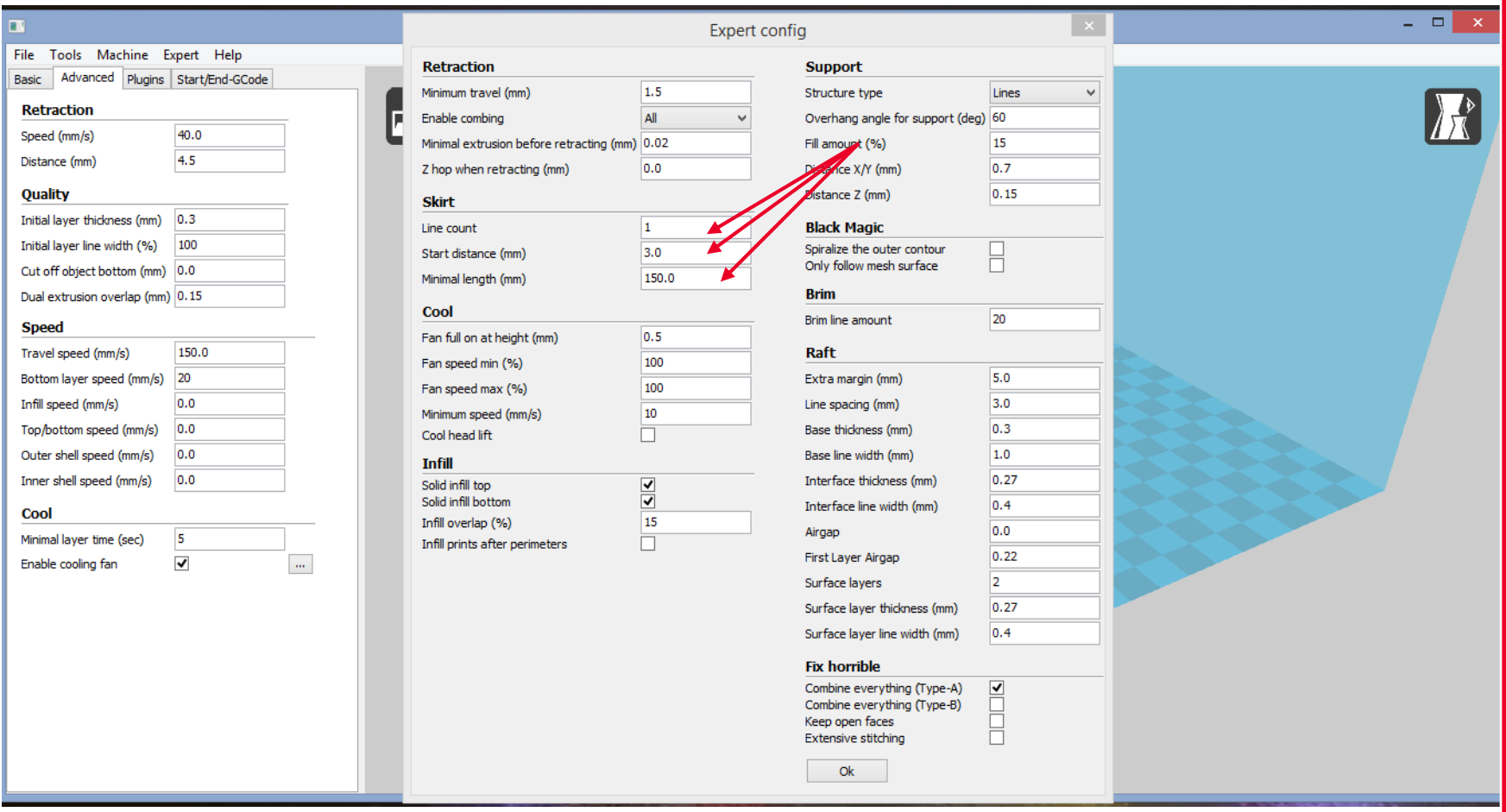


Z hop when retracting

It corresponds to the downward and upward axis Z movement (printing bed), when the printer makes a retraction.

This feature is useful especially to avoid friction between nozzle and bed, when the extruder makes a long “travel” (non-printing movement)

Sezione EXPERT/Skirt



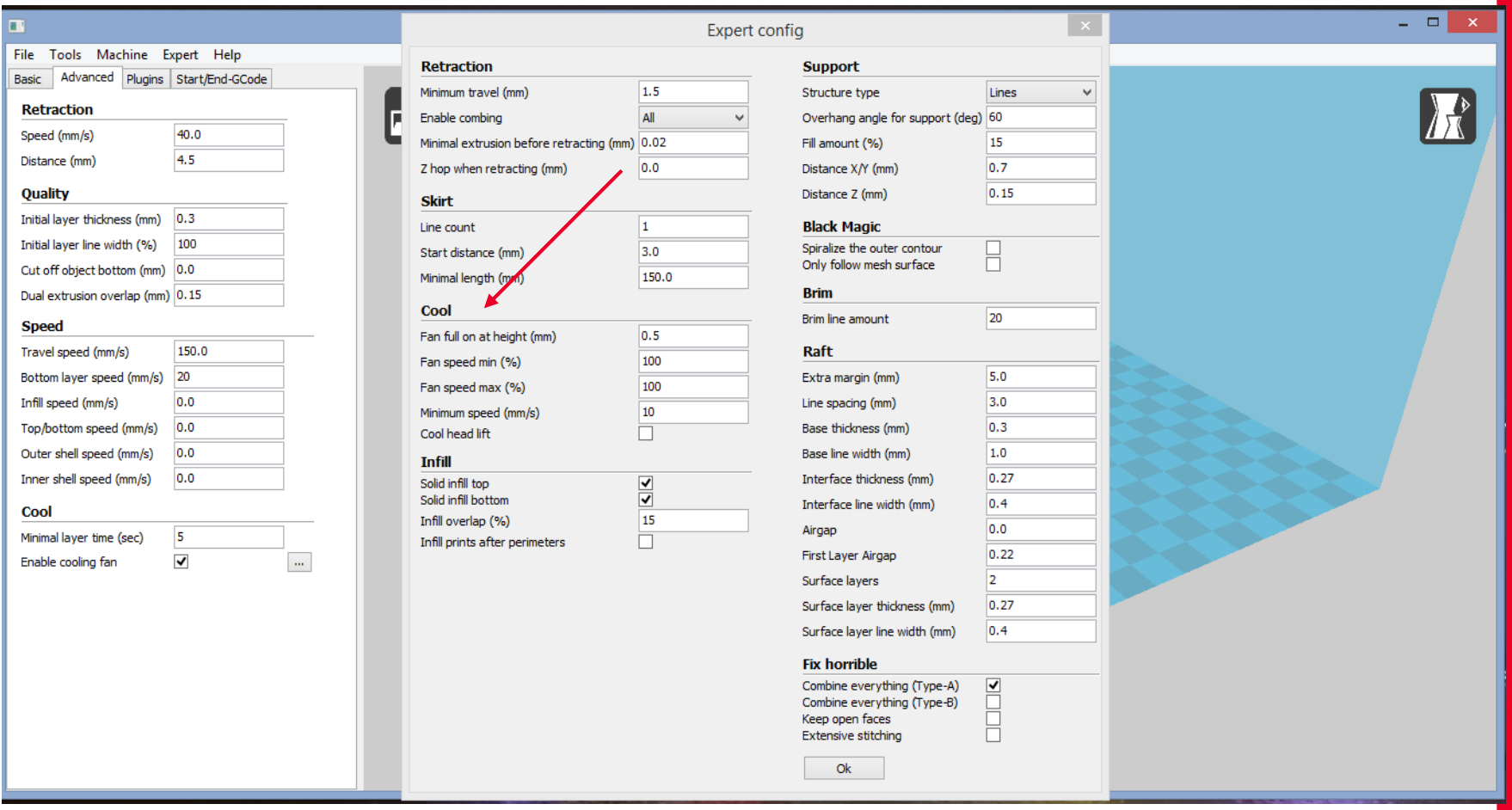
At the printing beginning, the skirt feature allows to create outer perimeters, to improve the initial printing quality

Line count
It sets how many lines make the skirt.

Start distance
It sets the distance between the skirt and the object

Minimal length
Skirt minimal length

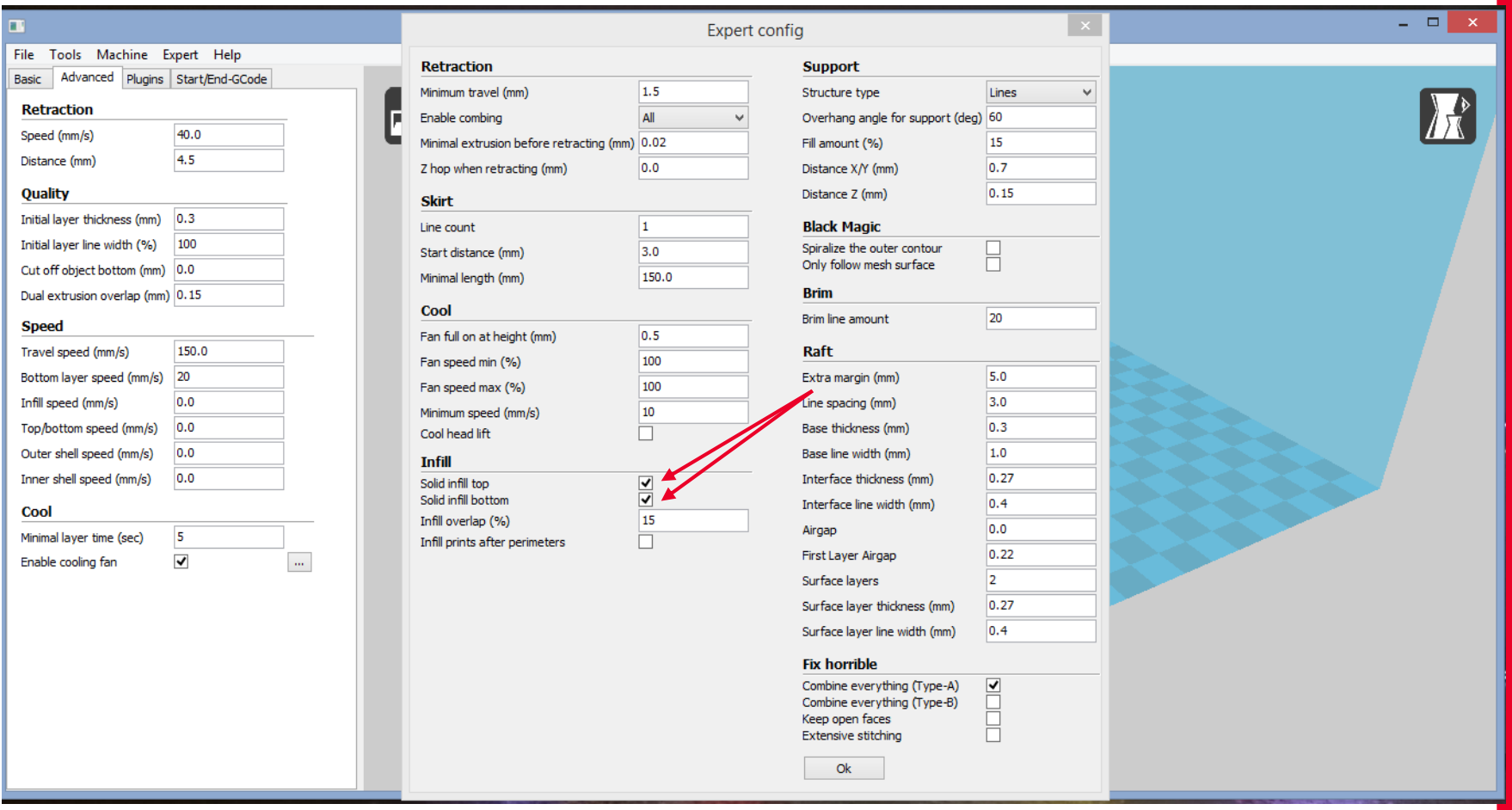
EXPERT/Cool section



The "Cool" functions are used to manage the cooling fan working.

It's advisable to keep the settings shown in the picture, to guarantee an effective cooling of the material.

EXPERT/Infill section - 1

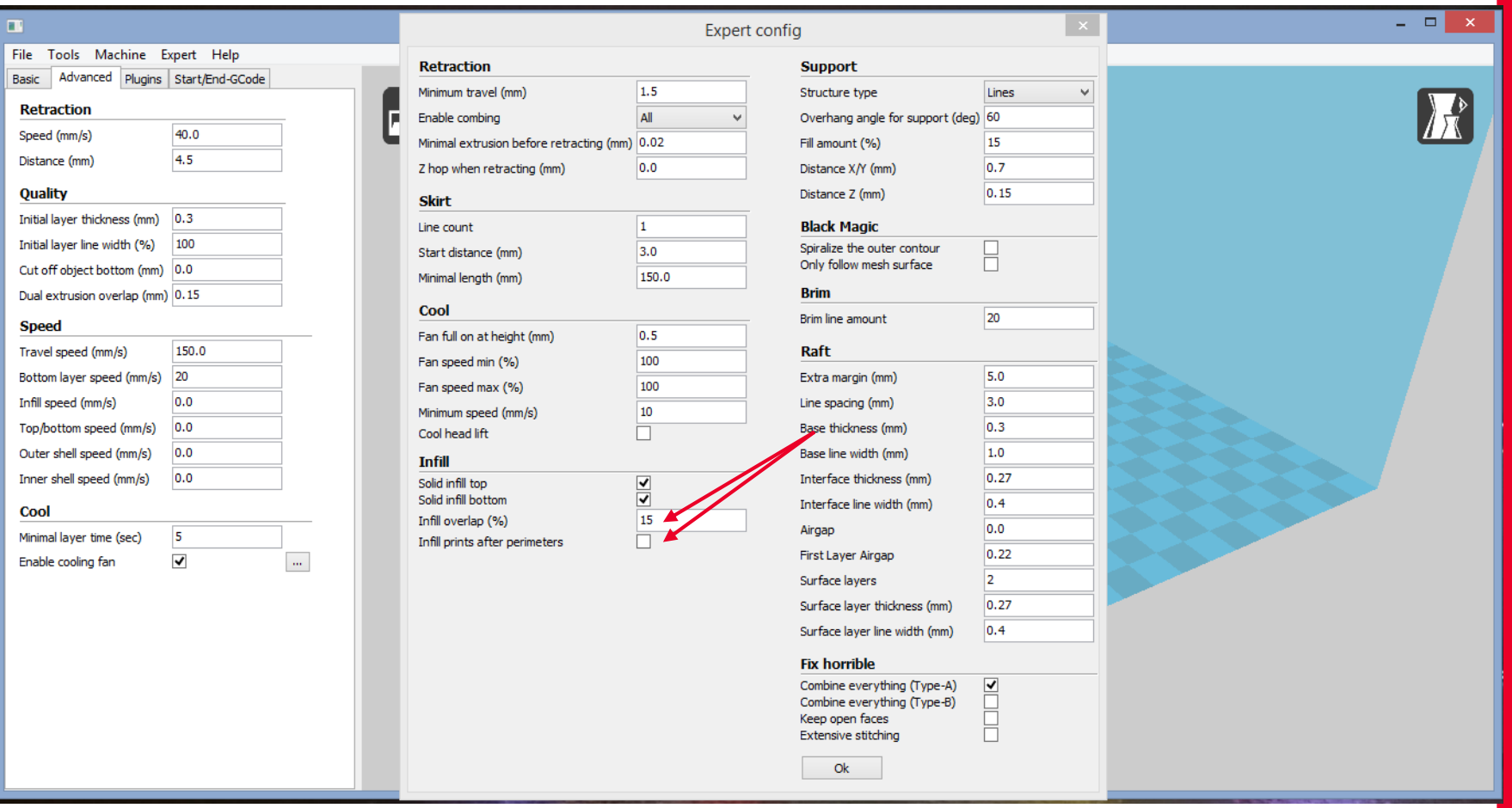


Infill

Solid infill Top
Solid infill Bottom

Flagging these items, the Top and bottom faces of the object will be filled; unflagging them, the faces will be empty.

EXPERT/Infill section - 2



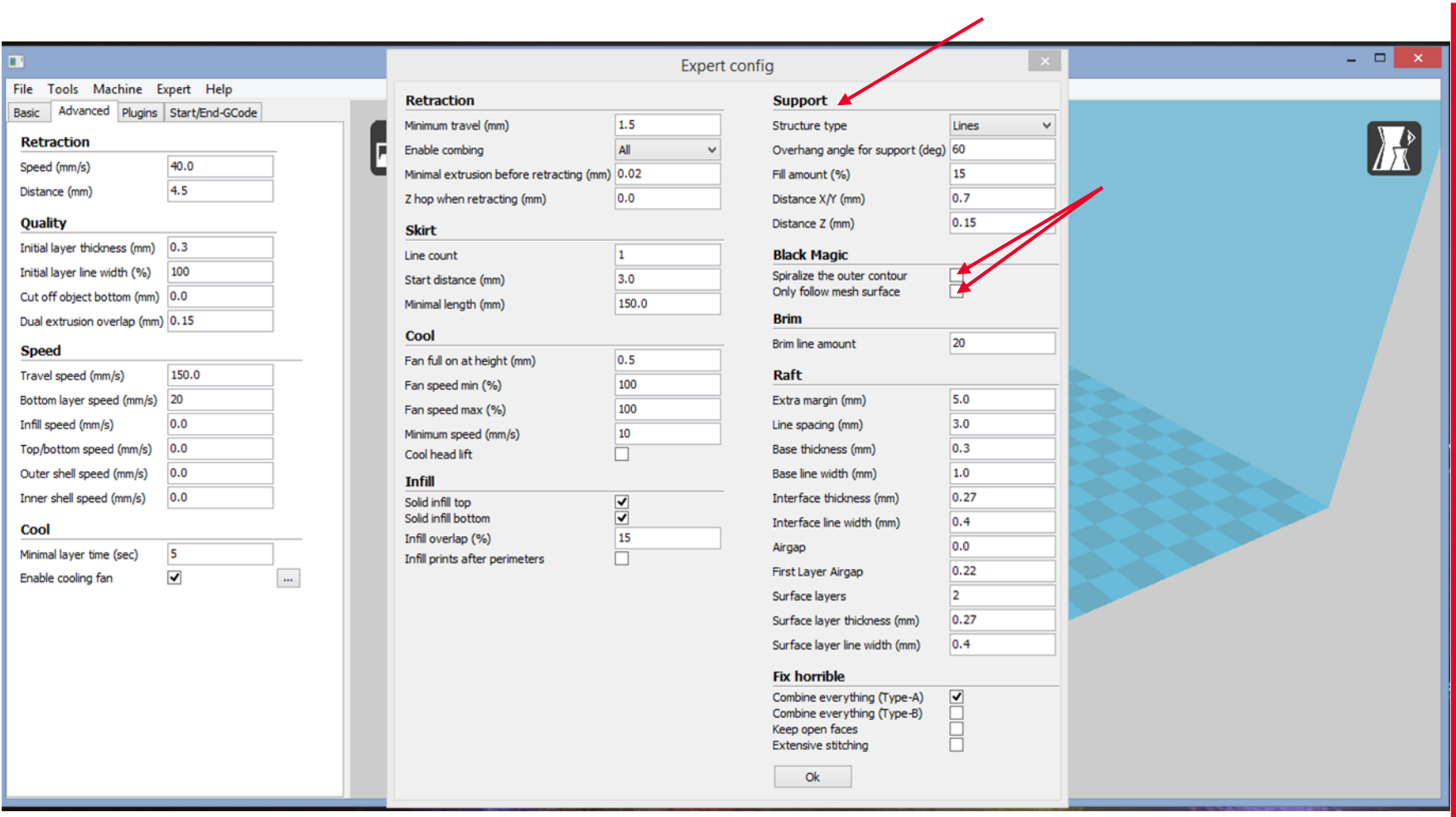
Infill overlap

Overlappping percentage between perimeters and filled layers of the object.

Infill print after perimeters

It allows to create the filled layers before/after the perimeters.

EXPERT/Support and Black Magic section



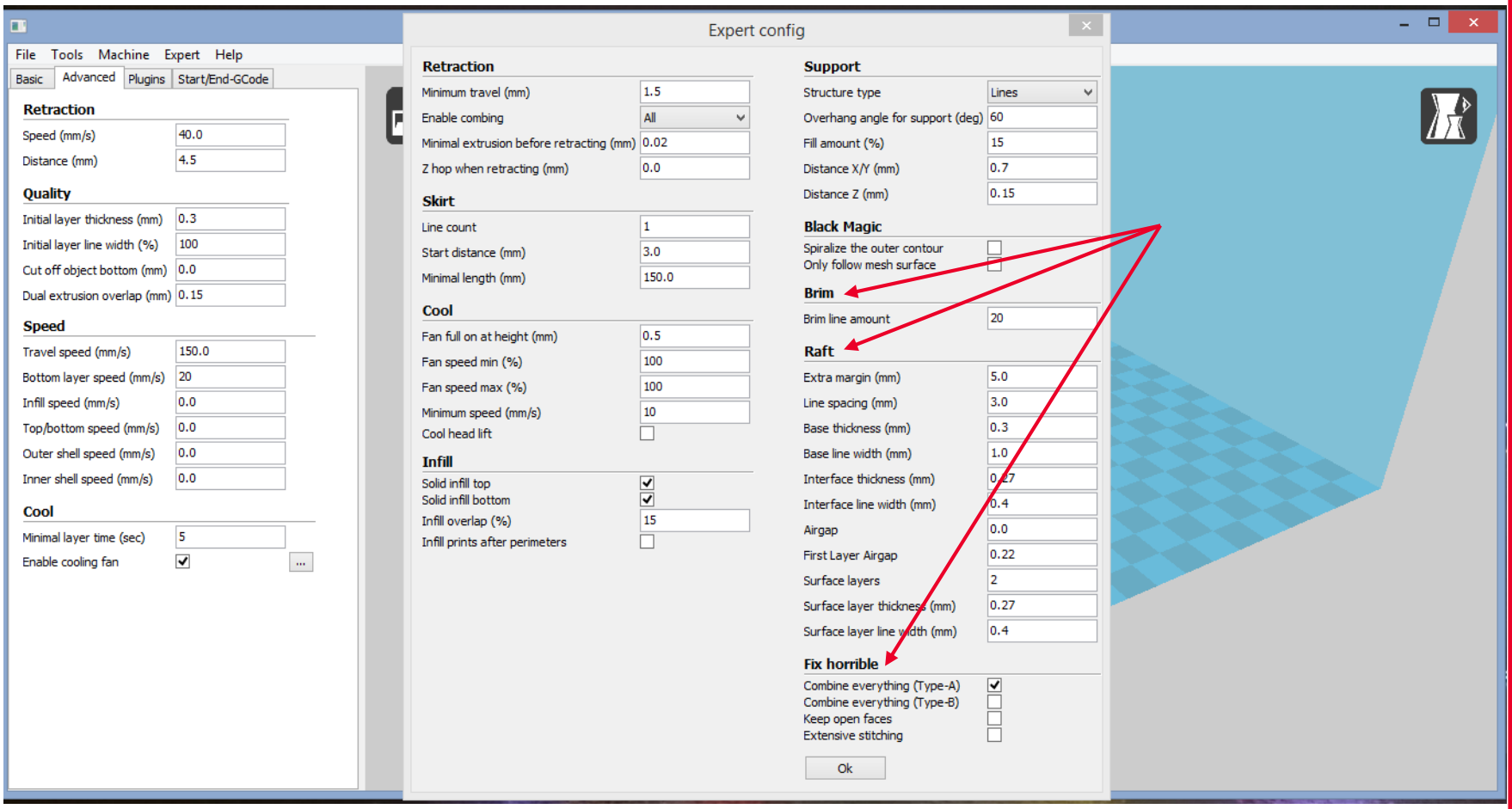
Support

Features already described at slides 31 and following

Black Magic

option available only for objects not filled (only perimeters), as the Z axis constantly moves down and not *layer after layer*.

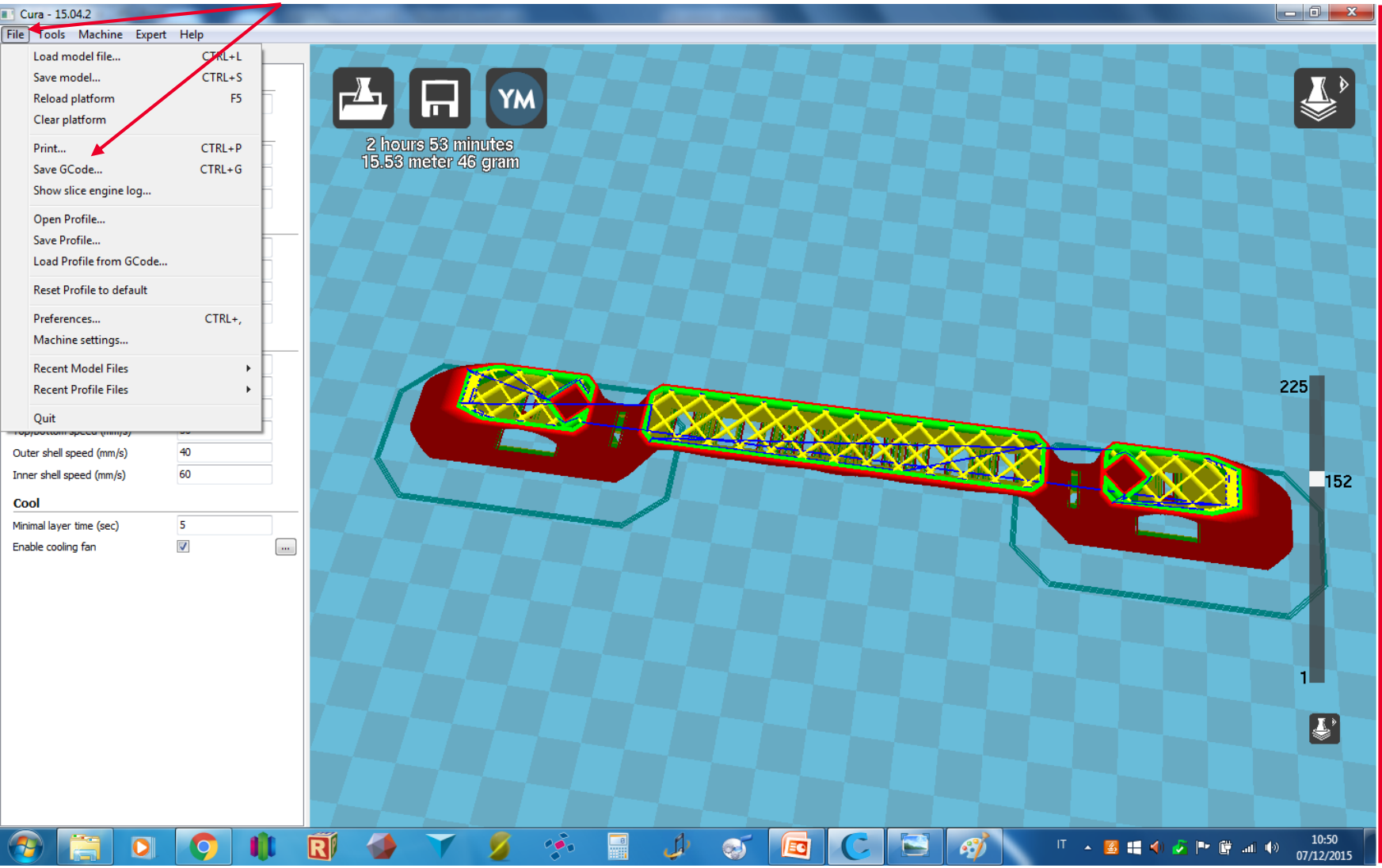
EXPERT/Brim, Raft and Foxx Horrible sections



Brim and Raft Features already described at slides 35-36 and following

Fix Horrible Features not available for the Olivetti 3D DESK

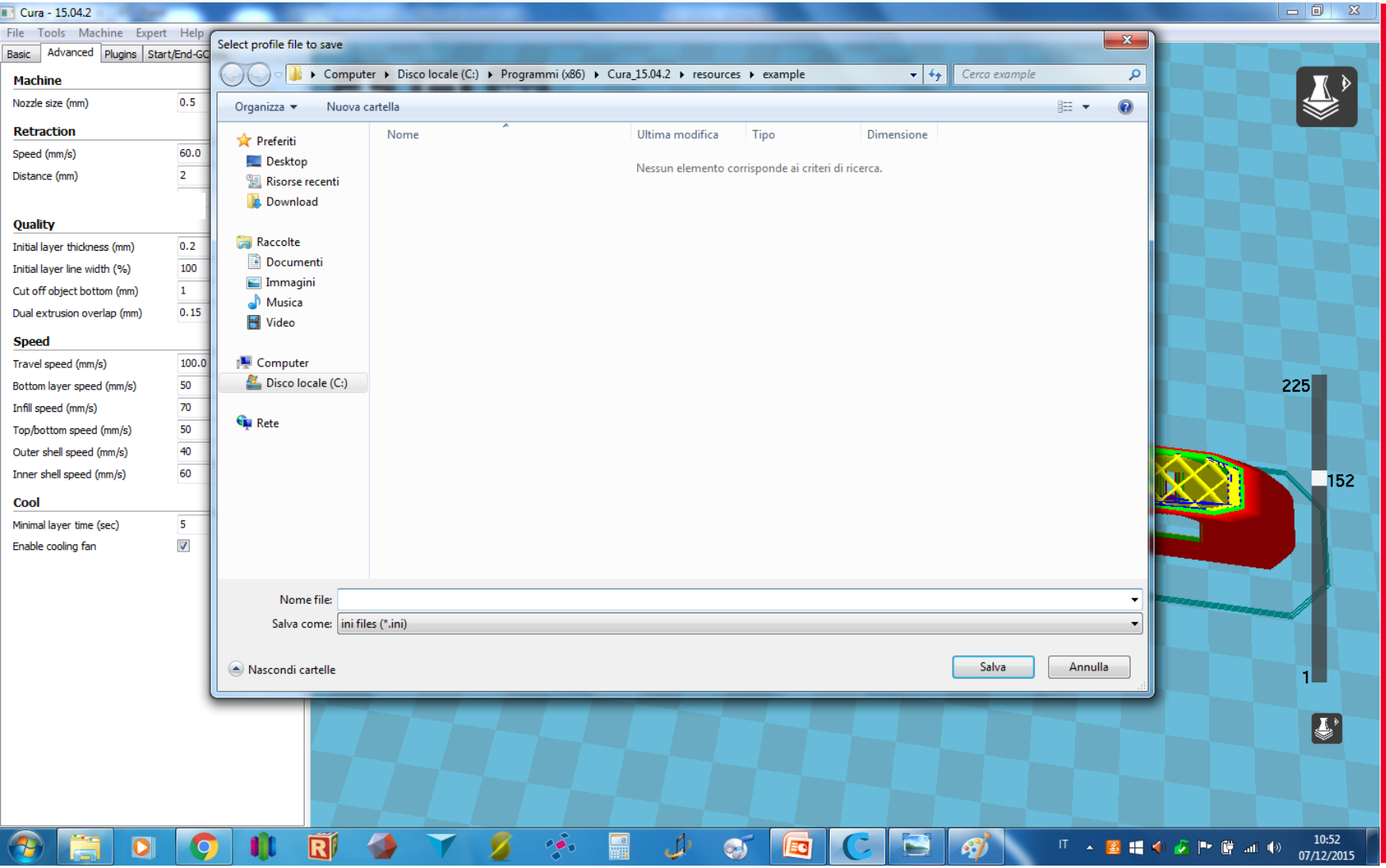
File saving - 1



Once the printing setup is complete, the file has to be saved, selecting File → Save Gcode, to save the file on your PC or directly on the SD Card.

The file extension is Tollpath (Gcode)

File saving - 2



Once the Gcode was saved, it's advisable to save also the printing profile, selecting File → Save Profile.

This way, it's possible to save the printing setup used for each object and/or object groups.

Slicer"CURA"

User Instructions for Olivetti 3D DESK