HEI MAKERS

LESSON #7
3D PRINTING SOFTWARE

Technical creativity in 3D printing module

Co-funded by the
Erasmus+ Programme
of the European Union
2017-1-LT01-KA203-035231

OUTLINE OF THE LESSON #7

- Topic 1 Workflow
- Topic 2 Slicers
- Topic 2 CURA
- Further learning
- Tasks for reflection

TOPIC 3.7 3D KEY COMPONENTS

- In this topic, you will learn about the basic features of 3D printing software
- Expected learning outcomes: basic knowledge on 3D printing software

Duration	1 academic hrs
Author / Lecturer	Justas ingelevičius, TEA
Delivery methods	Individual / Teamwork / P2P
Evaluation methods	Test / Report / Feedback / Exam etc.

WORKFLOW

Modelling object

Exporting from modelling software

Importing into slicer software

Adjusting printing parameters

Sending to 3D printer

SLICERS

Software that will allow 3D printing is called slicer. It takes a 3D drawing (most often in .STL format) and translates this model into individual layers. It then generates the machine code that the printer will use for printing.

In this lesson we will cover one of the slicer software - CURA

Source:
http://edutechwiki.unige.ch/en/Slicers_and_user_interfaces_for_3D_printers [2019]

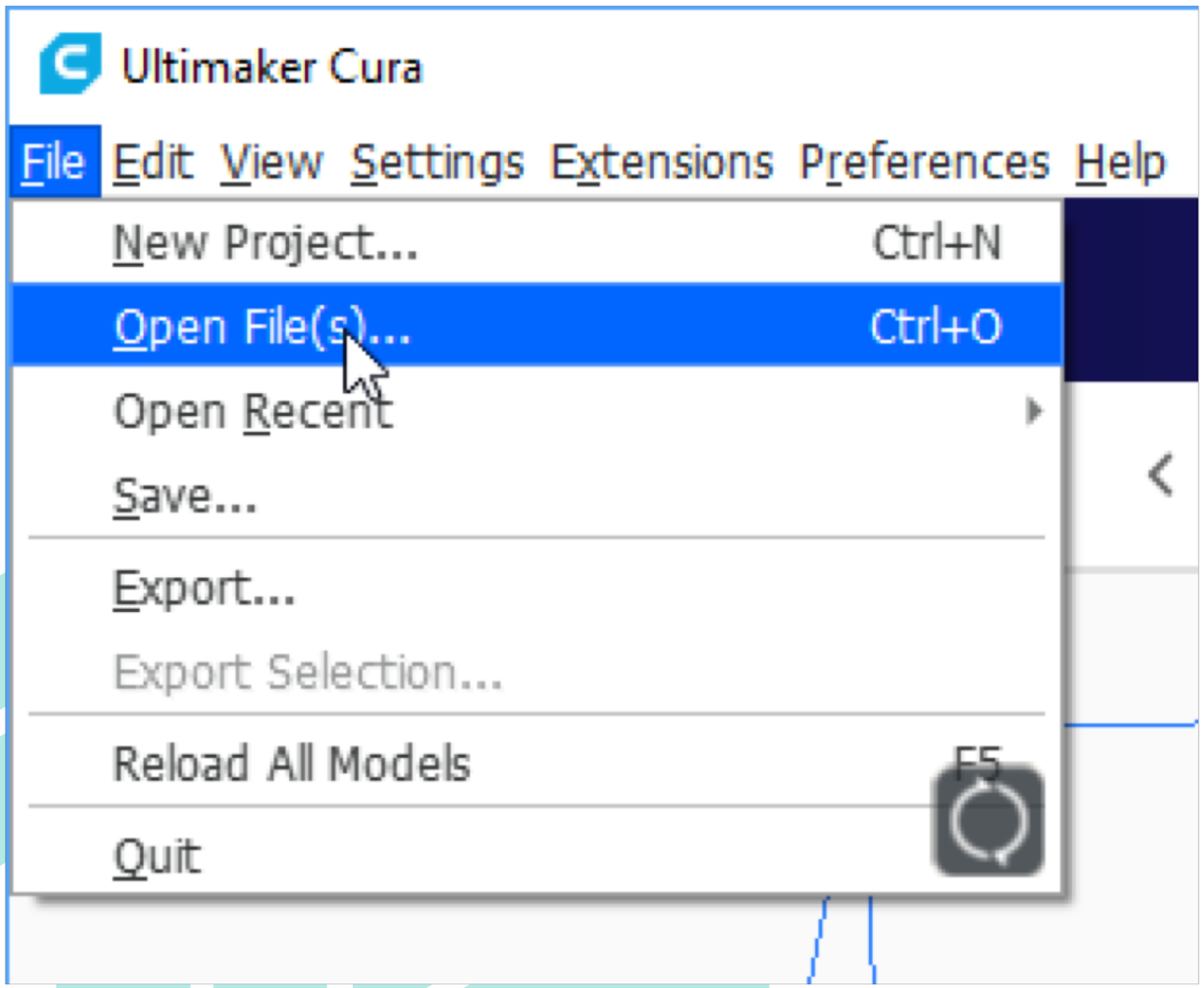
CURA

In this lesson, we will present one of the slicer software CURA 4.0

EXPORTING FROM MODELING SOFTWARE

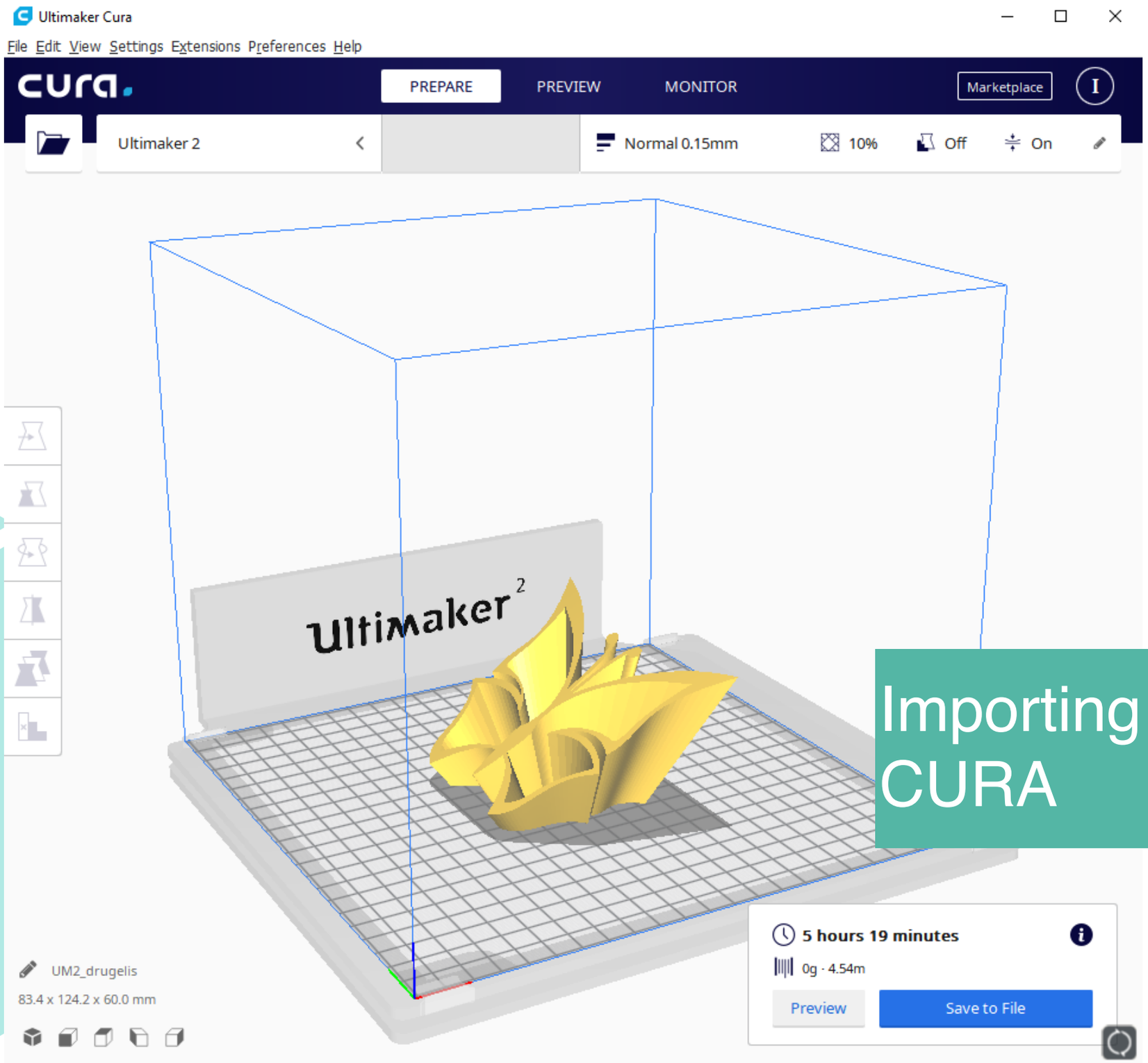
Firstly you need to obtain a file from your modelling software. In most cases, you need to go find “export” and chose .stl or .obj file extension

IMPORTING TO CURA



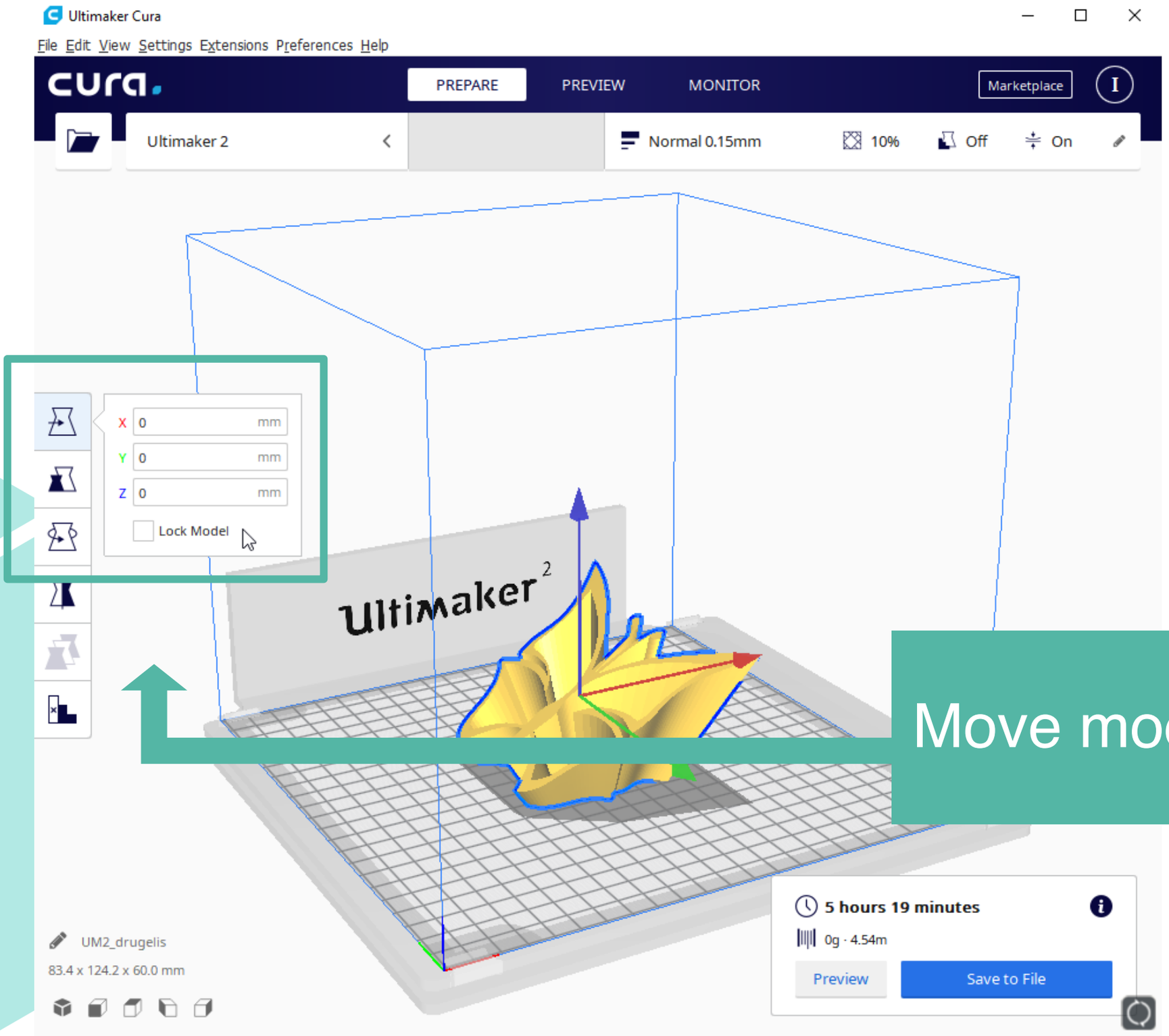
Importing file into CURA

IMPORTING TO CURA



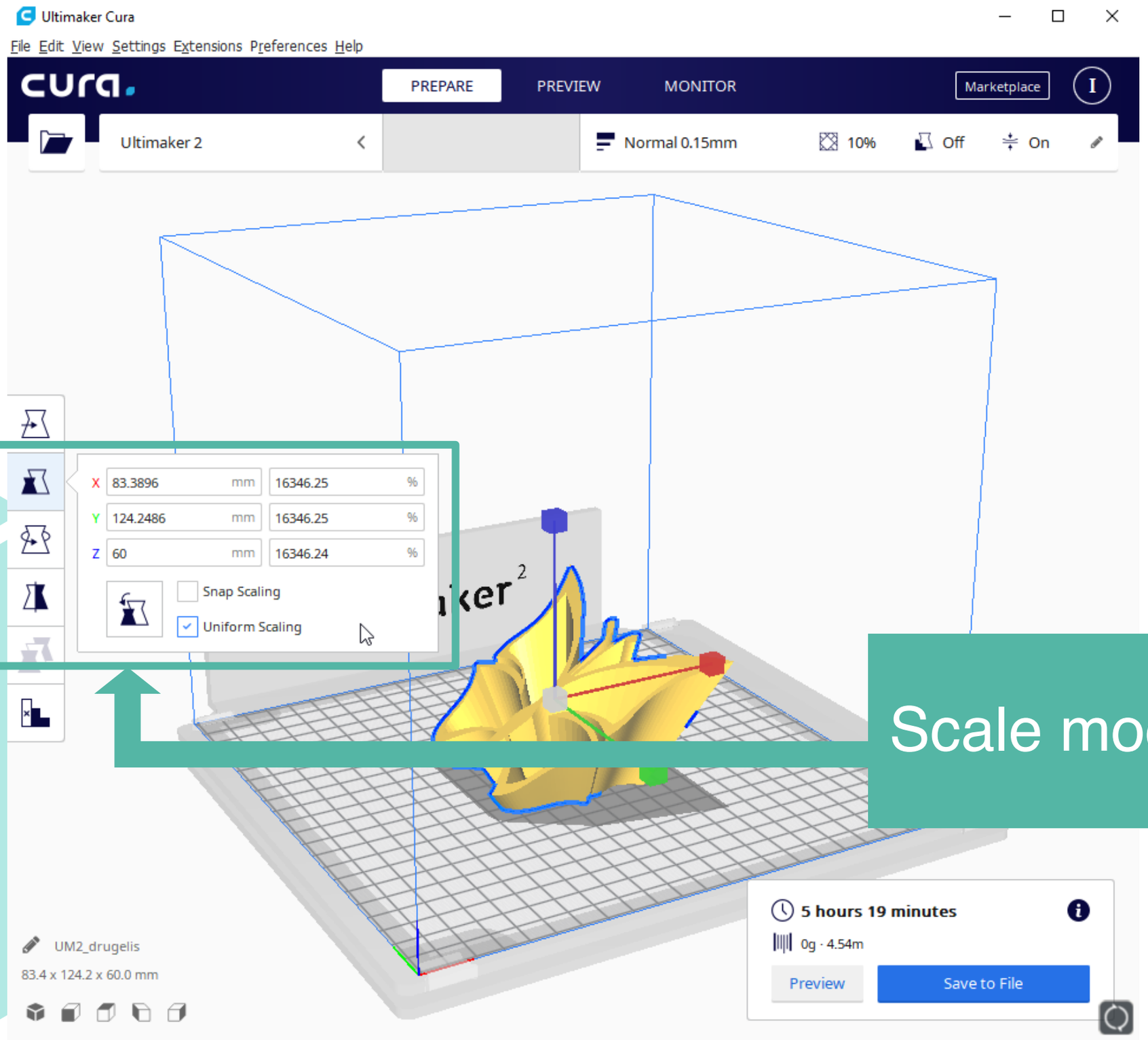
Importing file into CURA

MAIN TOOLS



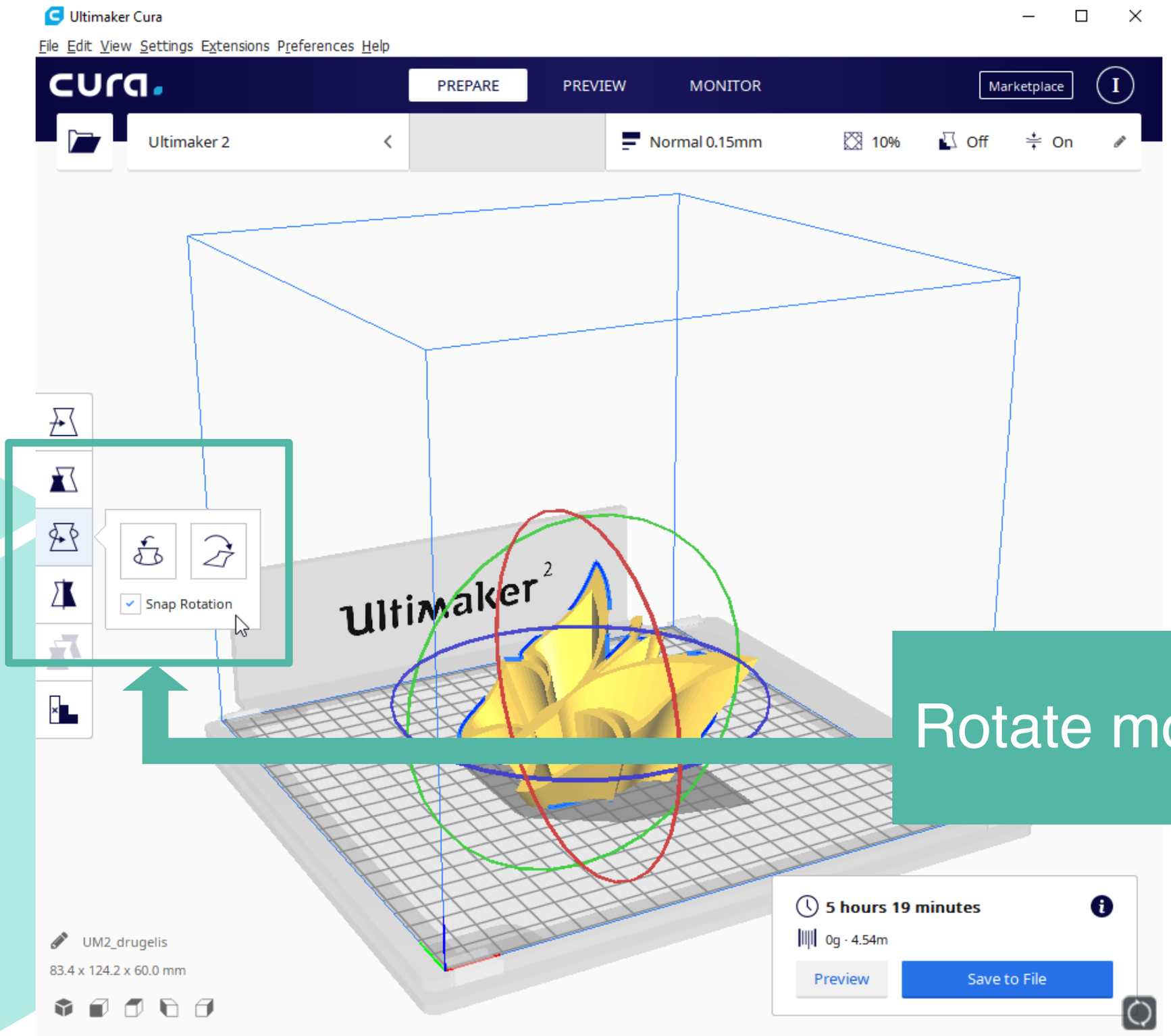
Move model

MAIN TOOLS



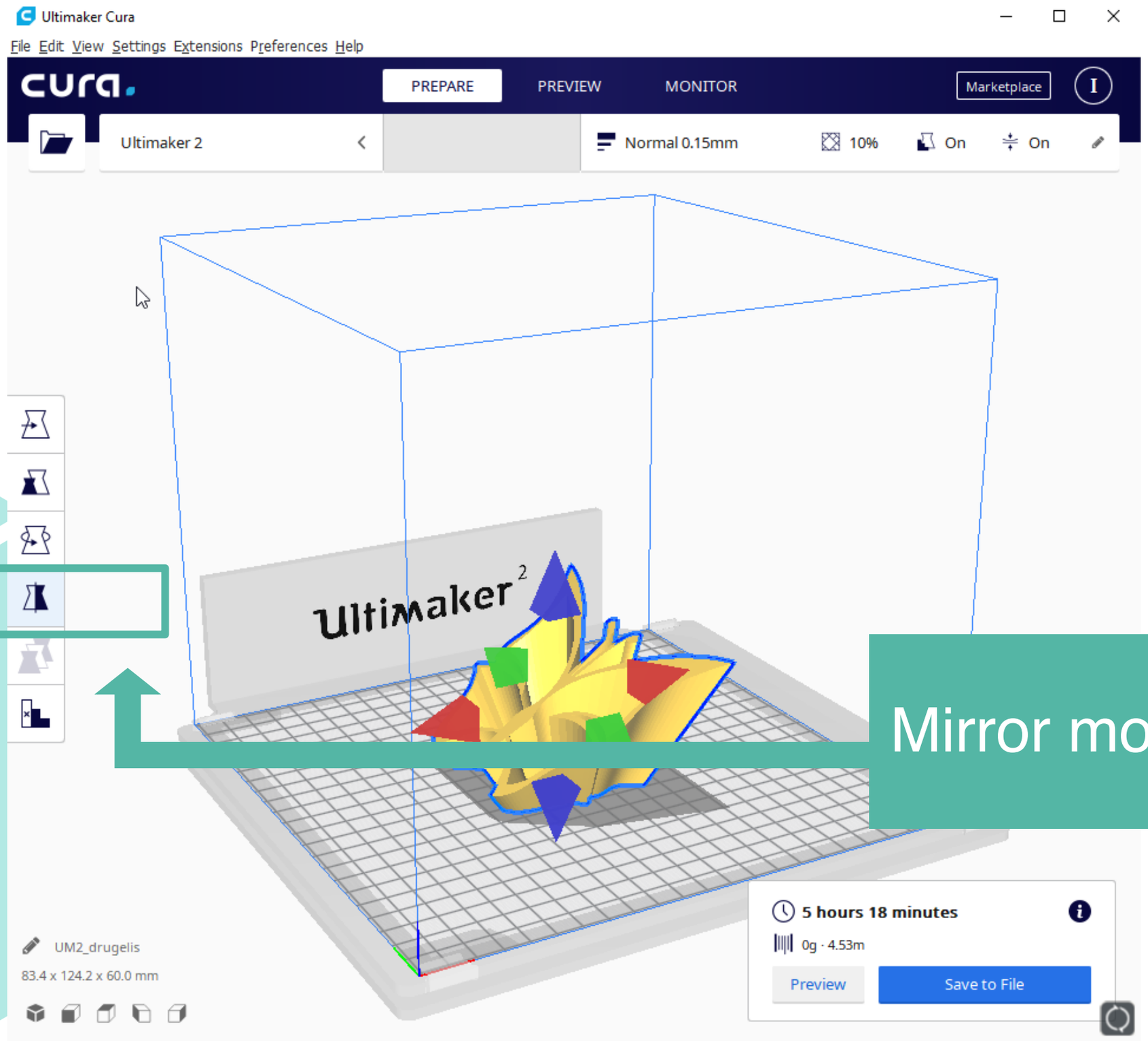
Scale model

MAIN TOOLS



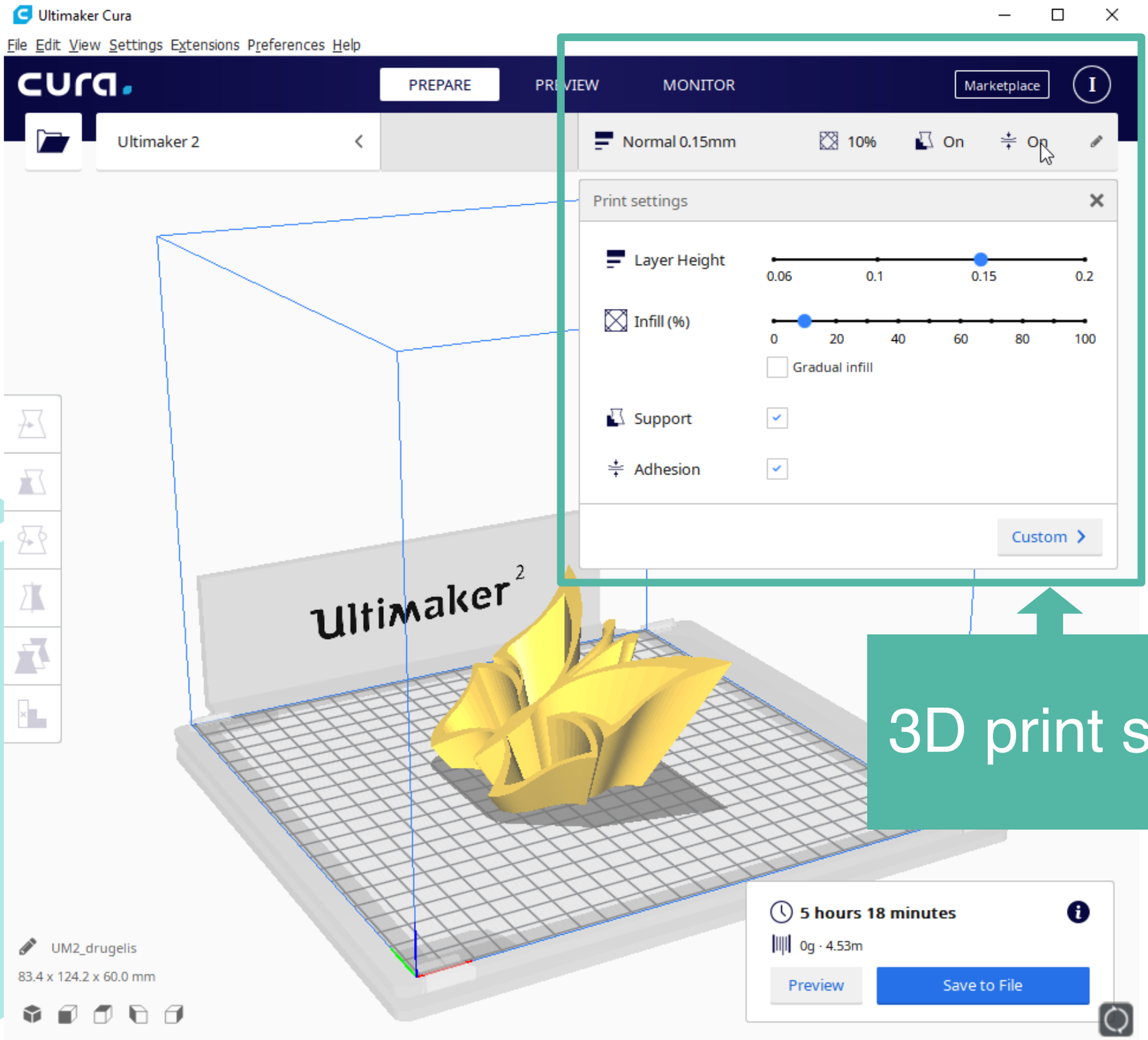
Rotate model

MAIN TOOLS



Mirror model

SETTINGS



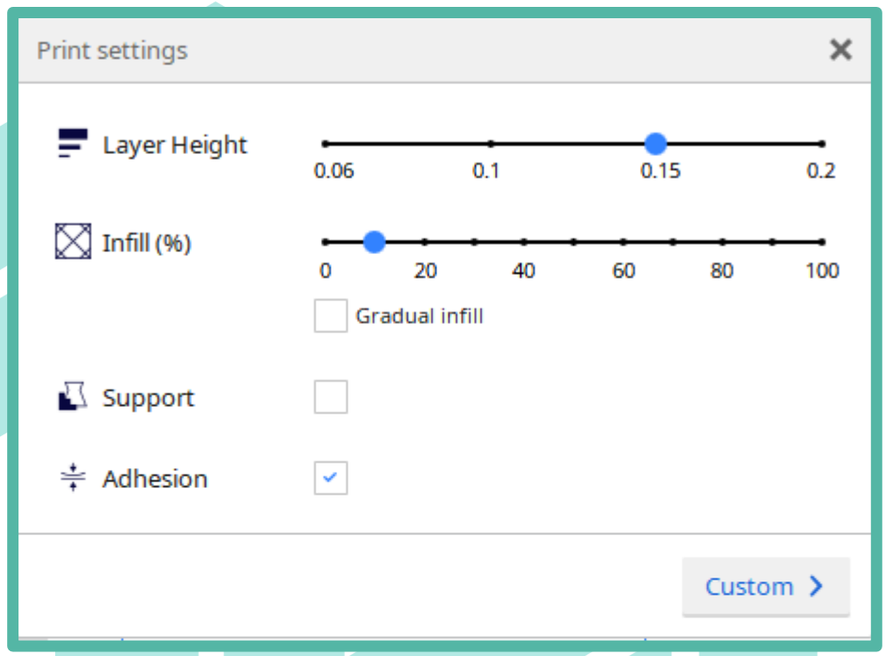
3D print settings

SETTINGS

Layer Height – choosing bigger Layer Height will result in less accurate model, but faster printing speed, and vice versa.

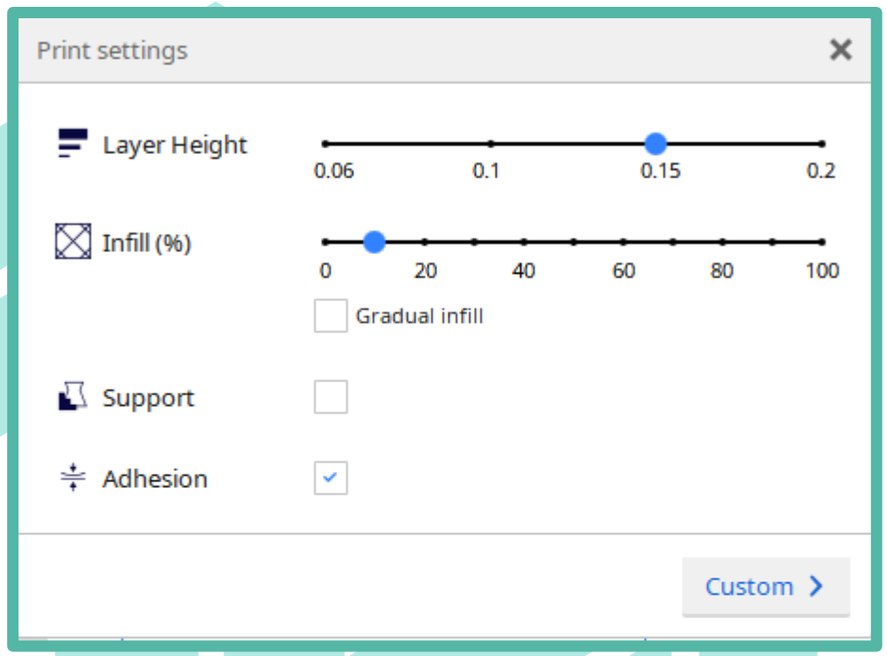
SETTINGS

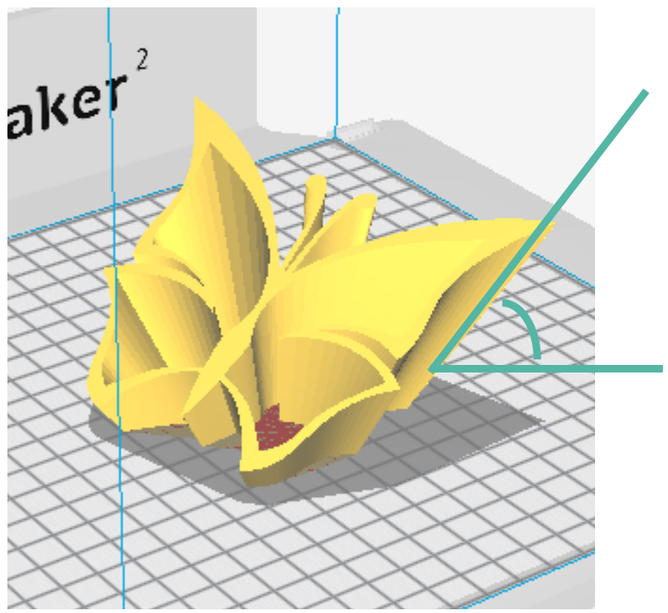
Infill – defines if the 3D print will be hollow or completely solid or filled with honeycomb like structure. More infill makes object stronger but use more material and printing time.



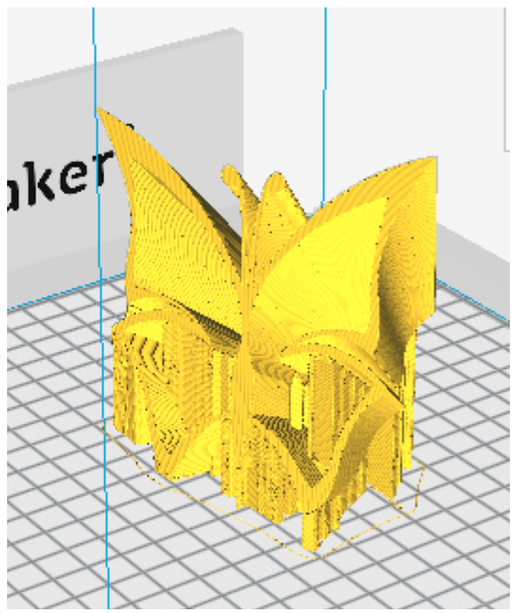
SETTINGS

Supports needed to hold in place cantilevered parts of the model. Sometimes it‘s possible to avoid supports by choosing the different model orientation.





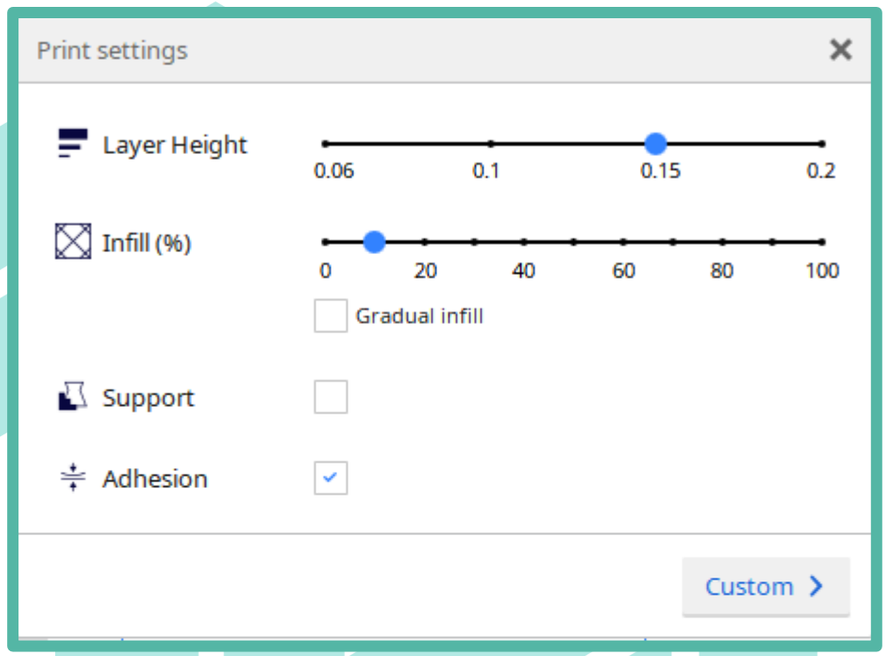
Object does not need supports

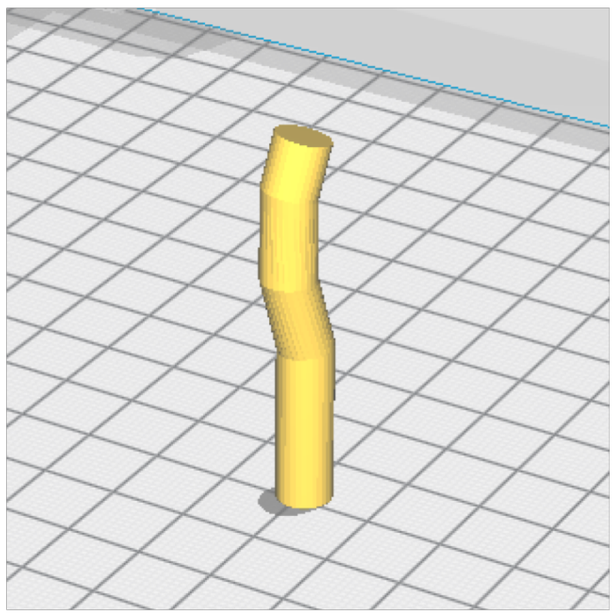


Same object in different position needs supports to print properly

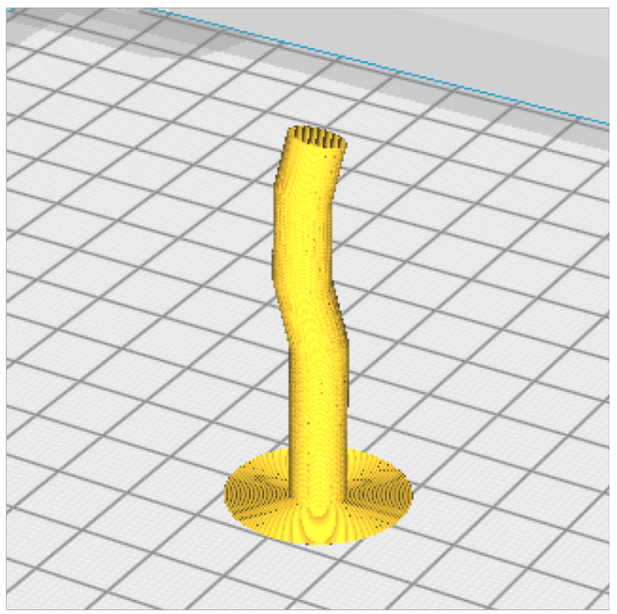
SETTINGS

Built Plate Adhesion option is useful when printing object which touches 3D printer build plate by very small area.



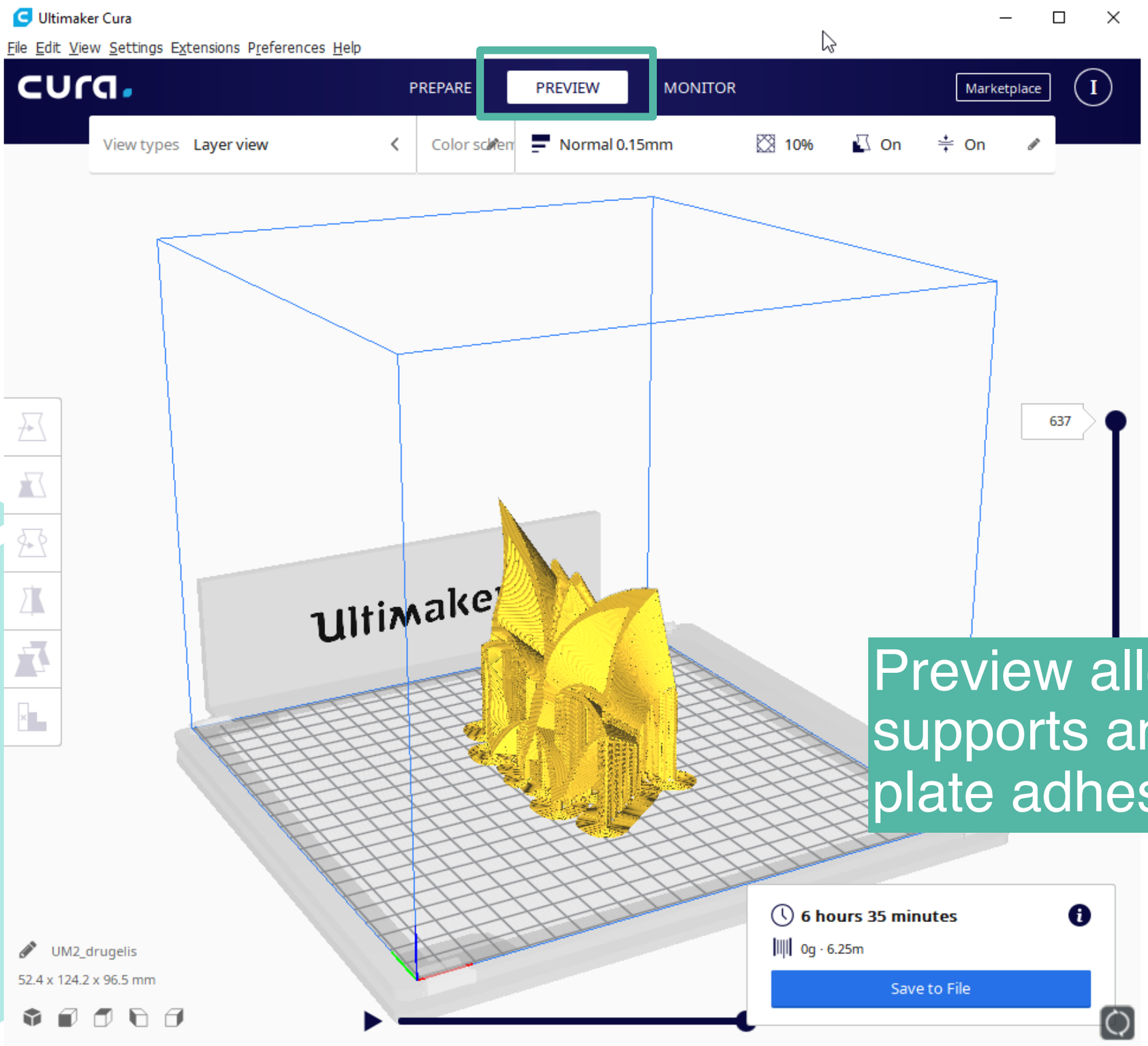


Object with small base area



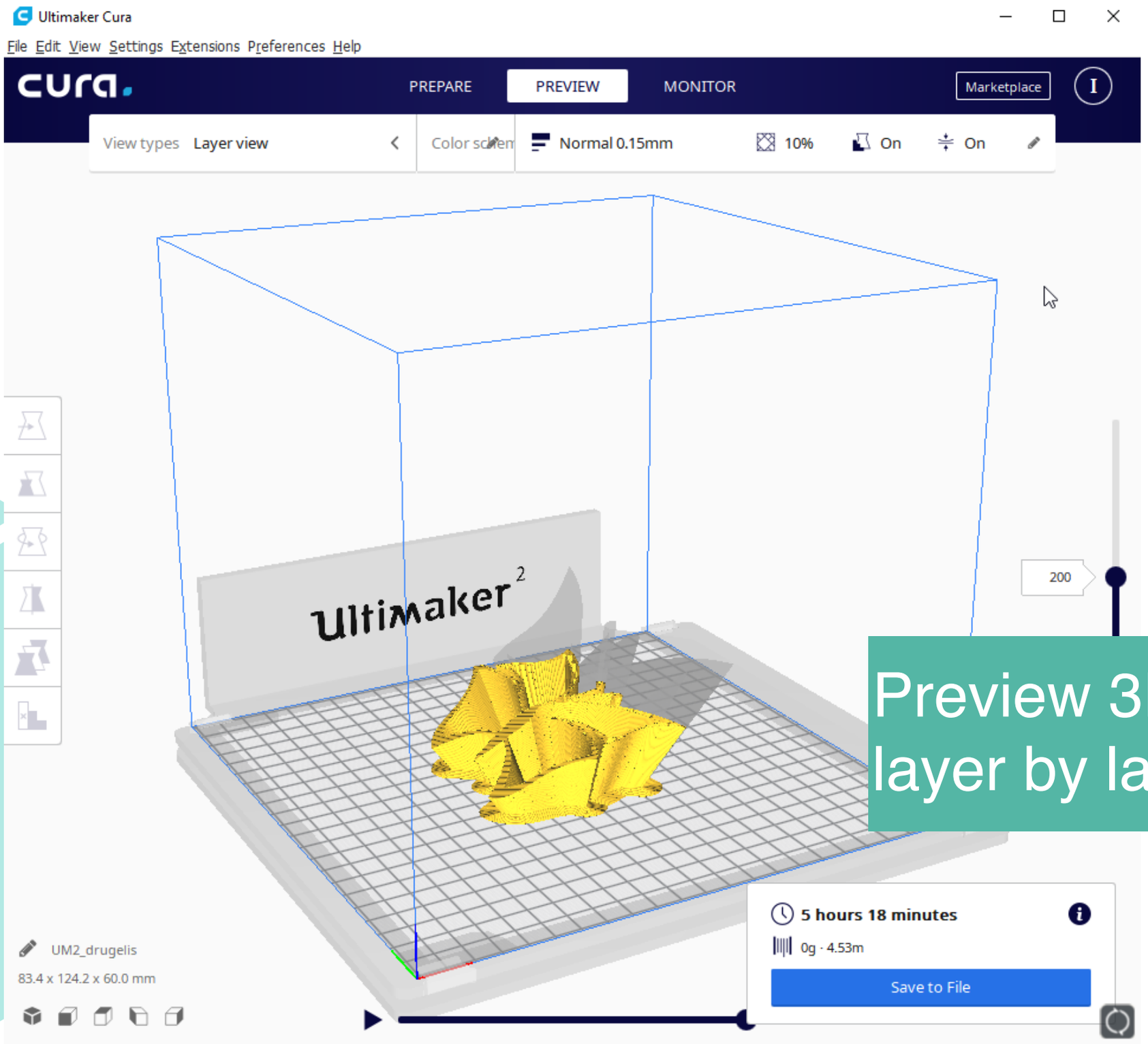
Generated layer to improve adhesion to buildplate

PREVIEW



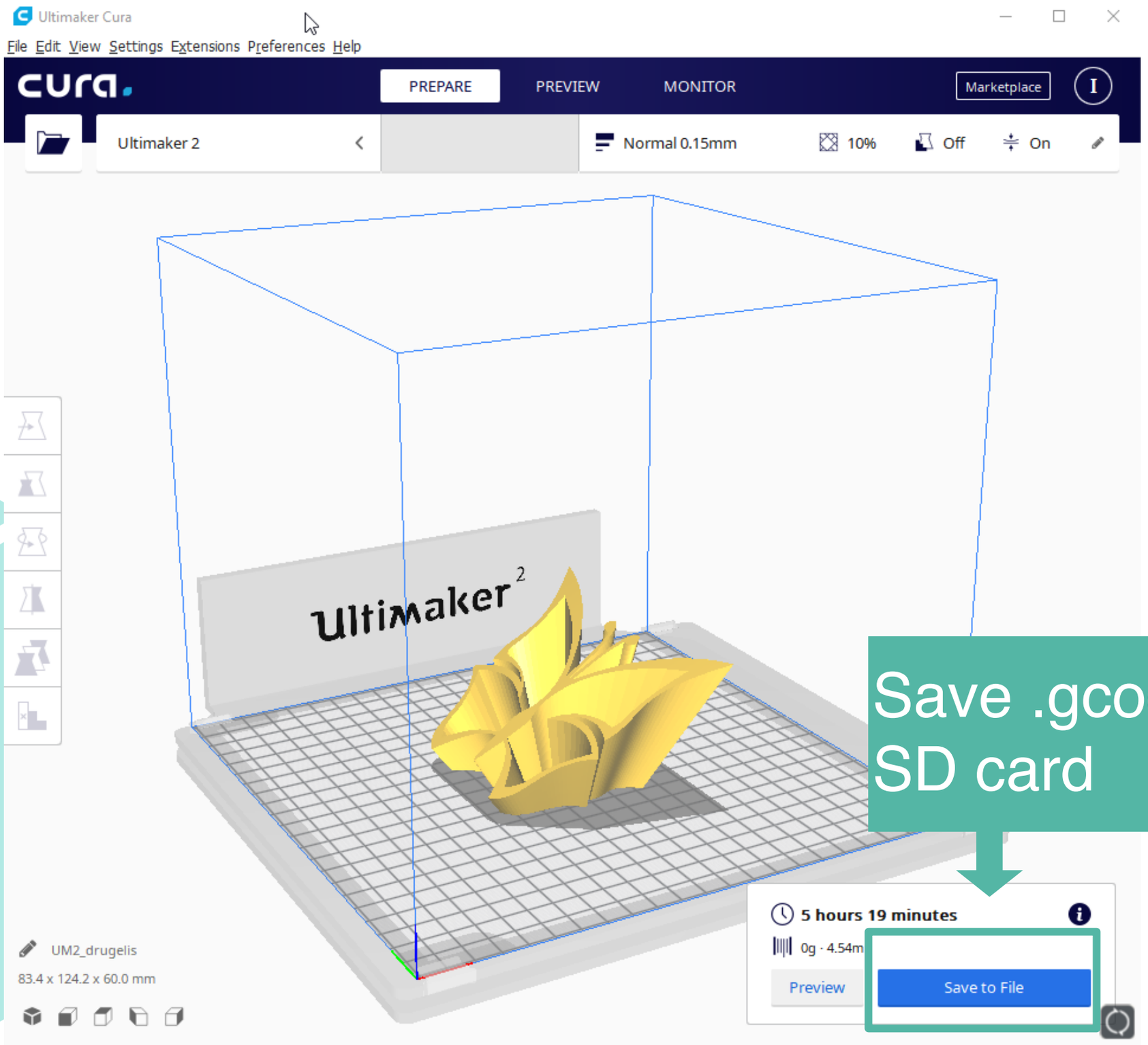
Preview allows to see supports and build plate adhesion layer

PREVIEW



Preview 3D print layer by layer

SENDING TO 3D PRINTER



SENDING TO 3D PRINTER

The best method to send a file to 3D printer depens on 3D printer model. It could be writing the .gcode file to SD card or USB drive inserting it into 3D or sending a file through wi-fi.

There is also an option to connect Ultimaker 3D printer via USB plug and print directly from CURA, but this option is not recomended, because any error at computer side or disconnection will end 3D printing.

FURTHER LEARNING

Deeper about CURA

https://ultimaker.com/en/resources/tips-tricks/software

TASKS FOR REFLECTIONS

Tasks:

- Export model from modelling software into .stl file
- Import file into CURA
- Choose best object orientation
- Choose 3D printing settings
- Export .gcode file to SD card

HEI MAKERS

www.heimakers.eu