



Print a 3D .stl file



1. Open Prusa Slicer



2. Import one or more files: File >

Importer STL/3MF/STEP/OBJ/AMF...

3. Choose the Prusa physical printer you wish to use

Imprimante :

MK3S Orange * Prusa MK3S - EssuyageOFF

4. OPTION: Add supports to print shapes and layers in the void

Supports: Supports sur le plateau uniquement

5. OPTION: Change the size at the bottom right of the screen :

Taille:

101

21,5

4

mm

6. The software will "slice" the 3D object, i.e. transform it into layers. To do this, at the bottom right of the screen, click on

7. Check the preview, the Filament used and the Estimated print time.

8. OPTION: Choose a bobbin from the cupboard and place it on the machine selected in (3.), removing the installed bobbin if necessary. To unload and replace a coil, use the button to select on the LCD screen :



, Then insert the wire and wait for it to come..

9. On Prusa Slicer click on

