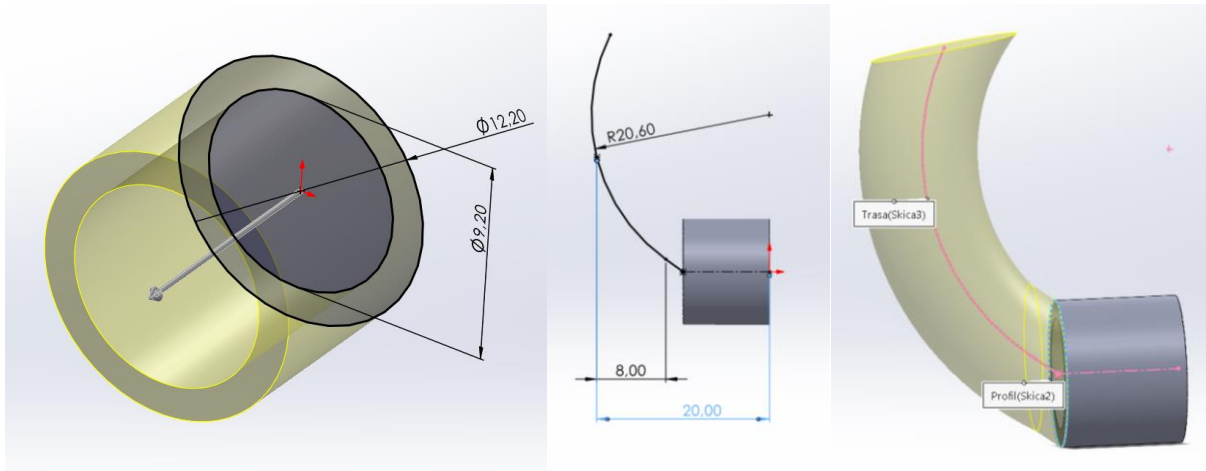
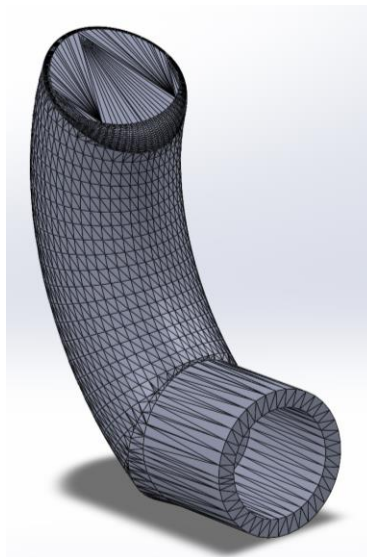


In this task, we will prepare a model in Solidworks, which we will then print using a filament 3D printer.

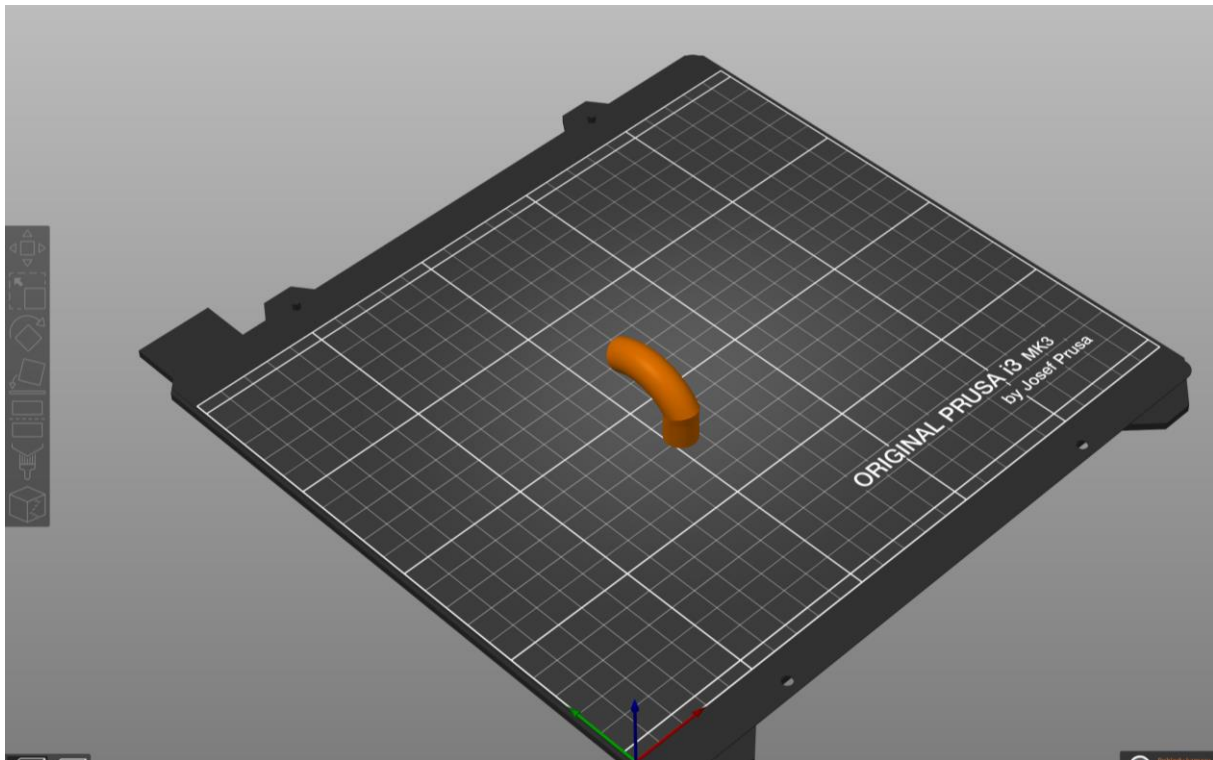
1. We will create the part for printing. Create a new part in Solidworks, according to the drawing. This is a part for our linear dispenser, specifically the hook holding the syringe.



2. For 3D printing, parts are exported in .stl format. This format converts the model into polygons, triangles, which are easier to understand for 3D printers. To export to .stl, save the file as .stl:



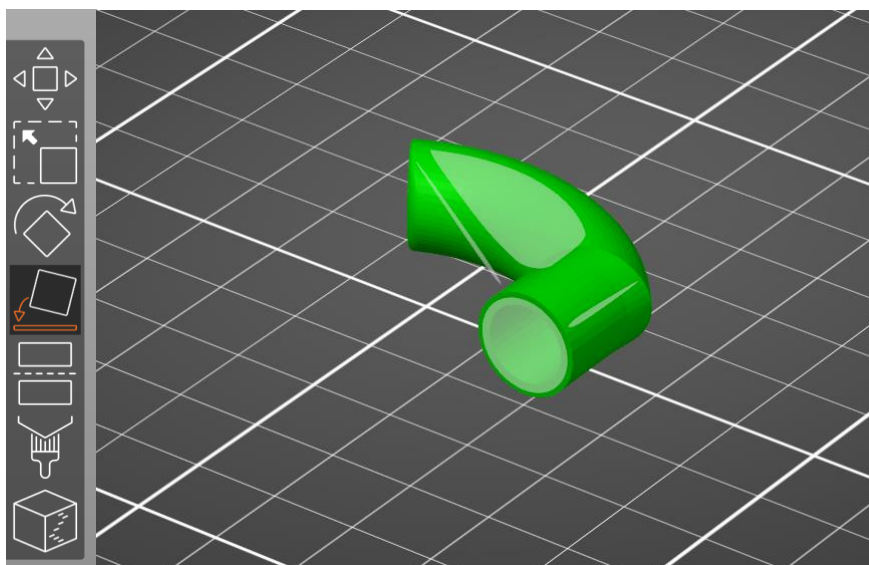
3. Subsequently, the .stl file is imported into a special program called a "slicer", which cuts the model into individual print layers according to the set parameters. We will work with PrusaSlicer software. Import the created part in .stl format.



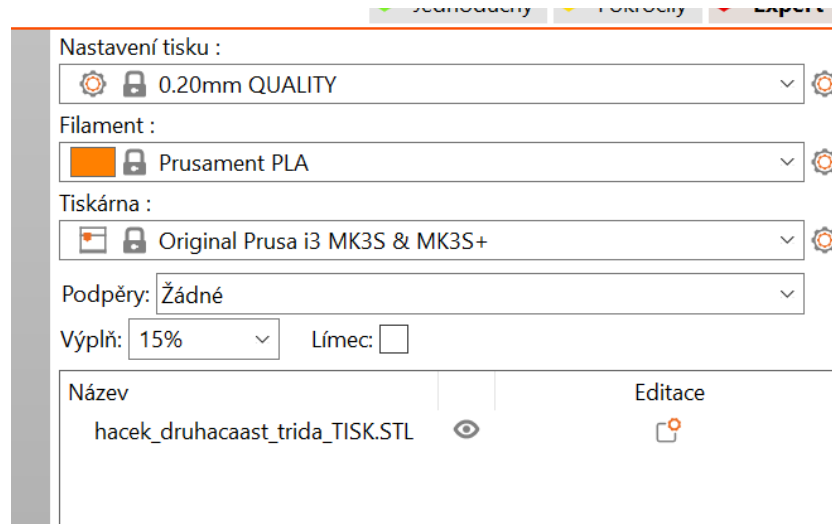
Tisk na filamentové tiskárně

Printing on a filament printer

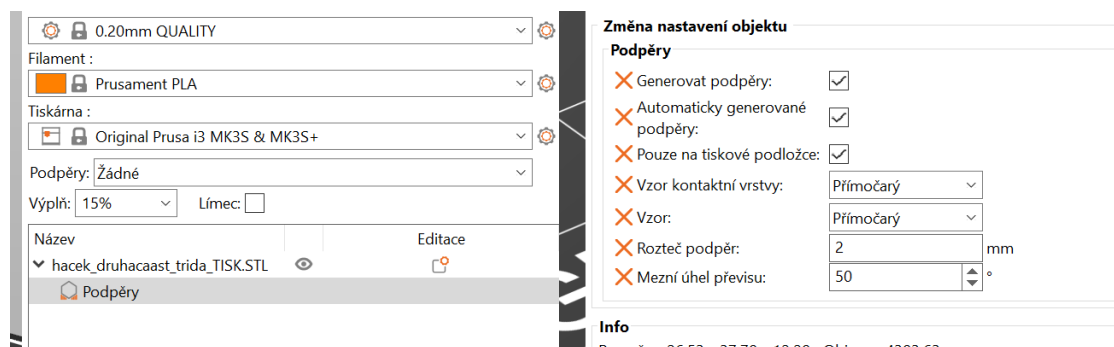
1. Think carefully about the direction of laying the part on the surface. Remember that the part will be printed from the bottom to the top. The position of the part lying down seems to be the most suitable.



- The creation of supports is also important. Supports are structurally weaker parts of 3D printing that are used to support parts of parts where overhangs occur. Supports can be generated automatically or manually in PrusaSlicer's advanced mode.

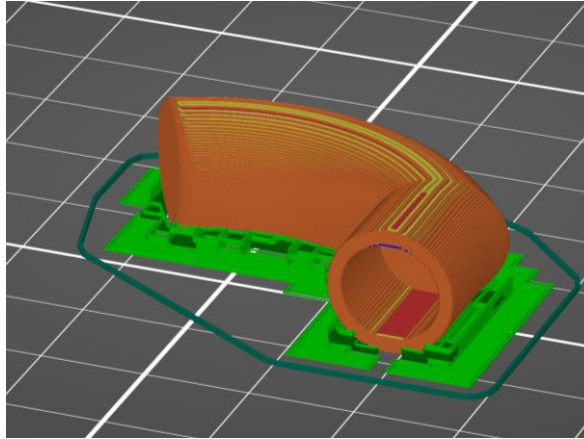


- Correctly set the type of printer and the type of filament we will use for printing - Original Prusa i3 MK3S+ and Prusament PLA type filament. Set the cutting height to 0.20 mm, type QUALITY.



- If you have thoroughly thought out the position of the part and set the supports, click "Slice". This converts the model into a print tip trajectory form. The horizontal and vertical sliders on the sides allow you to view the individual layers of the print. In the upper left corner you can see the percentage representation of individual print types. Pay particular attention to the amount of overhanging parts - they may not always print correctly. Hollow parts of parts where there are supports are also problematic, as such supports can be difficult to break out after printing.

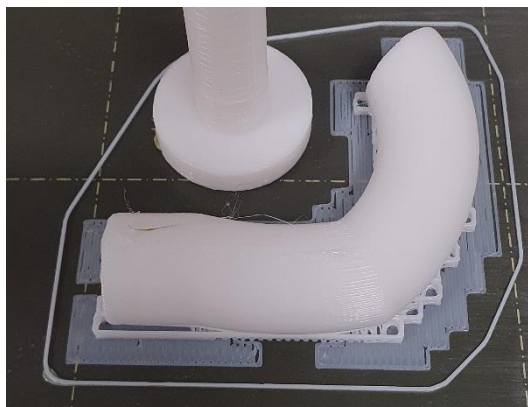
5. Export the resulting part in the form of .gcode to a memory card and insert it into the switched on filament printer.



6. Find the part in the menu and print it.



7. After printing and after the printing surface has cooled, you can remove the surface and remove the part from the surface. Next comes the part of breaking out the supports and adjusting the imperfections of the model.

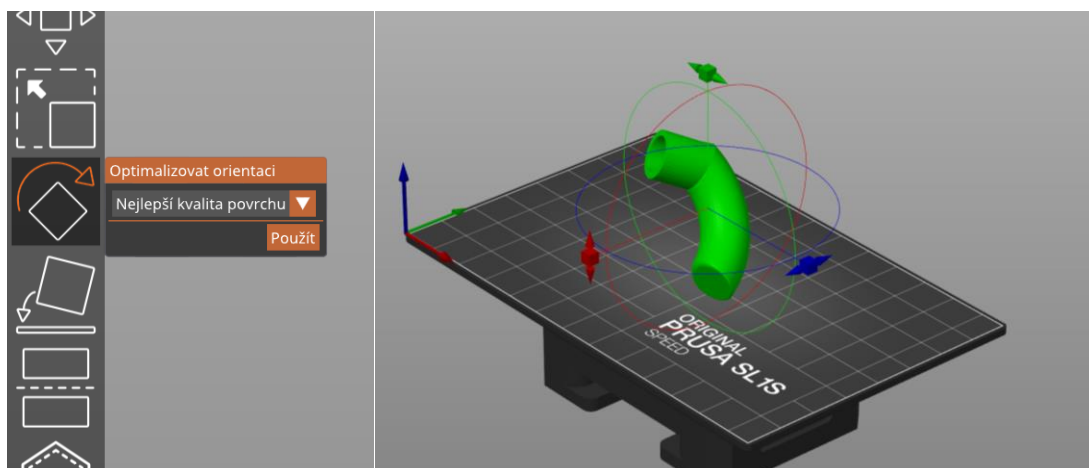


Printing on a 3D resin printer

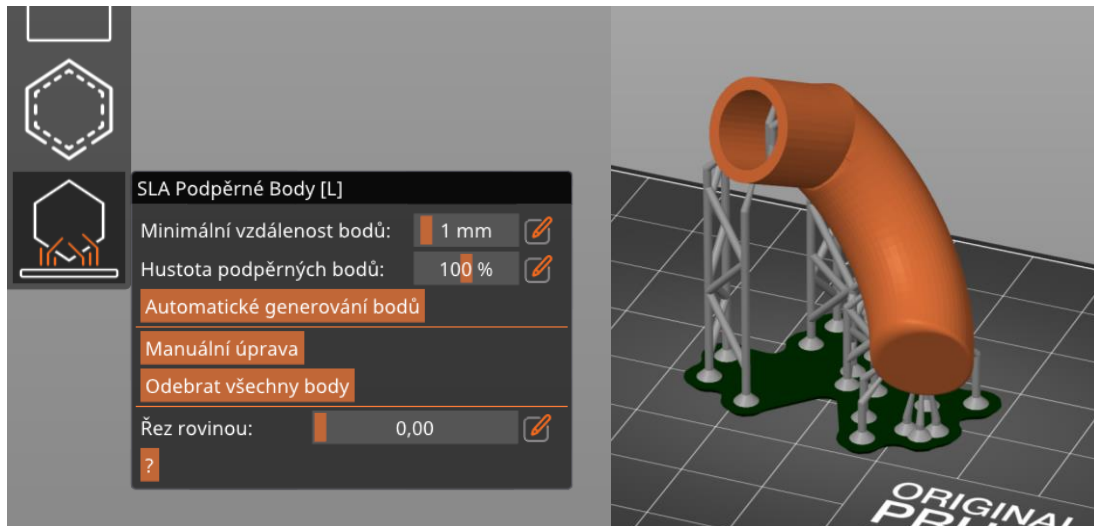
1. Resin 3D printer uses a different method of 3D printing. With the help of lighting up the LED screen in individual sections of the model, the entire layer of the model is cured in one moment. So the print speed does is not affectes by the amount of the model in the section, only in the total height. This makes printing with this printer faster and, thanks to the resolution of the LED screen, it is more detailed than printing on a filament 3D printer.



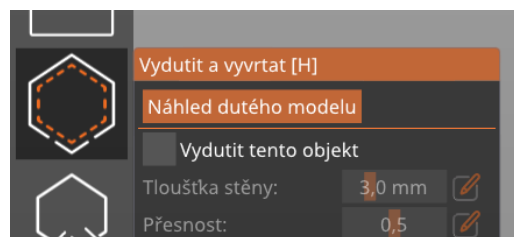
2. Load your created part into PrusaSlicer again. Choose the right printer and the type of resin you will use - Prusa SL1S SPEED and Prusa Orange Tough resin.
3. During preparation, it is necessary to pay attention to the fact that the resin printer does not print well large flat surfaces cured in one moment. In this case, the resin accumulates and the model curls. It is therefore good to tilt the part.



4. Parts also work with supports. The supports are generated under the entire model, so that the part does not lie directly on the printer bed. Again, it is possible to create your own supports.



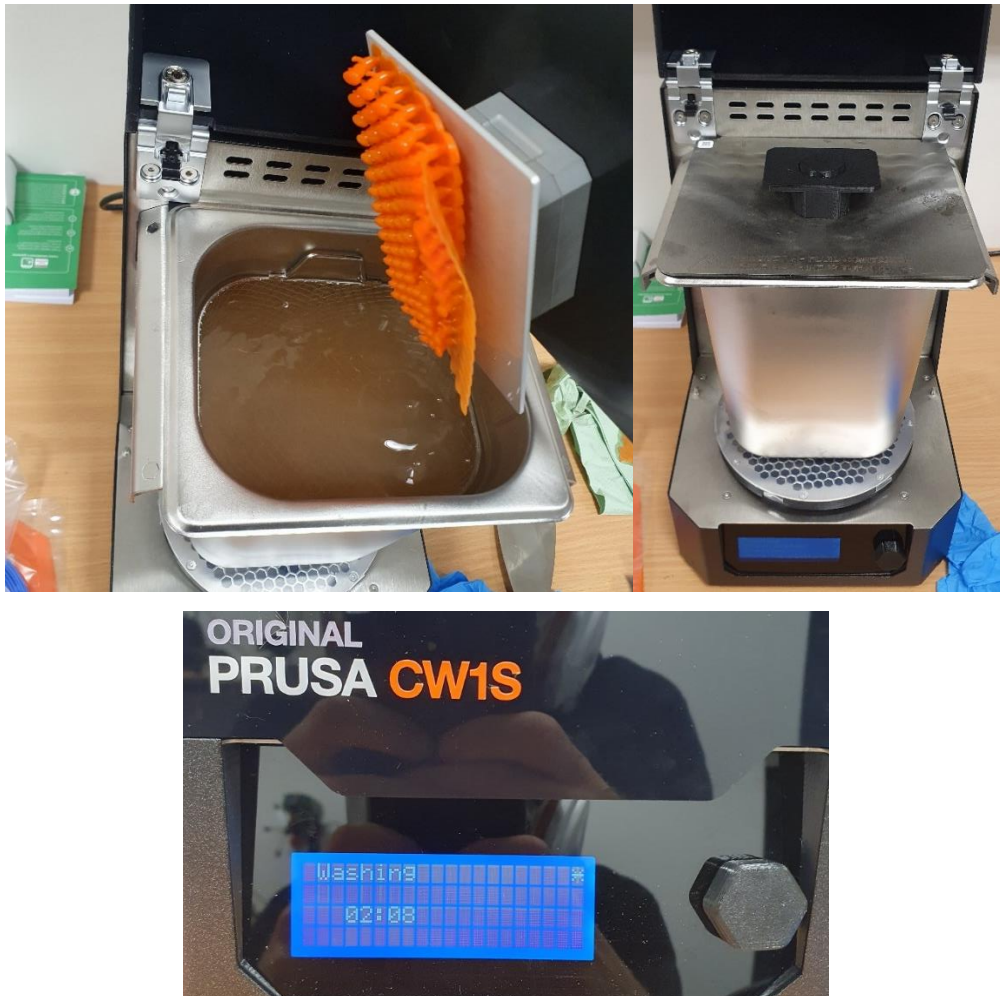
5. Hollow printing is used for printing on a resin printer. Printing filled models would be materially inefficient and deformations of the model would occur. Creating a hollow model can also be done during model preparation in PrusaSlicer.



6. After exporting the code to a flash drive, insert the drive into the printer. The printer itself will indicate how much resin should be poured into the tray. Wait for the model to print.
7. **WORK IN A WELL-VENTILATED ROOM AND ALWAYS WEAR NITRILE GLOVES. RESIN IS VERY TOXIC!**



8. After printing, the resin must be cleaned and cured. The hardening and cleaning stations are used for this. Place the model in the cleaning station so that it is completely submerged in the IPA liquid. You can either scrape the model off the mat and break the supports before cleaning, or leave it on the mat for cleaning as well. There must be a magnetic stirrer at the bottom of the container. Run the printer cleaning cycle.



9. Subsequently, remove the container and the part and place the part in the station without the container, directly on the pad. This is followed by a drying cycle and a subsequent curing cycle under UV light. This hardens the resin and should no longer be toxic to humans. The model is now complete.

