# Molding and Casting[link]

As with the previous guide, this is a general explanation of the designing process for a two part (positive) mold. This design will be machined onto a wax block, which will allow you to create a flexible (negative) mold. This guide was developed using FreeCAD, and it does not include the designing process for the part, you should already have an stl file with the part you want to convert to a mold.

First, create a new FreeCAD project and import your stl file.

Note: Remember that the stl object must have an appropriate size for the wax, the milling machine and the end mills that you will be using for the process. This mold must have walls, so you can't use all of the stock material. Also. you can't go too deep on the stock because you may hit the bed of the machine, or hit the stock with the collet. Also, be aware of the height of the tool (including flutes and shank).
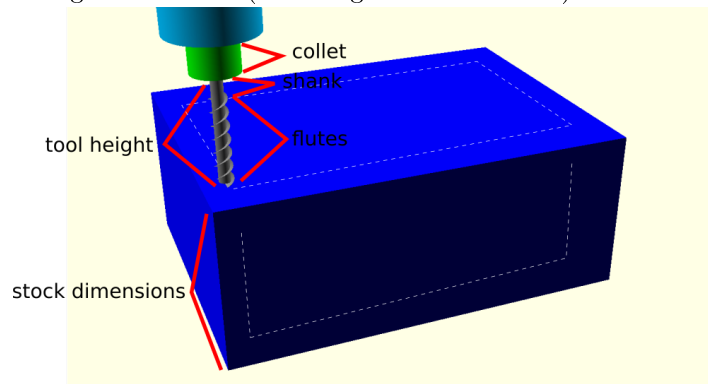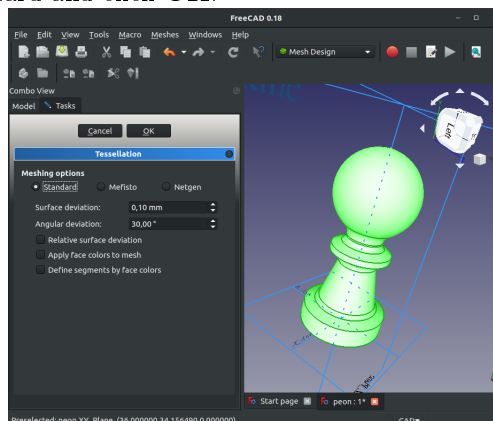

For this example, I will use a chess pawn model. It was created in FreeCAD. Its a simple profile and the tool "revolution" was applied to it:
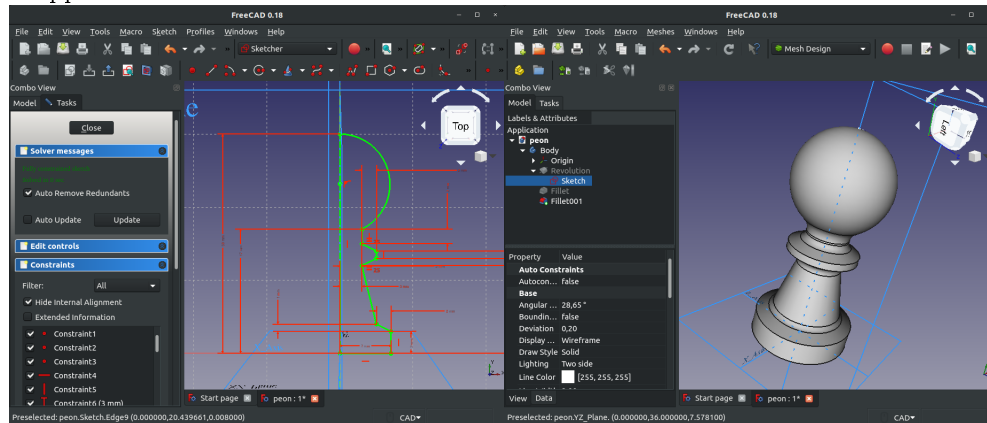

To export a model to stl in FreeCAD, switch to **Mesh design** workbench, then select the body you want to export and go to **Meshes > Create mesh from shape...**

In meshing options, select standard and click **OK**.

Now create a new project and import the model with **File > import**. Then Create a new body and a sketch.

In this case, the YZ_Plane will be used for designing the mold. The pawn for this example is symmetric along the Z axis, so the mold will be designed using this axis, and the two parts of the mold will look the same.
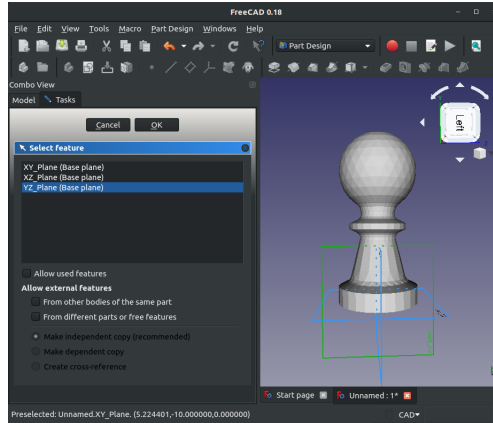

Draw the outline of the stock around the model, keep good track of the space around the model. Also, the plane of this sketch should be exactly placed so it crosses the model in the middle:
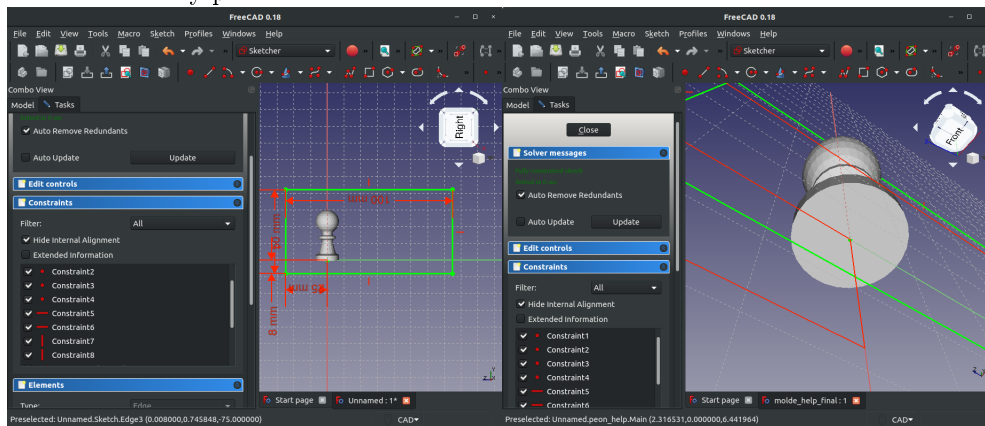

Now copy and paste the model, and move the copy to the right (keeping symmetry).
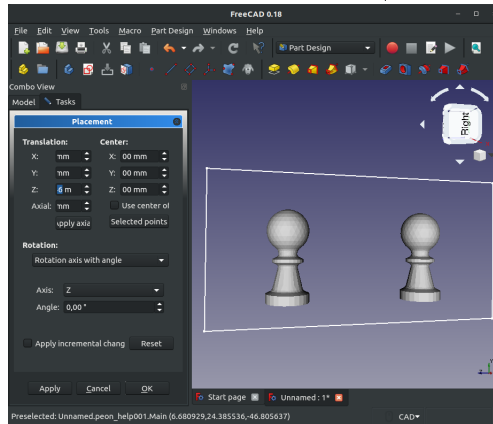
Then select the previous sketch and create a pad. In order to generate a pad around the models, select the **two dimensions** option in the **Pad paramenters**, and apply an appropriate length on each direction.
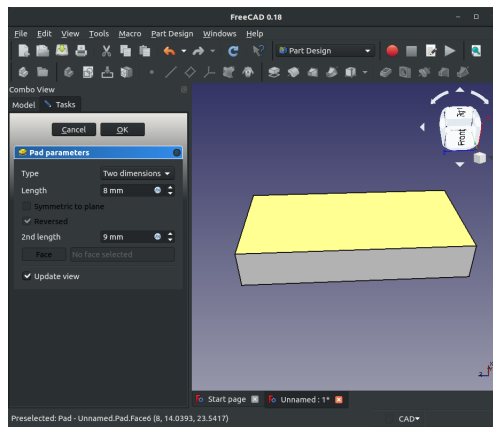
Now select the top face and create a new sketch. Create two rectangles, making sure the model is in the center of each rectangle. Then apply the pocket operation, making sure you uncover exactly half of the model.
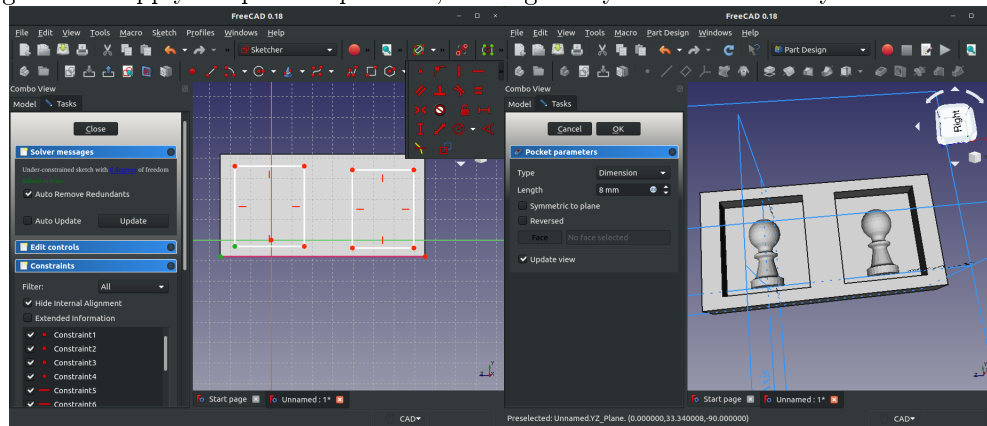

Now the mold needs some register marks to align the two parts. We will use half spheres for that. Go to **Part** workbench and add one sphere.
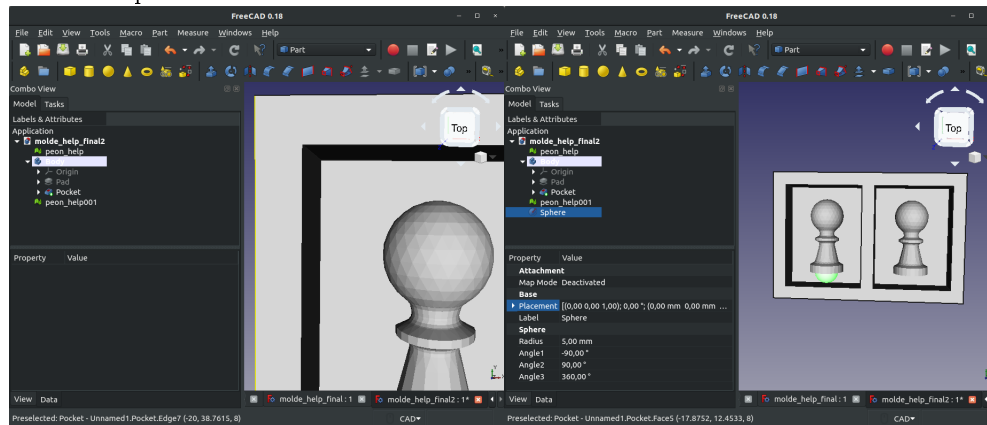
In the control panel on the left (in the **Data** tab) change the radius and modify the position of the sphere. Note that exactly half of the sphere should be "inside" in the stock. Move the sphere to a place near the corner, then create three more and place them near the other corners. Remember to always keep the symmetry.
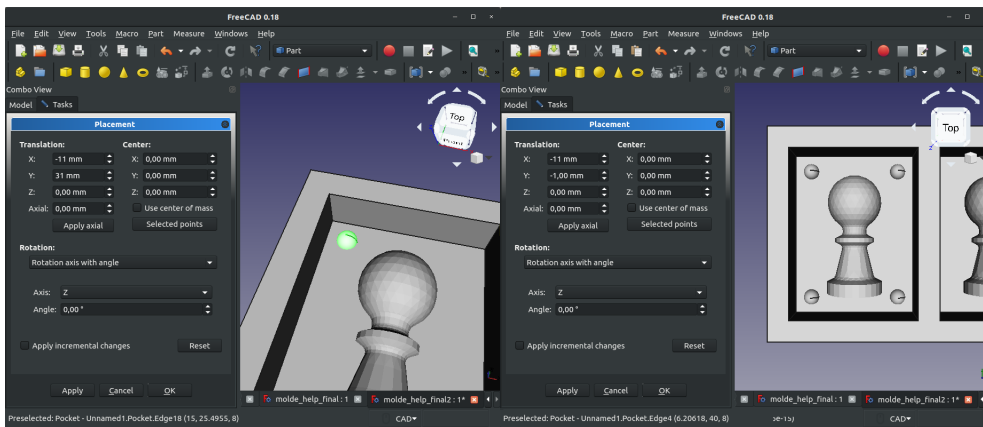
Now, from the project tree, select the four spheres and go to **Part** > **Boolean** > **Union**. Then copy and paste the Fusion object and move it to the other part of the mold (the positioning is very important because these marks must match).
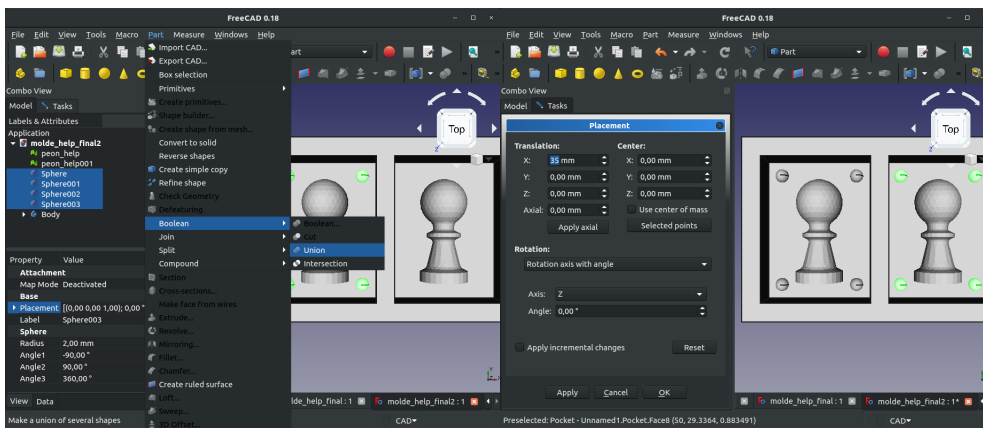
Finally, Select the body and the fusion object and go to **Part** > **Boolean** > **Cut**. The spheres will be subtracted from the stock and you'll have spherical holes on the other side of the mold.