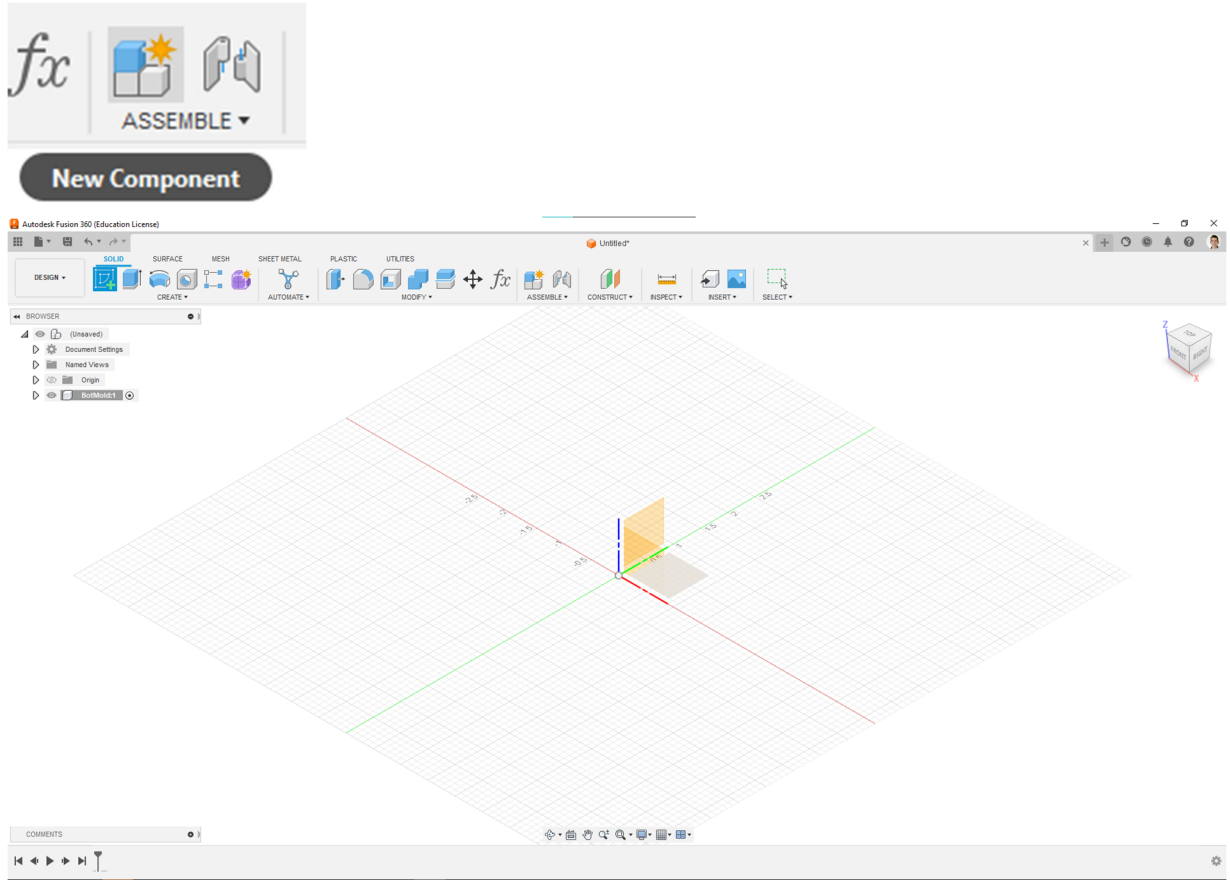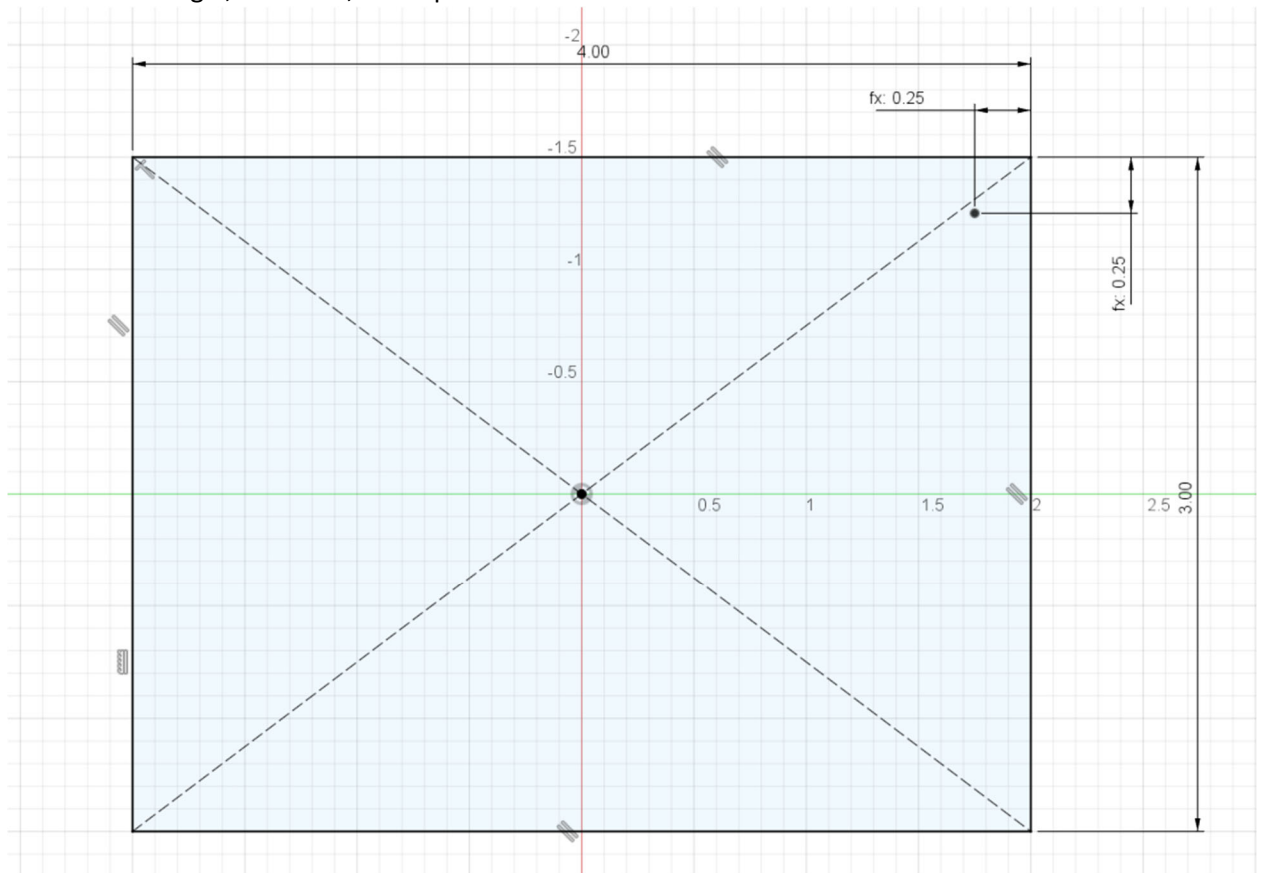# Teacher Instructions

1. It is best to start with a mold template to help students keep their lures at a manageable size. Begin a new Design, create a new Component, and start a sketch on the top plane.
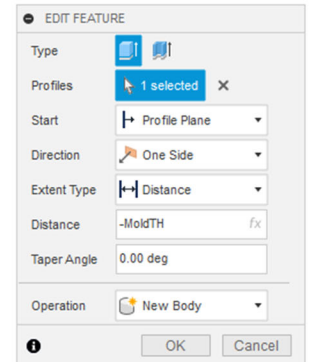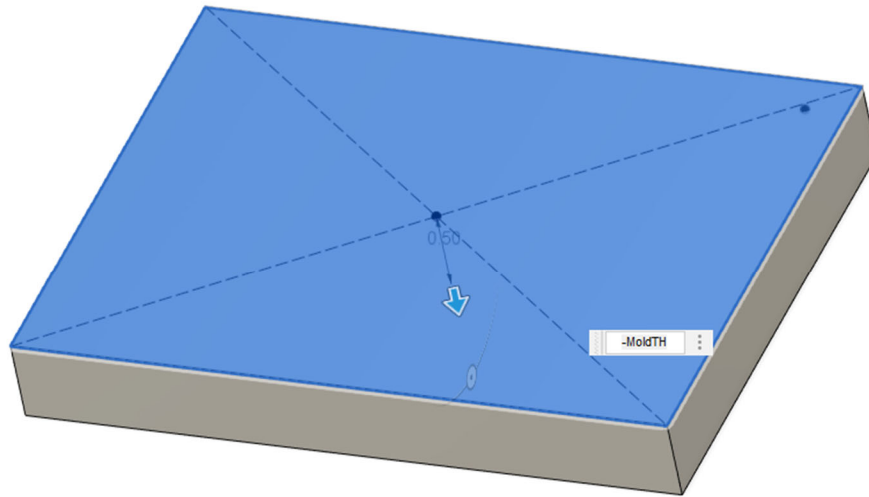
2. Create a rectangle, two lines, and a point as shown below.

3. Extrude the rectangle to a height of -1/2 inches. Make sure that the extrusion is going below the top plane.

4. Use the hole tool to create a through hole for the bolts. This model uses ¼-20 bolts, which can be readily found at most hardware stores. If needed, the diameter of the holes can be adjusted for other bolt diameters. Note that the hole diameter must be larger than the bolt diameter.

Note also that bolts must be longer than twice the mold thickness.
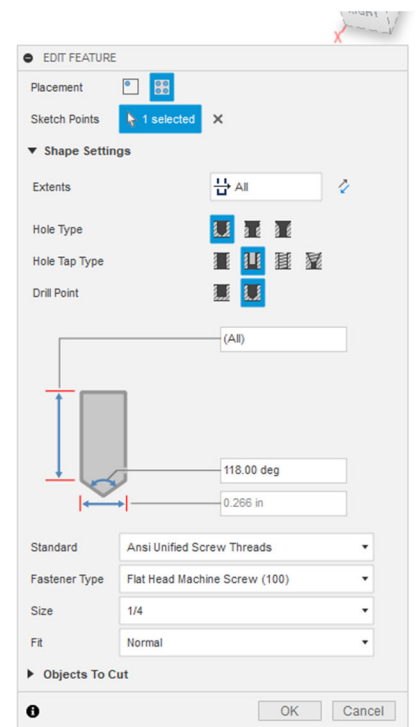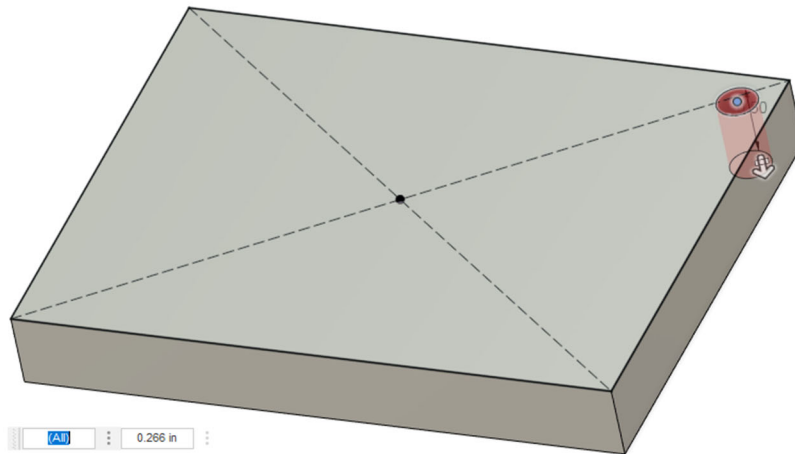
5. Create a rectangular pattern for the other three holes.
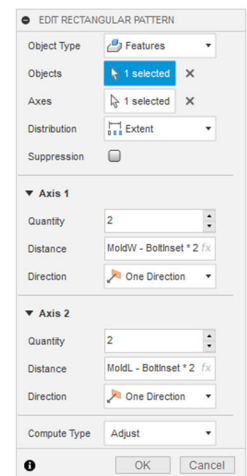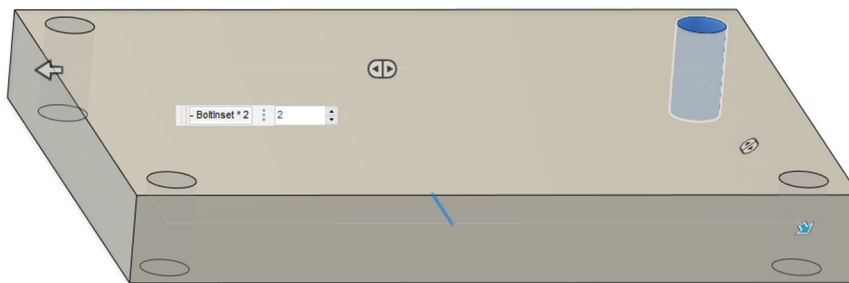
6. Right-click on the Component and select "Copy". Right-click in the space below this Component and select "Paste New". This will create a copy of the bottom of the mold, but without linking it to the original. This is so that edits can be made to the top of the mold.
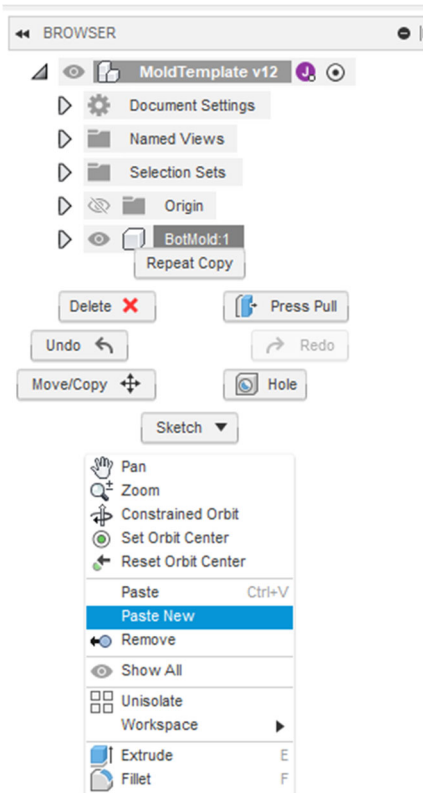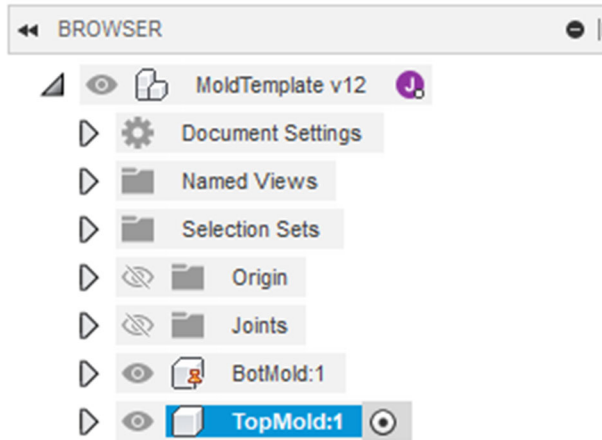

7. Rename the new Componet "TopMold" (or some other descriptive name).

8. Right-click on the bottom mold Component and select "Ground". This will fix the Component in space so that it cannot be moved.
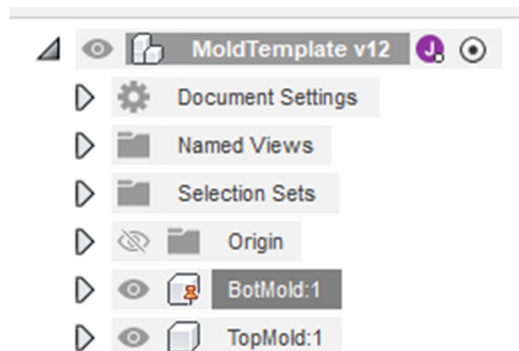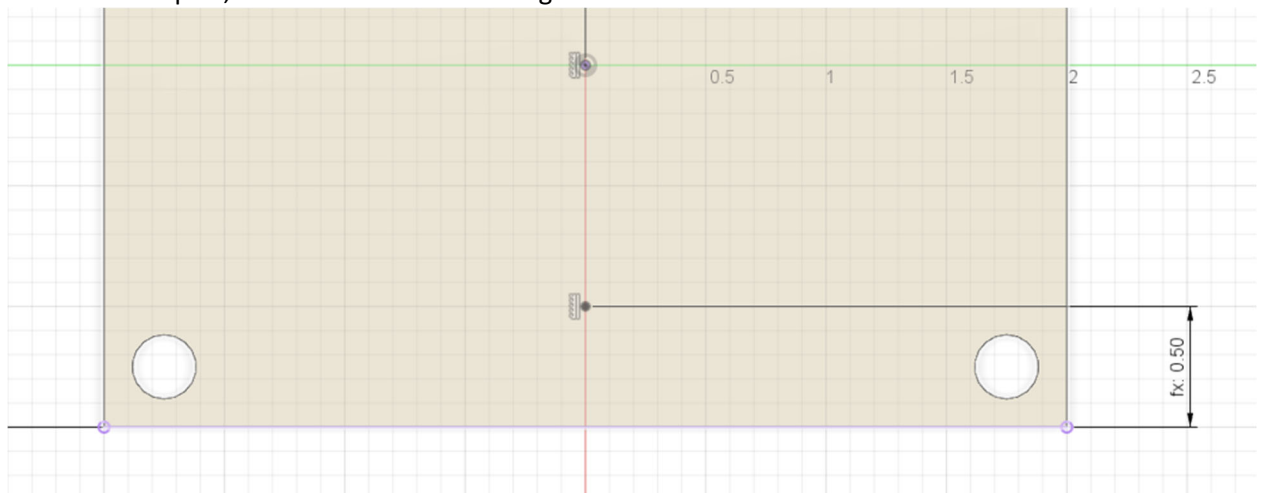
9. Hover over the name of the top mold Component and left-click on the radio button on the right. This activates the Component so that only it will receive the changes.


10. Create a sketch on the top face of the mold. This sketch only needs to have a single point, in the middle of the part, about ½ inch from the edge.

11. Create a hole at this location. The hole type should be countersink. The depth and angle of the hole will vary depending on the injector geometry that is used. Make sure to verify this with calipers and a protractor before printing the mold or create some sample parts to test the fit. Ensure that you are only cutting through the top of the mold by opening the Objects to Cut section and only selecting the top mold Component.
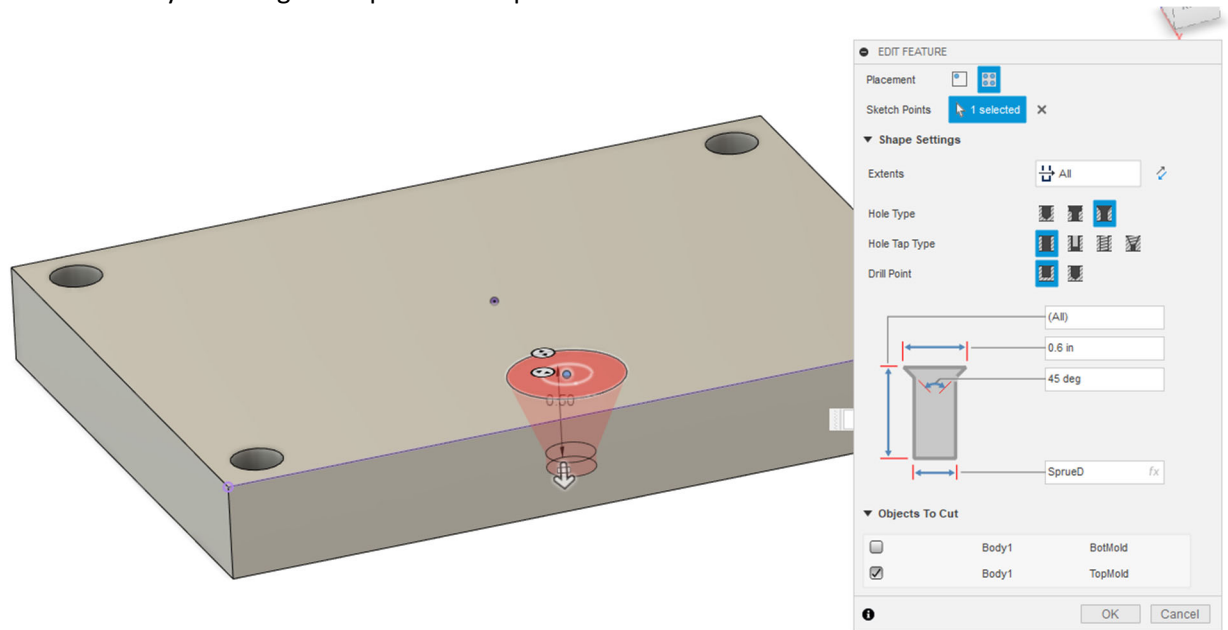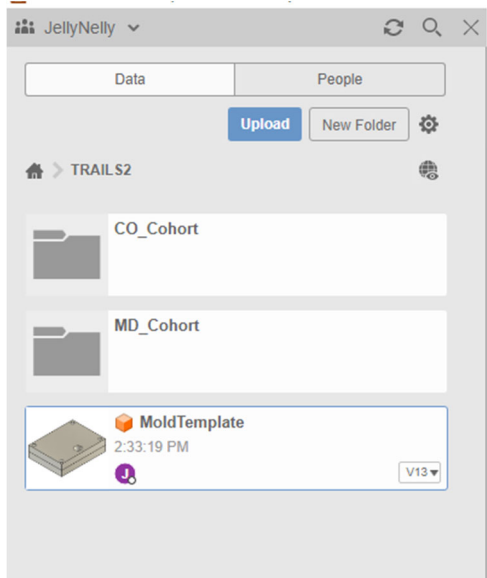

12. Use the joint command to create a rigid joint between the top and bottom halves of the mold. This will properly position them so that they are ready for use by students.
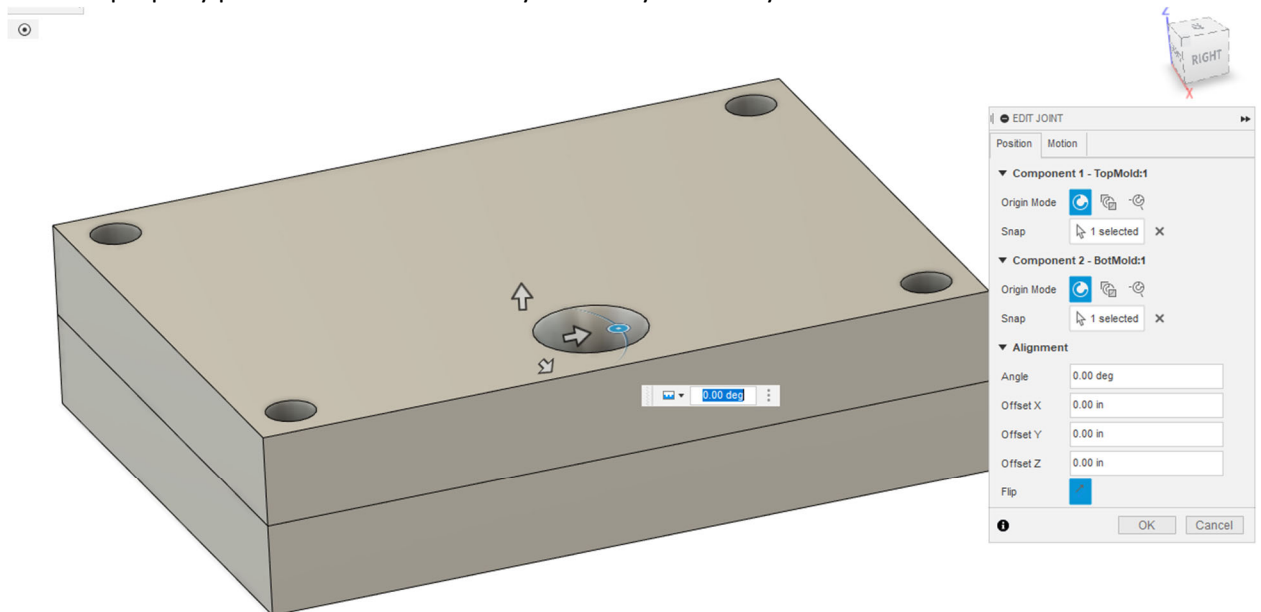
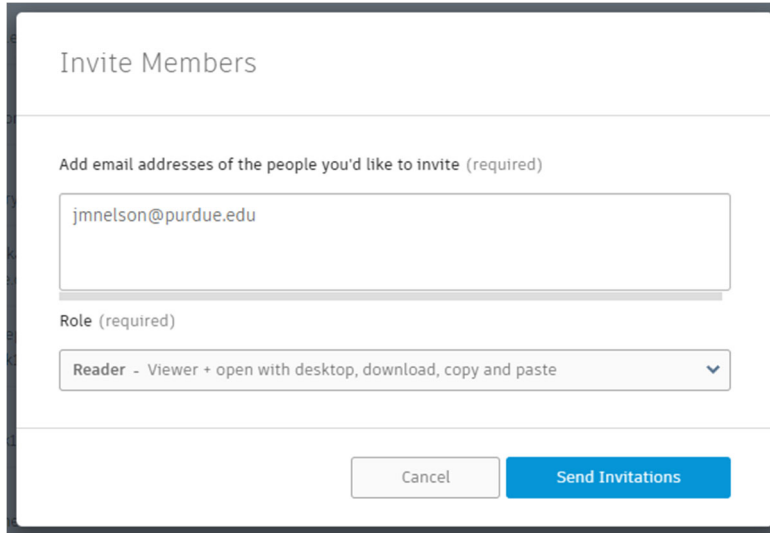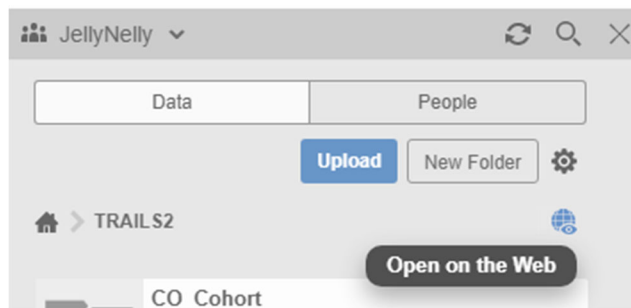13. Save the Design in a Project that you intend to share with your students.

14. Now the mold template is ready to share with your students. They will be able to copy the files and create a mold based on their lure design Click on the "Open in Web" icon to go to Fusion Team.

15. Make sure to invite all students to this Project. There is an Invite button at the top right of the screen. Input students' email addresses into this window. It is suggested that the Role be set to "Reader" to allow students to view, download, and copy files (but not edit them). Note that your school's IT department may need to get involved in order to allow students to receive emails from Autodesk and Fusion 360.

Invite Members

Add email addresses of the people you'd like to invite (required)

jmnelson@purdue.edu

Role (required)

Reader - Viewer + open with desktop, download, copy and paste

Cancel   **Send Invitations**

16. If needed, these permissions can be changed in Fusion Team.

17. Students should now be able to see this Project in their Fusion 360 Data Panel.

Student Instructions (starting with Step 18)

18. Students can open the template Design, save it to their own Project, and then insert their lure Design into the template by dragging and dropping from the Data Panel.
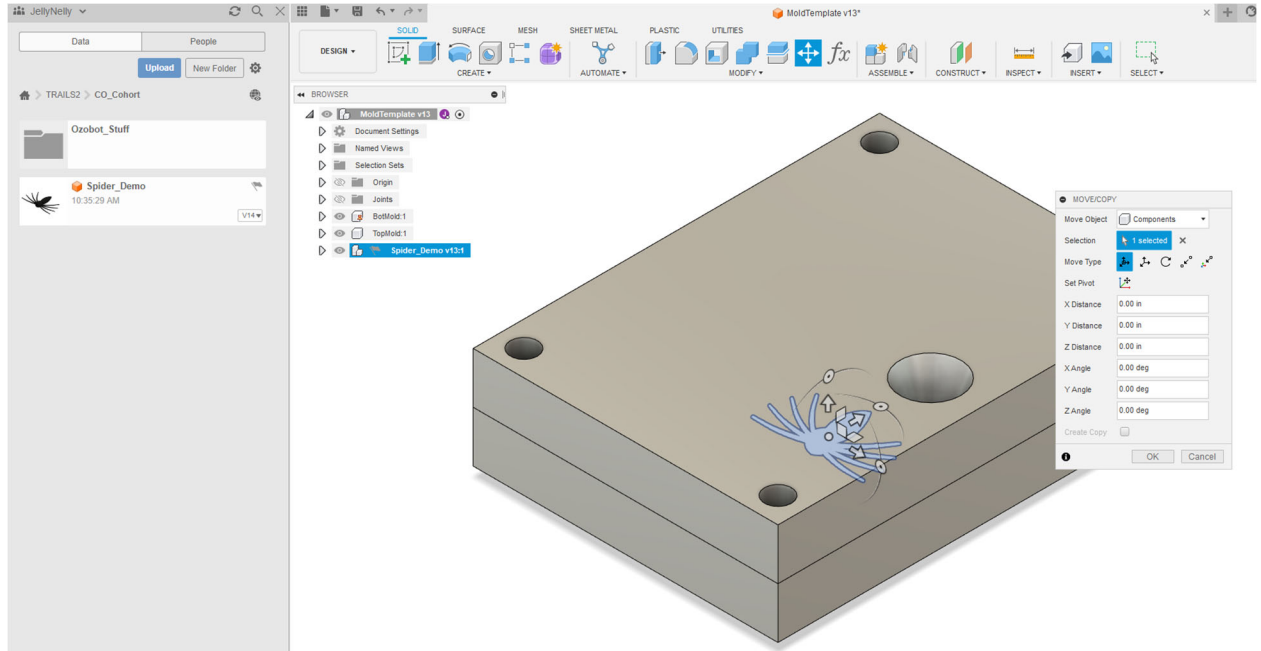

19. It is unlikely that the student lure Designs will be located in the correct place with respect to the mold. Probably the best thing to do is to align the origins (assuming the student Designs were created around the origin). Create a joint between the origin of the student Design and the top face of the bottom mold. Note that the visibility of the top mold Component was turned off to enable easy selection of its top face. The visibility of the origin of the lure was also turned on.
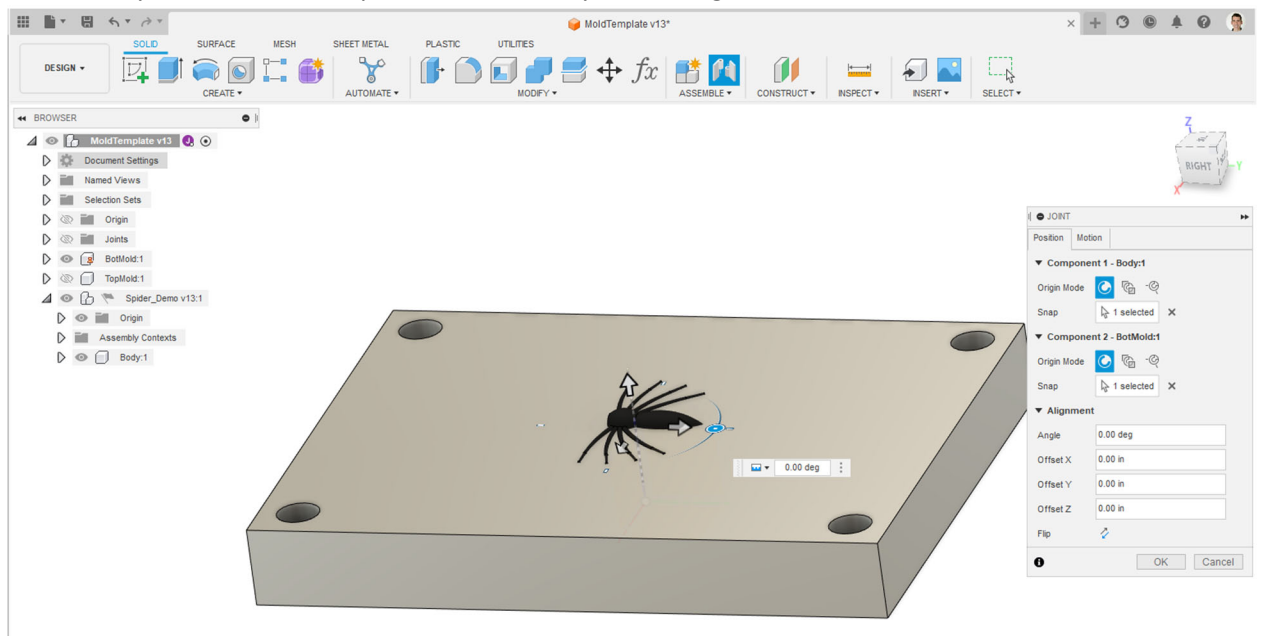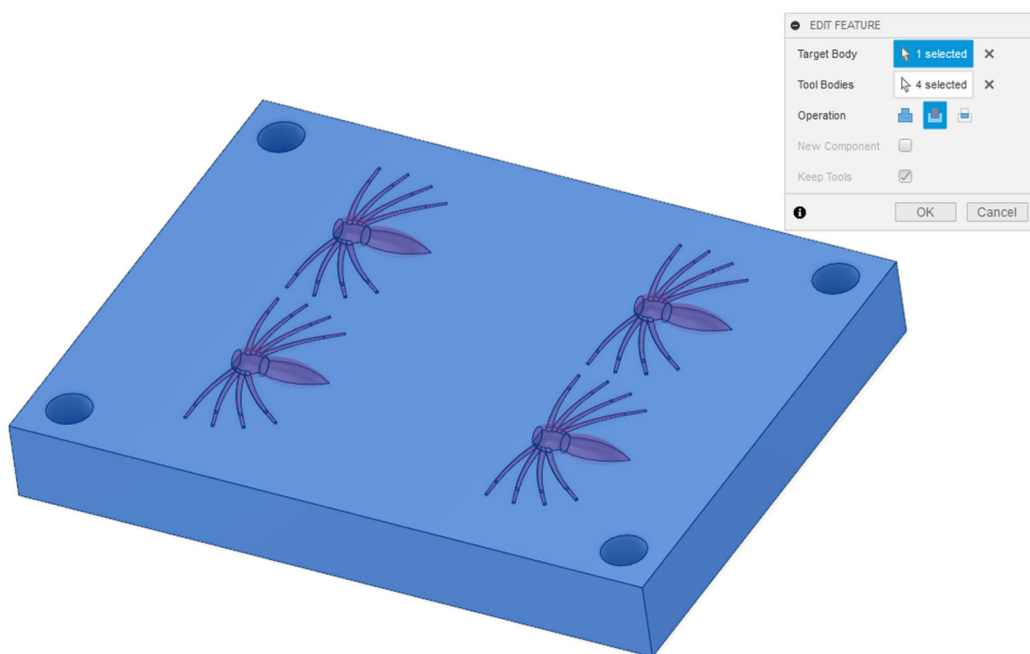
20. If more than one lure per mold is desired, position the lure accordingly (not at the center) and create a rectangular pattern.
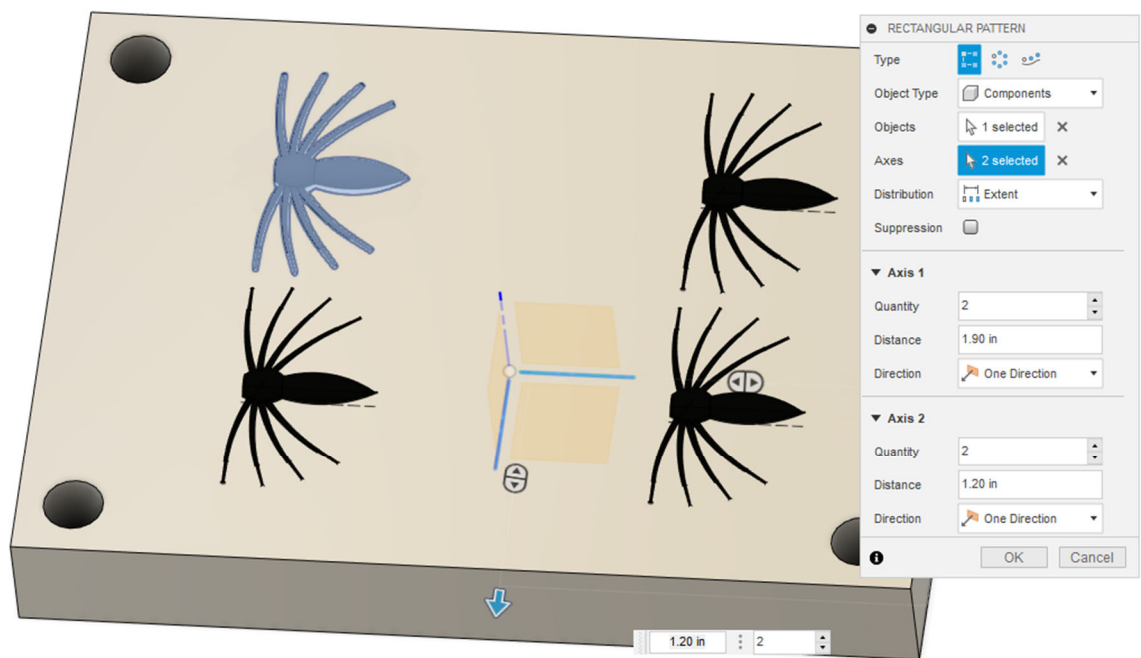
21. Use the Combine tool to cut negative spaces in the mold Components. Ensure to select the "Cut" option. This will need to be done for each mold half. It is easiest to disable the half of the mold that is not being used.

22. Hiding the lure Components reveals the result.
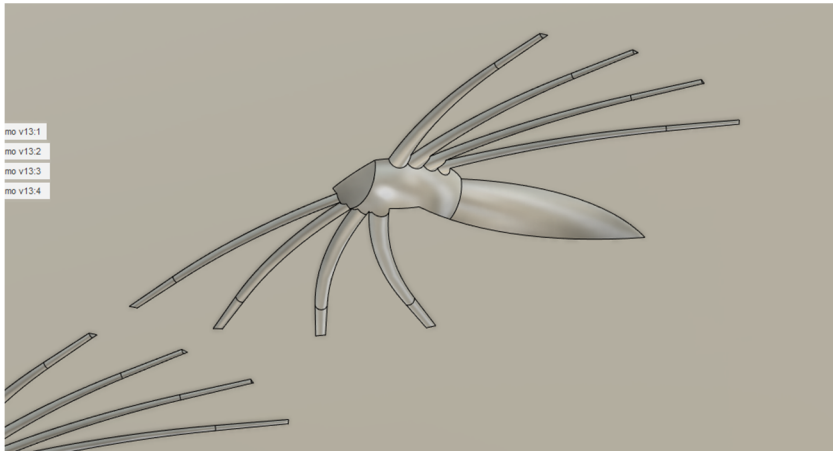

23. Repeat this for the top half of the mold.

24. There must be a path for the liquid plastic to flow from the injector to each of the mold cavities. This path is called a runner. Runners can be created in several ways, but one of the easiest is to create a circle somewhere in the middle of the cavity and then extrude it toward a central runner.

25. Create a sketch with a circle (or more than one) on the appropriate plane (the front plane in this example). Make the circles smaller than the lure bodies and locate them inline so that they will intersect when they are extruded.
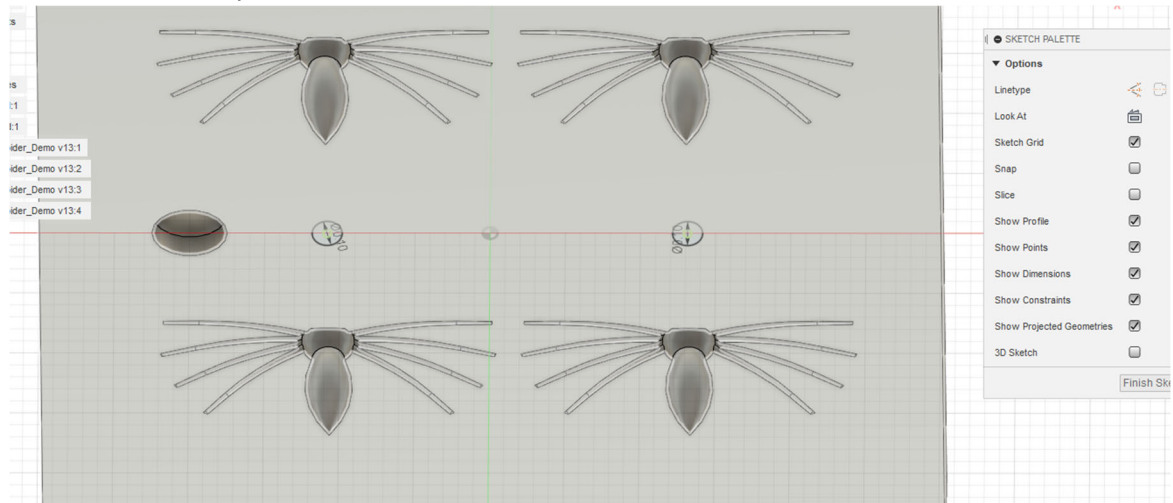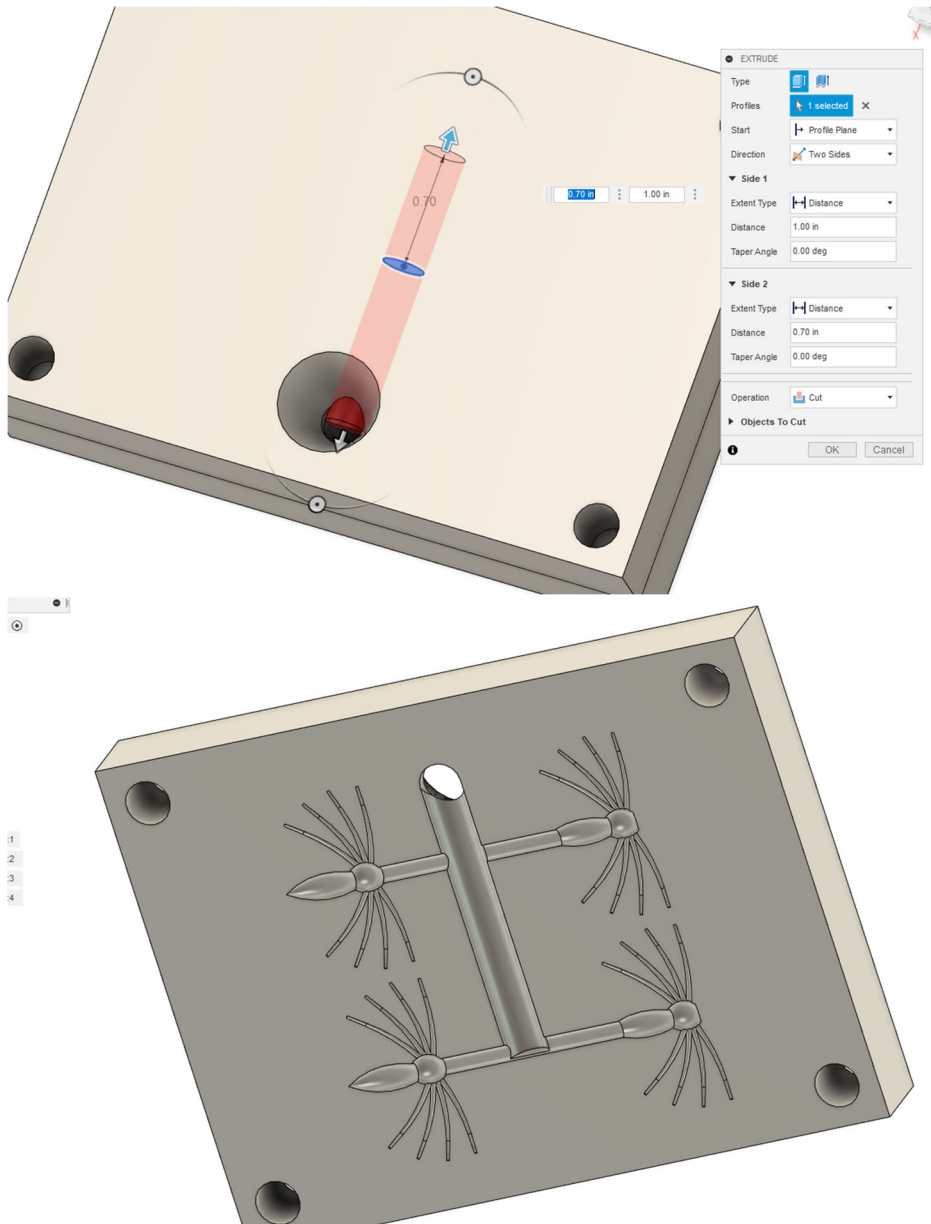
26. Make an extruded cut that extends all the way into the cavities. Ensure that both the top and bottom mold halves are cut.
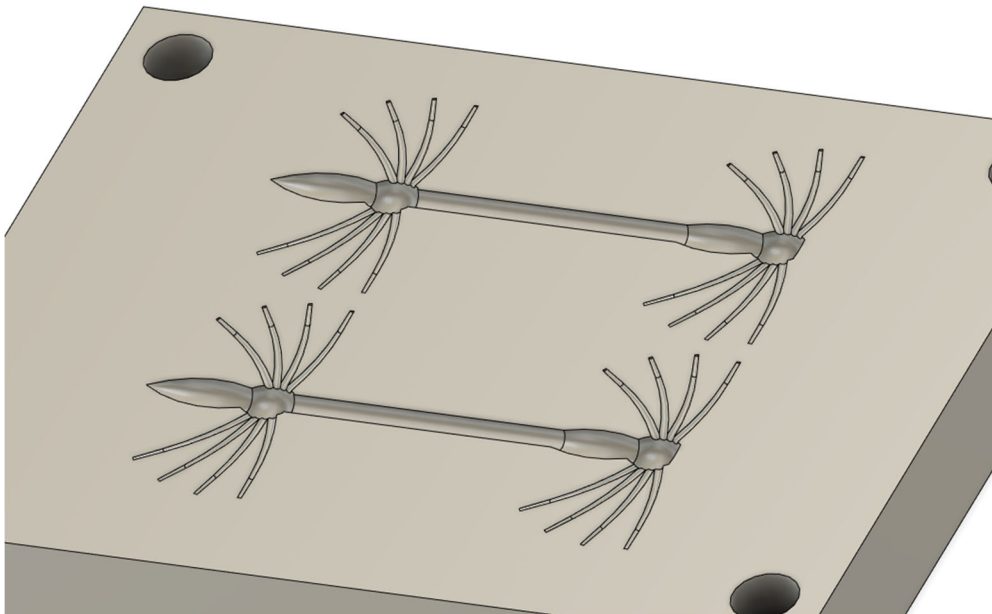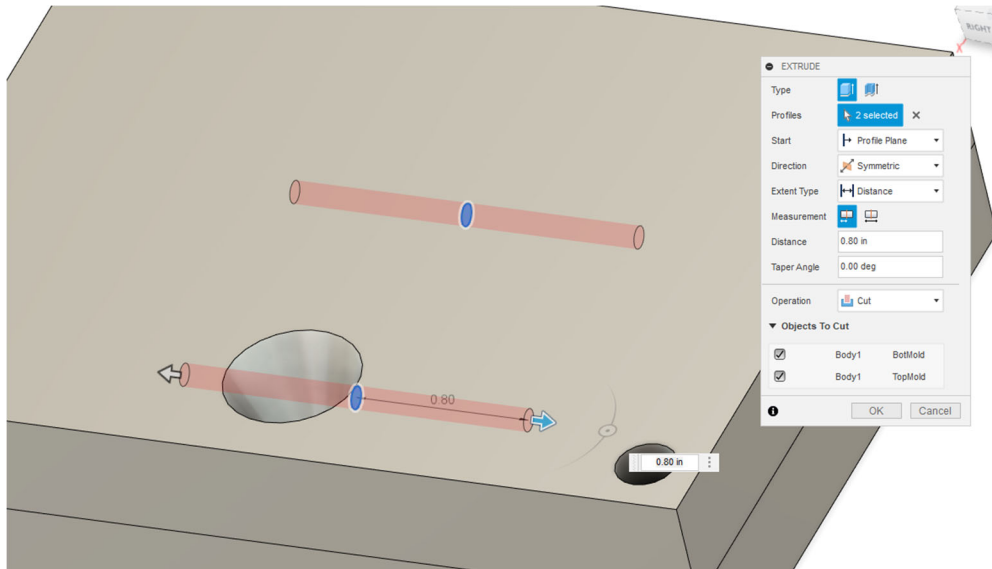
27. Next, do a similar cut to create a path from the sprue (where the plastic enters the mold) to each runner. This runner should extend from the other runners to the sprue. Once again, it needs to cut both mold halves.

28. Air can get trapped inside the mold when plastic is injected into it. The last step is to create vents through the top to minimize bubbles. Create a sketch on the bottom of the top mold face. Vents at the end of each leg, tail, wing, abdomen, etc. are suggested. Vents should be a similar size or smaller than the feature to which they are attached. Using a pattern again might be helpful.
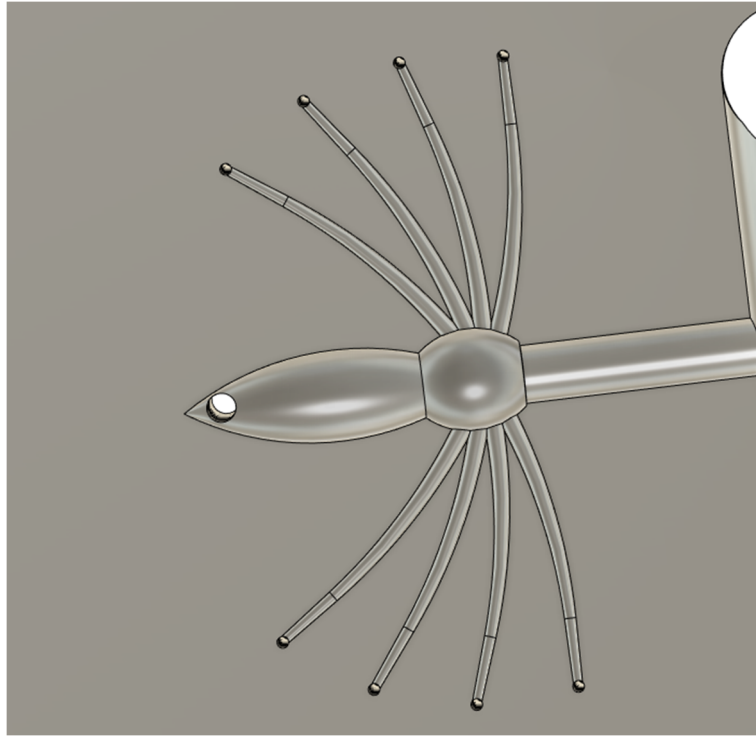
29. When preparing the molds for printing, select the Body in the Browser, right-click, and select "Save as mesh". Choose the "stl" filetype. Ensure the units are set to the appropriate units for the printer (typically millimeters). The default settings are generally sufficient.
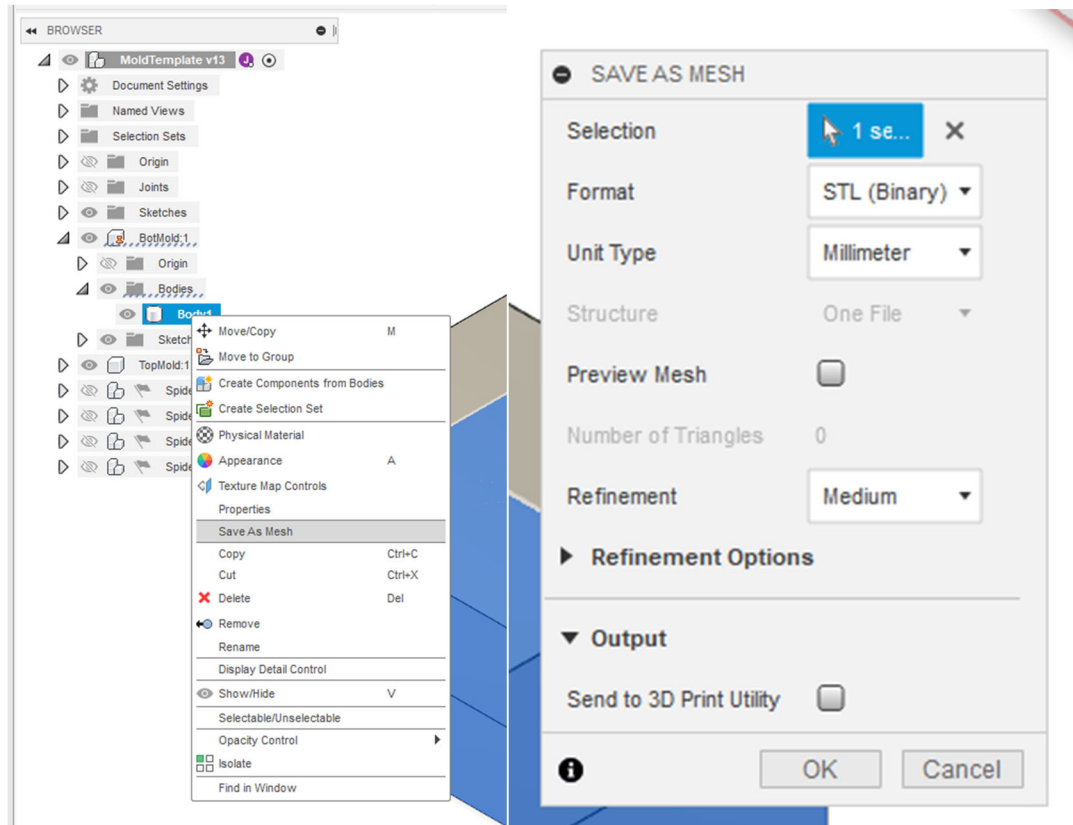
30. Repeat for the other half of the mold.