CP502618

Connected Approach to Designing Plastic with Autodesk Production, Design & Manufacturing Collection

Obiora Odugu
Engineering for change / Autodesk Foundation

Felix Amaefule
Generative CAD Services

Learning Objectives

Discover rules in design for injection molding

Discover and demonstrate knowledge of Autodesk Fusion 360 injection molding simulation

Learn ways to reduce lead time when designing and manufacturing with injection molding

Discover and demonstrate knowledge in Inventor mold design

Learn how to connect the plastic design workflow with Autodesk Fusion 360 desktop connector

Description

Design is rarely a perfectly linear and straightforward process. Often a single tool cannot accommodate all the workflows—and even when it does, it's rarely the right fit. By maximizing the different tools within the Autodesk Product Design & Manufacturing Collection (Autodesk Inventor, Autodesk Fusion 360, Autodesk Desktop Connector), designers and engineers can now design, make, and collaborate on products to reduce the lead time faced in today's manufacturing. This course will cover the complete workflow involved in designing parts for formative manufacturing—injection molding. From making a mold in Inventor to validating your part for injection molding in Autodesk Fusion 360. The Autodesk Product Design & Manufacturing Collection helps you connect these workflows right on your desktop using Autodesk Fusion 360 Desktop Connector

Speaker(s)

Obiora Odugu is an E4C fellow working with the Autodesk Foundation from Feb 10 – Sept 30. He provides technical support and assessment to Autodesk Foundation non-profits and start-ups. He is excited about this role as he gets to experience and work with multiple organizations developing products that are transforming human life and advancing the SDGs

He is passionate about designing robots and great products. He believes a great product should have qualities of a genuinely democratic government –Made by and for the people using it. He has been engaged in projects that demonstrate this belief. In 2021, he worked with ASME Engineering for Change (E4C) as an Autodesk design fellow helping Kheyti redesign their greenhouses for farmers at the bottom of the pyramid. He has also worked on multiple products within the furniture, electronic and energy sector.

Chukwubuikem Felix Amaefule is a leader of excellence and an inspiring being. He is very passionate about doing more with less which is deeply rooted in effectiveness. A fast-rising industry leader in the design and digital manufacturing space with the vision to empower the ecosystem.

He has over 5 years of experience in project and program management; over 8 years of experience in product development and over 3 years' experience in product management. He has over 8 years of experience in instructional training and management. He is also an intentional digital storyteller and a poet.

He has worked both in the non-profit and for-profit sectors and is passionate about making things and inspiring others to make even greater things. Learning through design and design for value creation. He believes when creative thinking meets innovative thinking, disruptive value is bound to be created. He has a strong passion for inspiring young minds into creative thinking through innovation and he lives out this dream daily.

He is currently leading the transformational team who are building an industry outcome-based Product Development school & On-demand Digital Manufacturing platform that promises to democratize making in Africa. He has a keen interest in People, Technology, Process and Product Relationships for Value Creation.

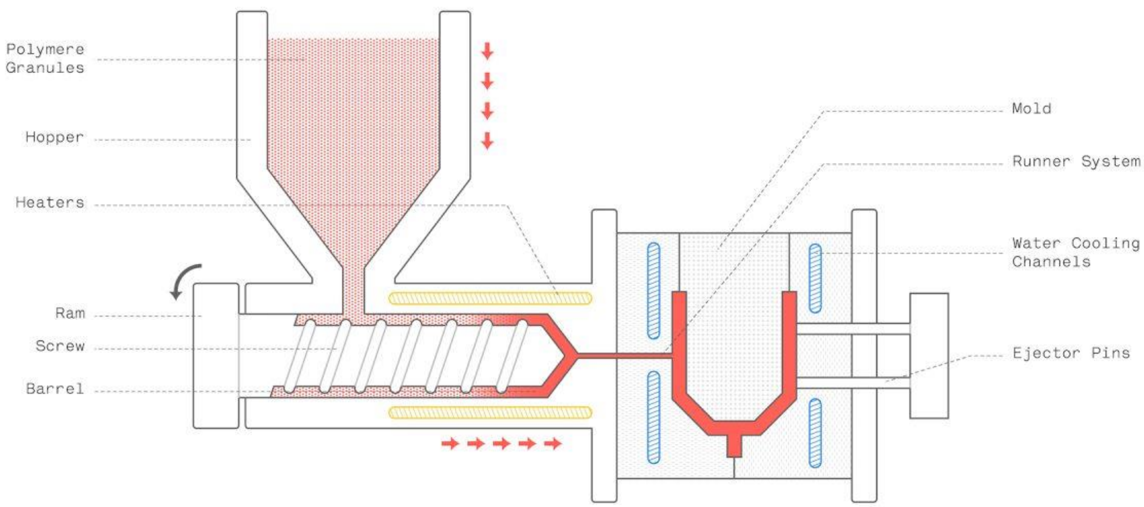
DESIGN FOR INJECTION MOLDING

Introduction

Injection molding is a formative technology for producing large volumes of parts. Compared to other manufacturing technologies—like CNC machining and 3D printing— it produces identical plastic parts with good tolerances. Injection molding is widely used for plastics, because of the dramatically low cost per unit when compared to other means of manufacturing plastic parts. The downside of this method is it requires an upfront, capital investment into tooling.

How Does Injection Molding Work?

Once an injection molding design is finalized, the next step in the manufacturing process is to machine the tooling, which is typically fabricated from steel or aluminum. Then CNC machine is used to carve out the negative part of the part.



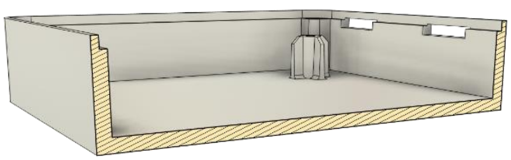
Injection molding process by Hub360

After making the mold, the actual production of plastic parts begins by pouring the plastic pallet into the hopper. The barrel temperature is raised to a melting temperature. This turns the pallet inside the barrel into a molten state. As the screw rotates it fees the molten plastic in the barrel to the mold via the runner system and gates. Finally, the part cools, solidifies and is automatically ejected from the mold the process starts over. It's an ideal manufacturing process for industries like medical devices, consumer products, and automotive to name a few.

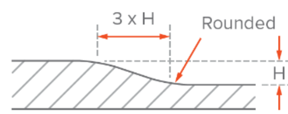
Design Rules for Injection Molding

RULE 1: Wall Thickness

- Favour a uniform wall thickness through the whole part.
- Apply smooth transitions in form of fillets between walls of different thicknesses if required. 3 times the difference in thickness is a good starting point.

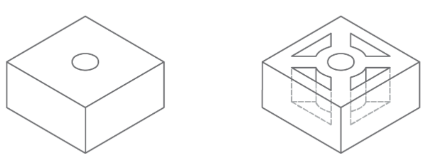


Incorrect

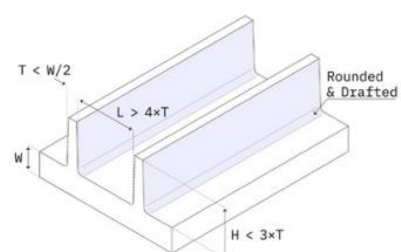


Correct

- Avoid thick sections by hollowing them out using ribs for adding strength.
  - Use a thickness equal to 0.5 × main wall thickness
  - Define a height smaller than 3 × rib thickness
  - Use a base fillet with radius greater than ¼ × rib thickness
  - Add a draft angle of at least 0.25° - 0.5°
  - Add a min. distance between ribs & walls of 4 × rib thickness

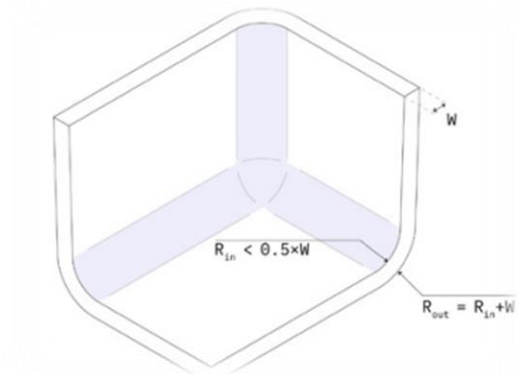


Incorrect Correct



RULE 2: Rounded edges

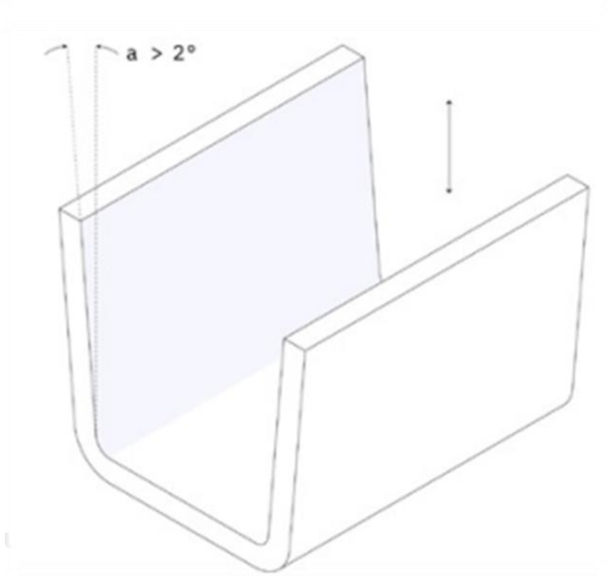
- Internal edges of part should be larger than 0.5 × wall thickness
- External edges: internal fillet + wall thickness



RULE 3: Add Draft angles

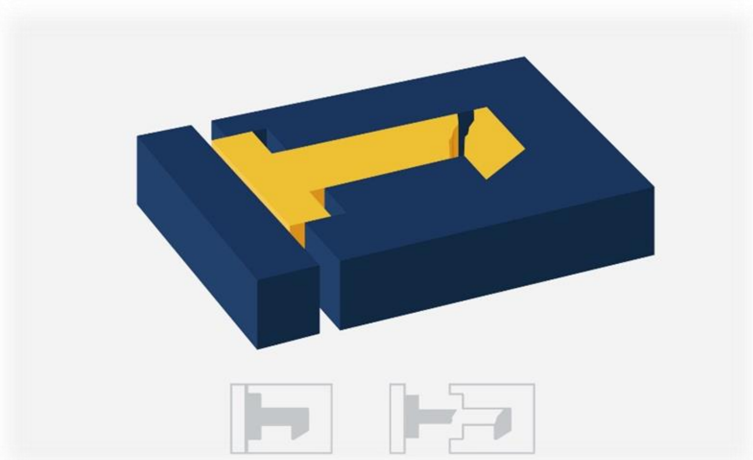
Recommended minimum: > 2°

- For parts taller than 50 mm, increase the draft
by 1° for every 25 mm
- For parts with a textured finish, increase the
draft by an extra 1°- 2°



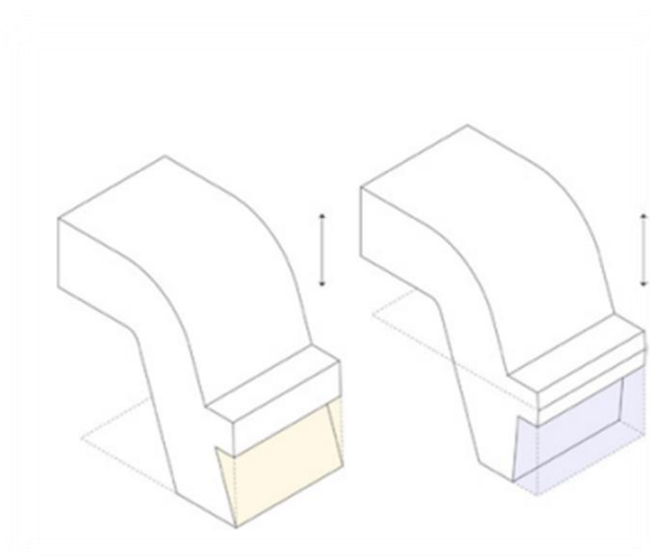
RULE 4: Watch out for

Undercuts in Injection molding are part features that cannot be manufactured with a simple two-part mold, because material is in the way while the mold opens or during ejection.



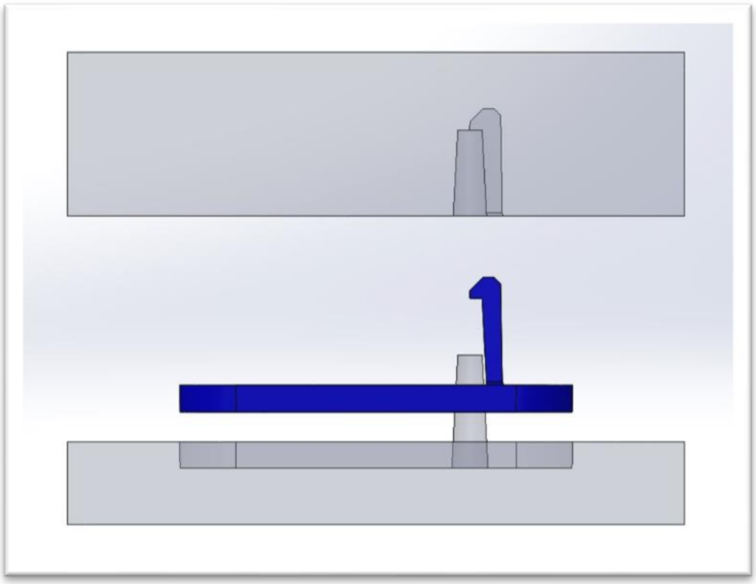
- **Moving the parting line**

The simplest way to deal with an undercut is to move the parting line of the mold to intersect with it



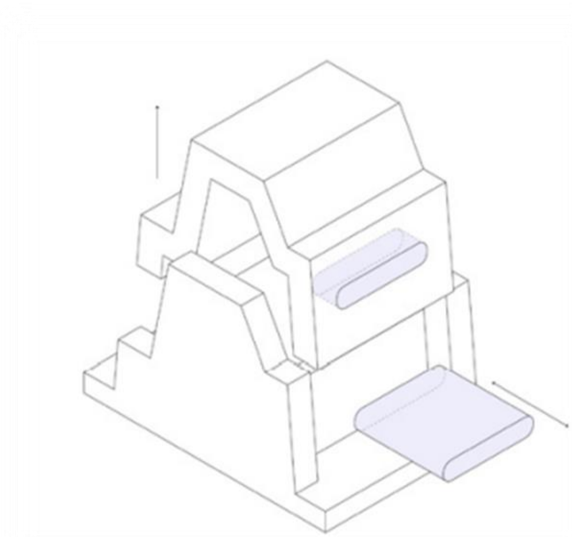
- **Using a shut-off**

Another way to deal with undercuts is to remove material from under or above the problematic area. This way the undercut is eliminated as the whole part can be directly supported by the mold.



- **Side-action cores**

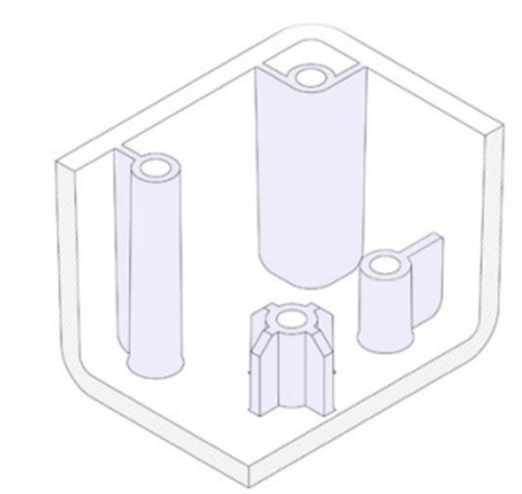
When none of the above solutions are viable, then cores can be used that slide out of the part from the side before it is ejected. Side-action cores should be used sparingly as they add complexity and increase the overall cost of a mold by 15% to 30%.



RULE 5: Bosses

Bosses are like circular ribs - the same general design guideline apply.

- Avoid designing bosses that merge into main walls.
- Support bosses with ribs or connect them to a main wall
- For bosses with inserts:
- Use an outer diameter equal to 2 × the insert's nominal size.



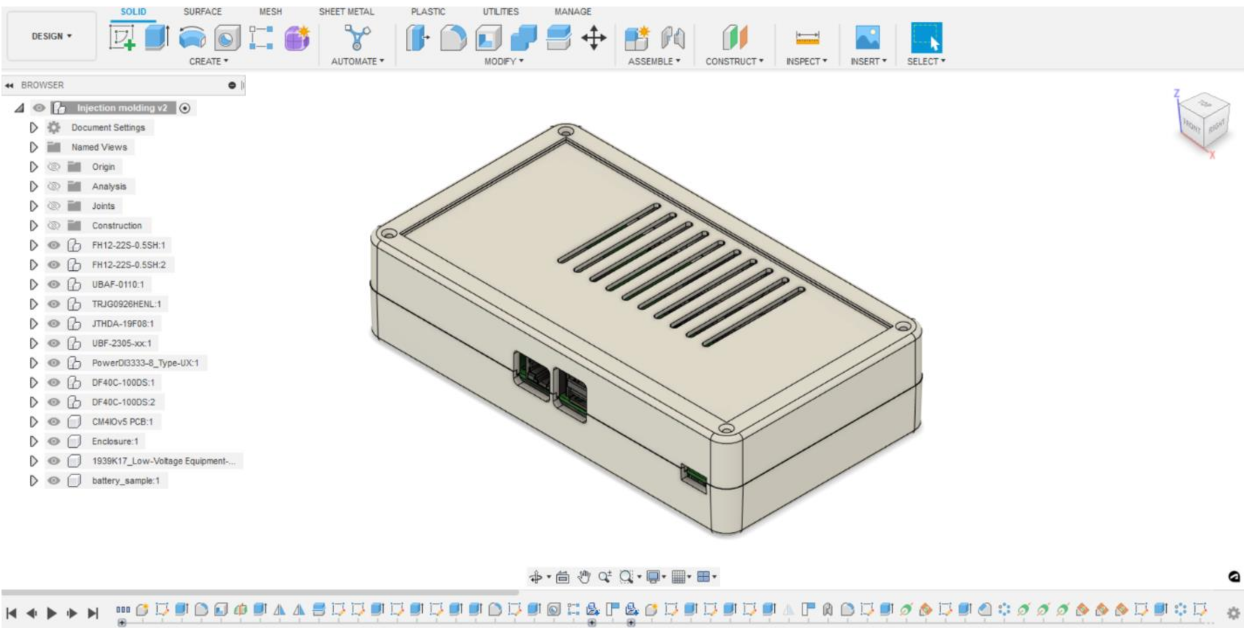
RULE 6: SNAP-FIT

use snap fit to join parts together without fasteners

- When designing snap-fits for Injection molding:
- Add a draft to the side-walls of the snap-fit.
- Use a thickness of 0.5 × main wall thickness.
- Adjust the width & length to control the deflection & force.
- Think how to deal with the created undercut.

Simulating a plastic part with Fusion 360 Injection molding simulation

Finding an Injection molding part can be pretty easy but getting the part you designed into a mold can be really hard. For example, designing a part and mold that is very easy to fill depends on many factors in the part design. With Fusion 360, all setup for an injection molding study is automatically done once you go into the simulation workspace. Fusion 360 also allows you customize and pick different values with over 11,000 materials to select from and counting.

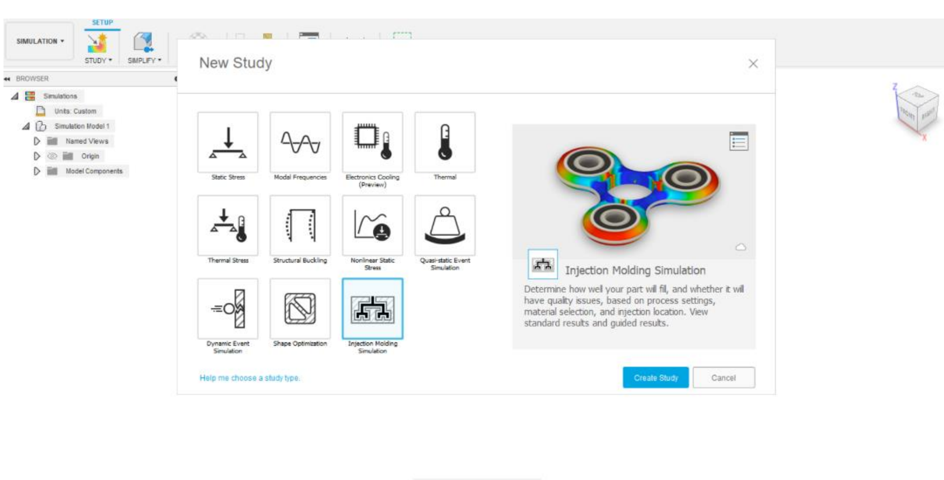


In the next paragraphs, we will show how one can achieve a simulation and interpret the results.

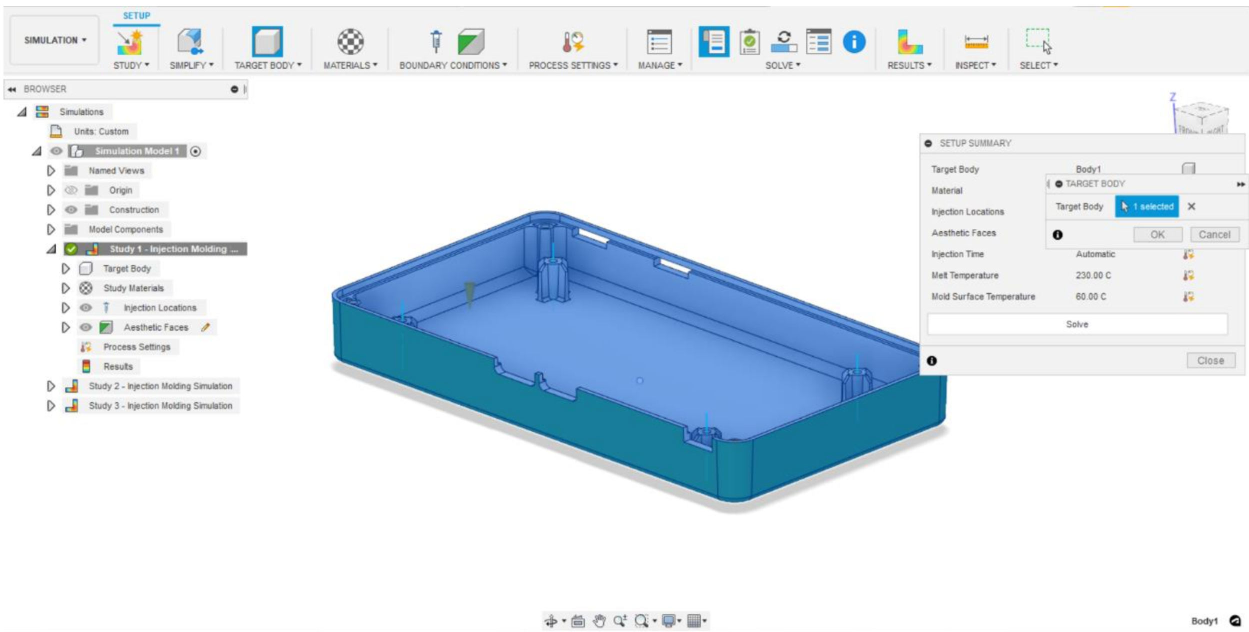
Setting up a simulation

In this step-by-step demonstration, we would be showing how to run a typical plastic simulation and gain insight from the results. The model being used here is a plastic enclosure of a PCB board used to control a camera system for remotely sensing the amount of content within stores

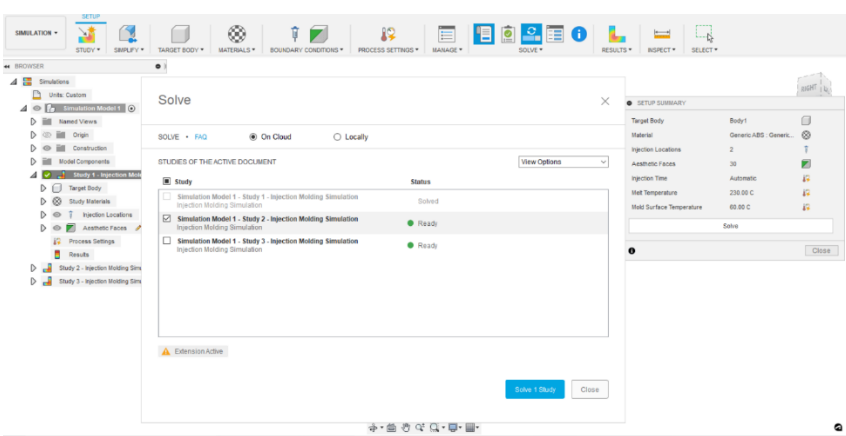
Step 1: Change your workspace to simulation and select Injection molding simulation



Step 2: Use the setup summary panel to pick the target body, material, desired injection location and faces designated as aesthetic faces. Aesthetic faces are faces that will be visible to the user. The remaining options like Injection time are subject to the manufacturer's specifications. Fusion 360 gives you default settings which are typical and good for a start.



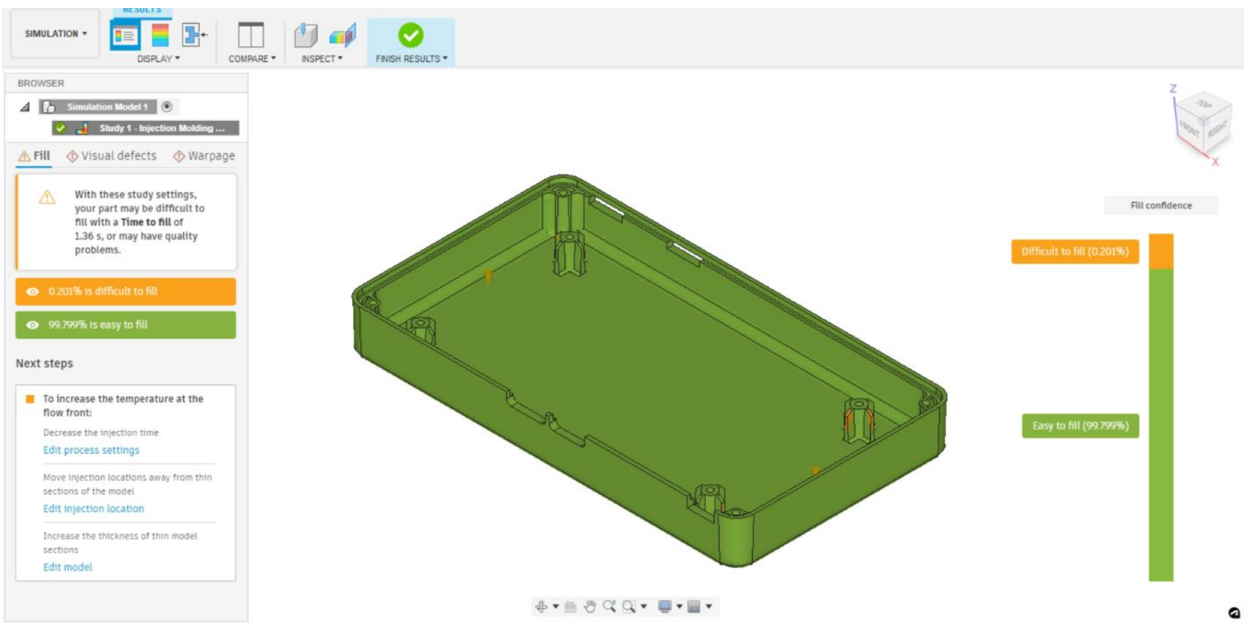
Step 3: Hit solve and select the study you would like to solve. To run a simulation in Fusion 360, you must have cloud credits. If you are running this for the first time, you can use the 7 days trial and proceed with your simulation. The option for 7 days trial is found under preferences in settings.



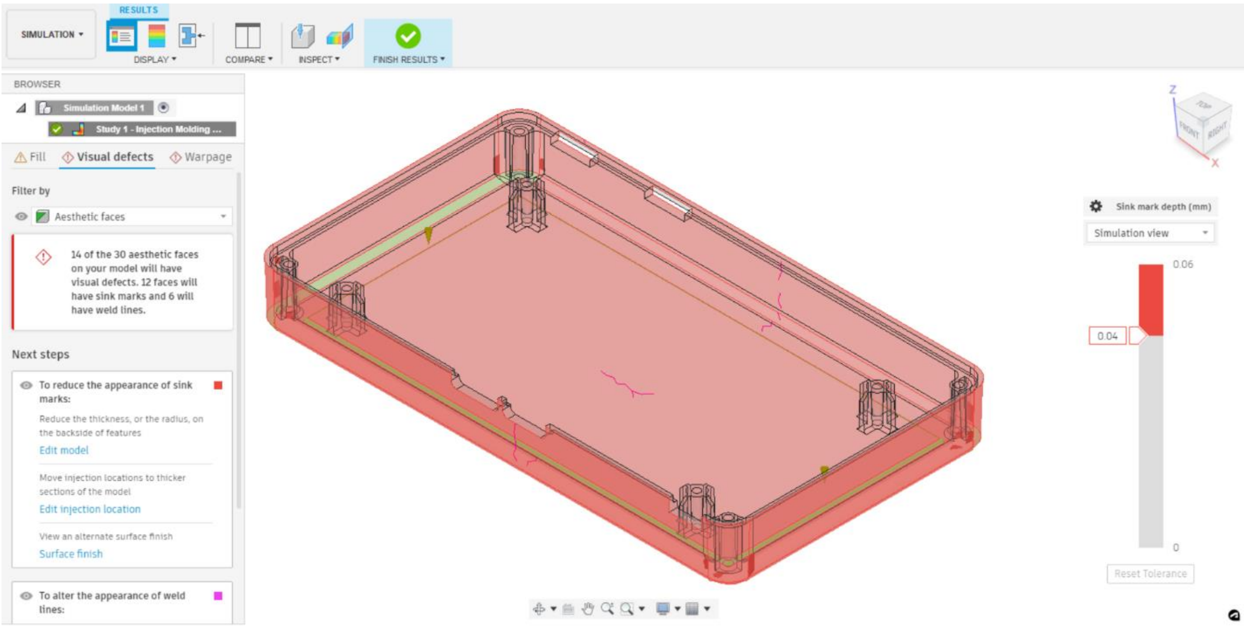
Simulation results and interpretation

After your solution has converged on the cloud, you can now view it on your PC. In the following paragraphs, we will show the different results.

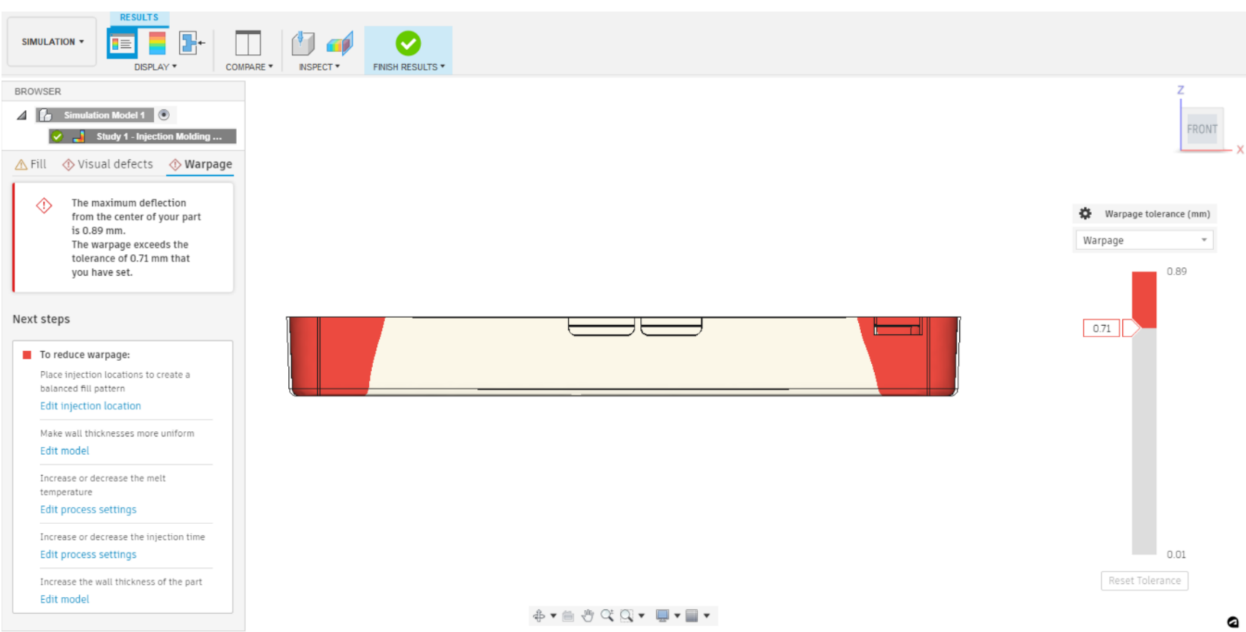
As seen on the left section of the figure below, the result page shows information such as fill confidence, visual defect and warpage. The enclosure design has a fill confidence of 99.799% which means that just 0.201% would experience difficulty filling. A closer look at the affected area shows that this are ribs used to support the bosses. Those ribs were drafted and has variable thickness which could point to the lower section getting solidified before the upper section eventually resulting in difficulty filling those section. Solution 3 as suggested by Fusion 360 is most likely the answer to the difficult sections



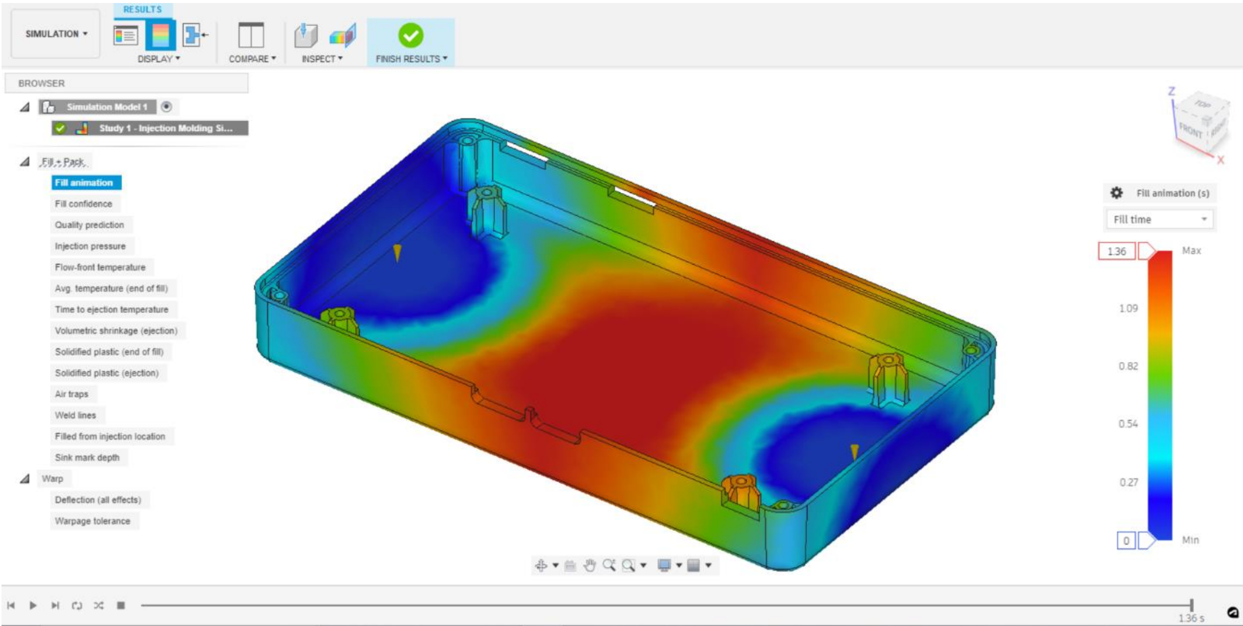
The next tab shows the visual defects that would be present on the part. 14 faces will have defects. A visual inspection of the part shows that these sink marks are location around the bosses at the edges of the part as indicated by the deep red marks. A potential reason for this is the large fillets that increased the thickness of the inner section and created uneven walls. These could have resulted to the inner section cooling faster than the outer. As suggested by fusion 360, reducing the thickness and radius on the backside of features would help resolve this.



Another useful insight in injection molding is warping. The third tap shows the amount of warpage the part will see. We have set a 0.71mm tolerance and any value below this is considered acceptable. Our part shows a warpage size of 0.89mm from the center. While this is a cause for concern, the deviation from the actual value can be attributed to



You can view this results in many ways within fusion 360, for example, on the Display tab in the top left, you can see three different displays. The second display helps you visualize how things work. The fill animation shows you the time it takes to fill the cavity of the mold and also has video demonstration of how the mold is filled. Other tabs also show useful information that provides into redesigning or optimizing the part.



The third tab under display shows all the details in on snapshot. It is also useful for visualizing how a typical injection molding process works. For this part the filling time is 1.35s with a weight of 0.07kg. It will take 23.49s to get a single part in your hands.

Connecting Autodesk Inventor to Fusion 360 & Fusion Team with Inventor using Desktop connector

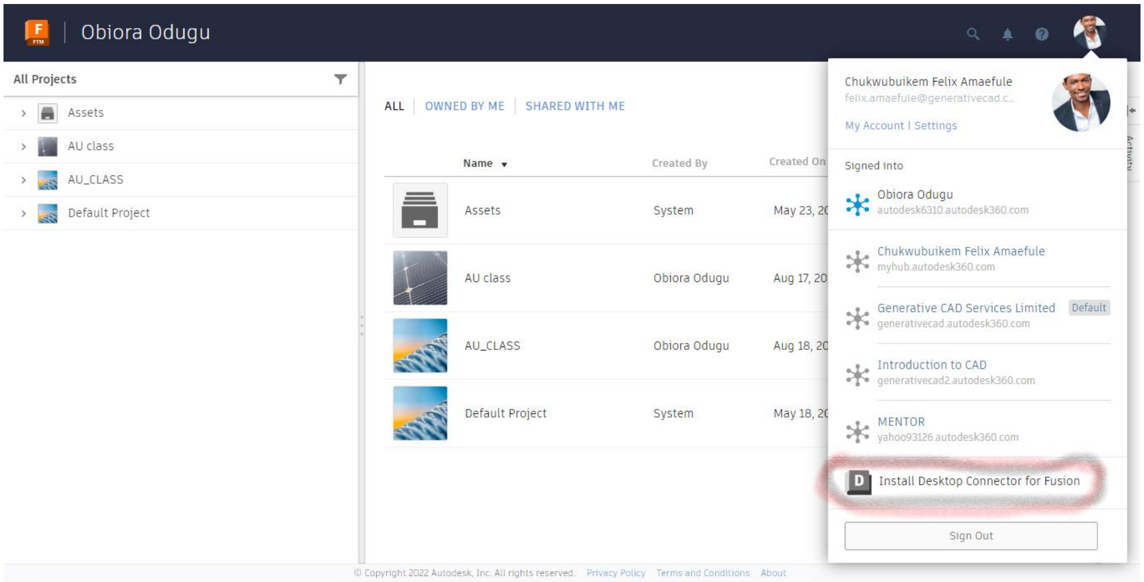
Designing the mold is done using Autodesk Inventor. To collaborate efficiently and ensure an integrated workflow between Fusion 360 and Autodesk Inventor, desktop connector is used.

What is Desktop Connector?

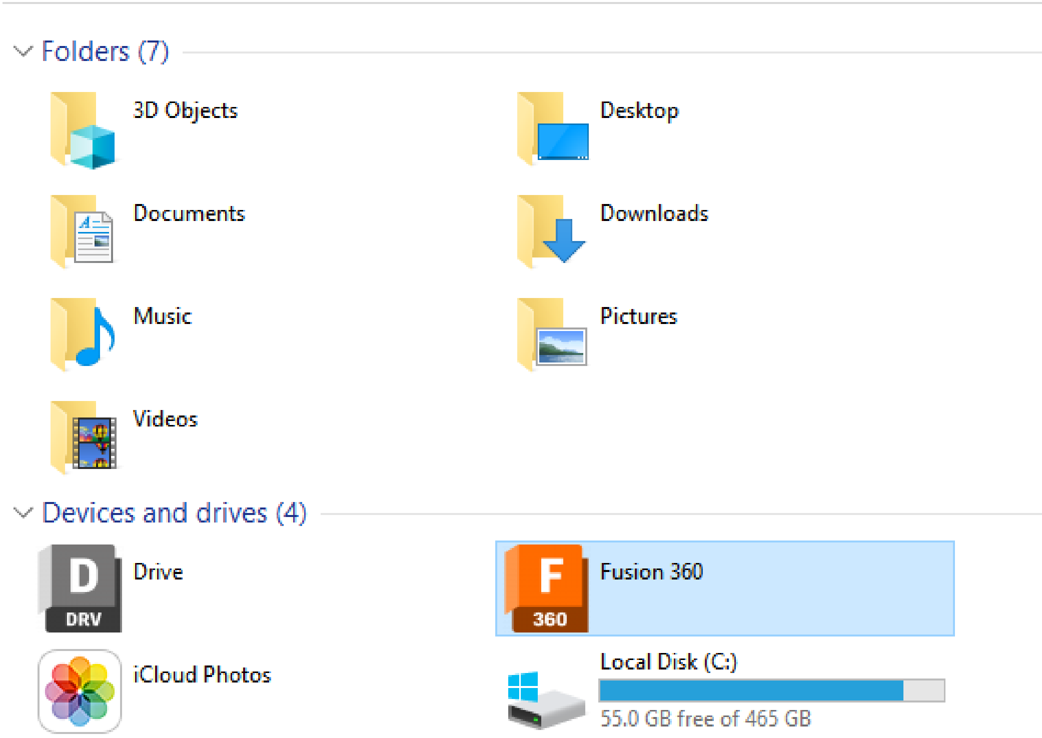
Desktop Connector is a desktop service that integrates an Autodesk data management source (or data source) with your desktop folder and file structure for easy file management.
The files in the data source are replicated in a connected drive. You can manage files in the data source through the connected drive, just as you would any other folder on your machine. Changes made in the connected drive are automatically uploaded to the data source.

- ✓ To get started download and install Desktop Connector for Fusion.

- ✓ To download the Desktop Connector for Fusion, use this [link]() and login to your Autodesk account. See the screenshot below.



- ✓ Sign in to your desktop connector with your Autodesk account.
- ✓ Make sure the signed in account has access to the Team and the Project in Fusion Team you are working on.
- ✓ Don't forget to add team mates on the Fusion team if you have any

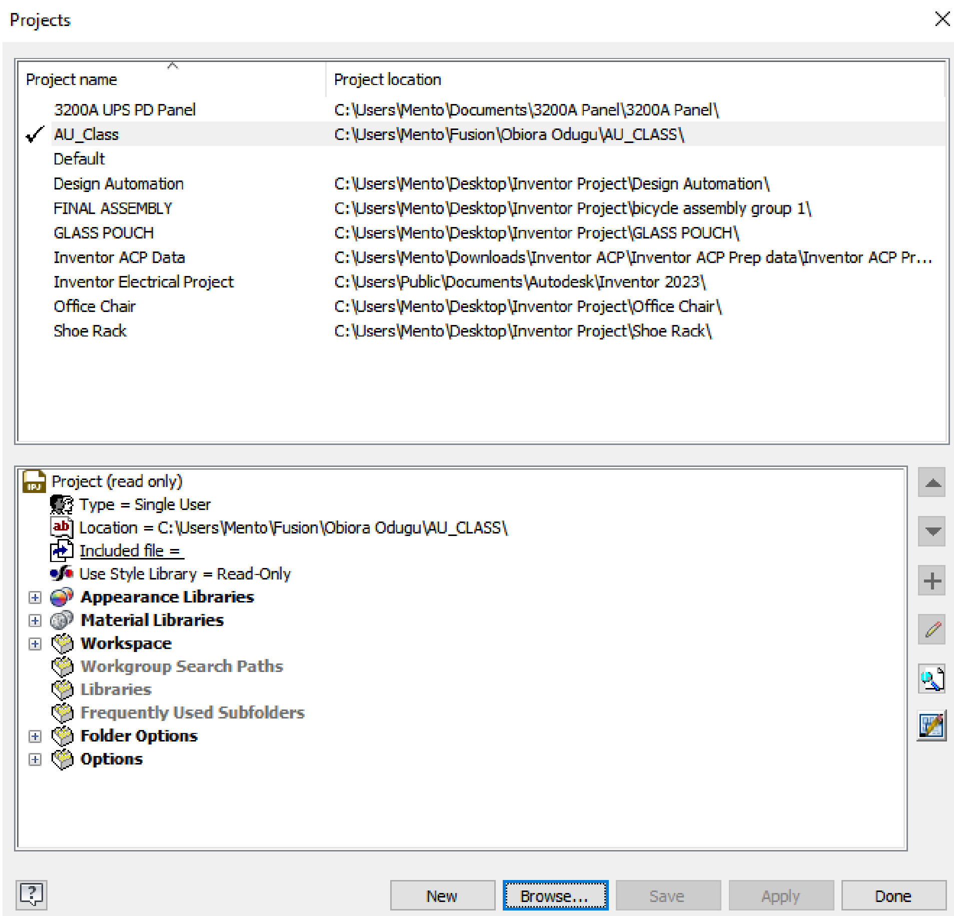


Designing the Mold with Autodesk Inventor

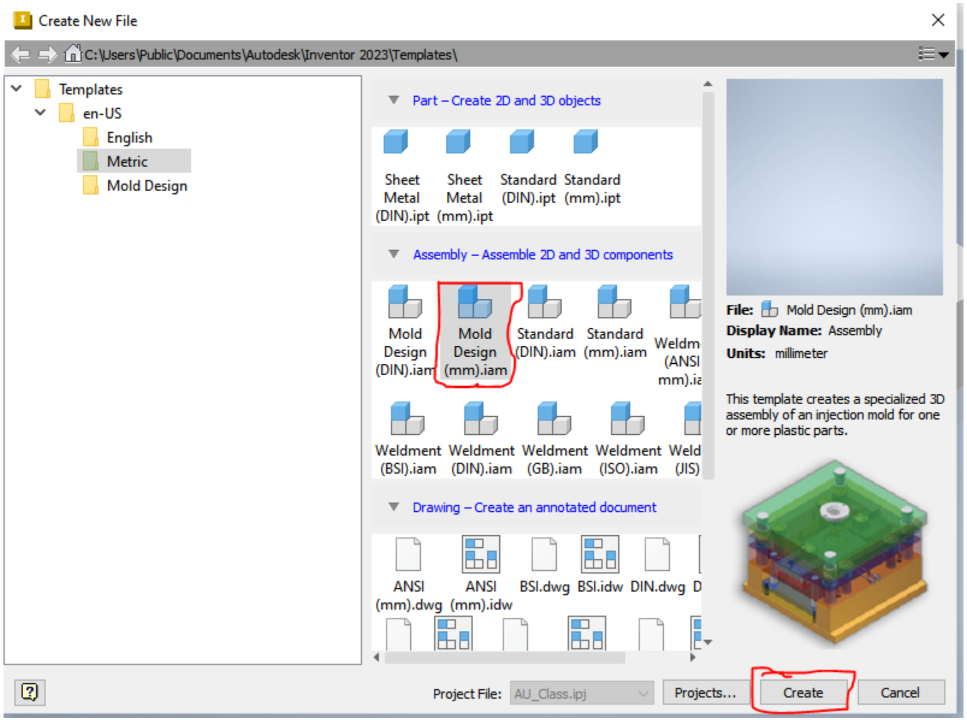
After setting up the desktop connector as described above, it is time to jump straight to Autodesk Inventor to get started with our mold design.

- ✓ Open Autodesk Inventor application
- ✓ Create an Inventor Project file and the file location should be the project in your Fusion Team (To do this you need to go through the Fusion desktop connector in your file explorer)

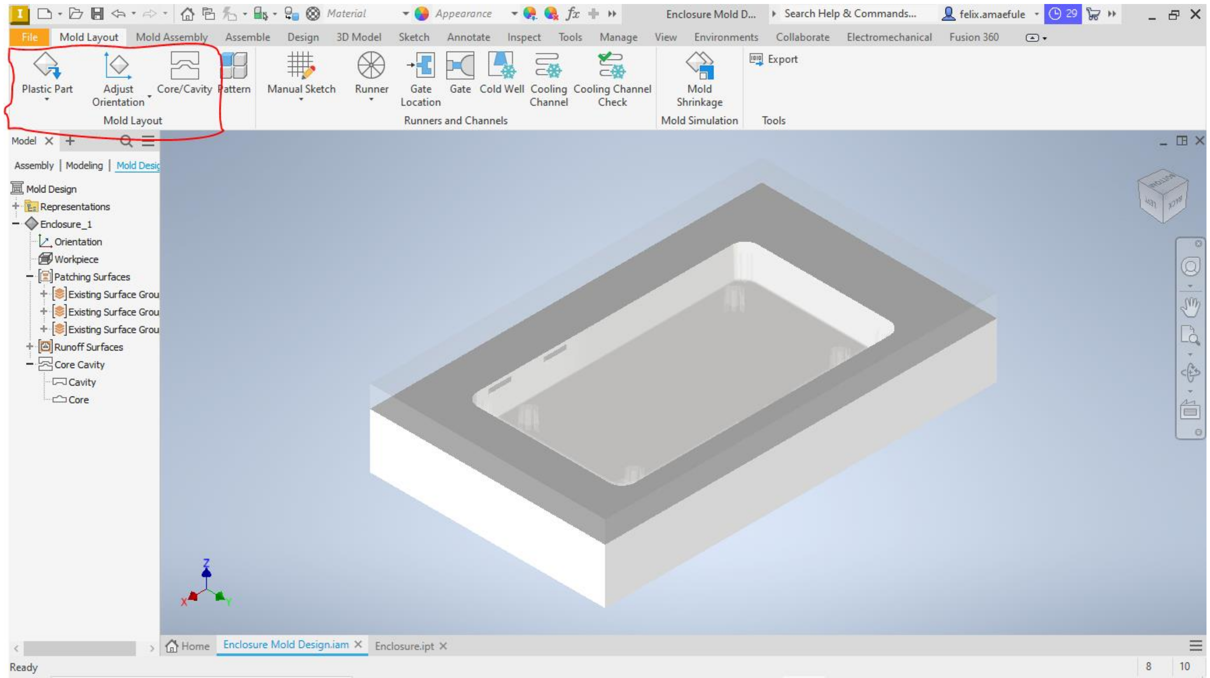
✓ Create a Mold file from the mold design (assembly) template.



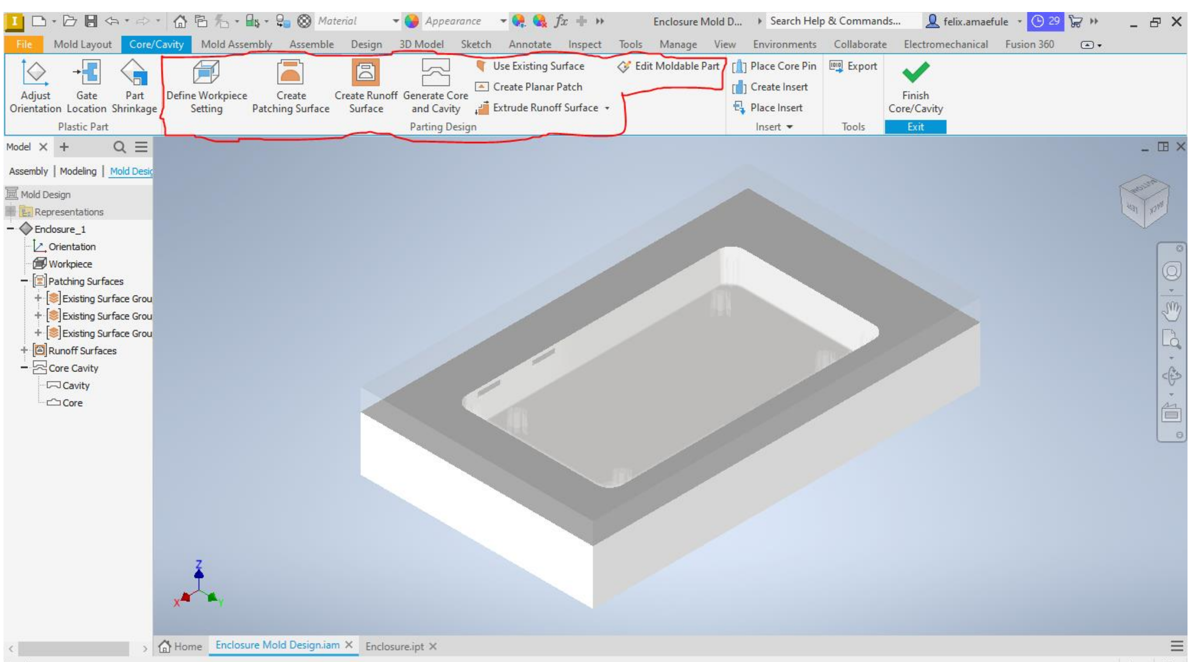
- ✓ As soon as you have the project created, create the Inventor mold design assembly file as seen in the screenshot below.

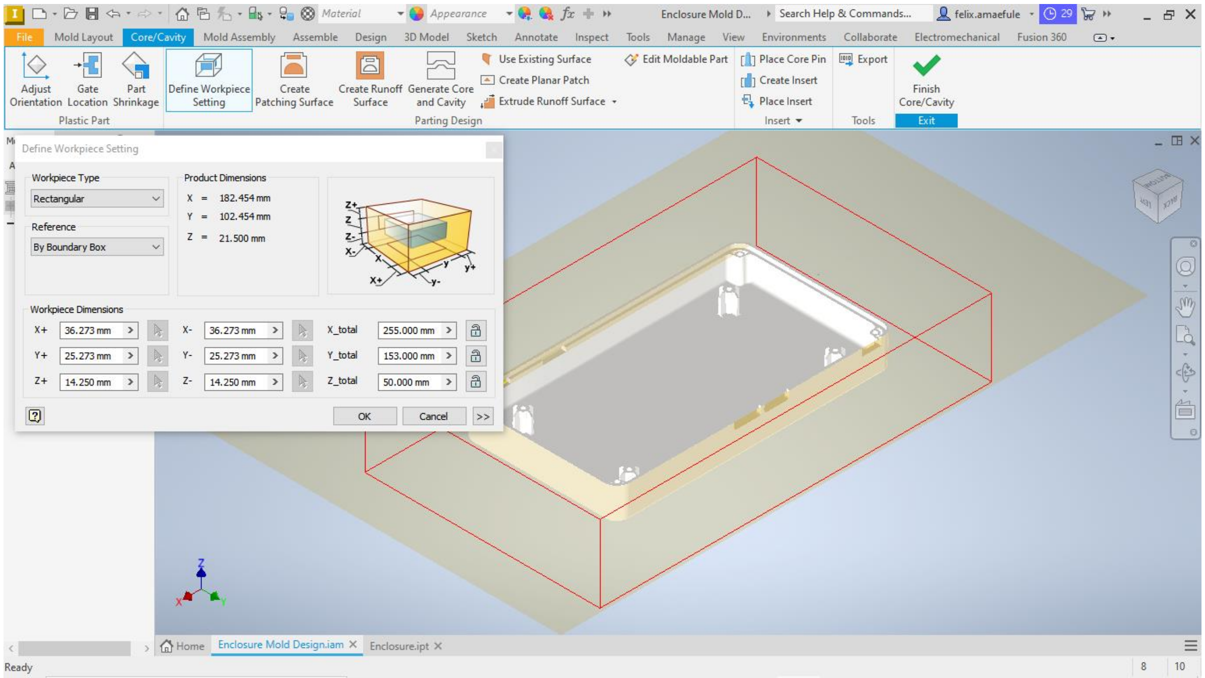


Next is to import the plastic part design from the fusion team (with the help of desktop connector) into the mold design assembly file using the plastic part tool. Now have the part imported, you will adjust the part orientation as desired, while considering how it will fit on the mold. Click on the Core/Cavity feature, it will open a dynamic tab called core/cavity.
define the mold (block) workspace using the "define workspace settings" feature.

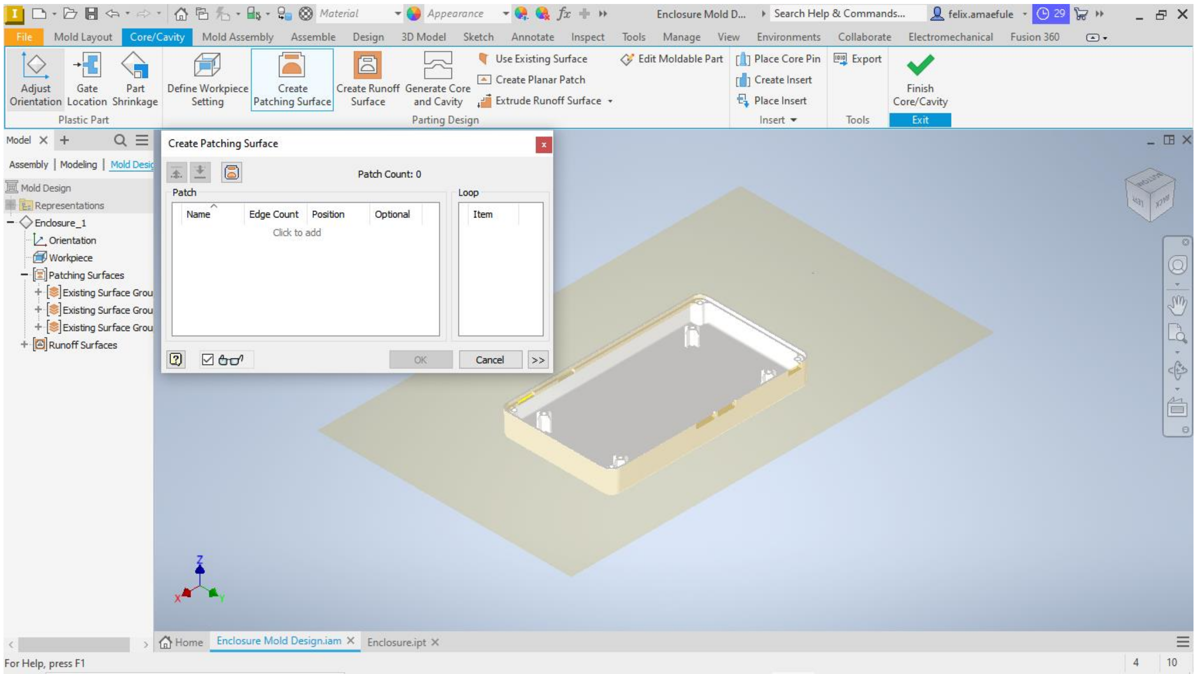


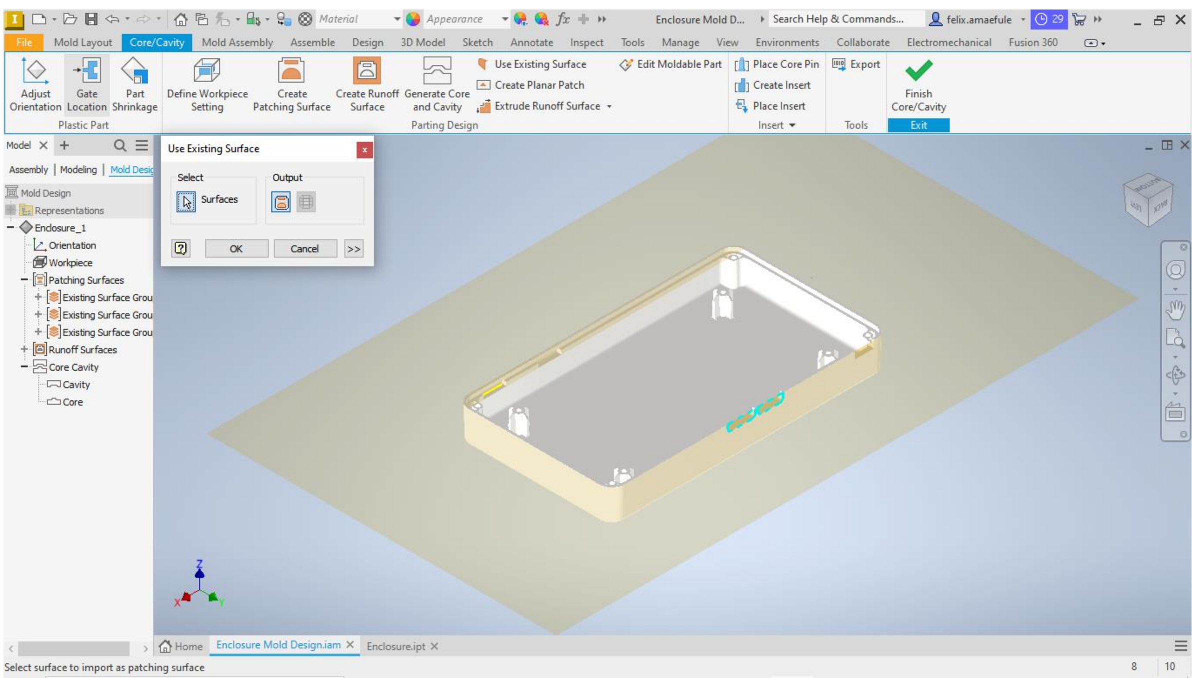
Then, define the mold (block) workspace using the "define workspace settings" feature. Make sure that you give appropriate allowance offset between the product size and the mold/metal block. While doing this think about the standard sizes of the metal block (aluminum or steel) in the market and the mold integrity while machining.



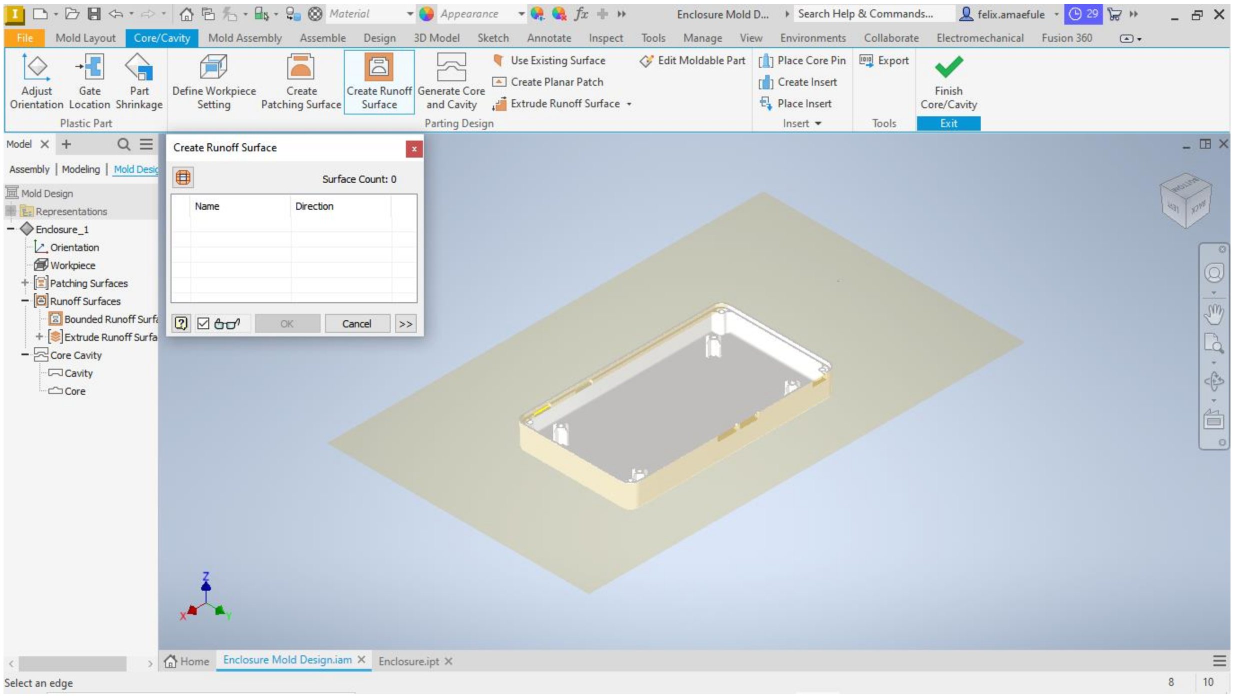


Patch the open surfaces using the "create patching surface" feature. You can either use the "create patching surface" feature to automatically generate the patch surface or the “Use Existing Surface” feature.to manually create the patch surfaces (with the help of “Edit Model Part” feature) or both when the “"create patching surface" feature did not generate all the needed patch surfaces.

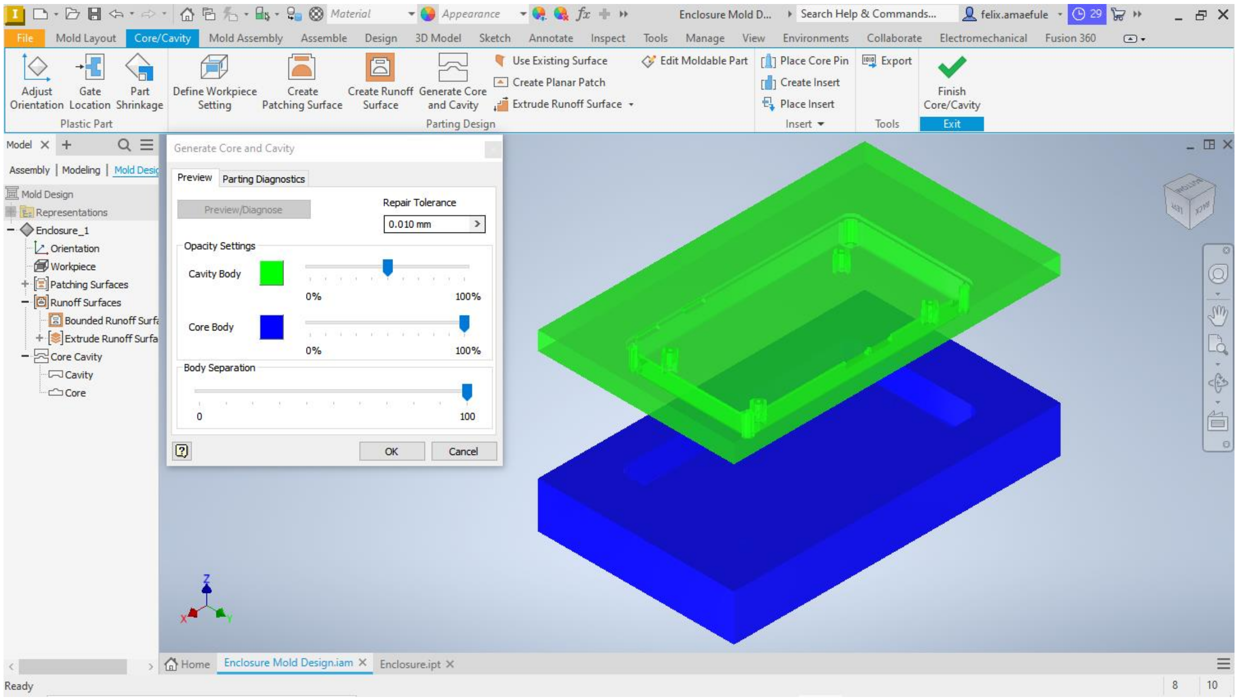




Use the “Create Runoff Surface” feature to create parting line for the mold. You can either use the "Create Runoff Surface" feature to automatically generate the parting line or the “Extrude Runoff Surface” feature.to manually create the parting line or both when the " Create Runoff Surface" feature did not generate all the needed surfaces for the parting line.



Lastly, use the “Generate Core and Cavity” feature to create the mold core and cavity. Use the “preview/diagnose” button to generate the preview of the mold core and cavity, while using the opacity and body separation settings to drive further insights.


Generate Core and Cavity
Preview
Parting Diagnostics
Preview/Diagnose
Repair Tolerance
0.010 mm
Opacity Settings
Cavity Body
Core Body
0%
100%
Body Separation
0
100
OK
Cancel
Mold Layout
Core/Cavity
Mold Assembly
Adjust Orientation
Gate Location
Part Shrinkage
Define Workpiece Setting
Create Patching Surface
Create Runoff Surface
Generate Core and Cavity
Use Existing Surface
Create Planar Patch
Extrude Runoff Surface
Edit Moldable Part
Place Core Pin
Create Insert
Place Insert
Export
Finish Core/Cavity
Plastic Part
Parting Design
Insert
Tools
Exit
Mold Design
Representations
Enclosure_1
Orientation
Workpiece
Patching Surfaces
Runoff Surfaces
Core Cavity
Cavity
Core
Enclosure Mold Design.iam
Enclosure.ipt
Ready