AUTODESK UNIVERSITY

Connected approach to designing plastic with Autodesk PDM collection

CP502618

Obiora Odugu
Autodesk foundation tech adviser | @obioraodugu

Felix Amaefule
CEO | @Felixamaefule

Safe Harbor Statement

The presentations during this event may contain forward-looking statements about our outlook, future results and related assumptions, total addressable markets, acquisitions, products and product capabilities, and strategies. These statements reflect our best judgment based on currently known factors. Actual events or results could differ materially. Please refer to our SEC filings, including our most recent Form 10-K and Form 10-Q filings available at www.sec.gov, for important risks and other factors that may cause our actual results to differ from those in our forward-looking statements.

The forward-looking statements made in these presentations are being made as of the time and date of their live presentation. If these presentations are reviewed after the time and date of their live presentation, even if subsequently made available by us, on our website or otherwise, these presentations may not contain current or accurate information. We disclaim any obligation to update or revise any forward-looking statements.

Statements regarding planned or future development efforts for our products and services are not intended to be a promise or guarantee of future availability of products, services, or features but merely reflect our current plans and based on factors currently known to us. Purchasing decisions should not be made based upon reliance on these statements.

PLEASE NOTE: All Autodesk content is proprietary. Please Do Not Copy, Post or Distribute without authorization.

Today’s Objectives

Reduce lead time with Injection molding rules

- We will review and discuss injection molding rules
- Learn ways to reduce lead time when designing and manufacturing with injection molding

Fusion 360 simulation & Inventor mold design

- Discover and demonstrate knowledge of Autodesk Fusion 360 injection molding simulation
- Discover and demonstrate knowledge in Inventor mold design

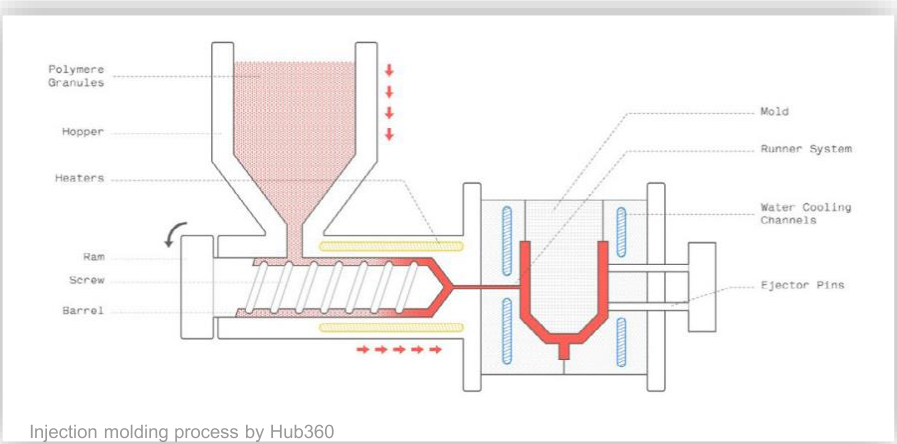
Seamless workflow with desktop connector

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

INJECTION MOLDING

What is Injection molding?

- Injection molding is a formative manufacturing technology for producing plastics parts
- Typically for producing large volumes of plastic parts
- Parts are always identical with good tolerances
- The process is shown on the left of the slide



Injection molding process by Hub360

INJECTION MOLDING RULES

RULE 1: Use a constant wall thickness

Recommended thickness: 1 mm and 3 mm

- Always design parts with the smallest possible (and constant) wall thickness
- 10% increase in wall thickness,30% increase in stiffness.
- Use ribs to hollow out parts

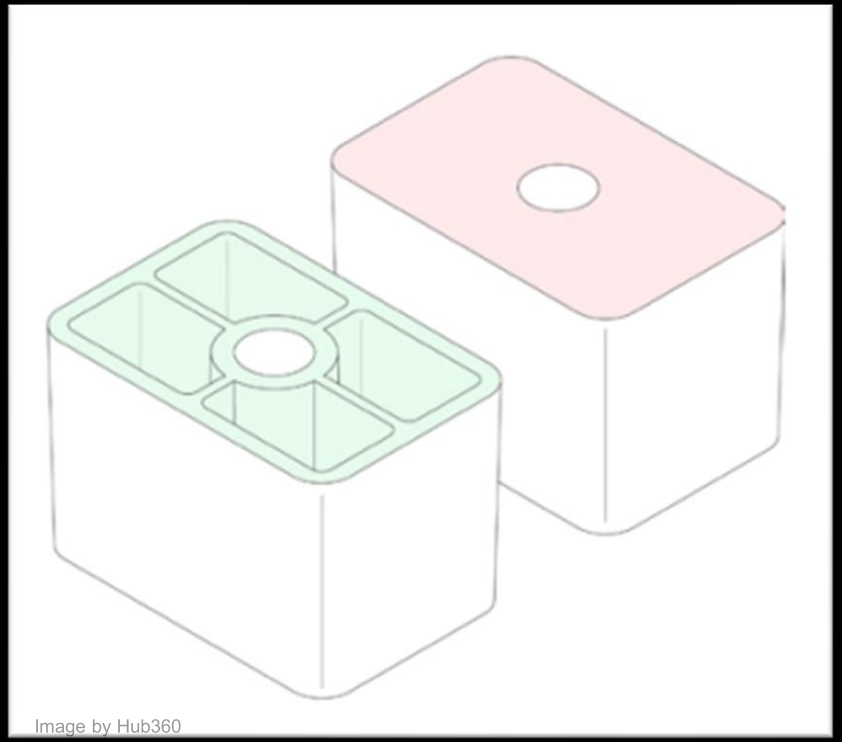
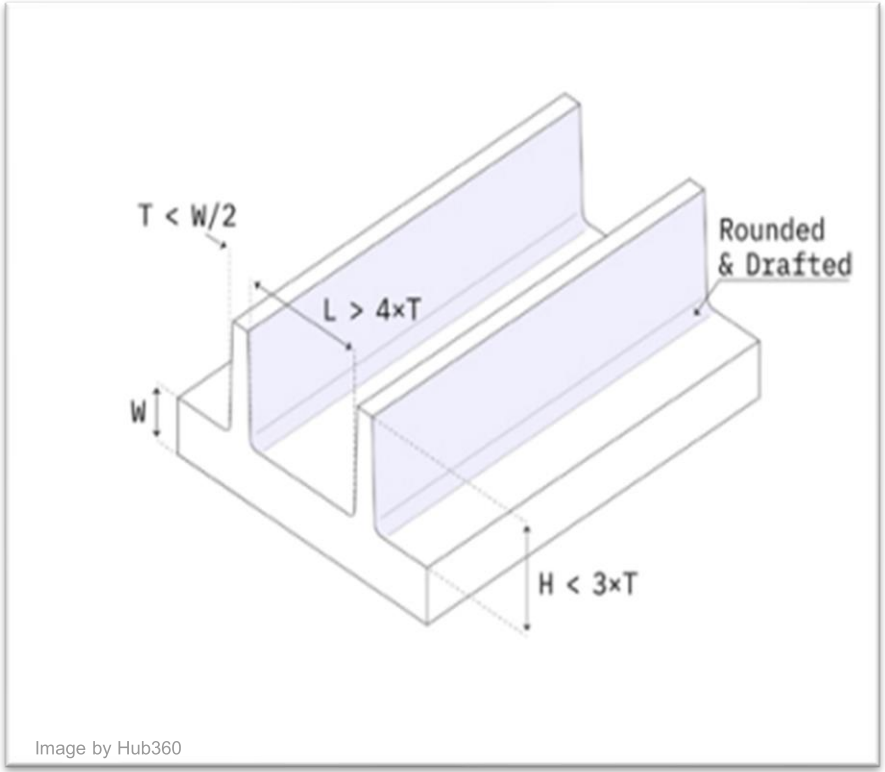


Image by Hub360

INJECTION MOLDING

RULE 2: Use Ribs to hollow out part

- Rib thickness = 0.5 × main wall thickness
- Height should be lesser than 3 × rib thickness
- Fillets greater then ¼ × rib thickness should be applied to the edges
- Add a draft angle of at least 0.25° - 0.5°
- When possible, use a minimum of 4 × rib thickness between ribs & walls



INJECTION MOLDING RULES

RULE 3: Round all edges

- Add a fillet with a radius that is as large as possible to all internal and external edges.
- The constant wall thickness rule must also be applied to the corners of the part.

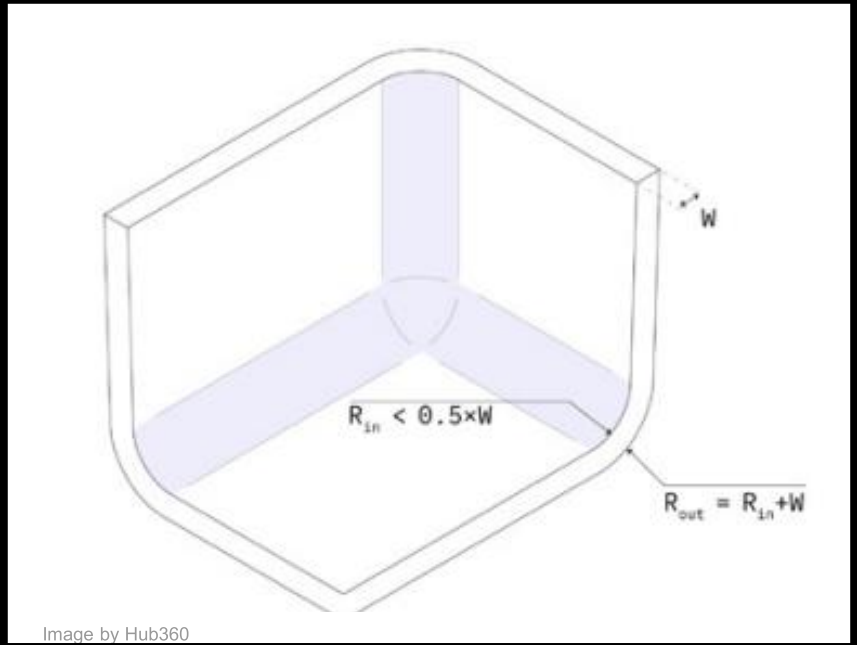


Image by Hub360

INJECTION MOLDING RULES

RULE 4: Add draft angles

- Add a draft to all vertical walls to make the ejection of the part easier and avoid drag marks.
- If they serve a functional purpose, external walls may be left undrafted

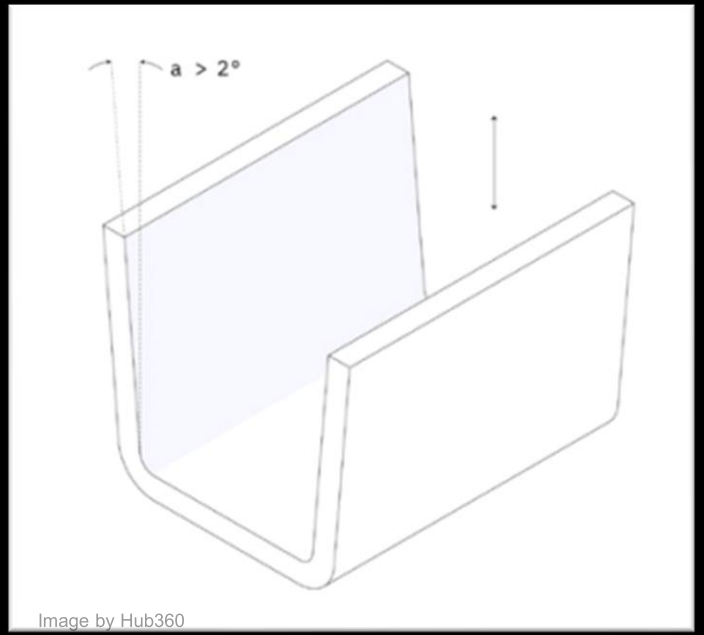
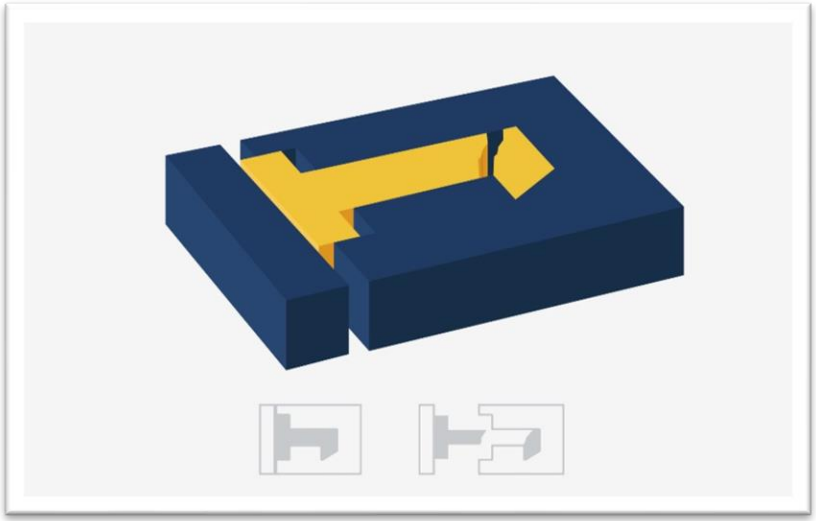
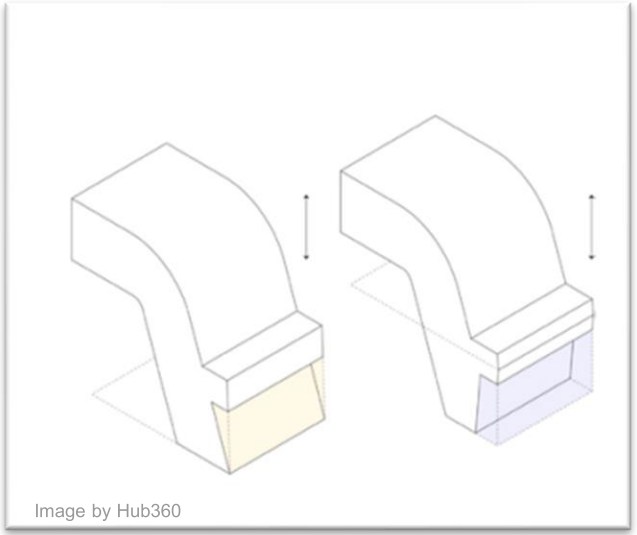


Image by Hub360

UNDERCUTS

RULE 5.0: Moving the parting line

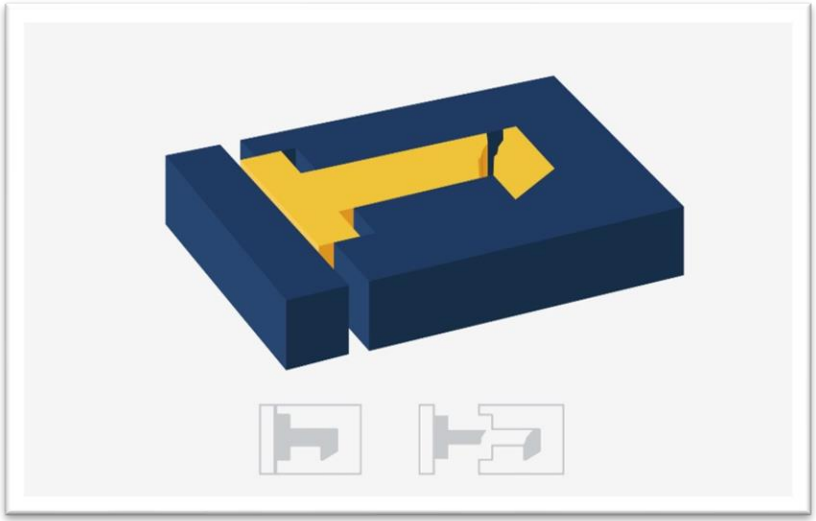


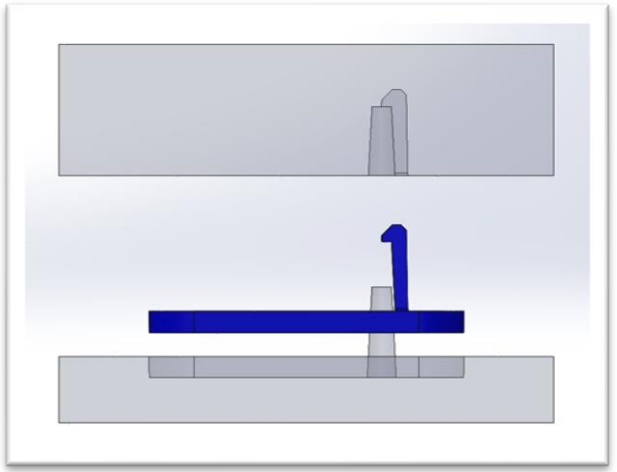


The simplest way to deal with an undercut is to move the parting line of the mold to intersect with it.

UNDERCUTS

RULE 5.1: Moving the parting line

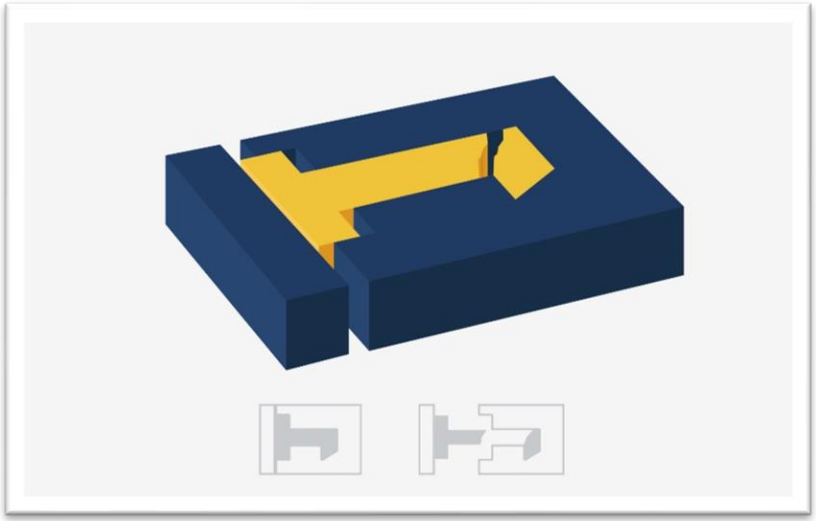


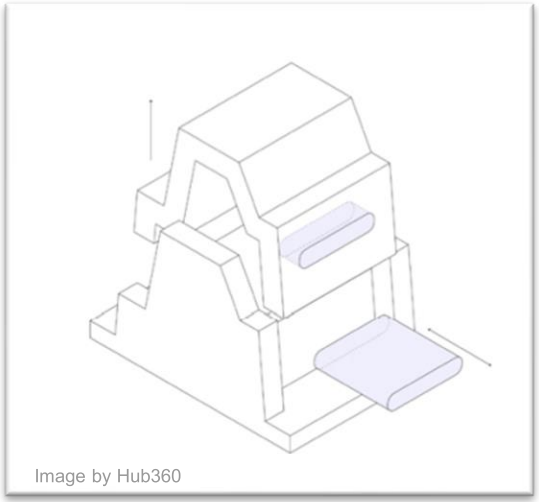


The next best way to deal with an undercut is to use shut off to remove material from under the area of interest

UNDERCUTS

RULE 5.2: Side-action cores



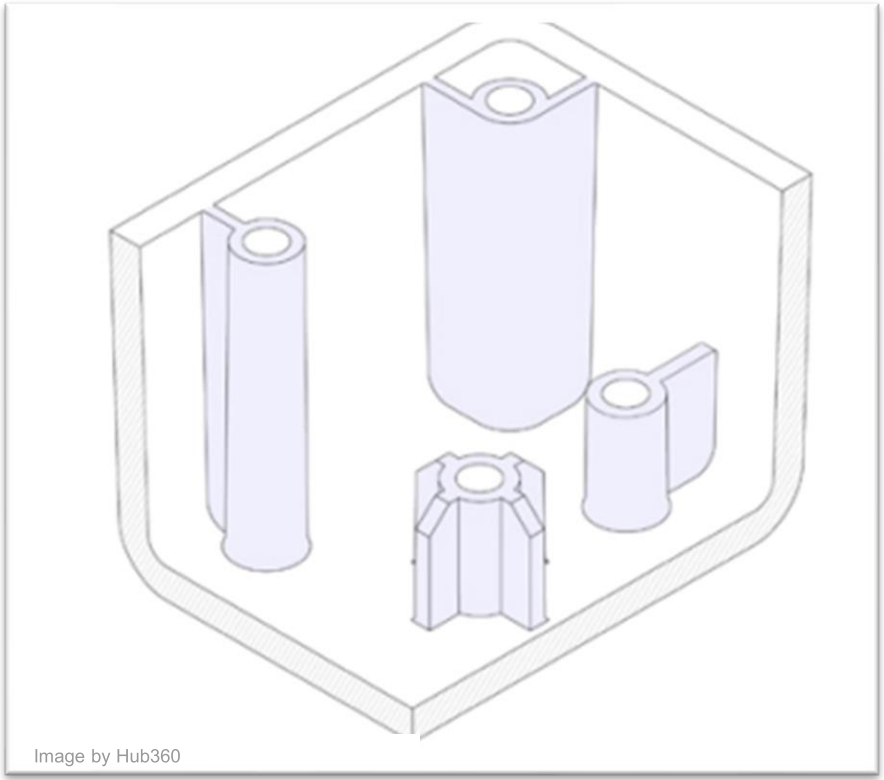


Side action is used in situation where undercut and shutoff are not possible

INJECTION MOLDING

RULE 6: BOSSES

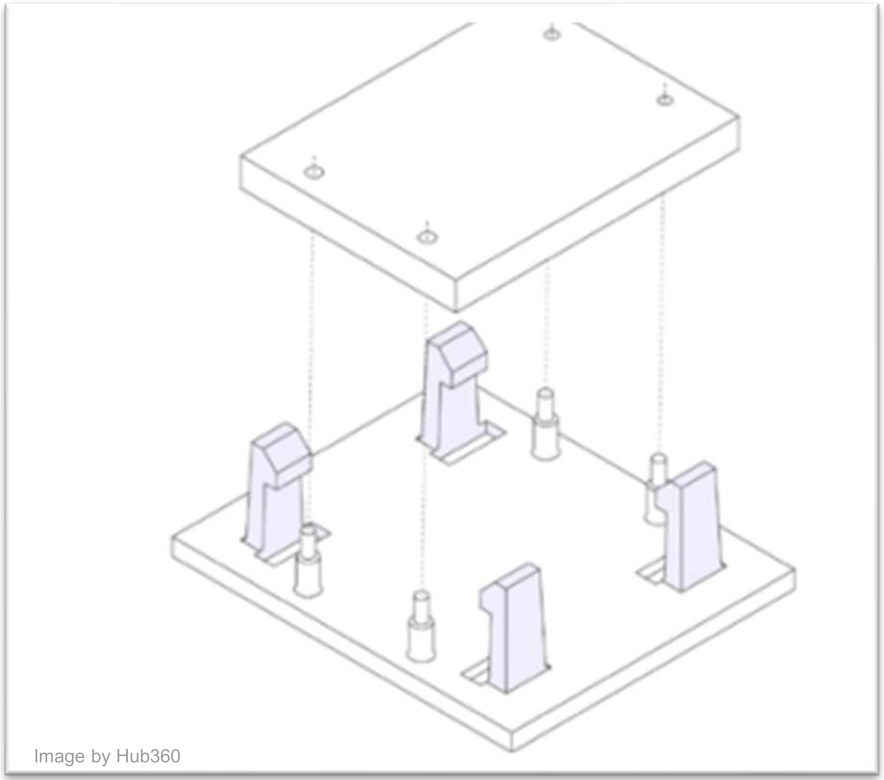
- Avoid designing bosses that merge into main walls.
- Support bosses with ribs or connect with main wall as shown
- For bosses with inserts: Use an outer diameter = 2 × the insert’s nominal size.



INJECTION MOLDING

RULE 7: SNAPFITS

- 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.



INJECTION MOLDING SIMULATION AND SETUP

A CASE STUDY

Autodesk and the Autodesk logo are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.

© 2022 Autodesk. All rights reserved.

Autodesk Confidential & Proprietary Information - Please do not post, copy or distribute without authorization.