Design for Manufacturing (DFM) Report-1-1P18022026-Upwork

Project: 1P18022026-Upwork

Designed For: Injection Molding

Unit System: Millimeters (mm)

CAD File Name: Plastic Enclosure

CAD File Version: A1

Prepared For: Spencer Yandrofski | Mecado

Prepared By: Bertrand Mezatio / Certified Mechanical Engineer (VDI - Association of German Engineers e.V.)

Date: 18 February 2026

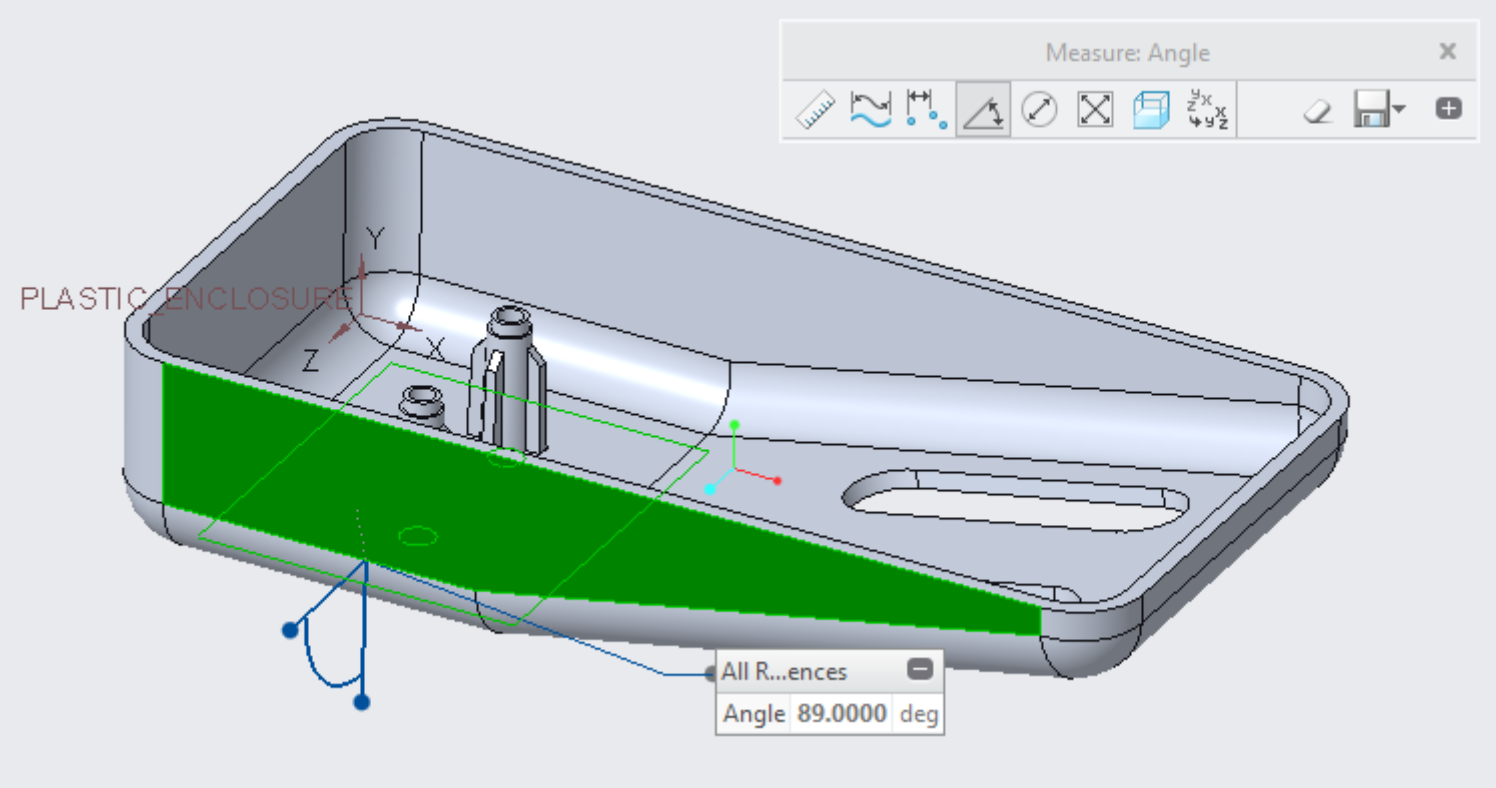
Design for Manufacturing (DFM) Report-1-1P18022026-Upwork

A Design for Manufacturing (DFM) analysis was conducted on the plastic enclosure to evaluate its suitability for injection molding. The analysis identified 75 specific areas requiring design adjustments to optimize moldability, reduce tooling costs, and prevent structural or cosmetic defects.

Label Class	Description (Indication)
draft_angle	Associated with adjacent faces and a draft angle
undercut	Associated with faces that constitute an undercut
lifter	Associated with faces that require a lifter
pull	Associated with faces that require a side pull
thin_steel_condition	Too thin of steel pins/features that would destroy the mold
blend	Either faces need blend between them or the blend needs to be changed
ejector_pin_loc	Associated with a point on a face (ejector pin location)
parting_line	Associated with a plane, offset from a face or cartesian plane
wall-thickness	Associated with faces that have a wall-thickness issue (geometric)
design_clean_up	Issues with the 3D model geometry

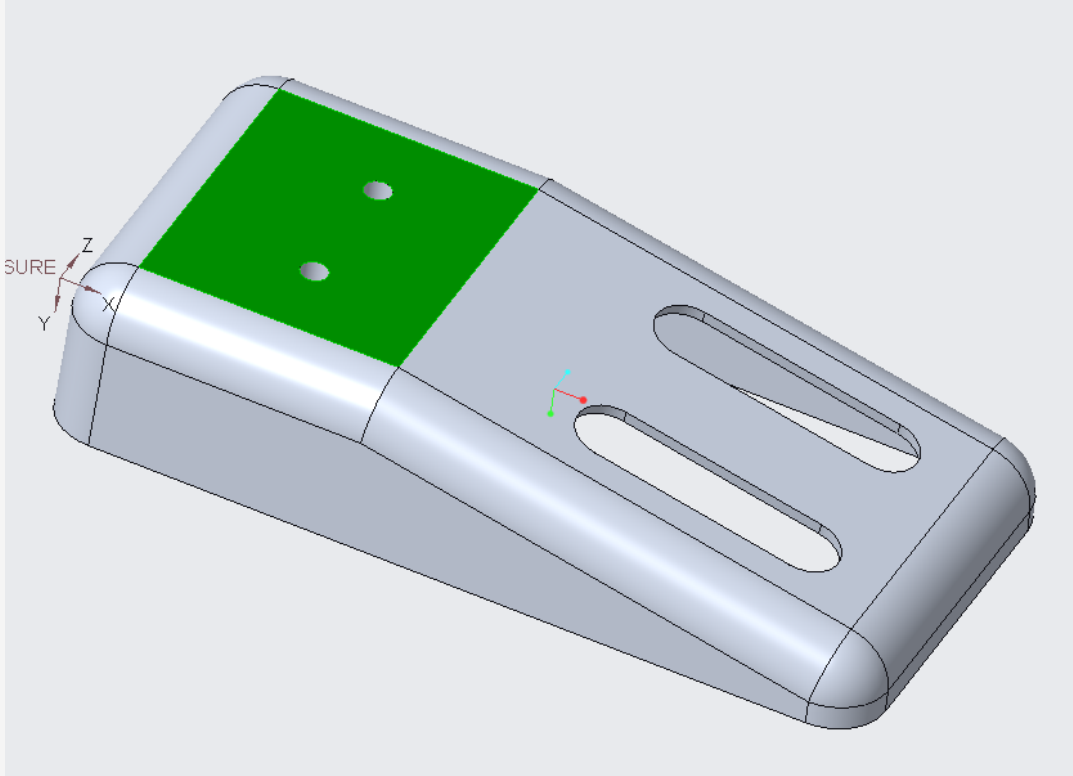
1-Draft_angle

Draft Angles (29 Instances)
Several cavity surfaces feature draft angles of 1.0° to 2.0°, falling below the recommended 5.0° minimum. Insufficient draft complicates part ejection, risks surface scuffing, and increases cycle times.



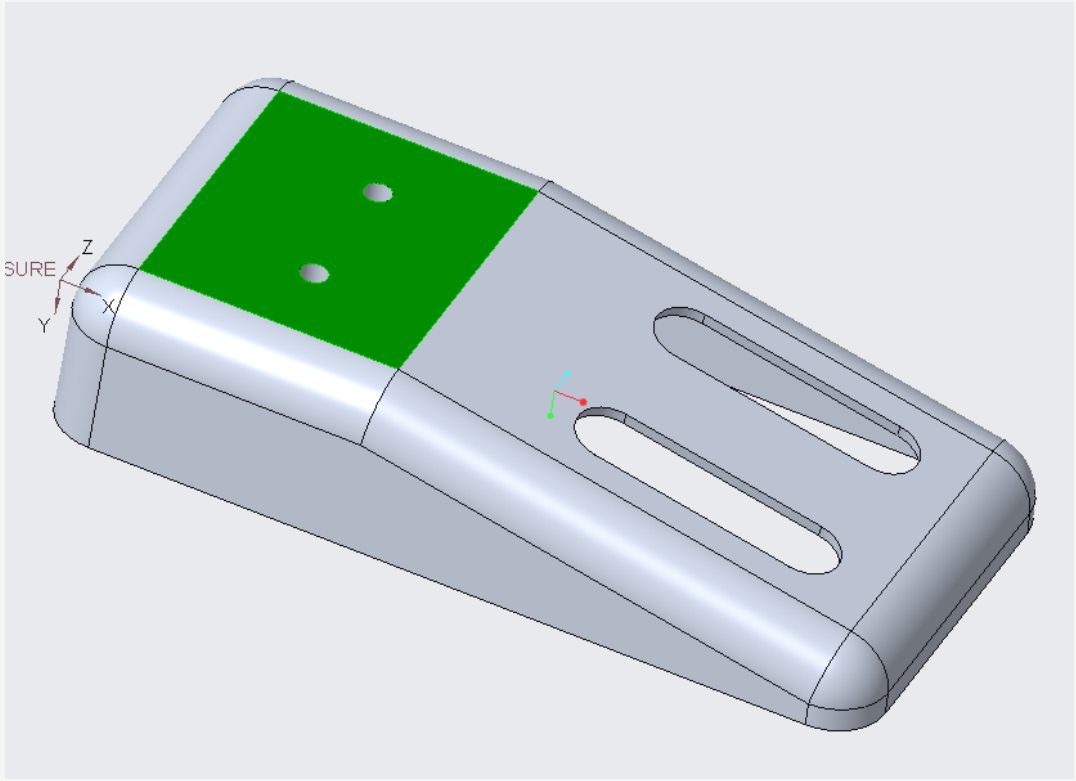
2-Undercut

No undercut was found because I assumed the pulling direction to be perpendicular to the green highlighted surface shown in the picture below.



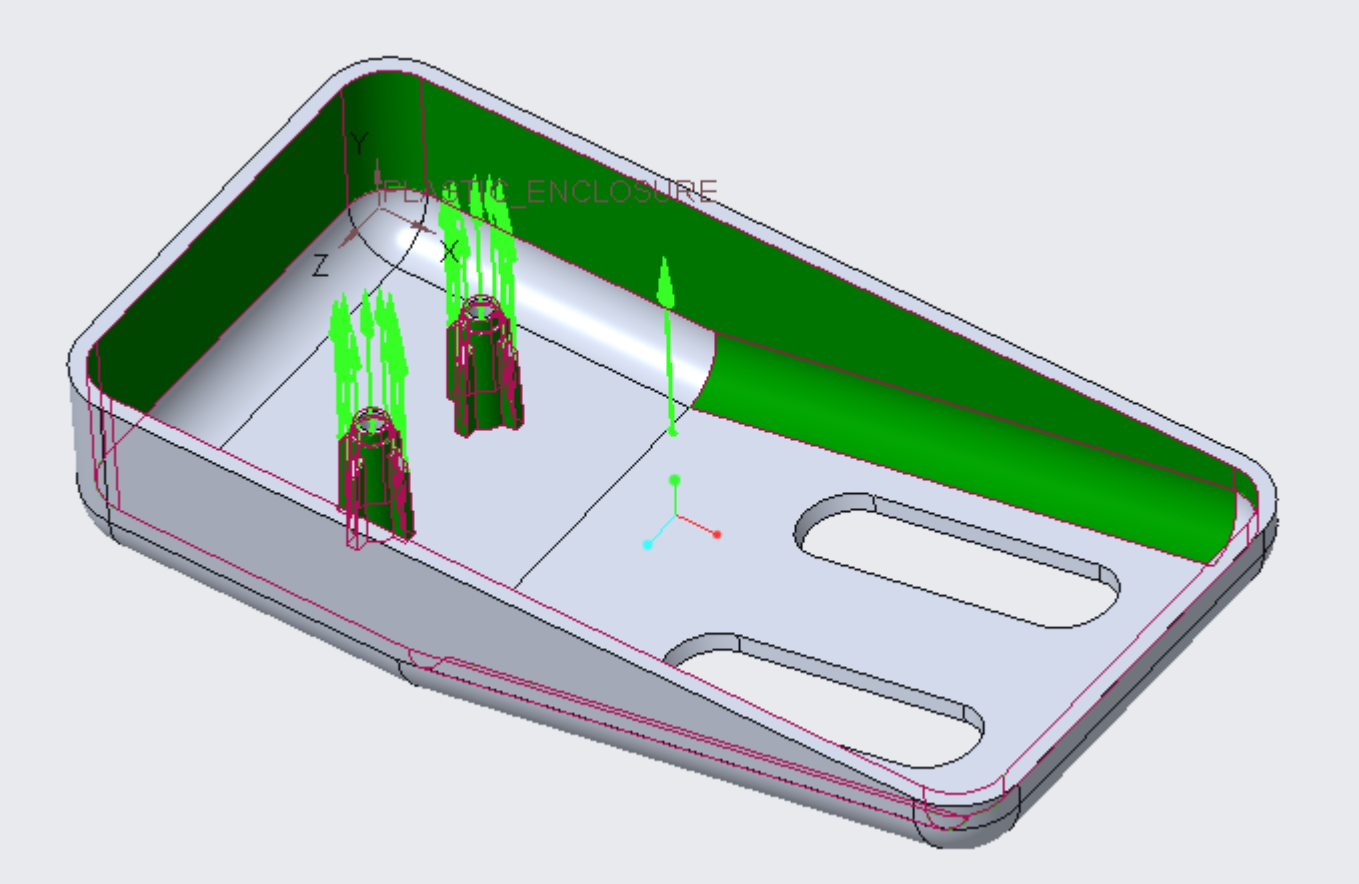
3-Lifter

Because no internal undercuts were found on the plastic enclosure, lifter requirements are currently not needed.



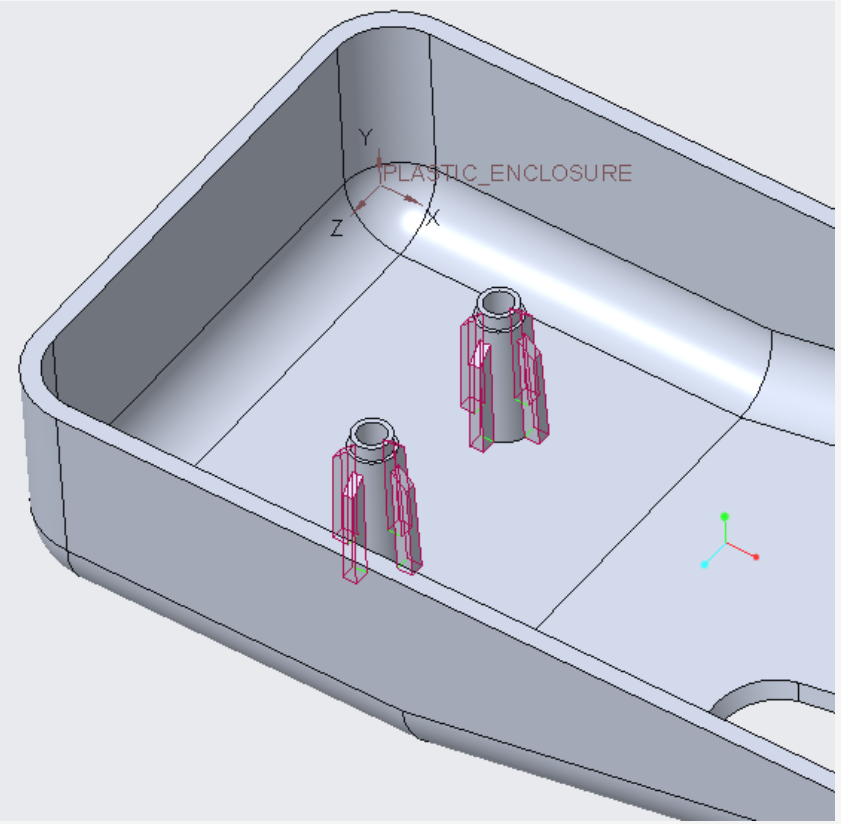
4-Pull

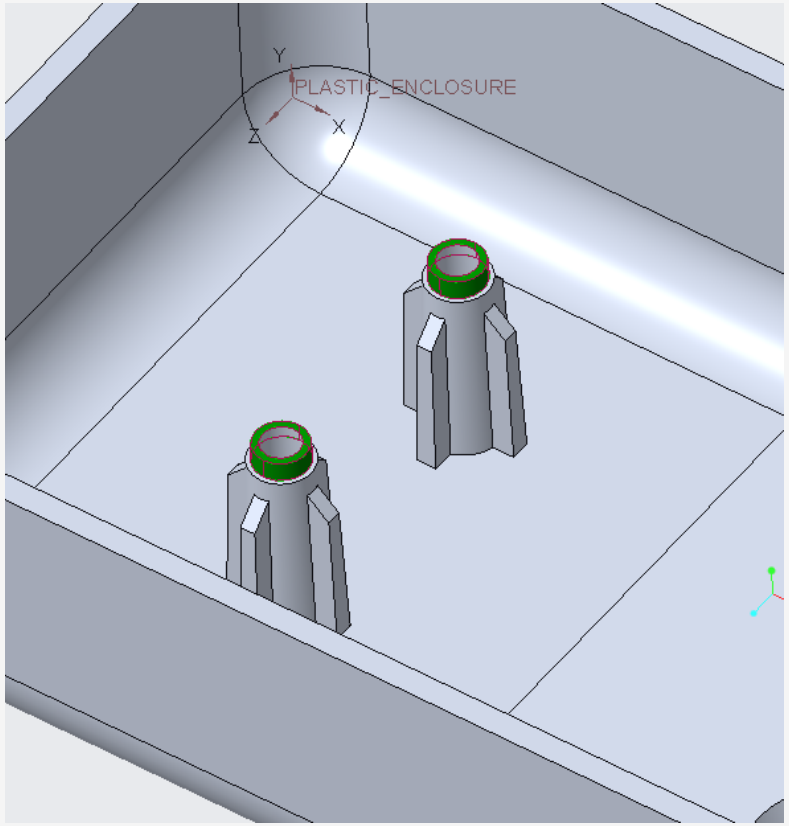
side pulls (or slide cams) are not required because not undercut have been detected on the plastic enclosure



5-Thin_steel_condition

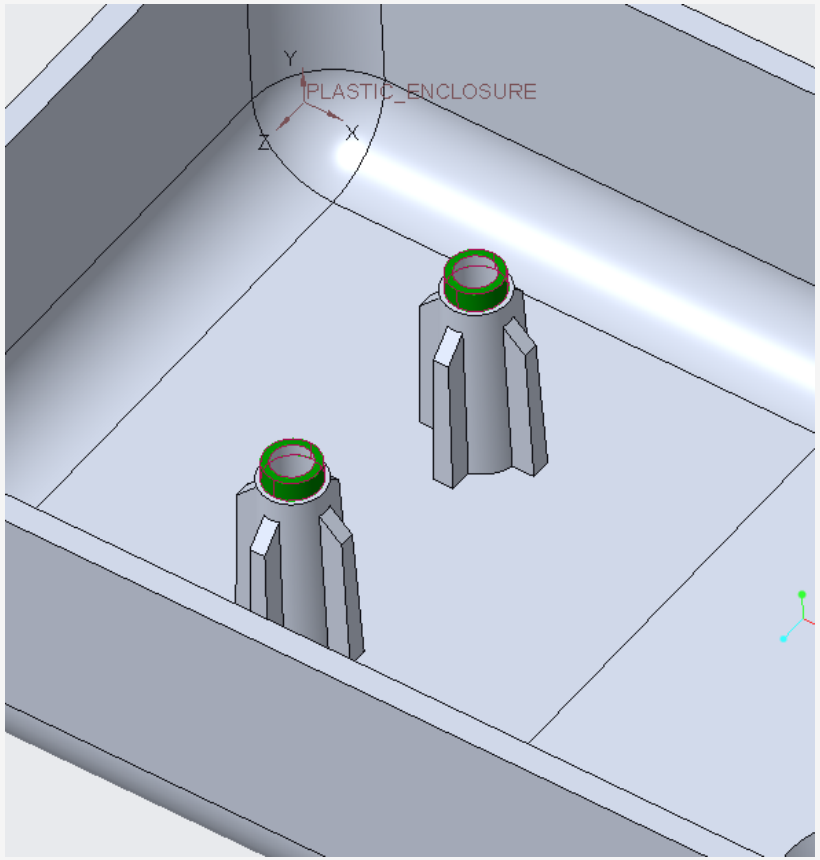
Deep, narrow bosses or ribs can create thin steel sections in the mold





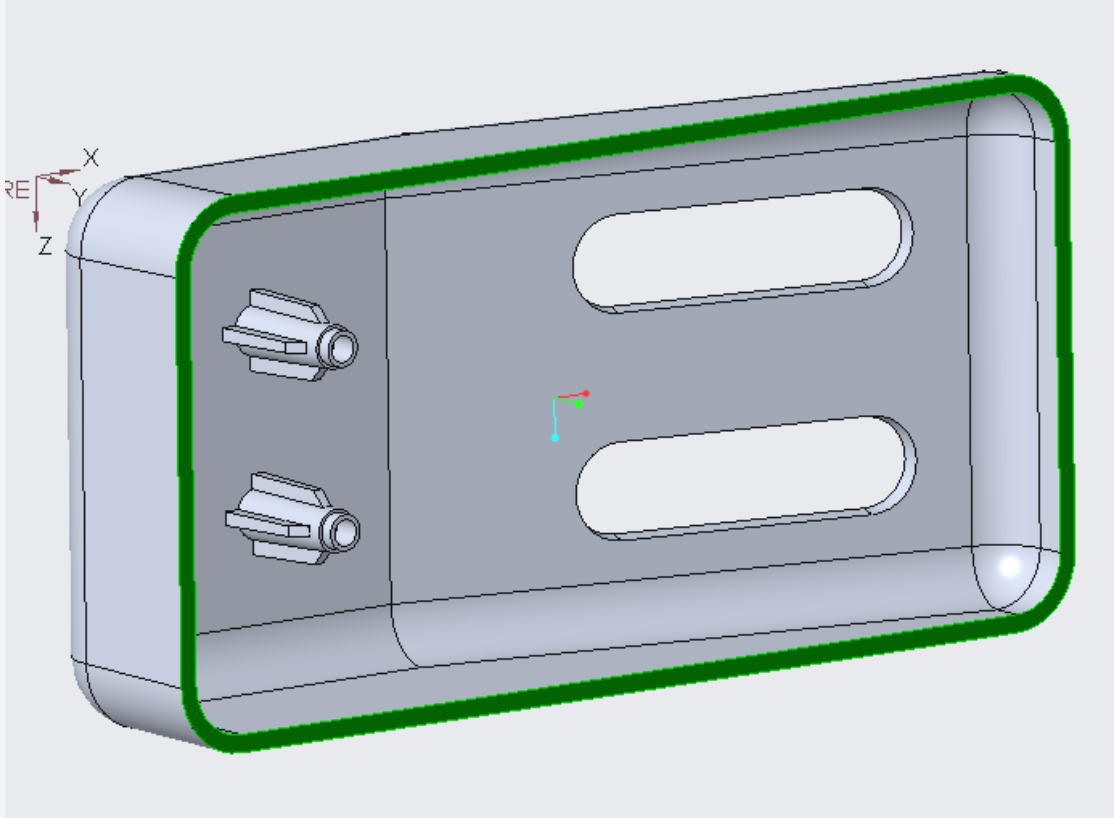
6-Blend

4 instances of missing blends (fillets) on the mounting bosses haved been found and that can be solve by adding a blend/fillet with a minimum radius of 0.492 mm (maintaining a radius-to-thickness ratio of >= 0.25).



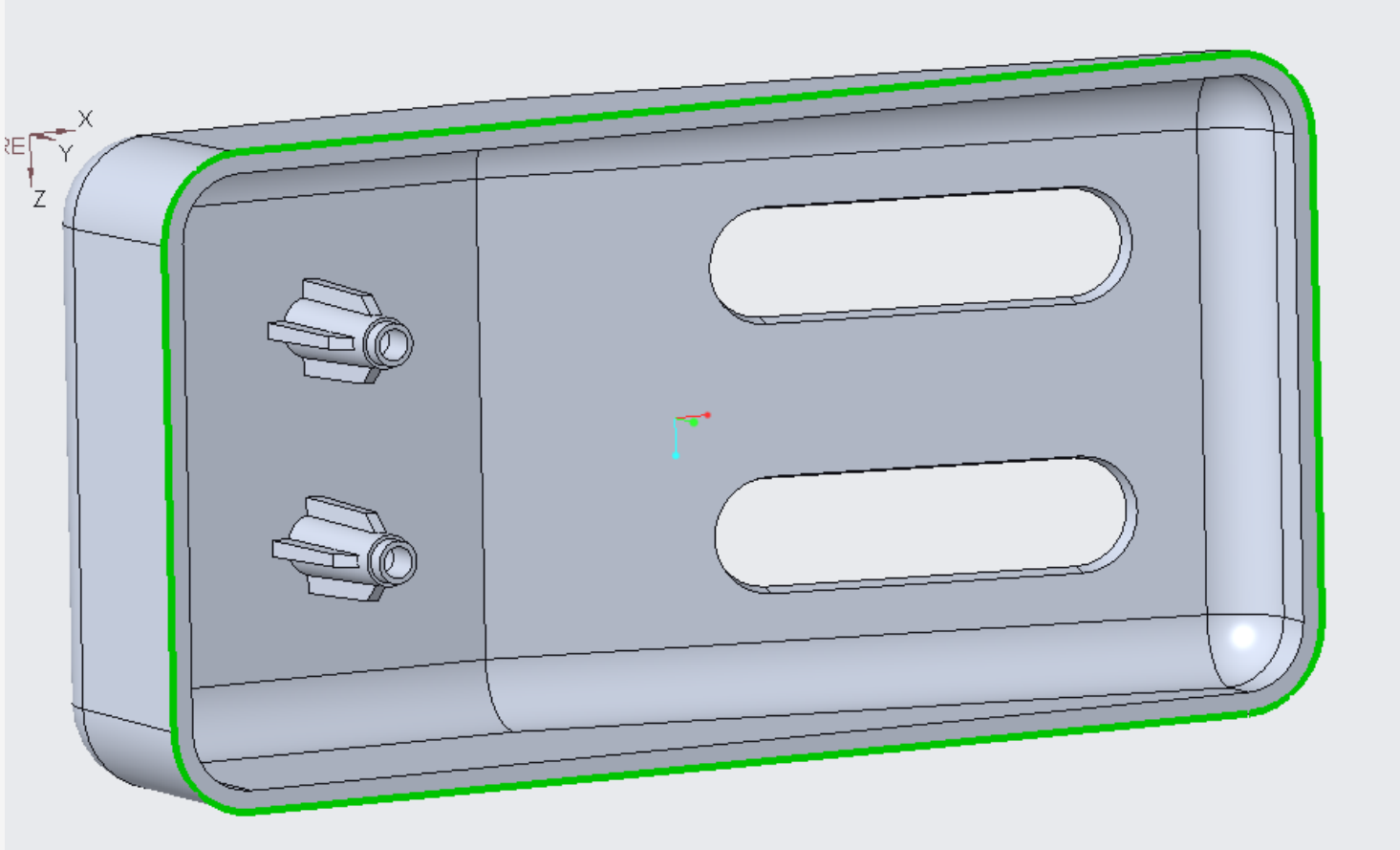
7-Ejector_pin_loc

Ejector pin are placed on a the flat surface at the base of the deep walls and bosses so that they can uniformly push out the plastic enclosure without punching through the thinned out 0.52mm wall sections. (See the green highlighted area in the image below)



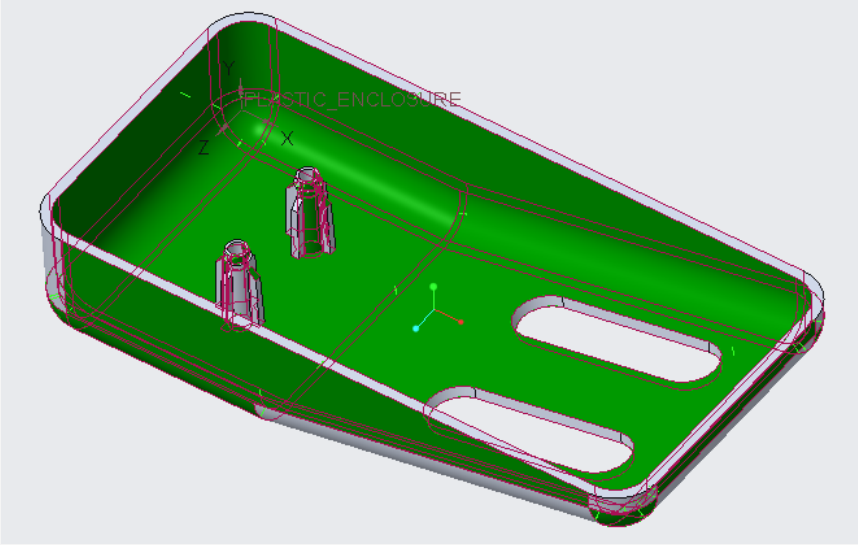
8-Parting_line

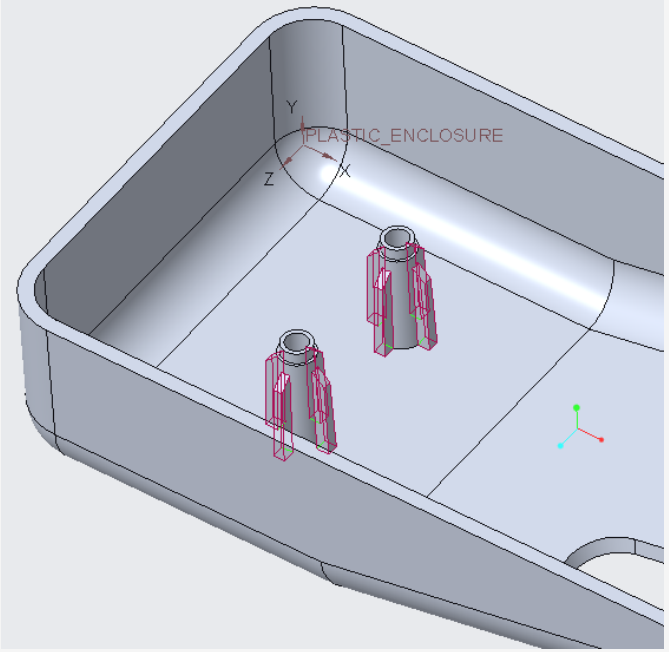
The parting line must be established cleanly around the perimeter of the enclosure. (See the green highlighted area in the image below)



9-Wall-thickness

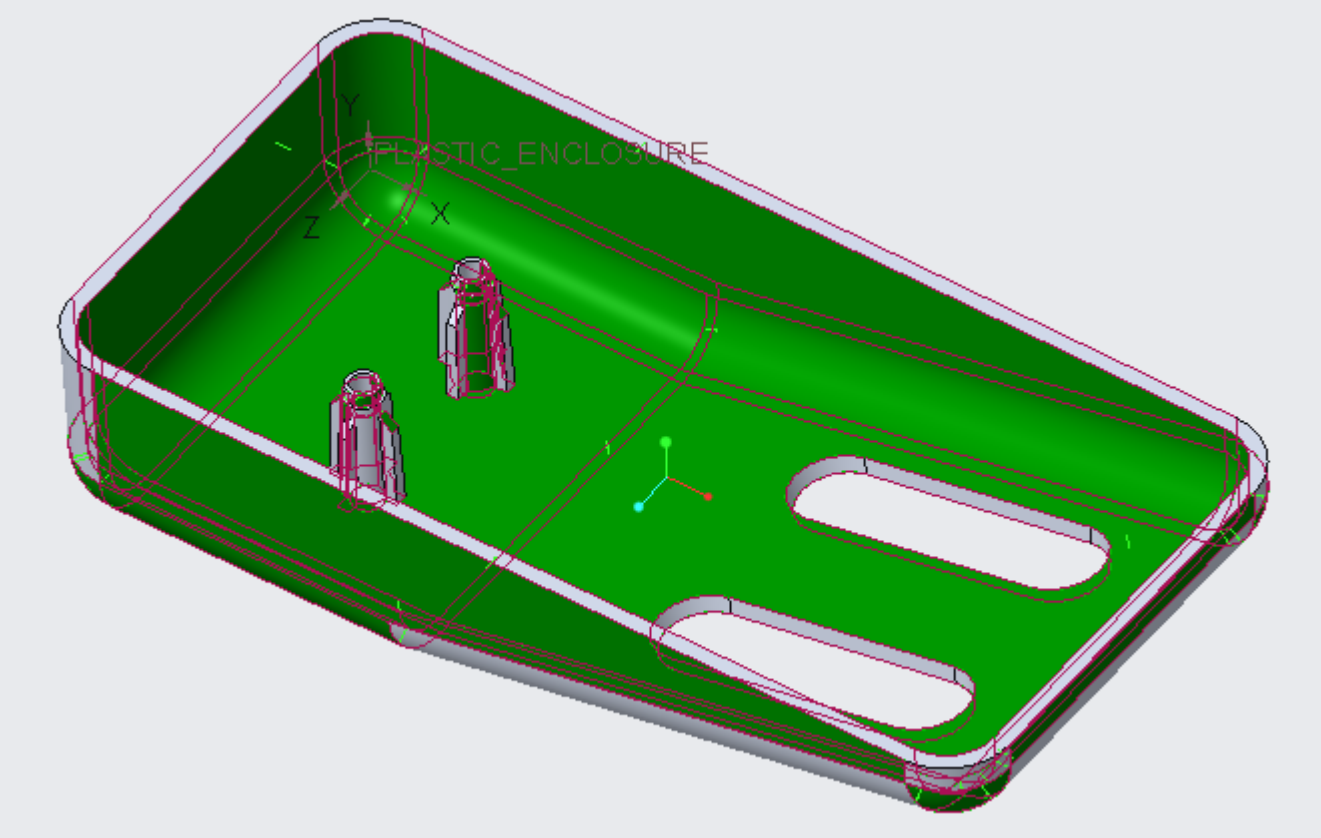
General Wall Thickness (34 Instances): The nominal wall is 1.97 mm, but drastically thins out to 0.52 mm and 0.69 mm in several areas. This will cause race-tracking (uneven plastic flow), short shots, and warping.
Rib Wall Thickness (8 Instances): Ribs are currently 1.55 mm thick (ratio of 0.79 to the main wall). This is far too thick and will cause massive sink marks on the visible exterior of the enclosure.
Action: Standardize the main walls to a minimum of 2.0 mm. Reduce rib thickness to a maximum of 0.788 mm (a ratio of ≤ 0.4 of the nominal wall).





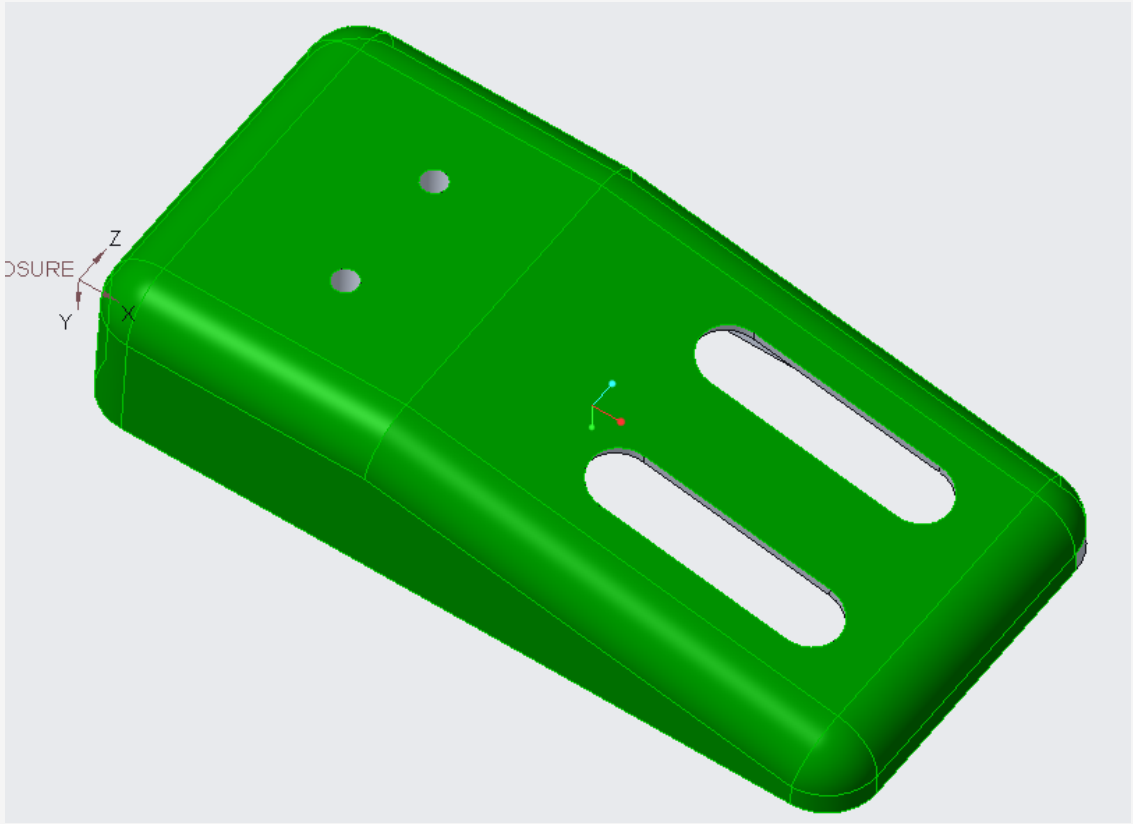
10-Design_clean_up

The engineer must run through the feature tree in SolidWorks to uniformly thicken walls, thin out the ribs, apply the missing 0.49mm and 0.2mm blends to the bosses, and apply the 5° draft angles.



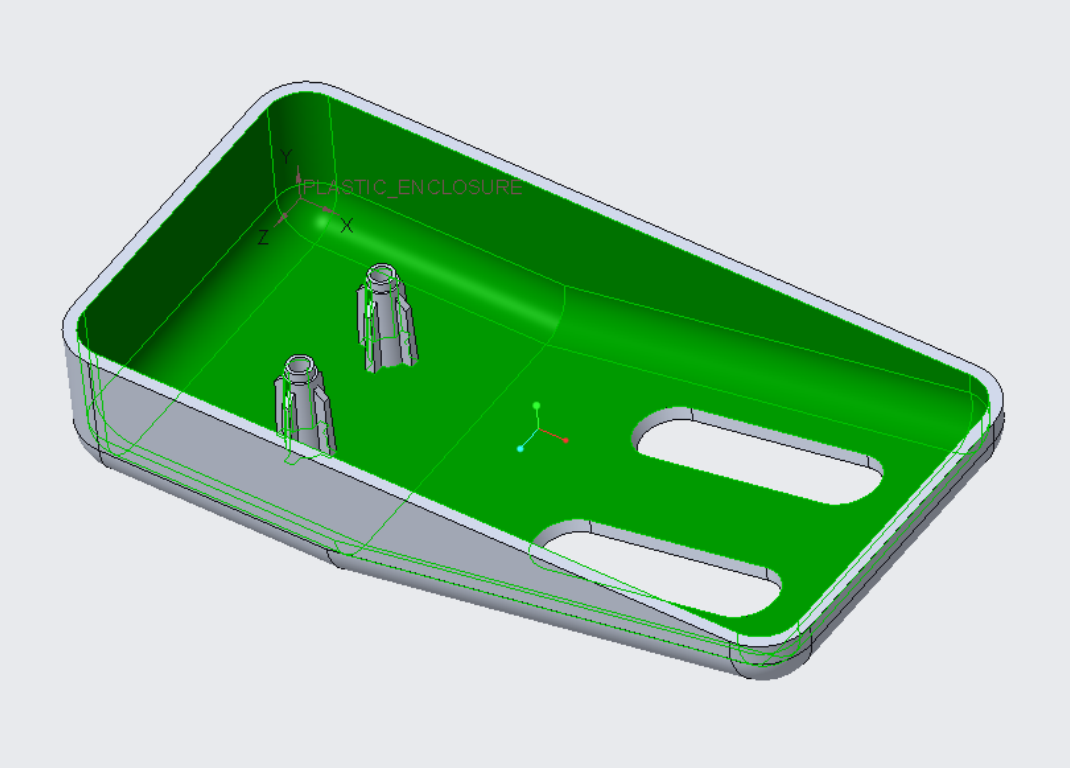
11-Cavity

The cavity corresponds to the face highlighted in green in the image below



12-Core

The core corresponds to the face highlighted in green in the image below



Modlfow analysis

Analysis Software: The simulation was conducted using the Moldex3D/Solid - Enhanced Flow module (Version R14.0 SP3).

Component Analyzed: Plastic Enclosure.

Part Dimensions: 121.31 x 25.00 x 60.87 mm.

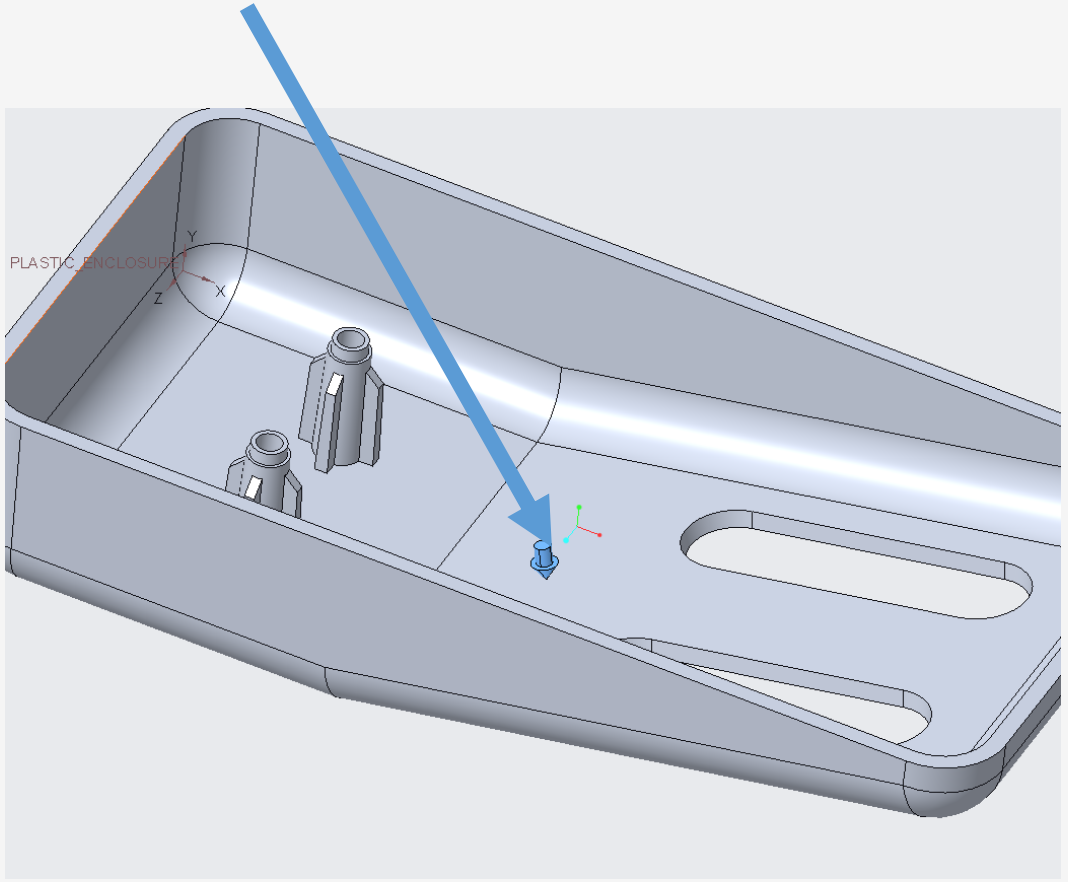
Mesh Details: Polyhedral mesh was generated, consisting of 165,781 total elements and 159,424 nodes to accurately capture the part and runner geometry .

Part Volume: The total cavity volume is approximately 22.85 cm^3.

Polymer Type: Polypropylene (PP).

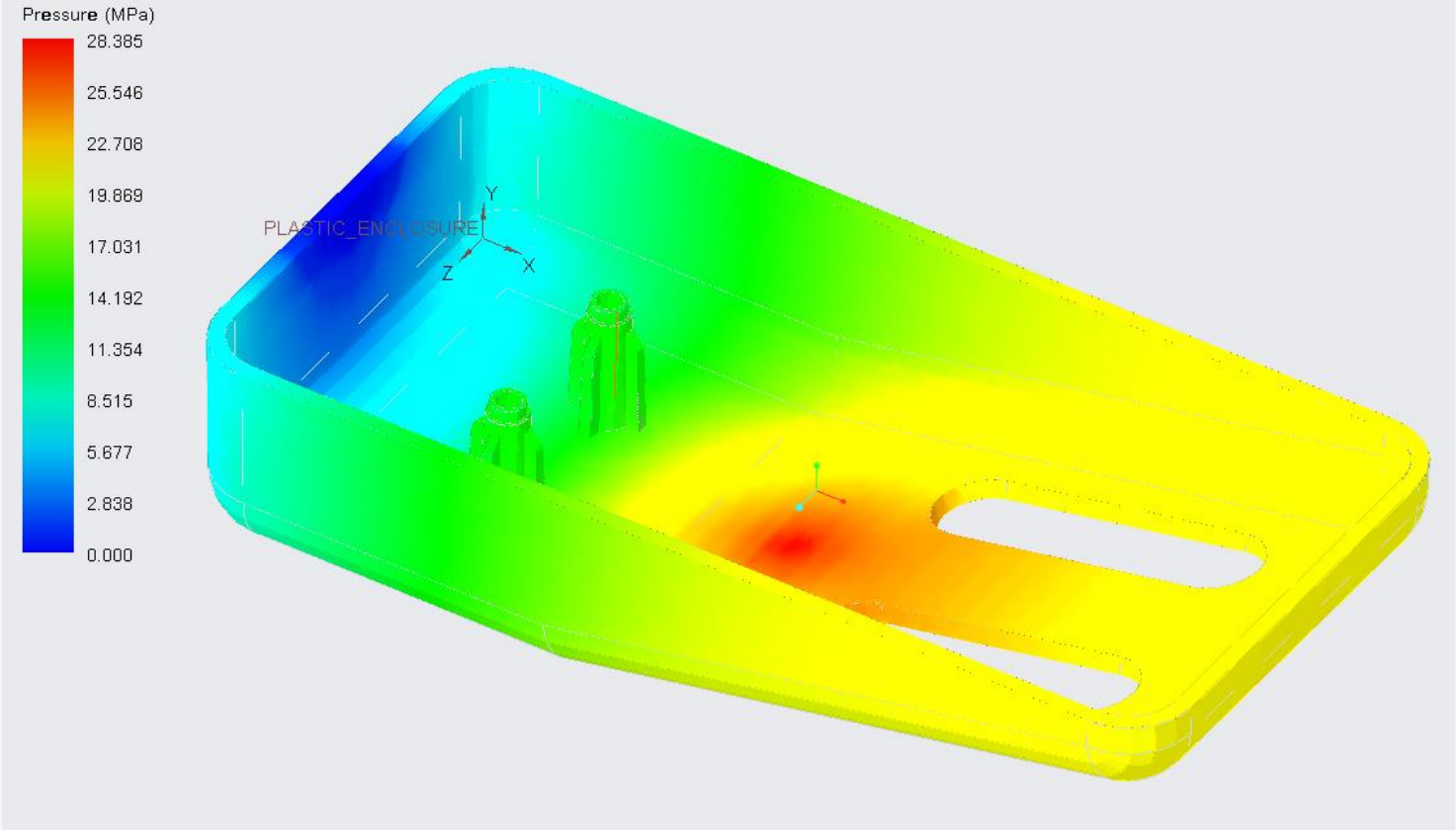
1a- Gate_location

This is the optimal injection point suggested by the software that provides a balanced fill pattern and minimizes overall pressure.



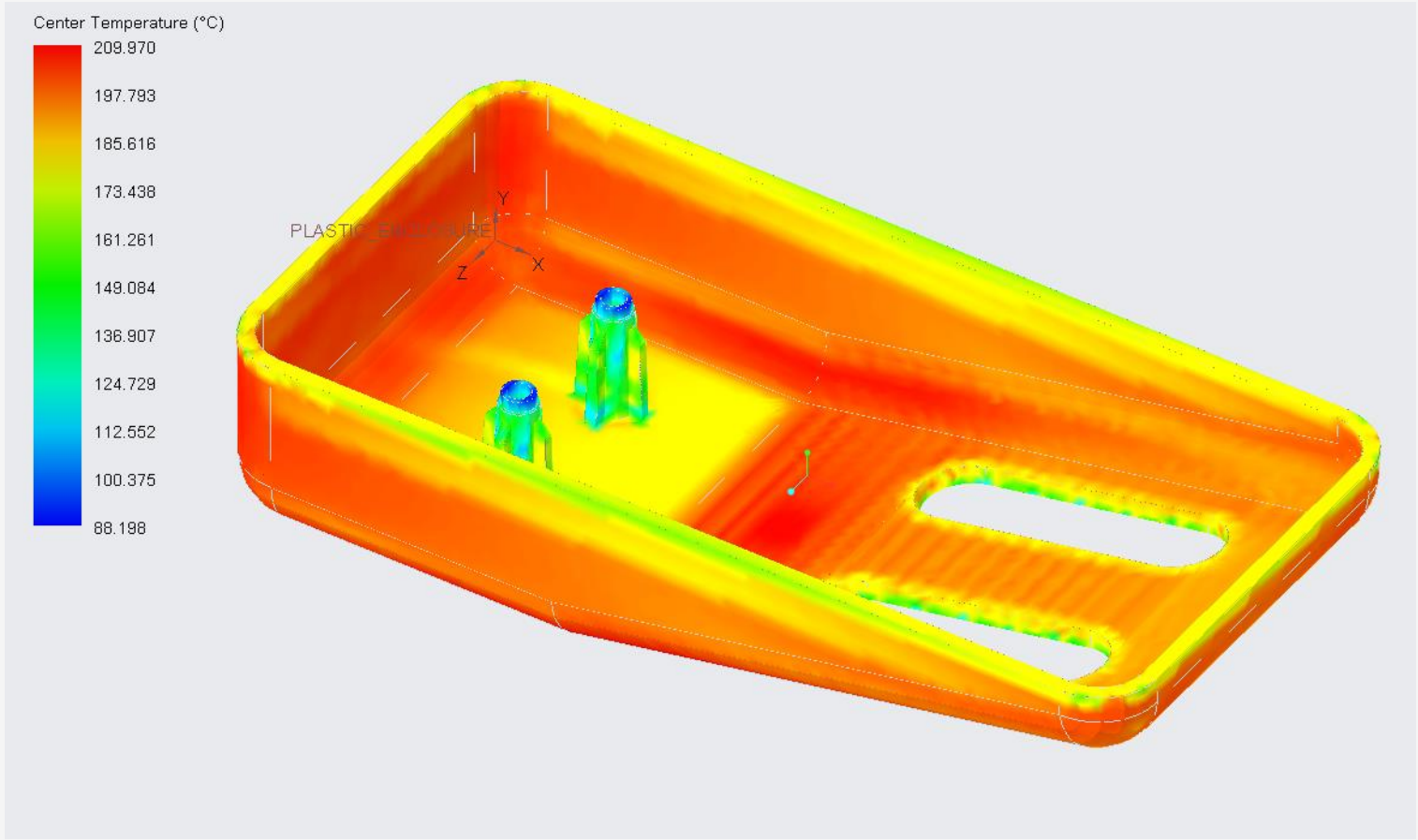
1b- Gate_location

The flow analysis data show that the current single-gate location achieves a 100% balanced fill of the 23.17 cc cavity in 0.647 seconds, requiring a peak injection pressure of only ~35.1 MPa, well below the 140 MPa limit



2- Warp

The simulation calculates an estimated cooling time of 10.3 seconds to reduce the maximum part temperature to 93.6 °C .so there is no risk of warping if cooling channel are placed correctly by the mould designer



3- Shrinkage

The highlighted area exhibit high volumetric shrinkage .