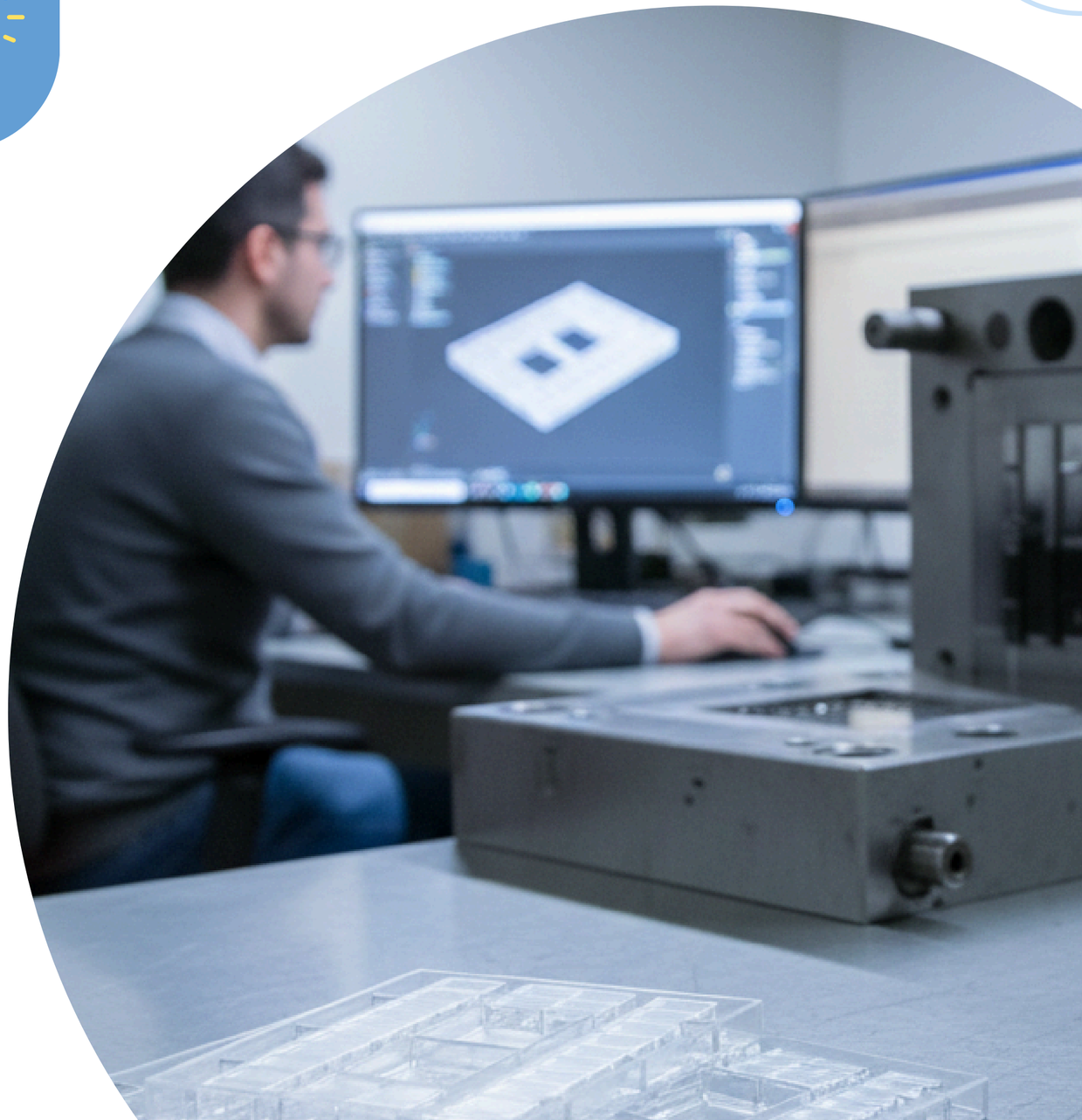
PRATICAL GUIDE

Design for injection molding

Guidelines for effective and manufacturable design



new product development

INDEX

03 INTRODUCTION

04 WALL THICKNESS

05 UNIFORM WALLS

06 HOLLOWS AND SHRINKAGE

07 WARPING

08 COLUMNS

09 RIBS

10 INTERSECTIONS RIBS

11 TEXTURES AND WRITINGS

12 SHARP EDGES

13 INSERTS

14 HINGES

Design for injection molding

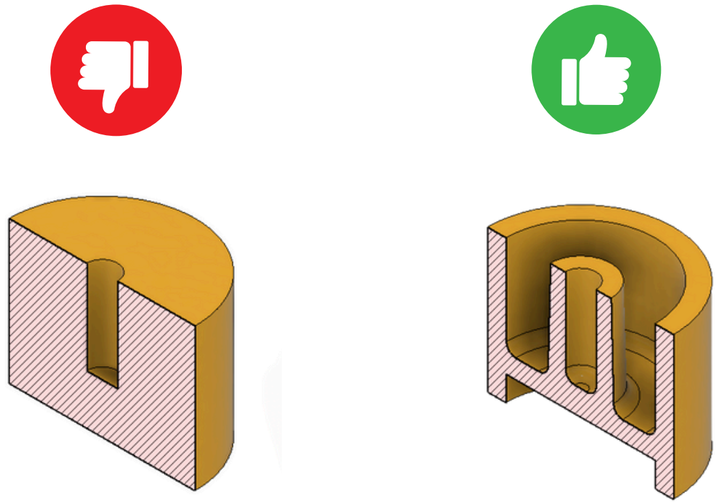
Injection molding is a manufacturing process used to mass-produce plastic parts with consistent quality and precision.

To **optimize time, quality, and costs**, it is necessary to properly design the 3D file to ensure production that fully meets the quality and cost expectations of the printed component.

In this ebook, we'll share useful tips and advice for designers to avoid production problems.

Design for injection molding

WALL THICKESS



DEFINITION

Wall thickness refers to the distance between two parallel surfaces. In injection molding, wall thickness refers to the distance between the inner and outer surfaces.

SUGGESTIONS

In injection molding, the wall thickness must be constant and between 2 mm and 4 mm (0.080 in. to 0.160 in.). This aspect is related to the molding material and the mechanical performance of the product.

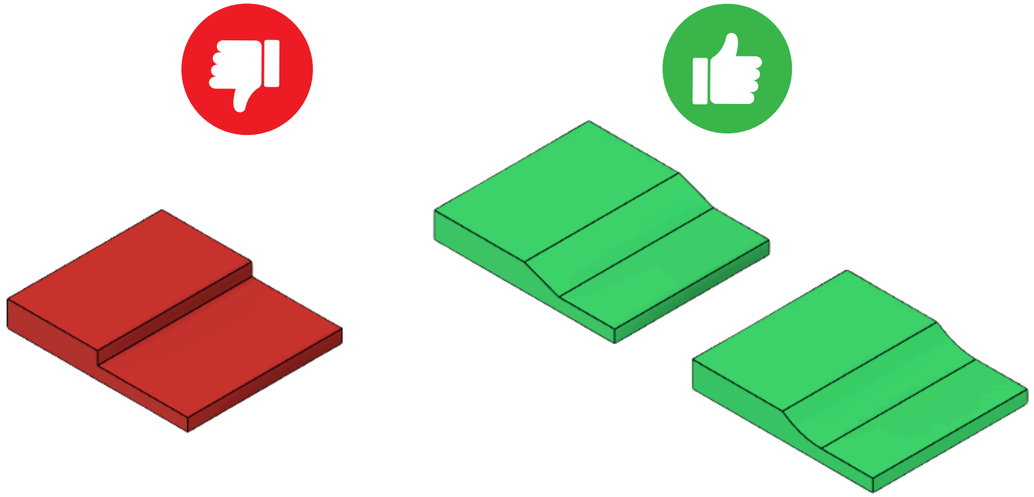
SIDE EFFECTS

Inadequate design could cause:

- Walls that are too thin can cause cracking, breaking, and warping.
- Walls that are too thick can cause an uneven filling of the printed piece, compromising its aesthetics and resistance.
- Uneven walls can cause warping, suction and twisting of the entire part.

Design for injection molding

UNIFORM WALLS



DEFINITION

The term uniform wall refers to a constant wall thickness. In injection molding, it's good practice to design gradual transitions between different wall thicknesses.

SUGGESTIONS

Wall uniformity ensures good molding performance. The use of a leveling system and triangular dowels helps provide support in the corners, preventing distortion. If the design cannot be optimized, post-process coring can be performed to remove material from the thickest areas.

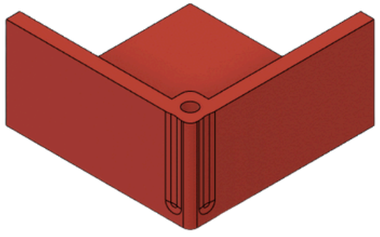
SIDE EFFECTS

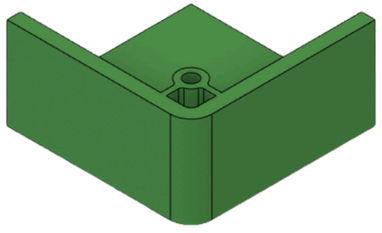
Inadequate design could cause:

- Distortion of the printed part
- Depression, subsidence and in the worst cases cavities on the surface
- Excessive shrinkage in areas that are too thin

Design for injection molding

HOLLOWS AND SHRINKAGE





DEFINITION

Hollows and shrinkage are defects that can appear on an injection molded component due to the intersection of walls that do not have uniform thickness.

SUGGESTIONS

To minimize shrinkage, we recommend keeping the thickness of the ribs between 50% and 60% of the walls to which they are attached.

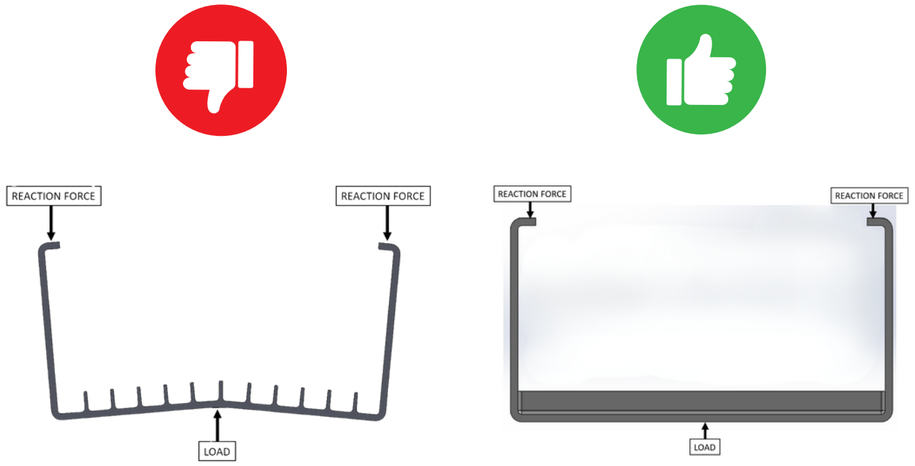
SIDE EFFECTS

Inadequate design could cause:

- Narrowing and depressions in some areas of the plastic piece
- High percentage of rejects on molding
- Reduction of structural strength due to the presence of hollows

Design for injection molding

WARPING



DEFINITION

In the plastic injection molding process, the term warpage refers to an unwanted distortion of the shape of the molded part relative to the mold cavity.

SUGGESTIONS

To minimize deformation, we recommend adding reinforcements such as ribs and fillets. The thickness of the ribs should be between 40% and 60% of the walls to which they are attached. The ribs must be designed taking into account the stresses to which the part is subjected.

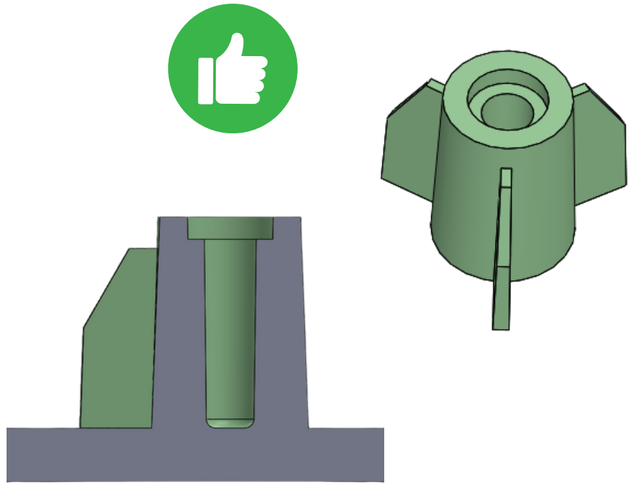
SIDE EFFECTS

Inadequate design could cause:

- High percentage of rejects on molding
- Low quality of molded parts
- Alteration of the aesthetics, functionality and mechanical properties of the printed component

Design for injection molding

COLUMNS



DEFINITION

These are cylindrical or conical elements used primarily to provide fastening points for screws, bolts, or threaded inserts, or to reinforce specific areas of the workpiece. These structural elements also serve to connect two or more parts of the component, maintaining precise alignment and increasing the rigidity of the workpiece.

SUGGESTIONS

Wall thickness: The diameter of the columns should be balanced with the thickness of the surrounding wall, usually maintaining a ratio that ensures good strength without creating defects such as sink holes.

Strengthening: Longer columns or those subjected to high loads can be reinforced with ribs from the base.

Draft angle: To facilitate removal of the part from the mold, it is necessary to insert an adequate draft angle for the pillars.

Generally, the pillar wall thickness should be approximately 40%–60% of the part's main wall thickness. The pillar's outer diameter should be approximately 2.5–3 times the diameter of the internal hole (if present).

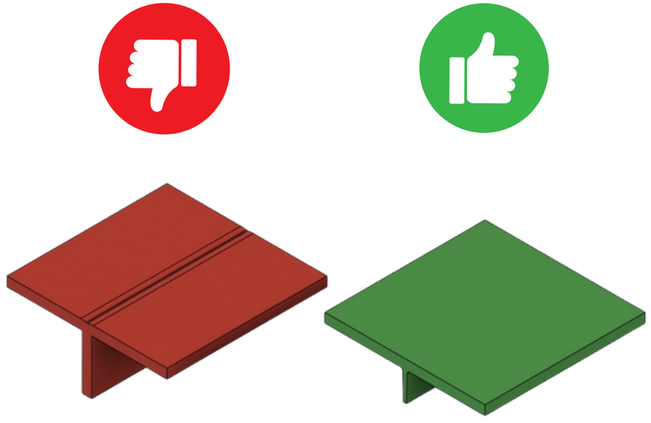
SIDE EFFECTS

Inadequate design could cause:

- Fragility at the fixing points
- Uneven shrinkage problems and non-standard tolerances
- Structural weakness and difficulty in assembly

Design for injection molding

RIBS



DEFINITION

Ribs in the design of a plastic part are thin, vertical structures added to the internal or external surfaces of the part to improve its mechanical strength and rigidity, without increasing the overall thickness of the component. These ribs act as reinforcements that better distribute forces and reduce the possibility of deformation or failure under load.

SUGGESTIONS

Their thickness is generally between 40% and 60% of the wall thickness to which they are attached to prevent surface defects.

The height of the ribs should not exceed three times their thickness to avoid weakening the part and causing difficulty in filling the mold.

The corners at the base of the ribs should be rounded to prevent stress concentration and improve the flow of plastic material in the mold.

The orientation of the ribs serves to ensure maximum rigidity against bending of the part. It is necessary to ensure that the ribs are arranged perpendicular to the bending load. An incorrect orientation may not provide any improvement in structural rigidity.

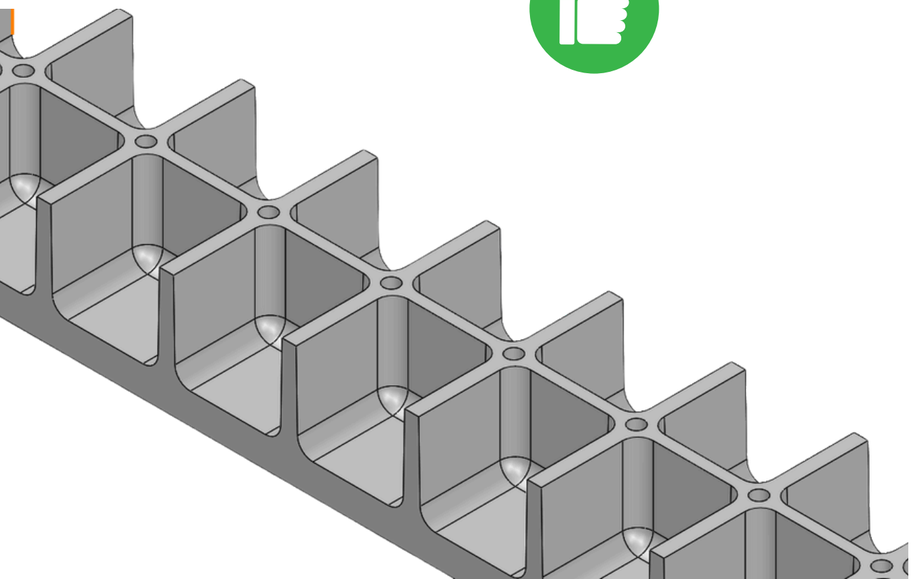
SIDE EFFECTS

Inadequate design could cause:

- Aesthetic defects on the printed piece
- If too thin, incomplete filling may occur.
- If ribs are not designed to the required tolerances, they can interfere with other components during assembly, causing misalignment or the need for rework.

Design for injection molding

INTERSECTIONS RIBS



DEFINITION

Rib intersections in injection molding refer to the points where two or more ribs, designed to strengthen a plastic component, meet or overlap within the part. These intersections play a crucial role in distributing mechanical stress and improving the structural rigidity of the part without significantly increasing its weight or wall thickness.

SUGGESTIONS

In addition to the ribs, it is useful to drill cores at intersections to reduce surface depressions. Drilling cores at intersections reduces the amount of excess material, promoting more uniform cooling and minimizing these aesthetic defects.

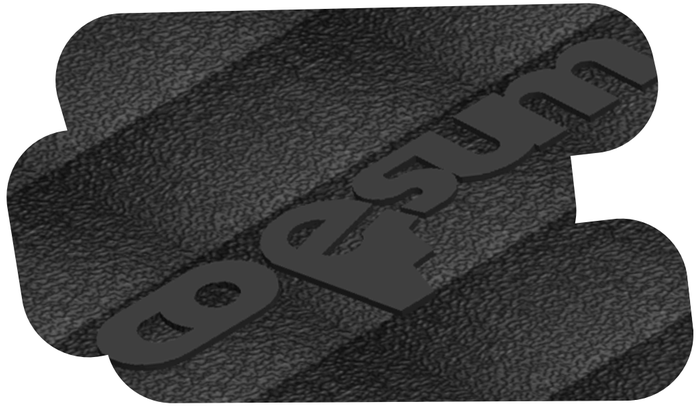
SIDE EFFECTS

Inadequate design could cause:

- Suction and bubbles on the molded part
- Deformation and warping of the component
- High waste with increased production costs

Design for injection molding

TEXTURES AND WRITINGS



DEFINITION

Texture refers to the surface finish applied to the part, which can range from smooth to rough. Textures affect the aesthetic appearance and functionality of the part, such as grip or resistance to scratches and fingerprints.
Letters and lettering in injection molding are primarily used to identify, mark, or personalize plastic components.

SUGGESTIONS

For textured surfaces, increasing the draft angle is essential to facilitate part removal from the mold. The deeper or rougher the texture, the greater the draft angle required. A good rule of thumb is to add 0.5° for every 0.1 mm of texture depth.
Letters must be high enough and deep enough to be clearly legible without compromising the quality of the part. Generally, a depth of 0.5-1.5 mm is recommended, depending on the size of the part and the level of detail required. Letters must also be designed with an adequate draft angle.

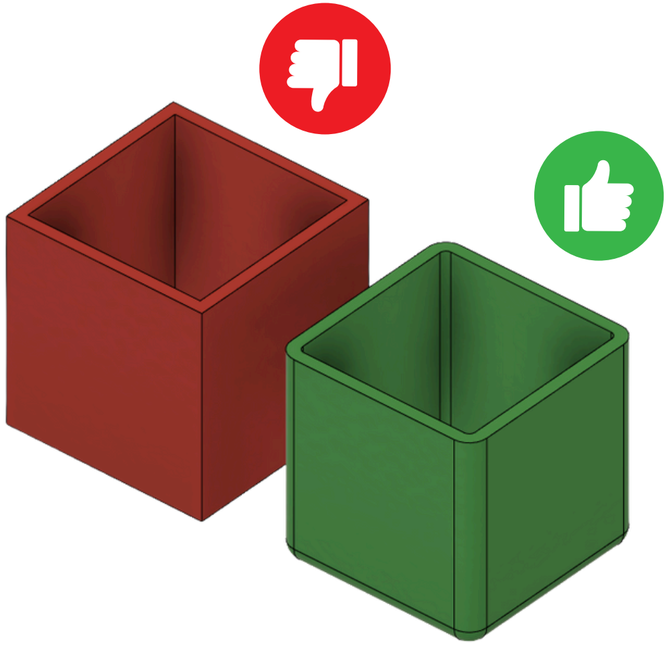
SIDE EFFECTS

Inadequate design could cause:

- Difficulty in extracting the piece with damage to the piece or mold
- Functional problems
- Increase in production costs

Design for injection molding

SHARP EDGES



DEFINITION

These are sharp corners or edges, without roundings or fillets, formed where two surfaces meet at 90 degrees or other sharp angles. These edges are characterized by an abrupt transition between the surfaces, without curves or chamfers.

SUGGESTIONS

Ideally, sharp edges should be replaced with fillets that allow for better distribution of mechanical and thermal stress and facilitate the flow of plastic into the mold. A radius of at least 0.5-1 mm is recommended, but ideally it should be larger depending on the size of the part and its structural function. If a sharp edge or a very sharp corner is necessary, ensure the draft angle is sufficient to facilitate the removal of the part from the mold without damage. The sharper the edge, the greater the draft angle may be necessary.

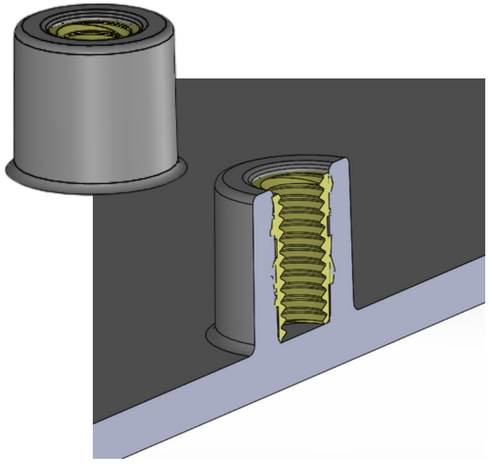
SIDE EFFECTS

Inadequate design could cause:

- Cracks, breaks or fractures on the workpiece caused by a concentration of mechanical stress
- Deformations, distortions or depressions caused by cooling
- Voids and weld lines due to filling difficulties

Design for injection molding

INSERTS



DEFINITION

Inserts are preformed components, typically made of metal or other materials, which are inserted into the mold before the plastic is injected. The goal is to integrate them into the finished part, improving its mechanical or functional properties. Their function is to support the tightening of screws. Plastic alone may not withstand repeated use or strong tightening, while a threaded insert offers a stable and durable connection.

SUGGESTIONS

It's essential to design the columns in accordance with the technical specifications provided by the manufacturer of the chosen insert, to ensure precise fits and the mechanical strength required for proper function of the product. Furthermore, it's essential to provide accurate positioning systems for the insert within the mold, thus ensuring a precise and stable final fit.

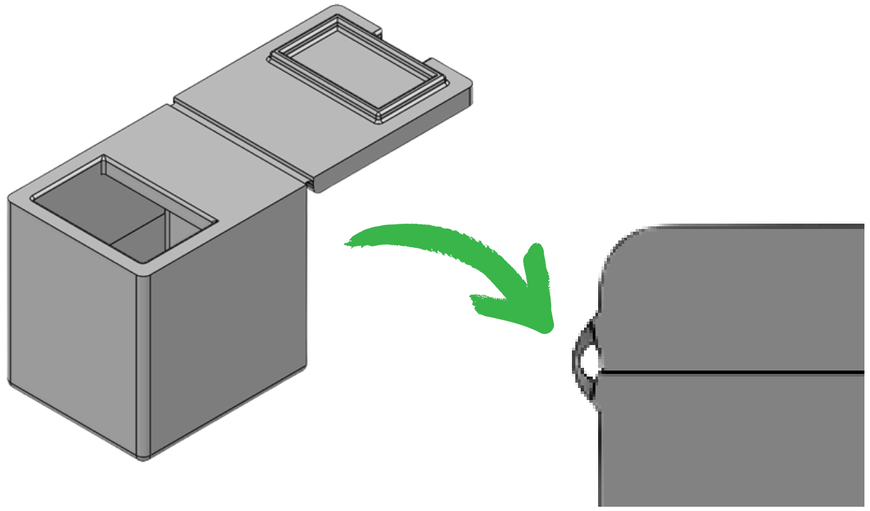
SIDE EFFECTS

Inadequate design could cause:

- Poor mechanical, functional and structural resistance of the product
- Detachment of the insert from the artifact
- Misalignment of the insert with difficulty during assembly

Design for injection molding

HINGES



DEFINITION

These functional elements are designed to allow rotational movement between two connected parts of a component, such as lids, doors, or moving closures. They can be created directly in the plastic part during the molding process, eliminating the need to add separate components for articulation.

SUGGESTIONS

When designing hinges, it is essential that the designer selects a suitable material with high fatigue resistance, capable of withstanding repeated bending without breaking or deforming. Furthermore, it is essential that the part be designed with the correct pull direction and an adequate draft angle. This facilitates removal from the mold, preventing damage during cooling and ensuring uniform filling of the hinge.

SIDE EFFECTS

Inadequate design could cause:

- Irregular movement and reduced flexibility
- Accelerated wear and premature failure of hinges
- Misalignment between the parts joined by the hinge

info@coesum.com

+39 0871 250230

Via Erasmo Piaggio, 75 - 66100 Chieti - ITALY

www.coesum.it

Follow us:

This document and the information contained therein are the property of IDEA & SVILUPPO Srl.