DESIGN GUIDE

Best Practices for Reducing Stress Cracking in Molded Plastic Parts

AVIENT™

TABLE OF CONTENTS

INTRODUCTION

DESIGN BEST PRACTICES TO REDUCE STRESS CRACKING IN MOLDED POLYMER PARTS

Recent viral outbreaks and pandemics such as SARS, Ebola, and COVID-19 have highlighted the importance of cleaning and disinfection for reducing disease transmission. Designers and engineers are increasingly concerned about the impact of more frequent cleaning and the resulting chemical exposure. Both can lead to material failure through a phenomenon called environmental stress cracking (ESC).

ESC is defined as the premature embrittlement and crack propagation of a polymer part. It is caused by the combination of stress and chemical exposure. It is believed that ESC is the leading cause of plastic component failure[1], accounting for 25% or more of observed failures in the field. Adding to the challenge, the applied stress that leads to failure can fall below the levels that normally cause mechanical damage, making it difficult to predict when failure might occur.

Factors that cause ESC include the type and frequency of chemical exposure, and any residual or applied stresses to the plastic part. Residual stresses occur when latent areas of weakness emerge as a result of the part's design and processing parameters. Increased stresses during processing localize in areas of changing geometry due to non-uniform flow and differential cooling rates. These stresses can accelerate the method of cracking by forming and propagating voids.

Strategies to counteract failure in this "new normal" of heightened disinfection are two fold: One, selecting materials with improved chemical resistance and two, modifying part design to reduce residual stresses. This guide will focus on the latter and highlight best practices to reduce areas of localized stress at common areas of concern, such as corners and edges as well as reinforcing and assembly features.

WALL THICKNESS

Uniform wall thickness will achieve a uniform cooling, fill, and shrink rate. Non-uniform walls will contribute to warp, stress, and cracking by creating areas of the part that cool faster. These areas then pull on slower cooling areas.

All walls should be the same thickness except for ribs, bosses, snapfits, and interlocks. Wall thickness should be as thin as possible for the application. It should be dictated by the structural and flow requirements of the part. The wall thickness of the part will dictate the cooling time as seen in Figure 1.[2]

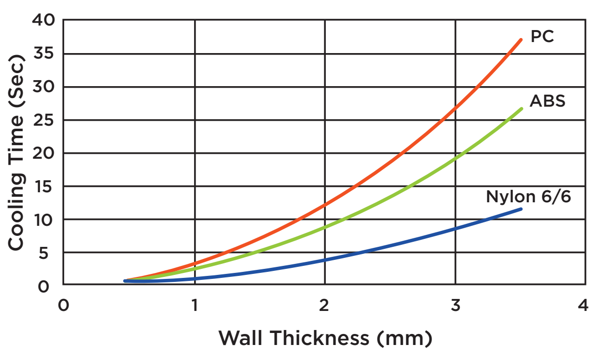


FIGURE 1 - Wall thickness vs. cooling time of various plastics

When designing for even wall thickness, it is important to transition the wall thickness using rounds or radii. Figure 2 shows the proper way to transition different wall thicknesses.

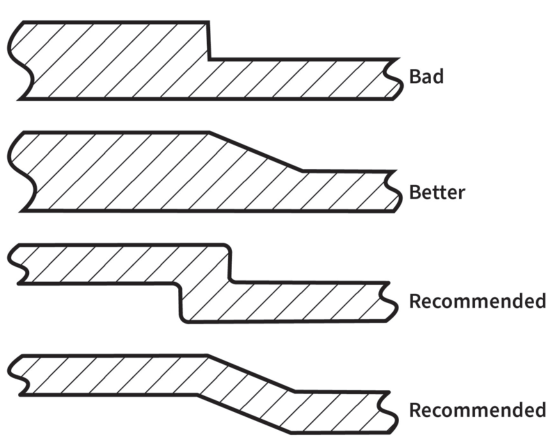


FIGURE 2 - Designing for wall thickness changes

When designing edges and corners, allow for generous radii to maintain constant wall thickness and reduce stress concentrations. Figure 3 shows the guidelines to use for internal and external radii. These radii need to be concentric. Figure 3 also shows how the wall thickness can suddenly change when outside and inside rounds are not concentric.

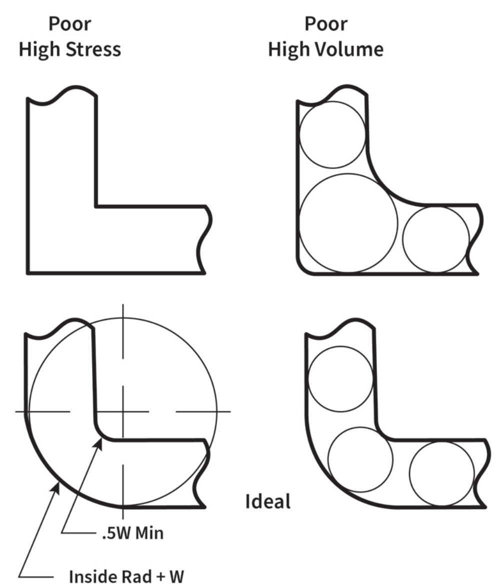


FIGURE 3 - Internal and external radius guidelines

Sometimes, tight radii are required to create specific device aesthetics or to allow for other components. A tight radius increases the wall thickness in the corner. Coring, or removing unnecessary material in the corner, will keep the radii concentric. Figure 4 shows a top and side view of a corner with constant wall thickness where coring has been applied.

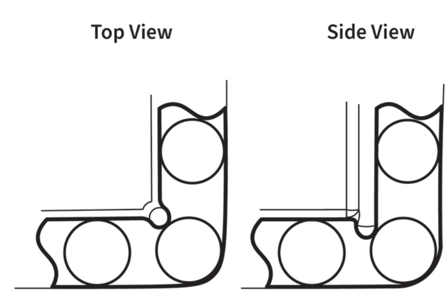


FIGURE 4 - Coring for tight radius design

CONTOUR

Where possible, contouring the products' exterior form, as shown in Figure 5, can help to eliminate pooling of fluids. This will allow fluids to bead and flow off of the product. Recess holes are recommended so that fluids do not ingress.

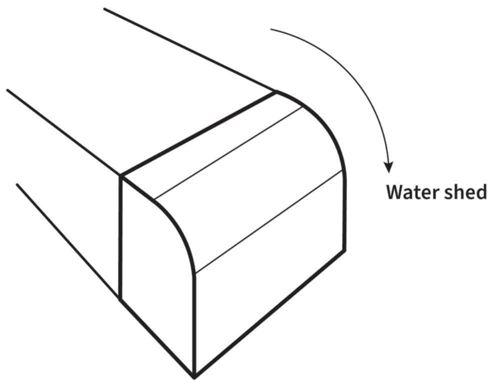


FIGURE 5 - Proper contour design

DRAFT

The draft on a part is a taper on the walls perpendicular to the parting line to allow the part to release from the mold. Usually, it is 2° for parallel walls. However, non-parallel walls should be ½°. Part walls need to remain parallel when adding draft. One thing to avoid in adding draft is a large variation in wall thickness. Figure 6 shows how to properly draft a part to keep a nominal wall thickness.

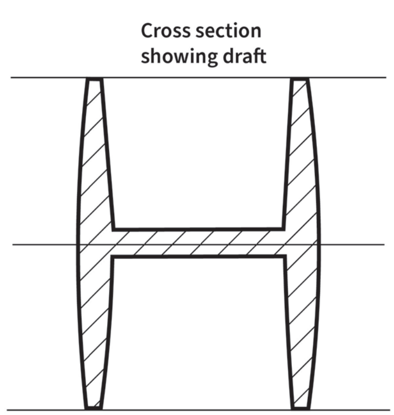


FIGURE 6 - Drafting guidelines for nominal wall thickness

RIBS

Ribs are thin, reinforcing features that can add structure and strength to a part. Ribs will also cause a larger wall thickness throughout the part. Figure 7 shows how the rib creates a thick region. These regions are excellent options for ejector pins because they present increased surface area.

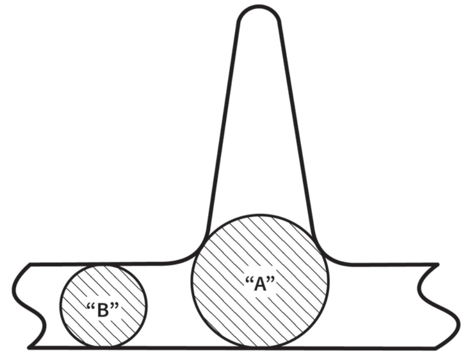


FIGURE 7 - Wall thickness changes due to rib placement

Figure 8 illustrates the recommended rib design guidelines. The minimum distance for rib spacing is three times the nominal wall thickness (3W). The maximum height should be about 2.5 times the nominal wall thickness (2.5W). Keep in mind, the width of a rib will be different depending on the material used. A 1:1 wall thickness will produce sink and cause stress where the ribs connect with the main wall. Ideally, ribs should be a little thinner so as to not create thick spots –75% of wall thickness for a low-shrink material and 50% for a high-shrink material. The round located at the base of a rib should be 25% of nominal wall thickness, and all ribs should have a draft angle from ¼° to 1°. The design of a rib should be derived from the structural, flow, and assembly requirements of the part.[2]

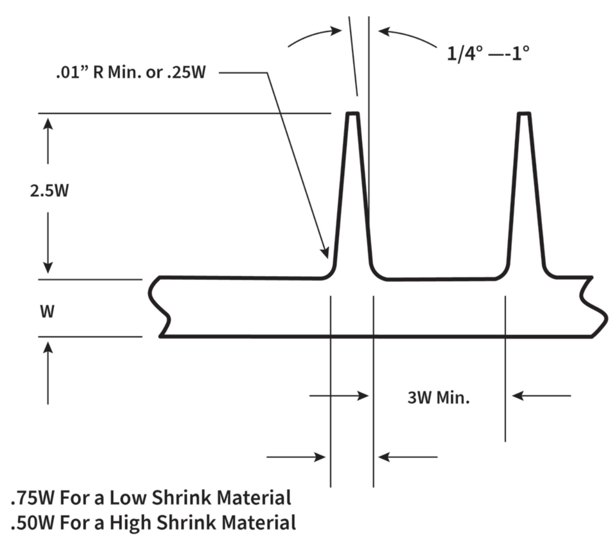


FIGURE 8 - Rib design guidelines

BOSSES

A boss, demonstrated in Figure 9, is a protruding feature that allows for fastening components. This fastener method keeps one smooth surface, and has the fewest parts required of mechanical fasteners. It also includes mating plastic threads formed during assembly. Unfortunately, this method has limited durability and assembly repeatability as fasteners may wear and stretch the plastic thread. Often a necessary feature for assemblies, it usually causes some warp or sink. As a result, stress cracking frequently happens around a fastening feature as the holding force adds stress. If bosses are improperly designed, the stress will be amplified.

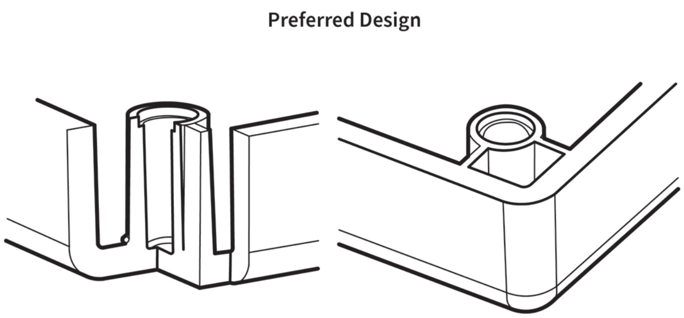


FIGURE 9 - Proper boss design

A boss design should be dictated by the core size and the wall thickness of the part. The inside diameter of the boss should be the diameter of the core or screw being used, and depends on the type of fit desired, interference, and/or clearance. The outside diameter of the boss should be 2.5 times the inside diameter of the core. The height of the boss should be 2.5 times the inside diameter of the boss. There should be a round at the intersection of the boss and a nominal wall that is 25% of wall thickness. The boss should extend into the nominal wall of the part, to avoid a thick region and possible sink area. The round at the bottom of the boss should be 10% of wall thickness. Draft on the inside of the boss should be ½°, with a 45° chamfer at the top of the boss. Draft on the outside of the boss should be 1°. If a boss is to be attached to a primary wall, avoid thick regions. Instead, use gussets to structurally support the boss.[2] Figure 10 illustrates these stated boss design guidelines.

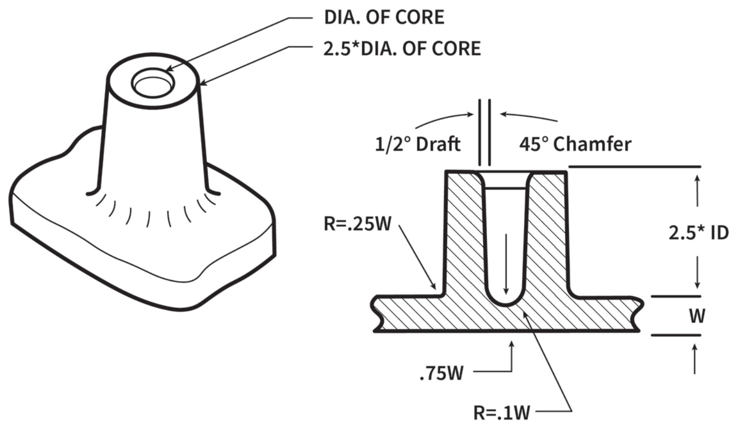


FIGURE 10 - Boss design guidelines

INSERTS

Size the inside diameter of bosses at 50-60% of thread engagement. These must be sized large enough to engage the thread only, but not so small that the boss will widen with the shaft of the screw. Use compression limiters where mechanical fasteners are needed to assemble the system. This helps to reduce added compressive loads or pressures on the plastic. Choose threaded inserts with properties that will minimize the stress on bosses. For example, this can be done by using inserts with thermal expansion rates matched as closely as possible to the polymer used to mold the boss. If using metal inserts, brass and aluminum are typically preferred over steel.

Stresses can also be reduced by minimizing sharp edges on the insert. Ideally, all polymer contacting edges should be round or radiused. Sharp edges on an insert can create sharp edges on the polymer that result in a stress riser. It may be difficult to minimize sharp edges due to the machining of the inserts but practically minimal sharp edges are acceptable. Figure 11 shows an example of a standard spiral molded-in insert that does a good job of minimizing sharp edges. Another option, which is more ideal for stress reduction, includes octagonal inserts with more rounded edges versus standard spiral inserts.

For press-in inserts, heat press installation is preferred. The heat will melt the plastic around it, allowing the plastic to relax, reflow, and solidify. This helps to reduce the ring stress around the insert.

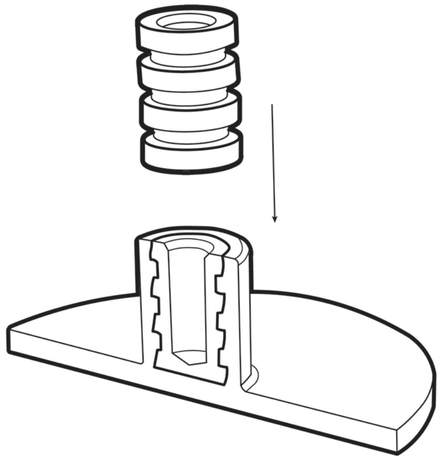


FIGURE 11 - Molded-in inserts

GUSSETS

Gussets are support features that reinforce the connection between vertical features (such as walls or bosses) and the part's floor. Figure 12 is one example of gussets used to help reduce warp on long flat faces by adding strength to the wall, preventing it from buckling or curling in on itself.

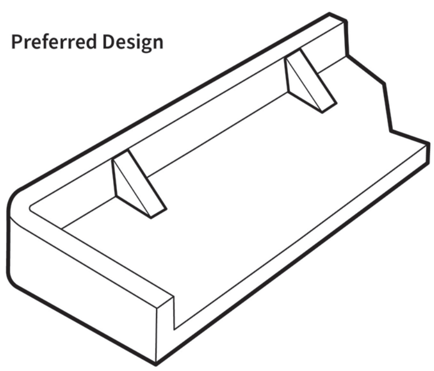


FIGURE 12 - Gusset placement on side wall

Guidelines similar to those for ribs can also be applied to gussets. Aim for 75% of wall thickness for a low-shrink material, and 50% of wall thickness for a high-shrink material, with a draft angle from ¼° to 1°.

Gussets are used to support a boss because they do not cause thick sidewall regions. Thick sidewalls will cause sink in a part. That's because the plastic will shrink away from the mold wall due to the increased heat in these areas. Figures 13 and 14 provide design guidelines that should be followed when designing for gussets and side wall bosses.

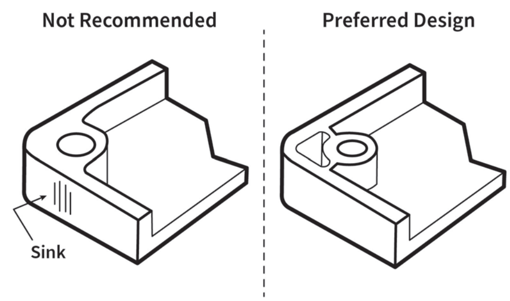


FIGURE 13 - Side wall boss design guidelines

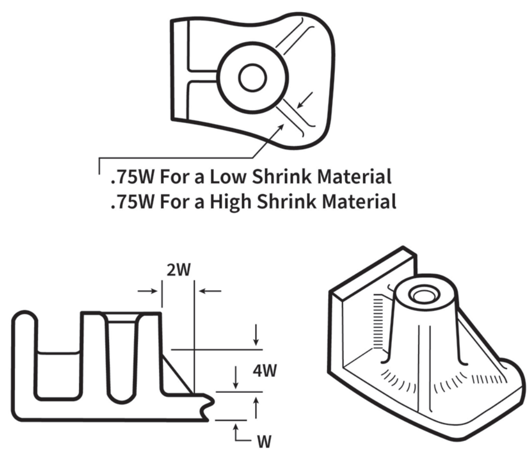


FIGURE 14 - Gusset design guidelines

THROUGH-HOLES

Minimize through-holes when possible to reduce plastic weld lines. These lines are an intrinsic point of weakness. Figure 15 shows how practically round or oval holes with a long axis parallel to the flow path would be preferred over square or rectangular holes. If rectangular openings are required, ensure that the long axis is parallel to the flow path and corners are rounded so that the polymer flows well around the hole and doesn't create too much turbulence. In some filled plastic materials, through-holes are best added in a post-process step.

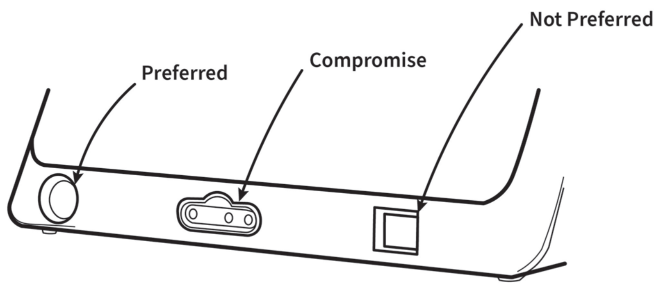


FIGURE 15 - Through-hole shapes

Holes should follow best practices for a polymer's viscosity and tensile strength when determining the ideal distance to the edge versus the wall thickness. If the hole is too close to the edge, the material may not flow well around it. The results are sink marks and cracking. Holes are placed in a part for either structural support or cosmetic reasons. There are different guidelines to follow for each. For structural support, such as for mounting or fastening, each hole should be placed at least one diameter of the hole apart. They should be placed two times the diameter from the outside wall and one times the diameter (D) from an inside wall. Figure 16 shows the proper design for structural holes.

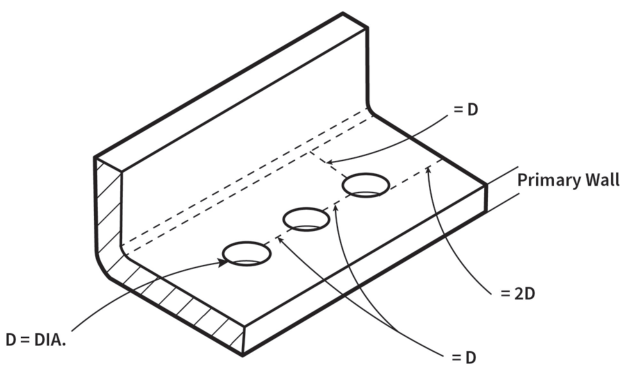


FIGURE 16 - Structural hole design guidelines

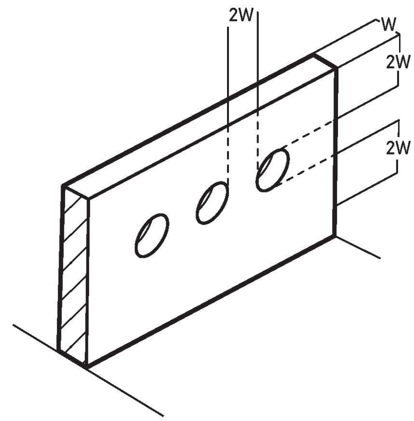


FIGURE 17 - Non-structural hole design guidelines

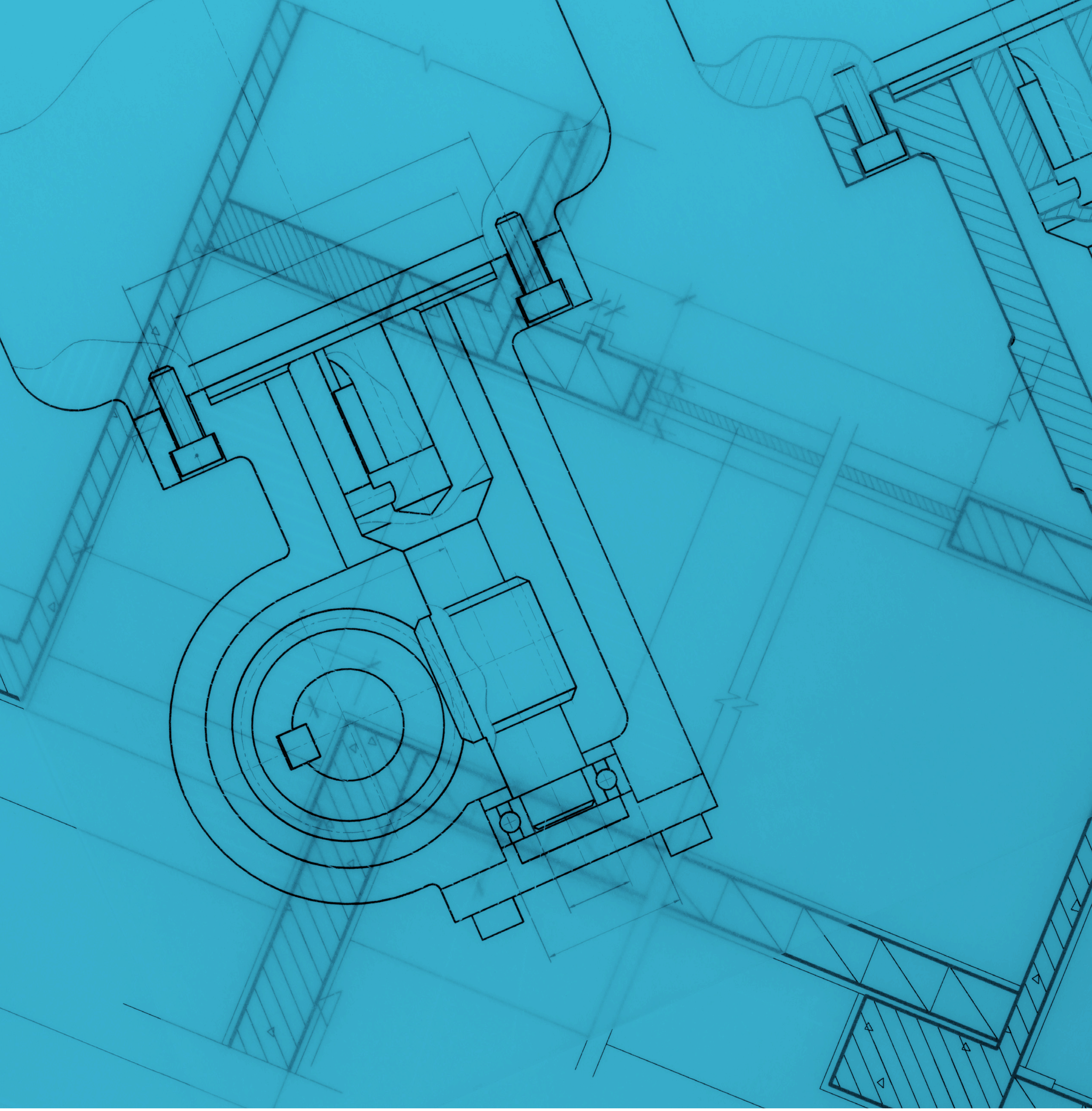
If the holes are not for structural design, they can be placed closer together and should all be placed two times the nominal wall thickness (W) apart from each other and from an outside or inside wall. Figure 17 shows how to design a non-structural hole.

For more design support for improving ESC resistance or other challenges, please contact us at **+1.844.4AVIENT**.

ENDNOTES

[1] Jansen, J. A. Advanced Materials & Processes 2004, 162 (6), 50–53

[2] PLET 350 – Plastic Part Design, Jason Williams, Spring 2014, Penn State Erie, The Behrend College.



1.844.4AVIENT
www.avient.com