

12 RULES OF THUMB FOR SUCCESSFUL PART DESIGN

Designing parts for plastic injection molding is a delicate balance of functionality, quality and cost. Follow these rules of thumb to increase your odds of success.

<p>1</p> <p>WALL DESIGN</p> <p>Thick walls utilize more material and extend the length of the molding/cooling cycle. They also increase part cost. Consider using ribs, gussets and bosses to improve wall strength rather than increasing their thickness.</p>	<p>2</p> <p>WALL UNIFORMITY</p> <p>Maintain consistent wall thicknesses to maintain a uniform flow of plastic, which will help to prevent flow lines and other defects.</p>	<p>3</p> <p>RIB THICKNESS</p> <p>To minimize wall thickness while maintaining strength, consider adding ribs – ideally 50-60% of the nominal wall thickness.</p>	<p>4</p> <p>RIB HEIGHT</p> <p>Rib height should be no more than 3x the nominal wall thickness. Since the rib will be thinner than the wall, it may be hard to fill it if it is taller.</p>	<p>5</p> <p>BOSS HEIGHT</p> <p>For best results, boss height should be limited to 3x its outer diameter.</p>	<p>6</p> <p>BOSS WALL THICKNESS</p> <p>To minimize sink, prevent voids and maintain optimal cycle times, the wall thickness of the boss should be approximately 60% of the nominal wall thickness.</p>
<p>7</p> <p>CORNERS</p> <p>Place radii on inside corners to enhance part strength, alleviate stress, improve material flow and reduce the potential for warping.</p>	<p>8</p> <p>DRAFT</p> <p>Include the proper amount of draft in your design to enable proper mold release. 1 degree of draft per 1 inch of cavity depth is recommended.</p>	<p>9</p> <p>INSERTS</p> <p>Metal or hard plastic inserts are an acceptable way to provide threads in a plastic molded part. These may be molded in place or inserted during a post-molding operation.</p>	<p>10</p> <p>UNDERCUTS</p> <p>Reduce/eliminate undercuts or features that require costly side actions to release the part from the mold.</p>	<p>11</p> <p>TOLERANCING</p> <p>Select tolerancing that is supported by the material selected and can be adhered to during the molding process, while still enabling the correct mechanical functionality of the part.</p>	<p>12</p> <p>MOLD DESIGN</p> <p>If possible, all the features in your part should be situated in the direction that the mold will pull apart. This will enable them to freely clear the cavity and the core as they separate.</p>