KAYSUN

INJECTION MOLDING & ENGINEERING SOLUTIONS

5 Key Considerations When COMPARING TOOLING QUOTES

PUTTING IT ALL TOGETHER

With virtually every injection molded product, the tool determines its success. It's why all components of the tool design and build need to come together – including a clear understanding of how the tool is quoted.

The process of gathering and comparing quotes is not always easy, especially since it's likely that no two quotes will be configured in the exact same way. The more information you share with the injection molder about your project — part or product design, type of material, functionality, production demands, production volume, timeframe, and total budget – leads to:

- Efficiencies in determining the best way to design your tool
- An accurate tool design and quote
- A more uniform approach to understanding the quotes provided

In the end, it leads to better decision-making and selection of the tooling quote that best meets objectives based on five key considerations:

1. Number of cavities
2. Tool material
3. Gating
4. Mechanical components
5. Quality/source of construction

1. What is the appropriate number of cavities for my program?

The cavity is the open space in the tool that is filled with the melted plastic, which then cools and hardens into the final product. Engineers can design one-cavity tools or multi-cavity tools, depending on project needs.

Building a tool with multiple cavities is more expensive and takes longer to build, but it can speed up production dramatically, and reduce part cost. A tool with fewer cavities results in higher part cost, but lower total tool cost and less tool maintenance.

Cavity quantity should be primarily based on the program's annual volume but there are certainly other factors to be considered — for example, part complexity, tooling complexity, and part tolerances.

The more cavities the tool has, the more difficult it is to control them all. Fewer cavities are often best when the parts are critical and tolerances are tight. In some cases, very complex tools with multiple slides and side actions will increase the physical size of tool to the point where only a limited amount of cavities can fit into the appropriate press.

The amount of plastic used in each process cycle, or shot size, is another factor in determining the number of cavities. There are minimum shot size limitations for every injection molding machine.

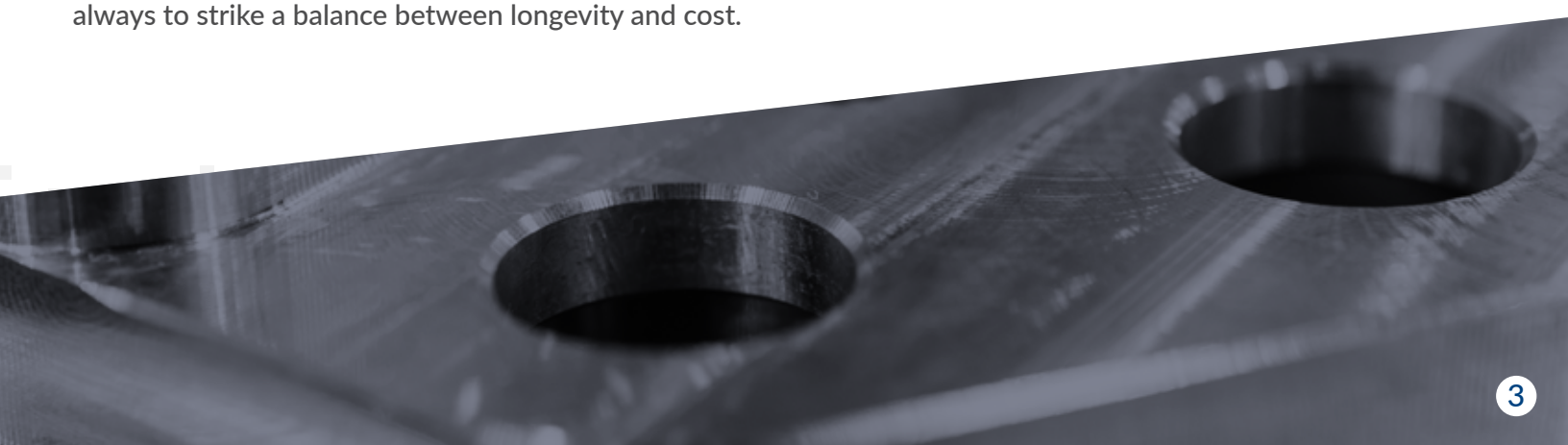
Different injection molders will quote different cavity numbers for the same volume of production, typically based on cycle time, press capacity, and press size constraints. This can be confusing. There may be cases where injection molders quote cavity quantity based on their specific equipment limitations, or intentionally over-quote the cavity quantity to drive down the part price.

The number of cavities will greatly impact part price and tool cost. Injection molders should be willing to explain their recommendations so the OEM can make the best decision for the project and budget.

2. The materials the tool is built from are a significant part of the total molding cost. Can I get by with a less expensive material?

Steel types vary, depending on project requirements and the expectations placed on the specific tool: How tight are part tolerances? How likely are design changes to occur after production starts? Is there the potential for increased volume down the road?

In some cases, lower-cost materials are more than sufficient. However, in other situations — especially with complex parts — going with low-cost/lower grade materials is not the best long-term option. The goal is always to strike a balance between longevity and cost.



WHAT ARE THE MATERIALS BEING MOLDED? HOW LONG IS THE TOOL EXPECTED TO LAST? HOW LONG IS THE RUN?

- Pre-hardened tooling surfaces are best for long life. Complex parts with difficult shut-off areas require pre-hardened tool steel like S7 or H13
- Hardened steels are ideal for glass-filled plastics and inserts for high-volume tools
- Stainless steel may be required for certain molding materials that are inherently corrosive
- Copper alloy tool materials can be used strategically to greatly improve cooling properties, which maximizes quality and speed
- Sintered metal inserts can be used to improve venting in critical areas

It's important to note that fully hardened steel inserts are best for longevity but they are more expensive than pre-hardened inserts (which are acceptable for medium volume and unfilled plastic projects).

ARE THE TOOLING SURFACES INSERTED?

Inserting the cavity and core tooling surfaces will cost more, but there are several advantages. The inserts can be removed from the tool and modified/repaired much more easily than if the cavity and core tooling surfaces are "in the solid."

Inserts can also be constructed from a higher-quality material compared to the tool base. If a major design change is required, the insert can be replaced instead of the entire tool.

3. How complex does the gating need to be?

Similar to the tool materials, the gate type and runner choices for a tool should be based on the specific program and its requirements including part tolerance, warp/flatness, and aesthetics. If any of these are a concern, be sure to clarify what type of gate/runner system the tooling quote is based on. The injection molder should be able to easily explain the logic behind the decision.

The first decision is gate location — typically the tool designer places the gate in the easiest and least-expensive location. Gates can sometimes cause cosmetic issues, so if the gate is going to be located at a visible part of the product, the tool designer can usually relocate it to prevent cosmetic flaws from being seen.

The runners transfer the molten plastic into the tool cavity. There are two types of runners — hot runners and cold runners. Hot runners are more complex systems and must maintain an even temperature; cold runners simply carry the molten plastic to the tool cavity. Hot runners are more expensive and require more maintenance but provide several advantages over cold runners, including less waste (greener), better process control, and shorter cycle times.

4. What is the best way to keep the tool as simple as possible to avoid problems and extra maintenance?

The easiest and cheapest way to build a tool is to eject the parts and let them fall out of the tool. Tools may be quoted in this manner if the OEM does not provide specific directions.

However, many parts can't be dropped into a bin or onto a conveyor since delicate or tight tolerance geometries might get damaged. Many parts also need to be cooled in a structured and consistent manner, which can't be done by dropping them.

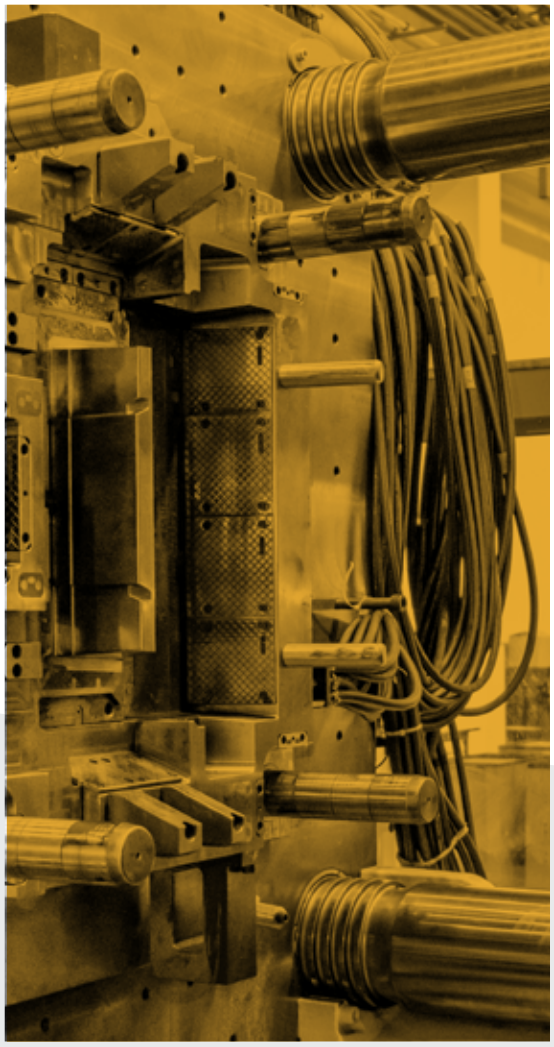
Extra features can be designed into a tool to ensure the parts do not drop. For example, parts can be held in place for robotic or operator removal. Even though these extra features cost more, they protect the final product and prevent damage which ultimately saves money. This is where experience pays off. At Kaysun, our in-house toolmakers work alongside our engineers — often right on the tool room floor — to immediately work through the best tooling approach. This eliminates the need for expensive tooling re-work, which can take anywhere from 3 to 5 weeks to complete.

Finally, injection molding is a harsh and jarring process. Tool parts wear out from time to time. Mechanical parts (locks, pins, and cylinders) should come from reputable manufacturers as this simplifies repair and maintenance. A forward-thinking injection molder will identify critical components that have long replacement lead times and purchase and store spare parts to minimize downtime. This practice may drive up the tool cost and could be a differentiator between quotes, but is vital for ensuring uninterrupted production. It also reinforces that the molder and toolmaker stand behind the tool and take care of any problems or repair costs.

5. Is it better to have the tool built locally or domestically, compared to overseas?

Tools that are made in the U.S. are often higher quality and easier to qualify than tools made overseas. Although tools manufactured outside of the U.S. are generally less expensive, exorbitant shipping costs, riskier steel and component quality control measures, and recently imposed tariffs could easily cause initial savings to evaporate. Having the tool made near the injection molding facility also makes it a lot easier and faster to fine-tune the tool after initial sampling, make engineering revisions, and make major repairs.

There's also the injection molder's relationship with the toolmaker to consider. Is the injection molder simply shopping around for the cheapest tool? Reputable, experienced injection molders have long-established relationships with high-quality toolmakers that facilitate communication and streamline modifications and repairs.



HELP THE EXPERTS HELP YOU.

Developing a tool is complicated given the technical aspects involved and the variety of ways an injection molder can go about it.

When gathering tooling quotes, help the experts help you. Don't just skim the surface and go for the low quote. Instead, approach it with a solid strategy; ask a lot of questions; and base decisions on useful information and facts.

In the process, look toward the future and establish a relationship with an injection molding partner that is respected in the industry, has proven experience in building the right tools for any job, and makes it a priority to earn — and keep — the trust of their OEM customers.

Contact Kaysun to learn more.

KAYSUN
INJECTION MOLDING & ENGINEERING SOLUTIONS

www.kaysun.com
920-686-5800

Follow us on: