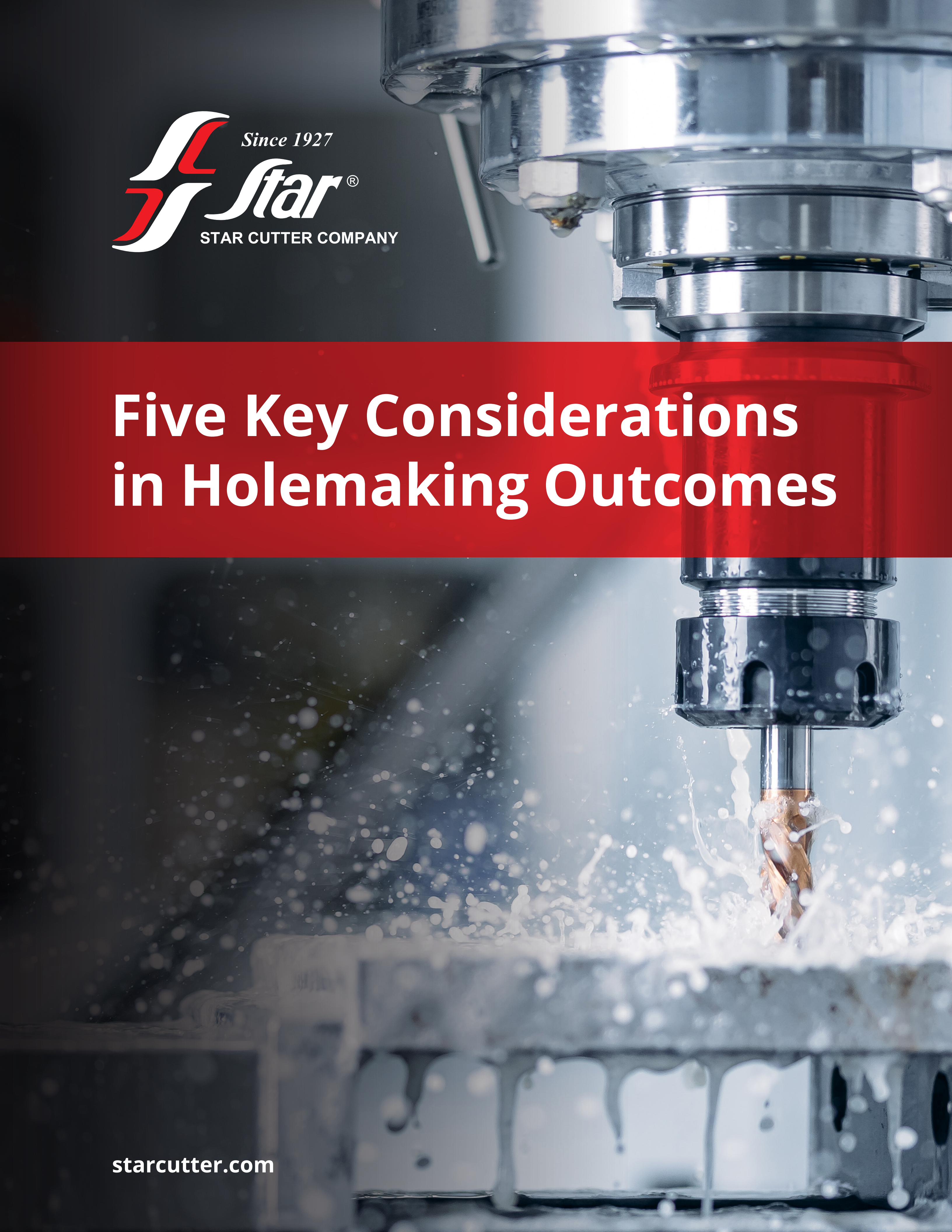
Since 1927

Star®

STAR CUTTER COMPANY

Five Key Considerations in Holemaking Outcomes



starcutter.com

Five Key Factors in Holemaking Outcomes

Drilling, or making holes, is the most common of all machining applications. There are as many ways to produce those holes as there are a variety of holes that need to be produced. But, optimizing any particular holemaking application needs to first start with the desired outcome. What is the most important to your company? Is it quality, ensuring optimal Geometric Dimension and Tolerance (GD&T) for each feature? Throughput? Cost per unit or tool life? Or maybe you just want a simple, standardized process that you can easily repeat over and over.

Once you know what you're trying to achieve, then it's time to examine the factors that influence the holemaking process.

Five basic factors that impact the drilling operation are Material, Machining Parameters, Holders, Tool Design, and Coolant. And all of these are dependent on one another; each factor has an influence on the others.

1. **Material:** Once you have established your objectives, the part characteristics are critical to your tooling decision. This begins with what is the material to be machined – type, treatment, hardness, surface condition? Soft, non-ferrous materials will, obviously, have quite different material properties and holemaking requirements than hardened materials and superalloys. Proper tool designs are developed based upon each of the ISO material grades machined.

2. **Machining Parameters:** Running the proper machining parameters is particularly important in order to control the heat, the formation of the chip produced by the tool, and tool life performance. Speeds and feeds are based upon running the tool at the proper surface footage per minute (SFM) and Inches Per Revolution (IPR). The SFM impacts the tool life the most as it creates the most heat in the application.

   Below you will find some basics for guiding the right speeds and feeds depending upon the workpiece material.

Rev (IRP) using a 0.2500" diameter drill

(based on deep hole drilling)

	Steels			
	Carbon	Alloy	17-4PH, 15-5PH	Stainless
	1010, 118, 1145	4140, 5120, 8620	300 Series	400 Series
ø = 0.2500"	0.0007"	0.0007"	0.0005"	0.0004"
	Irons		Aluminums	
	Gray Cast	Ductile	Cast	Heat Treated
	0.0015"	0.015"	0.0040"	0.0020"

Then, you can calculate the feed rate using the following formula:
Feed Rate = RPM x the IPR

Surface Feet/Minute (SPM)

(based on deep hole drilling)

Steels			
Carbon	Alloy	17-4PH, 15-5PH	Stainless
1010, 118, 1145	4140, 5120, 8620	300 Series	400 Series
350 SFM	325 SFM	225 SFM	200 SFM
Irons		**Aluminums**	
Gray Cast	Ductile	Cast	Heat Treated
200 SFM	175 SMF	600 SFM	600 SFM

From here you can calculate the RPM using the following formula:
Spindle RPM = SFM * 3.82 / by the diameter of the cutter

3. **Holders:** The next area influencing outcome is the tool holder. There are three basic types of holders used in machining practices – mechanical, hydraulic or heat shrunk. All of these holders are available with different interface options, depending on the interface of the machine spindle.

   In some cases, collets are used to standardize holder options and size down to the tool shank size (the shank is the part of the tool that goes inside the holder) instead of requiring additional holder options. In most cases, the shank tolerance of the cutting tool should have a h6 tolerance, which is considered standard for a shank.

   Mechanical holders normally use a set screw to lock the tool in place. This is best accomplished using a torque wrench that can lock to the specifications and eliminate the risk of the tool coming loose in the holder. With a mechanical holder, it's important to verify that runout does not become an issue.

   Hydraulic holders have a specific shank length that needs to be held for the distance inside the holder. If the tool gets pulled out too far out of the holder, the internal sleeve can become distorted over time as the hydraulic bladder gets clamped down. Not having the full surface of the shank in this area can risk distortion of the bladder and put runout into the assembly. This type of holder is good for applications that are prone to harmonics. This type of holder has mass that reduces the amount of residual harmonics.

   Heat-shrunk holders are commonly used and create a very rigid assembly. While providing particularly good runout characteristics, the rigidity can potentially create chatter in the part on longer tools.

There are three basic types of holders used in machining practices – mechanical, hydraulic or heat shrunk.

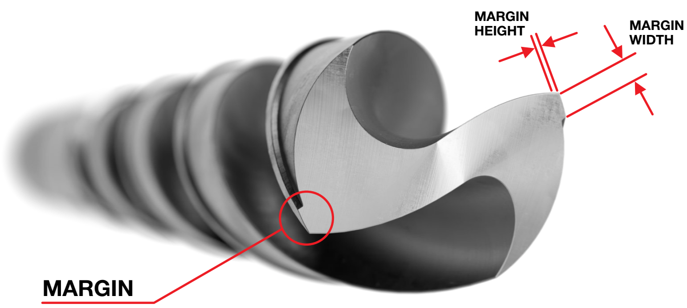
4. **Tool Design:** Here's where it really gets interesting. Matching the properties of the tool – such as the carbide substrate, coating, and coolant hole placement – to the needs of the workpiece is what modern metallurgy is all about. When done correctly, forces on the cutting edge are minimized, chip breaking is controlled and evacuation is complete, resulting in smooth finishes and controlled tool life.

   Proper tool geometry is designed around the material being machined and the hole specifications. Using the proper tool maximizes the part quality, as well as tool life. Each individual application determines the optimal design for the point geometry, clearances, coatings and margin widths. The margin is the part of the drill geometry located along the outer diameter trailing the cutting edge.

Margins are needed to stabilize the drill in the hole that provides support behind the cutting edge and the workpiece. Margins also help to produce a truer hole by essentially burnishing the workpiece if properly designed.

Single margins are common whereas multiple margins are a matter of engineering for part and material-specific requirements, accommodating part surface finish, hole location and bore size criteria.

Additionally, advanced tool coating options also need to be considered in order to maximize the tool life along with the number of times a tool can be reconditioned. Overrunning a tool will create excessive margin wear and reduce the amount of time a tool can be reconditioned.



Tool coating is just one of many considerations in optimizing your holemaking operations

The area between the cutting edges is referred to as flutes. Although straight flute tools can be used in deep-hole applications, most deep drilling requires helical tools depending on the diameter to depth of cut ratio. Straight-fluted tools can maintain proper hole diameter, less drift, and achieve good part surface finish. However, workpieces that feature intersecting bores can pose challenges to chip evacuation, whereas using a helical tool could be more beneficial.

Straight Flute vs. Helical Tool

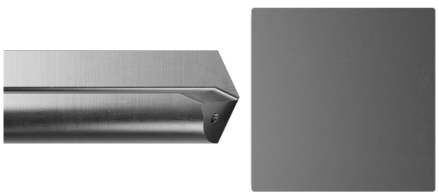
Then, you have to look at the actual operation:

Will the tool machine the part from a solid condition? Will there be through holes or blind holes (possibly cored), or any interrupted cuts?

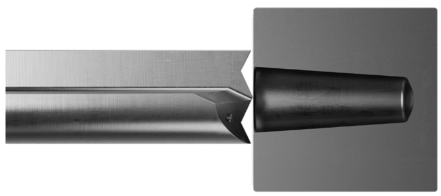
Together, these are the first identifiers for the choice of the ideal drilling tool.

Using die cast aluminum example, the below chart provides some basic point considerations for an application. The special points for machining cast cored holes assist in controlling tool deflection, which can result in tool drift or even catastrophic failure.

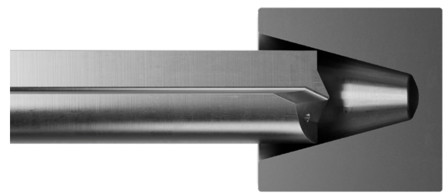
Knowing what your desired outcome is critical to the tool design. For example, you may be using different tools to drill your holes. However, if you want to optimize the cycle time and concentricity between multiple diameters for a machined feature, this can all be designed into a single tool. All of this plays a role in controlling the overall CPU.



Standard Point Geometry Cutting into Solid



Concave Point Geometry Cutting into a Cored Hole Where the Drill is Larger then the Cored Hole



180º Point Geometry Cutting into a Cored Hole Where the Drill is Smaller then the Cored Hole Entry

5. **Coolants:** Depending upon the machine tool, there are basic types of coolant delivery systems – a flood coolant system or a Through-the-Tool system. The flood coolant system has no Through-the-Spindle coolant supply, and is not ideal for the tool's cutting edge. The Through-the-Spindle coolant system flows the coolant through the internal channels of the cutting tool.

Flood Coolant System

Through-the-Spindle System

Today, more applications use the through-the-tool coolant approach. When the coolant is through- the-tool designed with proper tool geometries, the coolant keeps the part and cutting edge cool while in the cut, as well as aiding with flushing the chips up the flute channel to evacuate them out of the part as the tool is machining.

For this approach to provide optimal results, it is particularly important to have the pressure set at the proper settings — especially on deep hole drilling where the diameter-to-depth ratio is great. If the coolant pressure is too low, there is a risk of chips being packed together during the machining process, which creates a high probability of the tool breaking. Using the correct coolant pressure and proper clearances on the cutting tool aids in breaking the chip into smaller pieces, as well as helping prevent material build-up on the cutting edges.

There are three typical types of coolant used in machining:

- **Straight oil or neat oils**. Straight oils provide the best lubrication and therefore, are the best fluids to maximize tool life.
- **Semi-synthetic coolant** – a metalworking fluid that typically has 5-50% mineral oils, water and synthetic chemicals. This is more environmentally-friendly, but normally can have an effect in tool life in comparison to the straight oil.
- **Water soluble** – common type of coolant used that contains no mineral oils. This type of coolant requires a concentration level to be maintained that is recommended to fall between 8 and 12%.

It is always best to contact your coolant supplier for more specifics on your particular application.

Summary

As you can see, there are numerous factors to consider in selecting a tool for holemaking operations. **Below you will find a few case studies that can help illustrate the performance difference when a tool is optimized for a particular application.**

To discover the best way to address your holemaking application, work with a tooling solution expert who can engineer the optimal tool for your needs.

Reach out to a Star Cutter tooling solution expert, call (248) 474-8200 or email sales@starcutter.com

Case Studies

Material Machined: Aluminum

We designed a carbide tool using a special grade of carbide, overall coolant-hole configurations and special designed with tight tolerance geometries.

Actual Cost Savings Generated:

$10,500

Other supplier Tool Life: 10,000
Our Carbide Tool Life: 20,000

Actual Cost Savings Generated:

$23,100

Other supplier Tool Life: 1,500
Our Carbide Tool Life: 5,500

Application: Cast Iron

Old Technology

ø14mm Step Straight Flute Drill
Tool Life Per Grind: 1,800 holes (300 Heads)
RPM: 1,400
Feed: 225 mm/min

New Technology

ø14mm 30º Helix Step Drill
Tool Life Per Grind: 3,300 holes (550 Heads)
RPM: 1,800
Feed: 383 mm/min

Results

35% Increase in Throughput
40% Increase Tool Life

Many examples proven in Irons

Since 1927

Star®

STAR CUTTER COMPANY

Star Cutter Company

23461 Industrial Park Drive
Farmington Hills, MI 48335-2855 USA
Tel: 248-474-8200
sales@starcutter.com

Sold and distributed by **Star SU LLC**