HAYNES® HR-120® alloy

Machining and Grinding

HAYNES® HR-120® alloy can be readily machined using conventional techniques. Generally, the same practices are employed as those used with the 300 series austenitic stainless steels. Some minor adjustments in the machining parameters may be required to obtain optimum results. High speed steel tools are found to be satisfactory, although machining speeds can be substantially increased by using carbide cutting tools. As a general statement, grinding operations with HAYNES® HR-120® alloy are considered equivalent to those of the 300 series stainless steels. As with other alloys, grinding is recommended where a close tolerance is required. Basic “Do’s” and “Don’ts” that should be considered during machining are:

Do:

1. Use machine tools that are rigid and overpowered, where possible.

2. Insure work piece and tools are held rigid. In addition, minimize tool overhang.

3. Make sure tools are always sharp. Change to sharpened tools at regular intervals rather than out of necessity. Remember, cutting edges, particularly throw-away inserts, are expendable. Don’t try to prove how long they can last. Don’t trade dollars in machine times for pennies in tool cost.

4. Use positive rake angle tools for most machining operations. Negative rake angle tools can be considered for intermittent cuts and heavy stock removal.

5. Use heavy, constant, feeds to maintain positive cutting action. If feed slows and the tool dwells in the cut, work hardening occurs, tool life deteriorates and close tolerance is impossible.

6. Avoid conditions such as chatter and glazing. This can cause work hardening of the surface, making subsequent machining difficult.

7. Flood the work with premium-quality sulfochlorinated water soluble oil or water-base chemical emulsion oils with extreme pressure additives. Dilute per the recommendations of the manufacturer.

8. Use heavy-duty sulfochlorinated oil for drilling and tapping. Special proprietary tapping oils can also be used.

9. Use air jet directed on the tool when dry cutting. This can provide substantial increase in tool life.

Don’t:

1. Do not make intermittent cuts, if possible. This tends to work harden the surface, making subsequent cuts more difficult.

Detailed Machining Information
Turning, Boring and Facing
The table below represents a typical range of values for normal turning operations. The depth of cut (particularly for roughing operations) is quite large with relatively low feed rates. These parameters are equipment and component dependent. The larger depths of cuts and higher speeds are recommended only when using heavy, overpowered equipment on large rigid components.

Conditions	Roughing	Finishing
Depth of Cut	0.125-0.250 in.	0.020-0.040 in.
Feed Rate	0.008-0.010 ipr	0.006-0.008 ipr

Speed-HSS	30-50 sfpm	40-60 sfpm
Speed-Carbide	100-170 sfpm	140-180 sfpm

Drilling

Standard high-speed steel bits are normally used. For drill bits larger than 3/8", thinning the web may reduce thrust and aid chip control. The following are suggested speed and feed rates for various diameter drills.

Diameter	Speed	Feed Rate
1/8 in	250 RPM (max)	0.002 inch/rev.
1/4 in	250 RPM (max)	0.003 inch/rev.
1/2 in	250 RPM	0.005 inch/rev.
1 in	150 RPM	0.011 inch/rev.
1-1/2 in	100 RPM	0.013 inch/rev.
2 in	75 RPM	0.016 inch/rev.

For other diameters (above 1/2 inch diameter) the spindle speed may be calculated from the following: RPM = 150/Diameter (inches). This results in a cutting speed of about 40 sfpm. For drill diameters smaller than 1/2 inch, speed rates substantially below 40 sfpm are required.

Reaming

Standard fluted reamers of high-speed steel are generally used. Speeds should be about 20-25 sfpm for diameters above 1/2 inch. For small diameter reamers (less than 1/2 inch diameter) cutting speeds should be reduced substantially. Feed rates will range from 0.002 to 0.006 inch/revolution depending upon diameter. If carbide tipped reamers are used, the speed can be increased to 70 sfpm for reamers above 1/2 inch diameter. If chatter occurs, reduce speed.

Tapping

HAYNES HR-120 alloy is tapped using the same tooling and conditions as used with type 316 stainless steel. High speed steel taps work well. Cutting speed can be up to 20 sfpm for taps above 1/2 inch diameter. For small diameter taps (less than 1/2 inch diameter) cutting speeds should be reduced substantially.

Thread engagement can be reduced because of the high strength of this alloy. Generally, thread engagement of 60 to 75 percent is considered acceptable. Thread engagement is considered a design parameter and therefore should be left to the design engineer. As a general statement, 75 percent thread engagement is common for low strength materials, but only leads to increased tool wear and possible breakage in high strength alloys. It does not increase the holding strength in these alloys.

Milling

High speed steel cutters, with good impact strength, are recommended due to the interrupted nature of the cutting action. A cutting speed of 30 to 40 sfpm with feed rates of 0.002 to 0.005 inch/tooth is generally recommended. If carbide cutters are employed, speeds of 60 to 80 sfpm are possible.