



Innovative Tool Sales

755 E. Debra Lane, Anaheim, CA 92805
(714) 780-0730 • (714) 780-0735 Fax

Technical Support Page

MILLED MATERIALS MACHINABILITY

Aluminum & Magnesium Alloys

Aerospace Aluminum: When machined properly, will create little problems. Always make sure to maintain a healthy chip load, at the cost of the cutting speed (RPM). Tool life is dramatically increased when all of the cutting heat is taken away with the chip itself.

Standard Aluminum: Can be machined successfully, both with conventional HSS/Cobalt or Carbide end mills. However cast parts with hard scale or high silicon Aluminum, require a higher edge resistance than what regular HSS/Cobalt can offer. We, therefore, recommend either Titanium Carbo Nitride Coatings on HSS/Cobalt or Solid Carbide end mills in these materials.

Copper Alloys

In the pure Copper or Brass groups, we find materials that machine very much like Aluminum, with a need for sharp geometries and high removal rates. In cases where the material is very soft and gummy, a reduction in the number of flutes will always help

On the other hand Aluminum Bronze or Beryllium Copper have a machinability rating which is a third that of pure Copper or Brass. This removal rate is similar to a 303 Stainless steel and, therefore special care should be given to the machining of these alloys.

Since a lot of products which are made of this group of material require a nice finish and appearance, a way to improve floor surface finish in end milling, is to grind a very small corner radius or a chamfer of the corner edges. This will improve the finish drastically, particularly when runout exists due to poor tool holders or a long tool and/or extension.

Cast & Ductile Irons

Cast iron materials is a rather large family of ferrous materials in which the major components are Iron, Carbon and Silicon. The four basic types are White Iron, Ductile Iron, Gray Iron and Malleable Iron.

White Cast Iron may include additional elements such as Molybdenum, Nickel, Chromium etc.. and makes it, at same hardness conditions, the hardest to machine and should be approached just as an Alloyed steel.

Gray Cast Iron which the most commonly used for machine parts etc. is relatively easy to machine.

The graphite that is found in the structure introduces interruptions in the matrix and acts like a breaker in the chip formation, making it an easier to cut and creating small dust like chips. For this reason, it is possible to machine Cast Iron with multi-flute-end-mills, even in a channel (slotting) cut, since the chip evacuation is totally different.

Cast irons, because of the sand, are however very abrasive and Carbide tools (coated is even better) are the most efficient. Unlike for steels where recutting chips would destroy the edges of solid carbide roughers, the shape and texture of Cast iron chips make the usage of Carbide roughers very adequate and effective. They are highly recommended for high removal rates and wear resistance.

One important remark is to pay attention to the microstructure of Cast Irons. For mechanical and convenience purposes, the structure of the part to be machined may be made of Coarse Pearlite or of Ferrite (annealed condition): The Ferritic structure will machine 3 times easier than the Perlitic structure.



MILLED MATERIALS MACHINABILITY

Carbon Steels, Low & Medium Alloys

From all of the steel groups, Low Carbon steels are probably the easier to machine in comparative hardness. A few words of caution however should be kept in mind: All the recommendations in this manual are for Climb Milling, except for parts which have been flame cut. During the flame cutting process, the skin becomes hardened and offer an almost "heat treated" scale that may reach 60 HRC. It is only a superficial layer, but it is hard enough to damage and often destroy the edges of Cobalt and Carbide tools as will. In the Climb milling mode, the edges are entering the hard scale at each revolution and hit this tough wall causing chipping. By conventional milling, we are only hurting the tool, when we enter the material (Dragging), and after that the edge will go under the scale and crack it on the way out. Make sure, however to take radial cuts that will be beyond the thickness of the scale.

One way to avoid that first entering cut in the hard scale is to grind a small entry area. This can be done with a hand grinder but it will guarantee much longer toll life and edge integrity.

For a given hardness and condition, because of their free machining inclusions, the 12xx groups machine the best, next to the 11xx and then the 10xx.

Below are some rankings of machinability (from easy to difficult) for some popular alloys.

12L14 - M.R. 180
1212 - M.R. 100
1118 - M.R. 85
1018 - M.R. 70
1040 - M.R. 60
1050 - M.R. 55
Mold & Die Steels

This group, which in reality should also include tool steels, High Speed Steels etc.. is probably the most diversified one as far as machining is concerned. The parts can be machined in a variety of conditions and harnesses and the tools are required to have a good wear resistance, together with toughness and of course a good "Hot Hardness".

Here too, we often have flame cut parts with a hard scale where conventional milling will be better than Climb milling. TiCN coatings with a hardness of around 92 Rockwell has been extremely effective and will prolong tool life. As for High Temperature Alloys, the high Carbon, high Chromium groups such as the D2 require a very rigid setup and machine.

In order to offer the operator a simple guide for ranking these materials, hereunder is a chart ranking them for machinability starting from easy to machine to most difficult.

W Easiest
L
S
P
O
H
M
T
D
A Toughest

To place these factors in perspective a W group will be 3 to 4 times easier to machine than a D2 or an A7.



MILLED MATERIALS MACHINABILITY

High Alloy & High Strength Steels

When calculating horse power requirements (see formulas in the previous sections of this manual), quite often the operator may find that not enough Horse Power is available on the spindle for a particular operation.

It is wise to remember that from all the factors that affect HP consumption, such as Depth of cut, Width of cut, RPM and Feed rates etc., increasing chip load per tooth will affect HP consumption the least. For instance, doubling the Feed per tooth on a specific cut, will only increase the Energy required by 60%. On the other hand, doubling the depth of cut in the same operation will double the HP that is needed.

Roughing (Corn Cob) end mills are a simple way to reduce your Horse Power consumption. A coarse profile rougher will require nearly 33% less HP than a regular end mill in a given application. Roughing end mills are also chatter free and will permit much deeper cuts than regular finishing type tools.

However, they will not supply a good surface finish and often enough a finishing pass is need. Coarse profile end mills will provide a 400 RMS finish while Fine profile, which are more designed for hard materials and to too deep a cut, will provide 200 RMS finish.

The flat profile is a semi finishing tool that will guarantee a 125 RMS surface finish, however that type of tool will require more HP than a regular round knuckle rougher and the saving in energy versus a finishing tool will only be 10% (versus 33% for the Coarse Profile).

Roughers are also more efficient in re-sharpening versus so called "chip breakers" and mills. In the chip breaker case, you may loose the chip breaker depth after a couple of re-sharpening while a roughing end mill, thanks to its eccentric form relief, will keep its shape and profile till the end.

Stainless Steels & P.H. Series

Of all the Alloys in this group, the most unpredictable as far as their machining characteristics are the Precipitation (PH) Alloys. Depending on the hardness of the part, whether it is at the low end or high end of its permissible range, some tools may give at first, excellent results and suddenly tail. Therefore, it is recommended, whenever such a situation occurs, to first check the hardness of the lot prior to make hasty judgment on tool performances or operator's mistakes.

As for as the rest of the group, Stainless Alloys machine fairly well but they differ in machining ratings. The table below ranks the most popular grades of Stainless and their relative machinability rating:

| | |
|-------------|---------|
| 416 | Easiest |
| 303 | |
| 430 | |
| 410 | |
| 420 | |
| 304 | |
| 316 | |
| 17-4 | |
| 13-8 | Hardest |



Innovative Tool Sales

MILLED MATERIALS MACHINABILITY

Titanium Alloys

Titanium Alloys do not deserve the "Machining Fear" that many people associate with them. In reality they are, in their majority, very easy to machine if a few key elements are kept in mind.

First of all, the cutting tools need to be sharp and have very positive cutting geometries, similar to Aluminum. Tool life will be reduced drastically when tools with dull edges are used. The key to sharp tools is to avoid friction and positive edges will minimize the contact area between the tool and the part.

However, because of the materials strength, and the reduced amount of chips created compared to Aluminum, not two (2), but multiple flute end mills should be used.

End mills with high helixes and high rake angles made of Cobalt/HSS work the best. Solid Carbide end mills may be used for finishing passes but are not recommended for rough cuts because of their geometries.

Since avoiding friction is critical, anything that would improve the chip flow is critical. A coolant with a high viscosity may double the tool life in many cases. That is also where coatings make a lot of sense since most coatings offer better coefficient of friction in the interface zone.

TiN coating, contradicting all previous theories of Chemical reaction, works extremely well on Titanium, particularly in the Alpha and Alpha-Beta group. TiCN is recommended because of its superior hardness for the Beta group. The newer TiNAl coatings are also showing superior edge life at increased Sfm applications. More testing is being done, it looks very promising for the future.

There is an exception in the Alpha-Beta group and that is for the 6-2-2-2 which although belongs to this group, in reality it machines like a Beta group Alloy. We highly recommend TiCN coatings with Fine Pitch Roughers for this material on heavy removal cuts.

Another important factor beside tool geometry is chip load: A rather thick chip needs to be maintained guaranteeing a shearing action and eliminating rubbing. The most common problem in machining Titanium Alloys is to see high RPM and slow feed rates or chip loads. This absolutely destroys tool life and causes nasty Work-Hardening.

Just like in Aluminum, in the case of long tools, where table feeds are limited, the operator should reduce the RPM and not the Feed Rate. As a last remark, pure Titanium is extremely easy to machine and presents no problem whatsoever.

High Temperature Alloys

The High Temperature Alloys, also called "Super Alloys" include some of the most difficult to machine materials. Aerospace parts, turbines etc., require materials that perform well at very high temperatures and also offer extreme toughness wrapped into a nice clean package. All of these materials have very low thermal conductivity, resulting in trapping the heat at the cutting edge during the machining process. Tool life for these tools is relatively short and cutting speeds are very low.

Special care should be given to pull tools (from wear) in time for re-sharpening. An end mill with a slightly dull edge will soon turn to a broken tool and may damage a whole part. Watch for warnings that the tool needs to be changed such as appearance of burrs, chips turning blue, different noises, dimensional changes etc... Special care should be given to the fact that all though some of these Alloys carry the same "first name" such as Inconel, the number that follows the name is of utmost importance and differentiates the alloys as far as their machinability. Following are some examples:

Inconel 718 has 53% of Nickel and 1% Titanium while Inconel 750 has respectively 73% Nickel and 2.5% Titanium.

Haynes 556 has 20% Cobalt and 2.5% Tungsten while Haynes 25 has 50% Cobalt and 15% Tungsten.

Udimet 630 has 0% Cobalt and 0.7% Al, while Udimet 700 has 18% Cobalt and over 4% Aluminum.

High Temperature Alloys require strong tools that will not deflect. The core diameter of the end mill is a key factor and a 6 or 8 Flute end mill will have more Core Diameter than a 4 or 2-flute. Additionally, the Length of Cut Ratio is critical, and a short stubby tool will reduce tool deflection and breakage. Do not take short cuts and use longer than necessary flute lengths. Make tool changes if needed, even for only a couple of passes, since using as short a tool and as large of a diameter as possible will improve your productivity and will avoid premature tool breakage.