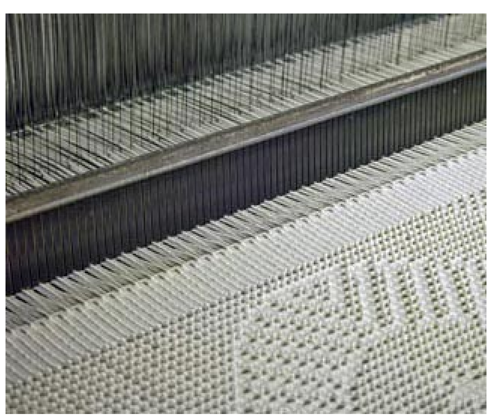
An intro to CNC Machining

CNC stands for Computer Numeric Control. CNC machining involves using a machine controlled by a computer to machine material. Generally the machine is either a milling machine or a lathe. The earliest record of a milling machine dates back to 1814 (Eli Whitney, England).

The earliest precedent to CNC machines may be the Jacquard loom, a mechanical process invented in 1801 by Joseph Marie Jacquard to simplify the process of manufacturing complex textile patterns. The Jacquard loom read punch cards. The holes in cards corresponded to the rows of design in the textile. This was pretty revolutionary, as an entire design could be stored on a set of cards, and changing production simply meant feeding a different set of cards through the system.



The first use of computer controlled machines arrives in 1956, via the Air Force. These machines used motors controlled by numbers from a computer to create movement in a machine. The convenience of being able to quickly change the program, unlike having to re-punch hundreds of cards, sets a new standard in manufacturing. By the 1960's as computer prices were dropping drastically, CNC machines were already being used across many different industries.

The CNC process

- The process starts with a 3D model, created in Rhino or virtually any other 3D package. This model is exported using .STL, .OBJ, .IGES or a variety of other file types that can represent 3D information.
- The exported file is imported into a computer aided machining program (**CAM** for short). CAM software recognizes the surfaces of the model and creates 3-dimensional paths which define the movements required by the CNC machine to cut material away from a solid block and recreate the model in physical space. CAM software exports these tool paths in code form (usually known as G-Code).
- The G-code file is fed into the CNC machine's controller. The controller reads the information in the code (which gives directions regarding movement of the machine's axis, speed of movements and the RPM of the machine's spindle among other variables).
- The machine's controller sends signals to the motors, resulting in the controlled movement and cutting of material.

The process looks like this…

CAD software	→	CAM software	→	CNC machine
Export model in a compatible format		Import model Create tool paths Export G-code		Import G-code into controller Machine's motors are sent co-ordinates to move to during cutting operations

There are many types of CAM software. These are often standalone programs, meaning they operate independently of your CAD software. Many CAD programs now have CAM plug-in support, meaning that the CAM operations occur within the modeling environment itself. Integrating CAD and CAM in the same software has several advantages. For one, the environment is already familiar to the user, which means there is no need to re-learn navigation within the interface. A second benefit is that the CAD model is not exported; rather, the model is worked on in its native format. This avoids problems which can result when exporting and importing between software. Finally, using CAM within CAD allows for changes to be made to the model much more quickly than would occur when exporting between separate CAD and CAM programs. The CAM software we will be using is a plug-in for Rhino 3D called Rhino Cam.

General Machining considerations

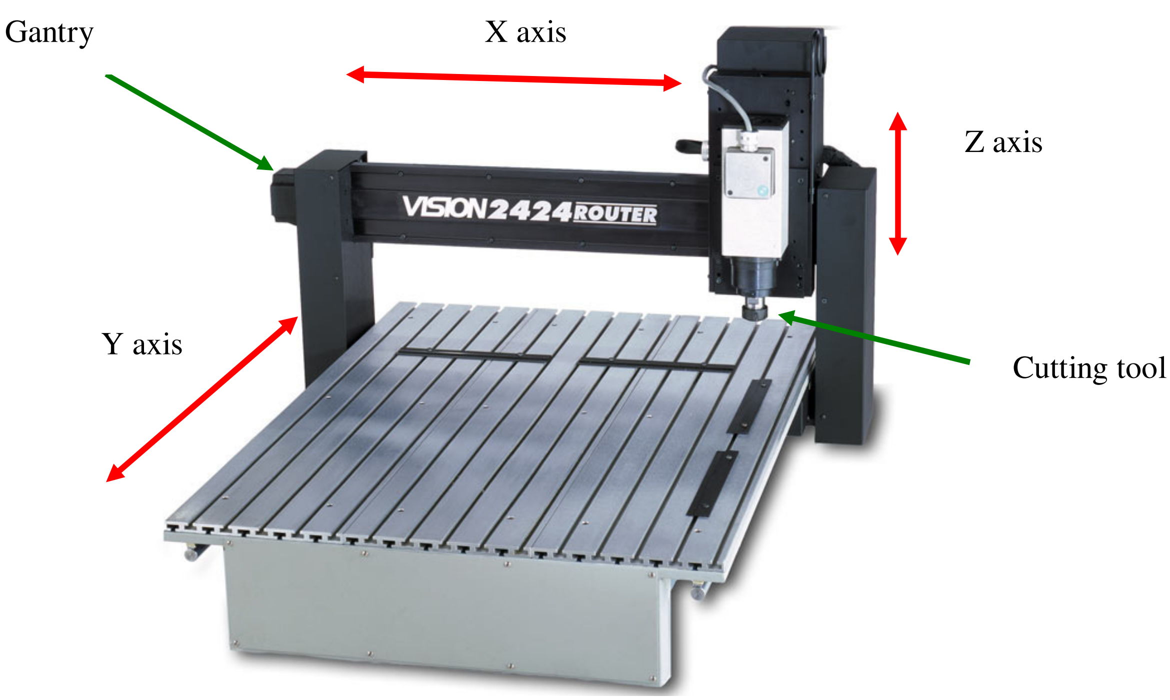
The material you are machining

There is a wide range of materials which can be machined. These include wood, wax, foam, plastic, metal, glass, graphite and ceramics. The type of material you want to machine will determine the type of machine you need to use.

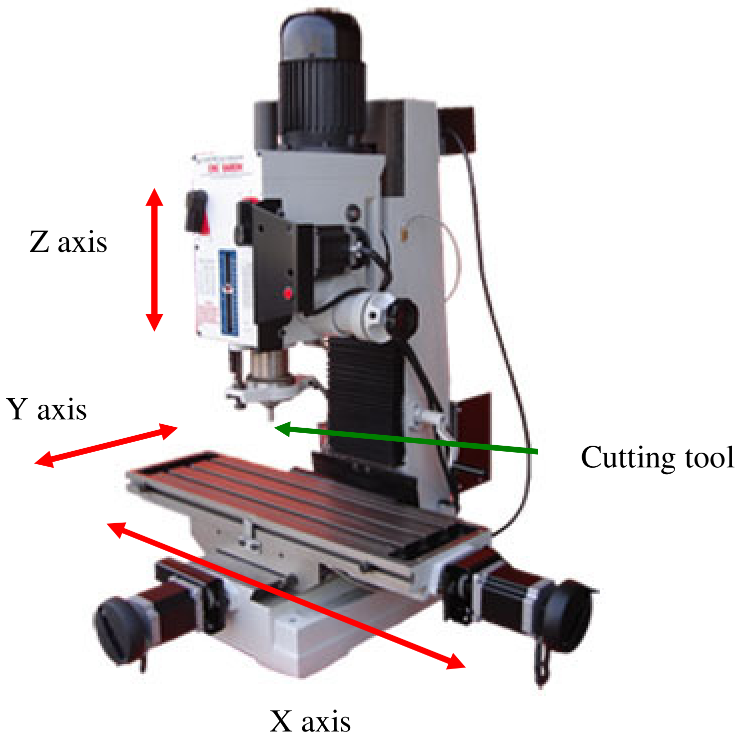
Machine types

In most cases, the principle is the same. The material you want to cut (called the "stock") is fixed to the machine's table, and a rotating cutter moves through the material, removing material as it moves. The most basic machines (the ones we are using) move in three axis… X, Y and Z.

- CNC Router-Normally made of an aluminum frame. The Z axis is attached to the X axis. This allows the cutter to move up and down, and side to side. The entire gantry moves in the Y axis, allowing for front to back movement. These types of machines are typically used in woodworking and sign making industries where they deal with large sheet materials. It's not uncommon to find 5x10 foot working areas (X and Y axis). This design is not particularly rigid, but is fine for wood, plastic, foam and other soft materials.



- CNC Milling Machine- Normally made of cast iron, or other dense materials. The table (ant the work attached to it) moves in the X and Y axis. The tool moves up independently of these, in the Z axis. Milling machines are much sturdier than Routers. They can cut much harder materials like ferrous metals, and stone. They normally move more slowly than routers.



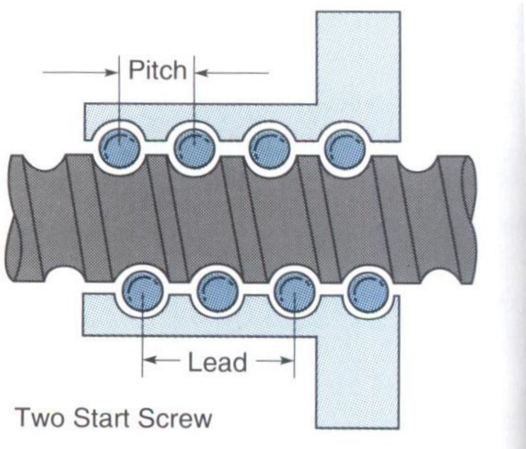
- Large scale machines- Often combine properties of both routers and milling machines. These are used for various applications, from automobile models to boat hulls. The principle is still the same as above… X,Y and Z.

Large scale machining… 131 x 26 x 15 FEET (!)

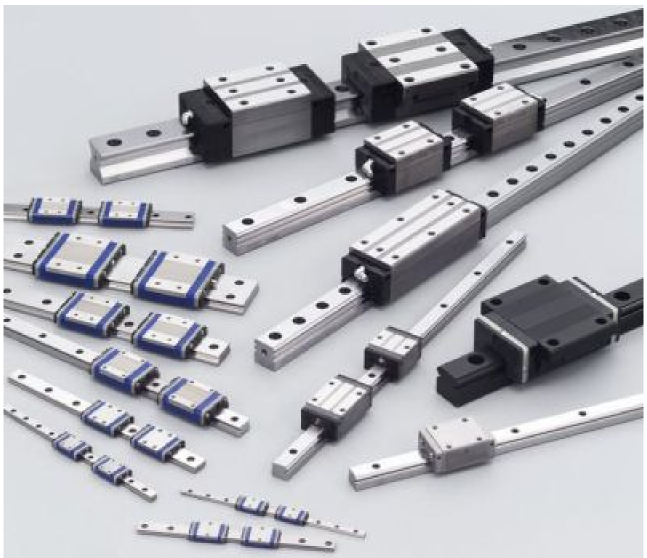
The base on this machine has tracks like a car kiln

Power transmission

Ball screws transfer motion from the rotation of a motor to a linear motion...

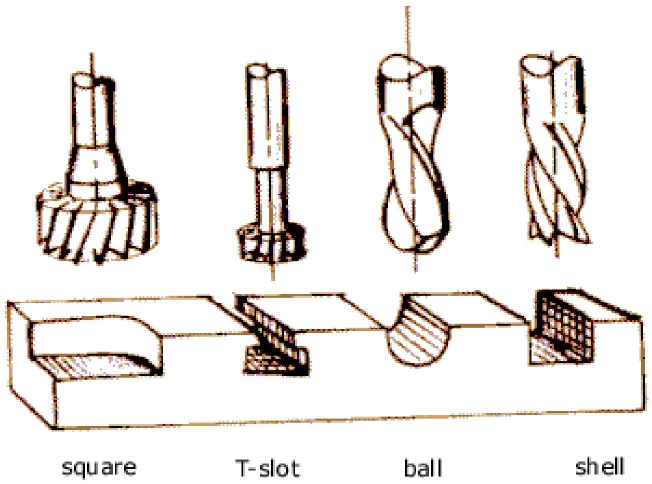


Linear rails ensure perfectly straight movement in each axis...



Cutting Tools

As the tool is what removes material from the stock, the size and shape of the tool is crucial to getting the results we want. In general, tools come in either a flat end or a ball end.



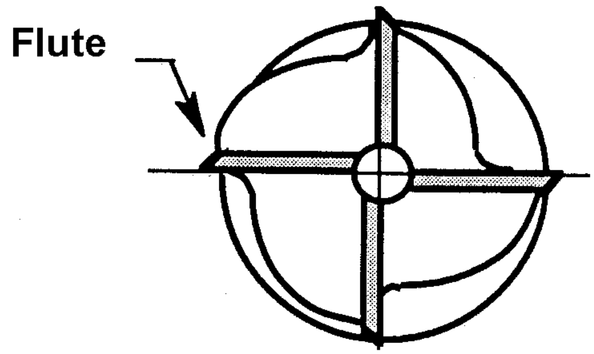
Ball end

Flat end

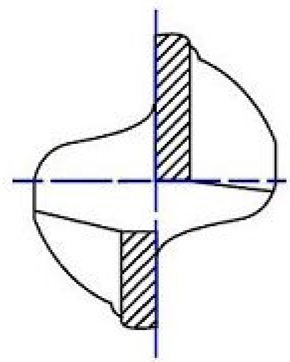
Flat end tools are perfect for defining flat surfaces while Ball end tools are better suited to defining curved surfaces.

The flutes on an end mill are the surfaces that remove material during the cutting operation. Having fewer flutes ensures easier removal of chips but as there are fewer cutting surfaces for each revolution of the tool, you must compensate by slowing down the speed with which the tools moves through the stock. We typically use either 2 or 4 flutes…

4 Flutes



2 Flutes



The most obvious difference between an end mill and a regular drill bit is that a drill bit can't cut on its sides like and end mill can.

The diameter of the tool dictates how long a tool you can use. Smaller diameters tend to flex more easily than larger diameters. This means that as you get into smaller and smaller diameter tools, the length of the tool itself will necessarily get shorter as well.

Smaller diameter tools tend to be shorter in length. This has to be considered when designing work for machining.

Feeds and Speeds

During cutting, the tool rotates, creating chips of material that are removed from the stock. The rate of chip removal is very important, because cutting generates heat. If the tool is fed too slowly through the stock, we risk not removing heat quickly enough which overheats the tool bit and ruins it. On the other hand, if the tool is fed too quickly through the stock, the flutes don't have enough time to remove material and can ultimately snap the tool. There is a happy medium for each given situation.

Factors involved include…

1-How fast the cutter moves through the stock… also known as "Feed Rate" or just "Feed". This is often indicated in Inches per Minute (IPM).

2-RPM of the cutter… also known as "Speed"

Actual values will vary but generally there are large differences between materials. For example, with a 1/4 inch (2-flute) cutter…

	Soft Wood	Plastic	Aluminum	Cast iron	Hardened steel
IPM	252	144	18.7	11.8	1.0
RPM	18,000	16,000	6,300	2,360	800

Softer materials can be machined more quickly than harder ones.

For a given material, Speed and Feed are interrelated. Slowing down your spindle speed means that there are less chips being cut in a given amount of time, which means you need to compensate by also slowing down the Feed of the machine.

For example, using a ¼ inch (2-flute) bit on aluminum…

IPM	44	29	20	10	1.2
RPM	18,000	12,000	8,000	4,000	500

Why wouldn't you just always run your feeds and speeds as fast as the chart allows? Because not all machines can run that fast. The spindle on the milling machine I have can only turn at 4,000 RPM. One of the machines in the Maintenance shop easily does 45,000 RPM. Being able to compensate for these differences in speed is essential.

A more universal method for calculating how fast to cut is to use Chip Load. Chip load is the thickness of material that is cut by one cutting edge of the tool. Useful formulas include the following:

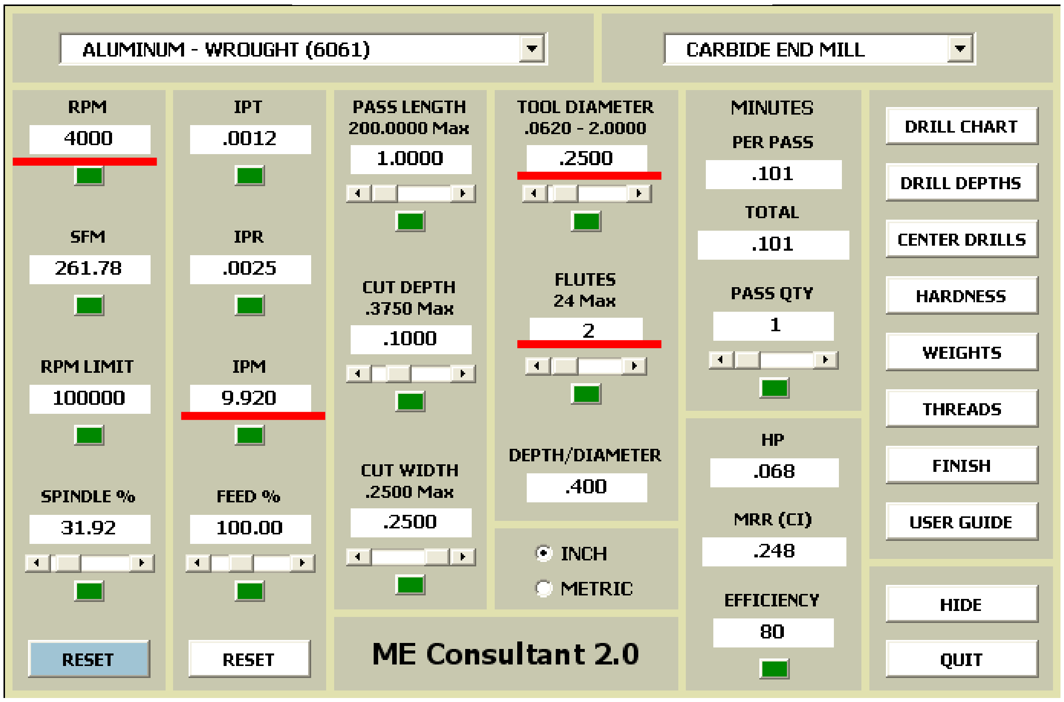
RPM = IPM/Flutes*Chip load
IPM = RPM * Flutes * Chip load
Chip load = IPM / RPM * Flutes

Table 2. Recommended Feed in

Material	Hardness, Bhn	End Mills						
		Depth of Cut .250"			Depth of Cut .050"			
		Cutter Diameter			Cutter Diameter			
		3/8	3/4	1 and up	1/8	3/8	3/4	1 and up
		Feed per Tooth, Inch						
Plain Carbon Steels, AISI 1010 to 1030	to 150 150 to 200	.002 .002	.004 .003	.006 .005	.001 .001	.003 .003	.006 .006	.008 .007
AISI B1111, B1112, B1113	140 to 180	.002	.004	.006	.001	.004	.006	.008
Plain Carbon Steels, AISI 1040 to 1095	120 to 180 180 to 220 220 to 300	.002 .002 .001	.004 .004 .002	.006 .005 .003	.001 .001 .0005	.003 .003 .002	.006 .006 .003	.008 .007 .004
All Alloy Steels Having .3% Carbon Content or Less	180 to 220 220 to 300 300 to 400	.002 .001 .0005	.004 .002 .002	.005 .003 .002	.001 .0005 .0003	.003 .002 .001	.006 .003 .002	.008 .004 .003
All Alloy Steels Having More Than .3% Carbon Content	180 to 200 220 to 300 300 to 400	.002 .001 .0005	.004 .002 .001	.005 .003 .002	.001 .0005 .0003	.003 .002 .001	.006 .003 .002	.008 .004 .003
Tool Steel	200 to 250 250 to 300	.002 .001	.004 .003	.005 .004	.001 .0005	.003 .001	.006 .002	.008 .003
Cast Iron	150 to 180 180 to 220 220 to 300	.003 .002 .002	.006 .005 .004	.008 .006 .005	.001 .001 .0005	.004 .003 .003	.007 .006 .005	.009 .007 .006
Zinc Alloys		.004	.008	.012	.002	.005	.008	.012
Brasses and Bronzes	100 to 150 150 to 250	.003 .002	.006 .004	.010 .006	.001 .0005	.004 .003	.008 .005	.010 .008
Free Cutting Brasses and Bronzes	80 to 100	.003	.008	.010	.001	.004	.008	.010
Cast Aluminum Alloy-as Cast		.003	.008	.010	.002	.003	.010	.012
Cast Aluminum Alloy-Hardened		.003	.006	.008	.001	.003	.008	.010
Wrought Aluminum Alloy-Cold Drawn		.003	.008	.010	.002	.003	.010	.012
Wrought Aluminum Alloy-Hardened		.003	.006	.008	.001	.003	.008	.010
Magnesium Alloys		.003	.008	.012	.002	.004	.010	.014
Ferritic Stainless Steel	135 to 185	.003	.004	.005	.001	.004	.006	.008
Austenitic Stainless Steel	135 to 185 185 to 275	.003 .002	.004 .003	.005 .005	.001 .0005	.004 .003	.006 .004	.008 .006
Martensitic Stainless Steel	135 to 185 185 to 225 225 to 300	.003 .003 .002	.005 .003 .002	.005 .005 .003	.001 .0005 .0005	.004 .004 .003	.006 .005 .003	.008 .006 .004
Plastics		.003	.008	.010	.002	.004	.010	.014

There are many online calculators available that can help us decide what our feeds and speeds should be for a given job. The one I use is called ME Consultant. It is available for download from the class site.

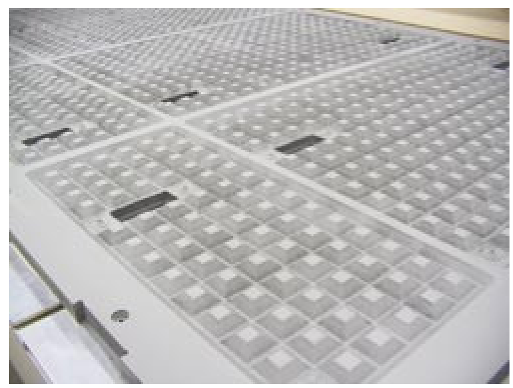
Screen shot from ME Consultant



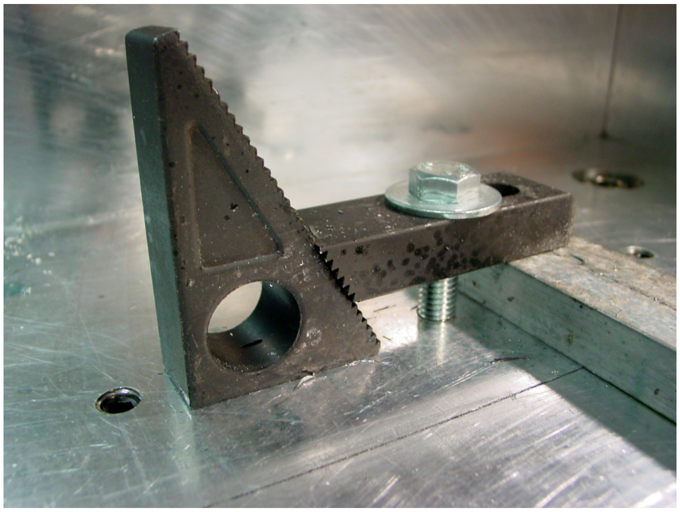
Holding the stock during cutting

The stock needs to be held securely during cutting. Deciding how to setup your stock for machining is critical. Clamps obviously can't be in the way during cutting. If you are cutting through the material, you may also need to consider a sacrificial bed to cut into.

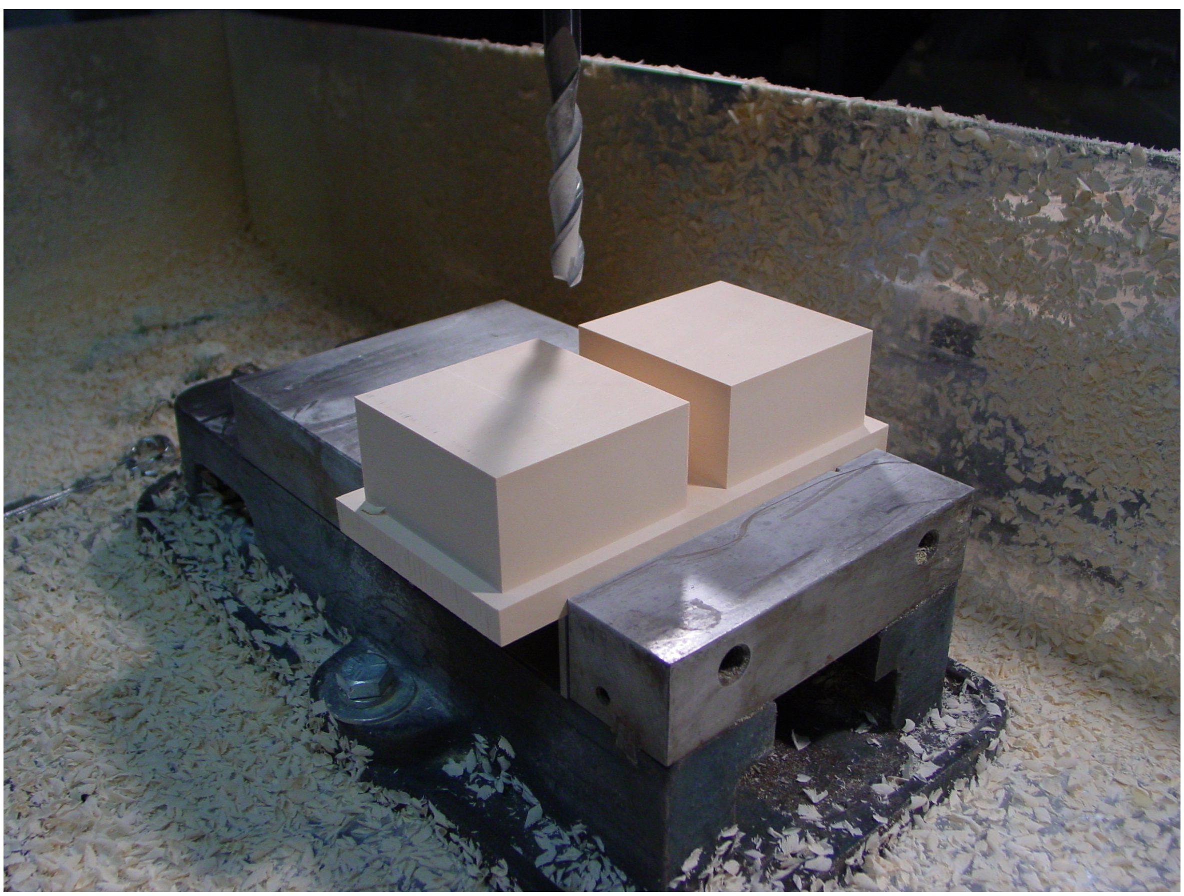
For large, flat stock in softer materials, a vacuum table is sometimes used to keep the material flat during machining. This only works for relatively large sheets of material, and usually where the cutter does not cut through the material entirely.



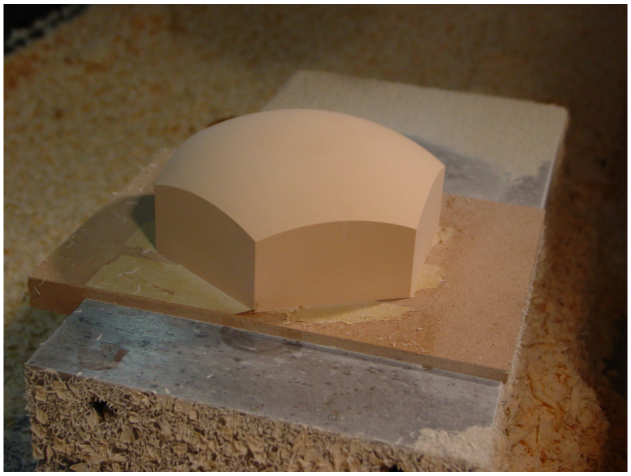
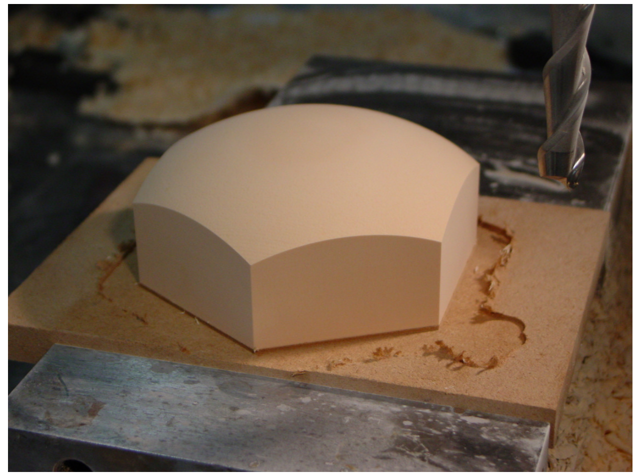
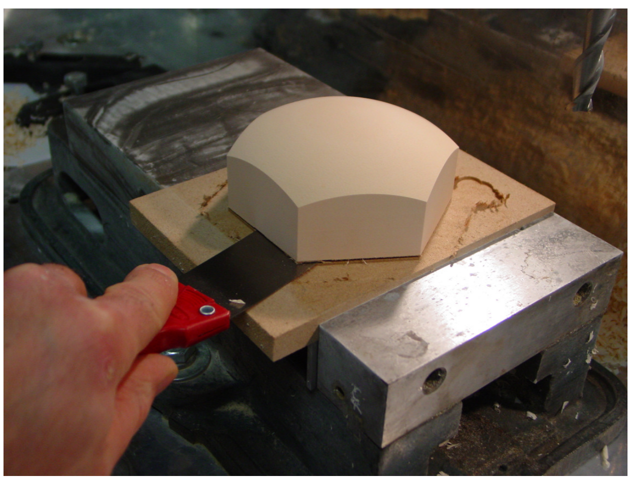
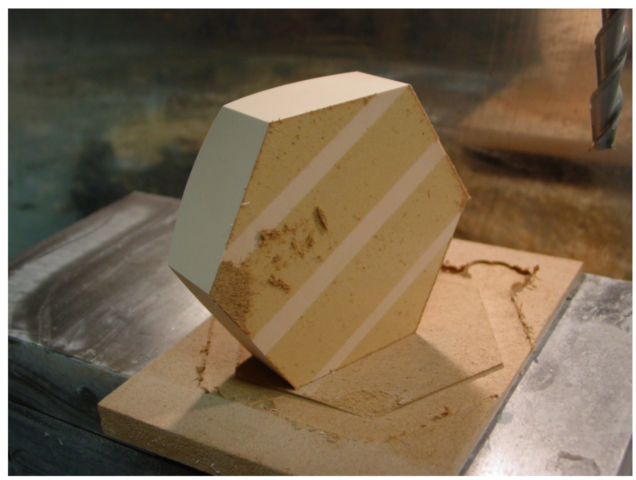
Some machines have t-slot tables that allow for very flexible positioning of the clamps over large areas…



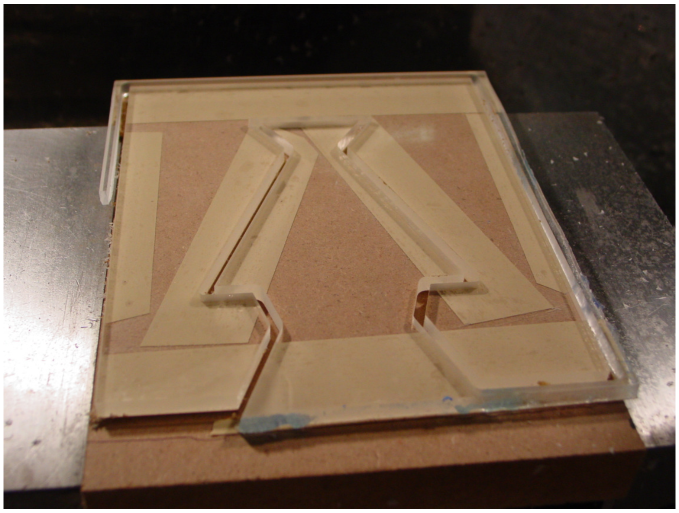
For smaller work, a vice is often used…

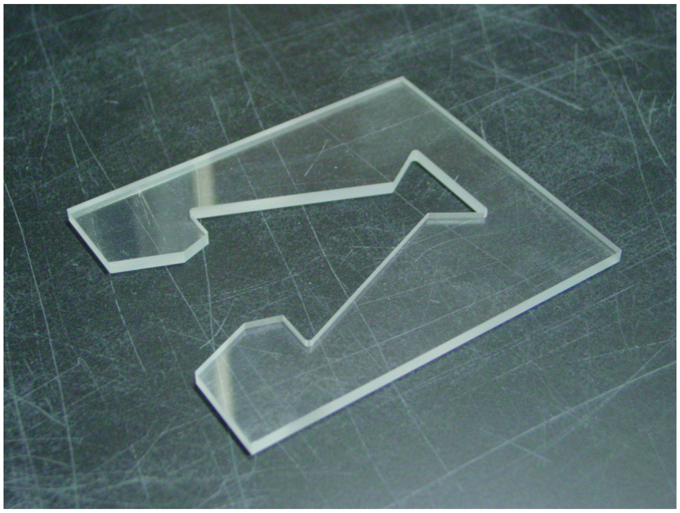


What if the vice gets in the way? Say we want to cut to the bottom of the piece all the way around? For soft materials, like foam, wood, and even aluminum you can get away with using double sided tape! We simply cut all the way through our stock and into the supporting base. The base is known as a “sacrificial board”…

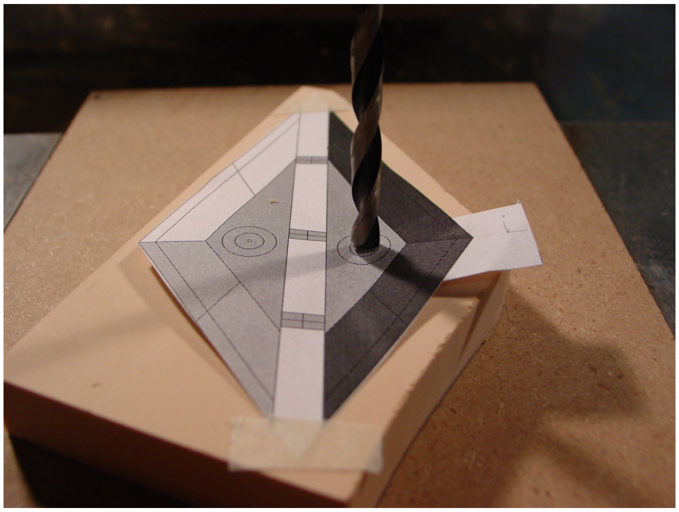





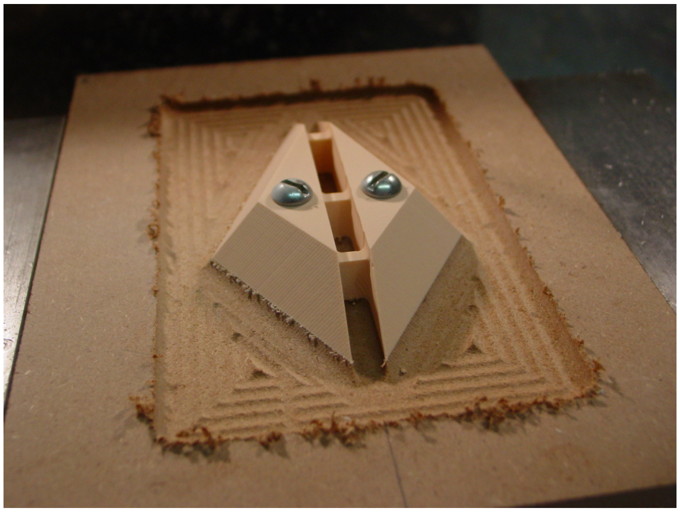
Plexiglas machines very nicely!



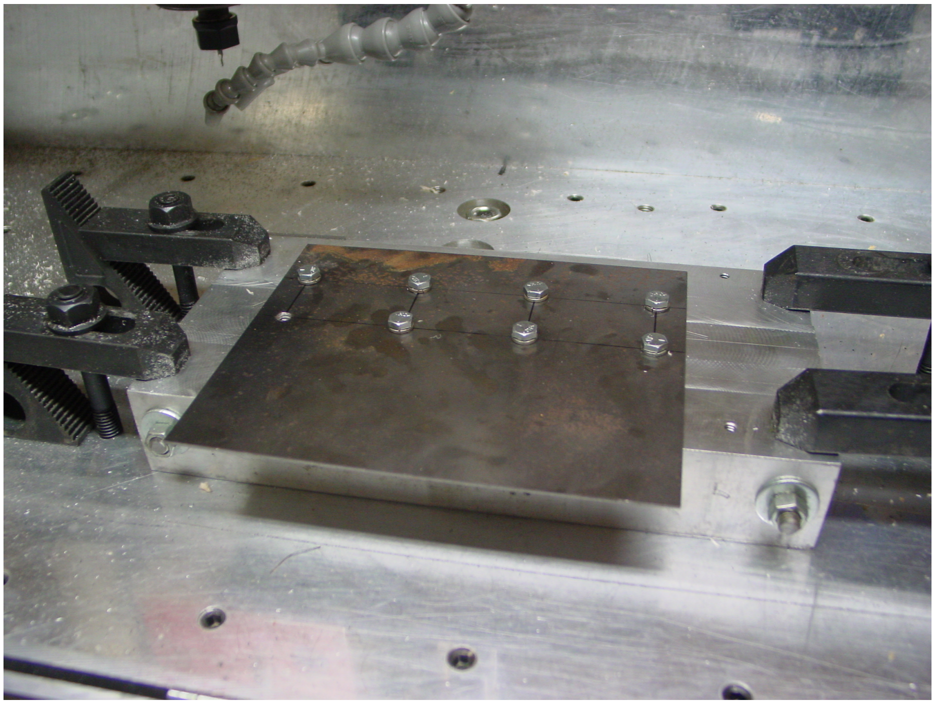


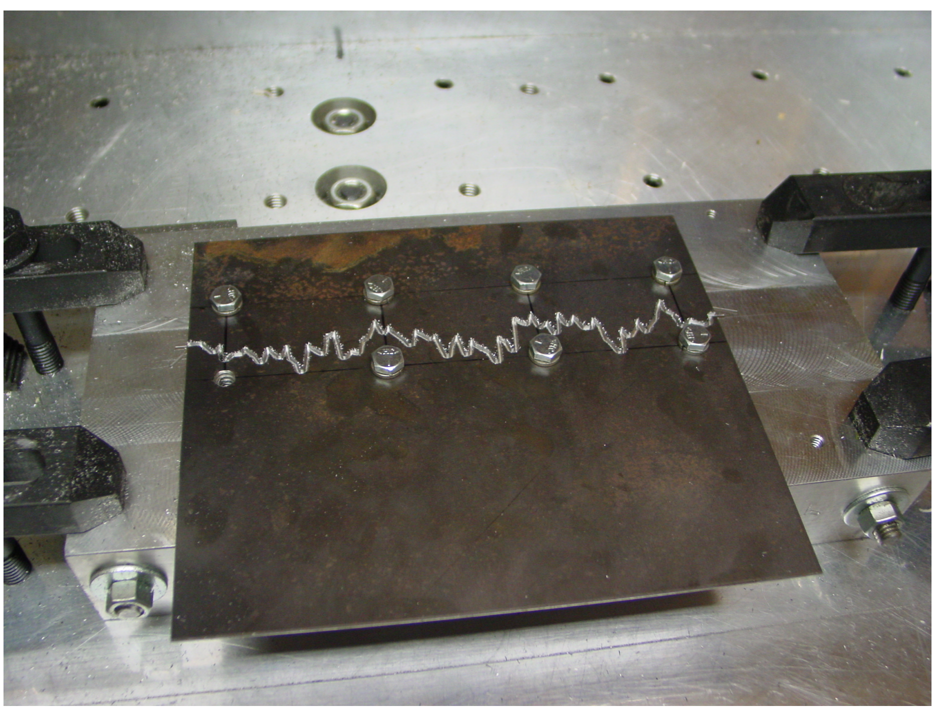
For very small objects, double sided tape is not enough. We may end up screwing or even bolting the object to hold it in place…





Bolting is quite common for metal working in general, as long as you don't mind some holes in the final object…





Videos to check out...

Really simple view of axis moving
http://www.youtube.com/watch?v=pbHPnTzXP2I&feature=related

Wood framed machine moving
http://www.youtube.com/watch?v=XL-aBAR0pFY&feature=related

Homemade CNC router
http://www.youtube.com/watch?v=xlGu51uqNgQ

5-axis clay machining
http://www.youtube.com/watch?v=TqokK03BJsM

Highspeed 5 axis helmet

http://www.youtube.com/watch?v=RnIvhlKT7SY

Linear motor by Sodick
http://www.youtube.com/watch?v=zgGYNiidL1I

Linear motor in use on 3 axis machine
http://www.youtube.com/watch?v=DCxZwSwf64U

Hexapod
http://www.youtube.com/watch?v=mU4mZKpvRLQ

Hexapod industrial example
http://www.youtube.com/watch?v=kS9oxp0mlw8

Hexapod machining a face (DIY)
http://www.youtube.com/watch?v=quN37YskoaM

Kuka arm freestylin'
http://www.youtube.com/watch?v=NDGQYTpPm_A

8 Axis arm for machining
http://www.youtube.com/watch?v=1JcvCZFG0L0&feature=related

Kuka + marble + rotary base
http://www.youtube.com/watch?v=yOp2MDeNLCA&feature=related

CNC router made from… plumbing
http://www.youtube.com/watch?v=6drMZqmyXQc&feature=related