Absolute Coordinates:
Also known as Machine Coordinates. The coordinates of the spindle on the machine based on the home position of the static object (machine). See Machine Coordinates

Absolute Move:
When the machine makes a movement based on the Absolute Coordinates or Machine Coordinates, instead of movements based on work offsets.

ATC:
Automatic Tool Change. When the machine has the capability to change the tool without operator assistance.

Axes:
The X, Y, Z, A, and B dimensions along with motions can be made and cuts can be produced.

Axis:
The singular form of Axes. Example: Just the X, Y, Z, A, or B axis

B-Axis:
An Axis that relates to turning and controls the pivot of the turning center

Backlash:
Backlash is caused by a lack of engagement or over-engagement between the rack and pinion. Also referred to as chatter.

Ball Screw:
The screw shaft in which the Z-axis and B-axis move on along a vertical plane.

Bull Nose:
The Bull Nose center is a 1.4” dia. X 1.25” Delrin, self lubricating, center designed to be placed on the tailstock shaft. The bull nose has been manufactured to an exact thickness to guarantee accurate Z- axis “0” settings when using the bull nose as a touch off method.

The bull nose is a tool touch off method used with the standard OM5 control software specifically designed to accurately set the Z-axis home position for A - axis turning when a homing location is not available. When choosing bull nose as a touch off method, a bull nose is placed on the tailstock shaft and the cutter is brought down to the bull nose surface, the Z-axis is zeroed, the .07 in. is subtracted for the Z to establish “0” for the center of the A work offset. Also known as Live Center

CAD:
Stands for Computer-Aided Design. Software used to create precision drawings or technical illustrations.

CAM:
Stands for Computer-Aided Manufacturing. Application technology that uses computer software and machinery to facilitate and automate manufacturing processes.

Carbide:
A durable material that bits and end mills can be made of to provide long lasting results with less maintenance. There are 3 types of hardness used in woodworking: 1, 2, 3. 1 is the softest with 3 being the hardest.

Chatter:
See Backlash

Chuck:
A device used to mount a piece into the turning center that allows for a solid grip and higher accuracy of diameter.

CNC;
Stand for Computer Numerical Control. A method of controlling tools such as a lathe, mill, router, or grinder.

Collet:
A device that will rest inside the mouth of the router or spindle, that when tightened, will grip the bit of choice and provide stability and security.

Constant Velocity Mode

A mode within the Mach3 software, that will allow the software to read a few lines of code ahead to compensate for changing feed rates and movements along an arc, so that the software may adjust to provide smoother arc movements and give the cut less chatter.

Contouring:
Movement in two or more axes at the same time in order to produce a smooth continuous surface or curve.

Conversational CAM:
A “Computer Aided Manufactucturing” program which uses specific questions to design turnings and flat milling, then produces the G-code needed to run the Legacy CNC milling machine. Conversational CAM was designed by Legacy to streamline the design process of commonly milled components without the use of **CAD** (Computer Aided Design) software program to draw the parts. CAD programs are design programs only and do not produce G-code.

Crash:
When the machine is attempting to move along an axis and is being obstructed.

Cycle Start:
The button within Mach3 that will start the process outlined by the code of choice loaded at that time.

Cycle Stop:
The button within Mach3 that will stop the process outlined by the code of choice loaded at that time.

DRO:
Digital Read Out

Edge Finder:
A tool that may be placed in a router or spindle that will project a laser dot for helping to find edges accurately and quickly.

E-Stop:
A physical button on the machine that when pressed, will cease any operations in place on the machine to prevent damage or unsafe work conditions. Must be disengaged to allow the machine to be used again.

Feed Rate:
The speed the cutter moves over the part, measured in IPM “inches per minute”. When using the A-axis , the feed rate is the speed the part moves past the cutter.

Guidelines: larger diameter cutter = slower feed rate
Deeper the cut = slower feed rate.
If the bit chatters, slow down the feed rate speed up the RPM’s
Recommended feed rates: flat stock 80 - 120 IPM
Turning 300 - 350 IPM

G-code:
G-code functions in the numerical control programming language. G-codes are the codes that position the tool and do the actual work. Under G-code there are several types of special codes each with a label.
M-codes: manage the machine
T-codes: represent a tool
S-codes: relate to tool speed
F-codes: represent feed rate.
H-codes: represent a tool height offsets. These are mainly used in establishing tool libraries.

Legacy CNC Control Interface:
The Legacy CNC Control Interface software has been custom designed by Legacy as an interface to Mach3 and is the control software used to run all the Legacy CNC Routers. The Legacy CNC Control Interface software was designed as a user friendly screen interface to help utilize the unique milling attributes of the Legacy CNC Routers.

Machine Coordinates:
Machine Coordinates represent the “home” location set on each axis when running the “reference machine” setup sequence. Machine coordinates are not tied to any offsets, in fact it runs behind any offset you use. When using machine coordinates the DRO’s will show you how far the spindle is away from the “home” location.

Part:
The Material to be cut by the machine.

Part Z-Zeroing:
The old fashioned method of slowly lowering the Z-axis, with your cutter in the spindle, until it barely touches the top of the part. Normally a piece of paper is wiggled between the cutter and part until the paper can no longer be moved.

Reference Machine:
A button in Mach3 which triggers a background code. This background code tells the machine to travel at a slow rate towards the Machine Home location. Each axis (X,Y,Z) will travel on its own until the machine triggers the limit switches. Each axis will trigger the limit switch twice. Once the reference has finished the machine now has its "home" location. **THE MACHINE MUST BE REFERENCED EVERY TIME IT IS TURNED ON AND AFTER A CRASH!**

RPM- Rotations Per Minute:
Recommend maximum spindle/router bit RPM. Adjust listed speeds according to your spindle/routers max RPM capabilities.

Bit Diameter - RPM

0.00"-1.00" - 22,000-24,000
1.01"-2.00" - 19,000-21,000
2.01"-2.50" - 16,000- 19,000
2.51"-3.00" - 14,000- 16,000

Smart Tool:
This Legacy designed programming process is placed within the Mach Control software and activated through the Legacy CNC control interface. Smart tool is designed to measure the Z zero plane on flat stock parts during smart tool setup and smart tool heights during tool changes.

Smart Tool Mobile Pad:
A mobile touch off pad, designed by Legacy to aid in the smart tool touch off process. Each machine comes with its own pad. If you switch out a smart tool pad the smart tool pad height must be changed in Mach3.

Smart Tool Touch Off:
This legacy designed process allows you to accurately set the Z-zero plane to the surface or bottom of your flat stock workpiece by using the Smart Tool Mobile Pad.

Spiral formula:
(diameter of blank x2)/ diameter of tool = number of starts

Stepper Motor:
A brushless, synchronous electric motor that can divide a full rotation into a larger number of steps. The motor's position can be controlled precisely without any feedback mechanism (open loop controller).

Surface Planning:
To smooth or level the uneven surface of a workpiece by using a flat bottomed bit (surfacing bit) to accurately remove material.

Soft Limits:
A setting within the Mach3 software that will provide a safe boundary that the machine will not exceed unless directed otherwise.

Tool Offset:(Auto Tool Change)
If you plan on using your tools as effectively as possible you will need to set up a tool library. In this library each tool set up in a tool holder will be given a specific height. This means that the machine will measure the tool once during the setup using the smart tool pad and establish the tool offset. After this offset is created the machine will never need to touch off for that tool after a tool change, so long as you do not adjust the the tool in the holder.

Tool Touch Off Methods:
These are the methods available to measure the tool height in relationship to the Z “0”. The four tool touch off methods used are Smart Tool, Tool Offset, Part, and Bull Nose.

Work Offset:
The work offset represents a location “offset” from the home coordinates where work will be performed. These will appear as 0’s in the DRO’s regardless of the Machine coordinates. The work offset for flat stock milling is manually set by moving the spindle head to the where the XYZ origin is programmed to be(we recommend a laser or a V-bit). Once the spindle is in correct location click the XY buttons to set them to 0. Then use one of the Z zeroing methods. For turning using the standard indexing hub enter 6 into the work offset box. Each machine can hold 256 work offsets.

XYZ Origin:
The XYZ origin is the the name of the “0” locations programmed for each part. Aspire can create a job setup sheet to show where the origin is located. The standard XY origin is a corner or the center while Z origin is either the top plane of the material or the bottom plane of the material.