

Dashboard™ Training

This Demonstration Dashboard will work on any PC or Mac, preferably in the Chrome browser. It will not work on an iPad unless you use the Photon browser. For the purposes of this demonstration we have added some buttons just below the Dashboard™ to speed you through the demo process. The Dashboard™ can be manipulated manually at anytime during the demonstration as you would with any delivered Dashboard™. You can click on, hold down and adjust the speed dial (you can click on the dial needle or use the slider bar), depth of cut, width of cut and feed rate. There are arrows for fine adjustments to the values. For low speed applications or to zoom into a stable pocket, you can change the resolution of the speed dial with the grey slider bar below the dial and speed adjustment bar. You can also highlight a value in one of the fields and type in a new value to change it. Simply put, if everything is green, the calculated parameters will provide a stable cut.

Let's begin.

The Dashboard™'s current state is where the user was running before BlueSwarf®. You can return to these values by clicking on the yellow "**Current Parameters**" button on the lower left, or the "**Reset**" button on the Dashboard™. NOTE: If the user did not provide the before speed, feed and depths of cut, everything would default to zero.

In this case, the user hit **chatter at 10,400 RPM and at 0.280" depth of cut in a full slot cut.**

Clicking on the red "**Catalog Recommendations**" button shows what the table in the cutting tool catalog suggested for speed and depth of cut. In this case it recommended a 1X cutter diameter axial depth of cut for a slot at the maximum speed of the machine for cutting aluminum. **This would exceed the power limits of the machine and was in a chatter pocket.**

The BlueSwarf Dashboard® recommendations are shown incrementally with the three green numbered buttons.

Press (1) and the best speed is selected. NOTE: We selected a speed that was not in the fastest green zone on the dial because this would have put us in the risk for resonance as indicated by one of four color codes in the "**Chatter Status**" bar that is at the top of the Dashboard™. The color codes are as follows:

- **CHATTER (Red):** Violent vibration and cutting forces, risk breakage of the cutter, poor surface finish and possible damage to the machine.
- **STABLE, RESONANT RISK (Orange):** Higher level vibrations. Not generally recommended but can be utilized if acceptable surface finish is found on the workpiece. Can possibly lead to poor tool life.

- **STABLE, CHATTER RISK (Yellow):** No chatter predicted but there is a high risk of experiencing chatter or resonant vibrations if conditions change, e.g. tool length, depth of cut, material hardness, tool condition, etc.
- **STABLE (Green):** Optimal cutting conditions. Lowest vibration levels. Will achieve best tool life, minimize tool failure and best surface quality.

Press (2) and with the speed stable we can increase the depth of cut. You can manually adjust the depth of cut by clicking and holding on the green bar. Note how the speed ranges on the RPM dial change as you change the **“Depth of Cut”**. Dial will also recalculate when you change the **“Width of Cut”**.

Press (3) and because we have sufficient power to increase the feed rate. Try moving the **“Feed Rate Bar”** manually by clicking and holding on the arrow and see the changes in the **“Power”** gage in the middle right of the Dashboard.

Even though we slowed the speed, we increased the depth of cut and feed rate. The result is an increase in metal removal rate from 26 cubic inches per minute to 55. Doubling the Productivity! The cut is stable, tool wear and energy consumption are more efficient. Finish and accuracy is improved.

PROCESS DAMPED – Process damping is when we run a slow enough spindle speed whereby the vibration of one tooth is fully dissipated before the next tooth comes through. It is actually be rubbed away from the cutter's edge prep or secondary relief geometry. At this slow speed, you can cut as deep as the cutter edge or flute length will allow. To demonstrate this, click on the **“Process Damped”** button. The speed will slow to 500 RPM and the depth of cut will increase to 0.7800. Even at full depth of cut you are removing only 3.5 cubic inches of material per minute as compared to 55 for the Dashboard™ calculated values.

b LIM or b CRITICAL – This also called the “unlimited speed” range and it is used for the class of milling tools called “Feed Mills”. In this case, if we cut shallow enough (low depth of cut) we are not deflecting the tool enough to cause chatter, so we can use a very fast spindle speed. However, resonance can still be a problem and may hinder performance. To demonstrate, click on the **“b LIM”** button and the depth of cut will be reduced to 0.100”. All of the speed zones turn green, but if you increase the speed above 9400 RPM you will be in resonance or resonance risk areas. Even at the maximum spindle speed of 12,000, and in resonance, your maximum metal removal rate is only 10 cubic inches per minute as compared to 55 for the Dashboard™ calculated values.

NOTE: In both Process Damped and b Limited ranges, the cut looks good, you are cutting either very fast or very deep, but look at the value in the Metal Removal Rate field. It is very low when compared to cutting in a stable lobe. Productivity is severely reduced when running in these regions.

CHIP THINNING – To demonstrate our automated Chip Thinning feature click and hold on the green width of cut bar and drag it until it goes below 50% engagement. Note how the **“Feed Rate”** field (in/tooth) automatically starts to increase to compensate for chip thinning.

ANALYTICS – Click on the **“Analytics”** tab to show how the performance increases will impact the shop's cost for this tool's process. The pie chart shows the industry average breakdown of a shop's cost by percentage. Some are variable, that is, they can be reduced with improved performance. Others, specifically **“Material”**, are fixed. Users can adjust the percentages to fit their shop, but the total must add up to 100%. The **“Shop Rate”**, **“Tool Cost”**, **Tool Life** and **“Tool Change”** time can also be edited by the user and are used to calculate true tool cost. We are using formulas developed by Boeing's R&D unit (the Phantom Works). For roughing we are calculating **“Cost per Cubic Inch Removed”**. Note that **“Tool Cost”** is defined as the replacement cost of the endmill or a set of inserts divided by the number of indexes (number of usable edges) per insert. For example, if you have a four insert cutter, the inserts cost \$10 each and have two useable edges per insert, your **“Tool Cost”** would be \$20, $(4 \times \$10)/2 = \20 . If they don't know the **“Tool Change”** time, you could leave it as a common value so as not to impact the comparison. The dials are the same as those on the **“Dashboard™”** tab, but we are adding the actual calculated values. For **“MRR”** and **“Tool Life by MRR”** we begin at zero and you can see what impact your changes you make will have on **“Productivity”** and **“Tool Life”**. For example tool productivity may go way up but actual tool life may go down because you are running much faster. If maximum tool life is more important to you than throughput you would pay attention to this dial. **“Spindle Efficiency”** is a calculated value based on the spindle load and the metal removal rate. Note: AC motors become increasingly inefficient when operating at less than 50% of their rated load. This value will ensure that you are properly utilizing your spindle's capabilities.