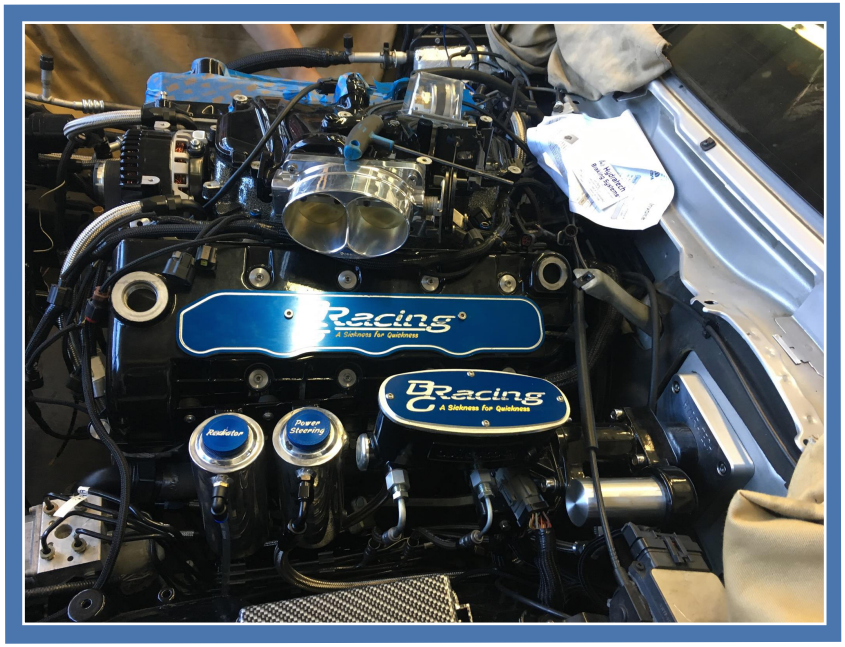
VisualCAD/CAM at BRG Racing Products

BRG Racing Products is located at 110 2nd Ave S, Ste D12 Pacheco, CA 94553 and specializes in high-performance custom vehicles including 1950,1960, and 1970 era muscle cars, Hary Davidson motorcycles, sport bikes, dirt bikes, and more! BRG Racing takes and finishes the custom jobs that other shops cannot or refuse to handle, all on time and under budget!

KC Gager, Owner/Operator started out as a mechanic in 1988 and quickly started his own auto repair shop. From there BRG Racing was born, starting out manufacturing lightweight aluminum wheels and motorcycle conversion kits.



The VisualCAD/CAM Difference

For years KC started out operating two 2½ Axis manual turning lathes. That led to his first 2½ Axis CNC machine from Laguna Tools that used the Bandit® control software. For CAD/CAM KC was using GeoPath® software from Solutionware. That worked for a while but soon became very cumbersome. To expand his operations into 3 Axis machining, KC migrated to a 3 Axis mill from CNCMasters and chose VisualCAD/CAM from MecSoft Corporation. We recently sat down with KC to discuss his recent success at using VisualCAD/CAM. Here is some of what KC had to say about his VidualCAD/CAM software.

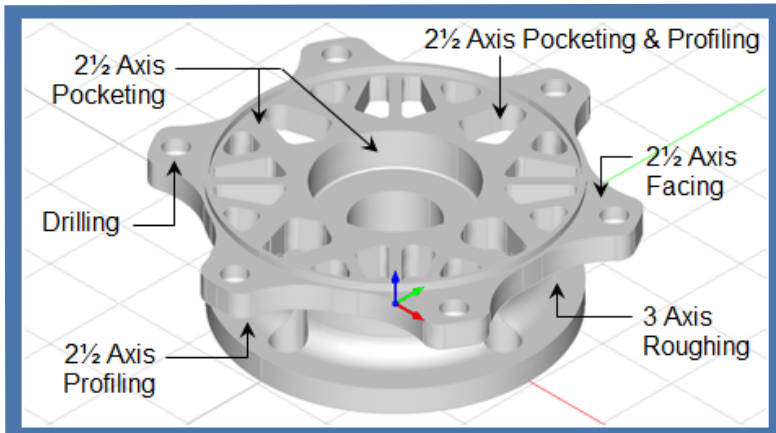
"Our experience with VisualCAD/CAM has been exceptional. The program is easy to learn and easy to use! Getting G-Code out to our CNCMasters 3-Axis mill is much faster now than our previous 2½ Axis GeoPath CAD/CAM system from SolutionWare."

K.C Owner/Operator,
BRG Racing Products and Dyno Services Pacheco CA

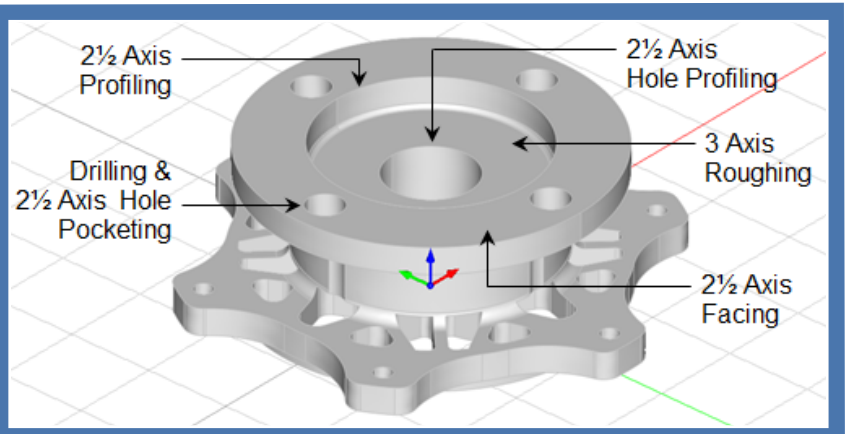
The VisualCAD/CAM Part

The part selected for this case study are custom wheel hubs that allow you to mount GSXR 600 brake rotors onto a BMW bike front wheel. This then will allow you to upgrade to a Suzuki front end that provides much better performance than the stock front end provided by BMW. he component is cut from 6061 aluminum and is 5.16” inches in diameter and 1.63” inches wide.

The part contains an interesting set of features including profiles, pockets, flanges, and holes as well as machining from two sides. The center thru-hole requires some tight machining tolerances. KC can CAM program all of these features to the required specifications using his VisualCAD/CAM software. We invite you to continue reading to learn more about this component and how it was programmed and posted to KC’s CNCMaster 3 Axis mill!



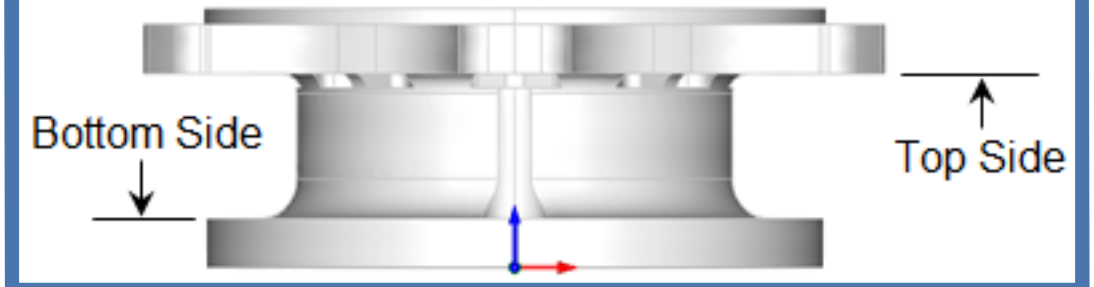
This component is 5.16” inches in diameter and 1.63” inches wide. The 3D solid model of the part oriented for machining the top side is shown above.



The bottom side of the component contains a flange with 4 mounting holes, a larger pocket, and a thru-hole. The 3D solid model is shown here oriented bottom side up.

The VisualCAD/CAM Setups

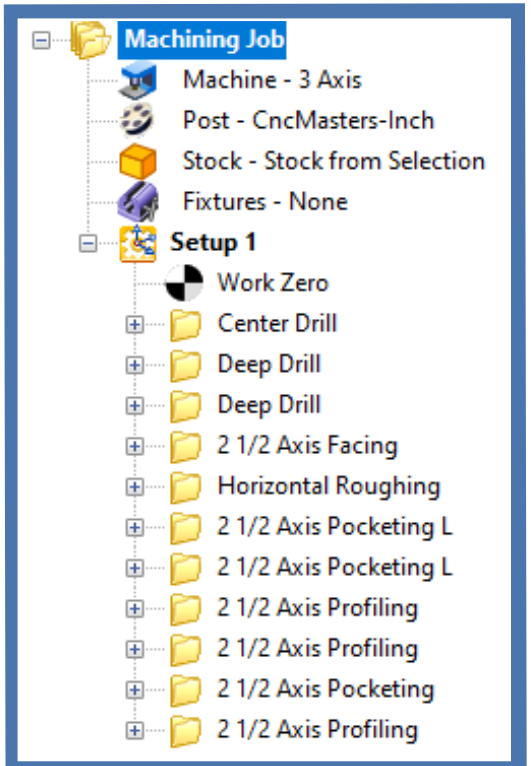
For this part, the stock blank was turned to part geometry specifications. The turned blank was then used to machine the top and the bottom of the part as illustrated here in this case study. The elevation view of the part is shown below with the scope of the top and bottom machining job setups defined. The machining of this part is illustrated using the Standard configuration of VisualCAD/CAM. The benefits of using the Professional configuration are discussed here also. Read on for some cool CNC machining!



An elevation view of the part is shown above. The scope of the top and bottom machining job is clearly defined.

Setup 1 (Top Side)

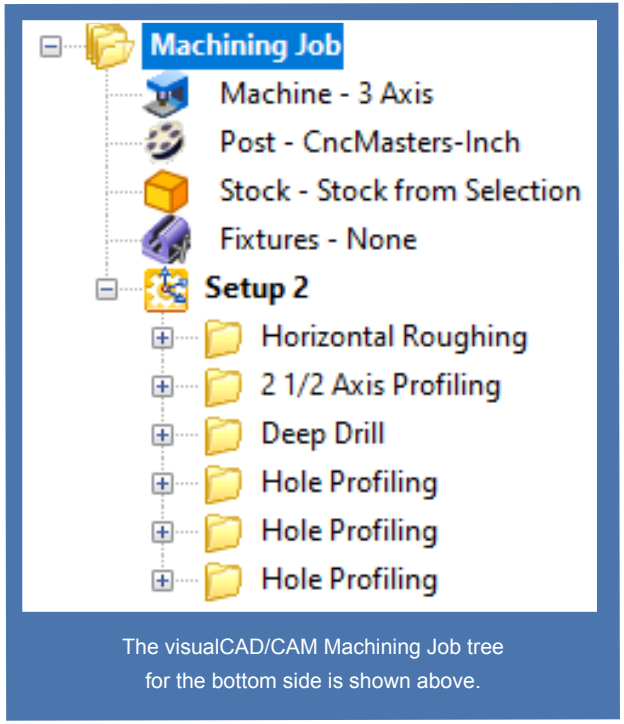
The machining job tree for the top side of the part is shown below. You will see that it contains Drilling operations first. Then followed with 2½ Axis Facing to level the top, followed with a 3 Axis Horizontal Roughing operation to remove the bulk of the stock material. This is then followed by 2½ Axis Pocketing and 2½ Axis Profiling operations. Note that the center thru-hole is left rough from the top side and is machined from the bottom side.



The visualCAD/CAM Machining Job tree for the top side is shown above.

Setup 1 (Bottom Side)

The machining job tree for the bottom side of the part is shown below. You will see that it begins with a 3 Axis Horizontal Roughing operation to remove the bulk of the stock material. This is followed by 2½ Axis Profiling and Drilling to pre-finish the 4 holes. This is followed by 2½ Axis Hole Profiling operation to finish the 4 holes. The last two 2½ Axis Hole profiling operations will finish the center thru-hole to specifications using tighter machining tolerances. The Bottom side is illustrated in detail below so please continue reading!



The visualCAD/CAM Machining Job tree for the bottom side is shown above.

Machining the Top Side

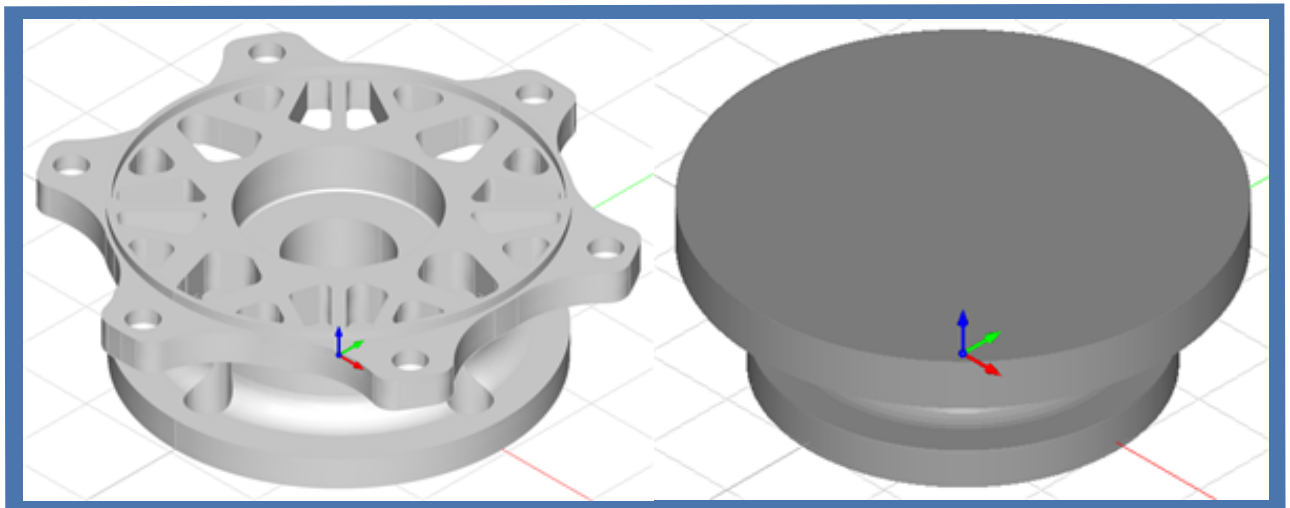
As we mentioned above this is a 2-sided (flip machining) part using the Standard configuration of VisualCAD/CAM. 2-sided components must be programmed in two separate CAM files in the Standard configuration of VisualCAD/CAM. The top side is one CAM file and the bottom side is a second file. The Machining Job for the top side is shown above. It should be noted that the method of fixing the stock is not discussed here.

2-Sided Machining Videos

If you are unsure if your parts require 2-sided machining, we invite you to watch a few of our videos on this subject. They are listed below. Enjoy!

1. CAMJam Short #253: Do I Need 2-Sided Flip Machining?
2. CAMJam Short #249: 2-Sided Flip Machining in PRO vs STD
3. CAMJam Short #250: Basic 2 Axis Bridge-n-Flip Method
4. CAMJam Short #251: Basic 3 Axis Box-n-Flip Method
5. CAMJam Short #252: Advanced 3 Axis Flip-n-Jig Method

In the left image (below) we see the part geometry model oriented to machine the top side. On the right side, we see the stock blank. It should also be noted that the actual stock blank is a complete cylinder turned to the actual part cross-section. The process of turning the blank is not discussed here.

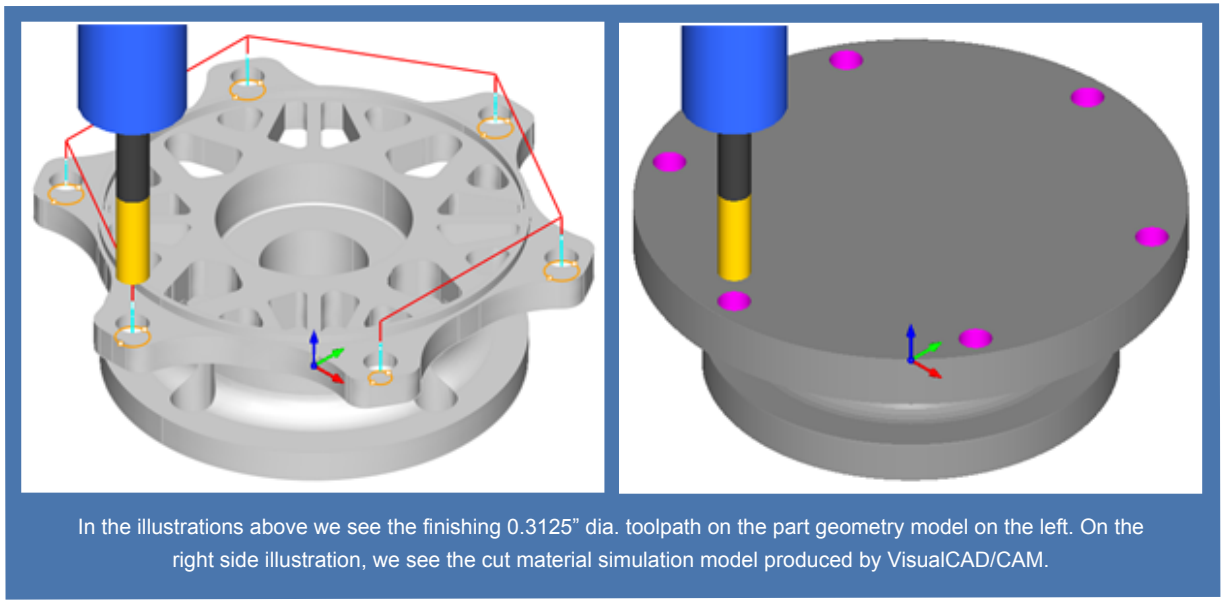


The CAD geometry model is shown on the left side image above. On the right side, we see the stock blank.

Drilling Operations

As shown in the Machining Job above the first 3 folders are Drilling operations. The first two are roughing operations. The first of these is a Center Drill of 0.125" dia. The depth of the cut is 0.01". The second is a Deep Drill operation using a #7 (0.201" diameter) drill cutting 0.35" deep with 0.1" peck increments.

The third is the finishing operation using a 0.3125" drill cutting 0.30 deep with 0.05" peck increments. Note that you do not need to use three drilling operations. We use them here to illustrate the use of pre-drilling operations. In the illustrations below we see the finishing 0.3125" dia. toolpath on the left and the cut material simulation illustrated on the right.

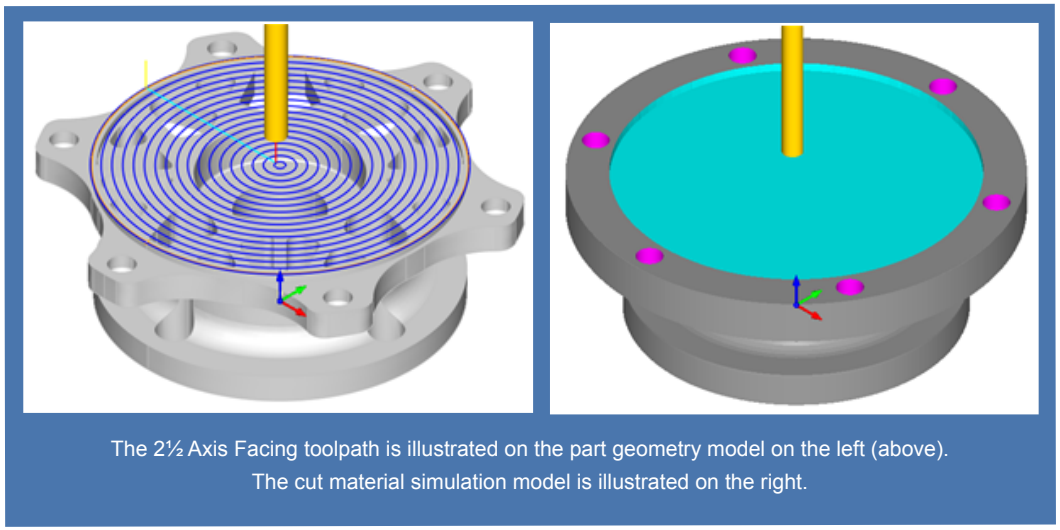


In the illustrations above we see the finishing 0.3125" dia. toolpath on the part geometry model on the left. On the right side illustration, we see the cut material simulation model produced by VisualCAD/CAM.

2½ Axis Facing

The next operation in the Machining Job is a 2½ Axis Facing operation. In this operation, a 0.250" dia. end mill cuts one level at the top of the part. The control geometry is the outer diameter of the upper lip. The cut parameters include 0.001" tolerance, Stock of 0.025" (in 2½ axis stock is applied to the X and Y directions only). The facing Cut Pattern is set to Island Offset Cuts, with a Mixed cut direction, and a stepover of 45% (of the tool dia.).

The location of cut geometry is set to At Bottom, the total cut depth, rough depth, finish depth, rough depth per cut, and finish depth per cut are all set to 0 (zero). This means that there will be just one cut level with the bottom of the cut residing on the top of the part lip. The 2½ Axis Facing toolpath is illustrated on the left (below) and the cut material simulation model is illustrated on the right.

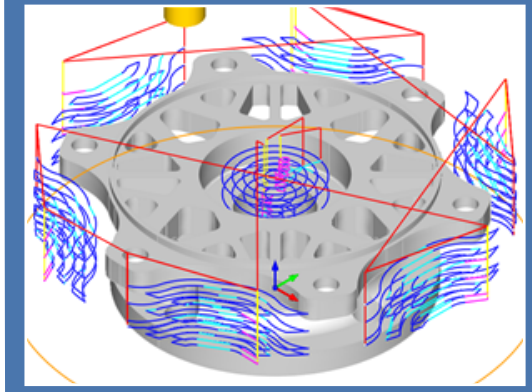


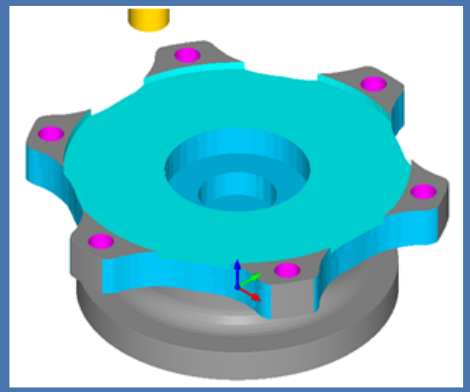
The 2½ Axis Facing toolpath is illustrated on the part geometry model on the left (above).
The cut material simulation model is illustrated on the right.

3 Axis Horizontal Roughing

The next operation in the Machining Job is 3 Axis Horizontal Roughing. It is used as a roughing operation to remove excess stock material. In this method, VisualCAD/CAM automatically calculates what material needs to be removed based on a 0.500” diameter end mill, the part, and the stock geometry. The cut parameters used include 0.01” tolerance, 0.025” stock allowance, an offset cut pattern, mixed cut direction, a 25% stepover.

Each cut level depth is set to remove 50% of the tool diameter (which is 0.25” inches), depth-first ordering, and a bottom limit of 1.024”. The cut engagement is set to helical motions. Arc Fitting is also enabled (blue-colored motions).
The resulting toolpath is illustrated on the part geometry model on the left (below). The cut material simulation is illustrated on the right.

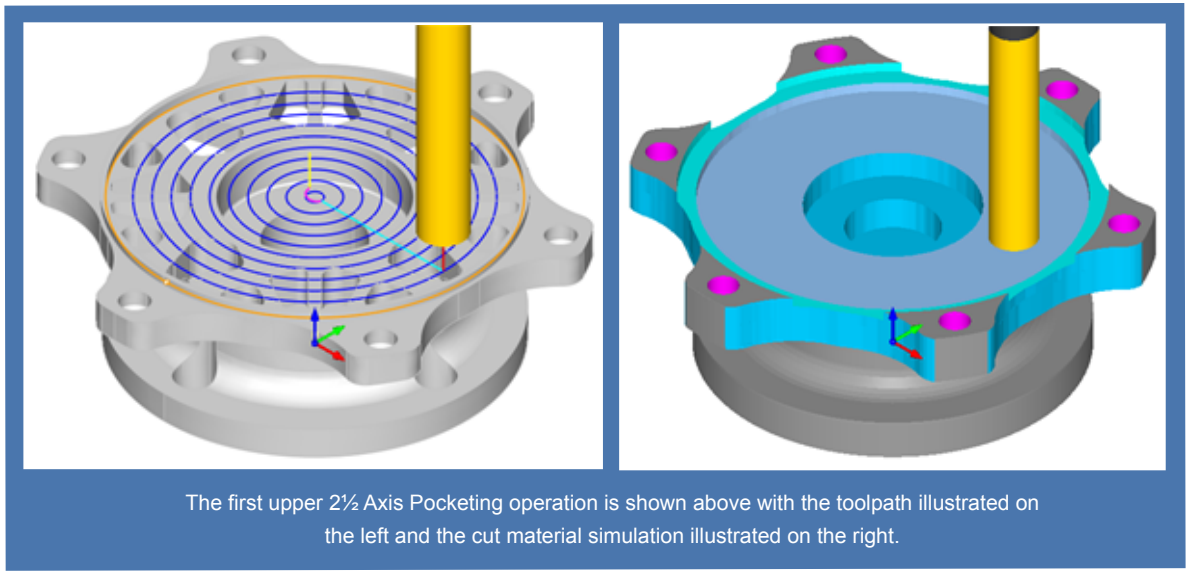




Here we see the 3 Axis Horizontal Roughing toolpath illustrated on the part geometry model on the left. The entry engagement is set to helical motions. Arc fitting is also enabled (blue-colored motions). The resulting cut material simulation model is illustrated on the right.

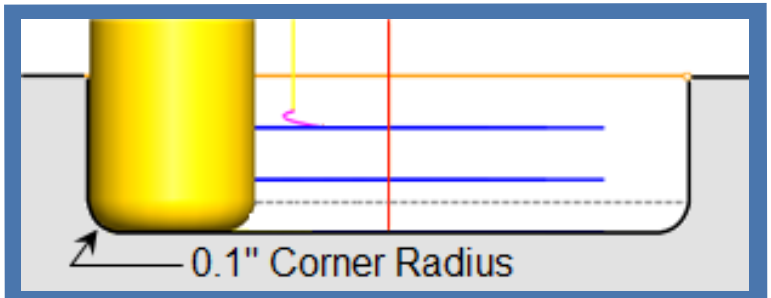
2½ Axis Pocketing

The next two methods in the Machining Job are 2½ Axis Pocketing operations. These will cut the upper 3.82” diameter pocket (on the inner side of the lip) and the smaller 1.80” diameter pocket below it. The first pocket is again one cut level with stock allowance set to 0 (zero), an offset cut pattern, climb cut direction, and a 35% stepover. The first pocket is shown below.

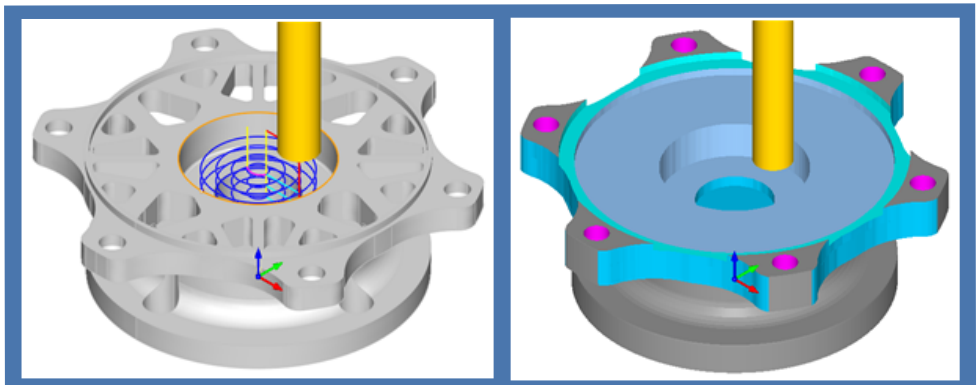


The first upper 2½ Axis Pocketing operation is shown above with the toolpath illustrated on the left and the cut material simulation illustrated on the right.

The second smaller 1.80” diameter pocket uses a 0.50” diameter corner radius mill (also referred to as a bull mill) with a corner radius of 0.05”. This radius matches the filet at the bottom of this pocket. The remaining cut parameters are similar to the first pocket but with three cut levels 0.188” deep each. Both pocketing operations employ a ramp entry motion and linear exit motion. The second pocket is shown below.



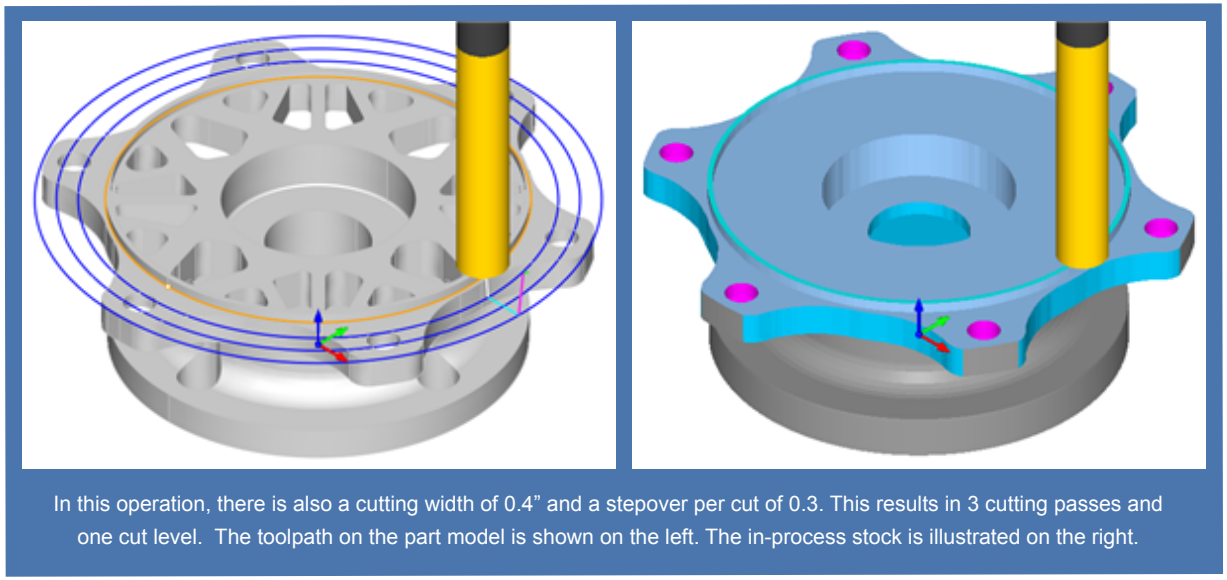
The inner base of this pocket has a 0.05” radius. It will be cut using a 0.50” diameter x 0.05” Corner Radius Mill.



The second, lower 2½ Axis Pocketing operation is shown above with the toolpath illustrated on the left and the in-process cut material simulation illustrated on the right.

2½ Axis Profiling

The next two methods in the Machining Job are 2½ Axis Profiling operations. The first cuts the top perimeter of the upper lip (level with the upper flange containing the drilled holes). This is shown in the left illustrations below. The cut parameters for the first Profiling operation include a 0.001" tolerance, a stock allowance of 0 (zero), a mixed cut direction, and arc fitting (dark blue motions). In this operation, there is also a cutting width of 0.4" with a XY stepover per cut of 0.3".This results in 3 cutting passes and one cut level. The resulting in-process cut material simulation is illustrated on the right below.



In this operation, there is also a cutting width of 0.4" and a stepover per cut of 0.3. This results in 3 cutting passes and one cut level. The toolpath on the part model is shown on the left. The in-process stock is illustrated on the right.

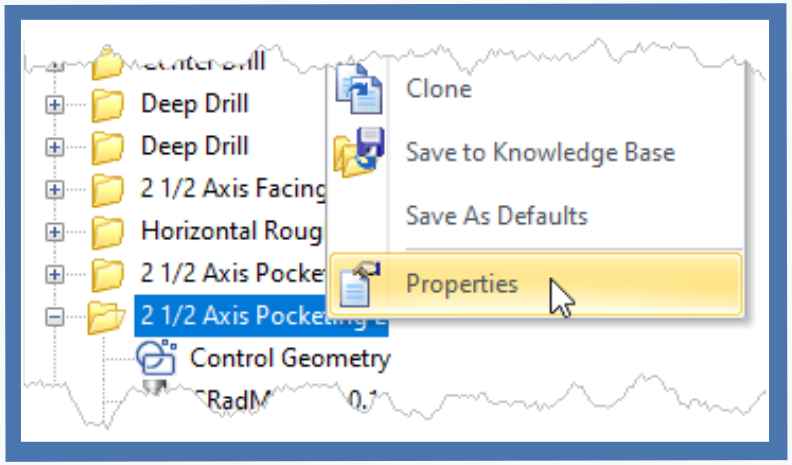
Note:

You may be asking yourself:

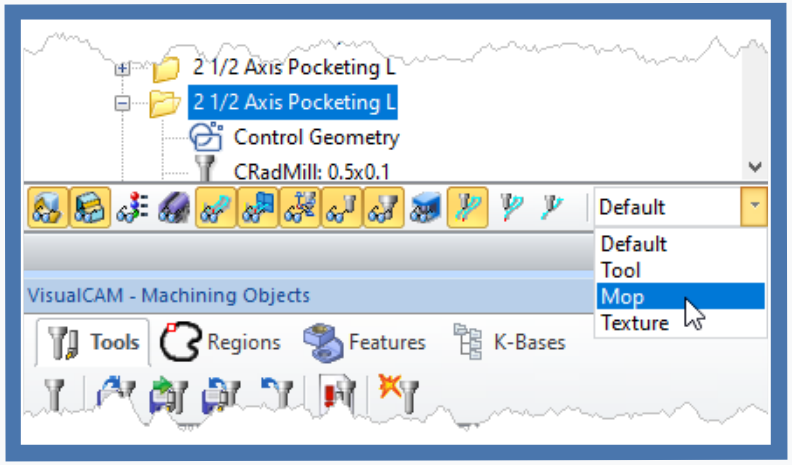
"How can I make my toolpaths simulate in different colors?"

Here are the steps to make this happen:

1. After defining the toolpath operation, right-click on the folder from the Machining Job and select Properties. Use the dialog to set the simulation color. Do this for each toolpath operation in the Machining Job tree.

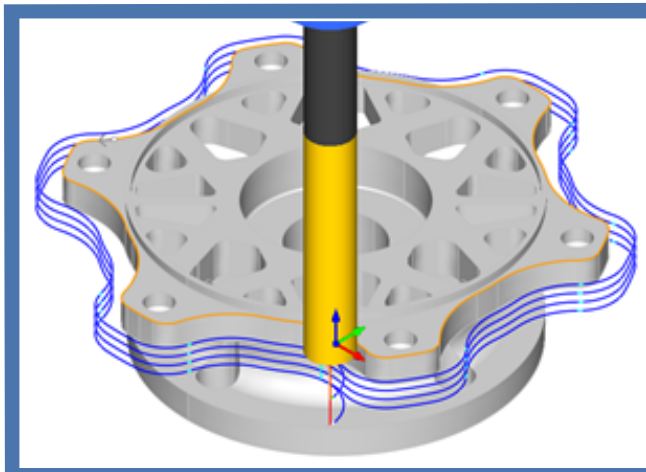


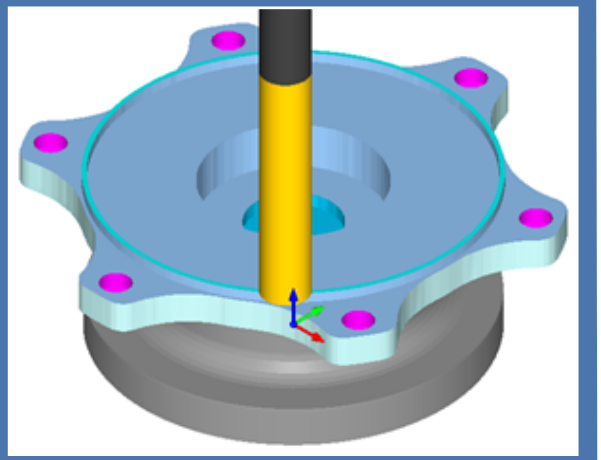
2. Then select the Simulate tab and look at the toolbar at the bottom. Where it says “Default” drop the menu down and select “MOp”. This stands for Machining Operation.



3. Now simulate your machining Job to see the colors applied.

The second Profiling operation cuts the outer perimeter of the part (around the flange containing the 6 mounting holes). The cut parameters for the second Profiling operation include a 0.001" tolerance, a stock allowance of 0 (zero), a mixed cut direction, and a total cut depth of 0.4 with 0.1" stepdown increments resulting in 4 cut levels. These can be seen in the illustrations below.



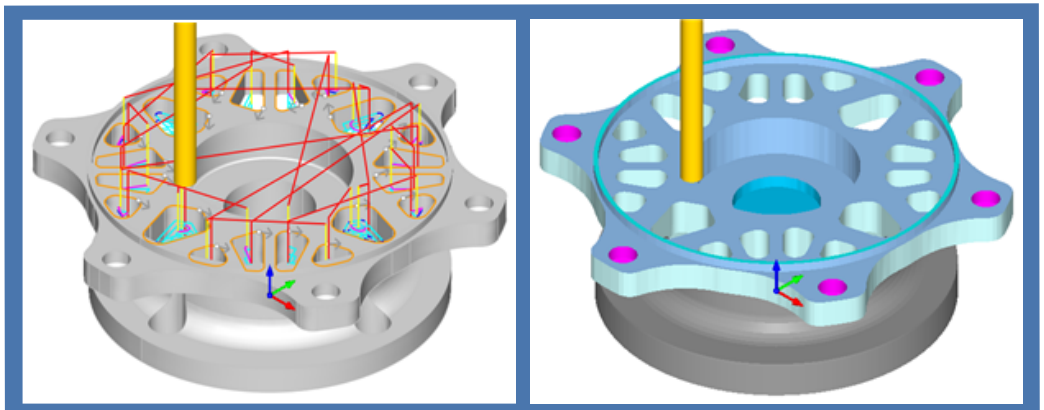


The cut parameters for the second Profiling operation include a total cut depth of 0.4" with 0.1" stepdown increments resulting in 4 cut levels. Refer to the illustrations above.

2½ Axis Pocketing & Profiling

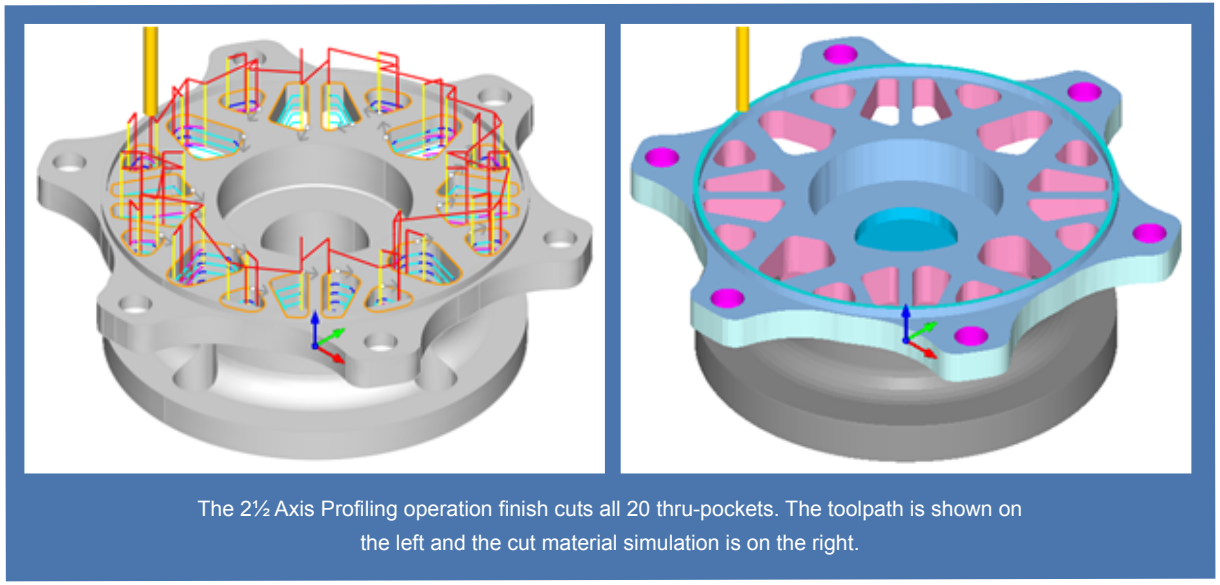
The last methods in the Machining Job will cut the pattern of thru-pockets located within the upper lip and around the flange. The first is a 2½ Axis Pocketing operation for roughing and the second is a 2½ Axis Profiling operation for finishing. Note that these pockets can be cut with just the Pocketing operation. However, we wanted to illustrate the combination of Pocketing and Profiling for roughing and finishing a pocket.

The 2½ Axis Pocketing operation rough cuts all 20 pockets. Cut parameters include 0.001" tolerance, 0.01" stock allowance (to be removed during the Profiling operation), offset cut pattern, climb cut direction, 25% stepover, and a cut depth of 0.4" divided into 4 cut levels. Pocketing entry is set to ramp and exit is set to linear. You can see the toolpath and the cut material simulation in the illustrations below.



The 2½ Axis Pocketing operation rough cuts all 20 thru-pockets. The toolpath is illustrated on the left and the resulting cut material simulation is on the right.

As we mentioned above, the 2½ Axis Profiling operation finish cuts all 20 pockets. Cut parameters include 0.001" tolerance, 0" stock allowance, mixed cut direction, and a cutting depth of 0.4" divided into 4 cut levels. Arc fitting is enabled with a fitting tolerance of 0.002" (dark blue motions). The operation includes a ramp entry and a straight retract exit motion. You can see the toolpath and the cut material simulation in the illustrations below.

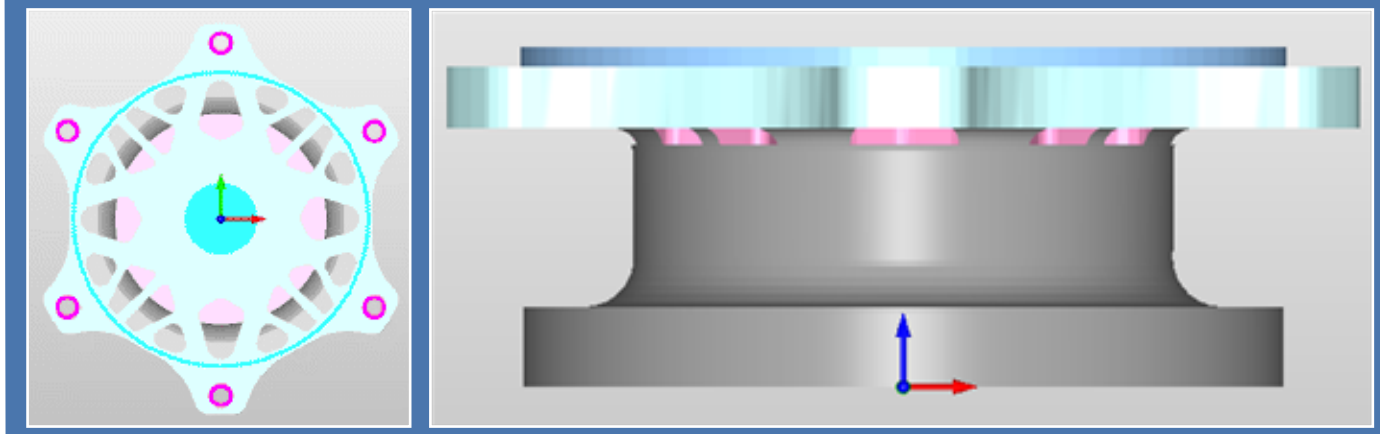


The 2½ Axis Profiling operation finish cuts all 20 thru-pockets. The toolpath is shown on the left and the cut material simulation is on the right.

This completes the top side for this part. The Setup1 as shown in the Machining Job tree above can be posted out in one g-code file (if you have an automatic tool changer) or posted out as separate g-code files, one operation at a time if you need to change tools manually for each operation.

The Top Side Completed In-Process Stock Model

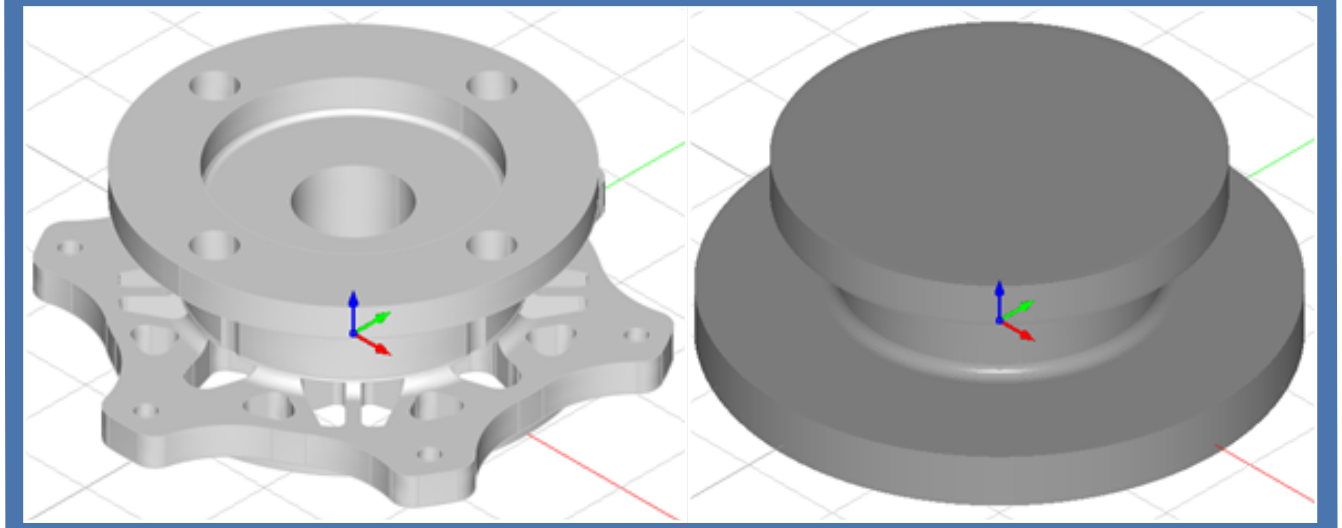
After simulating Setup 1 for the top side of the part, the in-process stock material simulation model will look like the illustrations shown below.



This completes the top side for this part. The completed cut material simulations are shown in the illustrations above.

Machining the Bottom Side

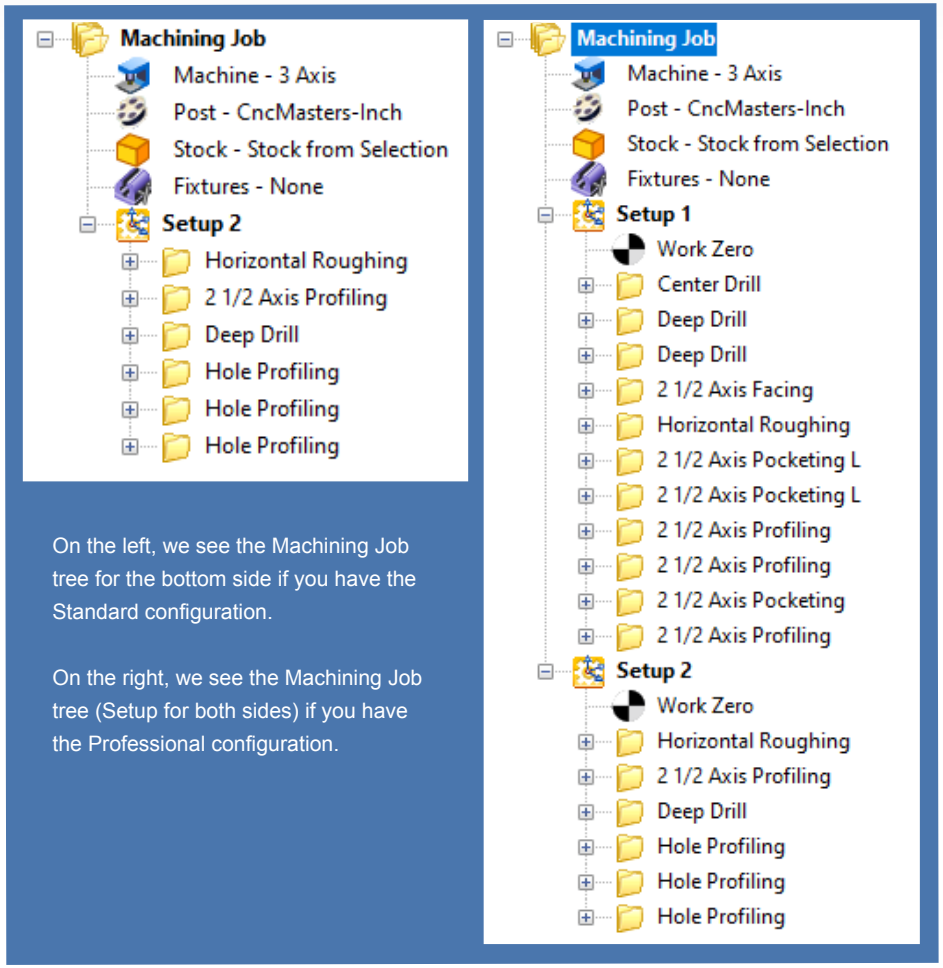
The configuration of MecSoft CAM you are running will determine how you will proceed with the bottom side. In the Standard configuration, you are limited to one setup orientation per CAD/CAM geometry file, so the bottom side is programmed using a second part file. In this second file, the part geometry is rotated so that the bottom side is up, facing the +Z axis of the WCS (World Coordinate System). The part and the stock orientation for the bottom side are shown in the illustrations below.



In the left image (above) we see the part geometry model oriented bottom side up for machining the bottom side of the part. On the right side, we see the turned stock blank. Note: In the Standard configuration, you cannot view the in-process stock model of the top side when programming the bottom side. You can in the professional configuration!

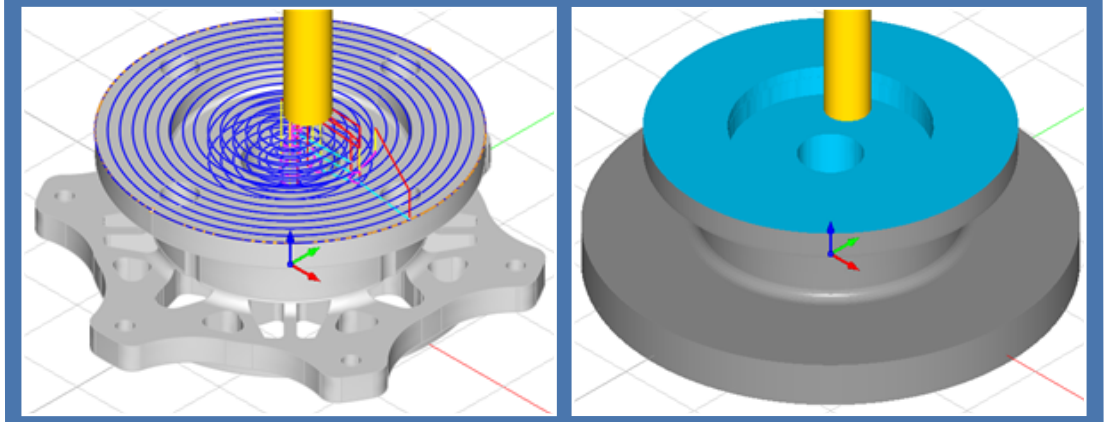
The Bottom Side Setup

This will be the first setup in the machining job, referred to as Setup 1 if you are running the Standard configuration. It will be the second setup, referred to as Setup 2 if you are running the Professional configuration. Both of the completed Machining Job trees are shown below. The Standard configuration is on the left and the Professional configuration is on the right.



3 Axis Horizontal Roughing

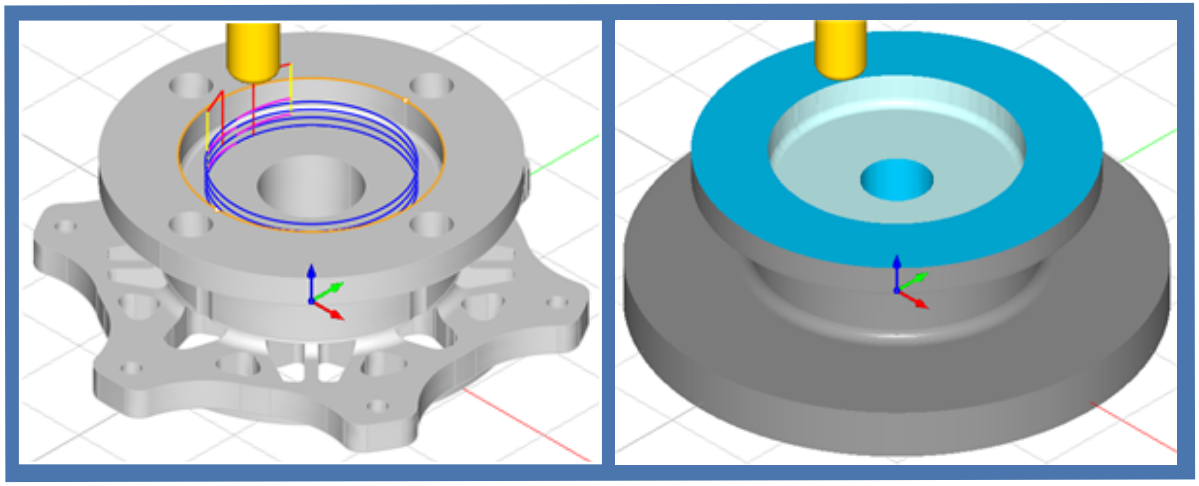
The first operation for the bottom side is 3 Axis Horizontal Roughing. It will remove most of the rough stock material needed. The cut levels tab of the operation allows you to control the maximum cut depth (referred to as the parameter "Bottom B" in the operation dialog (not shown). This is set to -1.06" since we do not need to go the full depth. Clear flats are enabled. Which will clear the stock down to the finished Z depth of the flat surfaces. Stepdown is set to 0.125" with depth-first cut level ordering. These parameters are reflected in the toolpath and cut material simulation illustrations below.



The first operation for the bottom side is 3 Axis Horizontal Roughing. It will remove most of the rough stock material. With clear flats enabled, we do not need additional finishing operations on the flat surfaces!

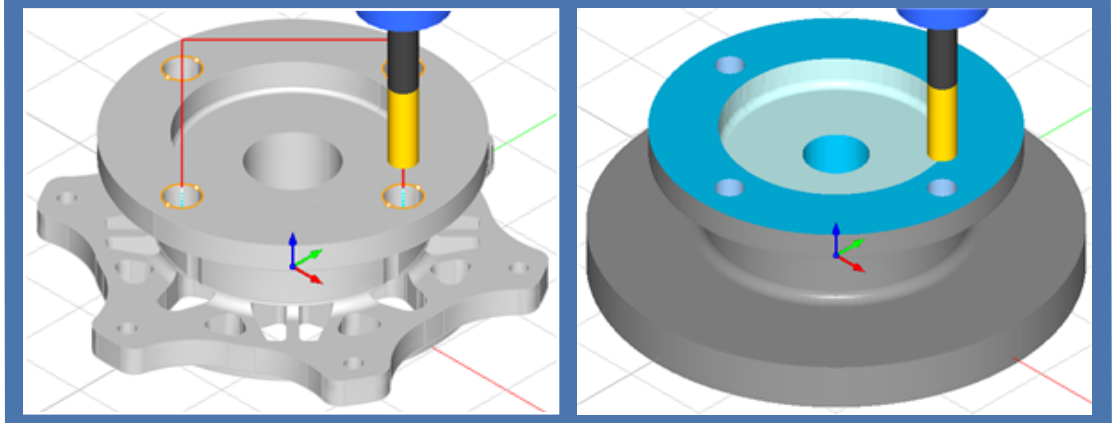
2½ Axis profiling

The next operation is 2½ Axis Profiling. It is used to finish the side wall of the circular pocket, again, using a corner radius mill. This is a 0.50” diameter cutter with a 0.05” corner radius. This is used to automatically machine the fillet located at the bottom of this pocket. Cut parameters include 0.001” tolerance, zero stock allowance, and cut width is zero (just one pass in X and Y) with a total cut depth of 0.35” at 0.10” depth per cut resulting in 4 cut levels. The toolpath and simulation are illustrated below.



Deep Drilling

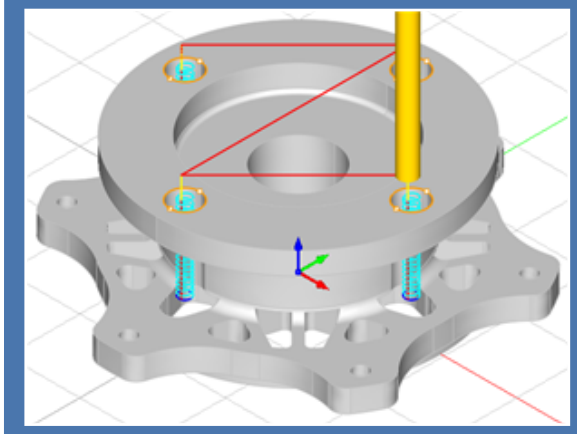
Next we rough cut the 4 holes on the flange using a 0.3125” diameter drill. The drill depth is set to 0.50” with a peck increment of 0.1 resulting in 5 increments. In Deep drilling, the tool retracts to an approach distance. We have it set to 0.1” above each peck increment. In this case, the tool is retracting 0.2” above the cut depth before pecking down again. This is different from Peck Drilling where the tool completely retracts to an approach distance above the hole. The toolpath and simulation are illustrated below.

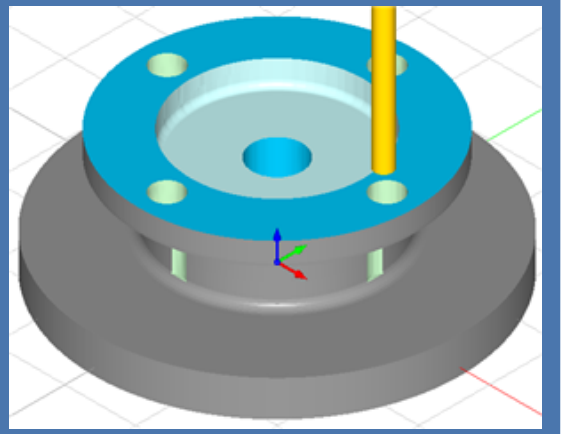


Here we see the toolpath and the simulation for Deep Drilling the 4 holes.

2½ Axis Hole Profiling (the Flange Holes)

The last three methods in the Machining Job are 2½ Axis Hole Profiling operations. In Hole profiling, the tool profile cuts each hole in a helical motion until it reaches the full depth of the cut. You can control the helical pitch which is the vertical distance between each full 360-degree helical motion. When the bottom of the hole is reached, a full circular motion is performed (dark blue), cleaning up the entire hole diameter. The toolpath and simulation are illustrated below.





In 2½ Axis Hole Profiling, you can control the helical pitch which is the vertical distance between each full 360-degree helical motion. When the bottom of the hole is reached, a full circular motion is performed (dark blue motion), cleaning up the entirety of the hole diameter. The toolpath and simulation are shown above.

2½ Axis Hole Profiling (rough and finish the center hole)

The accuracy of the thru-hole at the center of the part is critical in this case. We are using a combination of two 2½ Axis Hole Profiling operations to achieve this. The previous 3 Axis Horizontal Roughing operation left 0.015” of stock material around this 1.00” diameter hole. The first Hole Profiling operation cuts to a diameter of 0.950”.

The second Hole profiling operation will cut the hole to its full diameter. Two operations are used to minimize tool deflection during the final pass. Alternatively, you could use one profiling operation with two XY stepovers. However, using a second operation allows us to use a tool for the final cut! The hole depth is 0.75” and the helical pitch is set to a vertical depth of 0.06”. At the bottom of the hole, a full circular arc motion is added for increased accuracy. The tolerance is set to 0.0005” for the final operation.

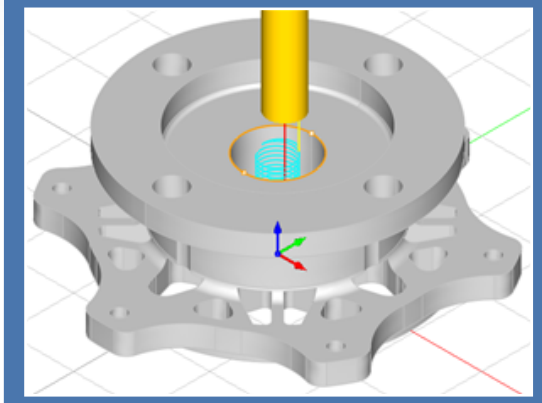
Note that this hole can also be machined using a Boring operation. However, since not everyone will have a boring bar tool, you can meet and exceed the same level of accuracy using 2½ Axis Hole profiling with a few tolerance adjustments. Refer to the note box below.

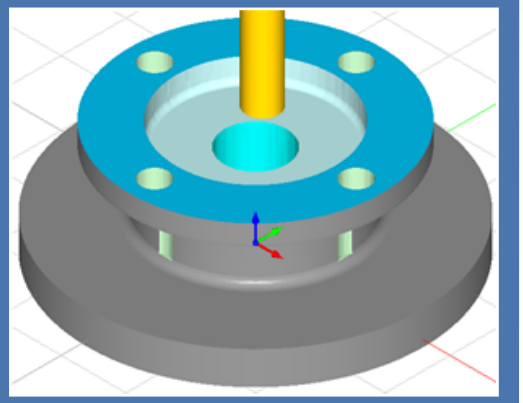
Achieve the exact level of accuracy required!

When close fits, in this case between a rod and a hole, are required, there are several parameter adjustments you can make in VisualCAD/CAM to achieve the exact level of accuracy required. You can refer to the following two resources:

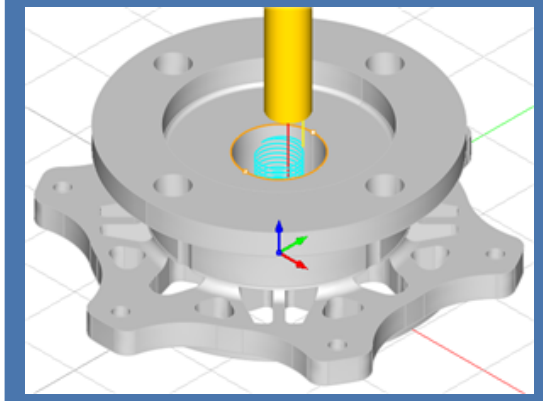
1. How to Increase Tool Path Accuracy
2. A case study for a real-world example

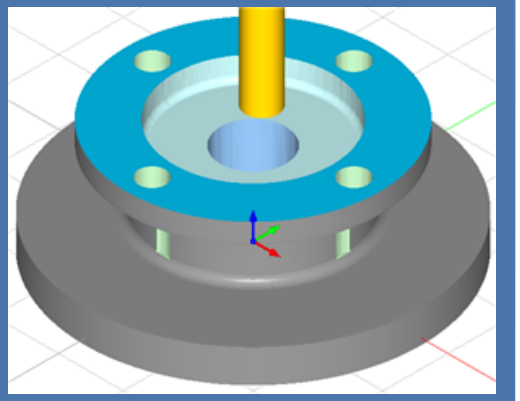
Note: These adjustments are the same for all MecSoft CAM plug-ins.



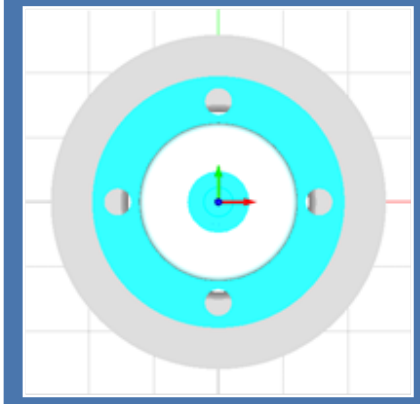


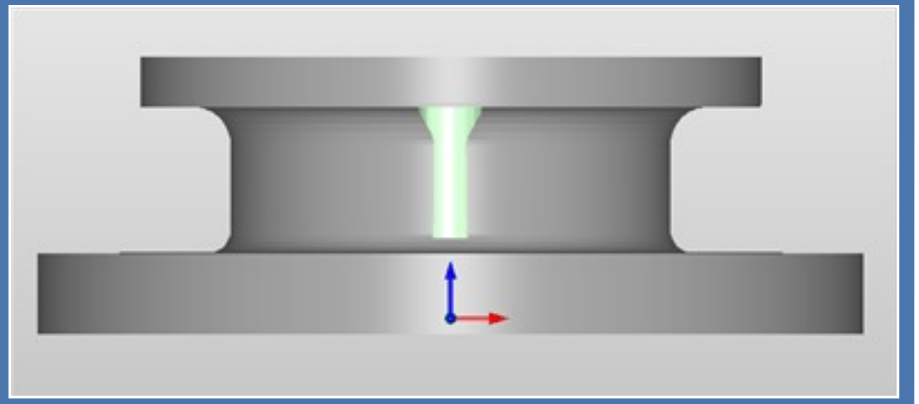
The first 2½ Axis Hole Profiling operation leaves 0.015" of stock around the 1" diameter hole.





The second 2½ Axis Hole Profiling operation (with a new tool) leaves zero stock remaining, finishing the 1" diameter hole to a tolerance 0.0005". Even tighter tolerances can be achieved if needed!



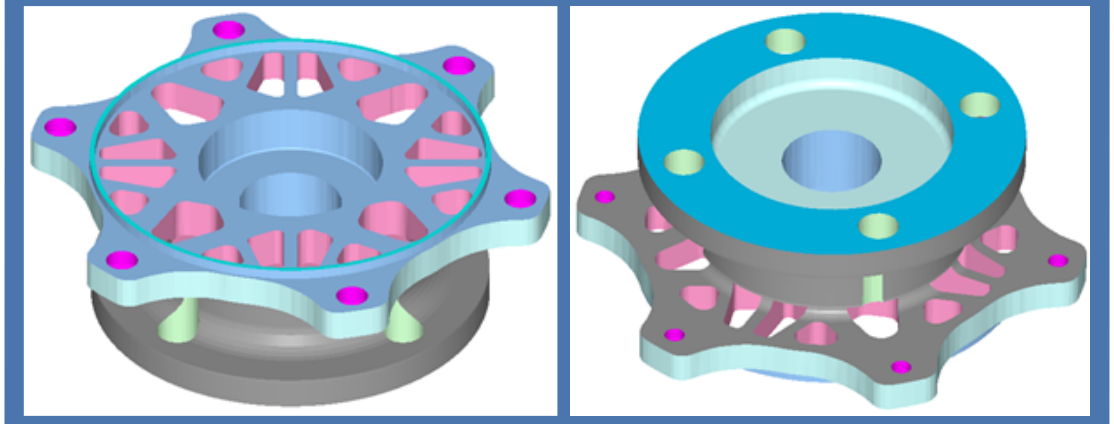


This completes the toolpath operations needed to machine the bottom side of the part.

The PRO Configuration

We mentioned that after the top side is machined, the in-process stock is flipped over on the CNC machine to machine the bottom side. We also mentioned that using the Professional configuration of MecSoft CAM will allow you to program both the top side and the bottom side in the same CAD/CAM part file, eliminating the need for extra files. It should also be noted here that in the Professional (PRO) configuration there is no need to re-orient the part model after Setup 1 (top side).

In PRO, Setup 2 is oriented and not part geometry. In our case, the X axis of Setup 2 is rotated 180 degrees so that the +Z axis of Setup 2 is facing the bottom side. This is all that needs to be done. The remaining toolpaths are programmed similarly to setup 1. In PRO, you can also simulate both setups at the same time, revealing the in-process stock model after each setup. The resulting simulations of both setups in the PRO configuration are shown below.



In PRO you can also simulate both setups at the same time, revealing the in-process stock model after each setup. The resulting simulations of both setups in the PRO configuration are shown above.

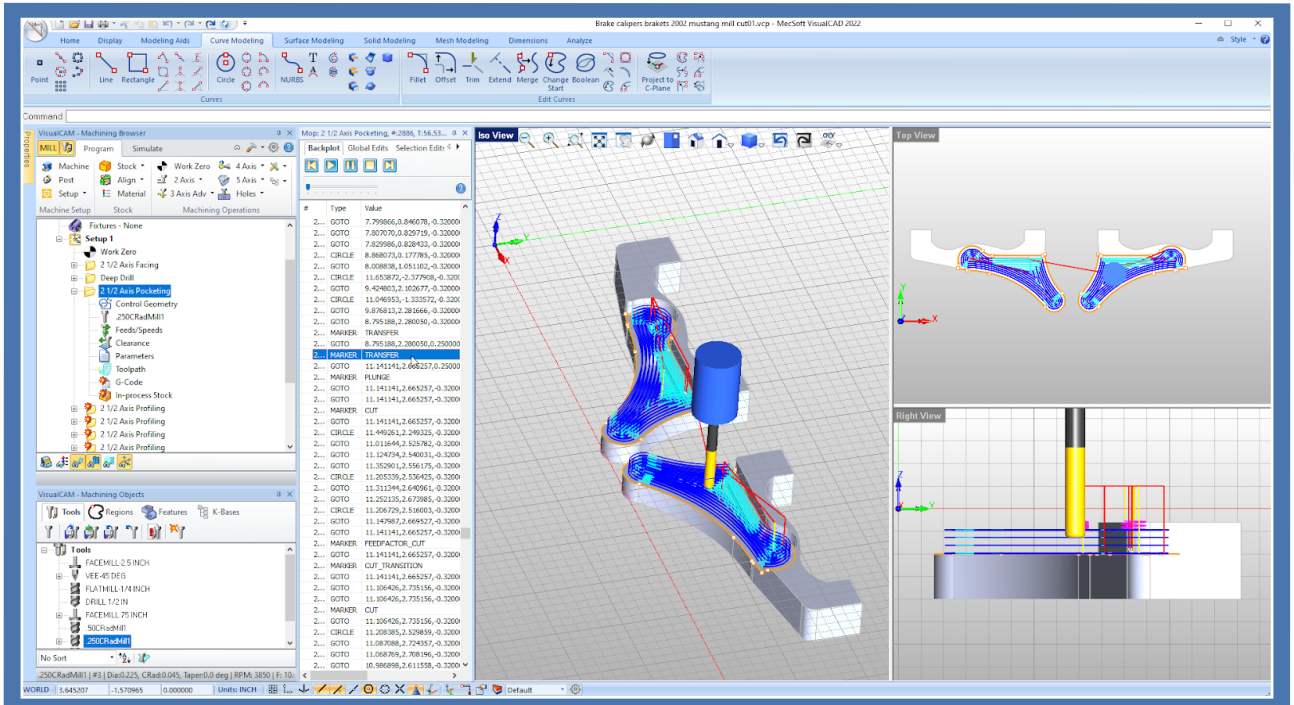
We hope you enjoyed reading about this fantastic project! We want to extend a very special thanks to KC Gager, Owner/ Operator, and his team at BRG Racing for allowing us to write about their cool work with VisualCAD/CAM!

Cool project KC!

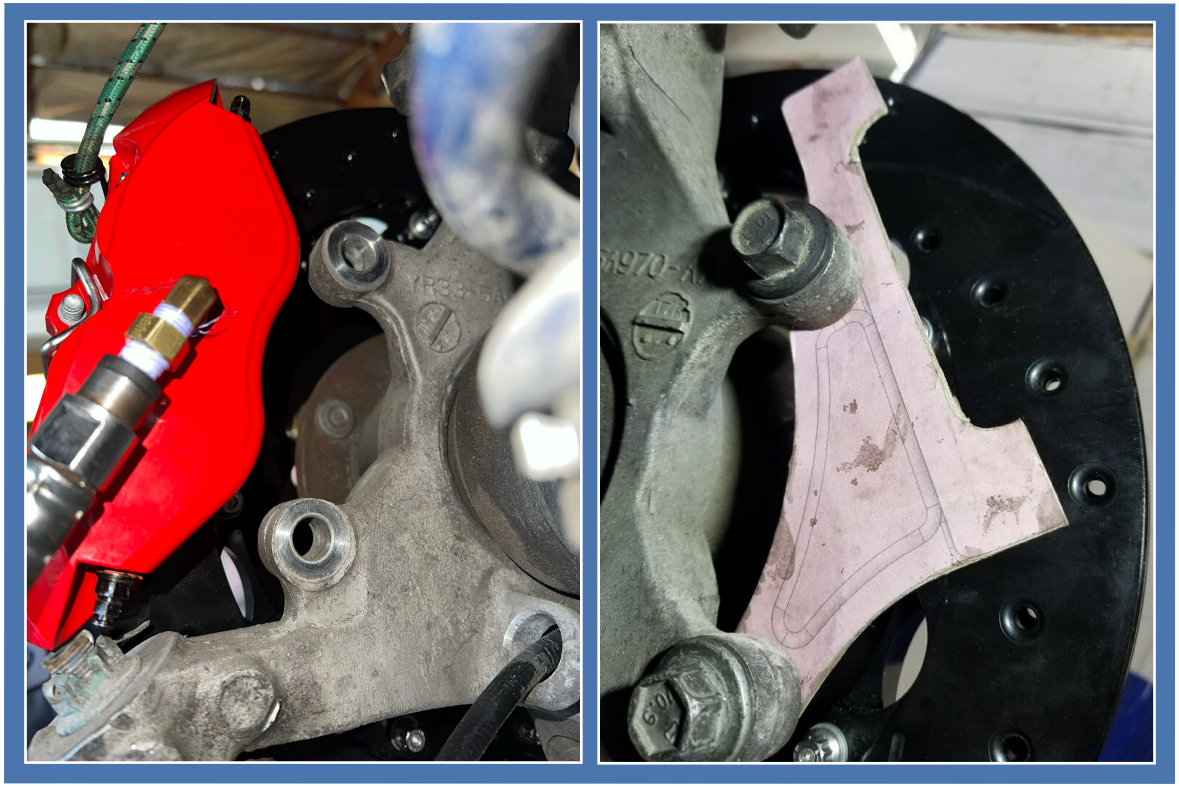
Thank you for allowing us to showcase your work!

More from BRG Racing

Here are some additional projects from BRG Racing with the use of VisualCAD/CAM.



Above we see custom right/left disk brake mounting brackets programmed in VisualCAD/CAM.
The actual finished parts are shown below.

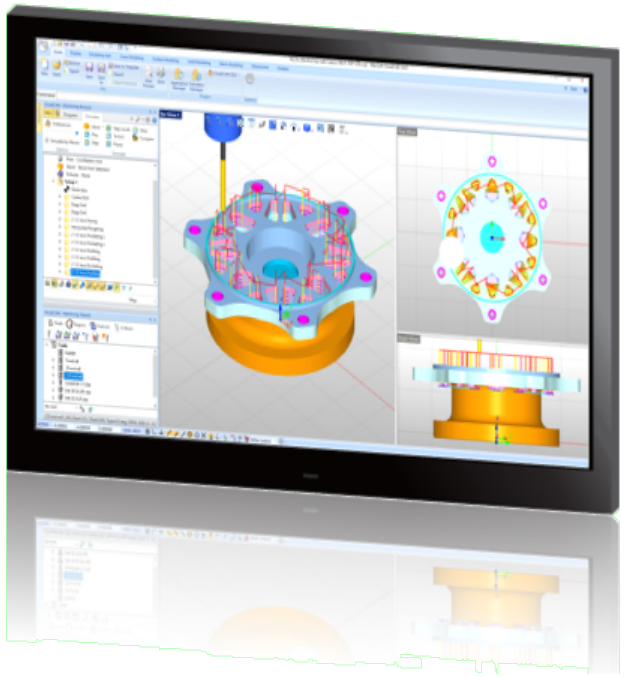


Want to see how VisualCAD/CAM can help you? Click Here to download a demo!

Follow BRG Racing Online:

More about VisualCAD/CAM

VisualCAD/CAM - MILL is available in 5 configurations (Express, Standard, Expert, Professional, and Premium). The parts shown here were programmed using both the Standard and the professional configurations. Here are some additional details about each of the available configurations. For the complete features list, visit the VisualCAD/CAM Product Page.



- **VisualCAD/CAM MILL Express**: This is a general-purpose program tailored for hobbyists, makers, and students. Ideal for getting started with CAM programming. Includes 2 & 3-axis machining methods. Includes ART & NEST modules as well!

- **VisualCAD/CAM MILL Standard**: This configuration includes everything that is in the Express configuration and additional 2-1/2 Axis, 3 Axis & Drilling machining methods.

- **VisualCAD/CAM MILL Expert**: Suitable for 4 Axis rotary machining. Includes the Standard configuration plus 4 Axis machining strategies, advanced cut material simulation, and tool holder collision detection.

- **VisualCAD/CAM MILL Professional**: Ideal for complex 3D machining. Includes the Standard and Expert configuration plus advanced 3 Axis machining strategies, 5 Axis indexed machining, machine tool simulation, graphical toolpath editing, and a host of other features.

- **VisualCAD/CAM MILL Premium**: Tailored for complex 3D machining with both 3 Axis and full 5 Axis methods. Includes the Standard, Expert, and Professional configurations plus 5 Axis simultaneous machining strategies.

For the complete features list, we invite you to visit the

VisualCAD/CAM Product Page

Try VisualCAD/CAM Today!

Powerful 2½ - 5 Axis machining capability on your desktop

More about MecSoft Corporation

MecSoft Corporation was founded in December of 1997 by a team of programmers from Unigraphics Solutions (now Siemens PLM) with the aim of providing affordable yet powerful CNC software solutions to the manufacturing industry. Our founding and operating principles are based upon the notion that our most important partner is our customer. This allows us to:

Develop quality products that meet or exceed customer needs and to
Deliver them at a price to performance value that is unbeatable in the industry
Provide excellent customer service and support. MecSoft Corporation is based in coastal Orange County, CA, south of Los Angeles.

VisualCAD/CAM®

MecSoft Corporation

Corporate Office Address:
34145, Pacific Coast Highway, #616
Dana Point, CA 92629-2808, USA

Phone: 949-654-8163
Support: 949-654-8163 x 2
Sales: 949-654-8163 x 201

For all other information call us at 949-654-8163
To contact us using our web form, please click here.

Follow MecSoft Corporation Online: