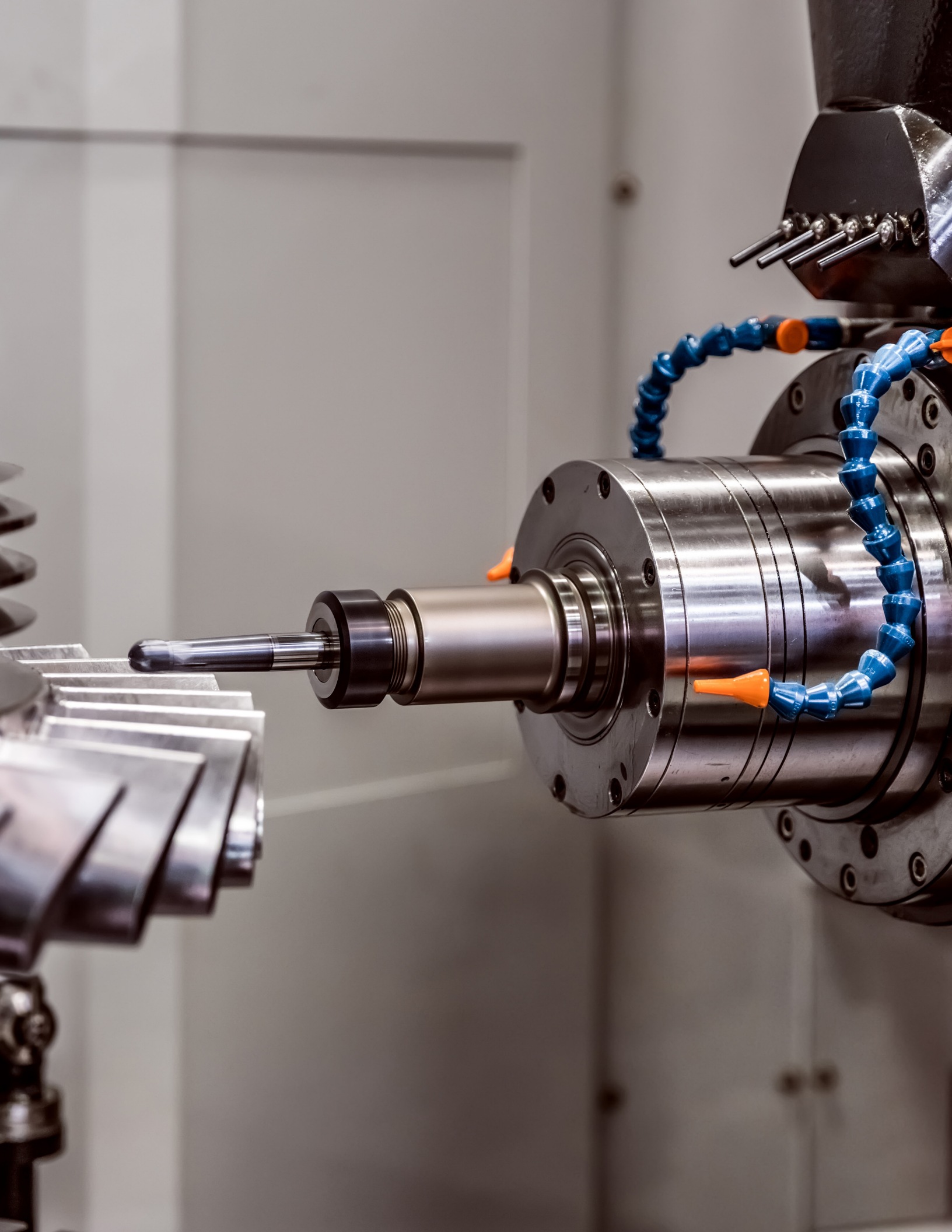# BRIDGERLAND MACHINING TROUBLESHOOTING GUIDEBOOK

National Science Foundation

BRIDGERLAND TECHNICAL COLLEGE

BECOMING AN EXPERT MACHINIST

Your capacity to be an expert machinist depends on your ability to produce parts efficiently and accurately. Throughout your career as a machinist, you will encounter problems that will require you to troubleshoot. The goal of this troubleshooting guidebook is to assist you in transforming yourself from a novice machinist into a more experienced machinist by learning more readily how to resolve entry-level machining problems.

Due to years of experience, expert machinists have the ability to troubleshoot problems efficiently. Over time expert machinists build decision trees in their heads that help them identify and solve problems. It takes years of experience as a machinist to develop these decision trees. Our machinists at Bridgerland Technical College (BTECH) have developed a troubleshooting flow chart and nine troubleshooting guides to assist you in developing your troubleshooting skills.

By internalizing these troubleshooting guides, you will develop the ability to more efficiently and effectively resolve entry-level machining problems. This will help you side-step years of troubleshooting through trial and error and assist you in more quickly becoming a valuable and sought-after machinist in the industry!

⚠️ **Problem Occurs!**

Begin Defining the Problem:

SETUP & TOOLING

MECHANICAL & ELECTRICAL

Attempt the most likely solution. If the problem continues, try the next potential solution.

G01 PROGRAMMING

FEEDS & SPEEDS

PROBLEM SOLVED!

Great work! Your ability to accurately diagnose problems is a useful skill in industry.

STILL UNRESOLVED?

Prepare to discuss the problem with your instructor by reviewing what solutions you have attempted and their accompanying results.

How to Use the Troubleshooting Flow Chart & Troubleshooting Guides

TROUBLESHOOTING FLOW CHART

The Troubleshooting Flow Chart (pictured on the left) was created to help you identify the category in which your problem falls under. There are four main troubleshooting categories in machining. These include:

MECHANICAL & ELECTRICAL

SETUP & TOOLING

FEEDS & SPEEDS

G01 **PROGRAMMING**

When you can identify the problem category you can then begin to narrow down the potential solutions.

For example, if you notice your part has a bad surface finish, this problem will likely fall under Feeds & Speeds.

Whereas if your machine won't turn on, this problem will likely fall under Mechanical & Electrical.

When a problem arises, pay close attention to your senses. What are you seeing, smelling, feeling, or hearing? Compare and contrast this experience with previous experiences you have had. All of these details will help you solve your problem! Using the evidence collected by your senses begin with the most likely solution. If that first solution is unsuccessfully continue to the next most likely solution until your problem is solved.

In addition to the Troubleshooting Flow Chart which guides the overall process of troubleshooting, some problems will require some additional troubleshooting resources. BTECH has also created Troubleshooting Guides targeted towards particular troubleshooting categories and machines. These Troubleshooting Guides will assist you when troubleshooting entry-level machining problems.

TROUBLESHOOTING GUIDES

Our machinists made sure to provide you with nine additional troubleshooting guides to help you dive deeper into troubleshooting. These charts are organized as Manual Operations, Programming Operations, and CNC Operations (See descriptions on the right).

These troubleshooting guides present you with a problem/issue in the left-hand column and accompanying troubleshooting steps in the right-hand column. These problems/issues are ordered from most common at the top of each guide to least common at the bottom of each guide. You will also notice that the icons and colors on the troubleshooting flow chart correspond to the troubleshooting guides, giving you direction on which category the problem falls under.

For example, if your machine won't turn on, the troubleshooting guide will provide you with the following troubleshooting steps.

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
Machine won't turn on	Did you turn on the main breaker?
	Are you pressing the cycle start instead of power on?

STILL CAN'T SOLVE YOUR PROBLEM?

If all of your attempts to resolve the problem on your own have failed, the next step is to reach out to your instructor for help. However, now that you have attempted the troubleshooting process you are better prepared to receive help from your instructor. Each troubleshooting attempt has given supplied you with more valuable information. Be prepared to discuss what solutions you attempted and their accompanying results with your instructor.

By following the Troubleshooting Flow Chart and working through the Troubleshooting Guides, you will begin to develop the decision trees of an expert machinist. Machinists who are able to troubleshoot and solve problems are incredibly valuable in the industry. These troubleshooting skills will be essential when you enter the workplace.

MANUAL OPERATIONS

In manual operations, various challenges arise. Troubleshoot **SETUP & TOOLING** 🔧 for proper alignment. Address issues with **FEEDS & SPEEDS** to prevent surface imperfections and tool breakage, and diagnose and resolve **MECHANICAL & ELECTRICAL** issues to ensure the smooth function of the manual machine. Overcoming these challenges is essential for achieving accurate and efficient machining in manual operations.

PROGRAMMING OPERATIONS

In the programming operations section, focus on solutions for **PROGRAMMING** errors that can occur. These errors include syntax errors, such as misspelled commands or incorrect parameter entry, as well as logic errors that result from mistakes in calculations, tool paths, or conditional statements. Through effective troubleshooting techniques and careful examination of the programming code, operators can overcome programming challenges and achieve smooth and efficient machining processes.

CNC OPERATIONS

In CNC operations, you'll encounter various challenges. Troubleshoot **PROGRAMMING** issues to ensure a smooth process. Optimize **SETUP & TOOLING** 🔧 for accurate alignment. Address problems with **FEEDS & SPEEDS** to prevent tool wear or vibration, and diagnose and resolve **MECHANICAL & ELECTRICAL** issues to maintain the proper function of the machine's components.

MANUAL OPERATIONS

MANUAL LATHE TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
Lathe spindle won't turn on	Did you turn on the main breaker? Is the brake pedal engaged? Are the pedal springs working? If the start lever is engaged, did you reset to neutral and restart the machine?
Part too big or too small	Did you have a bad tool touch-off? Retouch-off your tool and check for discrepancies. Is your digital readout set to the wrong mode (radius or diameter)? Make sure it is set correctly. Are you using the dials correctly? Reference back to instructions on the machine. Are you using the micrometer or caliper correctly?
Bad surface finish or chatter	Is your tool height too high or low? Tool tips need to be aligned with center. Is your tool sharp? If your tool is dull, index insert or reshape. Is your feed rate too fast or too slow? Adjust RPM.
Clearance/hitting the chuck or tool post	Change the setup. Hang part out further if operation permits. Adjust tool hangout if appropriate.
Carriage/cross slide is feeding in the wrong direction	Is the knob on the headstock set correctly? If the knob is incorrect, change direction.
Bad threads	Do you have the correct setting on the machine headstock? Correct tool setup? Correct compound setup? Did you engage the half nut lever on the wrong timing marks?
Chips are wrong shape/color	Is your spindle speed too fast? Change RPM to a lower speed. Adjust RPM and/or feed rate. Is the depth of cut too much? Reduce the depth of cut.
Tailstock moving during operation	Is your tailstock clamp tight enough? Tighten the nut on the bottom of the tailstock.

KEY

MECHANICAL & ELECTRICAL SETUP & TOOLING G01 PROGRAMMING FEEDS & SPEEDS

MANUAL MILL TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
Mill won't turn on	Did you turn on the main breaker? Is the E-stop engaged? Reset it.
Part too big or too small	Are you using the dials and measuring tools correctly? Is your part tapered? Check setup in vise and correct as needed. Is something moving during the operation? Check setup/clamping.
Bad surface finish/chatter	Adjust RPM and/or feed rate. Is your tool sharp? If your tool is dull, index insert or reshape. Check setup for rigidity. Is enough stock being supported or is the tool hanging out too far? Are you using the wrong cutting strategy (Climb/Conventional milling)? Is the cutter rotating in the wrong direction? Reverse cutter direction.
Collet won't go in/out of the spindle	Did you check collet alignment with key? Is the quill returned all the way to the top of travel so the drawbar will engage?
Head makes grinding noise when it's on	Is it in high or low gear and fully engaged before turning on the power to the motor?
Table/saddle/knee/quill hard to move	Did you check the appropriate locking handles for the axis that you are moving?
Part is tapered	Check mill head and vise alignment. Is your part seated correctly? Has it been taped down or is it sitting on any debris?
Drilled hole is too big	Did you center drill first?
E2/E3 error on DRO	Is the DRO scale dirty or oily? Clean it.

KEY

MECHANICAL & ELECTRICAL   SETUP & TOOLING   G01 PROGRAMMING   FEEDS & SPEEDS

PROGRAMMING OPERATIONS

MILL 2D TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
In toolpath geometry chaining, a full chain will not form.	**G01** **Is the geometry duplicated/overlapping?** • If so, delete any duplicate/overlapping geometry. • If any duplicated or overlapping geometry is needed for other operations and should not be deleted, use different geometry colors and/or levels to organize geometry so as to prevent mis-chaining issues. Otherwise, delete duplicate geometry. • If no duplicates/overlapping geometry exist, move to next step. **G01** **Is the chained geometry fully connected?** • Outside of toolpath chaining, using the Shift key, click on the chain in question to see if all entities of the geometry to be chained are selected as desired. • If the geometry is not all selected, zoom in on all entity joints to isolate the area(s) where a connection is needed and correct geometry as needed to assure that they are fully connected. **G01** If the geometry is able to fully chain, but will not chain properly in toolpath creation chaining, make sure that "partial chain", "window/area," "point," or "single entity" chain options are not enabled and "full chain" mode is enabled instead. • Make sure that the "CPlane" option is enabled rather than "3D".
Geometry created is not connected in wireframe design.	**G01** **Is 2D enabled in CPlane bottom menu bar?** • If your construction plane and the resultant geometry needs to be at Z0.0, you can choose to leave 2D enabled, but you must verify the ZPlane value is at Z0.0 otherwise, entities will be placed on that ZPlane value tied to 2D creation. • If your construction plane and the resultant geometry needs to be able to connect across different planes, 3D must be enabled. • If your construction plane needs to be at a different Z value, set to correct value and enable 2D for geometry creation on that plane.
Can't create arcs and lines tangent to each other.	**G01** Make sure tangency feature is enabled in AutoCursor options (Found under File, Configuration, Selection). **G01** Use the Shift key when trying to connect a line tangent to an arc. Make sure cursor highlights arc entity to be tangent to, but cursor does not actually touch arc.
Geometry created is hard to see or cannot be seen at all.	**G01** Change color of geometry to a color that provides better contrast with your screen background. **G01** Make sure the Active Level Always Visible option is enabled in the Settings icon which can be found in the Levels Tab.
When creating lines, arcs, points, circles, rectangles, etc., geometry tries to snap to an undesired point on the screen.	**G01** Go to File, Configuration, Grid, and change the radio button under Snap To from Always to Near.

KEY

MECHANICAL & ELECTRICAL	SETUP & TOOLING	PROGRAMMING	FEEDS & SPEEDS

LATHE TOOLPATH TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
Lathe Rough Toolpath will not machine into undercut geometry.	**G01** Ensure that your geometry is chained properly to include all undercut geometry. **G01** In Rough Parameters of toolpath parameters, change the Plunge Parameters to 2nd option to enable both X and Z axis undercuts, to the 3rd option for X only undercuts, or to the 4th option for Z only undercuts.
Lathe Rough Toolpath cuts large depth on first cut not consistent with programmed Depth of Cut.	**G01** Choose Use Stock for Outer Boundary or Remaining Stock in the Stock Recognition drop down options found in the bottom left corner of Rough Parameters in the Toolpath Parameters.
Tool Collision with stock in Lathe Rough Toolpath.	**G01** Go to Rough Parameters and change the Lead In/Out settings via the Entry Vector to a suitable angle that will allow the tool to not collide with the stock. This angle will vary but typically some increment of 45 will fix this.
Lathe Stock flip puts part and/or jaws in incorrect location.	**G01** Make sure the Transferred Position is placing the Z value at the new desired Z0.0 point of the part. This will not necessarily be a 0.0 value entered, but will instead be more likely the length of the completed part. **G01** Modify the Final Position Z value by choosing the furthest point you want the jaws to be in contact with. This position will vary per part, but will likely be less than or equal to the Z0.0 point, (i.e., a negative number here will place the jaws behind the part after the stock flip to allow more work on the flipped end of the part, such as for facing or left-hand type features not able to be cut from first op side).
Lathe Rough or Finish toolpath for ID work has tool collision with stock.	**G01** Make sure that you have a boring tool that is the appropriate size for the feature you are trying to create. A good rule of thumb is to only use as big of a tool as you can for the bored feature. **G01** Check the Lead In/Out settings for angles that will allow the tool to lead in and out safely without a collision. Modify as necessary. **G01** In some boring finish cases, it may be necessary to modify the geometry to shorten the path of the tool so as to prevent it from colliding with the other side of the hole.

KEY

MECHANICAL & ELECTRICAL   SETUP & TOOLING   **G01** PROGRAMMING   FEEDS & SPEEDS

2D TOOLPATH TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
Toolpath List does not display on left side of screen.	**G01** Use Alt-O to bring up Toolpath pane, or navigate Top Ribbon Menu to View Tab, and enable Toolpaths.
No machine group in Toolpath List.	**G01** Select a machine by navigating to Machine Tab in Top Ribbon Menu and select desired machine from list of drop down menus under Mill, Lathe, Wire, etc...
Stock Setup does not display at all or display correctly on screen.	**G01** Make sure your stock is defined properly under the Setup tab of the Machine Properties in the Machine Group, and Display checkbox enabled, or Stock Display turned on under the Toolpath Tab of the Top Ribbon Menu. **G01** Make sure the Z value used in the Stock Origin box of Setup page is in the correct location, as well as the correct point used in the Rectangular graphic. If this point is not located correctly, your stock display will not be aligned to your geometry properly.
Toolpath is not in the correct plane.	**G01** Make sure that you have enabled the appropriate WCS view and CPlanes/TPlanes for your toolpath by using the Planes tab and check boxes across row of desired plane to align all WCS associations properly. • If the toolplath still is in the incorrect plane, go to toolpath parameters and the planes tab to reselect the appropriate plane needed to regenerate the toolpath.
Drill toolpath is drilling a corner of part instead of hole geometry.	**G01** Check defining geometry entered in toolpath for any corner points or midpoints of lines, etc.. Drill toolpaths can also snap to quadrants or arc endpoints, so beware they are defined to arc centers or defined points.
Drill toolpath depth is too deep/too shallow.	**G01** Check Linking Parameters to verify depths from Clearance at the top to Depth at the bottom are in a descending order and that the appropriate depth is set in Absolute or, 0.0 for Incremental based on the geometry selected. **G01** In the Depth box in the Linking Parameters, check the Calculator button to verify thesettings entered in that will be added to or overwrite the Depth value. **G01** Check to make sure you are aware of any Break Through options that are enabled or disabled. Add or remove as necessary.
Drill Toolpath starts from too high.	**G01** Check the Retract and Top of stock options in the Linking Parameters page of the Drill toolpath parameters.
2D Contour toolpath is going the wrong way or on the wrong side of the part.	**G01** Check geometry within toolpath for proper direction. Arrows should be placed in a Climb Milling direction. • If going the wrong direction, right click the entity in the toolpaths selection box and click "Change Direction" option. • If the wrong side of the part is being milled, right click the entity in the toolpaths selection box and click "Change Side" option. Beware this may also change the direction of rotation around the chain, and may require clicking "Change Direction" as well.

KEY

MECHANICAL & ELECTRICAL   SETUP & TOOLING   **G01** PROGRAMMING   FEEDS & SPEEDS

2D TOOLPATH TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
2D Contour isn't going to the right depth in toolpath.	G01 Check Linking Parameters to verify depths from Clearance at the top to Depth at the bottom are in a descending order and that the appropriate depth is set in Absolute or, 0.0 for Incremental based on the geometry selected. It is important to understand how and when Incremental can or should be used. G01 Check to make sure you are aware of any Break Through options that are enabled or disabled. Add or remove as necessary. G01 Check the Cut Parameters for any stock to leave on floor options. This is another place that is commonly overlooked that will affect the depth of a 2D Contour toolpath. G01 Make sure Depth Cuts are being used (or not used) appropriately.
2D Contour has no Lead In/Lead Out even though it is enabled.	G01 Check starting location of chain on-selected geometry and use dynamic move to adjust start point to an entity or point where the lead in/lead out arcs/lines have enough room to engage. G01 Decrease length or percentage of lead in/lead out lines and/or arcs used if the starting location can't be adjusted or does not matter where it is moved to (such as interpolating the inside of a hole).
Pocket toolpath does not ramp or helix into depth as it should.	G01 Open toolpath Parameters, go to Cut Parameters, Roughing, and Entry Motion to check Ramp or Helix radio buttons instead of Off. • If Ramp or Helix are enabled put pocket toolpath still plunges, check to make sure settings entered are appropriate for tool type and material. Mininum Length or Maximum Length may need to be adjusted down to allow the Ramp entry to succeed. Maximum Radius may need to be adjusted smaller to allow Helix entry to succeed.
2D Dynamic Mill toolpath starts inside the part when it should be starting outside the part, or vice versa.	G01 In the Chain Options dialog box, Machining Region Strategy needs to use Stay Inside when cutting pocket type geometry, and needs Stay Outside when machining features that are external, such as a boss. Change radio button to select appropriate strategy for your feature(s).
2D Dynamic Mill toolpath does not cut in the right place(s).	G01 Check geometry used to define the toolpath in the Chain Options dialog box. Machining Regions are used to select geometry used to define the stock. If no geometry is chained, the toolpath will default to using the Stock Setup definitions. Avoidance Regions are used to define the geometry that the tool will cut to, but not beyond.

KEY

MECHANICAL & ELECTRICAL   SETUP & TOOLING   G01 PROGRAMMING   FEEDS & SPEEDS

3D GEOMETRY/SOLID TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
Can't create Revolved feature	**G01 Is axis of rotation selected or incorrect?** • In Solid Revolve feature, make sure you have selected the correct line to serve as the axis of rotation of the revolved solid body. **G01 Is the geometry chained properly?** • Check for any duplicate/overlapping geometry, or broken/disconnected chains. • Correct any geometry issues.
Extrude Solid Body will not extrude body/cut body correctly.	**G01 Is selected geometry chained properly?** • Correct any duplicate/overlapping geometry, or broken/disconnected chains. **G01 Is the direction or distance correct?** • Change direction or correct the distance to be extruded to. **G01 Is the chain fully closed?** • Make sure the chain is complete, otherwise the geometry will attempt to create a thin-wall feature instead of full body feature.
Loft feature has inconsistent number of syncs.	**G01 Do all profiles have an equal number of total entities?** • Trim/break geometry of any profile that has a lesser amount of entities to match the amount of entities of the other profiles. They must be equal. **G01** Check for any duplicate/overlapping geometry, or broken/disconnected chains. Correct as needed to create full chain.
Loft Feature fails to create loft.	**G01** Check to make sure all chained profiles will not create an overlapping surface issue due to a geometric condition. This may be done by using the trim/break command tool to modify the geometry as needed to allow the loft to form by bending the geometry.
Sweep Feature will not form or looks jagged/is not correct.	**G01** Check for any duplicate/overlapping geometry, or broken/disconnected chains. • Correct as needed to create full chains required for sweep profiles. **G01 Are all entities smoothed out and tangent as needed?** • Check geometry for proper creation by using Analyze tools to verify. • Correct geometry as necessary.
Cut Extrude only cuts a portion of the body of the part, even with Through All enabled.	**G01** Did Boolean correctly join multiple bodies to form one body? • Use Solid Tab to verify features created were properly added/subtracted via the Boolean feature in the Solid Tree.

KEY

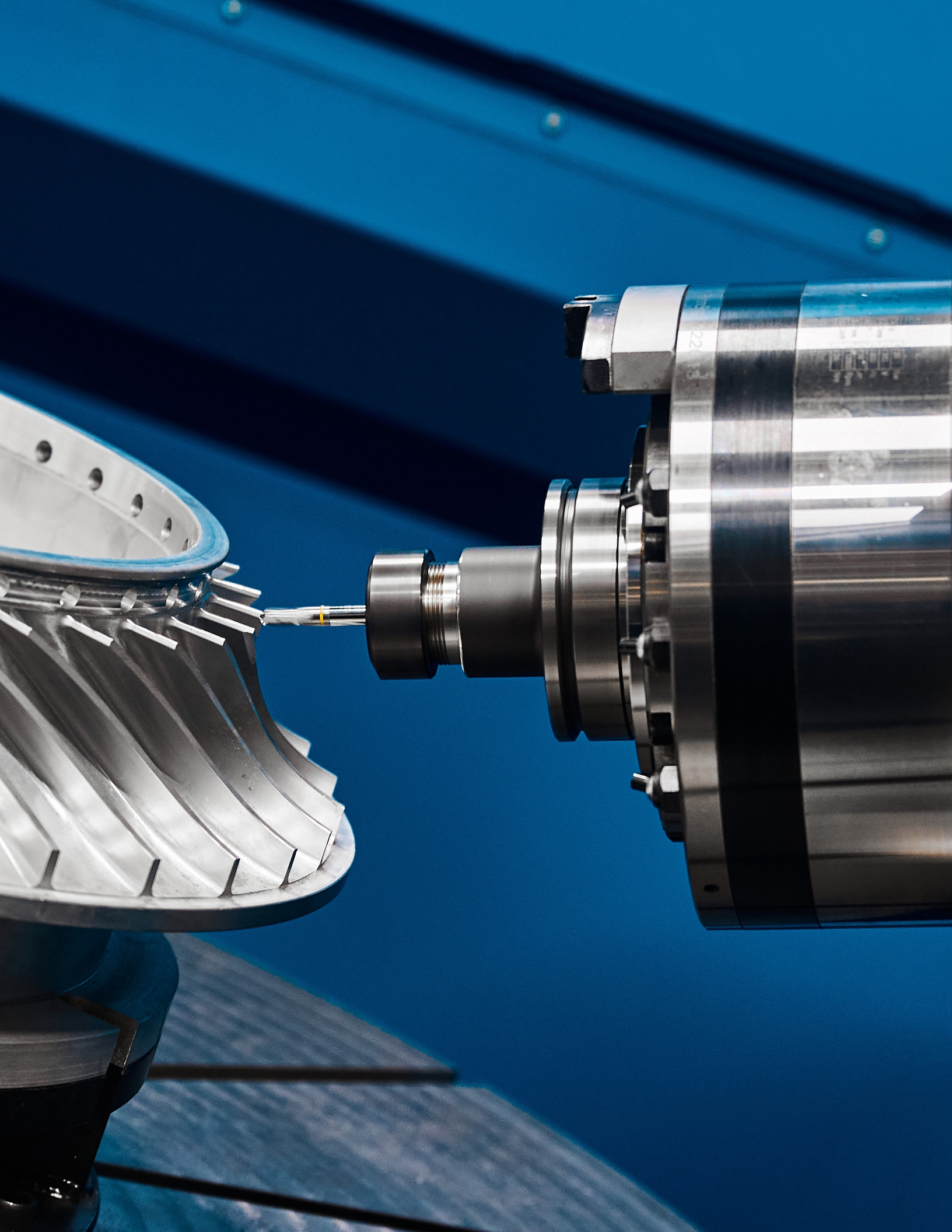| MECHANICAL & ELECTRICAL	SETUP & TOOLING	G01 PROGRAMMING	FEEDS & SPEEDS

3D TOOLPATH TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
3D OptiRough cuts outside of the surfaces it is supposed to be in.	**G01** **Is containment boundary selected properly?** • Add or correct the containment Boundary Chain in the Toolpath Control tab within Toolpath Parameters. **G01** **Are there any Drive Surfaces selected beyond the surfaces that are to be cut?** • Check selected entities/surfaces selected and deselect any that are outside of the desired area to be machined. • Add any critical avoidance surfaces to the Avoidance Surfaces selections in the Model Geometry tab within the Toolpath Parameters.
High Speed 3D Toolpaths leave too much material in Verify.	**G01** Adjust parameters for Wall Stock and Floor Stock in the Machining Geometry column of the Model Geometry tab of the Toolpath Parameters. **G01** This may also need to be fixed, but controls more in the avoidance geometry, adjust parameters for Wall Stock and Floor Stock in the Avoidance Geometry column of the Model Geometry tab of the Toolpath Parameters.
High Speed 3D Toolpaths do not machine far enough to the containment boundary chains that were selected, or machine past the boundary.	**G01** Check Strategy (From Outside/Stay Inside) settings and Compensate to (Inside/Center/Outside) settings entered in the Toolpath Control tab of the Toolpath Parameters. • Adjust as necessary for desired outcome of toolpath.
3D OptiRough toolpath creates excessively long program and calculated runtime is not efficient.	**G01** **Are your feeds and speeds optimized for your material based on your tool?** • Check with tooling manufacturer for appropriate SFM/Chipload of tooling for your material. **G01** **In Cut Parameters, are your Passes optimized in Stepover, Stepdown, and Stepup?** • Check with tooling manufacturer's recommended cut parameters such as with the "Machining Advisor Pro" app from Harvey Performance for Helical and Harvey Tool brand tools. **G01** **Are your Arc Filter/Tolerance settings set up for more arc filtering so as to create less program code?** • Modify by checking the Line/Arc Filtering Settings box and drag the left slide above away from the Cut Tolerance slider to 5%. This will decrease the number of small G01 lines created to form an arc and will opt for using more G02/G03 arcs.
3D High Speed Equal Scallop toolpath is jagged or noisy, not smooth.	**G01** Enable Projected Boundary Smoothing Tolerance in the Containment Boundary section under Toolpath Control of the toolpath parameters. Add a larger number to the box to apply a smoothing tolerance.
Surface finish is too rough/course in Verify.	**G01** Within Toolpath Parameters, modify the Stepover/Stepdown/Stepup/Scallop Height to a smaller increment to provide a better quality finish.

KEY

MECHANICAL & ELECTRICAL	SETUP & TOOLING	PROGRAMMING	FEEDS & SPEEDS

CNC OPERATIONS

CNC LATHE TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
Machine won't turn on	🔧 Did you turn on the main breaker? 🔧 Are you pressing the cycle start instead of power on?
Program won't load	G01 Check (file type, percent signs (top and bottom of the program), and correct O number for program number.) G01 Is your program in a subfolder on your USB drive?
Error - bad code	G01 Make sure that your code is G01 (number 0) not GO1 (letter O). G01 Check other number and letter assignments in each code to make sure they are correct.
Part is running off center	🔩 Is the chuck adjusted in the correct point of travel? 🔩 Are the jaws tight enough on the chuck?
Spindle is too fast or slow	G01 Do you have the correct speed on the headstock? G01 Do you understand G96, G97, and G50 and what they do controlling the spindle speed?
Bad surface finish or chatter	🔩 Is your height too high or low? Did you adjust accordingly? 🔩 Is your tool sharp? If your tool is dull, index insert or reshape. 🔽 Is your feed rate too fast or too slow? Adjust RPM.
Chips are wrong shape/color	🔽 Is your spindle speed too fast? Change RPM to a lower speed. 🔽 Adjust RPM and/or feed rate. G01 Is the DOC too much? Reduce the depth of the cut.
Part too big or too small	🔩 Did you have a bad tool touch-off? Retouch-off your tool and check for discrepancies. 🔩 Are you using the correct offset? 🔩 Is the tool nose radius set correctly? 🔩 Is the tool tip direction set correctly? G01 Is the cutter compensation set to the correct side (left or right?)
Z length of any features or the overall part wrong	🔩 Did you have a bad tool touch-off? Retouch-off your tool and check for discrepancies. 🔩 Are you using the correct offset? 🔩 Is your part clamped correctly? Do you have enough clamping pressure?

KEY

🔧 MECHANICAL & ELECTRICAL   🔩 SETUP & TOOLING   G01 PROGRAMMING   🔽 FEEDS & SPEEDS

CNC MILL TROUBLESHOOTING

PROBLEM/ISSUE	TROUBLESHOOTING STEPS
Machine won't turn on	🔧 Did you turn on the main breaker? 🔧 Are you pressing the cycle start instead of power on?
Program won't load	G01 Check (file type, percent signs (top and bottom of the program), and correct O number for program number.) G01 Is your program in a subfolder on your USB drive?
Error - bad code	G01 Make sure that your code is G01 (number 0) not GO1 (letter O). G01 Check other number and letter assignments in each code to make sure they are correct. G01 Check G01, G02, and G03 to make sure you are doing the correct motion (either line or arc).
Machine motion stops for "no reason"	G01 Check your feed rate. F3 is not the same as F3.0.
Spindle is too fast or slow	Do you have the correct spindle speed? If you have the incorrect speed, did you adjust to the correct speed?
Bad surface finish or chatter	Is your tool hanging out more than it should be? Is your tool sharp? If your tool is dull, index insert or reshape. Is your feed rate too fast or too slow? Adjust RPM and/or feed rate. Is the part clamped correctly in the vise?
Chips are wrong shape/color	Is your spindle speed too fast? Change RPM to lower speed. Is your feed rate too fast? Slow down feed rate. G01 Is the DOC too much? Reduce the depth of the cut. G01 Is your tool step over set to the correct amount?
Part too big or too small	Did you have a bad tool touch-off? Retouch-off your tool and check for discrepancies. G01 Are you using the correct tool diameter offset? G01 Is the cutter compensation set to the correct side (left or right?) G01 Are you using the correct tool height offset? Is the part clamped correctly in the vise?

KEY

🔧 MECHANICAL & ELECTRICAL   SETUP & TOOLING   G01 PROGRAMMING   FEEDS & SPEEDS