CAMDUCT MACHINE SETUP

PATRICK HENDRICKS

SENIOR SPECIALIST

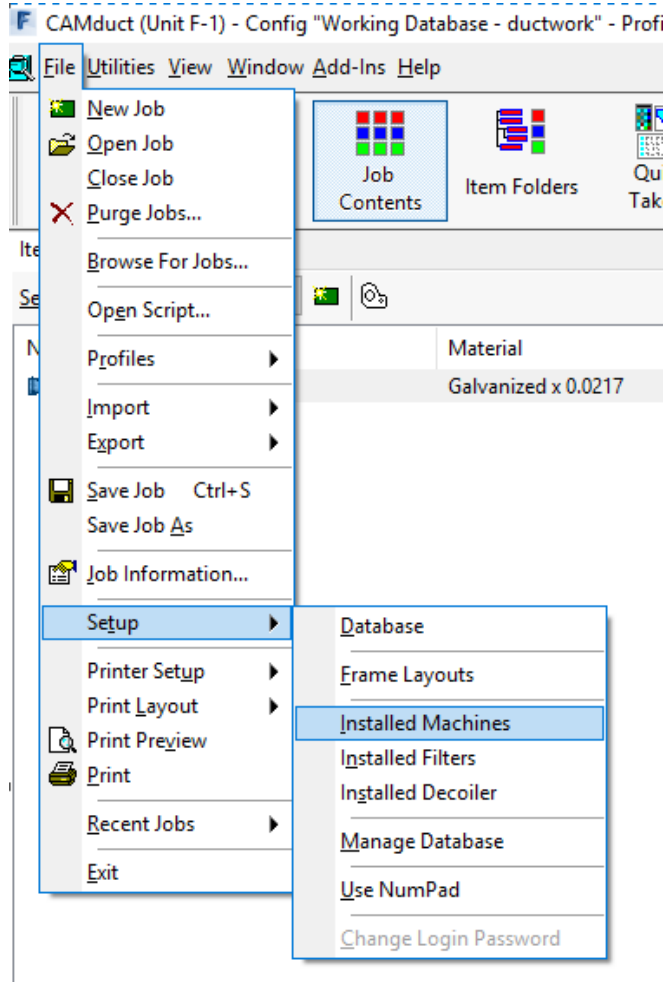
APPLIED SOFTWARE

IN THIS SESSION...

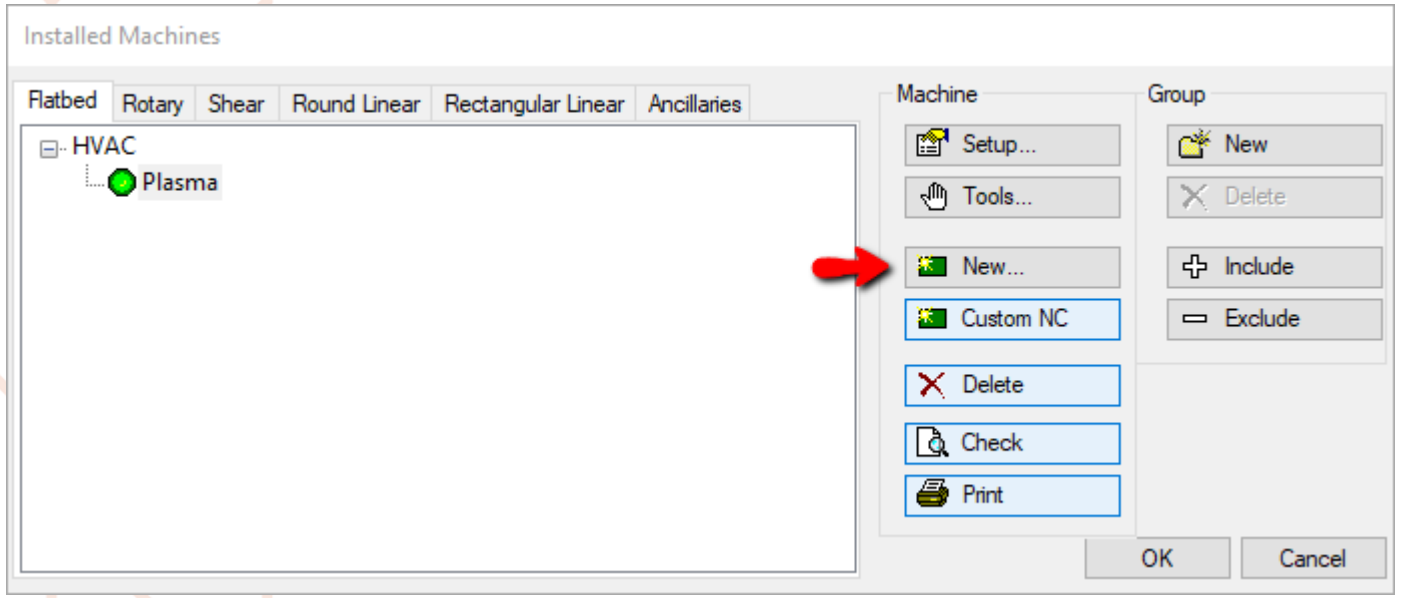
- Discuss preliminary machine setup
- Available post processors
- Common and uncommon settings
- Tool setup and variables
- Reviewing and testing NC files
- Custom NC

WHERE TO START

- File->Setup->Installed Machines



WHERE TO START



Select New for New Machine

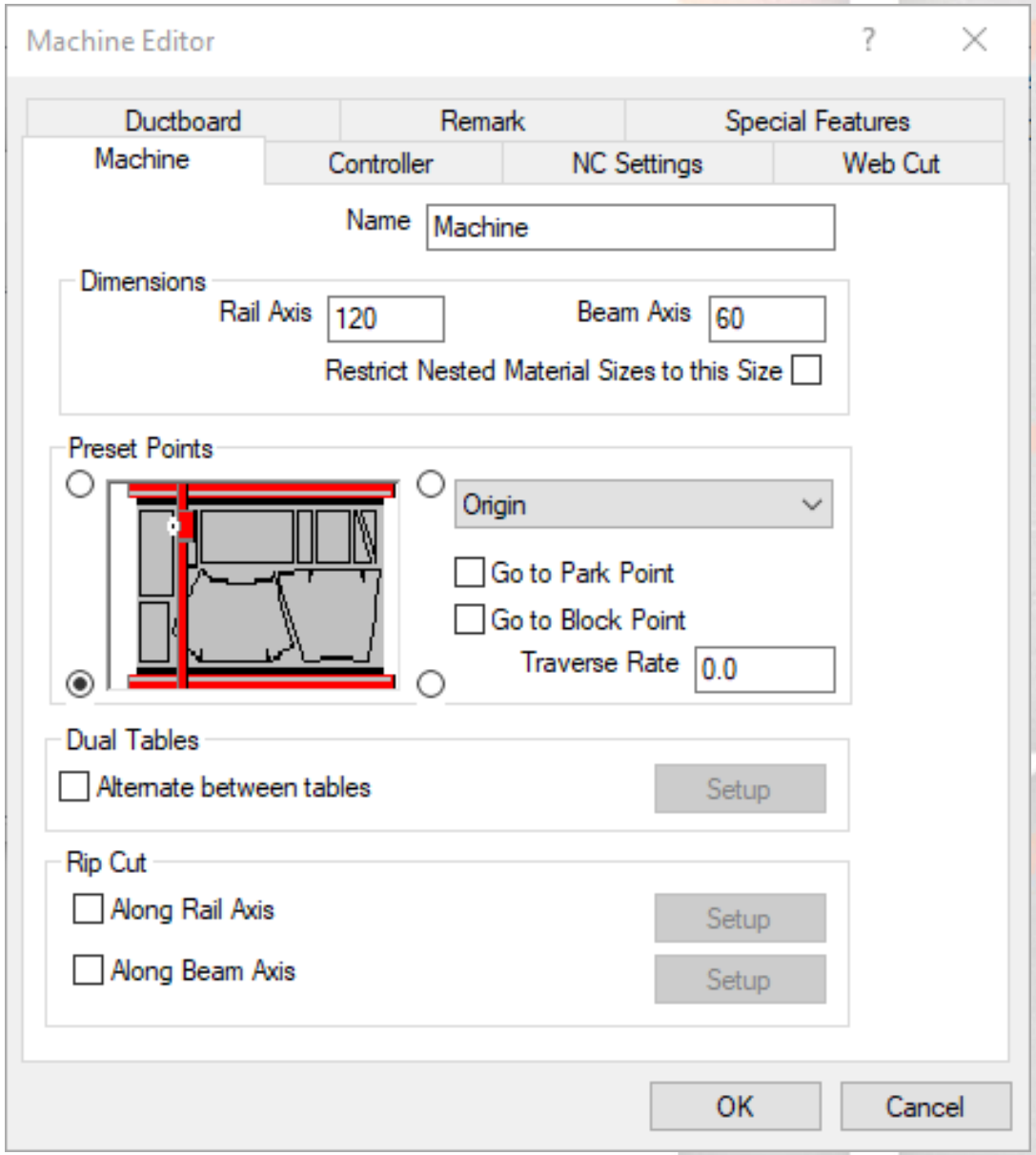
- Multiple types of machines
- Setup: Edit existing machine
- Tools: Edit selected machines tools
- New: Create New machine
- Custom NC: Create new custom NC machine
- Delete: Delete machine
- Check: Review any material data missing for tools
- Print: Print out machine and tool information

- Group: Can organize machines into groups
- New: New group
- Delete: Delete Group
- Include: Add machine into list that is in database
- Exclude: Remove machine from list that is in database

BASE SETTINGS

- Name
- Dimensions: Table dimensions (120x60, 240x60, varies)
- Preset Points:
  - Origin: For X and Y movements
  - Home Point: Starting head location
  - Start Point: Start of cut
  - Park Point: Location head returns to after cut
  - Block Point: Similar to park, but can be utilized with single parts that will be cut repeatedly
  - Nesting: Specify nesting location for this machine

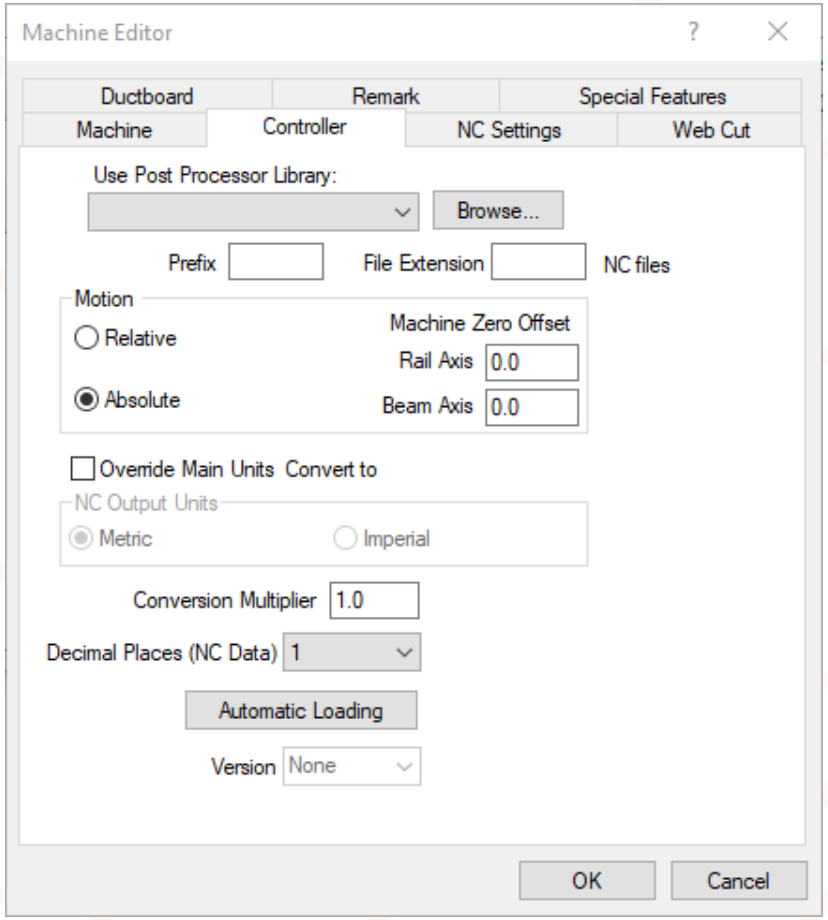
- Traverse Rate: Rapid traverse movements
- Dual Table: For tables that are two tables, or one long table capable of using two sheets
- Rip Cut: Complete cut from one end of sheet to the other



WHAT IS MY ORIGIN?

- The most important part of table orientation
- Depends on how the machine is configured
- Determines X and Y coordinate layout
- Bottom Left: X goes right down the length of table, positive. Y goes up the width of the table, positive.
- Top Left: X goes down the width of table, negative. Y goes right down the length of the table, positive.
- Top Right: X goes left down the length of table, negative. Y goes down the width of the table, negative.
- Bottom Right: X goes up the width of the table, positive. Y goes left down the length of the table, negative.
- As you can see tons of variation. Depending on your table settings on the floor, may get mirrored or flipped parts if Origin in CAM is not set up properly.

CONTROLLER TAB

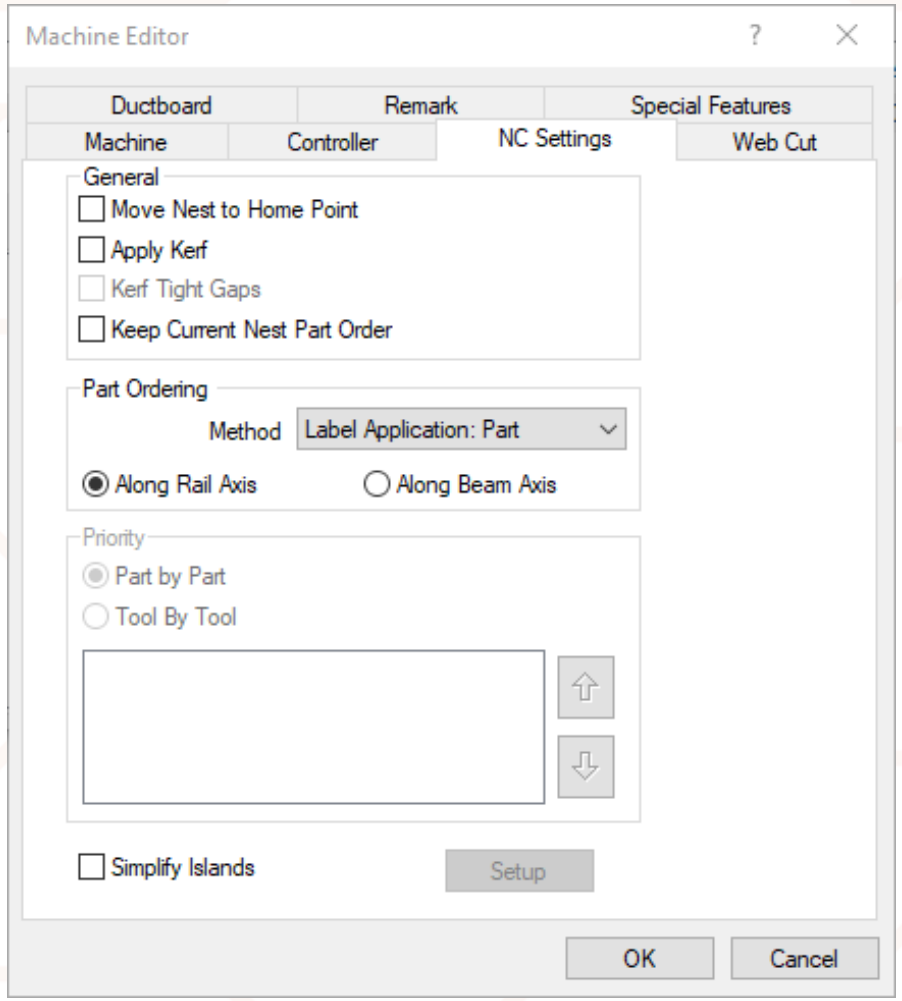


- Post Processor Library
- Prefix: Adds a prefix to NC data sent to CNC folder
- File Extension: File type your machine is looking for
- Motion
  Relative: Head moves up 5 inches, code calls for Y5
  Absolute: Head moves up 5 inches from a specific coordinate (X5,Y5), code calls for X5,Y10. Coordinate system
- Machine Zero Offset: Origin/Orientation correct, but parts off sheet, offset moves to fit the parts on the sheet
- Override units
- Conversion Multiplier: Used if code data is too large or two small. If part is typed in as 10 inches, but burned out 5 inches, multiplier of 2
- Decimal Places: How precise is the numerical NC data
- Automatic Loading: Machine specific. Prompts machine to load next sheet automatically
- Version: Post Processor specific

WHAT POST PROCESSOR?

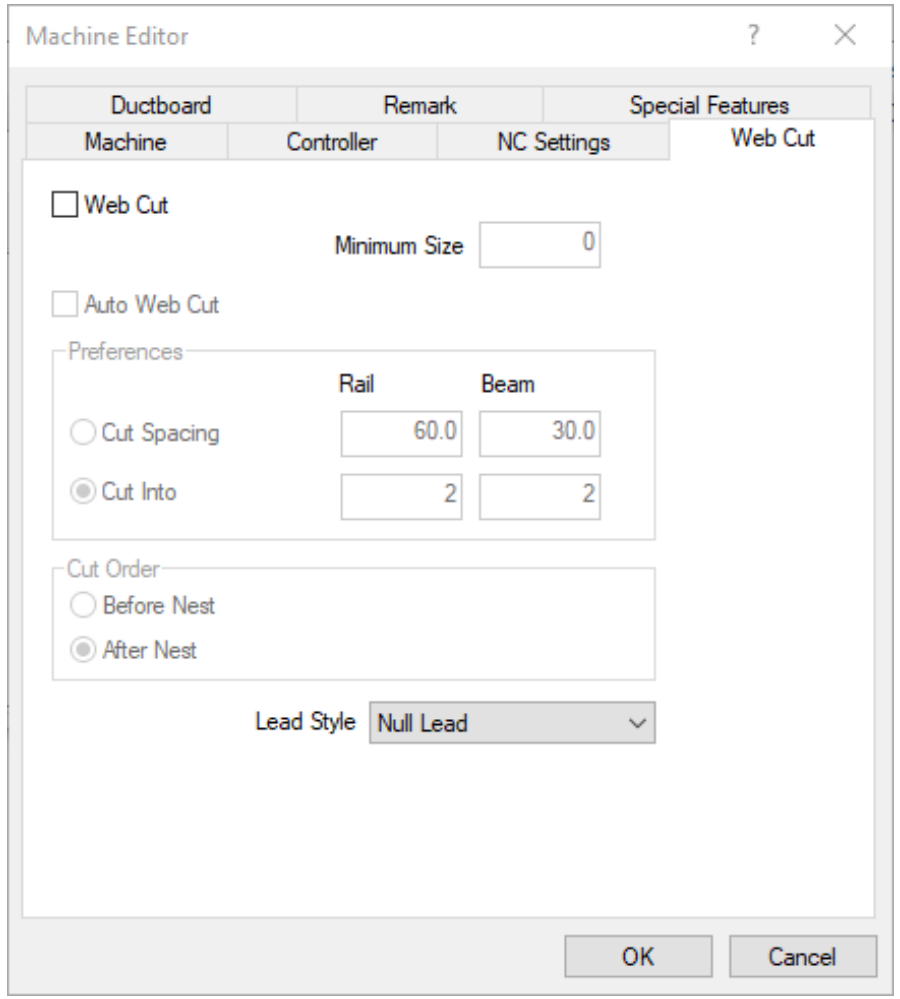
- Multitude available
- Located in C:\Program Files\Autodesk\Fabrication 2020\CAMduct\VPLs
- D6 is common (Koike, Hypertherm, older Lockformers)
- Updated Lockformers use lockformer1000D.
- PPI, Mach 3 for Lasermax.
- When in doubt, check with us!
- Link to Available Posts

NC SETTINGS TAB



- Move Nest to Home Point
- Apply Kerf (from tool data)
- Kerf Tight Gaps: Gap smaller than kerf, just use kerf.
- Keep Current Nest Part Order: When NC is written, part order remains unchanged from nest
- Part Ordering: Cut order of parts
- Priority: Part to Part, or use in order of Tools for multi-tool machines
- Simplify Islands: Reduces complexity to reduce amount of output data in code

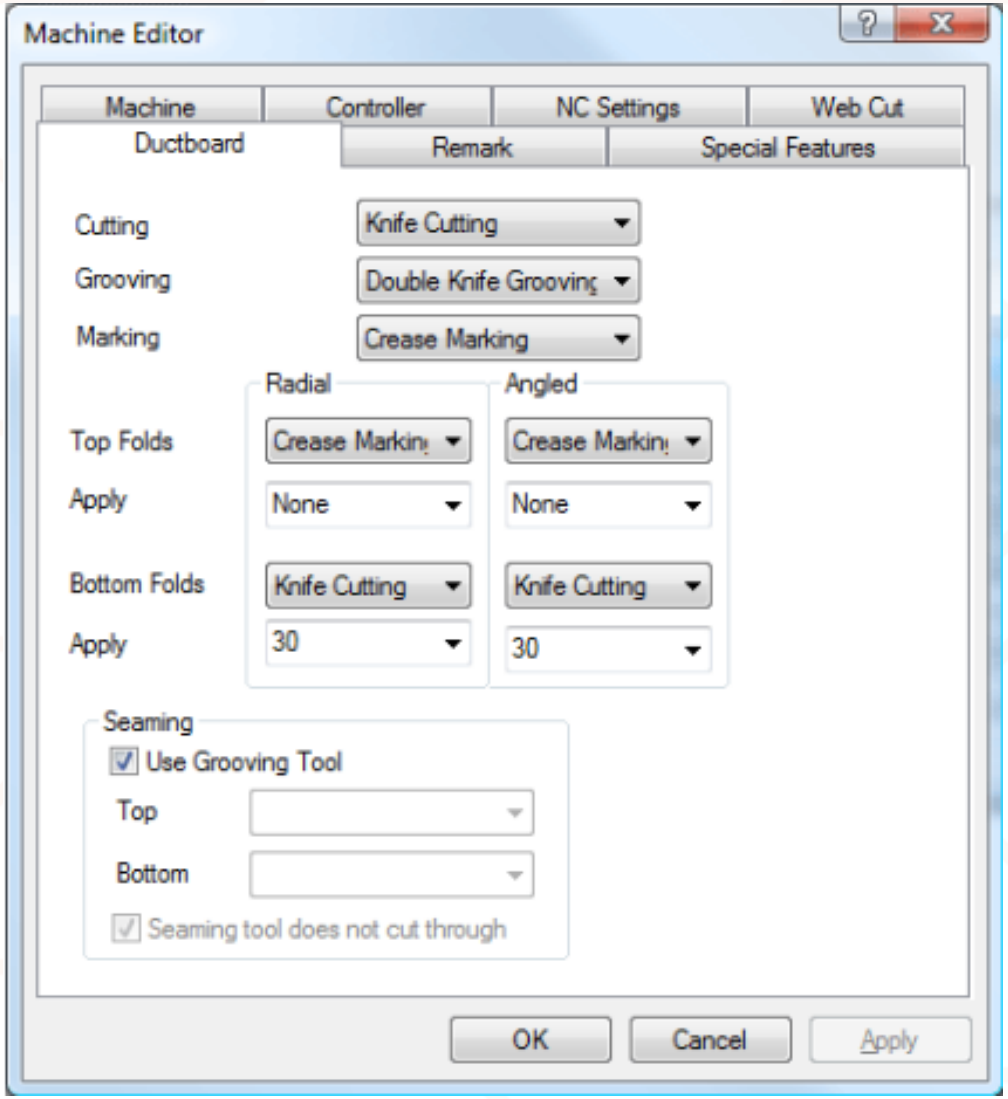
WEB CUT TAB



- Used specify web cutting material that remains on the sheet after parts are cut to manage scrap
- Usually used with oxyfuel, thick materials, or laser machines with small kerf
- Facilitates removal of parts from surrounding scrap

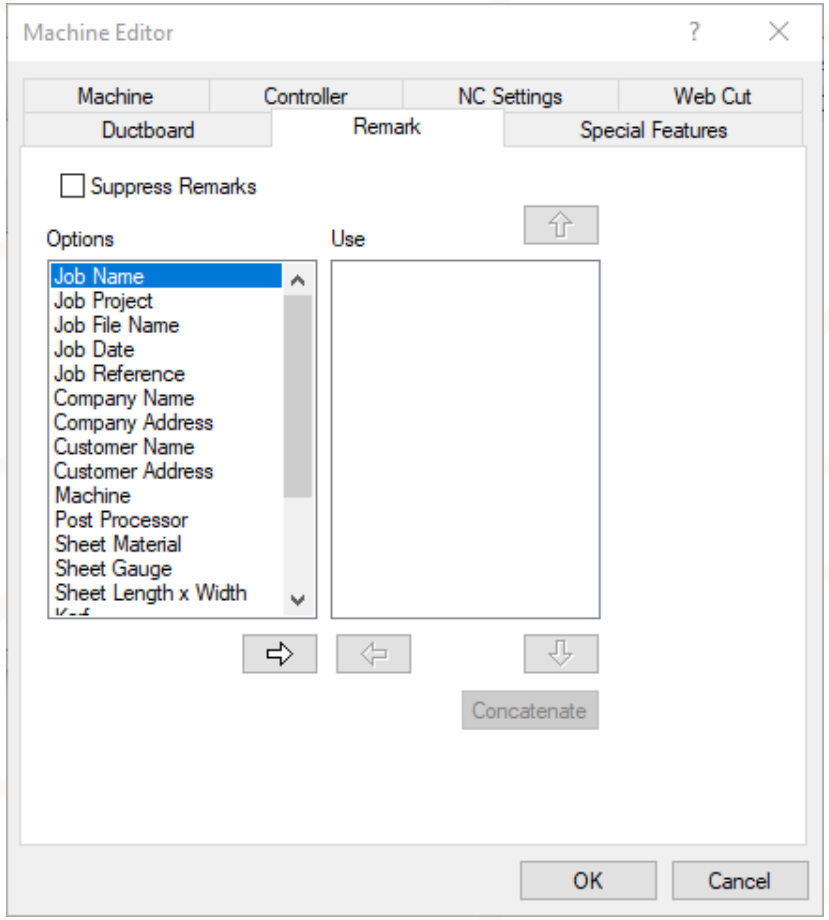
DUCTBOARD TAB

- Non sheet metal, pre-insulated duct
- Specify tool for Cutting, Grooving, and Marking
- Specify tools for Top Folds and Bottom Folds on radius or angled bends
- Seaming can specify whether seams get the default grooving, or alternate

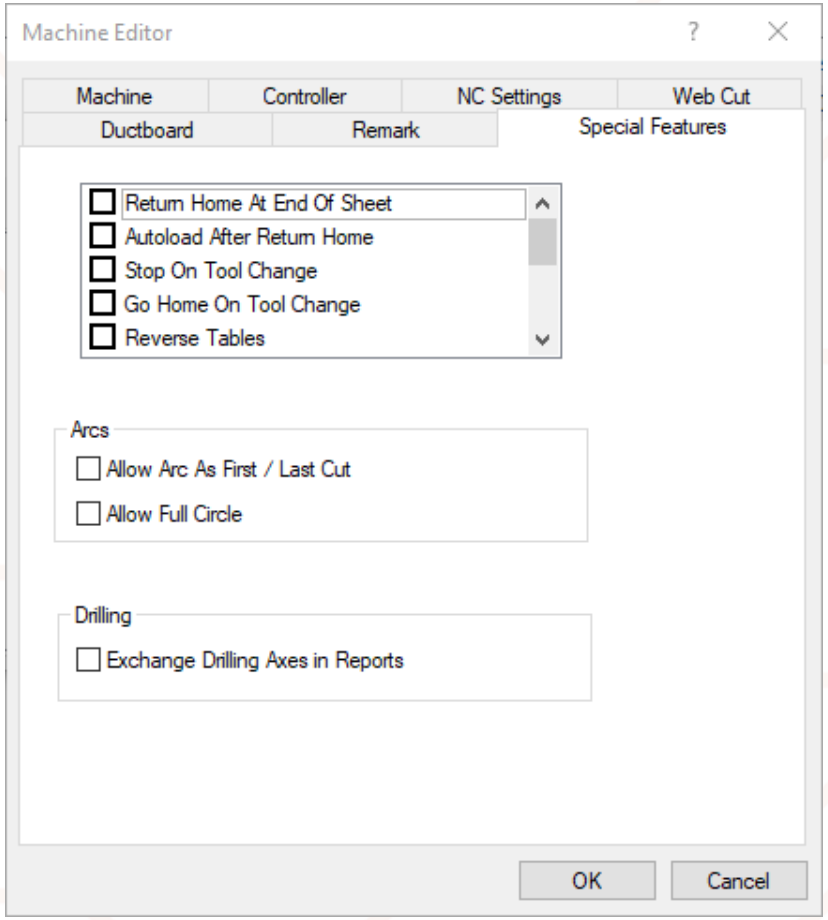


REMARK TAB

- Insert text data at start of NC code
- Suppress Remarks: All remarks are removed.
- Can specify various fields where NC code is marked.
- Useful for knowing how the code Is working.



SPECIAL FEATURES



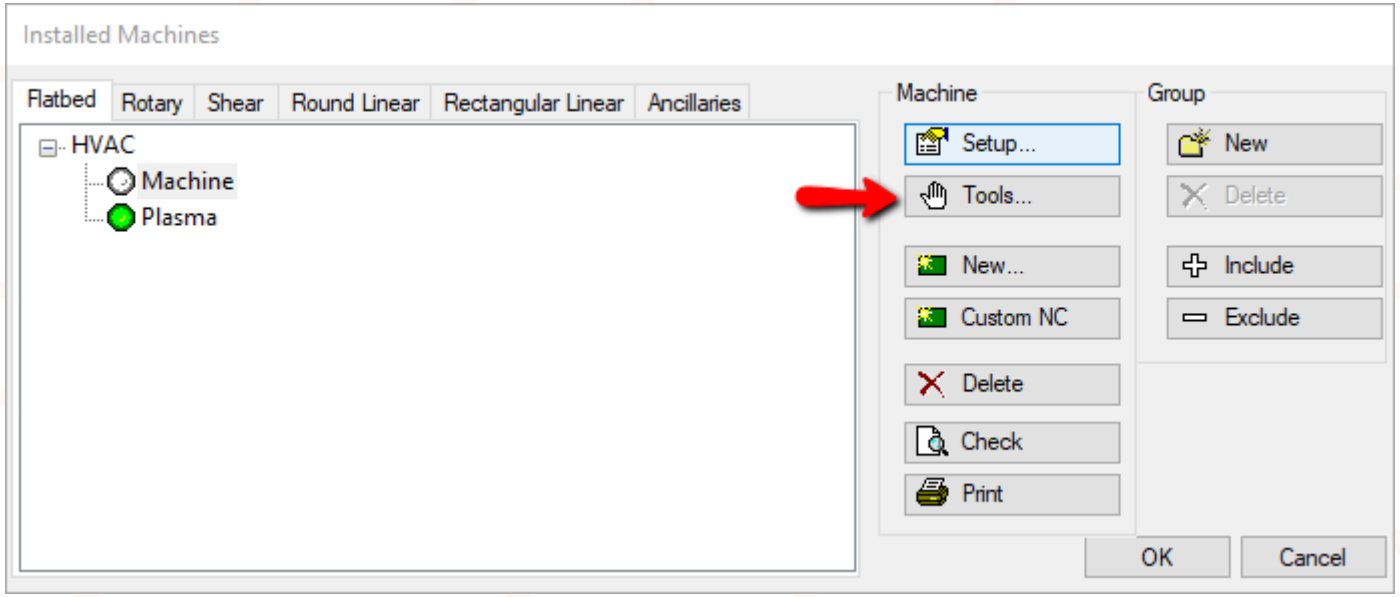
- Specific to each Post Processor
- Give different options that may be needed for your particular machine
- Arcs As First/Last Cut: Generates code using an arc command for the first and last cut part, instead of straight
- Allow Full Circle: Allows for full circle instead of many small straights
- Drilling: For machines with a drill. Allows for reporting on Drill positions, rather than part positions

NOT EVERY TAB HAS TO BE FILLED WITH DATA

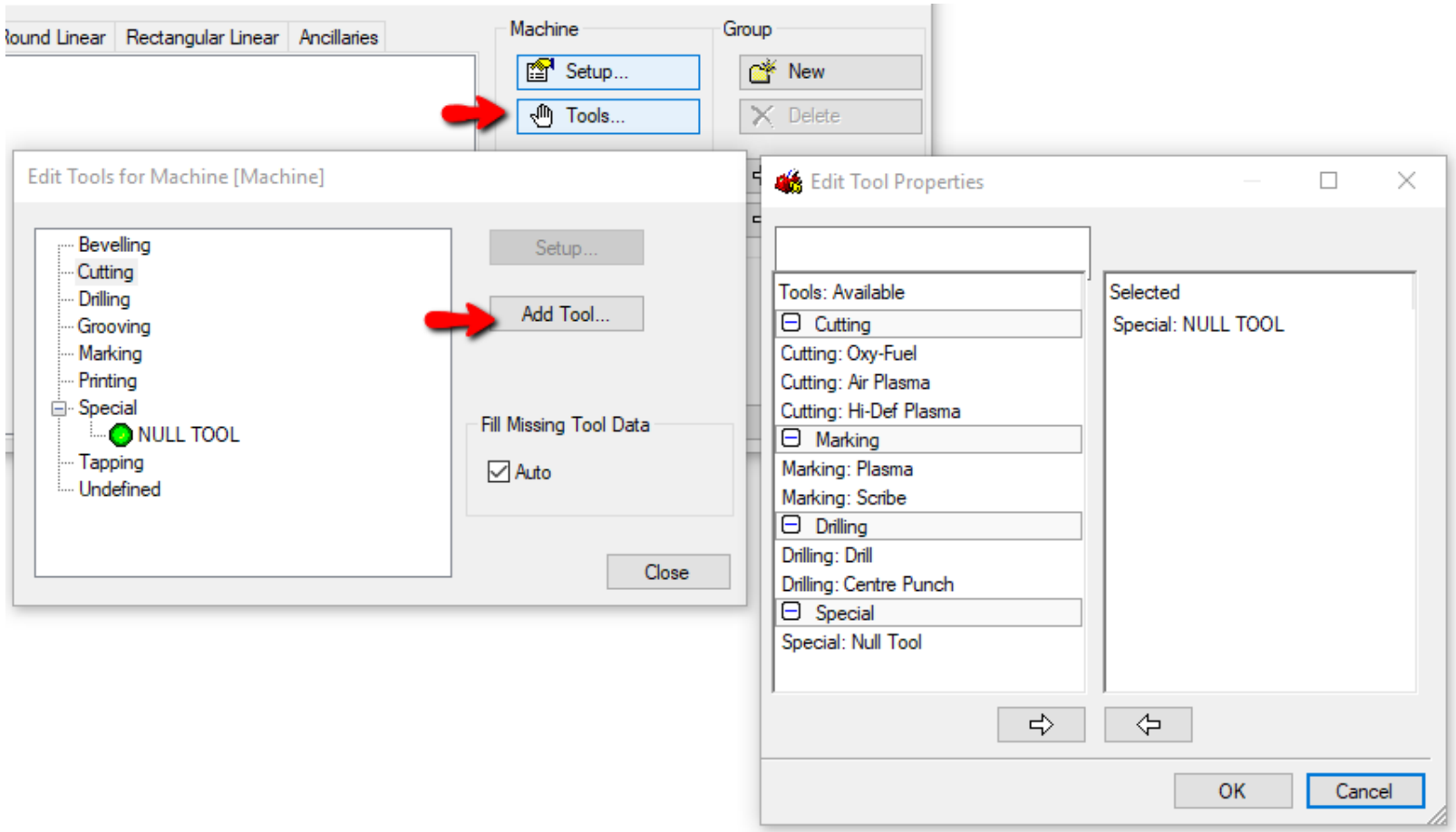
- For some machines the Web Cut, Ductboard, and Special Features tabs could be completely irrelevant
- If you have Ductboard, you know you need it, etc.
- Not every setting in every tab will be needed
- Case by case bases

TOOL SETUP

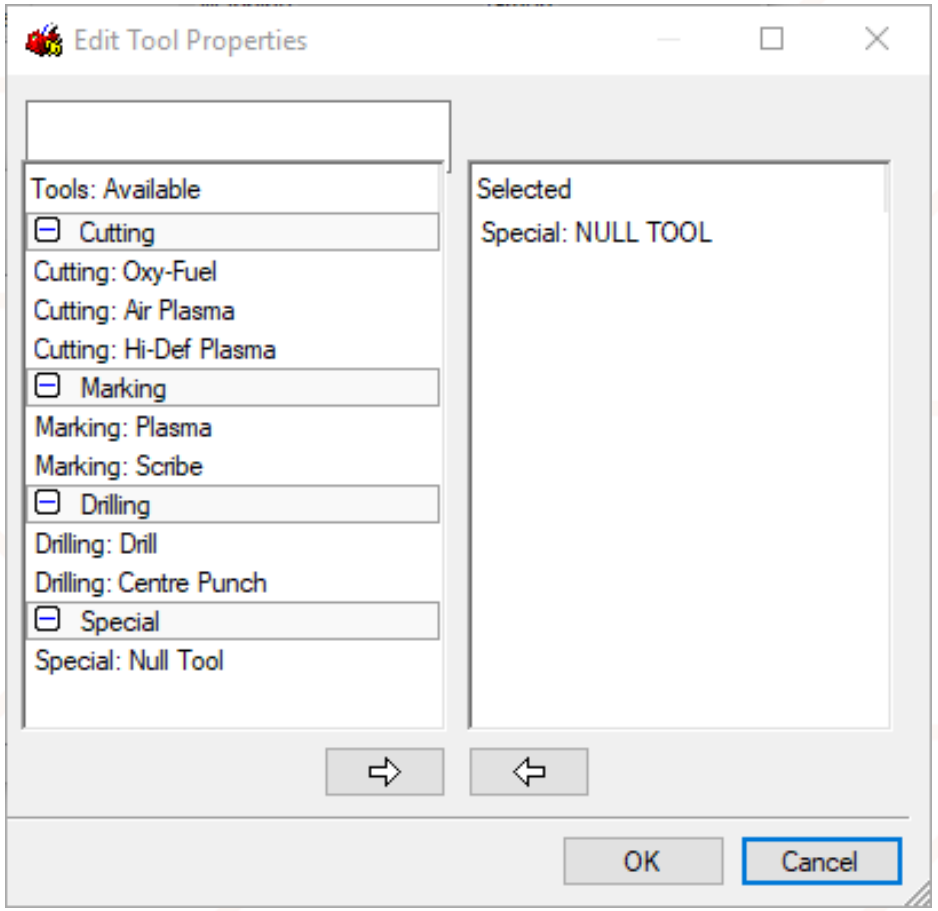
- Tools define basic parameters for your machine
- Tools must be made before NC code can be written
- Contains global settings and material specific settings
- Selected by highlighting machine in main Installed Machines window, clicking Tools



INITIAL TOOL SETUP



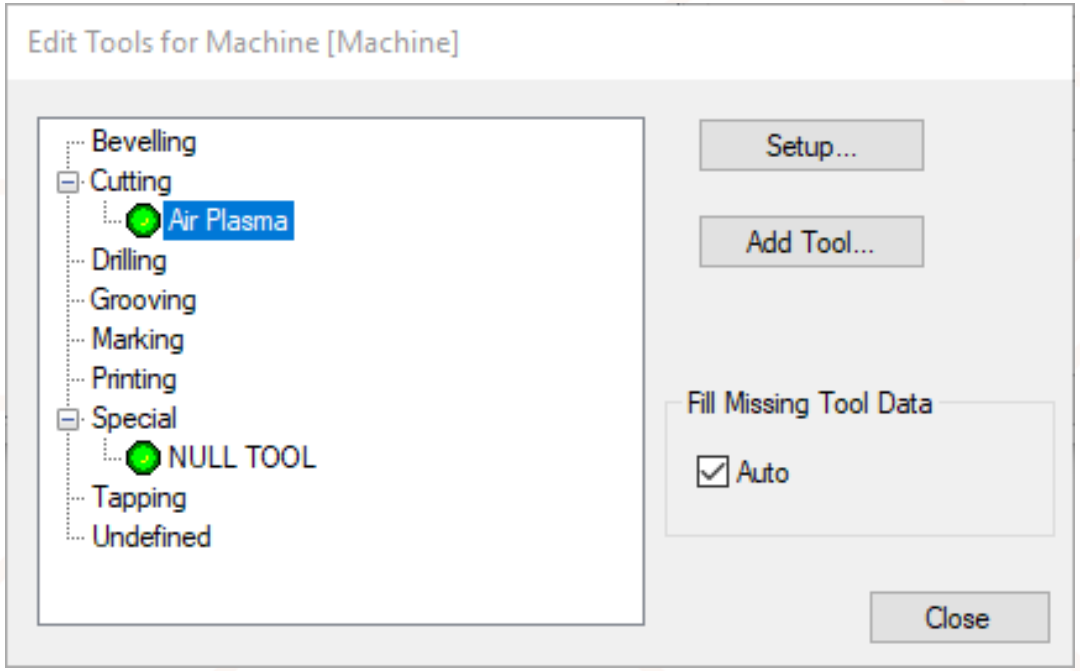
TOOL PARAMETERS



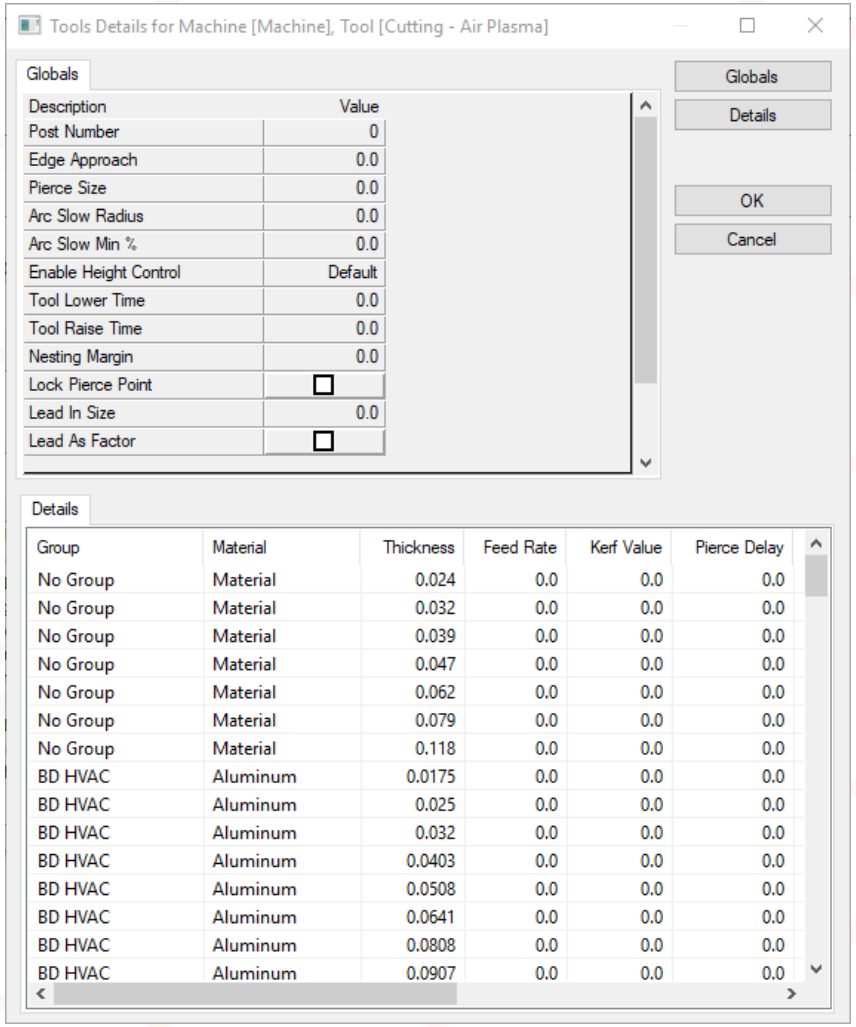
- Left is available tools (Post dependent)
- Right is currently selected tools
- Use the left and right arrows to add or remove tools for your machine

EDIT TOOL

- Once tool added, select tool and click on Setup



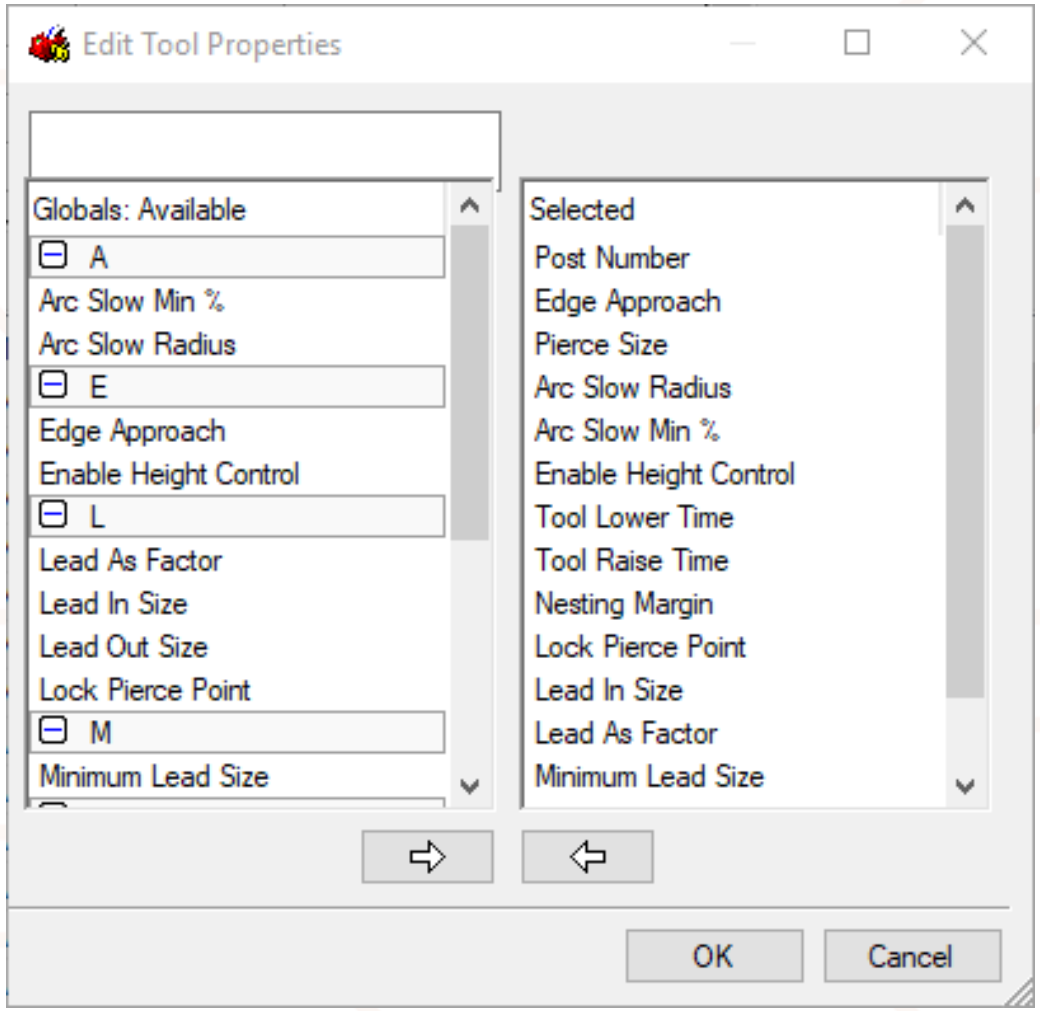
EDIT TOOL



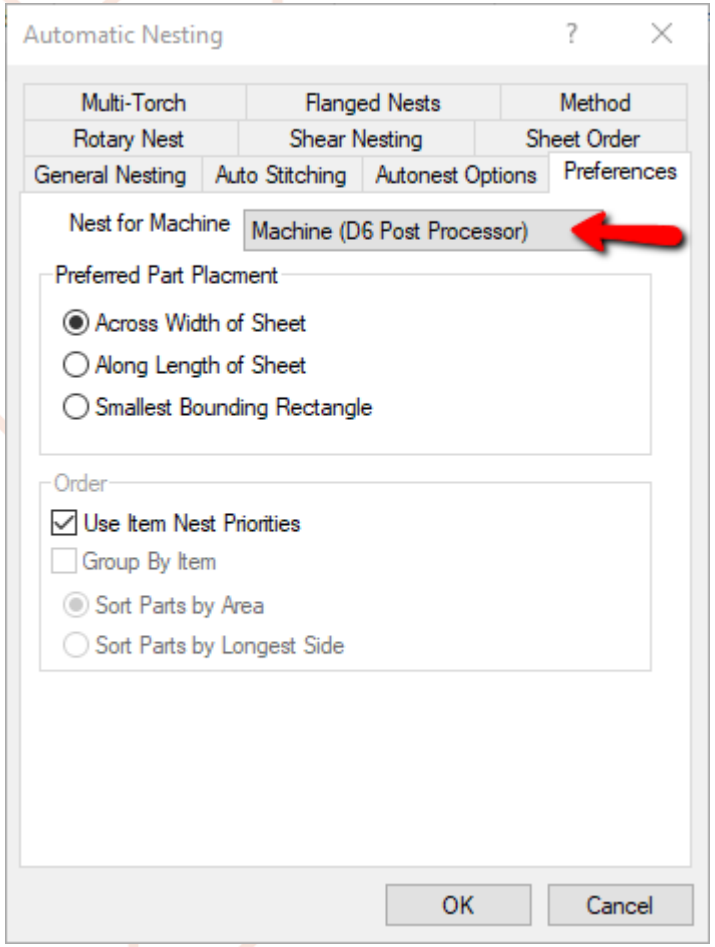
- Globals: At top. Parameters applied to all materials
- Details: At bottom. Parameters specific to materials and gauges
- Common parameters: Kerf, Feed Rate (cutting speed), Edge Approach, Tool Lowering/Raising Times, Nesting Margin
- More settings than what show in default
- Some tool settings interfere or override with other settings (like nesting). So if not actively used, could benefit from removal

EDIT TOOL

- Clicking on Globals or Details in the upper right allows to specify what is in which category
- Settings cannot be in both
- Use arrows to add and remove
- Once added, click on relevant setting in main tool screen to specify value



NOW WHAT?



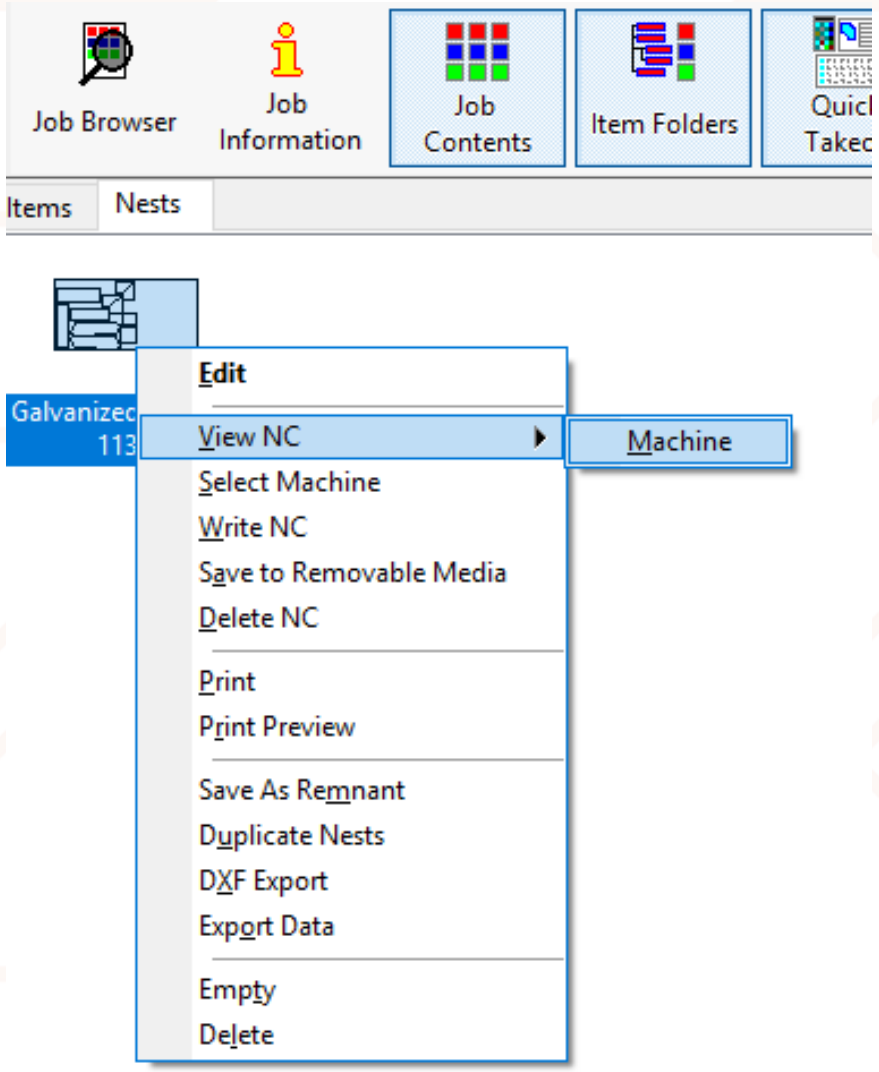
- Once all settings are applied, get to nesting
- Either in your Automatic Nesting window, or Setup Processes nesting settings, specify in the Preferences tab your new machine
- Write NC for the nested sheets to get code

VIEWING NC AND TROUBLESHOOTING

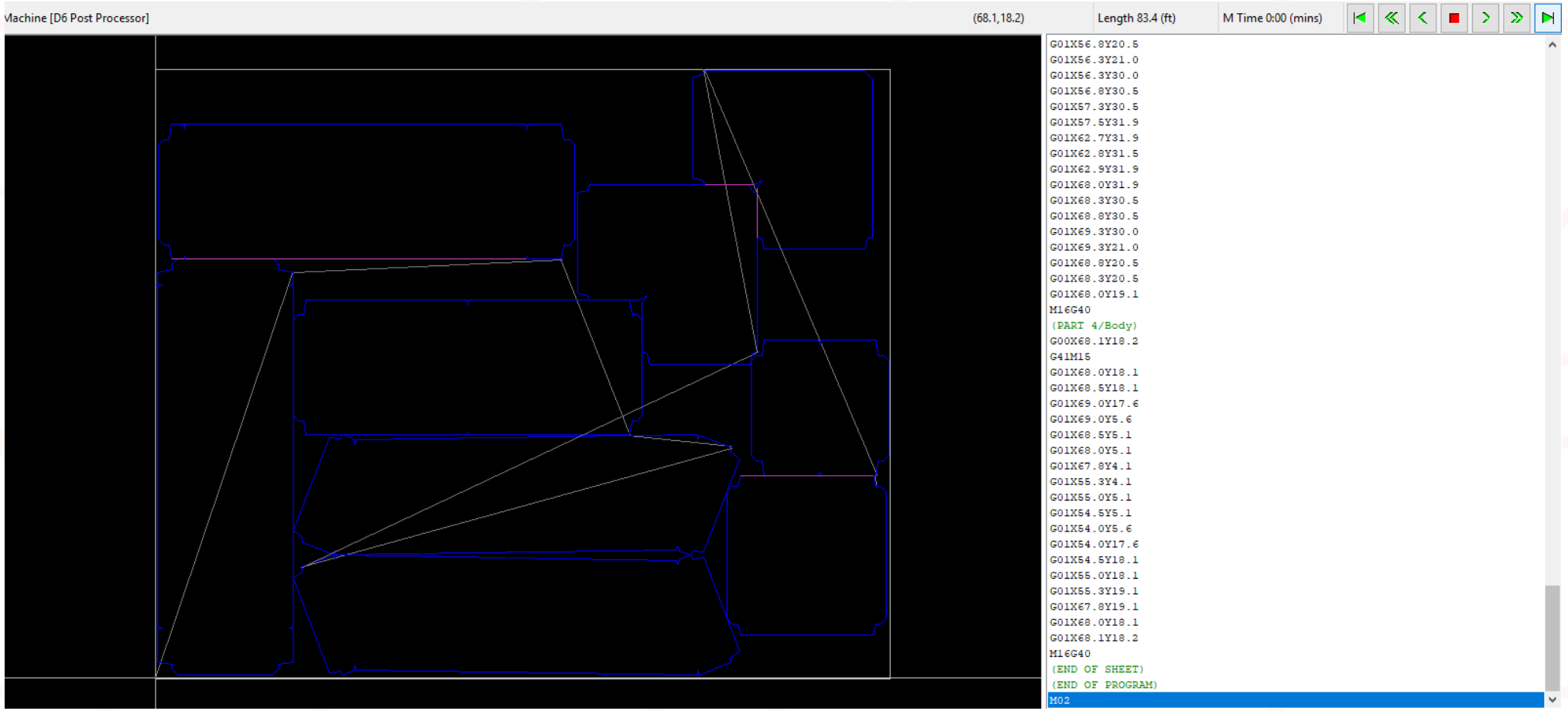
- Within CAMduct you can view the NC that CAMduct is pushing out
- Useful for viewing how CAMduct thinks the machine will cut
- Great for comparing with what is seen at the table

VIEWING NC

- Once nested, in Job Contents right click on sheet with NC written and select View NC



VIEWING NC


Machine [D6 Post Processor]
(68.1,18.2)
Length 83.4 (ft)
M Time 0:00 (mins)
G01X56.8Y20.5
G01X56.3Y21.0
G01X56.3Y30.0
G01X56.8Y30.5
G01X57.3Y30.5
G01X57.5Y31.9
G01X62.7Y31.9
G01X62.8Y31.5
G01X62.9Y31.9
G01X68.0Y31.9
G01X68.3Y30.5
G01X68.8Y30.5
G01X69.3Y30.0
G01X69.3Y21.0
G01X68.8Y20.5
G01X68.3Y20.5
G01X68.0Y19.1
M16G40
(PART 4/Body)
G00X68.1Y18.2
G41M15
G01X68.0Y18.1
G01X68.5Y18.1
G01X69.0Y17.6
G01X69.0Y5.6
G01X68.5Y5.1
G01X68.0Y5.1
G01X67.8Y4.1
G01X55.3Y4.1
G01X55.0Y5.1
G01X54.5Y5.1
G01X54.0Y5.6
G01X54.0Y17.6
G01X54.5Y18.1
G01X55.0Y18.1
G01X55.3Y19.1
G01X67.8Y19.1
G01X68.0Y18.1
G01X68.1Y18.2
M16G40
(END OF SHEET)
(END OF PROGRAM)
M02

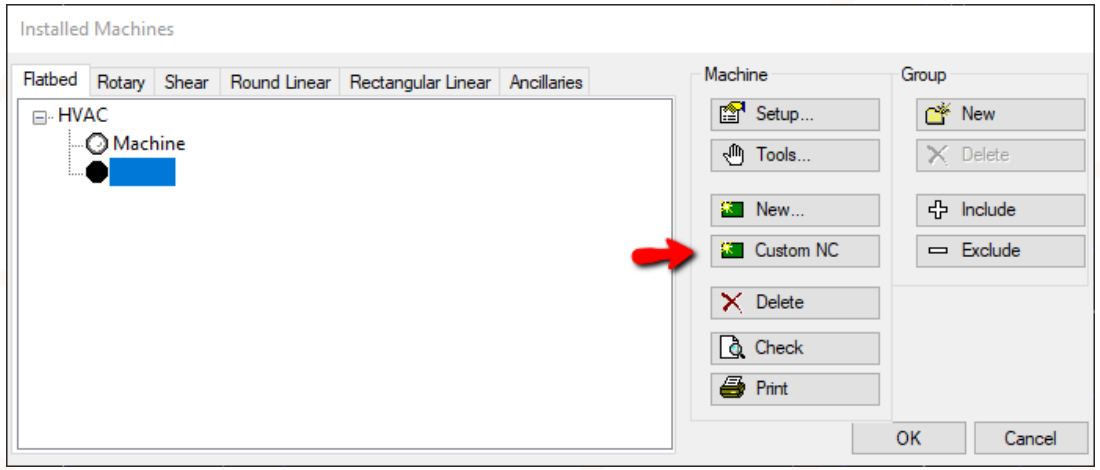
VIEWING NC

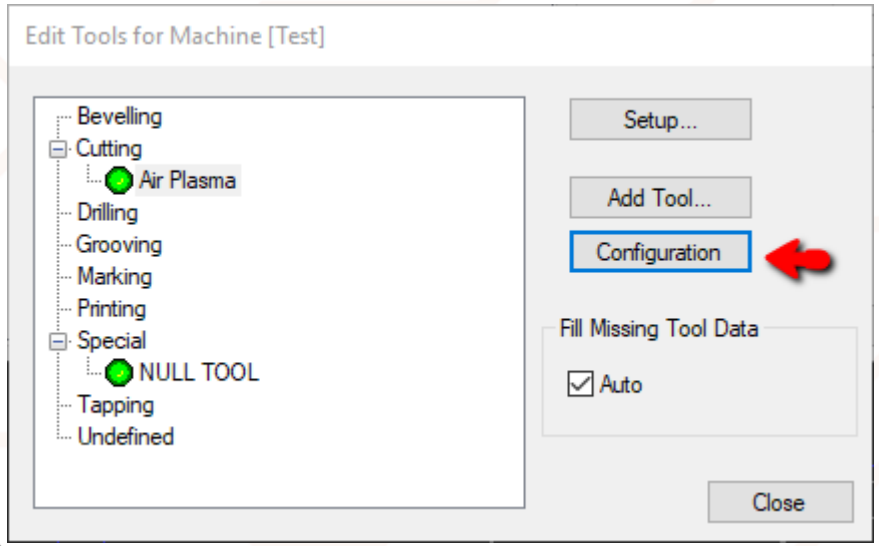
- Can view line by line, or tell the NC to play out and see how it will try to cut the parts
- Incredibly useful for machine troubleshooting
- If what you see in CAMduct and what you see at the machine line-up, usually good to go
- If it looks bad at the machine and looks bad in CAMduct, it's a CAMduct issue
- If it looks bad at the machine and good in CAMduct, could be either
- Let us know if things are amiss!
- Could be you identified a fix that must happen with post

CUSTOM NC

- Specify exact codes needed for tools and movement
- Typically used only if a base available post isn’t even close
- Some machines are still too complex for the Custom NC route
- While codes can be specified, if the issue with a post available in CAMduct is not just tool codes, Custom NC may not provide full solution. It might, though. Case by case bases
- Time consuming. Recommend base posts are tested to insure that Custom NC is needed

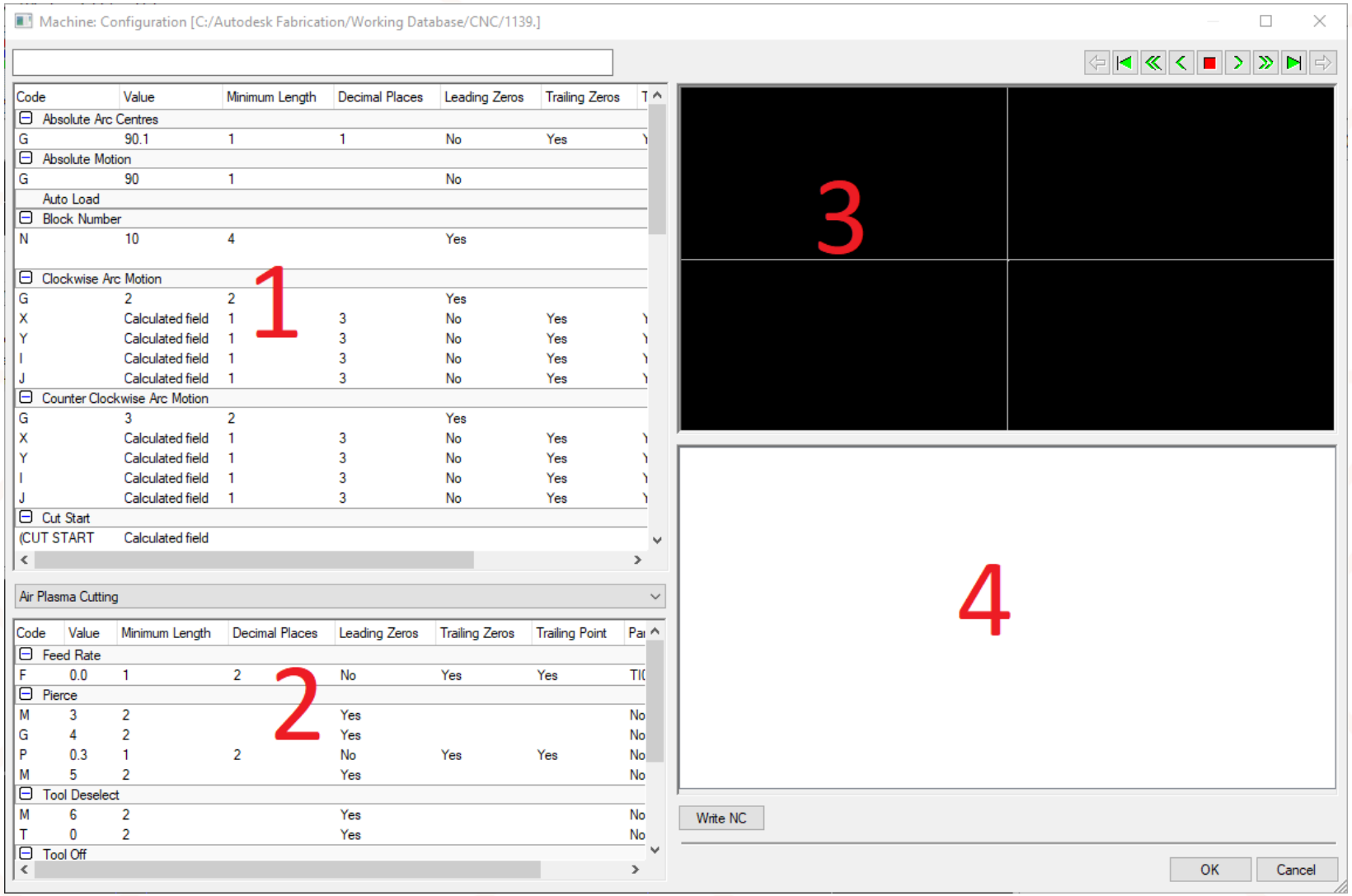
CREATING CUSTOM NC





- Instead of selecting New, select Custom NC
- Setup base table parameters (table size, origin, etc.) as shown earlier
- Once base parameters are done, select Tools and add a tool as before. More tools will be available, so choose that which you need
- Once tool is created, select the Machine, click on Tools, select the tool and go to Configuration

CUSTOM NC WINDOW LAYOUT



1.) BASIC MACHINE BEHAVIORS

- Set the NC code for specific actions
- Motion (Absolute, Relative)
- Kerf
- Program Start or End
- Subroutines
- Cut Start command

2.) SPECIFIC TOOL COMMANDS

- Use the dropdown to select which tool if multiple
- Specifies tool-specific behaviors
- Pierce
- Tool Select/Deselect
- On/Off
- Remember, some parameters (part ordering and the like will still be controlled by initial machine setup, not Custom NC)

3.) VIEW OF NESTED PARTS & 4.) VIEWING OF CODE

- If parts are nested, allows you to preview potential code without having to back out, Write NC manually, and View NC as mentioned earlier
- Immediate availability of seeing if the parameters you input are coming out correctly

CUSTOM NC AND FURTHER EDITS

- If you are creating a custom NC when a post is not available, there may still be more work to be done
- Custom NC is particularly useful for codes and some needed callouts for complex machines
- However, not all complex machines can be built with Custom NC alone
- Background scripting after NC is written can be used to fill in remaining data

QUESTIONS?

- Feel free to e-mail me at Phendricks@ASTI.com
- I am often on the road, so if I don't respond I'm not ignoring you!
- If you need assistance, reach out to FTGSupport@ASTI.com
- Case will be logged for you, and one of our many techs will be glad to assist.
- I'm around all week, so if you have any questions, or just want to shoot ideas around, look out for me.
- Thank you for your time, and enjoy the rest of the conference!

THANK YOU FOR ATTENDING OUR CLASS!

PLEASE FILL OUT THE SURVEY FOR THIS SESSION IN THE APP.

GO TO “MY SCHEDULE” → SELECT THIS SESSION → SCROLL DOWN TO “SESSION SURVEY”