COMFORT SYSTEMS USA

# CAMDUCT MACHINE POST PROCESSORS DEMYSTIFIED

WILLIAM TUCKER

COMFORT SYSTEMS USA

BIM TRAINER AND PRODUCT SPECIALIST

# CAMDUCT MACHINE POST PROCESSORS DEMYSTIFIED

- This class will focus on the machine post processors that ship with CAMduct. When you get a new machine, it can be a hassle figuring out how to get it up and running. We'll cover how to determine which Post Processor to use and verify and configure its output for your machine.

  - Reviewing sample CNC code
  - Testing Post Processor output
  - Common problems and how to over come them
  - Troubleshooting techniques

# DETERMINE THE POST PROCESSOR REQUIRED FOR YOUR TABLE

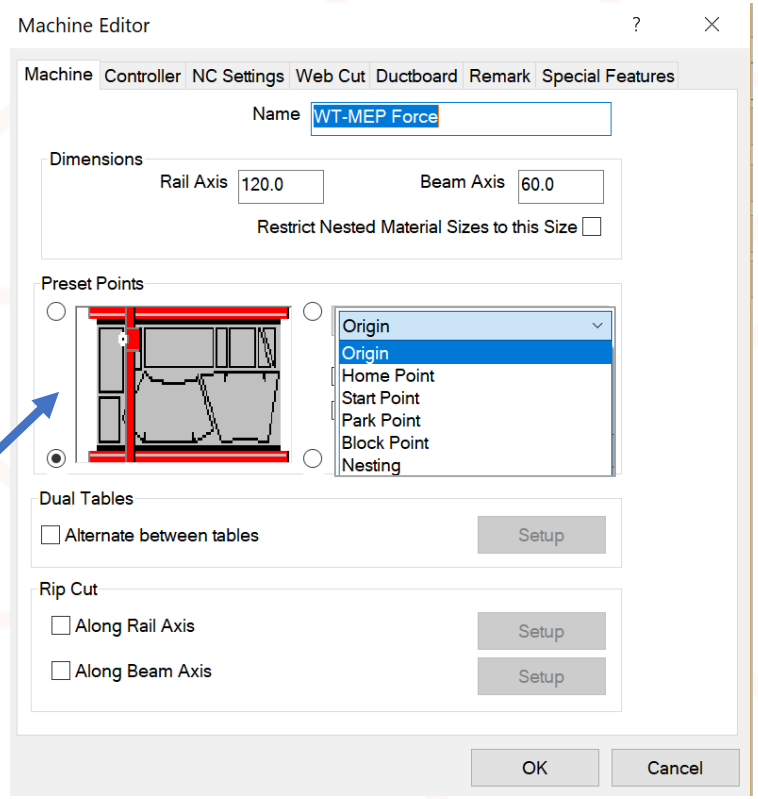
- CAMduct help "Supported Post Processors" will give a list of table manufactures and post processor file names.

| File Name | Post Processor | Default File Extension |
|---|---|---|
| acl64.vpl | ACL Post Processor for ACL 2000 & ACL 3000 series | CNC |
| ACSgen64.vpl | ACS Post Processor | ACS |
| actechgen64.vpl | AC Tech Post Processor | TAP |

- The D6 post processor will work for a lot of tables (Creonic Based)
  - Koike
  - Hypertherm Edge
  - Lockformer Vulcan
  - TechServ Controllers

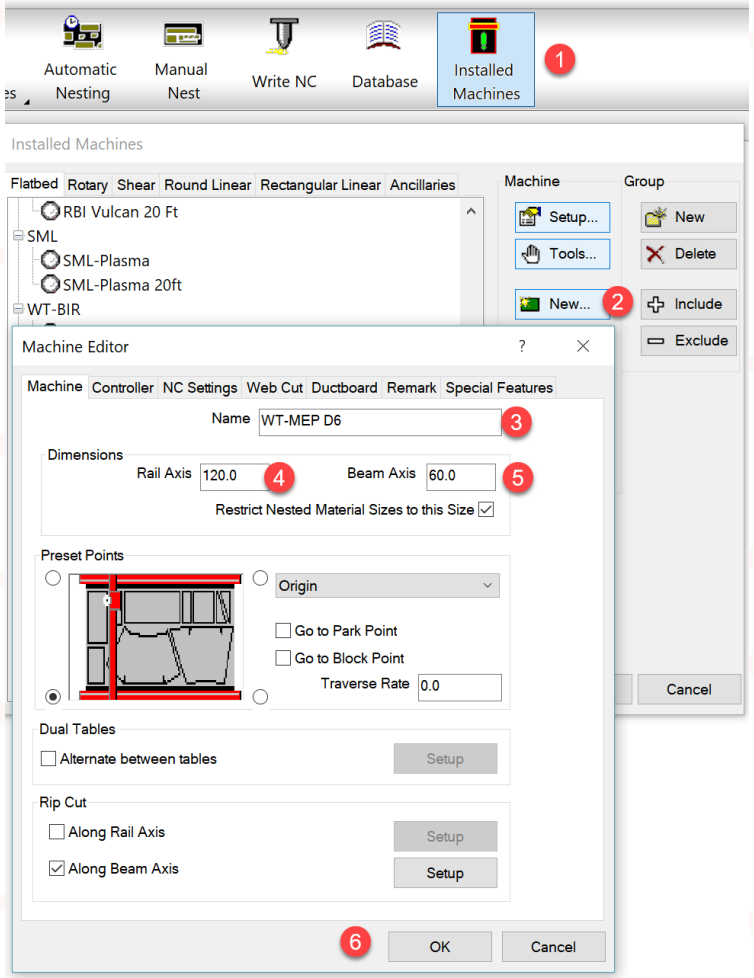
# SETTING UP A NEW MACHINE, LESSONS LEARNED

1. It's not an easy task
2. You are very lucky if you get it right the first time
3. I have never found 2 machines exactly alike
4. It is a trial and error task
5. Take it one step at a time
6. The time that it takes me to setup tables ranges from 5 minutes to 3 days.
7. I am not an expert…
8. This machine image can be deceiving!



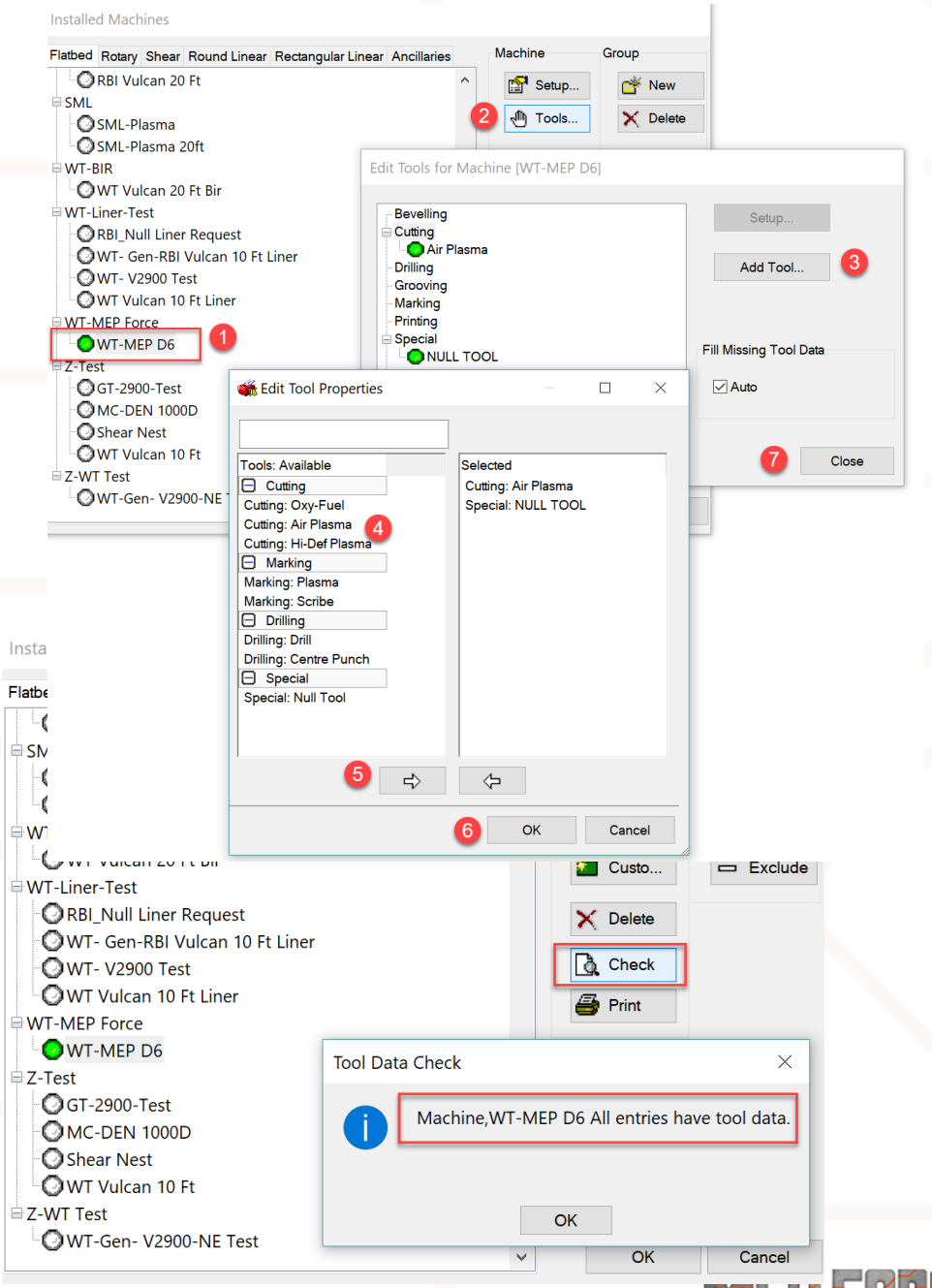
# SETTING UP A NEW MACHINE

1. Select Installed Machines
2. Select New
3. Name the Machine
4. Set Table Length
5. Set Table Width
6. Select OK



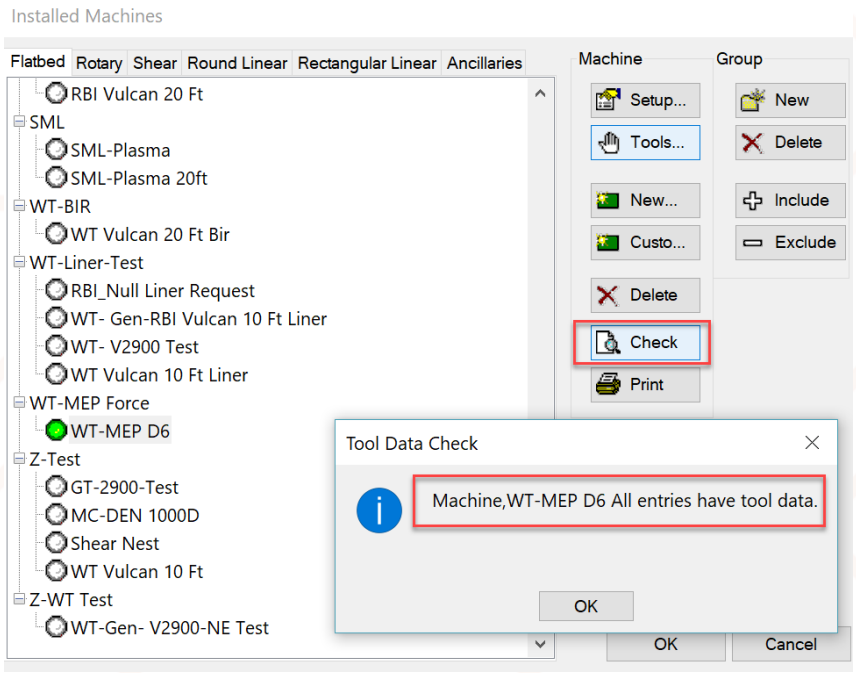
# ADDING TOOLS

1. With the new machine selected
2. Select tools
3. Add Tool
4. Select “Air Plasma” (Assuming that is what you are using
5. Add the tool
6. Select OK
7. Close



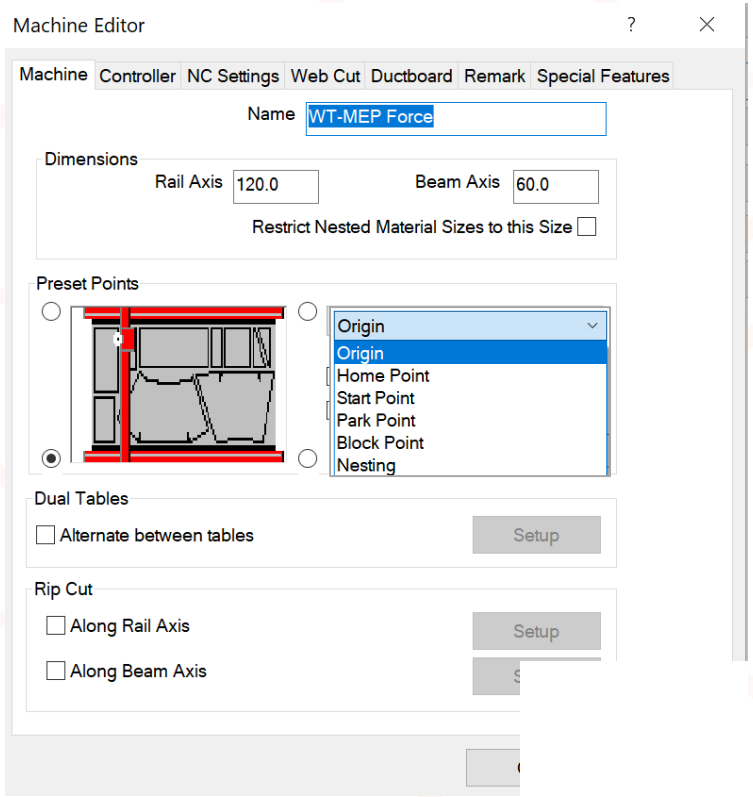
# ADDING TOOLS

- Select “Check”
- This will check and add tool data for all your materials



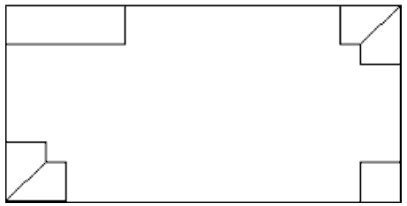
# MACHINE TAB

- Origin: The machines x0,y0 location
- Home point: The controllers home point
- Start Point: The starting point for the cut
- Nesting Point: The point that you will align the corner of the sheet.
- Block Point: This point is used for cutting a single part repetitively, maybe park the tool closer to the nesting point.

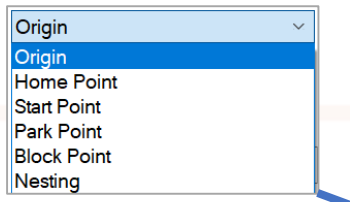


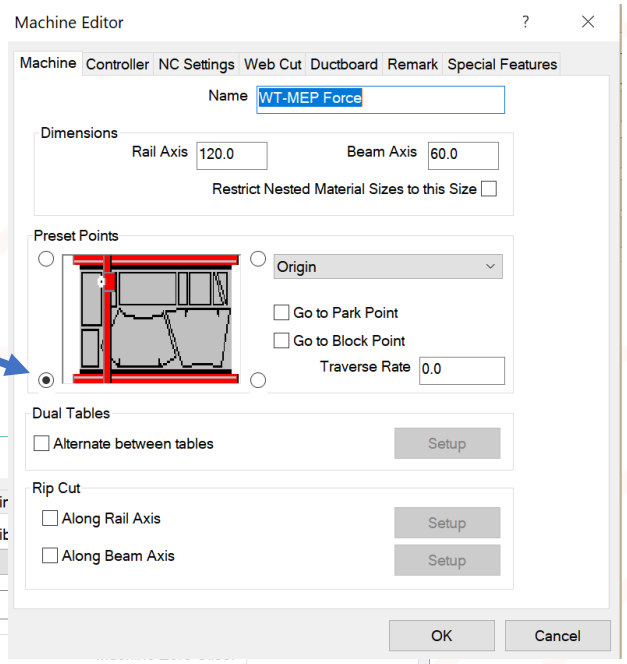
# WHERE IS THE ORIGIN AND DIRECTION OF TRAVEL

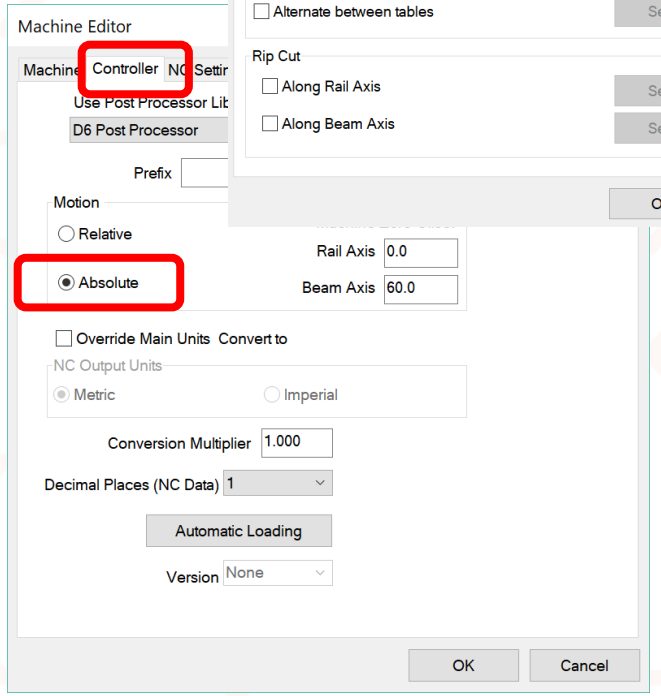
- When trying to determine the origin of the machine, I like to:
  - Manual nest a small square elbow without seams or connectors, one part at each corner of the sheet.



  - Select all points at one corner try to cut a part and see which direction the head moves.
  - I will keep moving all points to a different corner until I get the head moving toward the inside of all corners
  - Set the controller motion set to absolute for troubleshooting, this can possibly be changed to relative after the machine is setup



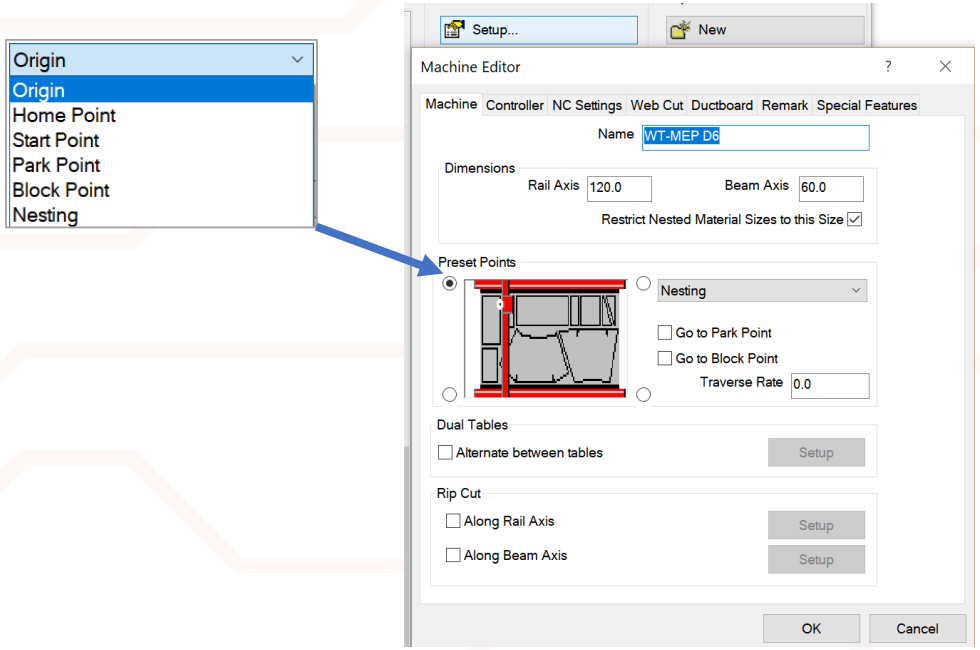
# TROUBLESHOOTING THE G-CODE

- Write the NC for the sheet
- View the NC
- Review the G-Code and verify that all coordinates are positive values
- Negative values at this point is a bad thing
- At the machine control panel call the NC
- Instruct the machine for a dry run

```
(1462)
(ORIGIN=BOTTOM LEFT)
(INCH)
G70
(ABSOLUTE)
G90
(PART 1/Top Cheek)
G00X101.9Y59.9
M15
G01X119.9Y59.9
G01X119.9Y41.9
G01X107.9Y41.9
G01X107.9Y47.9
G01X101.9Y47.9
G01X101.9Y59.9
M16
(PART 1/Throat Wrapper)
G00X107.8Y0.2
M15
G01X107.8Y12.2
G01X113.8Y12.2
G01X119.8Y12.2
G01X119.8Y0.2
G01X113.8Y0.2
G01X107.8Y0.2
M16
(PART 1/Back Wrapper)
G00X0.1Y47.8
M15
G01X0.1Y59.8
G01X18.1Y59.8
G01X36.1Y59.8
G01X36.1Y47.8
G01X18.1Y47.8
G01X0.1Y47.8
M16
(PART 1/Bottom Cheek)
G00X0.1Y18.3
M15
G01X12.1Y18.3
G01X12.1Y12.3
G01X18.1Y12.3
G01X18.1Y0.3
G01X0.1Y0.3
G01X0.1Y18.3
M16
(PARK)
G00X0.0Y0.0
(END OF SHEET)
(END OF PROGRAM)
M02
```

# WHERE IS THE ORIGIN AND DIRECTION OF TRAVEL

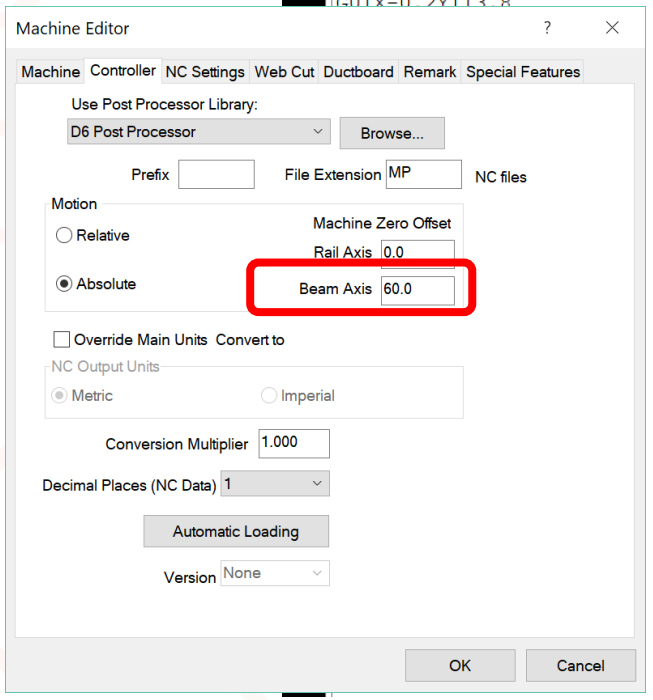
- If the tool moves towards the center of the table then you have located the origin
- If it fails, you need to change all points to another location



# REVIEW THE G-CODE AGAIN

- Write the NC again
- Notice the negative values and they are on the width of the table
- I don’t need to test this; I know it will fail
- Offset the beam by 60 or by the width of the table

```
(1462)
(ORIGIN=TOP LEFT)
(INCH)
G70
(ABSOLUTE)
G90
(PART 1/Top Cheek
G00X-59.9Y101.9
M15
G01X-59.9Y119.9
G01X-41.9Y119.9
G01X-41.9Y107.9
G01X-47.9Y107.9
G01X-47.9Y101.9
G01X-59.9Y101.9
M16
(PART 1/Throat Wrapper)
G00X-0.2Y107.8
M15
G01X-12.2Y107.8
G01X-12.2Y113.8
G01X-12.2Y119.8
G01X-0.2Y119.8
G01X-0.2Y113.8
```



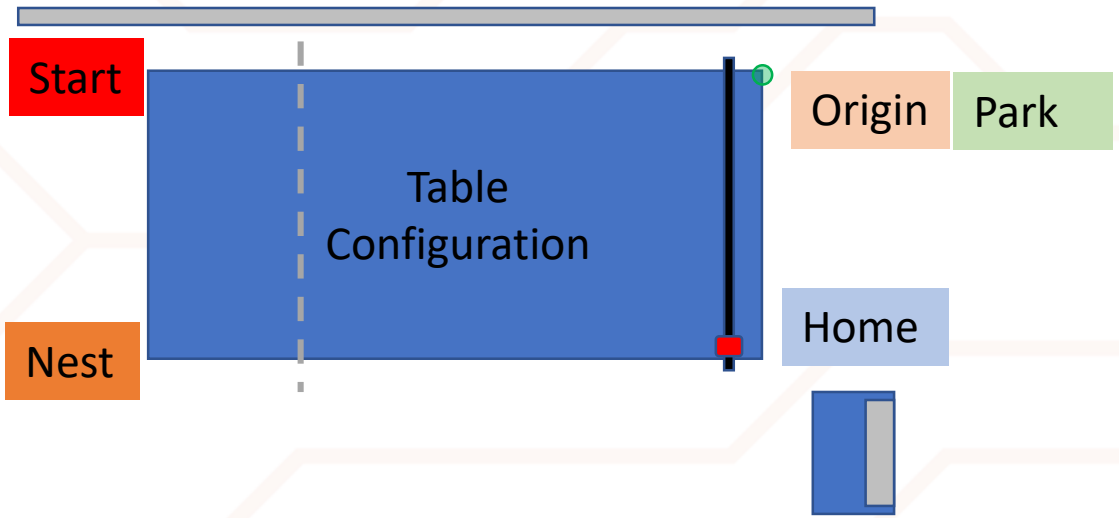
# REVIEW THE G-CODE AGAIN

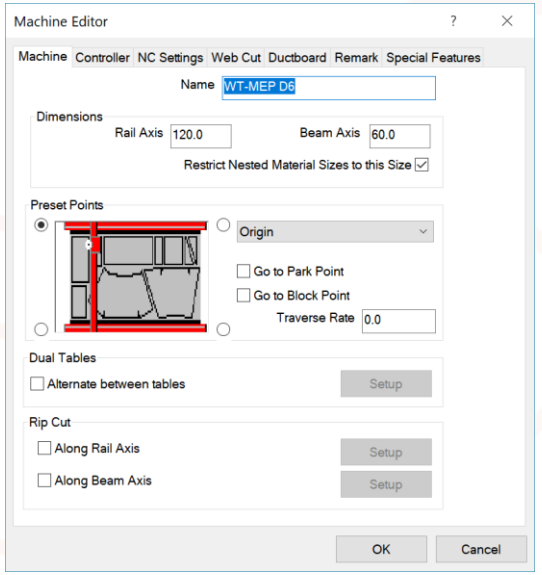
- Write the NC again
- Review the G-Code and verify that all coordinates are positive values
- At the machine control panel call the NC
- Instruct the machine for a dry run
- If the tool moves towards the center of the table then you have located the origin
- Let's say that it does

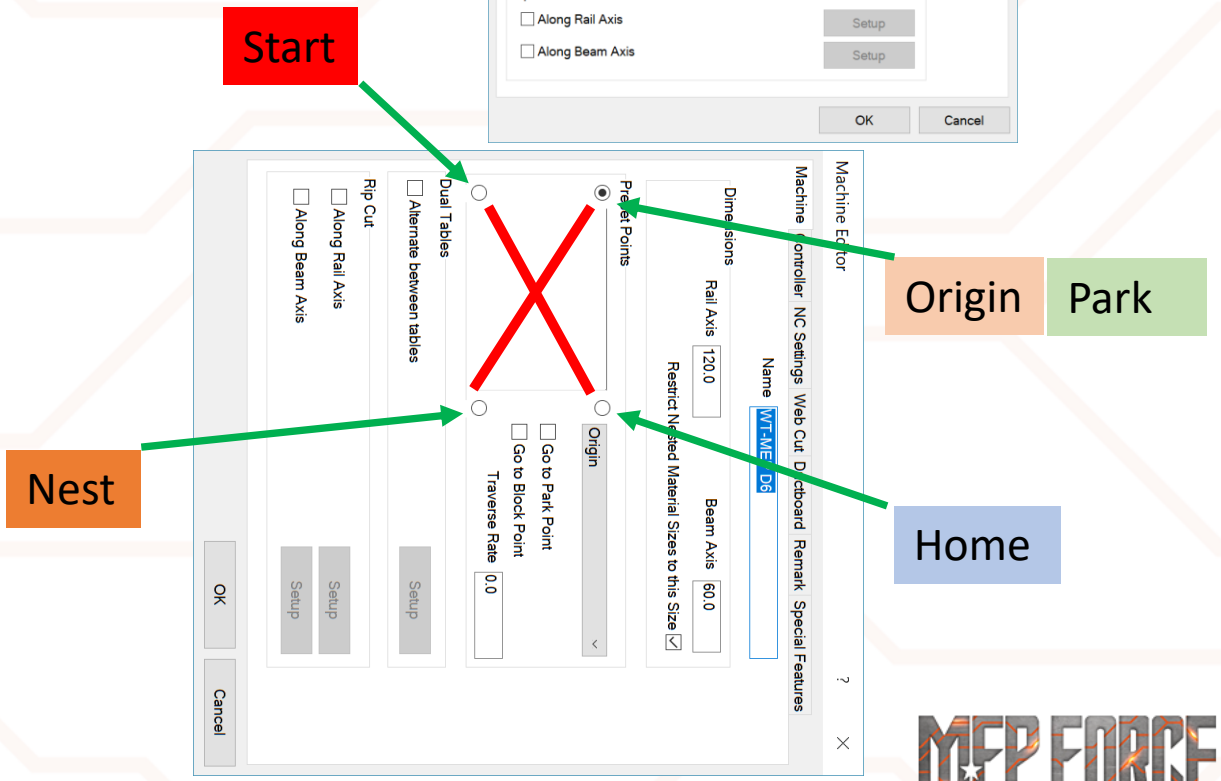
```
(1462)
(ORIGIN=TOP LEFT)
(INCH)
G70
(ABSOLUTE)
G90
(PART 1/Top Cheek)
G00X0.1Y101.9
M15
G01X0.1Y119.9
G01X18.1Y119.9
G01X18.1Y107.9
G01X12.1Y107.9
G01X12.1Y101.9
G01X0.1Y101.9
M16
(PART 1/Throat Wrapper)
G00X59.8Y107.8
M15
G01X47.8Y107.8
G01X47.8Y113.8
G01X47.8Y119.8
G01X59.8Y119.8
G01X59.8Y113.8
G01X59.8Y107.8
M16
(PART 1/Back Wrapper)
G00X12.2Y0.1
M15
G01X0.2Y0.1
G01X0.2Y18.1
G01X0.2Y36.1
G01X12.2Y36.1
G01X12.2Y18.1
G01X12.2Y0.1
M16
(PART 1/Bottom Cheek)
G00X41.7Y0.1
M15
G01X41.7Y12.1
G01X47.7Y12.1
G01X47.7Y18.1
G01X59.7Y18.1
G01X59.7Y0.1
G01X41.7Y0.1
M16
(END OF SHEET)
(END OF PROGRAM)
M02
```

# DETERMINE THE ORIENTATION OF YOUR TABLE

- Now that the origin is set you can set your other points
- The Origin and Home points are determined by the manufacturer
- Traverse Rate can be set to speed up the tool while not cutting

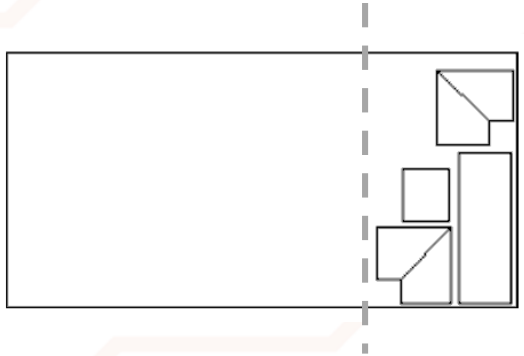


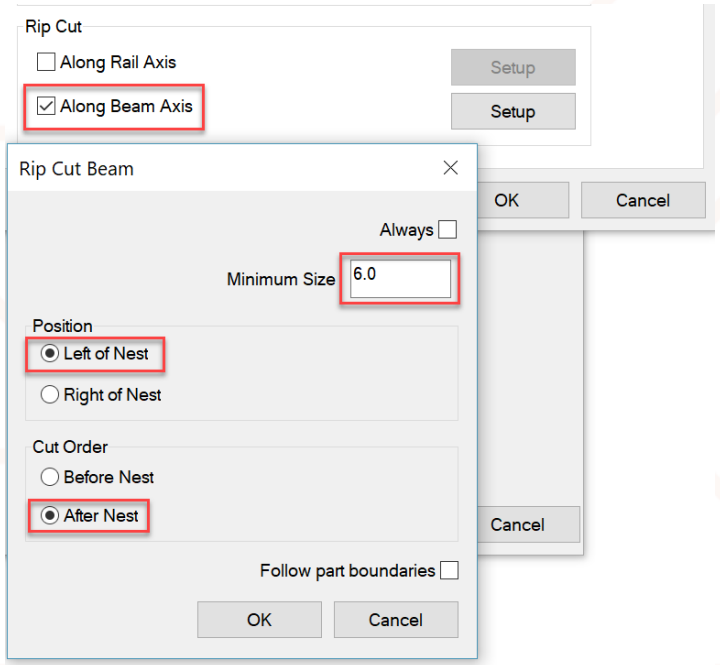



# COMPLETE THE TABLE SETUP

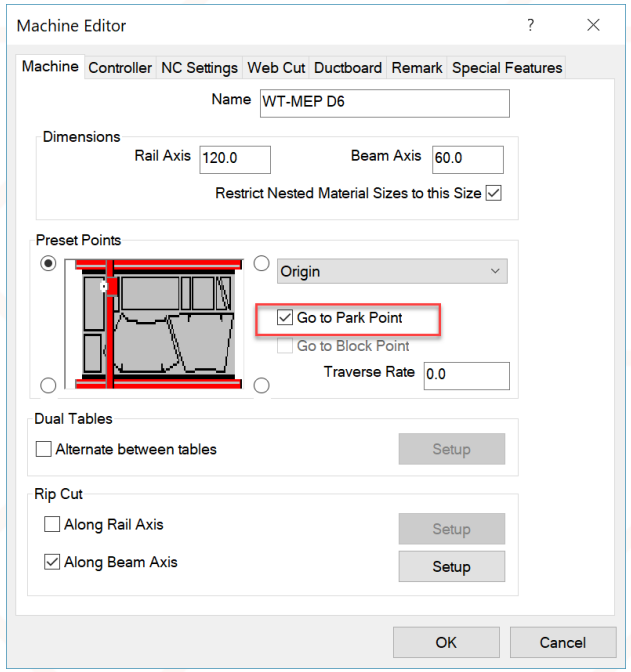
- Setup the rip cut
- If you are using sheets you will need to set the rip cut minimum length that you would want to scrap
- If you have a decoiler at the table, you will want to select “Always”



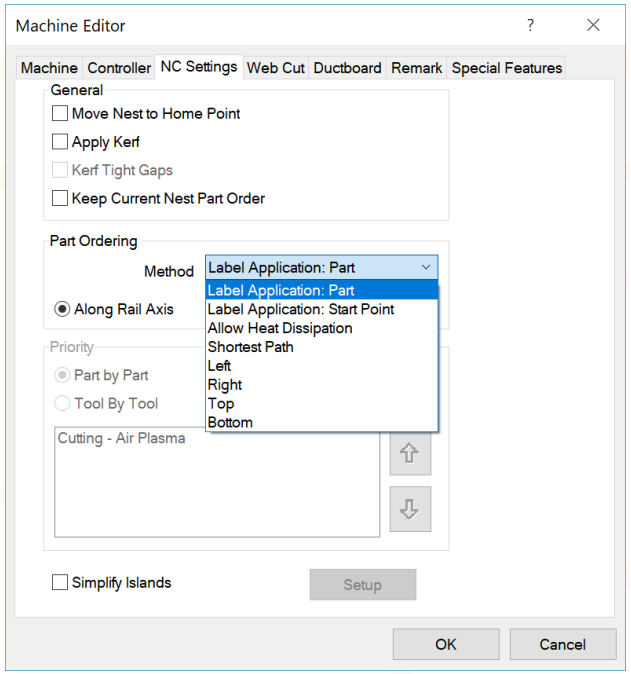


# COMPLETE THE TABLE SETUP

- Selecting “Go to Park Point” will have the tool return to the Park point after the sheet is cut.

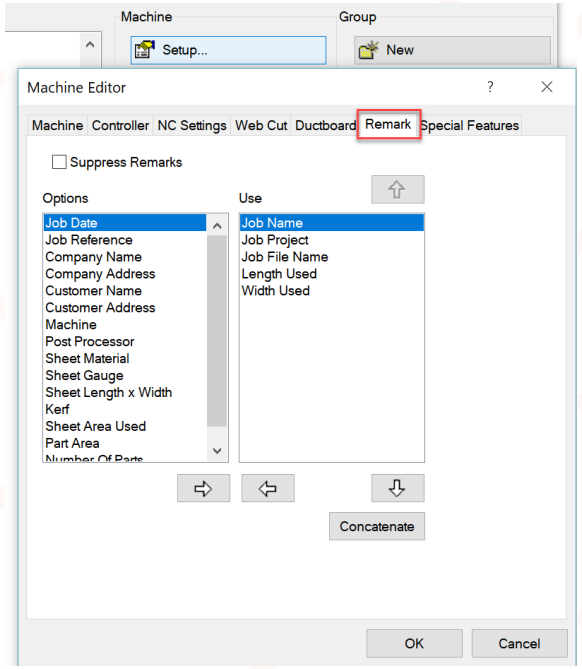



- NC Settings tab
  - Part Ordering: Load Application part is the most common: this will follow the order of the labels.
  - Shortest path may be the quickest cutting time



# COMPLETE THE TABLE SETUP

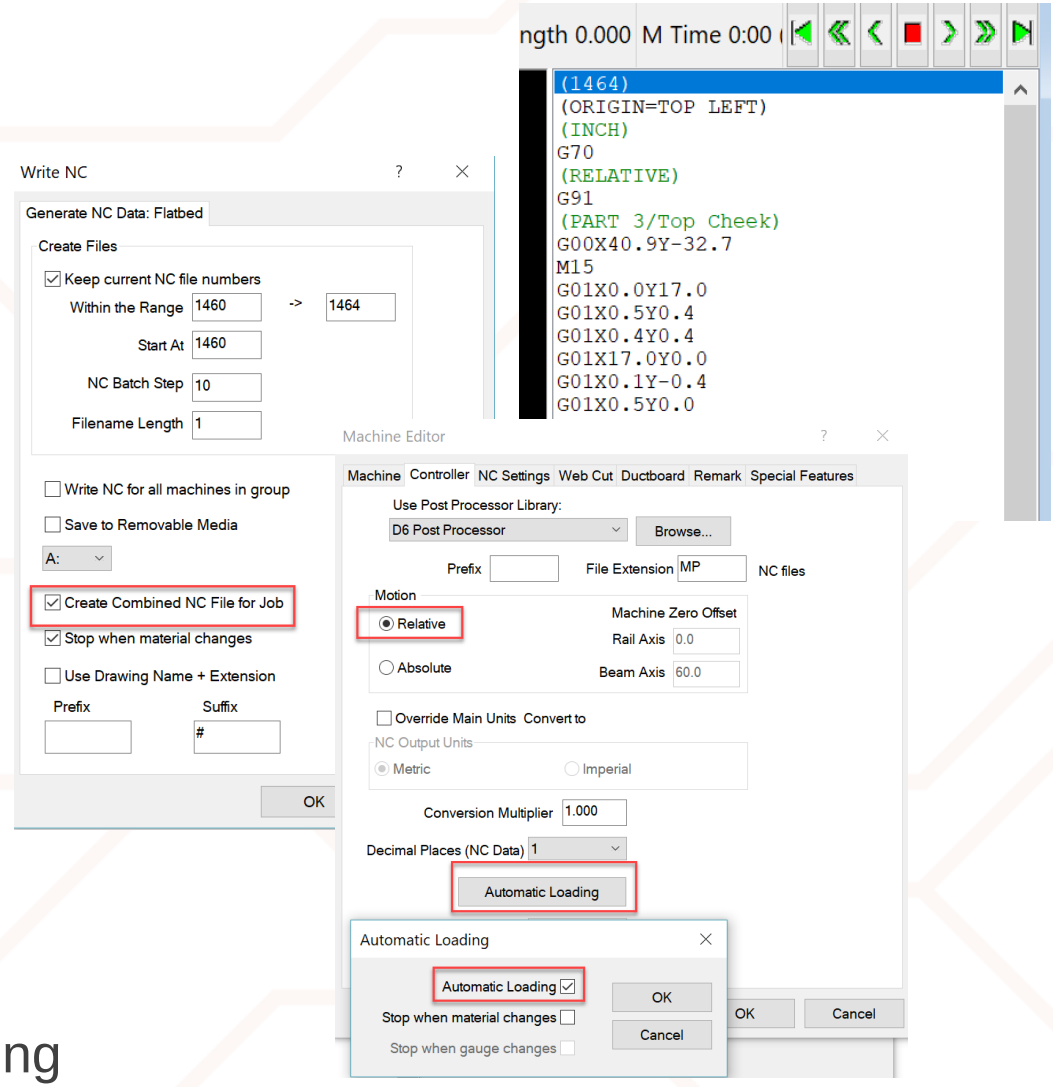
- Remark tab
  - This will add additional remarks to your NC file.
  - This can be useful if you wanted to query some data out of the file
  - Suppress Remarks will remove all remarks including Part Descriptions




```
(1464)
(Job Name=MEP Force)
(Project=)
(Job File Name=C:/Dropbox (Comfort Systems USA)/Au
(Length Used=33.680)
(Width Used=55.930)
(ORIGIN=TOP LEFT)
(INCH)
G70
(ABSOLUTE)
G90
(PART 3/Top Cheek)
G00X40.9Y87.3
M15
G01X40.9Y104.3
```

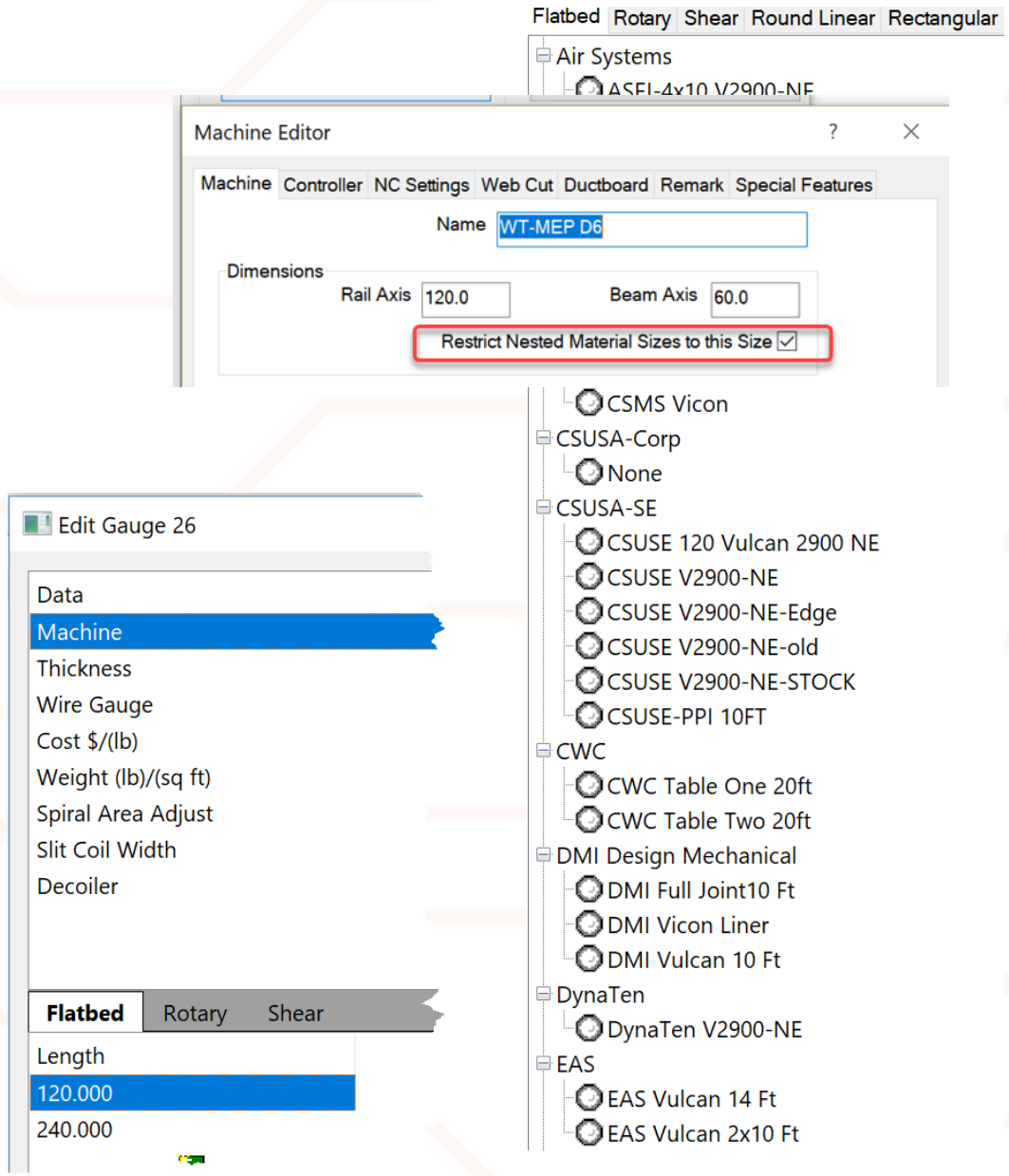
# COMPLETE THE TABLE SETUP

- Controller tab
  - You may want to change from Absolute to Relative Motion.
    - This can make it easier for verifying/troubleshooting part dimensions
    - Some Post processors will not allow relative motion
  - Automatic loading can be used with combining NCs.
    - Some Post processors will not allow Auto Loading



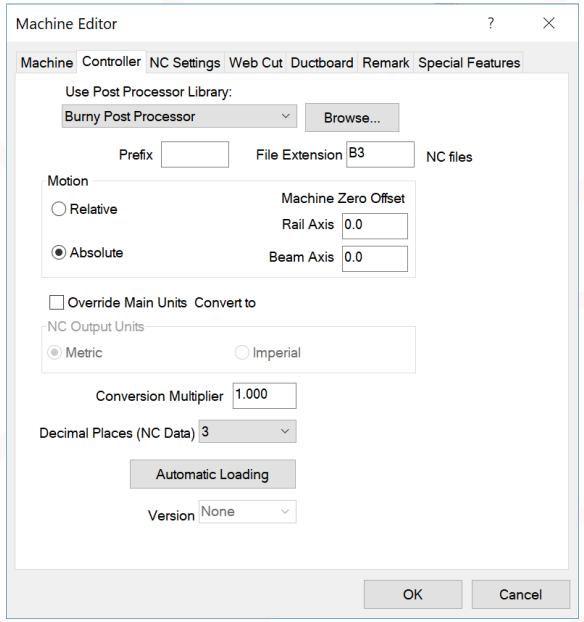
# OTHER HELPFUL INFORMATION

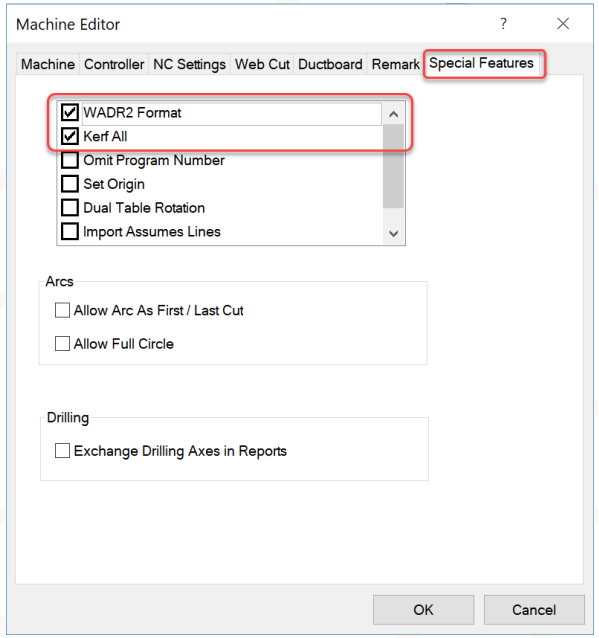
- If you have multiple tables of different sizes and multiple sheet lengths in your materials, you will want to check Restrict Nested Material to this size

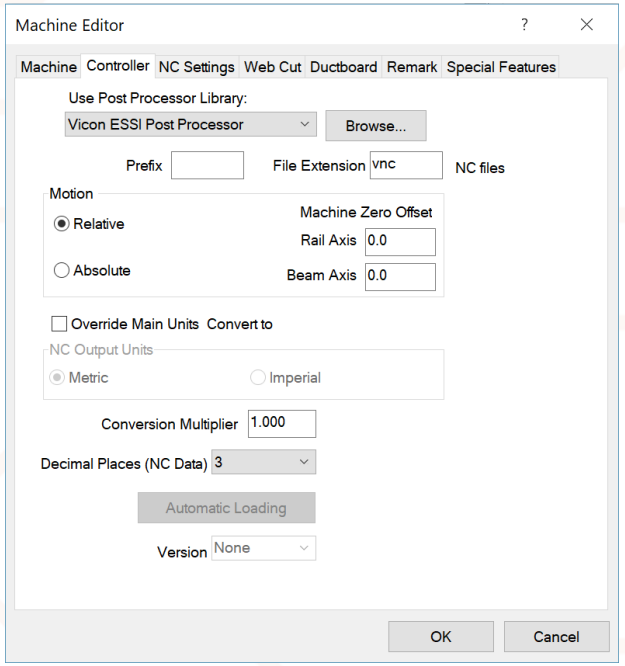


# OTHER HELPFUL INFORMATION

- Vicon Tables, you have 2 options
  1. Use the Burny Post Processor and send the nest directly to the table
  2. Use the Vicon Post Processor and the file through the Vicon cutting software for translation



# 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”