CAMCAD Premium Support - Typical Examples

Below are examples of the typical back n' forth dialog between one of our AEs and CAMCAD Premium Support Accounts. They read from bottom up.

Are you getting this level of application support?

CAMCAD Premium Support - Typical Examples

1. EXAMPLE #1

CAMCAD AE

> This part has a lot of details.   You'll need to finish with 7/16" ball all those 'pockets', and I think 30 min is not enough time.

CUSTOMER

> In your opinion do you believe that part can be machined in 30 minutes? That is the target number they have set, but I'm skeptical.

CAMCAD AE

> Yes, as long as the endmill is coated and you evacuate chips well (high pressure coolant is needed too). Remember for helical entry you should use more conventional speeds and feeds.

CUSTOMER

> Do you think 3/8ths high quality carbide will handle 600 sfm, 35 degree engagement, 50 ipm, .5" deep in 39Rc 4130 will work?

CAMCAD AE

> Here's your sample SCPRT file using a ¼" endmill on those vent ports.

CUSTOMER

> Please advise on using TRUEMill for the vent ports on this part. I made a new Cview, but didn't have any luck.

Custom Software | CAM Software | Shop Floor Technology | PostProcessors | Training | Reverse Engineering

CAMCAD Premium Support - Typical Examples

2. EXAMPLE #2

CAMCAD AE

Any endmill (flat-ball-bull) in any view. Create a 2-axis tool path in any Cview and project using the same Cview.

CUSTOMER

Would I use 3-axis project with flat end mill in top view?

CAMCAD AE

You can use 3axis cut (if this is a single surface) with ball mill, 3-axis planar if multiple surfaces, or 3-axis MultiCut with ball mill. You can also use 3-axis project of 2-axis tool path (ball mill or flat).

CUSTOMER

Can 2ax perform the cut in between the two tool paths I have shown? It is concave a little bit. Not really sure that a 3-axis routine would have any benefit in machining this tight area, what do you think?

CAMCAD AE

I think because part is big and the way the G68.1 is engage is the issue here. I'll discuss with Don to see if he can do some sort of motion analysis and see if this can be addressed that way.

CUSTOMER

It could cause problems sometimes if we position Z first every time.

CAMCAD AE

With addition of G0 Z0 are you operational? Please let me know so Don can add this in the post.

CUSTOMER

Only problem is, it posts x and y moves first, I had to add G0 Z0 line in there. I need to write a main program that will loop this program but index C 1.5 degree every time. I think this how g68.1 works, and why X is positive. This all new to me, thank you guys for putting up with me.

CAMCAD AE

If that is the case, change rapid plane to -6" or so and see if this help. And yes you can send me a video.

CUSTOMER

Can I text you a video of machine? I think codes are correct, just the starting positioning move (travel to cut area) is causing over travel condition. I can show you video if that will help us out for you to see machine motion.

CAMCAD AE

I need to talk with Don about it. You are right it doesn't look correct.

CUSTOMER

I re draw the geometry inside of Surfcam and still get the same codes posted. Looks like Cview is posting a mirror image of actual to me.

CAMCAD AE

I need to shut down for a while. I think we need to create the C-View in a very specific way. I'll work on it tonight.

CUSTOMER

Custom Software | CAM Software | Shop Floor Technology | PostProcessors | Training | Reverse Engineering

CAMCAD Premium Support - Typical Examples

> Solid works. I was thinking something was wrong with it also. Every other time cview has worked properly till this part. I did copy the spline to layer 5. Maybe I can recreate it inside Surfcam and bypass the model?

CAMCAD AE

> Can you tell us from what system this geometry is? There is chance this file may be corrupted and c-view is not recognized.

CUSTOMER

> The machine is moving away from part. Over travels up instantly. The x should be (-) in that 3d rotation shouldn't it? Since we rotated planes, x should be like mill axis, correct?

CAMCAD AE

> You are getting X in radius mode...(for milling mode)

```
O1234
(FILENAME  : CAMCAD-HD1)
(SCPRT NAME: step_lps02255-101_prt)
(MACHINE   : DMG-MORI NT4250)
(POSTREV   : 13-JAN-18)
(RUNDATE   : 05/14/2018)
#102=0.0000 (SAFETY_X)
#103=0.0000 (SAFETY_Z_SP1)
#104=0.0000 (SAFETY_Z_SP2)
G00 G20 G40 G80 M05
(--- MACHIN|PART IN MAIN SPINDLE AT STARTUP ---)
(--- BEGIN TOOL LIST ---)
(T 14 - CUSTOM 0:0.045:0)
(--- ENDOF TOOL LIST ---)
N1 (SAFE RESTART)
(PATH: 1 - SP1-HD1-2AX CONTOUR)
(VIEW: 10 - EM 144.1)
(.045 EM ROUGH 2)
M101     (SYNC CODE)
M303     (SP1 SELECT)
M45      (SP1 CAXIS ON)
G10.9 X0  (XAXIS RADIUS MODE)
G00 G40 G80 G98
(TOOL: 14 DIAM:.045 GL:0)
(DESC: CUSTOM 0:0.045:0)
T2014
G361 B-35.903 D0
G43 H14
M1
…
```

CUSTOMER

> Hi guys just checking in.

CAMCAD AE

> Delete all stuff other the one you have problems with. Try to ZIP it.

CAMCAD AE

> Please make sure your machine datum is set to View 5.

CUSTOMER

> I need a way to send you large file to look at. I created custom view, but I get very large x values in the program.

CAMCAD Premium Support - Typical Examples

3. EXAMPLE #3

CAMCAD

> Yes this (error with NCEdit) is a known problem with the editor in 2018R2 that happens with some Windows 10 systems with the latest Microsoft update. Vero knows about it. Change your editor to 2017 or 2018R1 in your PostMenu.cfg file. If you have problems please send me your {Surfcam_Data}\Config\PostMenu.cfg file and I'll try to help you.
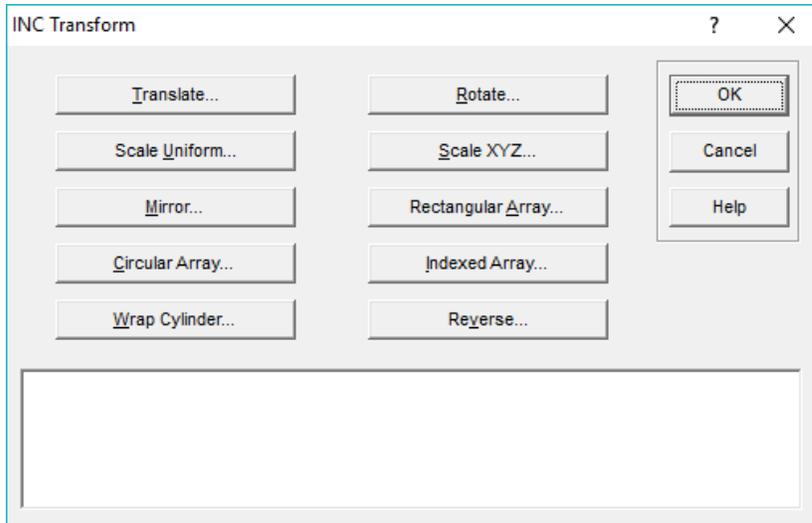
CUSTOMER

> Ok, our post doesn't like that. I'm getting a license error for editor every time I post process something now since the 2018r2 update.

CAMCAD Premium Support - Typical Examples

4. EXAMPLE #4

CAMCAD

CUSTOMER

> Last one for today. How do I wrap letters around a cylinder like with 4ax project? I attached file of exhaust tip I'm making.

Custom Software | CAM Software | Shop Floor Technology | PostProcessors | Training | Reverse Engineering

CAMCAD Premium Support - Typical Examples
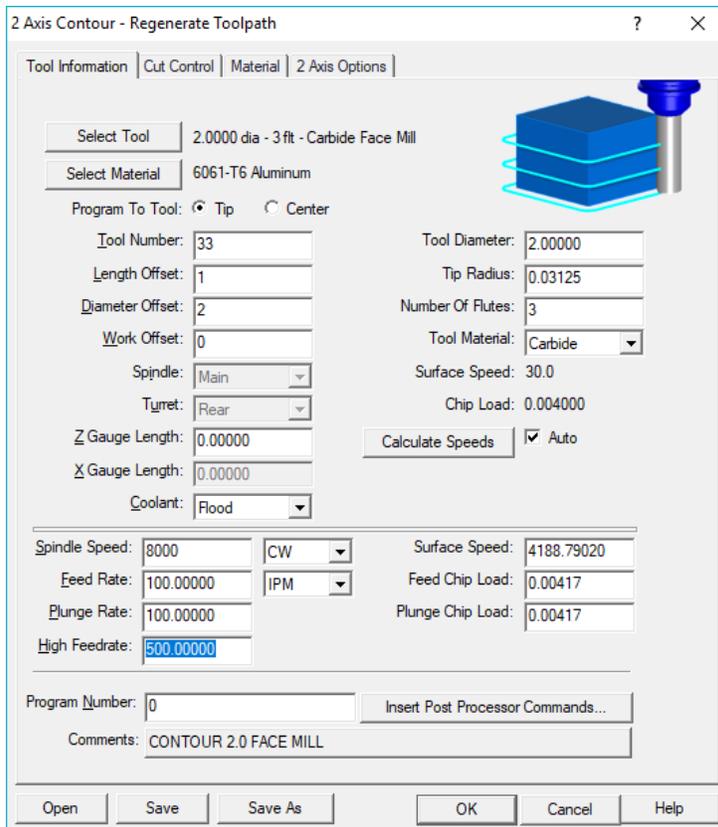
5. EXAMPLE #5

CAMCAD

> Good observation - TRUEMill is all arcs and this is point to point. Fanuc has codes for High Velocity Machining with settings for rough, semi-rough and finish G05 or G05.1 with a P parameter for settings that will smooth the motions.

CUSTOMER

> Also, I should mention, TRUEMill 500ipm rapid moves are very very smooth. Same controller and machine.
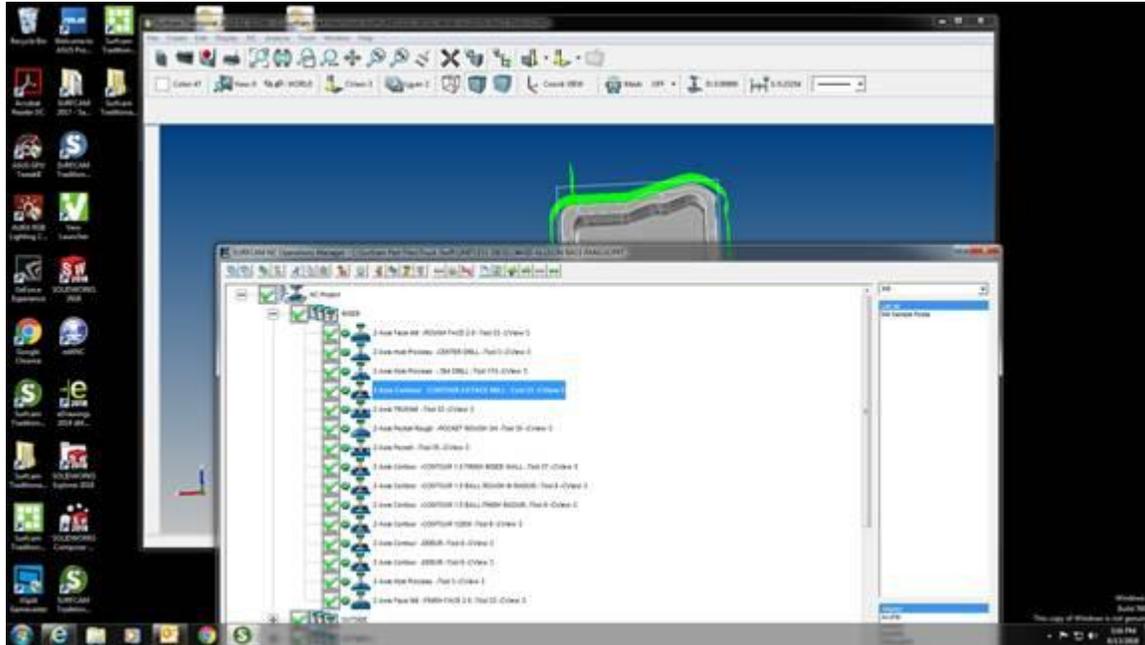
CAMCAD

> There is nothing wrong with your tool path. Those Y-Z axis 'arcs' are point to point and you programmed those with a 500 ipm (high feedrate). My guess is your controller can't handle 500 IPM with point to point (those motions are not tangent) if this is an older controller. I think maximum programable feedrate is 394 IPM. Try to programming this cut with ramping.

CUSTOMER

> I'm getting jerky machine movement in the z retract and plunge on this high speed 2ax contour. What settings do I have screwed up this time.

Custom Software | CAM Software | Shop Floor Technology | PostProcessors | Training | Reverse Engineering

CAMCAD Premium Support - Typical Examples

CAMCAD Premium Support - Typical Examples

6. EXAMPLE#6

CAMCAD

You have a lot of double geometry and you didn't have 'UNDERCUT' checked.

CUSTOMER

Hi, I am trying to cut the orange colored "teeth" on this part, I have tried top view and making a new cview, when I give tool the proper radius it doesn't want to profile the contour. If I leave tool nose sharp, it does the profile however this would leave part geometry wrong. What should I do?

Custom Software | CAM Software | Shop Floor Technology | PostProcessors | Training | Reverse Engineering