GibbsCAM

QUICK START GUIDE

Document Control

View Control

Visibility Control

Selection Control

Tool Tile (on/off)

CAM Tile (on/off)

Operation Simulation

Post Processor (GCODE)

Tool Tiles

Process Tiles

Operation Tiles

Origin (x=0,y=0,z=0)

Stock Boundary

Parametric Solid

Gc GibbsCAM®
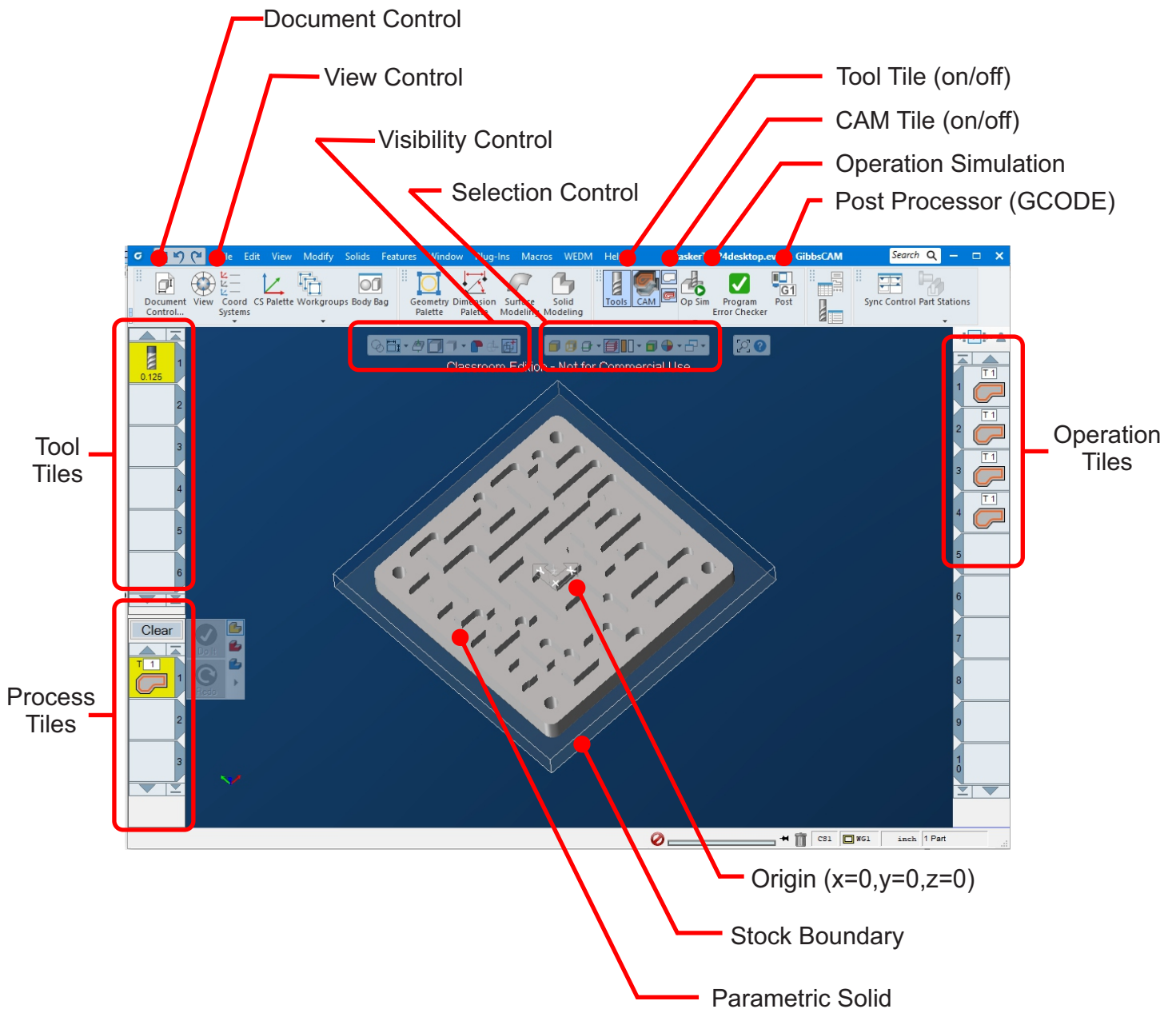
BU College of Engineering
Engineering Product Innovation Center

GibbsCAM

QUICK START GUIDE

LAUNCH GIBBSCAM
GibbsCAM License Window
1. Check Out License (select all, ensure boxes are checked)
2. Select "OK"

ProAXYS License Manager
1. Select "OK"

CREATE DOCUMENT
Pull down menu -> File / New Document
• Assign name and location for file to be saved

IMPORT PARAMETRIC SOLID
Pull down menu -> File / Import
• Select file from saved location
• NOTE: if you don't see your file in the expected location, confirm that filter selection is set to "All Files (*.*)"

ALIGN GEOMETRY WITH COORDINATE SYSTEM
• Select face of model to orient
• right click face / select "align to CS" (align to coordinate system)

Align bottom face of model with Z=0
• Modify pull down menu / translate
• enter value face must move to place at Z=0
• Select "do it"

Modify stock to match parametric solid geometry
• Modify pull down menu / Shrinkwrap

ESTABLISH CLEARANCE HEIGHT
Assign clearance height for tools to travel above part geometry
• Select "Document Control" button
• Assign absolute clearance value .100" above the absolute part height

DEFINE DRIVE GEOMETRY
• Right click face that contains drive geometry
• Extract edges
• "Do it"

ADD TOOLS, PROCESSES AND OPERATION
Select "tools" button from top banner icons
• This will reveal tiles along the left side of the screen

Select "CAM" button from top banner
• This will reveal tiles along the lower left side of the screen for processes
• This will reveal tiles along the right side of the screen for operations

ASSIGN TOOL PARAMETERS (Spindle RMP and tool feedrate)
• Double click tile 1 (top tile) and assign values for tool 1
• Double click tile 2 (2nd tile) and assign values for tool 2, etc.

See following chart for reference based on assigned tool numbers and speed / feed recommendations

Gc GibbsCAM®

BU College of Engineering
Engineering Product Innovation Center

GibbsCAM

QUICK START GUIDE

NOTE:

Check your tool path clearance planes
(.1" above stock height)

Check your fixturing clearances
(avoid putting clamps in tool paths)

ASK if you are cutting a different material

ALWAYS use a ramping tool path for hard stock

There are other tools in the machines,
ALWAYS ask for the feed/speed before using

ALUMINUM

TOOL	SPEED RPM	ENTRY FEED	CONTOUR FEED	Z STEP
1/8" End Mill	5000	16 ipm	18 ipm	.04"
1/4" End Mill	4500	18 ipm	20 ipm	.1"
3/8" End Mill	4000	18 ipm	20 ipm	.125"
1/2" End Mill	3500	18 ipm	20 ipm	.1875"
90 Spot Drill	3000	10 ipm	---	---

HDPE

TOOL	SPEED RPM	ENTRY FEED	CONTOUR FEED	Z STEP
1/8" End Mill	5000	18 ipm	20 ipm	.0625"
1/4" End Mill	4500	22 ipm	25 ipm	.125"
3/8" End Mill	4000	22 ipm	25 ipm	.1875"
1/2" End Mill	3500	22 ipm	25 ipm	.25"
90 Spot Drill	3000	10 ipm	---	---

ACRYLIC

TOOL	SPEED RPM	ENTRY FEED	CONTOUR FEED	Z STEP
1/8" End Mill	4465	15 ipm	17 ipm	.0625"
1/4" End Mill	4250	19 ipm	21 ipm	.125"
3/8" End Mill	4040	19 ipm	21 ipm	.1875"
1/2" End Mill	3825	19 ipm	21 ipm	.25"
90 Spot Drill	2600	8 ipm	---	---

CARBON STEEL (WROUGHT)

TOOL	SPEED RPM	ENTRY FEED	CONTOUR FEED	Z STEP
1/8" End Mill	3973	7.95 ipm	7.95 ipm	.04"
1/4" End Mill	1986	5.96 ipm	5.96 ipm	.083"
3/8" End Mill	1324	3.97 ipm	3.97 ipm	.125"
1/2" End Mill	993	2.98 ipm	2.98 ipm	.167"
90 Spot Drill	535	1.07 ipm	---	---

POLYCARBONATE

TOOL	SPEED RPM	ENTRY FEED	CONTOUR FEED	Z STEP
1/8" End Mill	5250	18 ipm	20 ipm	.0625"
1/4" End Mill	5000	22 ipm	25 ipm	.125"
3/8" End Mill	4750	22 ipm	25 ipm	.1875"
1/2" End Mill	4500	22 ipm	25 ipm	.25"
90 Spot Drill	3000	10 ipm	---	---

Gc GibbsCAM®

BU College of Engineering
Engineering Product Innovation Center

GibbsCAM

QUICK START GUIDE

CREATE MACHINING PROCESS
Machining processes define specific parameters of an operation.

Drag the desired tool from the tool tiles into a tile on the lower left side of the screen within the Operations tiles.
• select the operation type
• select tool heights
• define entry / exit moves, etc

CREATE MACHINING OPERATION
A machining operation applies a machining process to a specific piece of geometry.
The operation can then be post-processed to GCode in order to drive a CNC machine.

With a machining process selected, use the cursor to select drive geometry
• click on the drive geometry
• select proximity of tool to geometry (left / right / centered)
• select the direction of travel
• select "do it"
• an operation should now appear in the operations set of tiles on the right side of the screen

SIMULATE OPERATION
Simulating an operation visually confirms the operation runs as expected.

Select "Op Sim" button
Configure the variable to control:
• tool visibility
• stock transparency
• fixture transparency
• tool holders,.. and various other attributes

Adjust the speed control of the simulation
• Rewind / Play / Fast forward to help visualize the machining operation

TIP: To view model geometry rather than stock during simulation, select the model geometry before pressing the "Op Sim" button

CONVERT THE TOOLPATHS INTO GCODE
"GCODE" describes the code that directs a machine tool run your toolpaths. The process requires running a machine specific post-processor to convert the toolpaths into a code that the machining center can understand.

Select the part process tile(s) that you want to run
Press the "G1-Post" Button
Select the post-processor associated with the machine you'd like to operate
• Name the .NC file and identify the location to save the file
• Select the "process" button

This should generate a GCode file that auto-launches a GCode editor to review your file.

TRANSFER FILE TO MILLING MACHINE
Some machines in EPIC access GCODE files from a local server, and others from USB drives.
See an EPIC staff member if unsure of next steps.

Gc GibbsCAM®

BU College of Engineering
Engineering Product Innovation Center

GibbsCAM

QUICK START GUIDE

FAQ

Q: How do I modify an existing operation?
A: Double click the operation tile to re-activate the applied machining process. The process can then be modified,.. and by pressing the "redo" button, the revised process parameters are applied to the operation.