File Preparation for the Makino Wire EDM

Files provided for the Makino Wire EDM need to be provided as Part, Solid Part, STEP, IGES or IGS format. Dimensions need to be in millimetres (mm).

Please see following images for file export from Fusion 360, Solid Works and Inventor Programs. The image is a generic Image from NX 10, the main Camming Software used in the Digital Manufacturing Lab.

The Makino Wire EDM requires design orientation to take place in the XY orientation, with Z as depth / Height.



Fusion 360:



With Fusion 360 part needs to be assigned to home position on coordinates, this is start from 0 point of axis's. The Part still requires to be orientated in the XY plane. This can either be from beginning of modelling or on completion through the Move Function.

Once part is ready for export, select File (Displayed as ear-tagged page in image). Then select '**Export...**' using left key on mouse.

3	New Design				
	New Design From File				
V	New Drawing				
	Related Data				
F	Open Details on Web				
l	Save Ctrl+S				
Ē	Save as				
]	Recover Documents (3)				
Г	Export				
Ē	Live Review Session				
١	Share 🕨				
	3D Print				
	Capture Image				
	View 🕨				
	Scripts and Add-Ins				
	Fusion 360 App Store				
	Add-Ins (Legacy)				

This open the dialogue page for exporting part file to another location such as a thumb drive. Using the drop-down arrow under '**Type:**' select IGES File. Selecting the '...' next to original file location will open '**File browser**' to choose location for part to be saved to, eg Thumb-drive. Then simply hit '**Save**' and IGES will be created in designated file.

Export		×				
Name:						
Gear						
Туре:						
IGES Files (*.igs *.iges)	-					
Save to a project in the e	loud					
Liam's First Project > O master						
Save to my computer	G:/Liam CAine July16					
		Cancel				

Autodesk Inventor:

With Autodesk Inventor part needs to be assigned to home position on coordinates, this is start from 0 point of axis's. The Part still requires to be orientated in the YZ plane with X as Depth / Height. This can either be from beginning of modelling or on completion through the Move Function.



Once part is ready for export, select 'File'. Then select 'Export', 'CAD Format' using left key on mouse.



A new window will open, at bottom of page select 'Save as type', and then select 'STEP' using left key on mouse. Then select 'Save'.

	AutoCAD DWG Files (*.dwg)		
	CATIA V5 Part Files (*.CATPart)		
	IGES Files (*.igs;*.ige;*.iges)		
	JT Files (*,jt)		
	OBJ Files (*.obj)		
	Parasolid Binary Files (*x_b)		
	Parasolid Text Files (* x_t)		
	Pro/ENGINEER Granite Files (".g)		
	Pro/ENGINEER Neutral Files (*.neu*)		
	SAT Files (".sat)		
	SMT Files (".smt)		
	STEP Files (".stp;".ste;".step;".stpz)		
D 1	STL Files (".stl)		
File name:	XGL Hies ("xgl)		
	ZGL Files (".zgl)		
Save as type:	STEP Files (*.stp;*.ste;*.step;*.stpz)		

Solid Works:

TBA. Contact a technician for assistance.