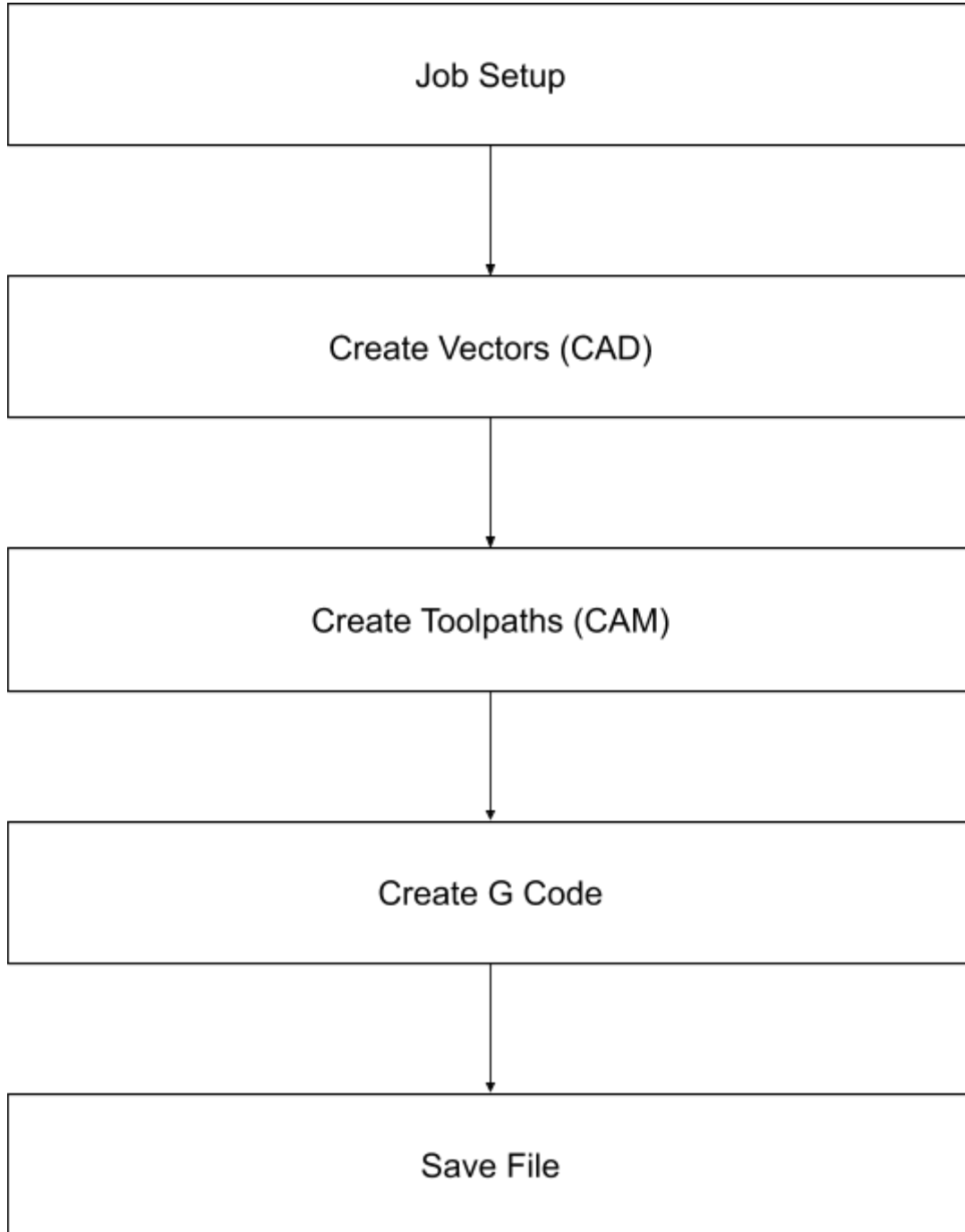


Getting Started Programming - Class 1

3-Axis Programming Sequence



Getting Started Programming - Class 1

CAD (Computer-Aided Design)

Job Setup

Starting a New File

- Open VCarve or Aspire
- Click the "Create a new file" icon - Startup Task section
- Select "Single Sided" - Job Type section
- Enter Width (x) 12, Height (y) 6 - Job Size section
- Enter Thickness (z) 0.75 - Job Size section
- Select "inches" - Job Size section
- Select "Material Surface" - Z Zero Position section
- Select the bottom left corner - XY Datum Position section
- Click the "OK" button

Create Vectors

Draw a Rectangle

- Click the "Draw Rectangle" icon - Create Vectors section
- Select "Radiused Internal" - Corner Type section
- Enter "0.75" in the Radius field - Corner Type section
- Enter "10" in the Width (X) field - Size section
- Enter "4.0" in the Height (Y) field - Size section
- Click the "Create" button
- Click the "Close" button
- Select the rectangle
- Click the "Align Selected Objects" icon - Transform Objects section
- Click the "Center both x and y" icon - Align to Material section
- Click the "Close" button

Draw Text

- Make sure the rectangle is selected
- Click the "Draw Text - within a vector box" icon - Create Vectors section
- Enter your name in the Text field box
- Select "Times New Roman" from the Font drop-down box
- Select "Normal" - Margin Size section
- Click the "Close" button

Getting Started Programming - Class 1

CAM (Computer-Aided Manufacture)

Create Toolpaths

Material Setup

- Click the “Switch to toolpath commands” icon - top left corner of the screen
- Click the “Set” button - Material Setup section
- Enter “0.2” in the Clearance (Z1) field - Rapid Z Gaps above Material section
- Enter “0.1” in the Plunge (Z2) field - Rapid Z Gaps above Material section
- Enter “0.5” in the “Z Gap above Material” Field - Home / Start Position section
- Click the “OK” button

Create a Profile Toolpath

- Select the rectangle
- Click the “Profile Toolpath” icon - Toolpath Operations section
- Enter “0.125” in the Cut Depth field - Cutting Depths section
- Check “Show advanced toolpath options” - Cutting Depths section
- Click the “Select” button - Tool section
- Select a 90-degree V-Bit - Tool List
- Click the “Select..” button
- Click the “Edit...” button - Tool section
- Enter “2” in the No. Flutes field - Geometry section
- Enter “24000” in the Spindle Speed field - Feeds and Speeds section
- Select “inches/min” in the Feed Units drop-down box - Feeds and Speeds section
- Enter “100” in the Feed Rate field - Feeds and Speeds section
- Enter “100” in the Plunge Rate field - Feeds and Speeds section
- Enter “1” in the Tool Number field
- Click the “OK” button
- Select “On” - Machine Vectors... section
- Select “Climb” Direction - Machine Vectors... section
- Click the “Ramp” tab
- Select “Add ramps to toolpath
- Select “Smooth” - Type section
- Select “Angle” - Specify Ramp... section
- Enter “20” in the degrees field - Specify Ramp... section
- Enter “Border” in the Name field
- Click the Calculate” button
- Click the “Preview Selected Toolpath” button
- Click the “Close” button

Getting Started Programming - Class 1

Create Toolpaths (continued)

Create a V-Carve Toolpath

- Click on the “2D View” tab - top left-hand corner of the screen
- Select the text
- Click the “V-Carve / Engraving Toolpath” icon - Toolpath Operations section
- Make sure “Flat Depth” is unchecked - Cutting Depths section
- Click the “Select” button - Tool: section
- Select the same 90-degree V-Bit - Tool List
- Click the “Select...” button - Tool section
- Click the “Edit...” button - Tool section
- Enter “2” in the No. flutes field - Geometry section
- Enter “24000” in the Spindle Speed field - Feeds and Speeds section
- Enter “100” in the Feed Rate field - Feeds and Speeds section
- Enter “40” in the Plunge Rate field - Feeds and Speeds section
- Click the “OK” button
- Enter “Text” in the Name Field
- Click the “Calculate” button
- Click the “Preview Selected Toolpath” button
- Click the “Close” button

Create G Code

Save Toolpaths

- Check “Toolpaths” to select both toolpaths
- Click the “Save Toolpaths” icon - Toolpath Operations section
- Check “Output all visible toolpaths to one file”
- Select “Legacy 3 Axis CV Arcs (inch)(* .txt)” from the Post Processor drop-down box
- Click the “Save Toolpath(s)...” button
- Select the “Documents” fold
- Select the “G Code” folder (Create this folder if necessary)
- Enter “Name Plaque G Code” in the File name: field
- Click the “Save” button
- Click the “Close” button

Save File

Save Vectric File

- Click “File” on the top menu bar
- Click “Save” from the drop-down menu
- Select the “Documents” folder
- Select the “Parts” folder (Create this folder if necessary)
- Enter “Name Plaque” in the File name: field
- Click the “Save” button