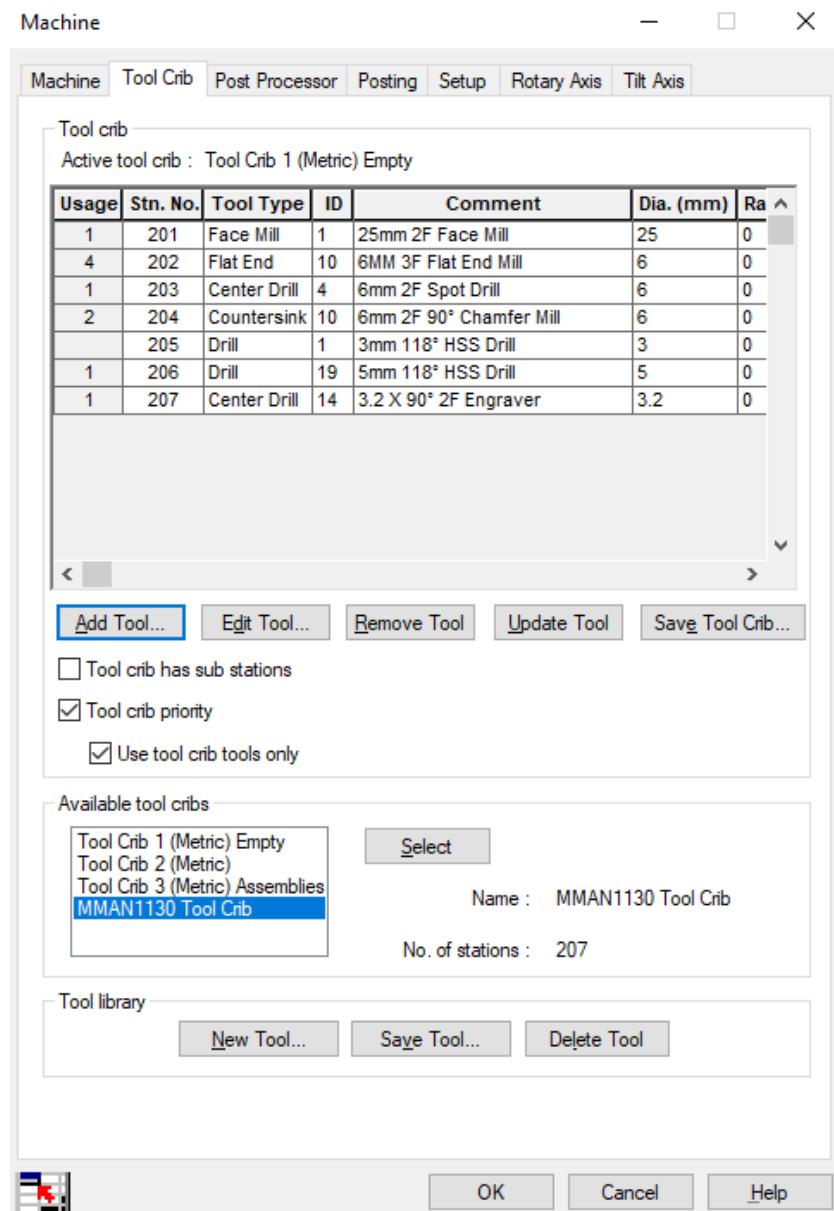


## MMAN1130 CAM Checklist

This document provides a series of important checks to ensure that your CNC Machining assessment parts are ready to be machined. Carefully adhering to these rules will minimise your chances of breaking an endmill or erroring out the machine. Don't forget to watch the MMAN1130 CNC Machining Setup video to get your part ready for machining!

### Preparing your set up (Click Define Machine):

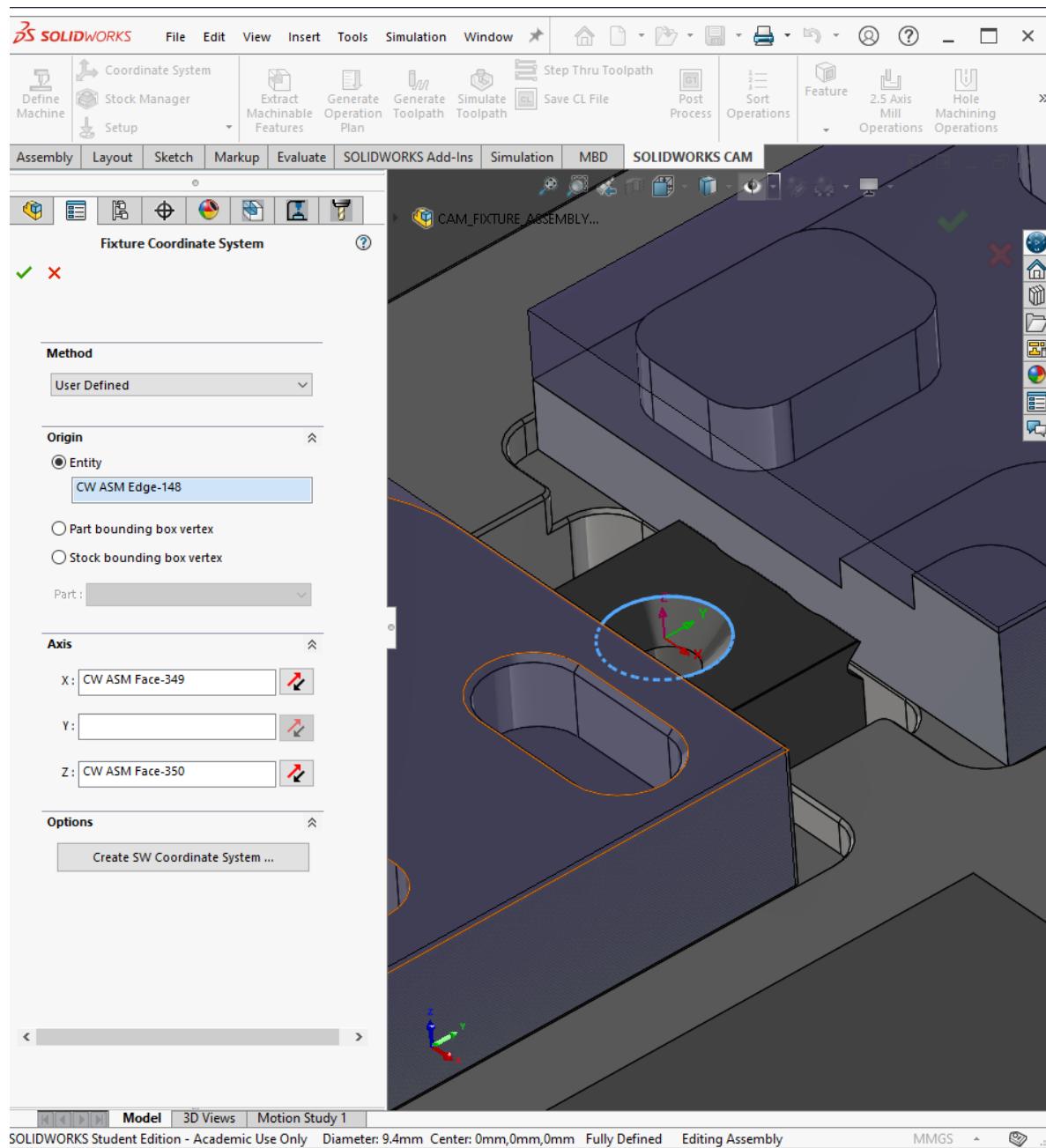
- In “Machine” tab, select “Mill – Metric”
- In “Tool Crib” tab, ensure you have
  - Selected “MMAN1130 CNC Assessment Tool Crib” for use, hit select to be safe!
  - “Tool crib priority” is ticked
  - “Use tool crib tools only” is ticked
- Ignore “Post processor”, “Posting”, “Setup”, “Rotary Axis” and “Tilt Axis” tabs.



## Coordinate System

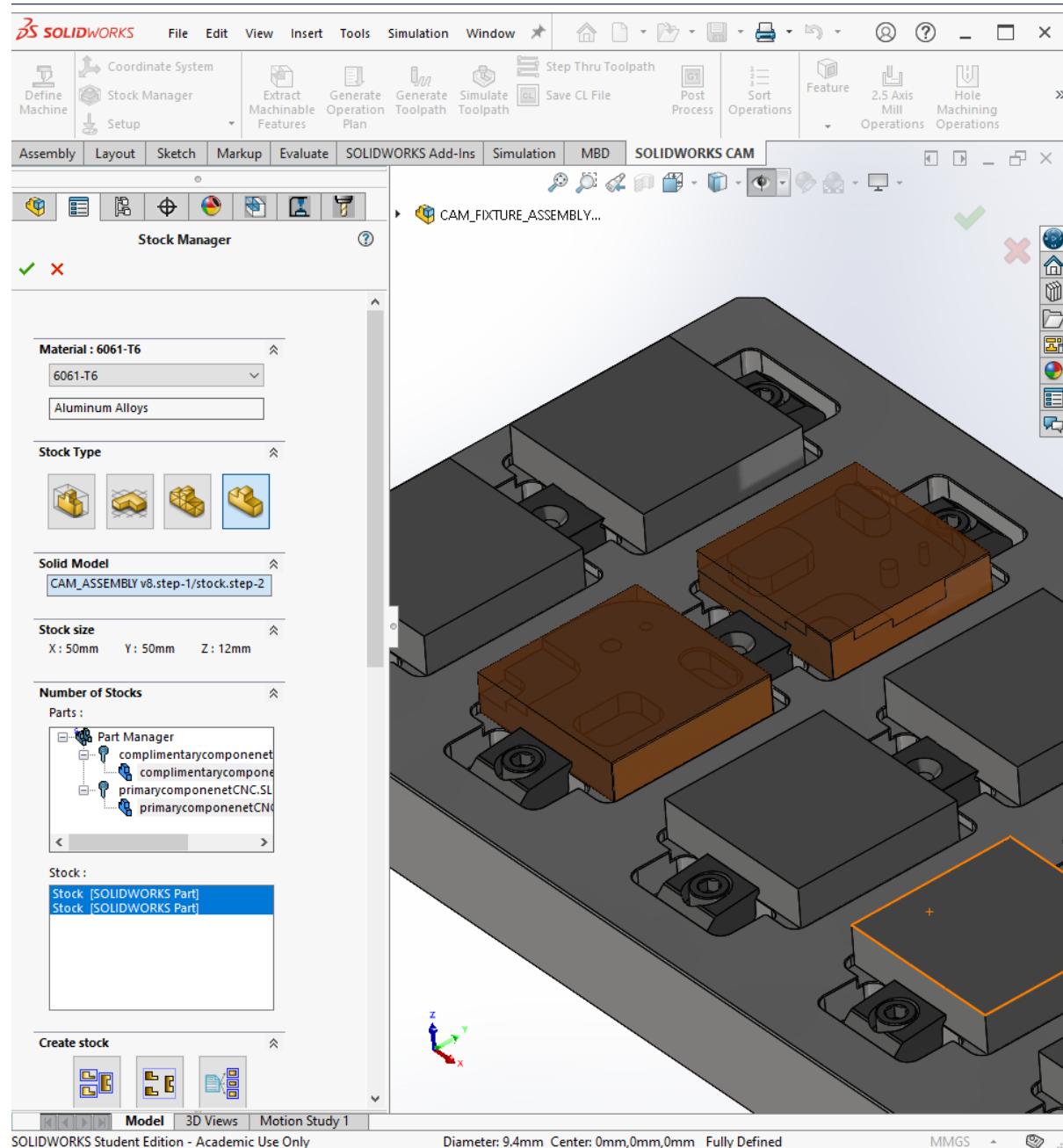
This is an extremely important step. Ensure that you have selected the correct placement for the origin. Failure to do this correctly will cause your job to error out and attracts a penalty to your CAM mark.

Fixture Coordinate System must be exactly as shown in the video and image below.



## Stock Manager

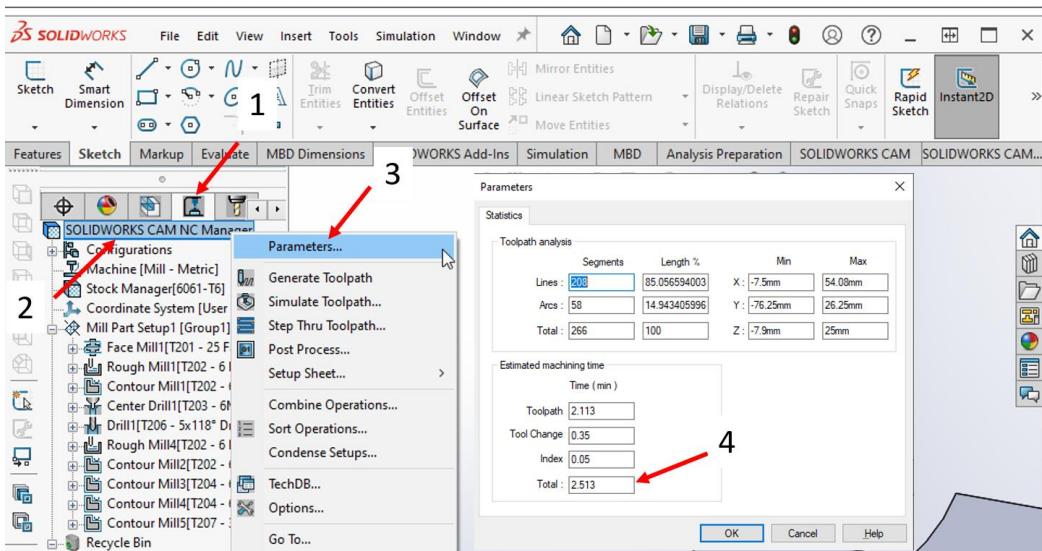
Stock set up is a little tricky for the assessment. Because we are machining two parts and are utilising a fixture assembly, you cannot simply pick bounding box. Please ensure that you watch the MMAN1130 CNC Machining Setup video to get the stock manager working.



## Checking Machining Time

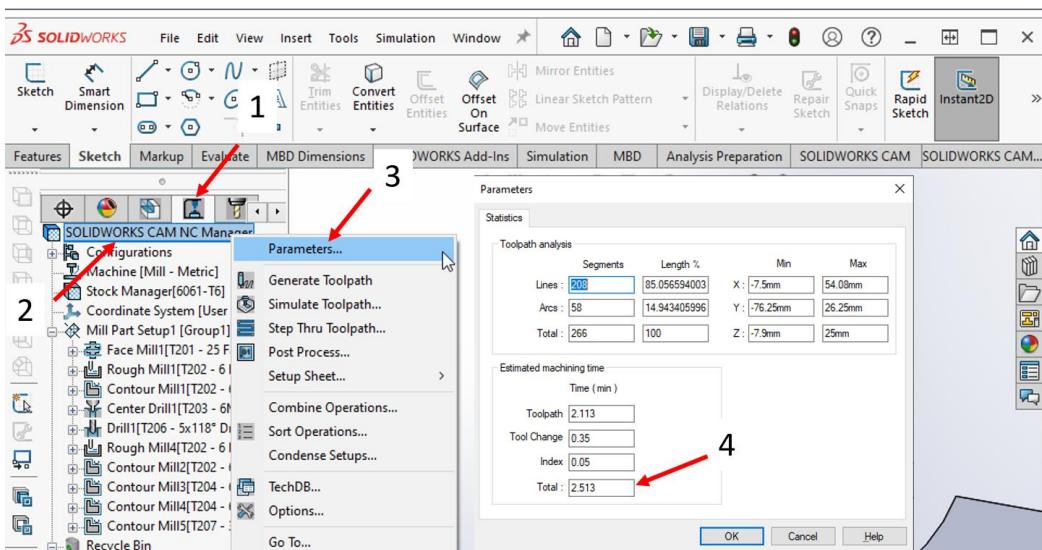
It is very important to be checking your total machining time. If your machining time exceeds a certain amount, your file will not pass the manufacturability review. Luckily, it is quite easy to check the machining time.

1. Ensure that you are in the CAM Operations Tab
2. Right click on “Machine[Mill-Metric]”
3. Click Parameters
4. You can find your estimated machining time here.



Note, you can also check the machining time for a single part. The process is similar but different to the CNC machining assessment assembly file.

1. Ensure that you are in the CAM Operations Tab
2. Right click on “SOLIDWORKS CAM NC Manager”
3. Click Parameters
4. You can find your estimated machining time here.



## **Checklist while doing CAM Programming**

For all operations, please ensure that you simulate and review the toolpaths carefully. Failure to do so may result in an tool breaking. This will cause you to fail the manufacturability review and penalty marks for the CAM component will apply.

**SIMULATE, SIMULATE, SIMULATE, SIMULATE!!!**

### **Facing Operations**

- In “Tool” tab, make sure “25mm 2F Face Mill” is showing in the comment.
- In “F/S” tab, make sure “Tool” is selected for “Defined by”.
- In “Facing” tab, ensure that the depth of cut never exceeds 0.5mm.
- In “Leads/Links” tab, ensure that the “Feedrate” in “Stepover <= link threshold” is set to 800mm/min

### **Roughing Operations**

- In “Tool” tab, make sure “6MM 3F Flat End Mill” is showing in the comment.
- In “F/S” tab, make sure “Tool” is selected for “Defined by”.
- In “NC” tab, ensure that CNC compensation is turned OFF.

### **Contouring Operations**

- In “Tool” tab, make sure “6MM 3F Flat End Mill” is showing in the comment.
- In “F/S” tab, make sure “Tool” is selected for “Defined by”.
- In “NC” tab, ensure that
  - CNC compensation is turned ON.
  - Toolpath Center can either be WITH compensation or WITHOUT
    - If WITHOUT, ensure “Gouge check” and “Add tool radius to leadin/leadout” is ticked

### **Center Drilling Operations**

- In “Tool” tab, make sure “6mm 2F Spot Drill” is showing in the comment.
- In “F/S” tab, make sure “Tool” is selected for “Defined by”.
- In “Center Drill” tab, ensure that
  - “Dwell” is set to zero (0).
  - “Canned cycle output” IS NOT ticked
- In “Feature Options” tab, machining depth is set to no more than 1mm.

### **Drilling Operations**

- In “Tool” tab, make sure “5mm 118° HSS Drill” or “3mm 118° HSS Drill” is showing in the comment.
- In “F/S” tab, make sure “Tool” is selected for “Defined by”.
- In “Drill Hole Parameters” tab, ensure that
  - In the “Type” dropdown box, select “Drilling”
  - “Dwell” is set to zero (0).
  - “Canned cycle output” IS NOT ticked

### **Chamfering Operations**

- In “Tool” tab, make sure “6mm 2F 90° Chamfer Mill” is showing in the comment.
- In “F/S” tab, make sure “Tool” is selected for “Defined by”.
- In “NC” tab, ensure that
  - CNC compensation is turned ON.
  - Toolpath Center can either be WITH compensation or WITHOUT
    - If WITHOUT, ensure “Gouge check” and “Add tool radius to leadin/leadout” is ticked

### **Edge Breaking Operations**

- In “Tool” tab, make sure “6mm 2F 90° Chamfer Mill” is showing in the comment.
- In “F/S” tab, make sure “Tool” is selected for “Defined by”.
- In “NC” tab, ensure that
  - CNC compensation is turned OFF.
  - Toolpath Center can either be WITH compensation or WITHOUT
    - If WITHOUT, ensure “Gouge check” and “Add tool radius to leadin/leadout” is ticked

### **Engraving Operations**

- In “Tool” tab, make sure “3.2 X 90° 2F Engraver” is showing in the comment.
- In “F/S” tab, make sure “Tool” is selected for “Defined by”.
- In “Feature Options” tab, machining depth is set to no more than 0.25mm.