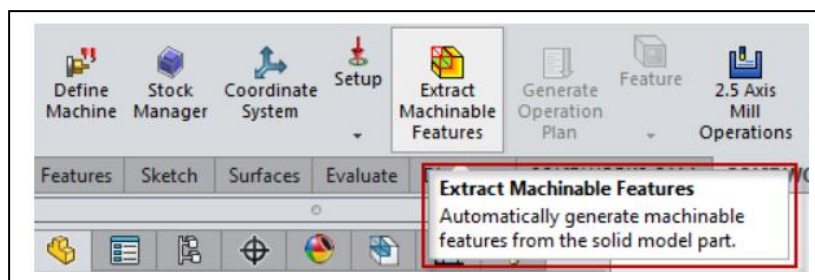| Module: Solidworks CAM Standard-Milling |
| --- |
| Topic 1: Introduction to Solidworks CAM Standard-Milling |

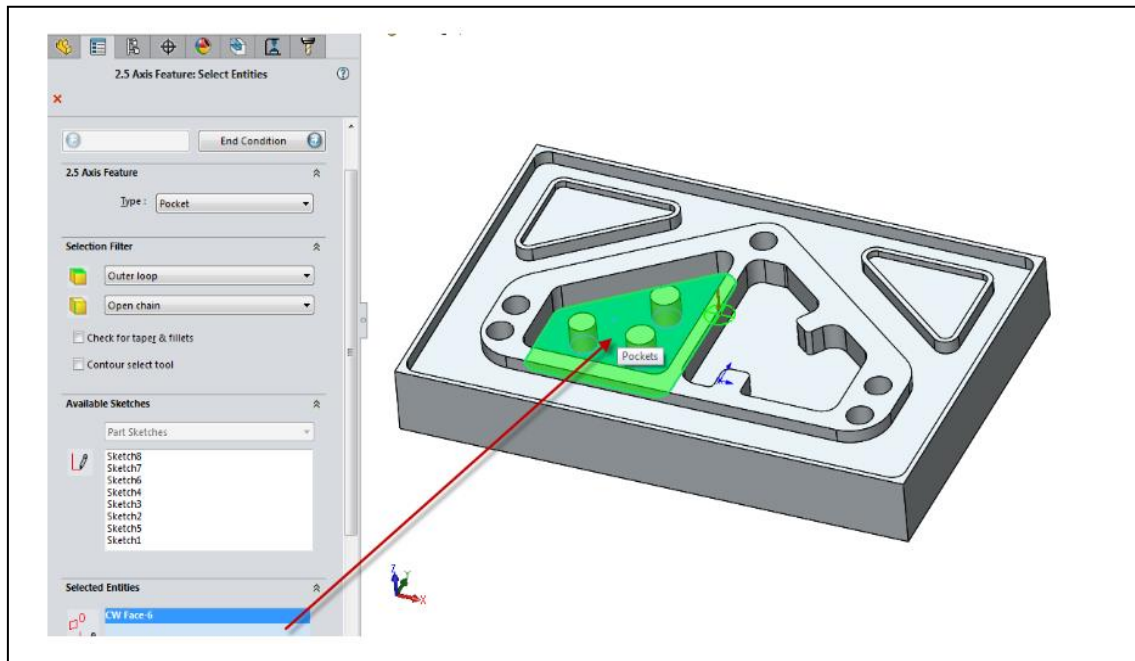### 1.0 Introduction to Solidworks CAM

- Solidworks CAM is a 2.5 axis milling and 2 axis turning solution that is powered by CAMWorks.
- SOLIDWORKS CAM will allow users to program in either part or assembly environments.
- Solidworks CAM is an add-on to all versions of SOLIDWORKS CAD.
- Solidworks CAM combines design and manufacturing in one application with an easy-to-use interface.
- All CAM data stored in the SOLIDWORKS CAD file, do program directly inside SOLIDWORKS.
- SOLIDWORKS CAM is a feature-based CAM package and can generate toolpaths using **Automatic Feature Recognition(AFR)** or **Interactive Feature Recognition(IFR)**
  - I. **Automatic Feature Recognition(AFR)** generates the features found on the geometry in order to apply toolpaths.
  - II. This saves time from having to do create every toolpath manually.
  - III. When SOLIDWORKS CAM runs automatic feature recognition it does not look at the SOLIDWORKS features found within the part.
  - IV. SOLIDWORKS CAM analyzes the geometry in order to apply its own machining features.

- At any time, additional machining operations can be **manually created** using the SOLIDWORKS model geometry.
- Faces to machine and areas to avoid can be defined using the comprehensive suite of selection options, eliminating the need to create additional reference geometry on the part.
- Applying features manually is as easy as selecting faces within the model. If you want more control over the machining area you can utilize SOLIDWORKS sketches for toolpaths, contain and avoid areas, as well as entry points and much more.
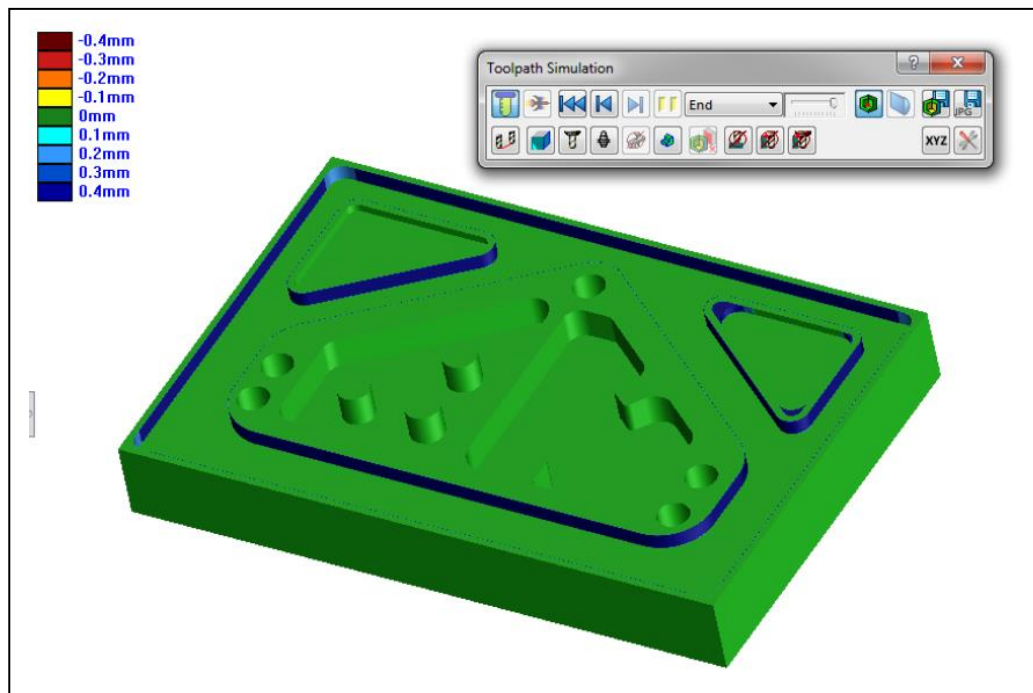
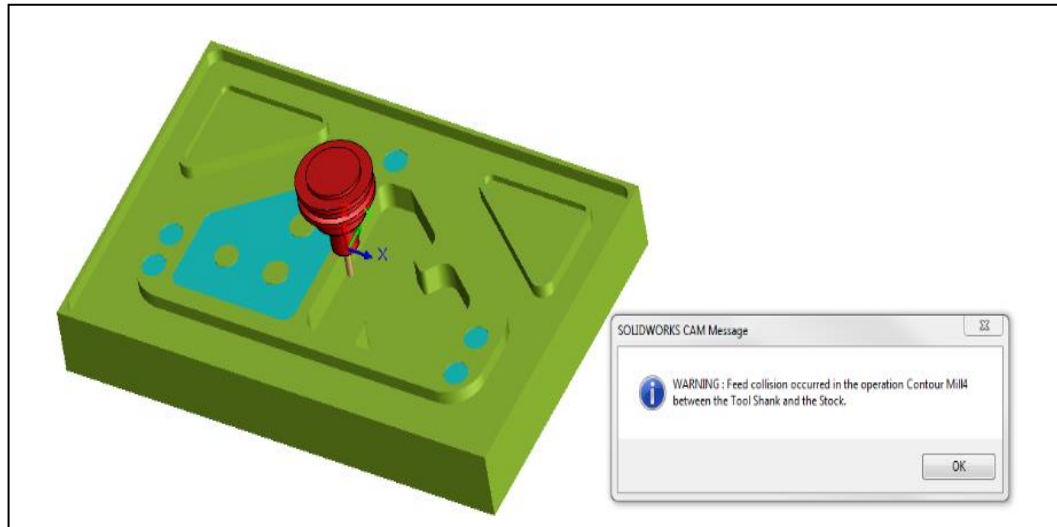## Module: Solidworks CAM Standard-Milling

Topic 1: Introduction to Solidworks CAM Standard-Milling
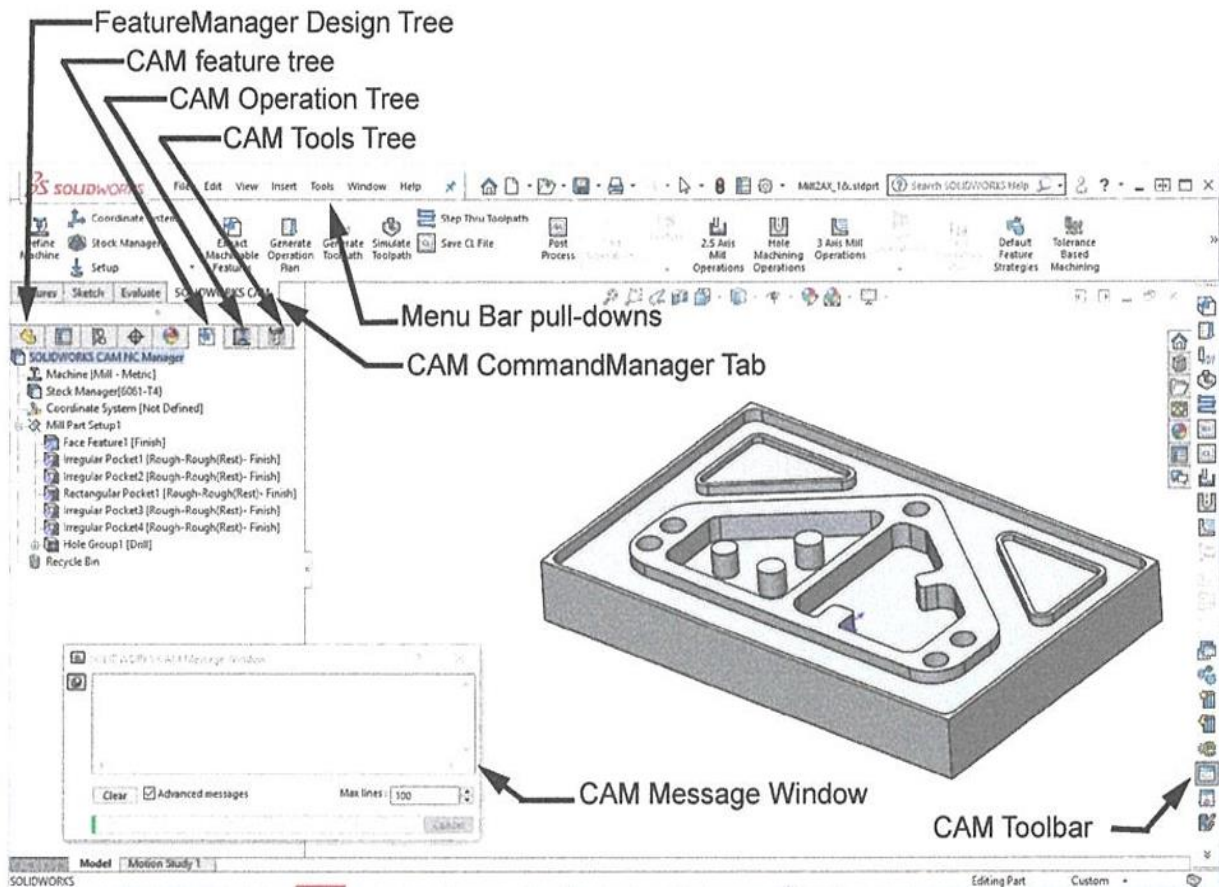

Selecting face from model

- Using SOLIDWORKS CAM's powerful **simulation capabilities**, cutter paths can be previewed and compared against the original model, helping you identify any potential problems, or areas requiring further refinement, long before the physical part is machined. Check out the screenshot below to see SOLIDWORKS CAM's simulation with the comparison tool enabled.

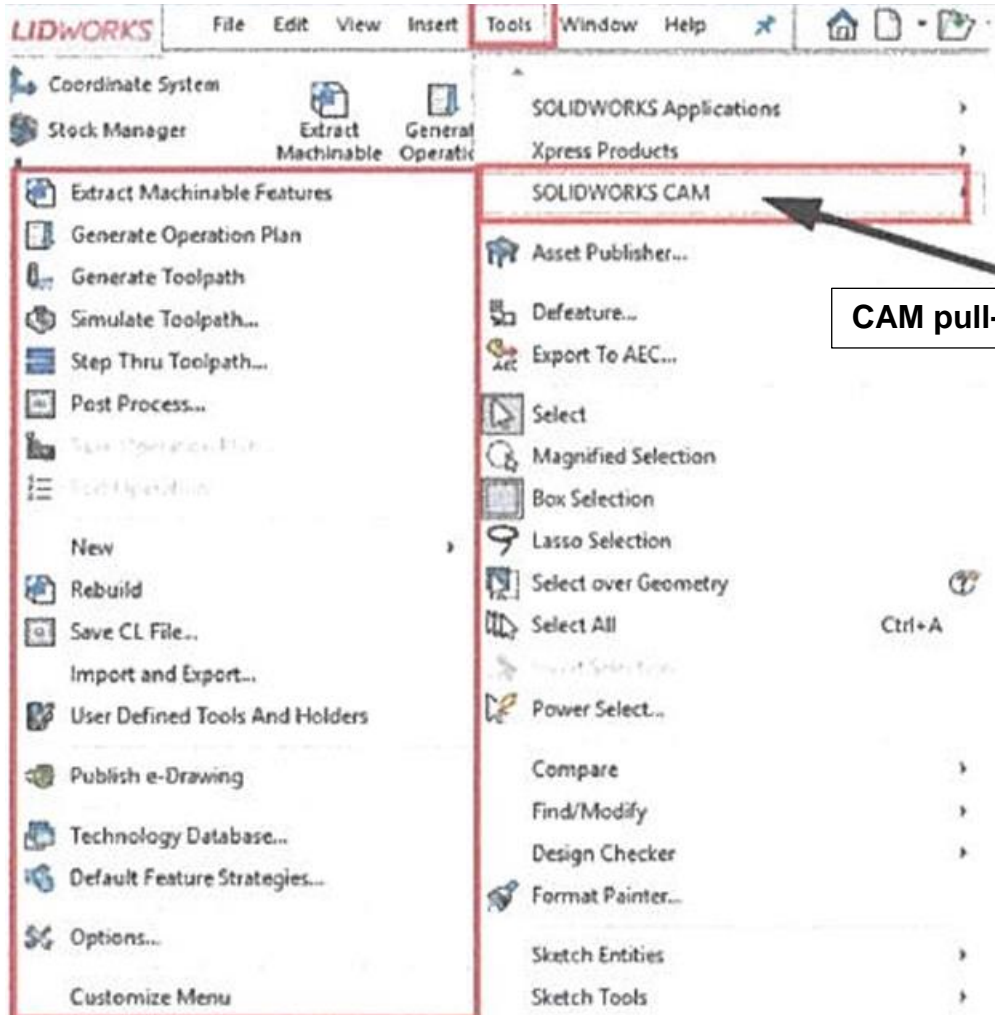| Module: Solidworks CAM Standard-Milling |
| --- |
| Topic 1: Introduction to Solidworks CAM Standard-Milling |

1.1 <u>SOLIDWORKS CAM USER INTERFACE</u>

The Solidworks CAM user interface is an extension to the Solidworks native Windows interface, and as such behaves in the same manner as other Windows applications. Some of the more important aspects of the interface are identified below.

Module: Solidworks CAM Standard-Milling
Topic 1: Introduction to Solidworks CAM Standard-Milling

1.2 Pull-down Menus

The Pull-down menus provide access to many of the commands that The Solidworks software offers. Access Solidworks CAM commands via **Tools → Soildworks CAM.**

CAM pull-down menu

Module: Solidworks CAM Standard-Milling

Topic 1: Introduction to Solidworks CAM Standard-Milling

1.3 Process Overview

The following steps are used to generate mill tool paths and post process them to generate NC codes

1. Model, open, or import the part file in Solidworks.
2. Click on Solidworks CAM feature tree.
3. Define the Machine and modify the controller parameter.
4. Define the stock.
5. Define machinable features.
6. Generate the operation plan and adjust operation parameters.
7. Generate toolpaths.
8. Simulate material removal.
9. Post process the toolpaths.