

## Module: Solidworks CAM Standard-Milling

### Topic 11: Introduction to Interactive Recognition Feature

#### 11.0 Introduction

In SolidWorks CAM, Interactive Feature Recognition is a tool that allows you to identify and define machinable features in a part based on the geometry. When you use this feature, SolidWorks CAM analyzes the part design and interacts with you to help identify features like holes, pockets, bosses, slots, and other machinable shapes.

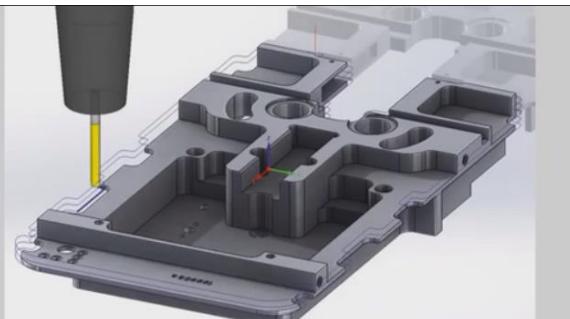
The process is called “interactive” because the software not only recognizes these features but also lets you review and adjust the feature recognition, ensuring it aligns with your machining strategy. Once identified, SolidWorks CAM can automatically generate machining operations based on the recognized features, simplifying the process of creating toolpaths and reducing manual input.

The key benefit of Interactive Feature Recognition is that it speeds up the setup process for CNC programming by automating the recognition of common machining features and by allowing users to change those features as needed.

#### 11.1 Different between 2.5D and 3D

##### 2.5D vs 3D

- 2.5D
  - Level Bottom Surface
  - Straight sides
  - Pockets, Slots, Holes, Engraving
  - Simple operations
- 3D
  - Curved Surfaces
  - Scallops



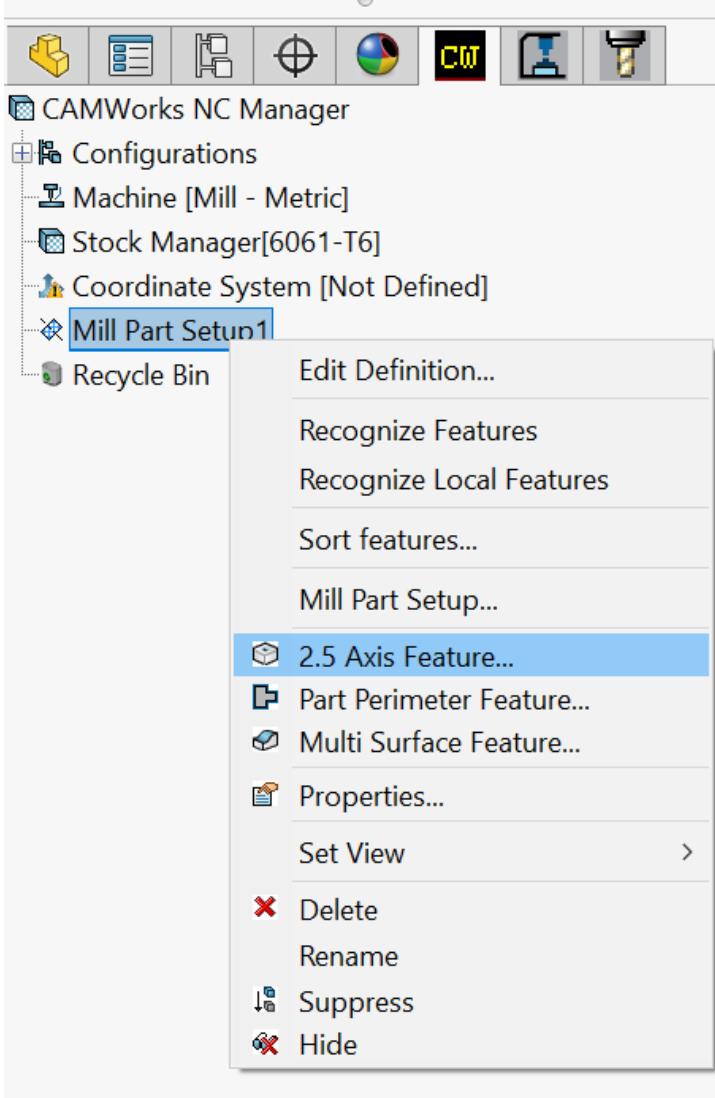
#### 11.2 Defining 2.5 axis features in Solidworks CAM

2.5 axis features have top and bottom of their feature flat and parallel to the mill part setup direction.

To define the 2.5 axis features, you can right click on the Mill Part Setup, then you can select the 2.5 Axis Feature from the context menu.

## Module: Solidworks CAM Standard-Milling

### Topic 11: Introduction to Interactive Recognition Feature



There are 10 types 2.5 axis features in SOLIDWORKS CAM:

- a). Face feature
- b) Open profile
- c) Pocket
- d) Slot
- e) Corner slot
- f) Boss
- g) Hole
- h) Open profile
- i) Engraving feature
- j) Curve feature

## Module: Solidworks CAM Standard-Milling

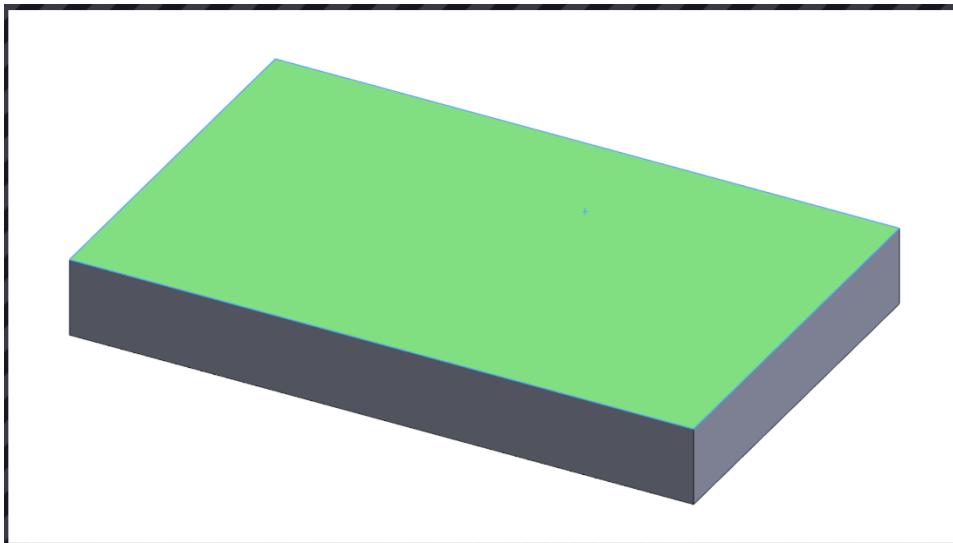
### Topic 11: Introduction to Interactive Recognition Feature

#### a) Face Feature

Face Feature is a planar face parallel to the Mill Part Setup. A Face Feature can be machined to the material extents or the extents of the feature itself.

Face features can be defined by Sketch, face, and edges that are parallel to the current Mill Part Setup. The perimeter of the feature must form a closed non self-intersecting loop.

Allowable operations: Rough Mill, Contour Mill

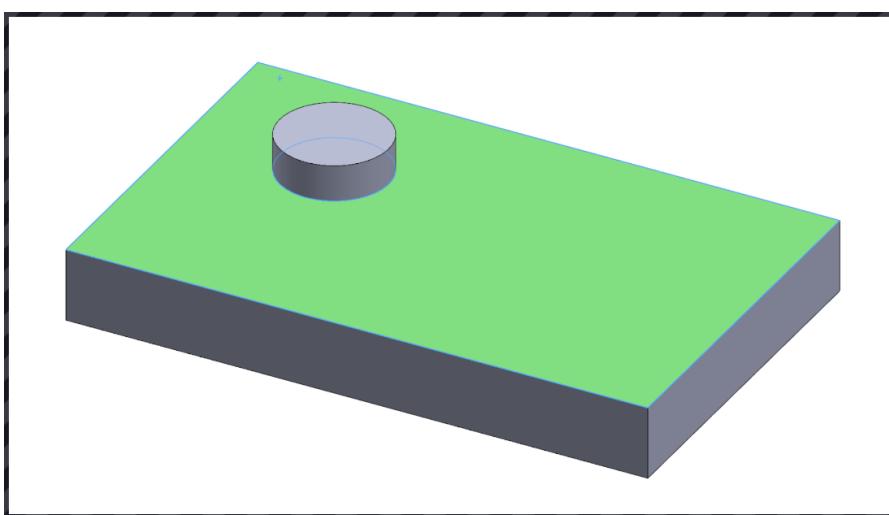


#### b) Open Pocket

An open pocket is an area to cut that is open on all sides. An Open Pocket feature allows machining to extend beyond the geometry that defines the perimeter of the pocket.

Open pocket can be defined by sketch, face, and edges that are parallel to the current Mill Part Setup. The perimeter of the feature must form a closed non self-intersecting loop.

Allowable operations: Rough Mill, Contour Mill

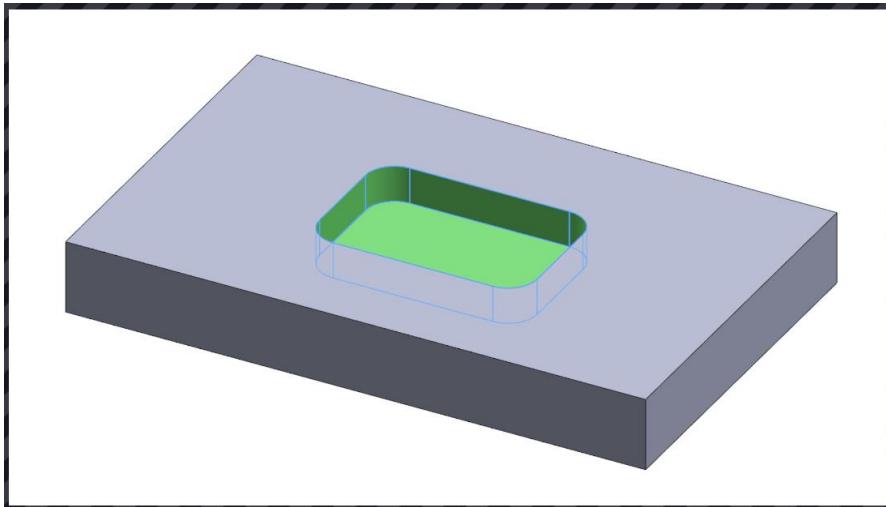


**c) Pocket**

A pocket is a recessed area where none of the sides are open to air. The tool will stay within the edges of the feature. There is rectangular pocket, circular pocket, obround pocket and irregular pocket.

Pocket can be defined by sketch, face, and edges that are parallel to the current Mill Part Setup. The perimeter of the feature must form a closed non self-intersecting loop.

Allowable operations: Rough Mill, Contour Mill, Thread Mill and all Hole Machining Operations

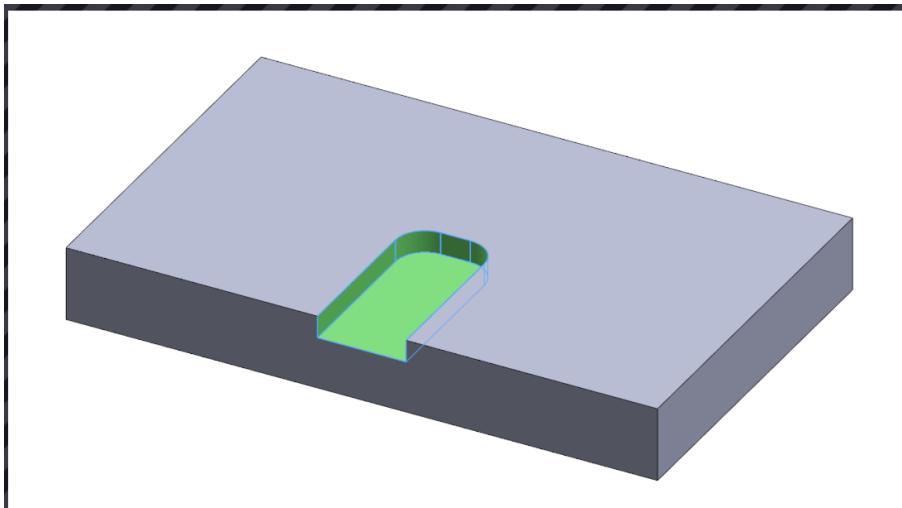


**d) Slot**

A slot is a recessed area with one edge open to the outside of the part. Machining is extended beyond the open segment of the slot, the tool will travel beyond the open edge to ensure the feature is fully cut. There is rectangular slot and irregular slot.

Slot can be defined by sketch, face, and edges that are parallel to the current Mill Part Setup. The perimeter of the feature must form a closed non self-intersecting loop.

Allowable operations: Rough Mill, Contour Mill



## Module: Solidworks CAM Standard-Milling

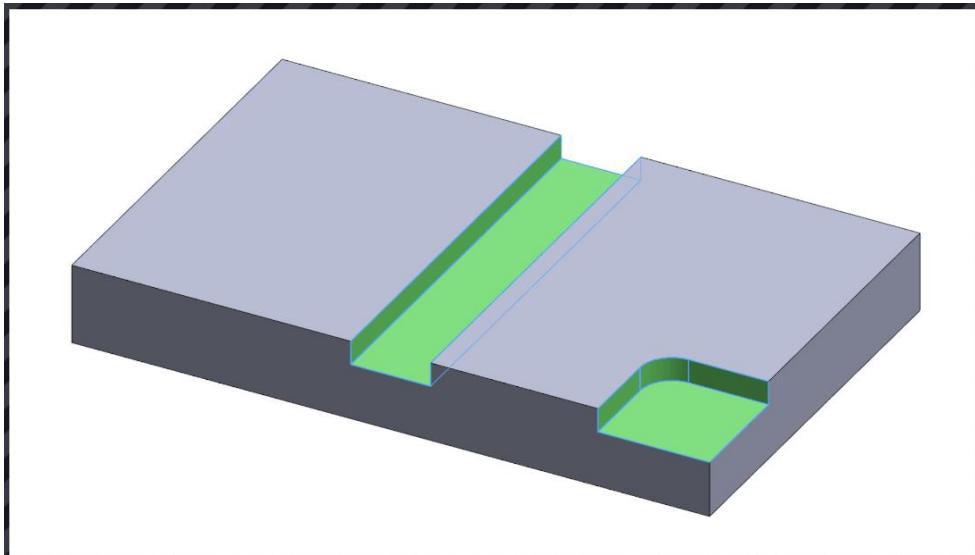
### Topic 11: Introduction to Interactive Recognition Feature

#### e) Corner Slots

A corner slot is a recessed areas with two sides open to the outside of the part. Machining is extended beyond the open segment of the slot, the tool will travel beyond the open edge to ensure the feature is fully cut. There is rectangular corner slot and irregular corner slot.

Corner slot can be defined by sketch, face, and edges that are parallel to the current Mill Part Setup. The perimeter of the feature must form a closed non self-intersecting loop.

Allowable operations: Rough Mill, Contour Mill

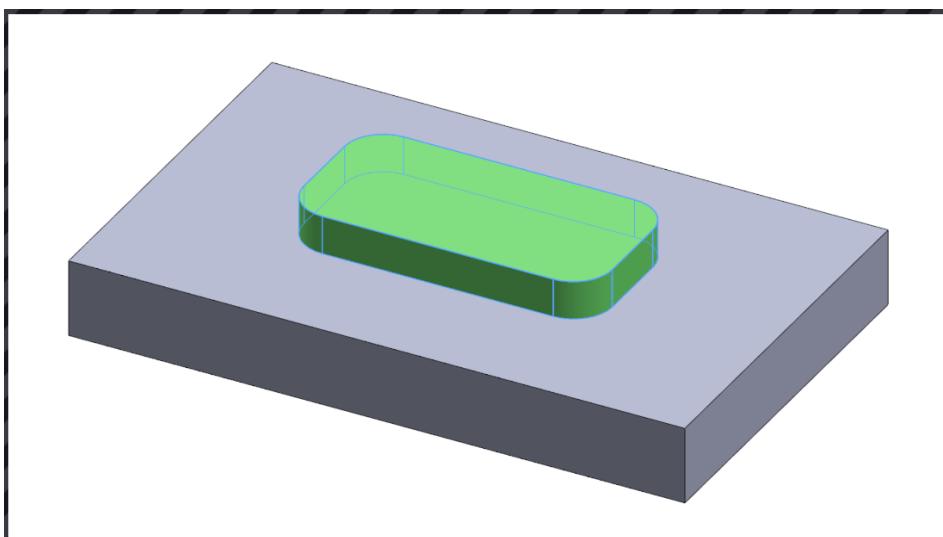


#### f) Boss

A boss is a raised area of material to be cut around. There is rectangular boss, circular boss, obround boss and irregular boss.

Boss can be defined by sketch, face, and edges that are parallel to the current Mill Part Setup. The perimeter of the feature must form a closed non self-intersecting loop.

Allowable operations: Rough Mill, Contour Mill



## Module: Solidworks CAM Standard-Milling

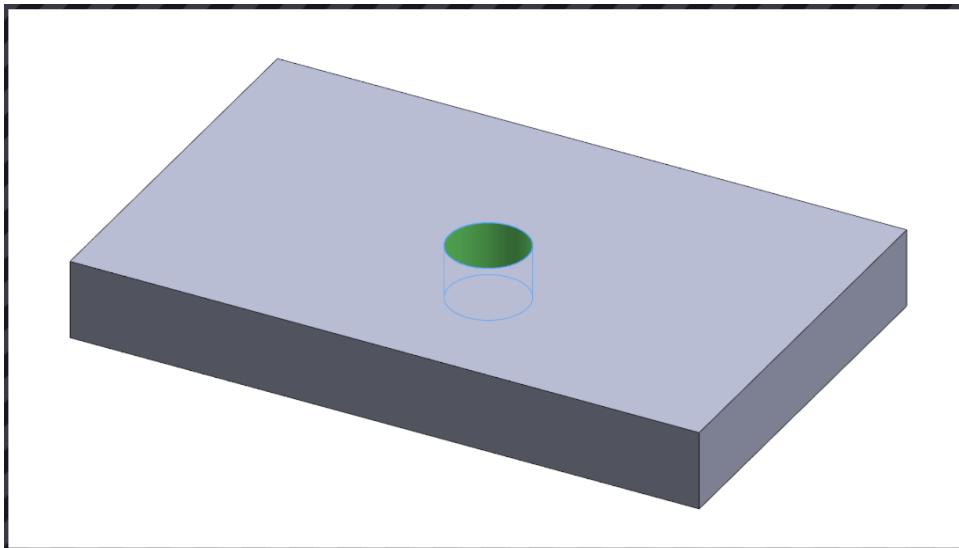
### Topic 11: Introduction to Interactive Recognition Feature

#### g) Hole

A Hole is a round recess to a depth or through the stock using point-to-point tools. There is blind hole and through hole.

Hole can be defined by sketch, face, and edges that are parallel to the current Mill Part Setup. The perimeter of the feature must circular.

Allowable operations: Center drill, Drill, Countersink, Bore, Ream, Tap, Rough Mill, Contour Mill, Thread Mill

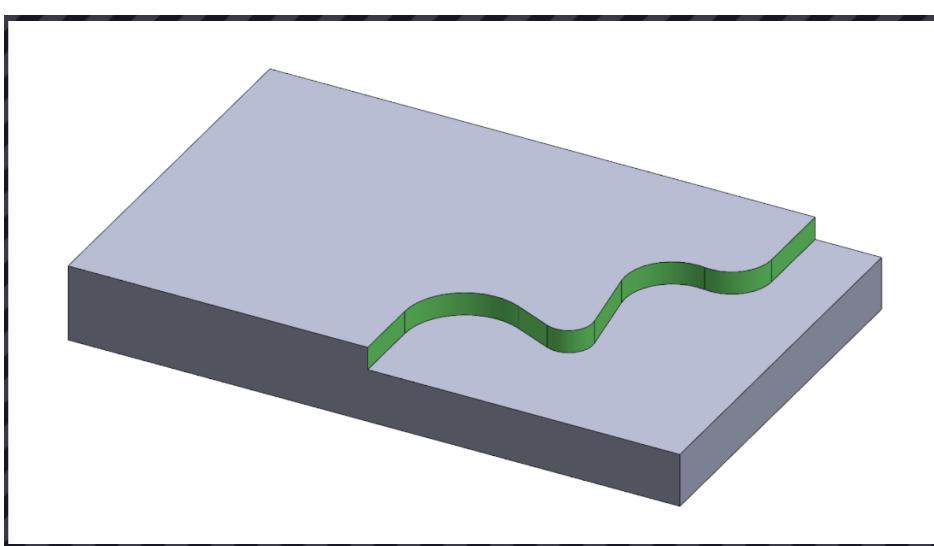


#### h) Open Profile

When it is required to machine an area where the start and end of the feature is not the same, an open profile can be defined. The open profile feature allows the tool to follow part edges or sketch lines. The tool will be offset from the edge or sketch line by the radius of the tool. This feature is commonly used to make a cut along the side of a part.

Open profile can be defined by sketch, edges that are parallel to the current Mill Part Setup. The feature shape must be opened and non self-intersecting.

Allowable operations: Contour Mill



## Module: Solidworks CAM Standard-Milling

### Topic 11: Introduction to Interactive Recognition Feature

#### i) Engrave Feature

With Engrave Feature, the tip of the tool follows the geometry in the feature. The toolpath cannot be offset nor can cutter compensation be applied.

Engrave feature can be defined by sketch, face, and edges that are parallel to the current Mill Part Setup. The geometry of the feature can be either open or closed and self-intersecting. The allowable sketch entities include lines, arcs, splines and text objects.

Allowable operations: Contour Mill



#### j) Curve Feature

With Curve Feature, the tool center will follow the selected 2D or 3D sketch, edge, or curve in three dimensions. This is often useful for chamfers.

Curve Feature can be defined by sketch, face, edges that are either 2D or 3D. The geometry of the feature can be either open or closed and self-intersecting.

Allowable operations: Contour Mill

