SolidWorks/CAMWorks Lab Manual

For

Manufacturing Systems Program

Prepared by
Professor Samuel C. Obi

Manufacturing Systems Program
Department of Aviation & Technology
San Jose State University
December, 2015

Contents

SolidWorks Commands

This section shows the five main groups of SolidWorks commands. They are arranged in five separate sections, namely: Sketch commands, Features commands, Evaluate commands, DimXpert commands, and SolidWorks Office commands. The purpose of this arrangement is to help students to be able to find and locate them easily. Students are encouraged to practice these commands on a regular basis as practice makes perfect. Many U-tube videos are also included here to help students see how various tasks and commands are employed in SolidWorks.

Sketch Commands

These are commands used to draw and dimension various shapes and geometries in SolidWorks. Figure 1 shows the Sketch tool bar, while Figure 2 shows a comprehensive listing of all the Sketch commands.
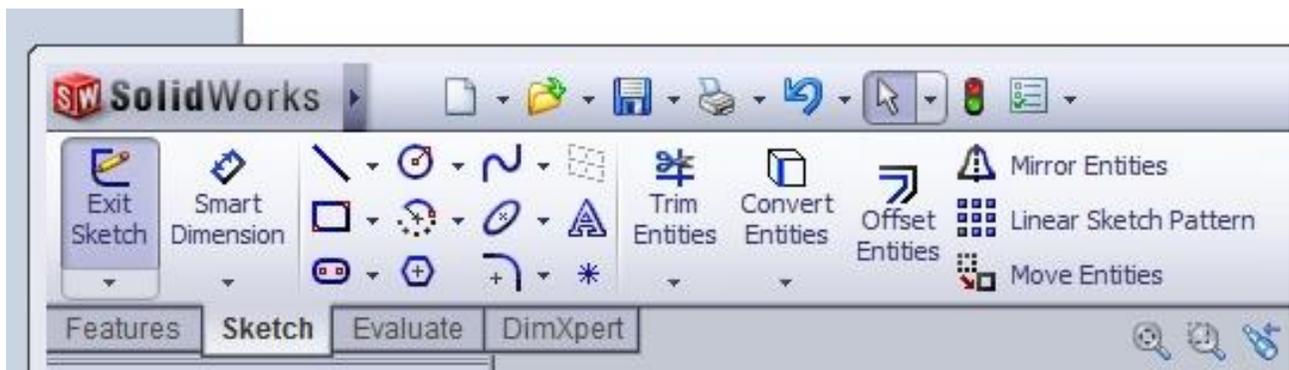
Figure 1: Sketch Commands Tool Bar

Tools Available:
Edit Sketch: Sketch, 3D sketch and Edit Sketch
Smart Dimensions: Smart, Horizontal, Vertical, Ordinate, Horizontal Ordinate, Vertical
    Ordinate Dimensions.
Sketch Tools:
    Line – Line, Center Line
    Rectangles: Corner, Center, 3 Point Corner, 3 Point Center Parallelogram
    Polygon
    Circle: Circle, Perimeter Circle
    Arc: 3 Point Arc, Centerpoint Arc, Tangent Arc
    Fillet: Sketch Fillet, Sketch Chamfer
    Spline
    Ellipse: Ellipse, Partial Ellipse, Parabola
    Point
    Plane
    Text
Trim Entities: Trim entities, Extend Entities

Convert Entities: Convert Entities, Intersection Curve
Offset Entities
Mirror Entities
Linear Sketch Pattern: Linear Sketch Pattern, Circular Sketch Pattern
Move Entities: Move, Copy, Rotate, Scale Entities
Display/Delete Relations: Display/Delete Relations, Add Relations
Quick Snaps: Point, Center Point, Midpoint, Quadrant, Intersection, Nearest, H/V
Point, Grid Snaps Rapid Sketch
View Origins
View Planes
View Temporary Axes
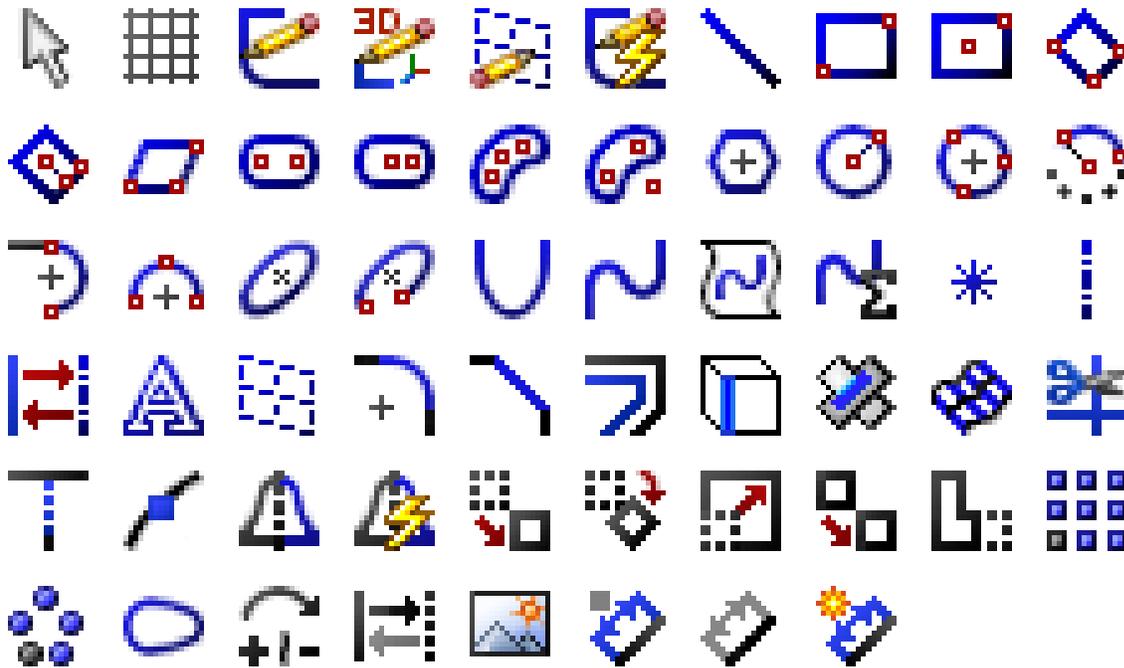
Figure 2: Sketch Tools

Review the above listed tools available in the Sketch command and try to identify them in Figure 1. Use your mouse to identify each of the commands and note their meanings and the functions they perform. The more familiar you get with using and recognizing these, the more proficient a designer you will become.

Features Commands
These are commands used to add 3-dimensional features and sizes to designs. Figure 3 shows the Features tool bar.
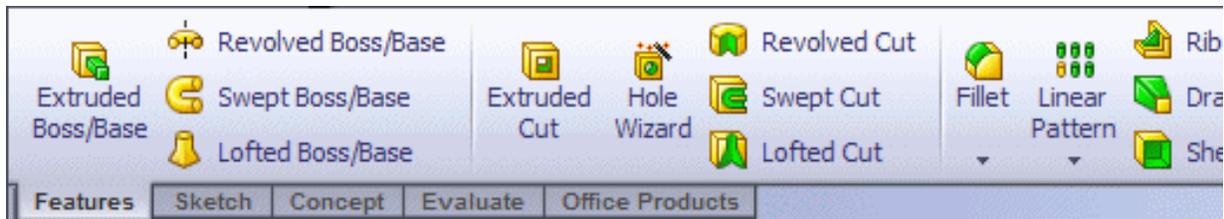
Figure 3: Features Commands tool Bar

Tools Available:
Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Fillet: Fillet, Chamfer
Linear Pattern: Linear Pattern, Circular Pattern, Mirror, Curve Driven Pattern, Sketch
    Driven Pattern, Table Driven Pattern, Fill Pattern
Rib
Draft
Shell
Wrap
Dome
Mirror
Reference Geometry: Plane, Axis, Coordinate System, Point, Mate Reference
Curves: Split Line, Composite Curve, Curve Through XYZ Points, Curve Through
    Reference Points, Helix and Spiral
Instant 3D View
Orientation Transparent
Tool Bar:
Previous View
Normal To
Section View
Front, Left, Right, Top, Bottom, Isometric Views
View Orientation: Top, Isometric, Trimetric, Dimetric, Left, Front, Right, Back, Bottom,
    Normal To, Single View, Two View Horizontal, Two View Vertical, Four View, Link
    Views
Display Style: Shaded With Edges, Shaded, Hidden Lines Removed, Hidden Lines
    Visible, Wire Frame
Hide/Show Items: Plane, Origins, Axes, Temporary Axes, Coordinate System, Points,
    Curves, Parting Lines, 3D Sketch Dimensions, All Annotations, Sketches, 3D Sketch
    Plane, Sketch Relations, Grid, Lights, Cameras, Routing Points

RealView Graphics
Shadows in shaded Mode

Evaluate Commands
These are commands used to perform analysis to determine various physical and mechanical characteristics about the design. Figure 4 shows the Evaluate tool bar.

Evaluate Tool Bar

Figure 4: Evaluate Commands Tool Bar

Tools Available: Measure
Mass Properties
Section Properties
Statistics
Check
Import Diagnostics Heal
Edges
Deviation Analysis
Zebra Stripes
Curvature
Draft Analysis
Undercut Detection
Equations
COSMOSXpress Analysis Wizard
COSMOSFloXpress Analysis Wizard
DFMXpress Analysis Wizard
DriveWorksXpress Wizard

DimXpert Commands
These commands are the dimensioning agents of SolidWorks. They enable dimensioning and tolerancing to your modeled parts. Figure 5 shows the DimXpert tool bar.

Figure 5: DimXpert Commands Tool Bar

Tools Available include:
Auto Dimension
Location Dimension
Size Dimension
Datum
Geometric Tolerance
Pattern Feature
Show Tolerances
TolAnalyst Study

SolidWorks Office Commands
SolidWorks Office commands list most of the commonly used Add-Ins with the enabled ones shown with the standard grey "push button" rectangular highlight as shown in Figure 6. The Office Products tab lets you load additional functionality that is available if you have installed SolidWorks Professional or SolidWorks Premium

Figure 6: SolidWorks Office Commands Tool Bar

Tools available include:
Design checker
SolidWorks simulation
SolidWorks motion
SolidWorks toolbox
SolidWorks utilities
PhotoWorks
Scan to 3D

CircuitWorks
3-D instant Website

SolidWorks Videos
This section has videos designed by SolidWorks to help you quickly learn how to use the design command tools. You are encouraged to view as many of them as possible. The contents of these videos will enhance your learning of SolidWorks commands.

This section is also geared to quickly introduce you to 3D parametric modeling. You will learn how to navigate the SolidWorks interface, create sketches, set up parametric relationships, and create 3D models. You will also learn the basics of creating assemblies, and how to generate 2D production drawings of both parts and assemblies. The numbers to the right of each video show the length of time in minutes and seconds of the video. The videos can be downloaded by googling the titles or by registering with SolidWorks who will provide access passwords with fees. If you have a password, the link is at: https://app.solidprofessor.com/lmsapp/#/library?software=1&courses=368

About This Course
Introduces and explains the scope of the course   1m 49s

What is SOLIDWORKS?
Provides some background info on the SOLIDWORKS software   0m 51s

Interface Tour
Covers the different areas of the SOLIDWORKS interface   5m 52s

View Manipulation
Shows some basic techniques for working with 3D models in the interface 3m 50s

Starting a Part
Shows how to start a new SOLIDWORKS Part file   2m 41s

Starting a Sketch
Introduces the sketcher and shows how to start a new sketch   7m 20s

Sketch Tools
Shows how to use some of the basic sketch tools   8m 9s

Geometric Relations
Explains how relations can be added to control a sketch   9m 23s

Dimensions
Explains how dimensions are used to define and control a sketch 7m 31s

Extrude Feature

Explains how to extrude a 2D sketch into a 3D part   2m 45s

Editing Geometry
Shows how to edit existing geometry   2m 58s

Basic Sketching
Walks you through the steps required to complete the Basic Sketching
exercise   3m 22s

Basic Extrude
Walks you through the steps required to complete the Basic Extrude
exercise   4m 32s

Editing Geometry
Walks you through the steps required to complete the Editing Geometry
exercise   2m 55s

Overview of Design
Introduces the bracket part that will be designed in this section   5m 4s

Creating the Base Feature
Shows how to create the first feature of the bracket part   3m 57s

The Tab
Shows how to add the tab feature to the bracket   14m 25s

Hole in Tab
Shows how to add the hole to the tab feature   5m 32s

Mirror Feature
Introduces the mirror tool which is used to take advantage of symmetry
in a design   3m 31s

Rollback Bar and Order of Features
Shows how the order that features are created in affects a design   3m 50s

Hole Wizard
Introduces the hole wizard which is used to add standard sized holes
to a part   7m 34s

Fillets
Shows how to use fillets to round off sharp edges   6m 28s

Mounting Block - Part 1
Walks you through the first few steps required to complete the
Mounting Block exercise   2m 40s

Mounting Block - Part 2
Walks you through the steps required to complete the Mounting Block
exercise   5m 55s

Mounting Block - Part 3
Walks you through the remaining steps required to complete the
Mounting Block exercise   5m 6s

Bracket - Part 1
Walks you through the first few steps required to create the Bracket part   4m 27s

Bracket - Part 2
Walks you through the steps required to create the Bracket part   4m 11s

Bracket - Part 3
Walks you through the remaining steps required to complete the
Bracket part   5m 34s

Revolve Feature
Shows how to use the revolved boss feature to create a wheel part   11m 19s

Cutout and Circular Pattern
Adds a cutout and patterns it around the wheel   5m 55s

Pinion Mount
Walks you through the steps required to complete the Pinion
Mount exercise   4m 9s

Strainer
Walks you through the steps required to complete the Strainer exercise   3m 51s

Assembly Overview
Introduces how parts can be put together to form assemblies   1m 36s

Starting an Assembly
Explains how to begin an assembly file   3m 54s

Inserting Additional Components
Shows various methods to insert components into an assembly   2m 33s

Move Components
Explains how to position parts within an assembly   3m 42s

Mating the Wheel
Shows how to add mates to lock down the wheel's position in an assembly 10m 27s

Mating the Pin
Shows how to mate the pin component in the assembly   5m 23s

Toolbox
Shows how to use the toolbox to add standard hardware to an assembly   4m 29s

Exploded View
Shows how to create an exploded view of the assembly   3m 57s

Working With Sub-Assemblies
Shows how to add an assembly within a larger assembly   2m 18s

Basic Moving Assembly - Part 1
Walks you through the first half of the Basic Moving Assembly Exercise   3m 26s

Basic Moving Assembly - Part 2
Walks you through the second half of the Basic Moving Assembly exercise 3m 8s

Brace Assembly - Part 1
Walks you through the first half of the Brace Assembly exercise   2m 58s

Brace Assembly - Part 2
Walks you through the second half of the Brace Assembly exercise   1m 43s

Scissor Jack Assembly - Part 1
Walks you through the first half of the Scissor Jack Assembly exercise   5m 13s

Scissor Jack Assembly - Part 2
Walks you through the second half of the Scissor Jack Assembly exercise 6m 30s

Drawings Overview - Starting a New Drawing
Explains how to begin a new drawing file   3m 39s

Model Views
Explains how to use the model view command to create views on your
drawing sheet   6m 59s

Projected Views
Shows how to create a projected view from an existing view   1m 13s

Model Items
Shows how to import dimensions and other model items to a drawing   3m 58s

Dimensions
Shows how to add dimensions to a drawing   2m 19s

Associativity
Explains the relationship between a part, assembly, and drawing and
how they are connected   1m 36s

Annotations
Shows how to add notes and other annotations to a drawing   3m 5s

Detail View
Shows how to create a detail view in a drawing   2m 5s

Section View
Shows how to create a section view of a model   2m 44s

Assembly Drawing
Shows how to create a drawing for an assembly file   4m 8s

Bill of Materials
Explains how to add a bill of materials in a drawing   4m 5s

Balloons
Shows how to add balloons in a drawing   2m 51s

Part Drawing - Clevis
Walks you through the steps required to complete the Clevis Part
Drawing exercise   5m 44s

Part Drawing - Holder
Walks you through the steps required to complete the Holder Part
Drawing exercise   7m 5s

Assembly Drawing - Brace
Walks you through the steps required to complete the Brace
Assembly Drawing exercise   6m 16s

CAMWorks

This section contains CAMWorks commands and instructions on how to use the
commands to successfully prepare your CNC programs. Supportive instructional videos
are also included. CAMWorks has many video examples designed to illustrate how to
perform special tasks and add or machine specific features to parts. Students are
encouraged to identify and watch specific videos that relate to their assignments or
needs in order to understand how to better apply CAMWorks commands to their specific
situations. It is recommended that, when necessary, students view particular videos to
help them understand respective sections. You can download the videos by googling
their titles or by going to the site if you have a password. The link to the official site is at:
https://app.solidprofessor.com/lmsapp/#/library?software=2&courses=230

This section will walk you through the fundamentals of creating tool paths and NC code in CAMWorks. You will learn how to create features, set up machining strategy, create avoids to limit toolpath movement, and run a simulation to review the final output.

CAMWorks Interface
Before getting started learning the fundamentals of creating toolpaths and NC codes in CAMWorks, it is important to familiarize with some important areas of SolidWorks interface. The CAMWorks interface runs directly in SolidWorks as shown in Figure 7. So some items should already be familiar including the graphics area, feature manage design tree, and command manager.
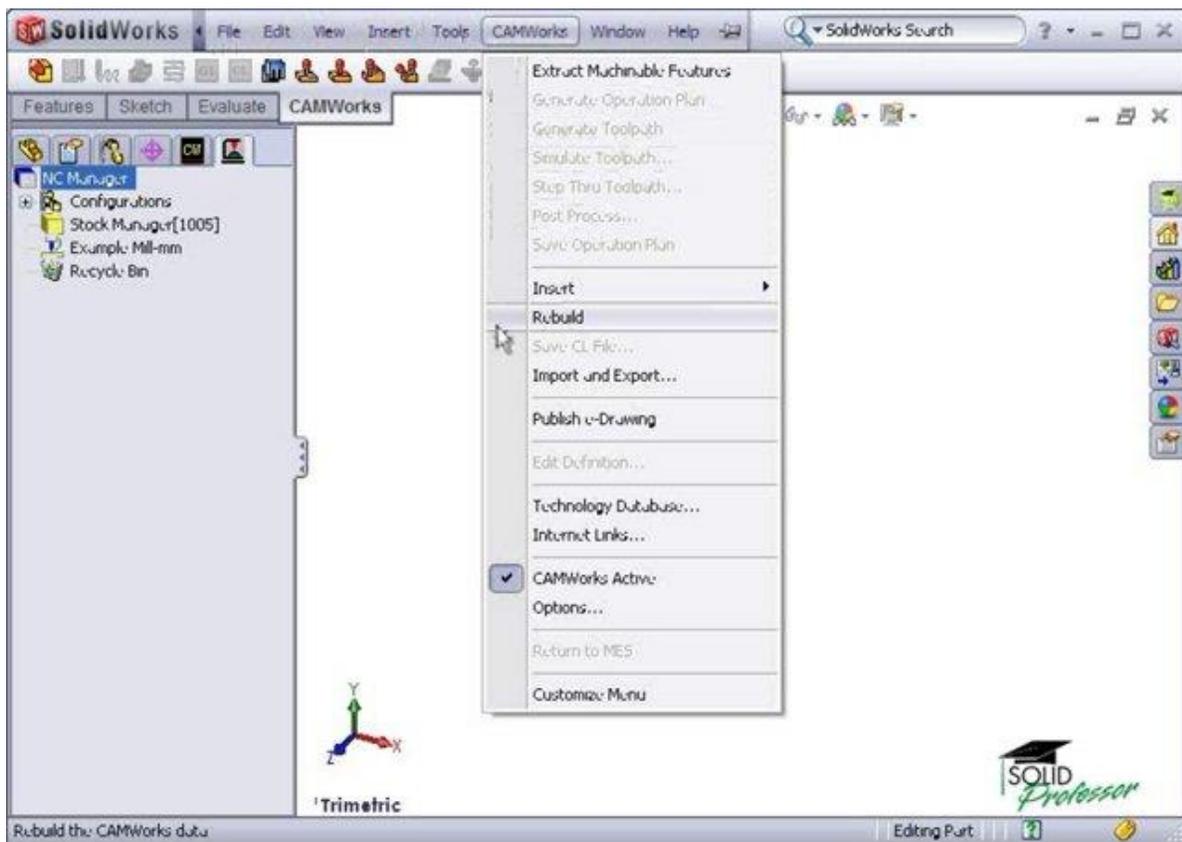
Figure 7: CAMWorks Interface in SolidWorks

CAMWorks is an embedded CAM program that is fully integrated within SolidWorks. It is a feature-based CAM software which leverages synchronous technology and provides users with reduced programming time.

Some important areas of CAMWorks interface include:
1. The graphics area
2. Dropdown menus

3. Command manager
4. Feature Manager Design Tree
5. CAMWorks tool bar
6. CAMWorks Workflow tool bar
7. Feature Tree tab
8. Operations Tree tab

Key icons in the CAMWorks Workflow include;
1. Machine Set Up icon
2. Stock Manager icon
3. Extract Machinable Features icon
4. Generate Operation Plan icon
5. Generate Toolpath icon
6. Simulate Toolpath icon

Some of the new items that come with CAMWorks interface include the CAMWorks tool bar, the CAMWorks workflow tool bar, the Feature tree tab, the Operations tree tab, and the CAMWorks dropdown menu. Many of the tools found in CAMWorks tool bar can also be found in CAMWorks workflow tool bar or by clicking one of the items in the tree. Command managers tool bars can be turned on and off by right clicking on the command managers' tab and selecting them from the drop down list.

Process Overview for CAMWorks
The CAMWorks interface works directly in SolidWorks. In addition to other SolidWorks features, it contains the CAMWorks toolbar, the CAMWorks workflow toolbar, the Feature tree tab, the Operations tree tab, and the CAMWorks drop-down menu. These features help to make the task of preparing the design for machining much easier.

To fully configure a part design for generating its toolpaths, NC codes and machining, the following 10 steps described here must be followed. These steps are generally helpful for both CAMWorks milling and turning procedures but there are specific differences which will be pointed out for each specific area in the sections which follow.

Step 1: Open a Part in SolidWorks
This is where the part drawing is opened in SolidWorks window. Different file formats (IGES, STL, CADKEY, ProE etc.) can be opened in CAMWorks. Figure 8 shows sample parts for milling and turning.
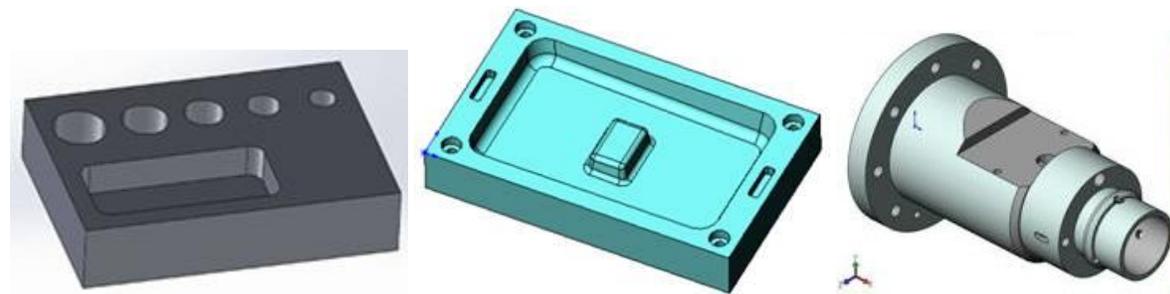
Figure 8: Sample SolidWorks Milling and Turning Parts

Step 2: Define the Machine

This is where the particular machine (Figure 9) is selected. This is important so that CAMWorks will customize the data to the machine's capabilities and requirements. The

Figure 9: Sample CNC Milling and Turning Machines

required tools are also selected and configured in this step to match the intended design characteristics. Selecting the machine also includes selecting tools from tool crib, and selecting the correct post processor and all the associated changes that will be made relative to the part and machine that have been selected.

Step 3: Define the Stock or Material

This is where the size of the part is specified. Also specified are the offset values of the material with which the part is made. These offset (extra) values are assumed to be the extra material the machine will remove to create the designed part to the specified size. Also defined here is the type of material with which the part is made. The material type is important because CAMWorks will use that information to determine the optimum speed (RPM) and feed values ideal for the operations.
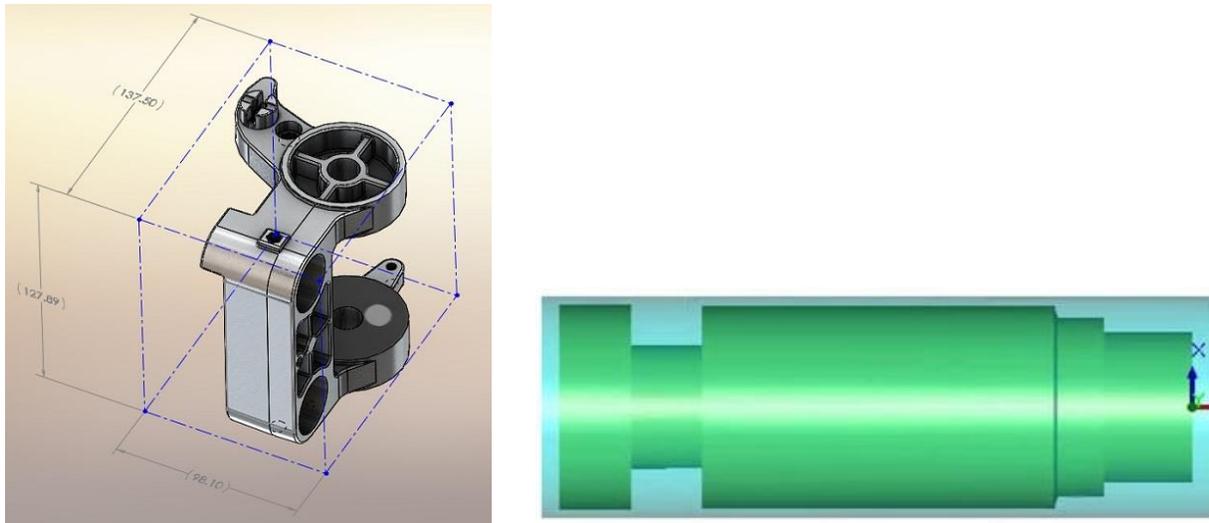
Figure 10: Selecting a Part by Bounding Box (Left) and Bounding Cylinder (Right)

Defining the stock can be accomplished by any of the following techniques:
1. By using the bounding box or bounding cylinder as shown in Figure 10.

2A. By using sketches (using existing sketch or by creating your own sketch). To do this, click at CAMWorks feature manager tree and expand the feature to see the existing sketch used to create the profile of the part. Click on CAMWorks feature tree and right-click on the stock manager to edit the definition. In the new window, click on CAMWoks sketch option to select it, and then select the existing sketch in the CAMWorks feature tree manager. Type in the needed value in the extrude properties section and you will see the selected part offset by same value.

2B. When the stock you are using does not match the sketch profile, then you have to create a new sketch that matches the stock. To do this, switch to CAMWorks feature manager by clicking at the tab. Select sketch from the sketch tool bar. Select the plane to use by clicking on the part's surface. Draw the sketch as needed. Add dimensions to the new sketch to fully define the sketch. Exit the sketch and the new sketch will be added to the feature manager tree. Click on CAMWorks flow manager tab and right-click on the stock manager icon and select the edit definition option to edit the new sketch.

3. By converting entities from the model. When no useful sketch is available on a part, its entities can be converted to help in defining the stock. To do this, insert a new sketch to the CAMWorks feature manager tree. Select the face you want to use as a reference, and then select all the lines on that face that make up its features by holding the control key and selecting the entities. Click at convert entities icon. The new sketch will be added. As you exit the sketch to complete it, the new sketch will be available to use.

In addition to defining stock as described above, stock can be selected as STL files and defined in CAMWorks. However, make sure the units, materials are set in CAMWorks to

match the units and materials used in creating the part in the first place. To import the STL file, make sure CAMWorks is active in sketch tab. Click at stock manager and select edit definition option. Select STL box. Select the particular file to import. Select the units and then click open. Specify the material and check the preview of the part. Click OK, and the STL file will be downloaded.

Step 4: Define the Features to Machine
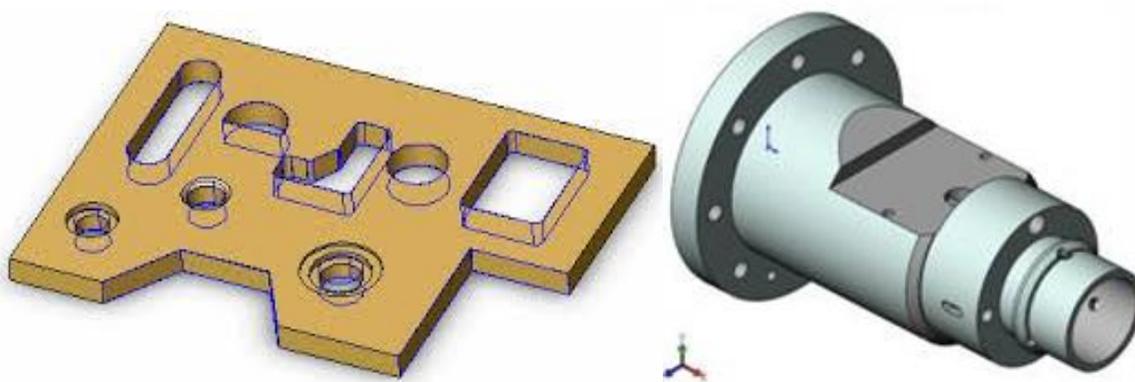
Figure 11: Sample Part Features

This is where the machinable features (holes, slots, diameters, chamfers, and other cuts) that define the shape of the parts are selected or identified on the parts (Figure 11). This information enables CAMWorks to know and identify what sections of the material to cut or shape as shown in the design. Defining the features can be done automatically or interactively.
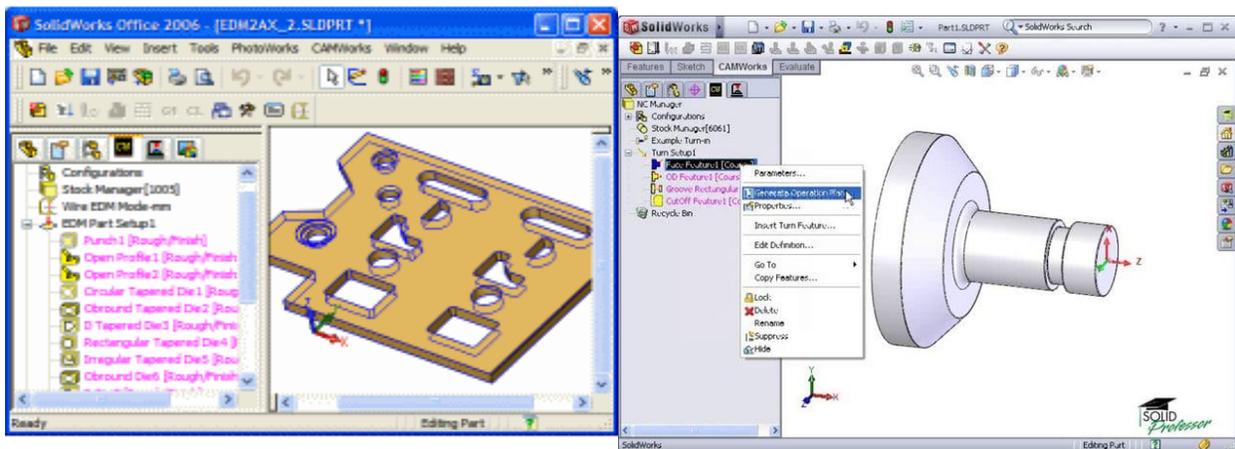
Figure 12: CAMWorks Generate Operation Output

18

Step 5: Generate Operations

CAMWorks incorporates knowledge-based machining technology and its automatic feature recognition (AFR) technology in its user-defined database to automatically define machinable features and generate a sequence of operations to machine them. To do this, click at the Generate Operation icon. CAMWorks recognizes these features and automatically generates an operation plan. These are shown in magenta color in Figure 12.

Step 6: Select the Origin

This is the point where you want the tool to start the cutting operation. You must indicate this point and align it with the point in your part design where the cutting operation must begin. For milling, this is accomplished by clicking at a selected vertex of the rectangle as shown in Figure 13. For turning, this is accomplished by using the setup option to indicate the orientation and the origin of the part (Figure 14).
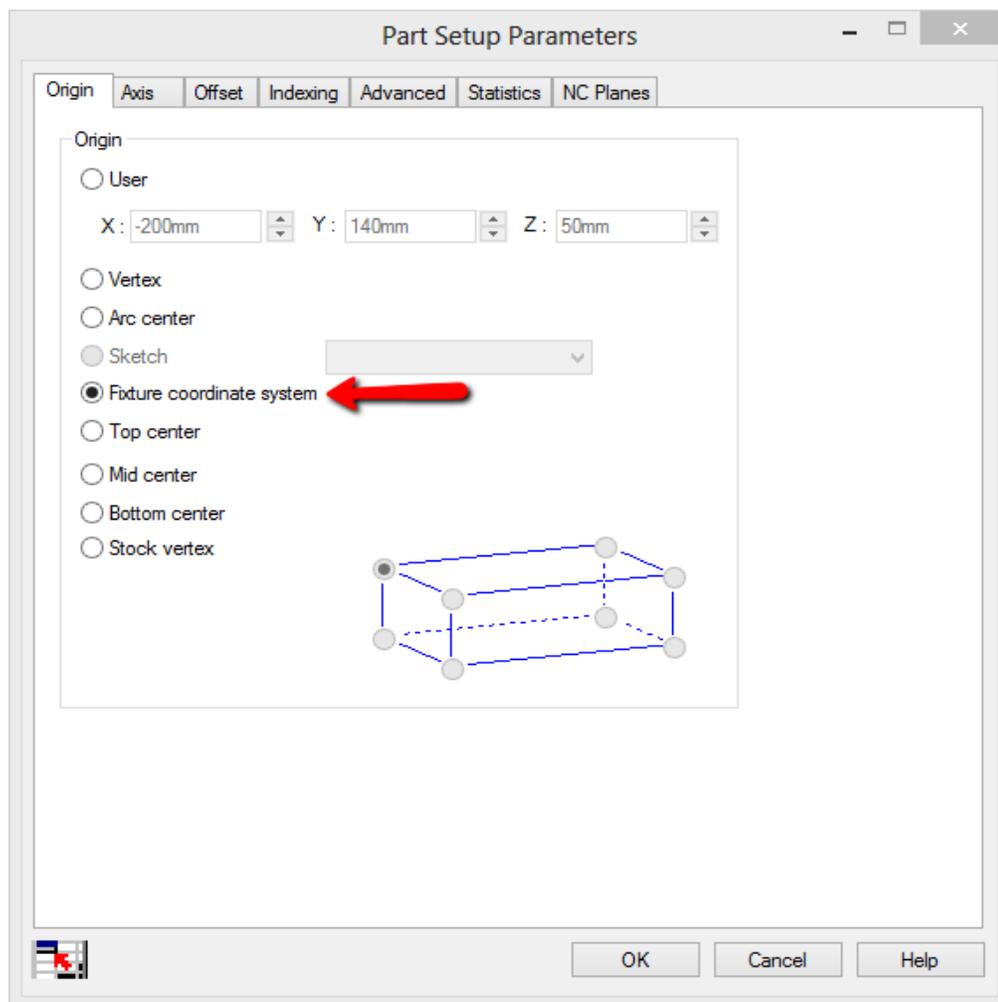
Figure 13: Part Origin Selection for Milling
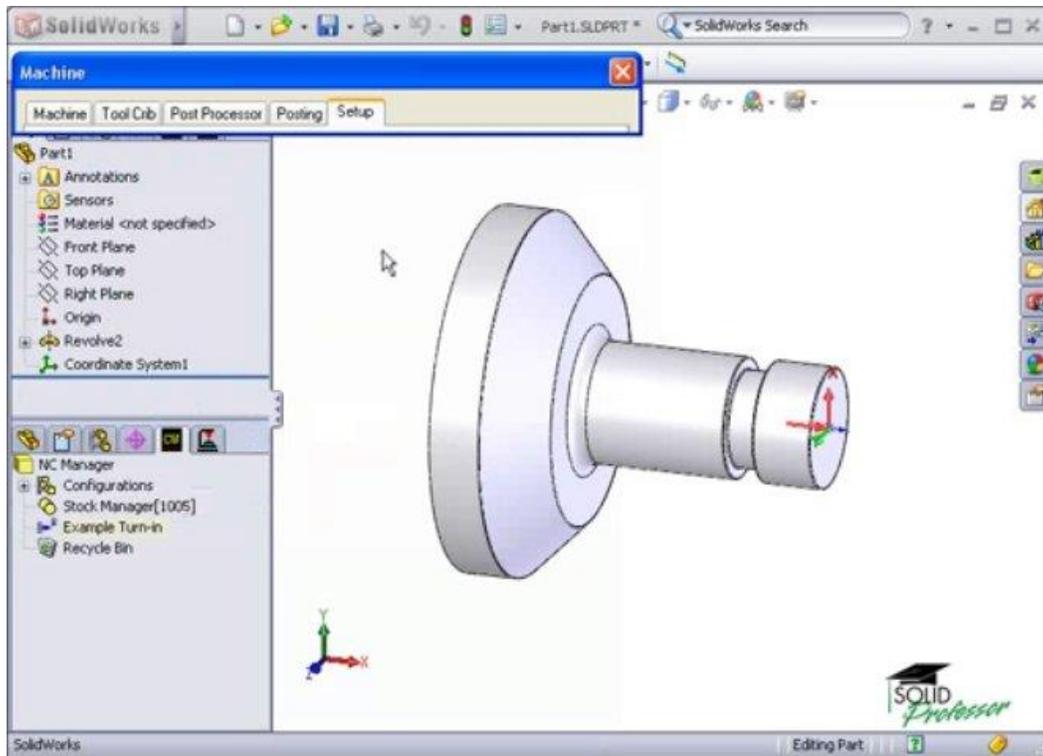
Figure 14: Part Origin Selection for Turning

Step 7: Generate Tool Paths

This pretty much generates all the tool paths needed to machine the part. The tool paths are generated from the features identified and defined previously and then stored in the CAMWorks system. To accomplish this, click on the Generate Toolpath icon. The toolpaths are displayed as shown in Figure 15.
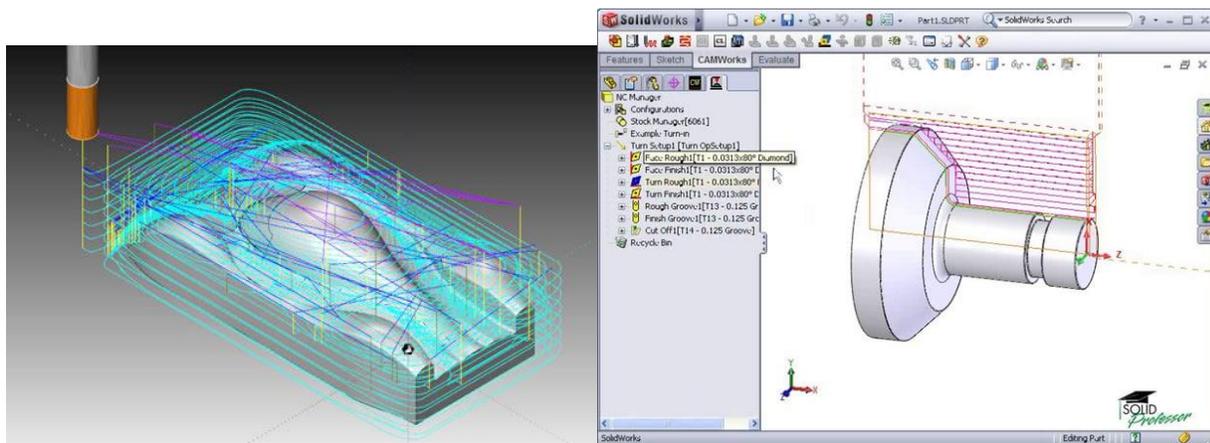
Figure 15: CAMWorks Toolpath Generation

Step 8: Simulate Tool Paths

This is where you directly output post-processed G-code to the simulation module to show the actual commanded moves of the machining processes. Accurate 3d models of your machine, toolholders, fixturing, and target part are automatically imported into the simulation environment. This allows for highly accurate visualization of cutter behavior as shown in Figure 16. Collision detection constantly checks for interference, and any errors found can then be edited directly in the G-code. To accomplish this, use the Simulation Toolpath play tool.
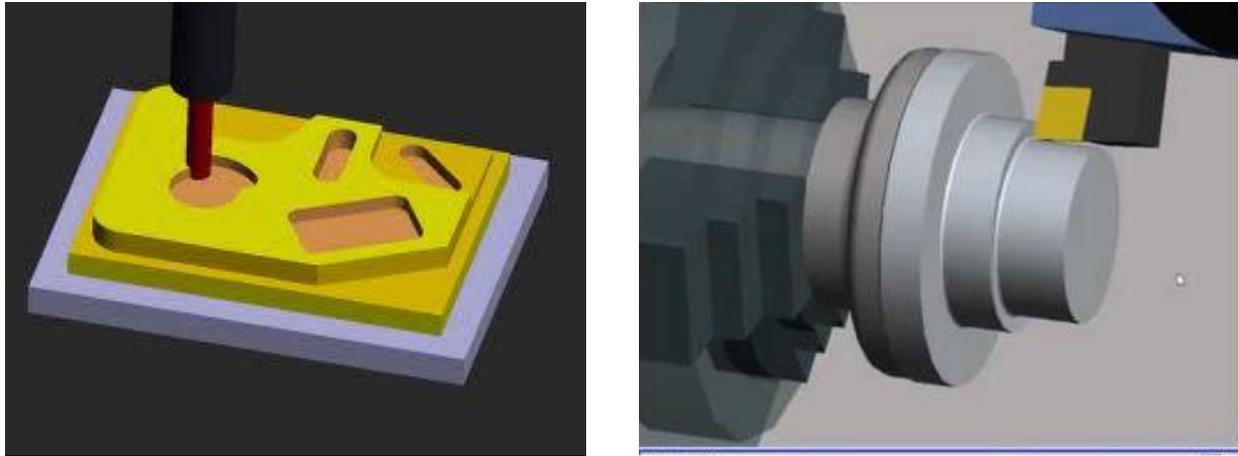
Figure 16: Toolpath Simulation in CAMWorks

Figure 17: Sample Set Up Sheet

Step 9: Create Setup Sheet
The setup sheet is a printable file that contains information that the machine tool operator can use to set up the stock and the tools required to produce a part. The information includes the machine, the controller, estimated machine time, the part material, and the tooling used to machine the part. The default set up sheets in CAMWorks contain common information a machine operator needs to setup a job. However, these can be customized for specific facilities. Set up sheets can include work in process images which allow the machine operator to graphically see the progress of the part for each operation. Figure 17 shows a sample set up sheet.

Step 10: Generate NC Code
Here the G-code needed to run the CNC machines that will make the parts are generated. G-code is a language which tells computerized machine tools how to make a part. It contains instructions on where to move, how fast to move, and through what path to move. CNC Machines have cutting tools which move according to these instructions through a toolpath, cutting away excess material to leave only the finished workpiece. Figure 18 shows a sample G-code.

```
%
O0001(PARTNAME=JSF300701-Plate)
G40G0
M5
G91G30Z0M19
G30X0Y0
T66 (1" 4 FLUTE HSS EM)
M6
T64
S4000M3
G0G90G54G17G49
G0X-4.2625Y3.9975
G332R2.
G05.1Q1
G43Z7.36H66M8
G1Z6.76F7.5M8
X-2.0109F60.
G3X-2.0232Y3.9856R.51
```

Figure 18: Sample G-Code

CAMWorks Milling Operations

Steps to Generating Your CNC Code (For 2.5 Axis Machining):
1. **Open a part in SolidWorks**
   a. Other types of files such as IGES, STL, sat files etc. can also be opened.

2. **Define the Machine (Found at CAMWorks Feature Tree Tab)**
   a. Switch to CAMWorks feature tree tab.
   b. Right click on the machine to define the opened machine dialogue box:
      i. Click on Machine tab to select your machine.
      ii. Click on Tool Crib to select or add your tool(s).
      iii. Click on Post Processor tab to select the type of processor to be used (Haas for CNC milling operation).
      iv. Click on Posting tab to enter any information that may be specific for your program that will appear on setup sheet.
      v. Click OK at the bottom to accept all your selections.

3. **Define the Stock or Material Bounding Box**
   a. Click on the stock manager icon or right click on stock manager and select Edit definition option to control the parameters of the stock (stock parameters include stock size, bounding box offsets, and material type).
   b. Enter the offset values as needed. (These offsets establish the actual size of the material to be cut)..
   c. Select the type of material to be machined. (This will help determine the speed and feeds to be generated)
   d. Click OK at the bottom to accept your selections.

4. **Define the Features to Machine**
   a. Click the Extract Machinable Features icon.
   b. (CAMWorks will automatically identify the machinable features on the part and recommend the operations needed to machine them).
   c. The features will be listed under part setup tree in the order recognized by CAMWorks.
   d. Features can be edited by right-clicking on each of them and selecting Parameter which opens a window for making changes.
   e. A second approach is to define features interactively by right-clicking on Part Setup and selecting Insert 2.5 Axis Feature (or other specific features) and then selecting the features manually/interactively from the window that appears.

   4B. Sometimes it is not possible to define machinable features automatically as described above. In such a situation, it is better to define the features interactively as follows:

   Defining Machinable Features Interactively
   1. Right click on Part Setup.

2. Select Insert 2.5 Axis Feature (the wizard feature window opens).
3. Select the type of feature from the Type dropdown list (often it is the Face feature if the feature is on top of the part).
4. Select the Entities for the Face feature by collapsing the dialog box and selecting the top Face of the part (by clicking at it with your mouse).
5. Expand the dialog box and the Face is added to the Entities window at the bottom.
6. Click Next to continue.
7. Select the End condition type in the next window.
8. Click Next.
9. Click Finish.
10. Click Close to close the window.
11. The new feature should now show as having been added to the feature tree.

5. Generate Operations
   a. Click on Generate Operation Plan icon (or right-click on Part Setup and select Generate Operation Plan).
   b. A list of operations will be generated under the setup.
   c. Sequence of operations can be changed by using the cursor to move them around as needed.
   d. Operations can be edited, renamed, hidden, suppressed/unsuppressed, deleted/restored etc.

6. Establish Program Zero or Reference Point
   a. Double-click on Part Setup from the operation tree.
   b. Click on the Origin tab.
   c. Use the XYZ coordinate system to specify the zero point for the tool (Stock Vertex option is recommended for rectangular parts).
   d. Click on the Axis tab to adjust the orientation of the triad to match values of the SolidWorks part drawing.
   e. Click on the Offset tab to specify the work coordinates.
   f. (Clicking on the Statistics tab will show information like length of time it will take to machine the part etc.).
   g. Click OK to accept and exit the window.

7. Generate Tool Paths
   a. Click on Generate Toolpath icon to create tool paths automatically (or right-click on the Part Setup and select Generate Toolpath).

8. Simulate Tool Paths
   a. Click on Simulate Toolpath icon (or right-click on the Part Setup and select Simulate Toolpath).
   b. (The simulation tool bar appears).

     c. Click on individual simulation play buttons to review (and edit) the simulated way the part will be cut.

9. **Create Setup Sheets**
     a. (This step is done ONLY after making all necessary adjustments to your operations and toolpaths).
     b. Right-click on the NC Manager.
     c. Select Setup Sheet.
     d. Select Generate.
     e. (A Setup Sheet Options window appears. Select the XML format).
     f. Click OK.
     g. (The setup sheet is generated).

10. **Generate NC Code**
     a. Click on the Post Process button in the CAMWorks toolbar (or right-click on the NC Manager and select Post Process).
     b. (The Post dialogue box appears so that the file name of the NC program can be specified).
     c. Save the file on a flash drive with a .nc file extension.
     d. (The postprocess output dialogue box appears).
     e. Click on the Play or Fast button and the NC code will be generated.
     f. Click OK to exit the window.

CAMWorks Milling Operations Videos

This section has videos designed by CAMWorks to help you quickly learn how to use the CAMWorks commands tools. You are encouraged to view as many of them as possible. The contents of these videos will enhance your learning of CAMWorks' various milling commands.

The numbers to the right of each video show the length of time in minutes and seconds of each video. The videos can be downloaded by googling the titles or by registering with CAMWorks who will provided access passwords with fees.

CAMWorks Interface
Introduces the various parts of the CAMWorks Interface   2m 3s

Process Overview
Outlines the steps required to machine a part using CAMWorks   0m 58s

Intro to 2.5 Axis Milling
Introduction to 2.5 Axis Milling   0m 37s

Defining the Machine
Shows how to define the machine that will be used to machine the part   4m 22s

Defining the Stock - Bounding Box

Shows how to define the stock using a bounding box   2m 21s

Defining the Stock - Sketch
Shows how to define the stock by using a sketch in SOLIDWORKS   4m 30s

Defining the Stock from STL
Shows how to define the stock by importing an STL file   1m 19s

Defining Features Automatically
Shows how to use Automatic Feature Recognition to define part features   4m 34s

Defining Features Interactively
Shows how to define part features interactively   3m 22s

Generate Operation Plan
Shows how to create an operation plan with the operations required to
machine the part   3m 23s

Establishing Program Zero
Shows how to establish the program zero about the part   2m 24s

Generating Toolpaths
Shows how to create toolpaths   0m 54s

Simulate Toolpaths
Shows how to simulate toolpaths   5m 10s

Reordering Operations
Shows how to reorder operations in the Operation Tree   0m 39s

Operation Parameters- Tool Adjustments
Shows how to change tools for operations   2m 35s

Operation Parameters- Feeds and Speeds
Shows how to set the feeds and speeds for the part   1m 10s

Adjusting Operations - Roughing Options A
Shows how to adjust operation parameters for roughing operations   3m 56s

Roughing Options B
Shows how to adjust operation parameters for roughing operations   3m 13s

VoluMill
Explains the options for the VoluMill add-in   4m 5s

Adjusting Operations - NC Parameters

Shows how to adjust the NC Options   2m 23s

Feature Options
Shows how to adjust feature options such as the entry type   3m 44s

Optimizing Toolpaths
Shows how to use options for optimizing toolpaths   3m 30s

Contour Operations
Shows how to adjust operation parameters for finishing operations   5m 16s

Creating Setup Sheets
Shows how to create setup sheets   1m 18s

Post Processing Toolpaths
Shows how to create the NC Code by post processing toolpaths   2m 21s

Inserting Additional Operations
Shows how to insert additional operations into the Part Setup   3m 1s

Sorting Operations
Shows the options for sorting operations   2m 44s

Combining and Linking Operations
Shows how to combine and link similar operations   2m 25s

Inserting Avoid Areas
Shows how to define areas that should be avoided by the toolpath   4m 47s

Inserting Contains
Shows how to define areas that toolpaths should be contained within   1m 45s

Part Exercise IA
Part Exercise   4m 7s

Part Exercise IB
Part Exercise continued   2m 6s

Part Exercise IC
Part Exercise continued   2m 13s

2.5 Axis Features
Introduces the various types of 2.5 axis features   0m 25s

Machinable Features
An Overview of the different Machinable Features   1m 14s

Face Feature
Identifies face features and shows how to define them interactively   1m 10s

Pockets I
Identifies pocket features and shows how to define them interactively   3m 14s

Pockets II
Identifies pocket features and shows how to define them interactively   2m 18s

Pockets III
Shows another application of the pocket feature   1m 44s

Bosses
Identifies boss features and shows how to define them interactively   3m 15s

Part Perimeters
Identifies boss features and shows how to define them interactively   1m 42s

Open Profiles
Identifies open profile features and shows how to define them interactively 1m 47s

Slots
Identifies slot features and shows how to define them interactively   1m 3s

Holes
Identifies hole features and shows how to define them interactively   1m 47s

Linear Pattern Feature
Shows how to define features that are arranged in a linear fashion   2m 59s

Circular Pattern Feature
Shows how to define features that are arranged in a circular fashion   2m 48s

Engraving
Shows how to create engrave features for machining   2m 42s

Introduction (to Technology Database)
Introduction to the Technology Database   1m 50s

Adding Machines
Shows how to add a machine to the Technology Database   2m 59s

User Defined Strategies - A
Shows how to begin setting up user defined attributes in the
Technology Database   8m 39s

User Defined Strategies - B
Shows how to associate specific features and operations with user
defined attributes   3m 44s

Adding Tools
Shows how to add a custom tool to the technology database   2m 8s

Setting Up Tool Cribs
Shows how to set up tool cribs in the Technology Database   2m 2s

Linking the Database
Shows how multiple users can link to a shared Technology Database
on a network   1m 45s

Restoring the Database
Shows how to restore the Technology Database   2m 13s

CAMWorks Options
Explains some helpful options in CAMWorks   4m 15s

CAMWorks Turning Operations

Steps to Generating Your CNC Code (For 2.5 Axis Turning):
  1. **Open a Part in SolidWorks**
     a. Other types of files such as IGES, STL, sat files etc. can also be opened.

  2. **Define the Machine (Found at CAMWorks Feature Tree Tab)**
     a. Switch to CAMWorks feature tree tab.
     b. Right-click on the machine in CAMWorks feature manager tree and select Edit Definition to bring out the machine dialogue box.
     c. Select Turn-in and click Select.
     d. Click on Tool Crib tab to select or add the tool of your choice.
     e. Click on Post Processor tab to define the post processor for the machine that will be needed.
     f. Click OK to accept all the inputs.
     g. Click on setup tab to select the origin that will be used to generate the G-code.
     h. Select automatic for method and XZ plane for turn feature plane and turn feature display plane.
     i. Click OK to accept and exit the window.

  3. **Define the Stock or Material Bounding Cylinder**
     a. Click on the stock manager icon or right click on stock manager and select Edit definition option to control the parameters of the stock (bar stock option, revolved sketch, work in process sketch, and STL options).

b. Select the option for round bar Stock (the preview will highlight the stock in the CAMWorks graphic window area).
c. Click on the material tab and select the type of material for the part.
d. Use the arrows to adjust the bar stock parameters and bounding cylinder as needed.

4. **Define the Features to Machine**
   a. Click the Extract Machinable Features icon.
   b. (CAMWorks will automatically identify the machinable features on the part and recommend the operations needed to machine them).
   c. The features will be listed under part setup tree in the other recognized by CAMWorks.
   d. Features can be edited by right-clicking on each of them and selecting Parameter which opens a window for making changes.

   4B. Sometimes it is not possible to define machinable features automatically as described above. In such a situation, it is better to define the features interactively as follows:

   Defining Machinable Features Interactively
   a. Right click on Turn Setup.
   b. Select Insert Turn Feature (the wizard feature window opens).
   c. Select the type of feature from the Feature dropdown list. (Select other options as needed).
   d. Specify the location of the selected feature by selecting from the Part Profile Method dialogue menu how you want the features to be selected.
   e. Click on the part's graphic section that you want to select and it will be highlighted.
   f. Click on Selected Entities and Extend boxes to further the selection as needed.
   g. Click OK (green check) to add the turn feature to the Feature Manager tree. (Note that these features can be rearranged in the order needed by click-dragging them).
   h. The new feature should now show as having been added to the feature tree.
   i. Repeat b-g for the other features you need to add.

5. **Generate Operations**
   a. Click on Generate Operation Plan icon (or right-click on Part Setup and select Generate Operation Plan).
   b. A list of operations will be generated under the setup.
   c. Sequence of operations can be changed by using the cursor to move them around as needed.
   d. Operations can be edited, renamed, hidden, suppressed/unsuppressed, deleted/restored etc.

e. Once operations have been generated, it is important to check to ensure the operations are sequenced in the order desired. Operations can be click-dragged and placed as needed. The pink color indicates that CAMWorks is waiting for next operation.

f. To setup the chuck, right-click on turn setup and select edit definition.

g. In the operations setup parameters window that appears, select appropriate options under origin and chuck location as needed. Be sure to select or set the appropriate chuck parameters.

5B. Sometimes the system does not generate all required operations automatically as described above. Or the programmer may need to add certain operations to the original design. In such a situation, it is better to manually insert the operations interactively as follows:

Inserting Operations
a. While in Feature Tree window, right click on the operation that will precede the needed operation and select the particular Insert Operation.

b. Select type of operation under Operation Type option.

c. Associate the selected operation with a feature by selecting the feature from the Features option.

d. Click the forward arrow to see the next windows and then define the operation.

e. Define the operation by copying from the existing operations under Parameter Default or by using a default parameter found in the database.

f. With the Edit Operation on Creation option checked, the Operations parameters window will appear when you click on the green check mark.

g. Make changes as needed in this window and click OK and notice that the selected operation has been added to the Operation Tree.

h. The operation under this tree can be click-dragged to correctly sequence them in the order they will be machined.

i. Repeat steps a-h as needed until the required operations have been added.

j. Right-click on Turn Setup to regenerate the operations.

k. You should see that the new operations have been added to the tree.

6. **Establish Program Zero or Reference Point (If not Done in Step 5 Above)**
   a. To setup the chuck, right-click on turn setup and select edit definition.
   b. In the operations setup parameters window that appears, select appropriate options under origin and chuck location as needed. Be sure to select or set the appropriate chuck parameters.
   a. Click OK to accept and exit the window.

7. **Generate Tool Paths**
   a. Click on Generate Toolpath icon to create tool paths automatically (or right-click on the Part Setup and select Generate Toolpath).

8. **Simulate Tool Paths**
    a. Click on Simulate Toolpath icon (or right-click on the Part Setup and select Simulate Toolpath).
    b. (The simulation tool bar appears).
    c. Click on individual simulation play buttons to review (and edit) the simulated way the part will be cut.

9. **Create Setup Sheets**
    a. (This step is done ONLY after making all necessary adjustments to your operations and toolpaths).
    b. Right-click on the NC Manager.
    c. Select Setup Sheet.
    d. Select Generate.
    e. (A Setup Sheet Options window appears. Select the XML format)
    f. Click OK.
    g. (The setup sheet is generated).

10. **Generate NC Code**
    a. Click on the Post Process button in the CAMWorks toolbar (or right-click on the NC Manager and select Post Process).
    b. (The Post dialogue box appears so that the file name of the NC program can be specified).
    c. Save the file on a flash drive with a .nc file extension.
    d. (The postprocess output dialogue box appears).
    e. Click on the Play or Fast button and the NC code will be generated.
    f. Click OK to exit the window.

CAMWorks Turning Operations Videos

This section has videos designed by CAMWorks to help you quickly learn how to use the design command tools. You are encouraged to view as many of them as possible. The contents of these videos will enhance your learning of CAMWorks various commands.

CAMWorks Turning walks you step by step through the entire process of generating G-code for a lathe using the CAMWorks Add-in. Users will be able to set the stock size, define the machinable features, create, simulate, and adjust toolpaths, and automatically output G-code.

The numbers to the right of each video show the length of time in minutes and seconds of each video. The videos can be downloaded by googling the titles or by registering with CAMWorks who will provided access passwords with fees.

CAMWorks Interface
Introduces the various parts of the CAMWorks Interface   2m 3s

Process Overview
Outlines the steps required to machine a part using CAMWorks   0m 58s

Introduction (to CAMWorks Turn Turning)
Introduction to Turning   0m 22s

Defining the Machine
Shows how to define the machine that will be used to machine the part   5m 2s

The Setup Tab
Finish defining the machine using the setup tab   2m 3s

Defining the Stock - Bar Stock
Shows how to define bar stock   1m 55s

Defining the Stock - Revolved Sketch
Shows how to define stock using a revolved sketch   2m 38s

Defining the Stock - WIP file and STL
Shows how to define stock using Work in Process or STL file   0m 56s

Defining Features Automatically
Shows how to use Automatic Feature Recognition to define part features 2m 4s

Defining Features Interactively
Shows how to define part features interactively   3m 39s

Generating an Operation Plan
Shows how to create an operation plan with the operations required to
machine the part   3m 7s

Generating Tool Paths
Shows how to create toolpaths for all of the operations   0m 52s

Simulating Tool Paths
Shows how to simulate toolpaths   5m 49s

Tool Adjustments
Shows how to adjust tools to get the correct cuts   3m 49s

Roughing Options
Shows how to adjust Roughing Options   3m 59s

Finishing Options
Shows how to adjust Finishing Options   2m 38s

Rough Grooving Options
Shows how to adjust Rough Grooving options   5m 17s

Finish Grooving Options
Shows how to adjust Finish Groove Options   0m 54s

NC Options
Shows how to adjust Numerical Control Options   3m 23s

Lead In / Lead Out Options
Shows how to adjust lead in and lead out options   1m 55s

Create Setup Sheets
Shows how to create setup sheets   1m 46s

Post Processing
Shows how to create the NC Code by post processing toolpaths   2m 27s

Inserting Operations
Shows how to insert additional operations   3m 1s

Setting up a Chuck
Shows how to set up a chuck for your part   5m 20s

Part Exercise 1A   1m 41s

Part Exercise 1B   4m 57s

Part Exercise 1C   6m 21s

Face Feature
Gives an overview, and shows how to add a face feature   1m 40s

OD Feature
Explains the outer diameter or OD feature   2m 47s

Grooves
Shows and explains how to add a groove feature   1m 45s

ID Feature
Explains and shows how to add an inner diameter or ID feature   1m 44s

Cutoff Feature
Shows how to add and apply a cutoff feature   2m 8s