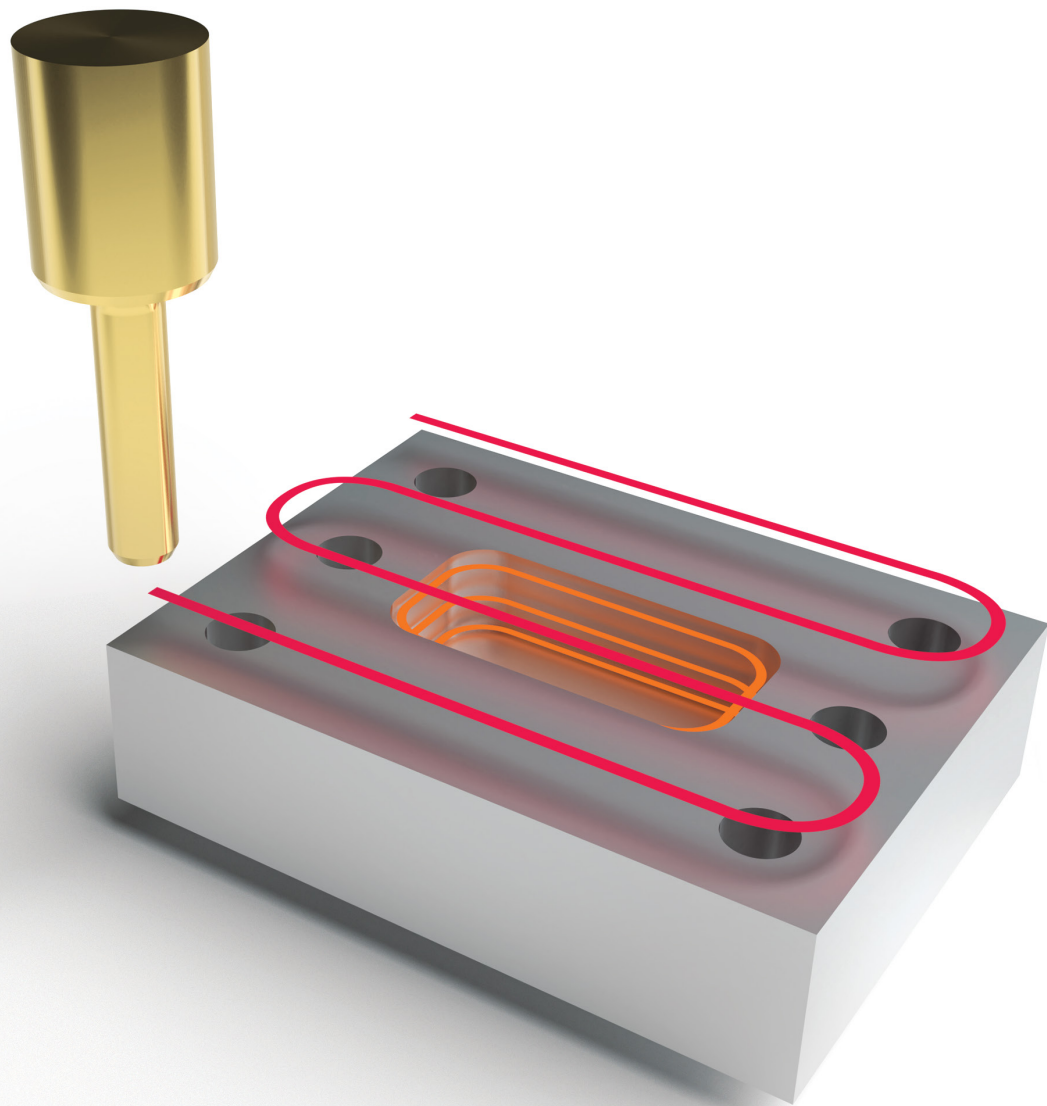Kuang-Hua Chang, Ph.D.

MACHINING SIMULATION USING
SOLIDWORKS® CAM 2018

Visit the following websites to learn more about this book:
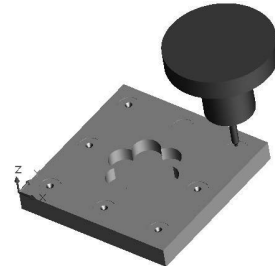
Lesson 1:   Introduction to SOLIDWORKS CAM

1.1   Overview of the Lesson

SOLIDWORKS CAM (www.solidworks.com/product/solidworks-cam), powered by CAMWorks (www.camworks.com), is a parametric, feature-based virtual machining software offered as an add-in to SOLIDWORKS. Such an add-in module supports users to integrate design and manufacturing in one application, connecting design and manufacturing teams through a common software tool that facilitates product design in 3D solid model. By defining areas to be machined as machinable features, SOLIDWORKS CAM is able to apply more automation and intelligence into CNC (Computer Numerical Control) toolpath creation. This approach is more intuitive and follows the feature-based modeling concepts of computer-aided design (CAD) systems. Because of this integration, you can use the same user interface and solid models for design and later to create machining simulation. Such a seamless integration completely eliminates file transfers using less-desirable standard file formats such as IGES, STEP, SAT, or Parasolid. Hence, the toolpaths generated are on the SOLIDWORKS solid model, not on an imported approximation. In addition, the toolpaths generated are associative with SOLIDWORKS parametric solid model. This means that if the solid model is changed, the toolpaths are changed automatically with minimal user intervention.

One unique feature of SOLIDWORKS CAM is the AFR (automatic feature recognition) technology. AFR automatically recognizes machinable features in solid models of native format or neutral file format; including mill features such as holes, slots, pockets and bosses; and turn features such as outside and inside diameter profiles, faces, grooves and cutoffs. This capability is complemented by interactive feature recognition (IFR) for recognizing complex multi-surface features, as well as creating contain and avoid areas.

Another powerful capability found in SOLIDWORKS CAM is its technology database, called TechDB™, which provides the ability to store machining strategies feature-by-feature, and then reuse these strategies to facilitate the toolpath generation. Furthermore, TechDB™ is a self-populating database, which contains information about the cutting tools and the machining parameters used by the operator. It also maintains information regarding the cutting tools available on the shop floor. This database within SOLIDWORKS CAM can be customized to meet the user's and the shop floor's requirements. This database helps in storing the best practices at a centralized location in support of machining operations, both in computers and on the shop floors.

We set off to discuss NC part programming and learn to use SOLIDWORKS CAM NC Editor and other NC viewers to review and verify the NC part program or G-code. With the basic understanding of NC part programming, we then start the main topic of the book: virtual machining simulation. We discuss the topic by exploring and learning capabilities offered by SOLIDWORKS CAM. The lessons and examples offered in this book are carefully designed and structured to support readers becoming efficient in using

SOLIDWORKS CAM and competent in carrying out virtual machining simulation for general machining applications.

We assume that you are proficient with part and assembly modeling capabilities in SOLIDWORKS, have some knowledge about NC part programming and the format and semantics of the G-code, and understand the practical aspects of setting up and conducting machining operations on CNC machines at the shop floor. Therefore, this book starts with a brief review on NC part programming so that readers will be able to read and revise, if necessary, the G-code generated from SOLIDWORKS CAM. We then focus on presenting virtual machining simulation using SOLIDWORKS CAM for toolpath and G-code generations. In addition to learning SOLIDWORKS CAM, we present two applications lessons, one for milling and one for turning operations, in which we load the G-code generated to CNC mill and lathe, respectively, and carry out physical machining operations at the shop floors. In these two lessons, we discuss the important steps in transition from virtual machining to physical material cutting at the shop floor, point out numerous practical aspects of CNC machining, and verify the accuracy of the G-code post processed by SOLIDWORKS CAM.

Although SOLIDWORKS CAM is useful and powerful in support of most machining assignments, capabilities offered in the 2018 version are somewhat limited for machining freeform surfaces often found in the die and mold machining operations. There are other third-party CAM modules fully integrated into SOLIDWORKS that offer more capabilities than SOLIDWORKS CAM. At the end of the book (Lesson 14), we introduce three such modules, including CAMWorks, HSMWorks (www.autodesk.com/products/hsm/overview), and Mastercam for SOLIDWORKS (www.mastercam.com/en-us), to show readers options available when it comes to machining parts that involve freeform surfaces.

1.2   Virtual Machining

Virtual machining is a simulation-based technology that supports engineers in defining, simulating, and visualizing machining operations in a computer environment using computer-aided manufacturing (CAM) tools, such as SOLIDWORKS CAM. Working in a virtual environment offers advantages of ease in making adjustments, detecting errors and correcting mistakes, and understanding machining operations through visualization of machining simulations. Once finalized, the toolpaths can be converted to G-code and uploaded to a CNC machine at the shop floor to physically machine parts.

The overall process of using SOLIDWORKS CAM for creating machining simulation, , as illustrated in Figure 1.1, consists of several steps: create design model (solid models in SOLIDWORKS part or assembly), choose NC machine and create stock, extract or identify machinable features, generate operation plan, generate toolpath, simulate toolpath, and convert toolpath to G-code through a post processor. Note that before extracting machinable features, you must select an NC machine; i.e., mill or lathe, choosing tool cribs, selecting a suitable post processor, and creating a stock.

The operation plan involves the NC operations to be performed on the stock, including selection of part setup origin, where G-code program zero is located. Also included is choosing cutting tools, defining machining parameters, such as feedrate, stepover, depth of cut, etc. Note that operation plans can be automatically generated by the technology database of SOLIDWORKS CAM when a machinable feature is extracted or created manually beforehand. Users may make changes to any part of the operation plan, for instance, choosing a different cutting tool (also called cutter or tool in the book), entering a different feedrate, adjusting depth of cut, and so on. After an operation plan is complete, SOLIDWORKS CAM generates toolpath automatically. Users may carry out material removal simulation, step through machining toolpath, and review important machining operation information, such as machining time that contributes largely to the manufacturing cost.

The design model (also called part, design part or target part in this book), which is a SOLIDWORKS part representing the perfectly finished product, is used as the basis and starting point for all machining operations. Machinable features are extracted or created manually on the design model as references for individual toolpaths. By referencing the geometry of the design model, an associative link between the design model and the stock is established. Because of this link, when the design model is changed, all associated machining operations are updated to reflect the change.

The following example, a block with a pocket and eight holes shown in Figure 1.2, illustrates the concept of conducting virtual machining using SOLIDWORKS CAM. The design model consists of a base block (a boss extrude solid feature) with a pocket and eight holes that can be machined from a rectangular block (the raw stock shown in Figure 1.3) through pocket milling and hole drilling operations, respectively.

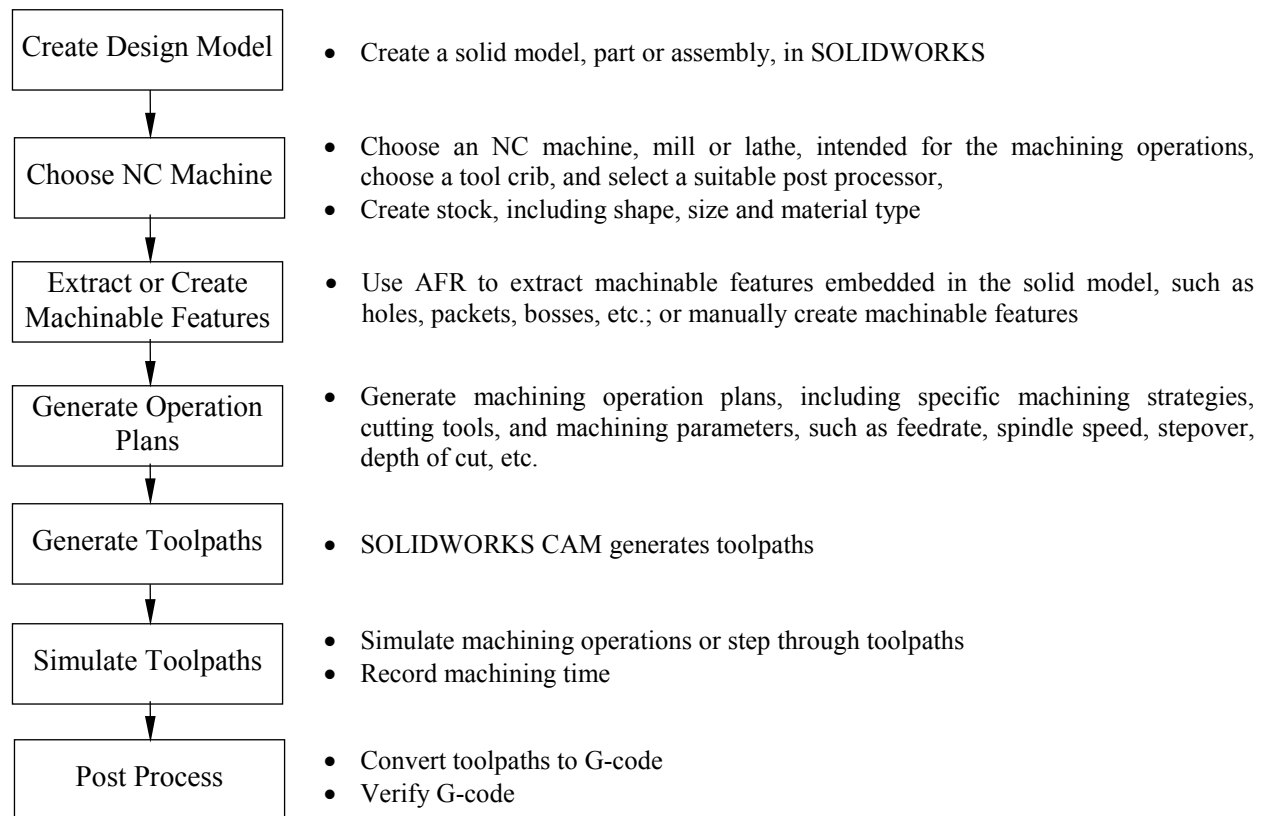|	
Create Design Model	• Create a solid model, part or assembly, in SOLIDWORKS
Choose NC Machine	• Choose an NC machine, mill or lathe, intended for the machining operations, choose a tool crib, and select a suitable post processor, • Create stock, including shape, size and material type
Extract or Create Machinable Features	• Use AFR to extract machinable features embedded in the solid model, such as holes, packets, bosses, etc.; or manually create machinable features
Generate Operation Plans	• Generate machining operation plans, including specific machining strategies, cutting tools, and machining parameters, such as feedrate, spindle speed, stepover, depth of cut, etc.
Generate Toolpaths	• SOLIDWORKS CAM generates toolpaths
Simulate Toolpaths	• Simulate machining operations or step through toolpaths • Record machining time
Post Process	• Convert toolpaths to G-code • Verify G-code

Figure 1.1 Process of creating machining simulation using SOLIDWORKS CAM
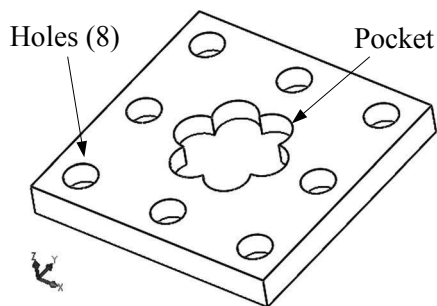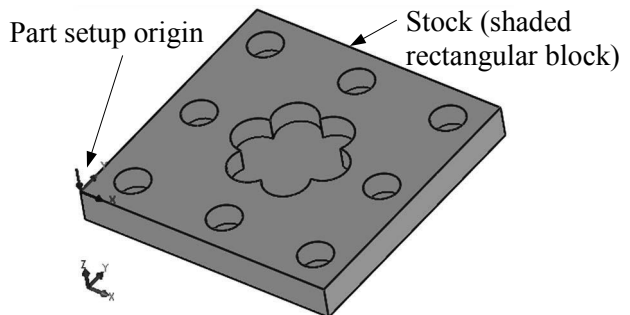

Figure 1.2 Design model in SOLIDWORKS

Figure 1.3 Stock enclosing the design model

A generic NC machine *Mill-in.* (3-axis mill of inch system) available in SOLIDWORKS CAM is chosen to carry out the machining operations. For example, toolpaths for machining the pocket (both rough and contour milling operations), as shown in Figure 1.4, can be generated referring to the part setup origin at the top left corner of the stock (see Figure 1.3). Users can step through the toolpaths, for example, the contour milling operation for cutting the pocket with tool holder turned on for display, as shown in Figure 1.5. The material removal simulation of the same toolpath can also be carried out like that of Figure 1.6.

Rough toolpath (shown in orange color)

Contour toolpath (shown in blue color)
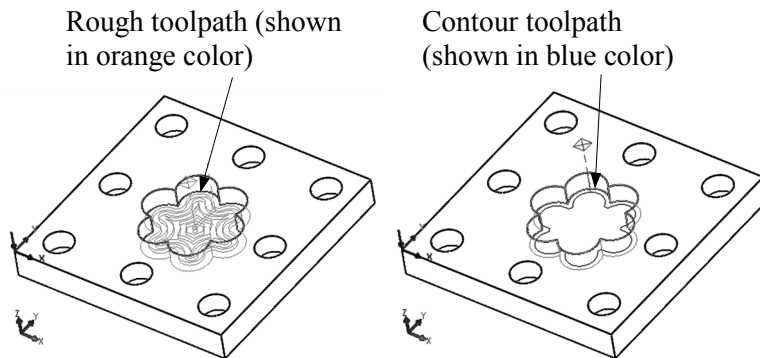
Figure 1.4 Toolpath of the pocket milling operations
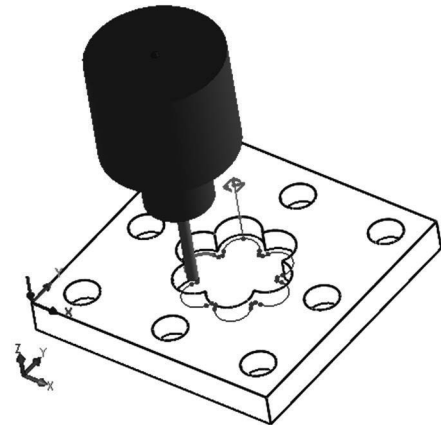
Figure 1.5 Step through toolpath

1.3   SOLIDWORKS CAM Packages

SOLIDWORKS CAM offers two packages, Standard and Professional. SOLIDWORKS CAM Standard supports you to quickly program individual milled parts and configurations without leaving the SOLIDWORKS 3D CAD environment. You have full access to defining rules within SOLIDWORKS CAM to create and build to your own standards. SOLIDWORKS CAM Standard also supports Tolerance-Based Machining (TBM).

SOLIDWORKS CAM Professional builds on the capabilities of SOLIDWORKS CAM Standard with increased programming capabilities. The additional features include High-Speed Machining (HSM), configurations, assembly machining, turning, and 3+2 programming to drive four- and five-axis machines[1].
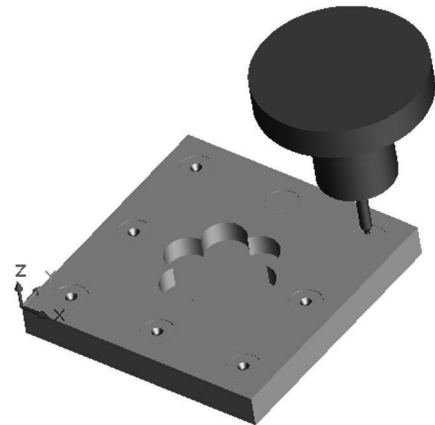
Figure 1.6 The material removal simulation

Please note that, as of 2018, SOLIDWORKS CAM offers 2½ axis (or 2.5 axis) milling capabilities. The 3+2 programming mentioned above is misleading. 3+2 programing really should be called 2½ + 2. It is not 4 or 5 axis machining. All it does is use the 4th and 5th axis for positioning. They do not move once positioned and all subsequent operations are really just 2½ axis motion. Users have to use other third-party CAM modules, such as CAMWorks, HSMWorks, or Mastercam (please review Lesson 14), for full three axis or more simultaneous axes of motion.

---

[1] Extracted from SOLIDWORKS website: www.solidworks.com/product/solidworks-cam

Overall, the machining modules included in SOLIDWORKS CAM support 2.5 axis milling and 2-axis turning; that is:

- 2.5 axis mill: includes roughing, finishing, thread milling, face milling and single point cycles (drilling, boring, reaming, tapping) to machine prismatic features;
- 2 axis turning: includes roughing, finishing, grooving, threading, cutoff and single point cycles (drilling, boring, reaming and tapping).

All the above capabilities are discussed in this book and are illustrated using simple yet practical examples. In addition, SOLIDWORKS CAM supports machining of multiple parts in a single setup. Parts are assembled as SOLIDWORKS assembly, which includes parts, stock, clamps, fixtures, and jig table in a virtual environment that accurately represents a physical machine setup at shop floor. A multipart machining example, as shown in Figure 1.7, with ten identical parts in an assembly will be introduced in Lesson 7. Furthermore, machining features on multiple planes of parts mounted on the respective four faces of the tombstone, as shown in Figure 1.8, in a single setup is supported. More about multiplane machining operations can be found in Lesson 8.
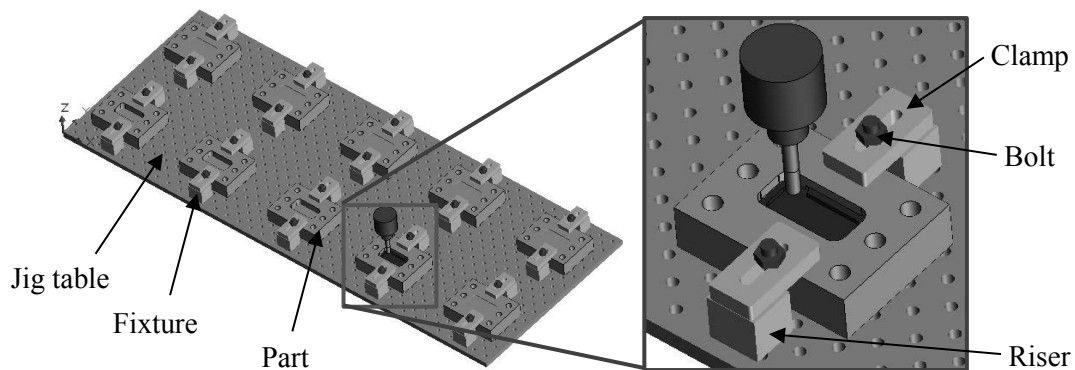

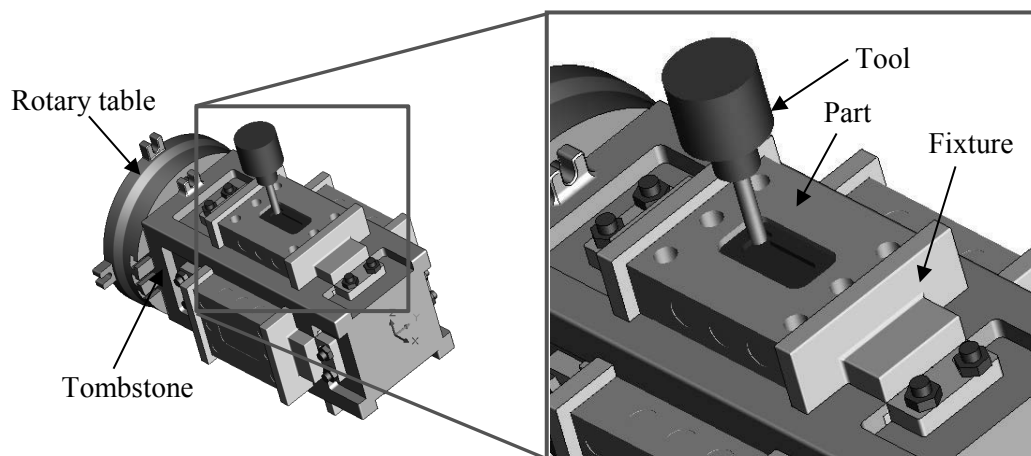Figure 1.7 The material removal simulation of a multipart machining example

Figure 1.8 Material removal simulation of the multiplane machining example

1.4   User Interface

The overall design of SOLIDWORKS CAM user interface, as shown in Figure 1.9 that includes the layout and windows, buttons, menu selections, dialog boxes, etc., is identical to that of SOLIDWORKS CAD. SOLIDWORKS users should find it is straightforward to maneuver in SOLIDWORKS CAM.

As shown in Figure 1.9, the user interface window of SOLIDWORKS CAM consists of pull-down menus, command buttons, graphics area, and feature manager window. An example file, *Lesson 1 with Toolpath.SLDPRT*, is prepared for you to browse numerous capabilities and become familiar with selections, buttons, commands and options of SOLIDWORKS CAM user interface. This file (and all example files of the book) is available for download at the publisher's website (www.sdcpublications.com). You may review Section 1.5 for the detailed steps to bring the example into SOLIDWORKS CAM.

The graphics area displays the solid model and machining simulation on the solid model with which you are working. The pull-down menus provide basic solid modeling functions in SOLIDWORKS and machining functions in SOLIDWORKS CAM. The command buttons of SOLIDWORKS CAM tab above the graphics area offer all the functions required to create and modify virtual machining operations in a generic order. Major buttons include extract machinable features, generate operation plan, generate toolpath, simulate toolpath, step through toolpath, and save CL file. When you move the mouse pointer over these buttons, a short message describing the function will appear. Some of the frequently used buttons in SOLIDWORKS CAM and their functions are summarized in Table 1.1 for your reference.
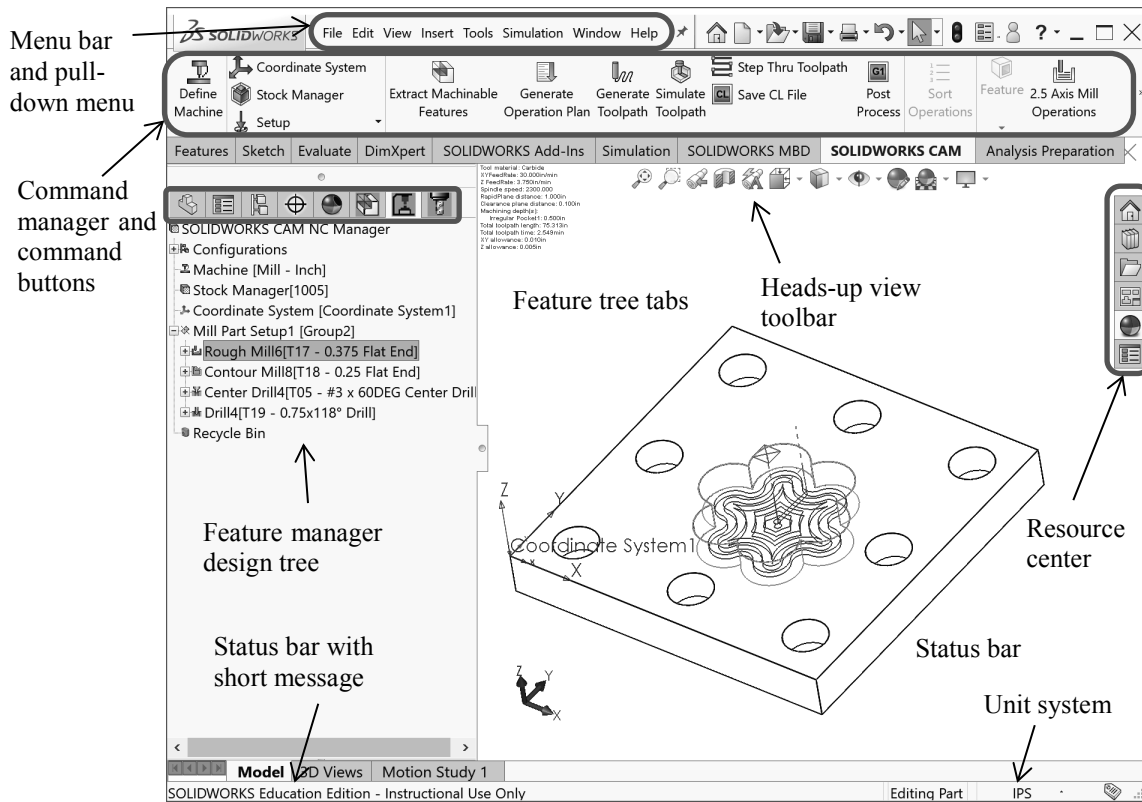

Figure 1.9 User interface of SOLIDWORKS CAM

Table 1.1 The major command buttons in SOLIDWORKS CAM

Button Symbol	Name	Function
Define Machine	Define Machine	Allows you to define the machine tool that the part will be machined on, such as 3-axis mill.
Coordinate System	Define Coordinate System	Allows you to define a coordinate system and assign it as the Fixture Coordinate System for the active machine.
Stock Manager	Stock Manger	Allows you to define the mill stock from a bounding box, an extruded sketch or an STL file.
Setup	Mill Part Setup	Allows you to create Mill Part Setups that defines (1) tool orientation (or feed direction), (2) G-code program zero, and (3) the X direction of tool motion.
Extract Machinable Features	Extract Machinable Features	Initiates automatic feature recognition (AFR) to automatically extract solid features that correspond to the machinable features defined in the technology database (TechDB^{TM}). The types of machinable features recognized for mill and turn are different. SOLIDWORKS CAM determines the types of features to recognize based on the NC machine selected. The machinable features extracted are listed in the feature manager window under the SOLIDWORKS CAM feature tree tab .
Generate Operation Plan	Generate Operation Plan	Generates operations automatically for the selected machinable features. The operations and associated parameters are generated based on rules defined in TechDB^{TM}. An operation contains information on how the machinable features are to be machined. The operations generated are listed in the feature manager window under the SOLIDWORKS CAM operation tree tab .
Generate Toolpath	Generate Toolpath	Creates toolpaths for the selected operations and displays the toolpaths on the part. A toolpath consists of cutting entities (line, circle, arc, etc.) created by a machining operation that defines tool motion.
Simulate Toolpath	Simulate Toolpath	Provides a visual verification of the machining process for the current part by simulating the tool motion and the material removal process.
Step Thru Toolpath	Step Through Toolpath	Allows you to view toolpath movements either one movement at a time, a specified number of movements or all movements.
Save CL File	Save CL File	Allows you to save the current operation and associated parameters in the technology database as CL (cutter location) data for future use.
Post Process	Post Process	Translates toolpath and operation information into G-code for a specific machine tool controller.

There are four feature tree tabs on top of the feature manager window that are highly relevant in learning SOLIDWORKS CAM. The left most tab, FeatureManager design tree 🎕 (see Figure 1.10), sets the display to the SOLIDWORKS design tree (also called model tree or solid feature tree), which lists solid features and parts created in SOLIDWORKS part and assembly in the feature manager window.

The third tab from the right, SOLIDWORKS CAM feature tree tab 📝 (see Figure 1.11), shifts the display to the SOLIDWORKS CAM feature tree, which lists machinable features extracted or manually created from the solid model. The tree initially shows only the *Configurations*, *Machine* (for example, *Mill-in* in Figure 1.11), *Stock Manager*, *Coordinate System*, and *Recycle Bin*. The *Machine* entity indicates the current machine chosen. You will have to select a correct machine before you begin working on a part. If you click any machinable feature, an outline view of the machinable feature appears in the part in the graphics area. For example, the sketch of the pocket appears when clicking *Irregular Pocket1* in the feature tree, as shown in Figure 1.11. Note that a symbol ⊗ (called tool axis symbol) appears indicating the tool axis direction (or feed direction) of the current mill part setup.
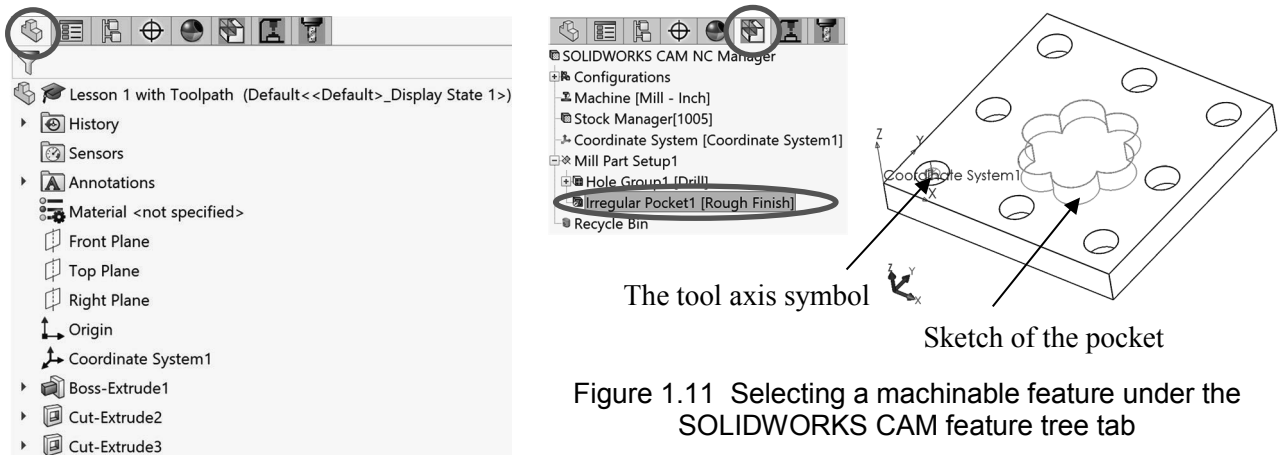

Figure 1.10 Solid features listed in the feature manager window under the FeatureManager design tree tab

The tool axis symbol

Sketch of the pocket

Figure 1.11 Selecting a machinable feature under the SOLIDWORKS CAM feature tree tab
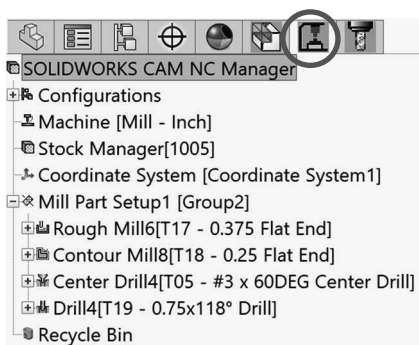

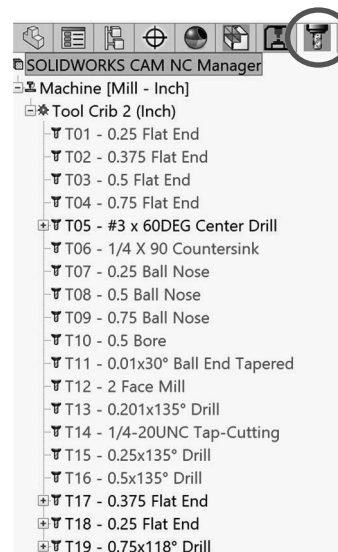Figure 1.12 SOLIDWORKS CAM operation tree tab

Figure 1.13 SOLIDWORKS CAM tools tree tab

The second tab from the right, SOLIDWORKS CAM operation tree ⬜ (shown in Figure 1.12), shifts the display to the SOLIDWORKS CAM operation tree. After you select the *Generate Operation Plan* command, the operation tree lists machining operations for the corresponding machinable features.

Similar to SOLIDWORKS, right clicking a machining operation in the operation manager tree will bring up command options that you can choose to modify or adjust the machining operation, such as feedrate, spindle speed, and so on. Clicking any operations after selecting the *Generate Toolpath* command will bring up the corresponding toolpaths displayed in the part in the graphics area, like that of Figure 1.4.

The rightmost tab, SOLIDWORKS CAM tools tree ⬜ (shown in Figure 1.13), changes the display to SOLIDWORKS CAM tools tree. SOLIDWORKS CAM tools tree lists tools available in the tool cribs chosen for the machining task.

1.5 Opening Lesson 1 Model and Entering SOLIDWORKS CAM

A machining simulation model for the simple block example shown in Figure 1.2 has been created for you. You may download all example files from the publisher's website, unzip them, and locate the model under Lesson 1 folder. Copy or move Lesson 1 folder to your hard drive.

Start SOLIDWORKS, and open model file: *Lesson 1 with toolpath.SLDPRT*. You should see a solid model file like that of Figure 1.2 appear in the graphics area.

Entering SOLIDWORKS CAM from SOLIDWORKS is straightforward. You may simply click the SOLIDWORKS CAM feature tree tab ⬜ or operation tree tab ⬜ to browse respective machining entities. You may right click any entity listed in the machining feature or operation tree to review or modify the machining model. You may also choose options under the pull-down menu *Tools > SOLIDWORKS CAM* to launch the same commands of those listed in Table 1.1 (and beyond) that support you to extract machinable features, generate an operation plan, and so on.

If you do not see the SOLIDWORKS CAM feature tree or operation tree tab, you may have not activated the SOLIDWORKS CAM add-in module. To activate the SOLIDWORKS CAM module, choose from the pull-down menu

Tools > Add-Ins

In the *Add-Ins* dialog box shown in Figure 1.14, click *SOLIDWORKS CAM 2018* in both boxes (*Active Add-ins* and *Start Up*) and then click *OK*. You should see that SOLIDWORKS CAM tab appears above the graphics area like that of Figure 1.9 and the three SOLIDWORKS CAM tree tabs (feature, operation, and tool) added to the top of the feature manager window.

If you still do not see SOLIDWORKS CAM tab appear above the graphics area, you may need to restart SOLIDWORKS to activate newly selected SOLIDWORKS CAM modules. Before going over this tutorial lesson, you are encouraged to check with your system administrator to make sure SOLIDWORKS and SOLIDWORKS CAM have been properly installed on your computer.

Another point worth noting is that the auto saving option might have been turned on in SOLIDWORKS CAM by default. Often, it is annoying to get interrupted by this auto saving function every few minutes asking if you want to save the model. You may turn this auto save option off by choosing from the pull-down menu

Tools > SOLIDWORKS CAM > Options

In the *Options* dialog box (Figure 1.15), select *Disable Auto Saving* under the *General* tab to turn it off. Then click *OK*.

To browser an existing SOLIDWORKS CAM model, you may click any machinable features listed under the SOLIDWORKS CAM feature tree tab 🔲 to display the feature in the graphics area. For example, the pocket profile sketch like that of Figure 1.11 appears in the solid model after clicking *Irregular Pocket1*.

You may also click any machining operation under SOLIDWORKS CAM operation tree tab 🔲 to display the toolpath of the selected operation. For example, clicking *Rough Mill* or *Contour Mill* displays toolpaths on the solid model like those of Figure 1.4. You may step through the toolpath of an operation (for example, see Figure 1.5) by right clicking it and choosing *Step Through Toolpath*. You may right click an operation and choose *Simulate Toolpath* to simulate a material removal process of the operation like that of Figure 1.6.

1.6   Extracting Machinable Features

Machining operations and toolpaths can be generated only on machinable features. A unique and appealing technical feature in SOLIDWORKS CAM is the automatic feature recognition (AFR) technology, which analyzes the solid features in the part and automatically extracts mill features such as holes, slots, pockets and bosses; and turn features such as outside and inside diameter profiles, faces, grooves and cutoff. The AFR technology helps in reducing the time spent by the designer to feed in data related to creating machining simulation.
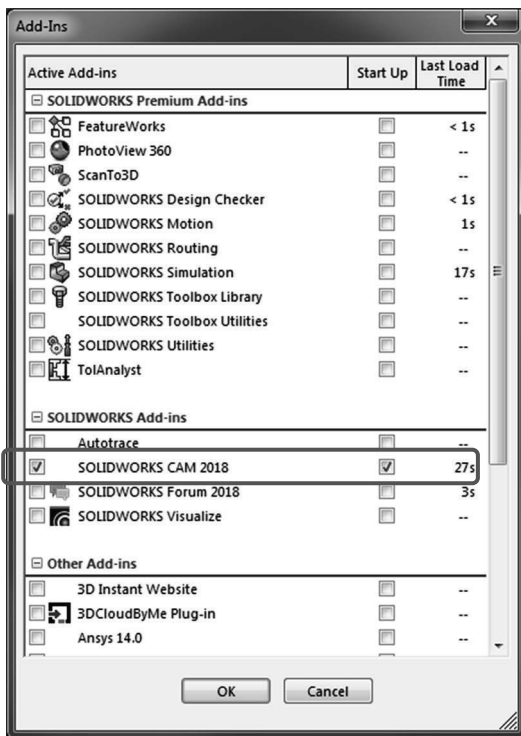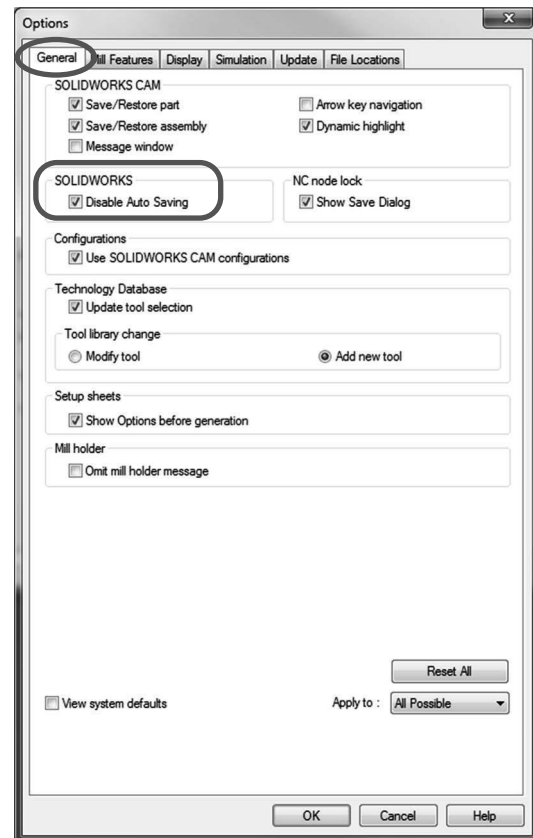

Figure 1.14 The *Add-Ins* dialog box

Figure 1.15 The *Options* dialog box

A set of machinable features for milling and turning operations that can be extracted by AFR are summarized in Appendix A. The associated machining strategies of individual machinable features can be found in Appendix B.

The *Extract Machinable Features* command initiates AFR. Depending on the complexity of the part, AFR can save considerable time in extracting 2.5 axis features, such as holes, pockets, slots, bosses, etc., either prismatic (with vertical walls) or tapered.

AFR cannot recognize every single feature on complex parts and does not recognize features beyond 2.5 axis. To machine these areas, you need to define machinable features manually using the interactive feature recognition (IFR) wizard. For example, you may define a *Multi Surface* feature manually by selecting faces to be cut and faces to avoid in the design model. More on this topic will be discussed in this book, for example, in Lesson 2: Simple Plate, and Lesson 6: Freeform Surface.

1.7 Technology Database

SOLIDWORKS CAM technology database, TechDB^{TM}, is a self-populating database which contains all the information about the machine, cutting tools and the parameters used by the operator, rules of repetitive NC operations (called strategy in SOLIDWORKS CAM) for the respective machinable features. This database within SOLIDWORKS CAM can be customized easily to meet the user's and the shop floor's requirements. This database helps the best practices at a centralized location in the tool room; thus, it eliminates the non-uniformity in practicing virtual and physical machining operations.

Using a set of knowledge-based rules, SOLIDWORKS CAM analyzes the machinable features and determines the machining process plan for a design model. In this approach, features are classified according to the number of possible tool approaching directions that can be used to machine them. The knowledge-based rules are applied to assure that you get desired cutting operations. The rules that determine machining operations for a respective machinable feature can be found in Appendix B, for both milling and turning operations.

The technology database is shipped with data that is considered generally applicable to most machining environments. Data and information stored in the database can be added, modified, or deleted to meet the user's specific needs in practice.

1.8 Tutorial Examples

There are fourteen lessons included in this book. In addition to the machining simulation example of the current lesson, we go over two lessons discussing NC part programming before resuming our discussion on more machining simulation lessons. In Lesson 2, we discuss (or more like review) NC part programming, in which we review NC address codes, preparatory functions, and auxiliary functions, before going over examples of pocket milling, trajectory milling, profile milling, and canned cycles. In Lesson 3, we introduce SOLIDWORKS CAM NC Editor, in which we learn to use the Editor to review and verify G-code.

The next eight lessons, Lessons 4 to 11, illustrate step-by-step details of creating machining operations and simulating toolpath capabilities in SOLIDWORKS CAM. We start in Lesson 4 with a simple plate example, which provides you with a brief introduction to SOLIDWORKS CAM 2018, and offers a quick run-through for creating a contour profile mill operation using a 3-axis mill.

Lessons 5 through 9 focus on milling operations. We include examples of machining 2.5 axis features using a 3-axis mill in Lesson 5, machining a freeform surface of a solid feature in Lesson 6, machining a set of identical parts in an assembly in Lesson 7, and machining features of parts on multiple planes using a 3-axis mill with a rotary table in Lesson 8. In Lesson 9, we discuss tolerance-based machining (TBM), which leverages SOLIDWORKS dimensions, tolerance ranges and surface finish annotations to select machining strategies for operations and machine parts to the mean of asymmetric tolerances.

Lessons 10 and 11 focus on turning operations. In Lesson 10, we use a simple stepped bar example to learn basic capabilities in creating turning operations and understanding G-code generated by SOLIDWORKS CAM. In Lesson 11, we machine a similar example with more turn features to gain a broader understanding of the turning capabilities offered by SOLIDWORKS CAM.

Lessons 12 and 13 discuss transition from virtual to physical machining, which is no small matter. In Lesson 12, we present an application extracted from a student project that involves milling operations for machining robotic rover forearm members. SOLIDWORKS CAM was employed to conduct virtual machining and toolpath generation for machining not only the forearm member but also a custom fixture. G-code generated by SOLIDWORKS CAM is then loaded to a HAAS CNC mill to machine the fixture and part. In Lesson 13, we discuss a turning application, in which we turn a scaled baseball bat on a HAAS CNC lathe using G-code created from SOLIDWORKS CAM. The goal of these two lessons is to offer readers a flavor of the role that SOLIDWORKS CAM is able to play in practical machining assignments. Moreover, we point out important factors for you to consider before transporting the results of virtual machining to CNC machines at the shop floor to physically cut the part.

In Lesson 14, we introduce third-party CAM modules that are fully integrated with SOLIDWORKS, including CAMWorks, HSMWorks, and Mastercam for SOLIDWORKS. In particular, we discuss milling operations involving full three axis or more simultaneous axes of motion, which are not supported in SOLIDWORKS CAM in the 2018 version. Note that you may be able to download trial versions of HSMWorks and Mastercam for SOLIDWORKS, install them on your computer, and go over Lesson 14.

One thing we emphasize in this book is the verification of the G-code converted from machining simulation. Learning the menu selections and button clicking of SOLIDWORKS CAM for generating machining simulation is important. On the other hand, the simulation must lead to something useful at the shop floor. That is, the G-code converted from machining simulation must be accurate and compactible with the NC machines at the shop floor. The G-code must be ready and able to produce parts as desired without major hurdles. Please note that no software is 100% error-proof and bug-free. Therefore, it is extremely important that we carefully review and verify the G-code before loading it to the NC machine for material cutting. At the end of most lessons, we review and verify the G-code converted. Readers are strongly encouraged to do the same while applying the skills learned from this book to their own machining projects.

All examples and topics to be discussed in individual lessons are summarized in Table 1.2.

<div align="center">Table 1.2 Examples employed and topics to be discussed in this book</div>

Lesson	Example	Machining Model	Problem Type	Topics
2	Trajectory milling, profile milling, pocket milling, hole drilling		Manual NC part programming	1. NC programming, 2. G-code and M-code, 3. Tool radius compensations, 4. Canned cycles, subroutines, and more.

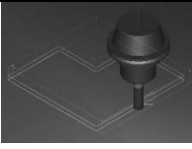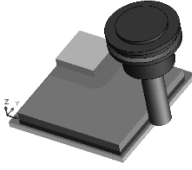Table 1.2 Examples employed and topics to be discussed in this book (cont'd)
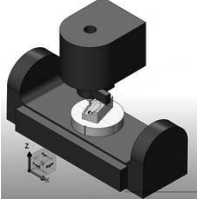
Lesson	Example	Machining Model	Problem Type	Topics
3	Trajectory milling, profile milling, hole drilling, turning operation		Review and verify G-code using SOLIDWORKS NC Editor	1. User interface of SOLIDWORKS NC Editor, 2. Backplotter, tool setup, coordinate system, and more, 3. Tool radius compensation, 4. Turning operations, 5. Toolpath simulation, 6. Other G-code viewers and editors.
4	Simple Plate		3-axis contour (or profile) milling	1. A brief introduction to SOLIDWORKS CAM 2018, 2. A complete process of using SOLIDWORKS CAM to create a milling operation from the beginning all the way to the post process that generates G-code, 3. Extracting machinable feature using interactive feature recognition (IFR), 4. Reviewing G-code generated.
5	2.5 Axis Features		3-axis pocket milling and hole drilling	1. Extracting 2.5 axis machinable features using automatic feature recognition (AFR), 2. Manually creating machinable feature for face milling operation, 3. Defining part setup origin for G-code generation, 4. Modifying machining parameters to regenerate toolpath, 5. Reviewing G-code generated.
6	Freeform Surface		Machine a freeform surface using 3-axis mill	1. Creating a multi-surface machinable feature and select avoid surface feature to restrain area of toolpath generation, 2. Creating *Area Clearance* (rough cut) and *Z Level* (finish cut) operations, 3. Creating section views for a closer look at the freeform surface in material removal simulation, 4. Limitations of SOLIDWORKS CAM.
7	Multipart Machining		Machine a set of identical parts in an assembly	1. Creating instances of part for machining operations, 2. Defining stocks for individual instances, 3. Selecting components in the assembly for the tools to avoid, 4. Reviewing G-code generated.

Table 1.2 Examples employed and topics to be discussed in this book (cont'd)

Lesson	Example	Machining Model	Problem Type	Topics
8	Multiplane Machining		Machine features on multiple planes using a 3-axis mill with a rotary table	1. Machining parts with machinable features on multiple planes, 2. Setting a rotation axis of the rotary table as the 4th axis, 3. Selecting components to be included in material removal simulation, 4. Rotating tool vs. rotating stock in material removal simulation, 5. Reviewing G-code generated.
9	Tolerance-Based Machining		Operation and toolpath generation based on tolerances	1. Defining dimension tolerances using *DimXpert*, 2. Tolerance-based machining settings, 3. Result interpretation.
10	Turning a Stepped Bar		Basic turning operations using 2-axis lathe	1. A complete process in using SOLIDWORKS CAM to create a turning simulation from the beginning all the way to the post process that generates G-code, 2. Manually creating machinable features, 3. Reviewing G-code generated.
11	Turning a Stub Shaft		Advanced turning operations	1. Extracting machinable features for turning, including face, groove, thread, and holes at both ends using AFR, 2. Manually creating turn thread feature, generating toolpath, 3. Reviewing G-code generated.
12	Machining Robotic Forearm Members		Practical application for milling operations	1. Solid model of the forearm member, 2. Machining strategy, 3. Custom fixture and stock-in-progress, 4. HAAS mill setup and stock installation, 5. G-code generated by SOLIDWORKS CAM, 6. Finished part.
13	Turning a Baseball Bat		Practical application for turning operations	1. Solid model of a scaled baseball bat, 2. Turning operations, 3. HAAS lathe setups and round stock installation, 4. G-code generated by SOLIDWORKS CAM, 5. Finished part.

Table 1.2 Examples employed and topics to be discussed in this book (cont'd)

Lesson	Example	Machining Model	Problem Type	Topics
14	Freeform surface example of Lesson 6		Freeform surface milling of 3-axis and multiaxis, and Machine Simulation	1. Third-party CAM modules for Multiaxis Surface Machining, 2. CAMWorks 5-axis surface milling operation and Machine Simulation in a setup with a tilt rotary table, 3. Volume milling, local milling, and surface milling using HSMWorks, 4. Volume milling, local milling, and multi-axis surface milling using Mastercam for SOLIDWORKS, plus Mastercam Simulator of a setup with a tilt rotary table.