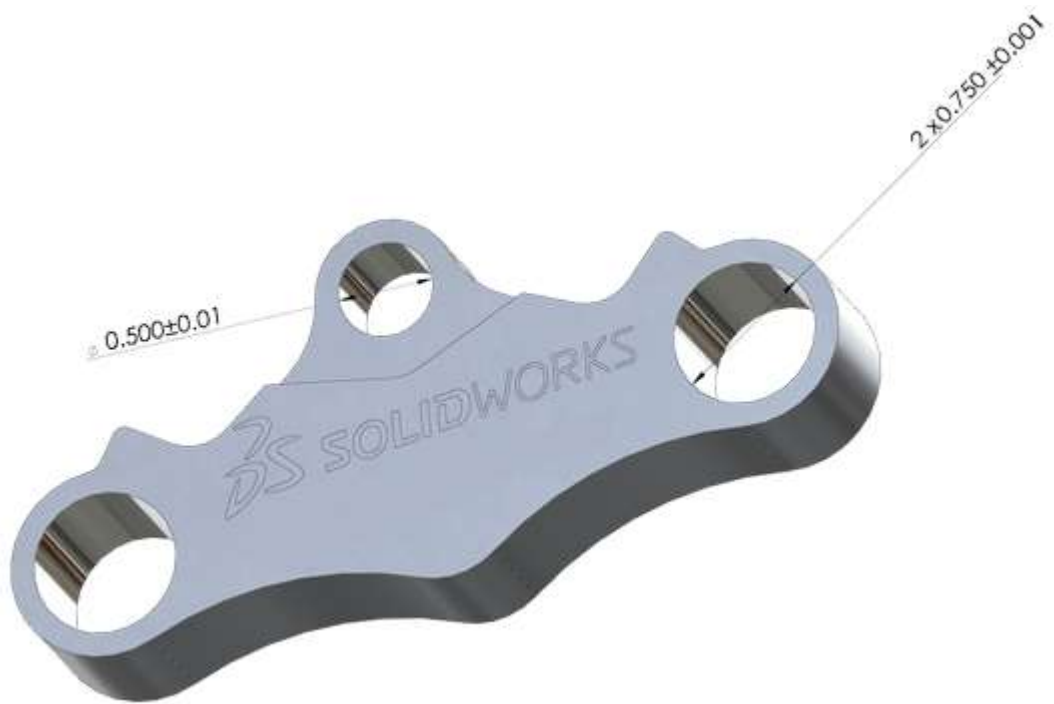


Tolerance Based Machining (SOLIDWORKS CAM - TBM)



User Guide & Tutorials

Copyright

© 1995-2019, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

Disclaimers

The information and the software application discussed in this document are subject to change without prior notice and are not commitments by Dassault Systemes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.




Trademarks

- SOLIDWORKS, 3D ContentCentral, 3D PartStream.NET, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.
- CircuitWorks, FloXpress, PhotoView 360, and TolAnalyst are trademarks of DS SolidWorks.
- FeatureWorks is a registered trademark of Geometric Americas, Inc.
- Adobe, the Adobe logo, Acrobat, the Adobe PDF logo, Distiller and Reader are registered trademarks or trademarks of Adobe Systems Inc. in the U.S. and other countries.
- Other brand or product names are trademarks or registered trademarks of their respective holders.





Product Name: SOLIDWORKS CAM 2020

TABLE OF CONTENTS

1. Introduction to 'SOLIDWORKS CAM - Tolerance Based Machining'.....	6
What is SOLIDWORKS CAM Tolerance Based Machining (TBM)?	6
Salient Features of SOLIDWORKS CAM TBM Plug-In.....	6
Pre-requisites for using SOLIDWORKS CAM TBM Plug-In	6
1. Embedded MBD/PMI Data on part to be machined	6
2. Run SOLIDWORKS CAM application on a supported CAD Platform	7
3. Adobe Reader XI	7
Launching the SOLIDWORKS CAM TBM Plug-in	7
Commands in SOLIDWORKS CAM TBM Command Manager	8
Run Tolerance Based Machining 	8
Tolerance Based Machining - Settings 	8
About SOLIDWORKS CAM TBM 	9
2. Assigning Tolerance Ranges & Associated Machining Strategies .	10
How is the Tolerance Range and Mean Tolerance value for a Feature computed?.....	10
Where are Tolerance Ranges and associated Machining strategies defined within SOLIDWORKS CAM TBM?	11
Parameters in the Tolerance Range tabs	11
Table of Features & Default Strategies.....	11
Feature	11
Default Strategy	11
No. of Tol based conditions	12
Tolerance Based Conditions grid.....	13
Tolerance Range.....	13
Undersize Strategy.....	13
Nominal Strategy.....	13
Oversize Strategy.....	13
Reassigning the Undersize, Nominal and Oversize Strategies.....	13
Defining /Editing Tolerance Ranges	13
How to use the Range dialog box to define /edit Tolerance Ranges.....	14
Special Case: Inputting only one single Tolerance limit.....	15
Flowchart: Assigning Tolerance Ranges & Strategies	16
3. Tutorial 1: Machining a Solid Part based on Tolerance Ranges	17
Step 1: Open the tutorial part model	17
Step 2: Defining Tolerance Ranges for the features	18



Step 3: Reassigning Strategies for Tolerance Ranges	21
Step 4: Execute the 'Run Tolerance Based Machining' Command	22
Analyzing how the Machining Strategies have been assigned	24
Interpreting the information in the Tolerance Range tab	24
Table of Features and Default strategies	25
'Tolerance range(s) available in the current part' grid	26
Interpreting the Integers within parentheses adjacent to Tolerance Range field	27
Computing Mean Tolerance Values for the Features on the part	27
Computing Tolerance Range and Mean Tolerance Value for the first Rectangular Pocket instance	28
Computing Tolerance Range and Mean Tolerance Value for the second Rectangular Pocket instance	29
Computing Tolerance Range and Mean Tolerance Value for the third Rectangular Pocket instance	30
Reassigning the Machining Strategies	31
4. Assigning Machining Strategies Based on ISO 286	32
What is ISO 286?	32
How does ISO 286 fit with SOLIDWORKS CAM TBM?	32
ISO 286 tab of Tolerance Based Machining – Settings dialog box	32
Table of Features and Default Strategies in the ISO 286 tab	33
Feature	33
Default Strategy	33
No. of ISO 286 Strategy Conditions	34
ISO Strategy Conditions grid	34
Designation	34
Size Range	34
Tolerance Values	34
Strategy	34
Add ISO 286 Strategy Conditions button	35
Steps to add/edit/delete ISO 286 based Strategy Conditions	35
Editing an ISO Strategy Based Condition/ Reassigning Machining Strategies	35
Deleting an ISO 286 Strategy Based Condition	35
Adding an ISO 286 Strategy Condition in the ISO 286 tab	35
Strategy applied when both Tolerance Based Conditions & ISO 286 Strategy Conditions are defined	36
Flowchart: Assigning ISO 286 Strategy Conditions	38
5. Tutorial 2: Machining a Part Based on ISO 286 Strategy Conditions	39
Step 1: Open the tutorial part model	39
Step 2: Execute the Extract Machinable Features Command	40
Step 3: Open the ISO 286 tab	41
Step 4: Assign ISO 286 Strategy Conditions	42

Step 5: Execute the Run Tolerance Based Machining Command	44
Selecting the Options in Tolerance Based Machining –Run dialog box.....	44
Interpreting the information in the ISO 286 tab of TBM-Run dialog box.....	45
Table of Features & Default Strategies.....	45
ISO 286 Strategy Conditions Grid	46
Executing the Run-TBM Command	47
6. Recognizing Multisurface Features Based on Surface Finish	48
Recognizing Multisurface features using SOLIDWORKS CAM TBM	48
Multisurface Features tab of Tolerance Based Machining – Settings dialog box.....	48
Parameters in the Multisurface Features tab	49
Surface Finish	49
Strategy	49
Color	50
Edit Surface Finish 	50
Edit Color button 	50
Thumbnail view of the selected color.....	50
Range dialog box to add Surface Finish ranges in Multisurface Features tab	51
Example illustrating how to add Surface Finish Ranges using Range Dialog box.....	51
Adding/Editing/Deleting the entries in the Multisurface Features tab	52
Editing the entries in the Multisurface tab	52
Steps to change the color associated with the Strategy condition for a Multisurface Feature	52
Reassigning a Strategy	53
Deleting entries in the Multisurface Features tab.....	53
Adding new entries in the Multisurface Features tab	53
Flowchart: Recognizing Multisurface Features using TBM	54
7. Tutorial 3: Recognizing Multisurface Features Based on Surface Finish.....	55
Step 1: Open the tutorial part model	55
Step 2: Assign Settings in Multisurface Features dialog box	56
Step 3: Execute the Run Tolerance Based Machining Command	60
Legal Notices	63

1. INTRODUCTION TO ‘SOLIDWORKS CAM - TOLERANCE BASED MACHINING’

What is SOLIDWORKS CAM Tolerance Based Machining (TBM)?

For CNC Programmers, machining 3D models with close tolerances is a tricky proposition as it requires them to machine to the mean of the tolerance. It is very time-consuming to generate a CNC Program for such parts.

SOLIDWORKS CAM Tolerance Based Machining (TBM) is a newly introduced plug-in provided with the *SOLIDWORKS CAM* application. It leverages SOLIDWORKS dimensions, tolerance ranges and surface finish annotations to select the correct machining strategies for operations and machine them to the mean of asymmetric tolerances. It works for 2.5 Axis Mill Features as well as Multisurface Mill features on Mill parts.

SOLIDWORKS CAM TBM automatically adjusts asymmetric tolerances to mean tolerances for cutting tool strategies. This machine to mean capabilities eliminates long-standing issues surrounding differences between design practices required to tolerance parts based on fit, form and function versus manufacturing's need to machine geometry based on mean dimensions and tolerances. This significantly reduces CNC Programming time and enables you to engage in smart manufacturing.

Salient Features of SOLIDWORKS CAM TBM Plug-In

- Reads SOLIDWORKS DIMXpert dimensions
- Reads SOLIDWORKS Machining Based Definition (MBD) information
- Reads Surface Finish annotations
- Adjusts asymmetric tolerances to mean values
- Selects tools, feeds and speeds based on *DimXpert* and Machining Based Definition (MBD)

Pre-requisites for using SOLIDWORKS CAM TBM Plug-In

1. Embedded MBD/PMI Data on part to be machined

Ensure that the part model to be machined has **Machine Based Definition (MBD)** and/or **Product Manufacturing Information (PMI)** data. To do so, use the **SOLIDWORKS DimXpert** functionality (which comes with every seat of SOLIDWORKS) to easily insert dimensions and tolerances manually/automatically to create a critical dimension 3D model.

Annotations need to be added only for close dimensions, unique surface finishes and asymmetric dimensions. As SOLIDWORKS CAM employs knowledge-based machining, there is no need to add annotations for dimensions and surface finishes that fall within the standard tolerance block. Dimensions and surface finishes that fall into the standard tolerance block will use the standard machining strategies that are already saved in the Technology Database.


2. Run SOLIDWORKS CAM application on a supported CAD Platform

The **SOLIDWORKS CAM Tolerance Based Machining** plug-in is automatically installed when you install the SOLIDWORKS CAM application. In order to use this plug-in, the SOLIDWORKS CAM application must be run as an Add-In within the SOLIDWORKS application.

3. Adobe Reader XI

The **Adobe Reader** application is necessary to view, read, navigate, search and print the *User Guide* for **SOLIDWORKS CAM Tolerance Based Machining** plug-in.

Launching the SOLIDWORKS CAM TBM Plug-in

1. Launch the SOLIDWORKS CAM application as an Add-In within the SOLIDWORKS application.
2. Open the *Mill* part to be machined.
3. Ensure that this part model has *Machine Based Definition* (MBD) and/or *Product Manufacturing Information* (PMI) data. [Use the **SOLIDWORKS DimXpert** functionality for this purpose. Note that annotations need to be added only for close dimensions, unique surface finishes and asymmetric dimensions.]
4. On the SOLIDWORKS CAM Command Manager, click on the **Tolerance Based Machining** command button .
5. This command activates the SOLIDWORKS CAM TBM Command Manager. Click on this Command Manager to peruse commands associated with the SOLIDWORKS CAM TBM plug-in.

The SOLIDWORKS CAM TBM Plug-In is now ready for use.

Commands in SOLIDWORKS CAM TBM Command Manager

Run Tolerance Based Machining



Executing this command opens the **Tolerance Based Machining – Run** dialog box. Use the options provided in this dialog box to:

- Select the *Tolerance Based Machining* options that are to be executed when the OK button in this dialog box is clicked.
- Optionally select the processes (such as feature recognition, generation of operations, sorting of operations and generation of toolpaths) to be executed prior to generation of NC code for machining the part.
 - Depending on the unitary system selected (MMGS or IPS) in SOLIDWORKS, when the **Recognize Tolerance Range** checkbox option is selected in the **Run** tab of the **Tolerance Based Machining – Run** dialog box, the **Tolerance Range (mm)** or **Tolerance Range (inch)** tab will be displayed within the dialog box. How to interpret and use the information displayed in the *Tolerance Range (mm)/Tolerance Range (inch)* tab is explained with the help of an illustrative tutorial in [Chapter 3](#) of this *User Guide* under the section “[Interpreting the information in the Tolerance Range tab](#)”.
 - If the active unit system is MMGS and the **Recognize ISO 286 limits and fits** checkbox option is selected in the **Run** tab of the **Tolerance Based Machining – Run** dialog box, the **ISO 286** tab will be displayed within the dialog box. How to interpret and use the information displayed in this tab is explained with the help of an illustrative tutorial in [Chapter 5](#) of this *User Guide* under the section “[Interpreting the information in the ISO 286 tab of TBM-Run dialog box](#)”.

Tolerance Based Machining - Settings



Executing this command opens the **Tolerance Based Machining – Settings** dialog box.

- Use the **Tolerance Range** tabs in this dialog box to setup different tolerance ranges for different features and specify which machining strategy you want to use for each tolerance range. Once the tolerance ranges have been defined and the strategies have been assigned to each tolerance range, the system is ready to automatically select the correct strategies based on the [MBD or PMI data](#) and program the part.

[[Chapter 2](#) in this Guide explains how to use the [parameters within the Tolerance Range tabs](#) of this dialog box while [Chapter 3](#) provides an illustrative tutorial that explains how to machine a mill part using tolerance ranges and machining strategies defined in the Tolerance Range tab.]

- Use the **ISO 286** tab in this dialog box to machine *Hole* and **Circular Boss** features based on *ISO 286 Strategy Conditions* (where machining strategies are assigned based on Size Ranges and Tolerances values defined per the in **ISO 286** standard).
[[Chapter 4](#) in this Guide explains how to use the [parameters within the ISO 286 tab](#) of this dialog box while [Chapter 5](#) provides an illustrative tutorial that explains how to machine a mill part with hole features based on ISO 286 Strategy Conditions.]
- Use the Multisurface Features tab in this dialog box to recognize multisurface features and assign specific machining strategies to such multisurface features on the basis of their Surface Finish values.
[[Tutorial_3](#) in this Guide illustrates how to machine a part using tolerance ranges define din this tab.]

About SOLIDWORKS CAM TBM

Executing this command opens the *About SOLIDWORKS CAM - Tolerance Based Machining* dialog box. This dialog box gives information about the *SOLIDWORKS CAM TBM* plug-in.

2. ASSIGNING TOLERANCE RANGES & ASSOCIATED MACHINING STRATEGIES

If [MBD/PMI data](#) is embedded in a Mill part model to be machined, then using *SOLIDWORKS CAM TBM*, machining strategies can be assigned to individual tolerance ranges defined for the 2.5 Axis Mill features.

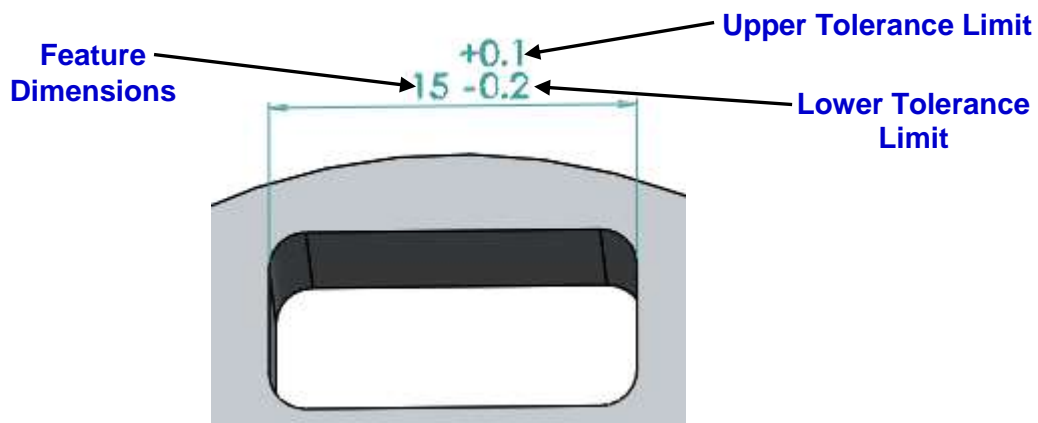
When the **Run Tolerance Based Machining** command is executed, the correct machining strategies will be assigned to the 2.5 Axis Mill Features that are recognized on the part. These machining strategies will be assigned based on the tolerance range match and mean tolerance value. Operations will be generated on the basis of these assigned machining strategies.

How is the Tolerance Range and Mean Tolerance value for a Feature computed?

The **tolerance range** for a feature is the difference between its upper tolerance limit and lower tolerance limit.

The **mean tolerance value** is the sum of the upper and lower tolerance limits of the feature divided by two.

These definitions are best understood via an illustrative example:




MBD data for a Rectangular Pocket Feature

For the above rectangular pocket feature, the dimension is 15 units, the upper tolerance limit is 0.1 units and the lower tolerance limit is -0.2 units.

The **tolerance range** is difference between upper and lower limits of the feature. In this case, it would be $[0.1 - (-0.2)]$ which is equal to 0.3 units.

The **mean tolerance value** is the sum of the upper and lower tolerance limits of the feature divided by 2. In this case, it would be $[(-0.2 + 0.1)/2]$ which is equal to -0.05 units.

Where are Tolerance Ranges and associated Machining strategies defined within SOLIDWORKS CAM TBM?

Tolerance ranges for the different 2.5 Axis Mill feature types are defined in the **Tolerance Based Machining – Settings** dialog box. This dialog box is displayed when the *Tolerance Based Machining – Settings* button  on the *SOLIDWORKS CAM TBM Command Manager* is clicked.

Two separate tabs are provided in this dialog box to define the tolerance ranges.

- Use the **Tolerance Range (mm)** tab to define the Tolerance based conditions in millimeters (MMGS unit system). This is the default tab displayed when you the dialog box is displayed.
- Use the **Tolerance Range (inch)** tab to define the Tolerance based conditions in inches (IPS unit system).

Parameters in the Tolerance Range tabs

Table of Features & Default Strategies

Feature

The *Feature* column lists all the types of 2.5 Axis Mill features for which Tolerance based conditions have been defined.

- If the tolerance based conditions have not been defined for a particular feature type, then the font color for that particular feature type's listing within this tab will be **magenta** in color.
- If the tolerance based conditions have been defined for a particular feature type, then the font color for that particular feature type's listing within this tab will be **black** in color.

Default Strategy

For every feature type listed in the table of the *Tolerance Range (mm)* and *Tolerance Range (inch)* tabs, the corresponding entry in the *Default Strategy* field indicates the default machining strategy assigned to the feature in the Technology Database.

On executing the *Run Tolerance Based Machining* command, the Default Strategy will be applied to a recognized 2.5 Axis feature in the following cases:

- If [tolerance based conditions](#) have not been defined at all for the particular 2.5 Axis Mill Feature type. (Such feature types can easily be identified in the *Tolerance Range* tab by the **magenta** font color for that particular feature type's listing.)
- Feature types for which the mean tolerance computed doesn't fall within any of the tolerance ranges defined for that feature within the *Tolerance Range* tab.

No. of Tol based conditions

For every feature type listed in the table within the *Tolerance Range* tab, the corresponding entry in the **No. of Tol based conditions** column indicates the number of tolerance ranges defined for that particular feature type.

If no tolerance based conditions have been defined for a particular feature type (i.e. the *No. of Tol based conditions* field indicates the value of '0'), then the font color for that particular feature type's listing within this tab will be **magenta**. If defined, then the font will be **black** in color.

When a particular feature type is selected in the grid, the tolerance based conditions (indicating the tolerance range and associated machining strategies) for this selected feature will be displayed in the *Tolerance based conditions* grid at the bottom of this tab.

SOLIDWORKS CAM – Tolerance Based Machining – Settings

Tolerance Range (mm) | Tolerance Range (inch) | ISO 286 | Multisurface Features

Feature	Default Strategy	Tol based conditions
Hole	Drill	3
Countersink Hole	Drill	2
Counterbore Hole	Drill	2
Rectangular Pocket	Rough-Rough(Rest)- Finish	2
Circular Pocket	Rough-Rough(Rest)- Finish	2
Irregular Pocket	Rough-Rough(Rest)- Finish	2
Rectangular Slot	Rough-Rough(Rest)- Finish	0
Irregular Slot	Rough-Rough(Rest)- Finish	0
Rectangular Corner Slot	Rough-Rough(Rest)- Finish	2
Irregular Corner Slot	Rough-Rough(Rest)- Finish	2
Circular Boss	Finish	3
Irregular Boss	Finish	3
Rectangular Boss	Finish	3
Obround Pocket	Rough-Finish	2
Obround Boss	Finish	3

Tolerance Range	Undersize Stra...	Nominal Strategy	Oversize Strat...
0 to 0.05 mm	Bore	Bore	Bore
0.05 to 0.25 mm	Ream	Ream	Ream
0.25 to 0.5 mm	Drill	Drill	Drill

Selected Feature

Table of features and Default Strategies

Tolerance Based Conditions grid (displayed for selected feature)

About... OK Close Help

'Tolerance Range' tab of Tolerance Based Machining – Settings dialog box

Tolerance Based Conditions grid

When a desired feature type is selected in the grid of the *Tolerance Range* tab, the tolerance based conditions (indicating the tolerance ranges and associated machining strategies) for the selected feature will be displayed in the *Tolerance based conditions* grid at the bottom of this tab.

Tolerance Range

This column indicates the tolerance ranges defined for the [feature type](#) selected in the grid. For each range, the lower and upper limit of the tolerance range is displayed along with the units (mm or inches).

Undersize Strategy

This strategy will be applied when the [mean tolerance value](#) computed for the feature has a negative value.

Nominal Strategy

This strategy will be applied when the [mean tolerance value](#) computed for the feature is "0".

Oversize Strategy


This strategy will be applied when the [mean tolerance value](#) computed for the feature has a positive value.

Reassigning the Undersize, Nominal and Oversize Strategies

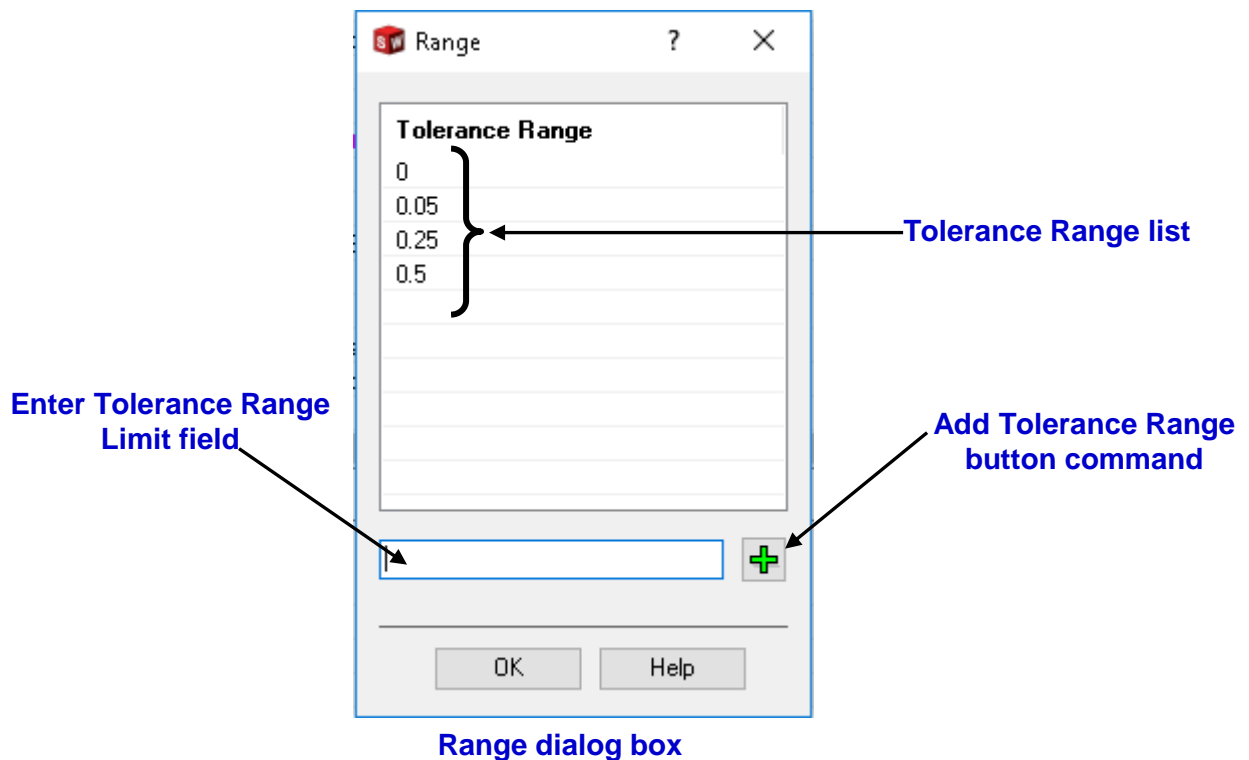
When you [define a new tolerance range](#) for a selected feature in the *Tolerance Range* tab, the [default strategy](#) defined for the feature in the Technology Database will be assigned as the *Undersize*, *Nominal* and *Oversize* strategies for that particular tolerance range.

You can change/edit the *Undersize*, *Nominal* or *Oversize* strategy for a tolerance range by simply clicking on that desired *Strategy* field. Clicking on the *Strategy* field displays a dropdown list of all machining strategies defined in the Technology Database for that particular feature type, including user-defined strategies. Select the desired strategy from this dropdown list to reassign the strategy for a tolerance range. The changes will be saved when you click the *OK* button of the dialog box.

Defining /Editing Tolerance Ranges

To define/edit the tolerance ranges for a selected 2.5 Axis Mill feature type, clicking on the **Edit Tolerance Range** button  at the bottom left corner of the *Tolerance Range* tab. Clicking on this button displays the **Range** dialog

box. Use this dialog box to define/edit the tolerance ranges. How to use this dialog box is explained in the next section via an illustrative example.




How to use the Range dialog box to define /edit Tolerance Ranges

How to use the **Range** dialog box to define/edit tolerance ranges for a feature is best understood through an illustrative example.


Consider that you want to define the following three tolerance ranges for a particular 2.5 Axis feature type:

	Lower Limit	Upper Limit
Range 1	0.0	0.02
Range 2	0.02	0.05
Range 3	0.05	0.1

The steps to define these tolerance range limits are as follows:

- Click on the **Edit Tolerance Range** button  at the bottom left corner of the **Tolerance Range** tab. The **Range** dialog box will be displayed.
- Input each unique tolerance range limit in the **Enter Tolerance Range Limit** field. (In this example, there are four unique tolerance range limits viz. 0, 0.02, 0.05 and 0.1.)

Note: Enter only the numerical value in this field. Do not enter the associated units as separate Tolerance Range tabs are provided for MMGS and IPS unitary systems.

- iii. After you input a tolerance range limit, click on the **Add Tolerance Range** button command .
- iv. The tolerance range limit you had input will be listed in the Tolerance Range list of this dialog box. Repeat [Steps i. and ii.](#) for all remaining unique tolerance range limits. The **Tolerance Range list** will automatically sort and display the tolerance range limits in ascending order of value.

Note: When adding multiple tolerance range limits in this dialog box, it is not necessary to add the limits in ascending or descending order of value. The **Tolerance Range list** will automatically sort and list the input values in ascending order from top to bottom row.

- v. In case you accidentally input an incorrect tolerance range limit, you can delete its entry from the **Tolerance Range list** by selecting the entry with a single mouse-click and pressing the **Delete** button.
- vi. After all tolerance range limits have been input, click on the **OK** button in the **Range** dialog box.

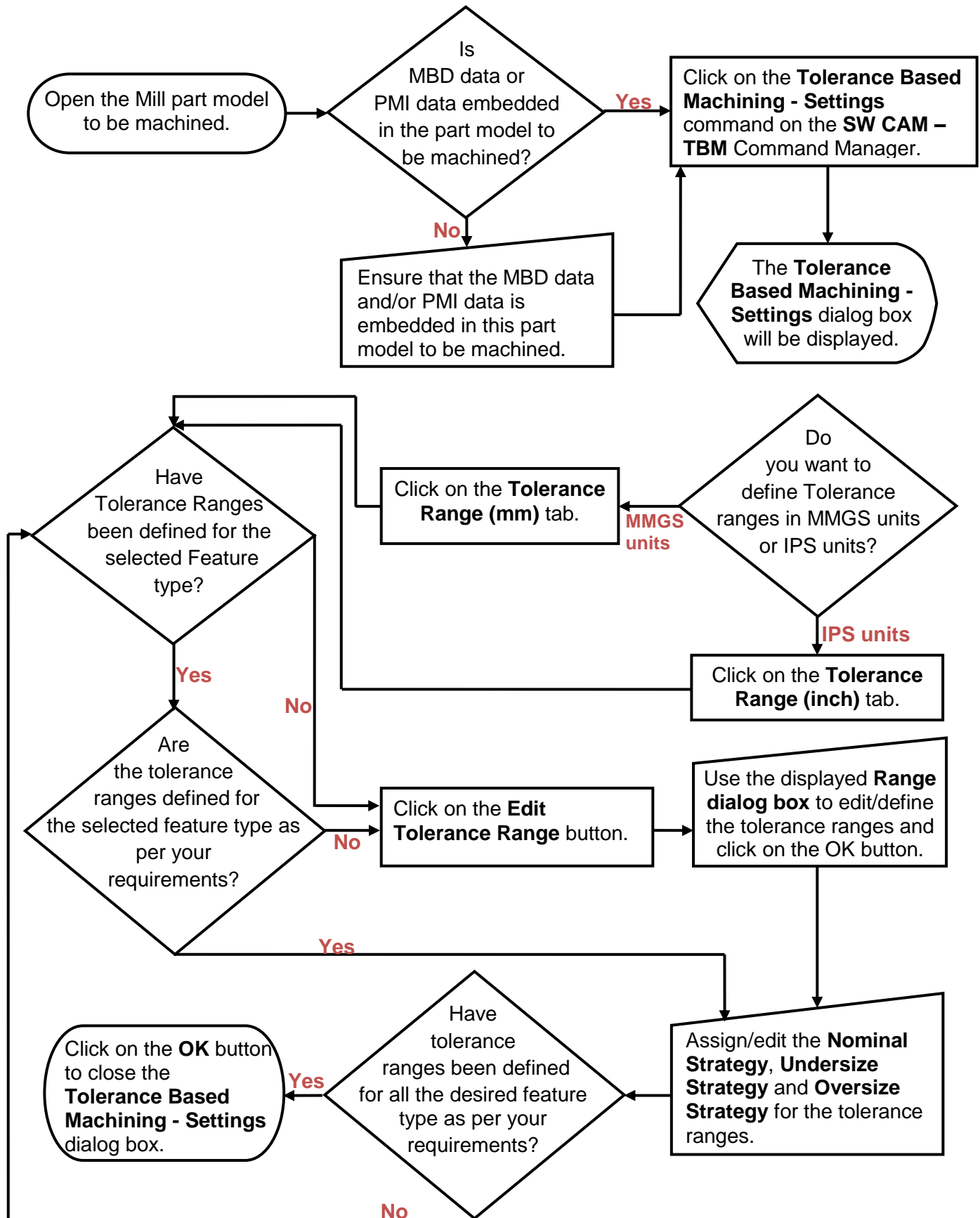
The *Range* dialog box will close and the new Tolerance Ranges defined will be listed in ascending order in the [Tolerance Based Conditions grid](#) of the *Tolerance Range* tab.

Special Case: Inputting only one single Tolerance limit

A lower and upper limit is required to define a tolerance range. If you press the OK button when only one limit is listed in the [Tolerance Range list](#), then the value of zero will be assigned as the lower limit and the Tolerance range limit entered will be assigned as the upper limit the subsequent Tolerance Range.

Given on the next page is a flowchart that summarizes the concepts explained in this chapter viz. how to assign Tolerance Ranges and associated Machining Strategies using the *Tolerance Based Machining – Settings* dialog box.

Flowchart: Assigning Tolerance Ranges & Strategies



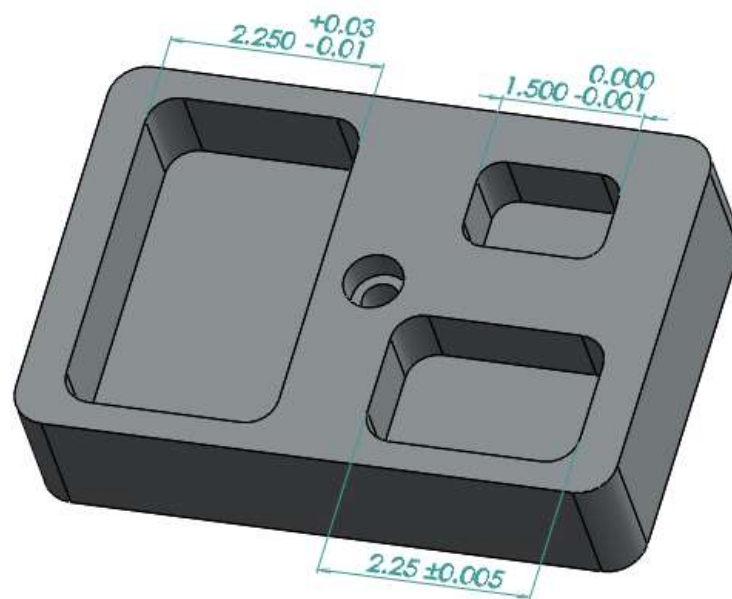
3. Tutorial 1: Machining a Solid Part based on Tolerance Ranges

This tutorial illustrates how to machine a mill part containing embedded MBD data using the SOLIDWORKS CAM TBM functionality to assign machining strategies based on tolerance ranges defined for the 2.5 Axis Mill features.

Before proceeding with this tutorial, ensure that you have understood the concepts explained in [Chapter 2](#) of this document.

Step 1: Open the tutorial part model

- i. Launch *SOLIDWORKS CAM* application as an Add-In within the *SOLIDWORKS* application.
- ii. Click on the *File* menu of *SOLIDWORKS* application and select the *Open* menu option.
- iii. The *File Open* dialog box will be displayed. Browse to the following folder location:
C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 201x\CAM Examples\TBM_SampleParts\Tutorials Parts
- iv. Select the ***TBM_Tutorial_1.SLDPRT*** file and click on the *Open* button.
- v. The selected tutorial solid model part will be displayed in the graphics area of the *SOLIDWORKS* application.





TBM_Tutorial_1.SLDPRT

- vi. Observe that this part model has three rectangular pockets and a Counterbore hole feature. Also observe that the dimensions and tolerance limits have been defined for each of the three rectangular pocket features.

Step 2: Defining Tolerance Ranges for the features

In this step, we will use the parameters within the *Tolerance Based Machining - Settings* dialog box to assign different tolerance ranges for the different feature types and then specify the machining strategies for each tolerance range defined.

- i. On the *SOLIDWORKS CAM Command Manager*, click on the **Tolerance Based Machining** command button .
- ii. The **SOLIDWORKS CAM TBM Command Manager** will be activated. Click on the **Tolerance Based Machining - Settings**  command on this *Command Manager*.
- iii. The **Tolerance Based Machining – Settings** dialog box will be displayed. In this tutorial, the dimensions and tolerance limits are expressed in inches (IPS units). Hence, click on the **Tolerance range (inch)** tab within this dialog box.
- iv. The solid part model under consideration in this tutorial has three rectangular pocket features. Hence, in the [table of Features and Default Strategies](#), click on the entry for **Rectangular Pocket** in order to select it.

Tolerance Range (inch) tab


Feature	Default Strategy	Tol based conditions
Hole	Drill	5
Countersink Hole	Drill	2
Counterbore Hole	Drill	2
Rectangular Pocket	Rough-Rough(Rest)- Finish	2
Circular Pocket	Rough-Rough(Rest)- Finish	1
Irregular Pocket	Rough-Rough(Rest)- Finish	2
Rectangular Slot	Rough-Rough(Rest)- Finish	0
Irregular Slot	Rough-Rough(Rest)- Finish	2
Rectangular Corner Slot	Rough-Rough(Rest)- Finish	2
Irregular Corner Slot	Rough-Rough(Rest)- Finish	0
Circular Boss	Finish	3
Irregular Boss	Finish	3
Rectangular Boss	Finish	3
Obround Pocket	Rough-Finish	2
Obround Boss	Finish	3

Tolerance Range	Undersize Stra...	Nominal Strategy	Oversize Strat...
0 to 0.002 in	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.002 to 1 in	Rough-Finish	Rough-Finish	Rough-Finish

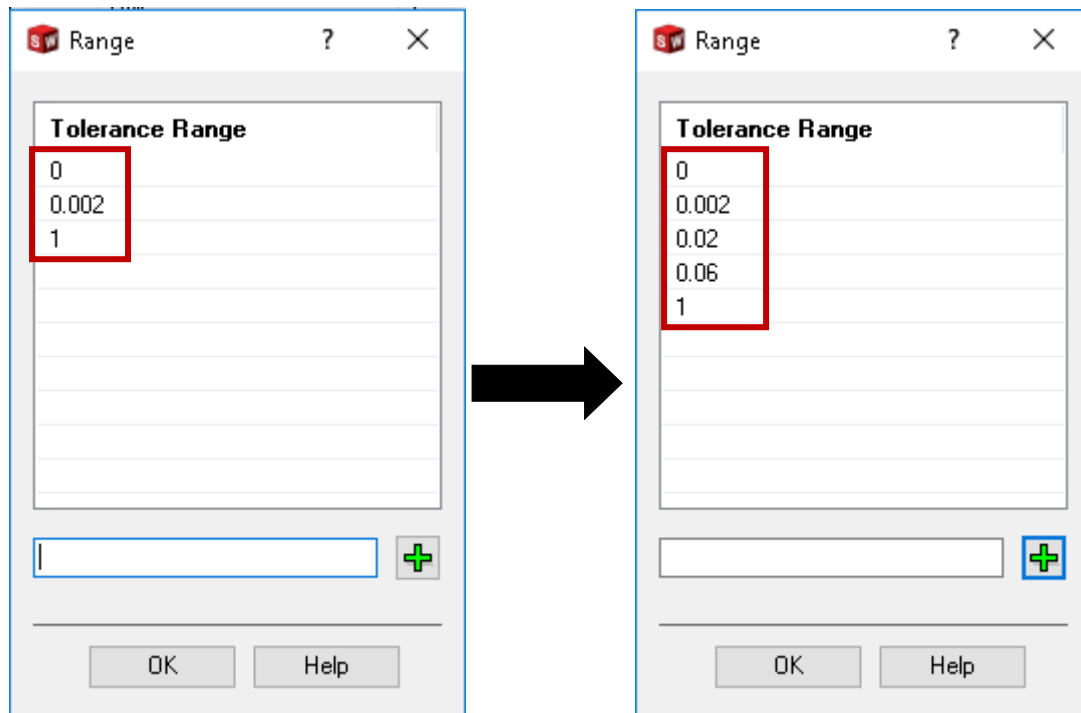
Tolerance Based Conditions corresponding to selected entry

- v. Observe the [Tolerance Based Conditions grid](#) for the *Rectangular Pocket* entry. Only two ranges have been defined. In this tutorial, the existing Tolerance Ranges will be edited and replaced with the following four tolerance ranges:


	Lower Limit	Upper Limit
Range 1	0.0	0.002
Range 2	0.002	0.02
Range 3	0.02	0.06
Range 4	0.06	1

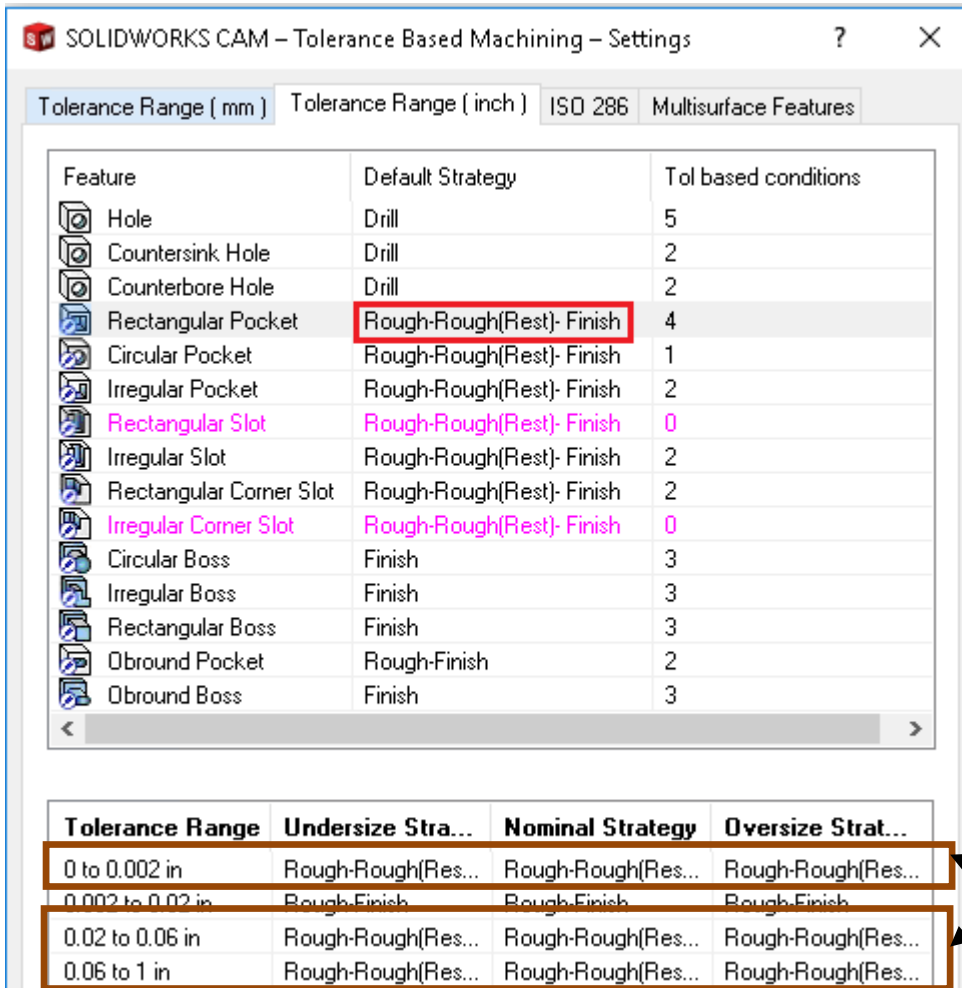
- vi. To edit the Tolerance Ranges, click on the **Edit Tolerance Range** button  at the bottom left corner of the *Tolerance Range* tab. The **Range** dialog box will be displayed. Three Tolerance limits (viz. **0.0**, **0.002** and **1**) are already displayed within this dialog box.
- vii. Based on the above Tolerance Range table, the new set of tolerance ranges to be input has 5 unique values viz. **0.0**, **0.002**, **0.02**, **0.06** and **1**. Hence, the only two tolerance limits missing in the *Range dialog box* are

0.02 and **0.06**. In the **Enter Tolerance Range Limit** field, input the value **0.02** and click on the **Add Tolerance Range** button command .



Adding Tolerance Limits to the Range dialog box

- viii. The value **0.02** will be added to the *Tolerance Range list* and will be automatically sorted and listed in ascending order within the *Range* dialog box. In the **Enter Tolerance Range Limit** field, input the value **0.06** and click on the **Add Tolerance Range** button command .
- ix. This value too will be automatically sorted and listed within the *Range* dialog box. Click on the **OK** button to apply the changes and close the *Range* dialog box.
- x. Observe the table of Features and Default Strategies. In the No. of Tol based Conditions field for the rectangular Pocket feature, the value has changed from **2** to **4** indicating that four tolerance ranges are now defined for the **Rectangular Pocket** feature.
- xi. Observe the *Tolerance Based Conditions grid*. Of the four tolerance ranges listed, three are new ranges. For all these new tolerance ranges that have been defined, the machining strategy assigned to the **Undersize Strategy**, **Nominal Strategy** and **Oversize Strategy** are the same as the default strategy for the *Rectangular Pocket* feature.



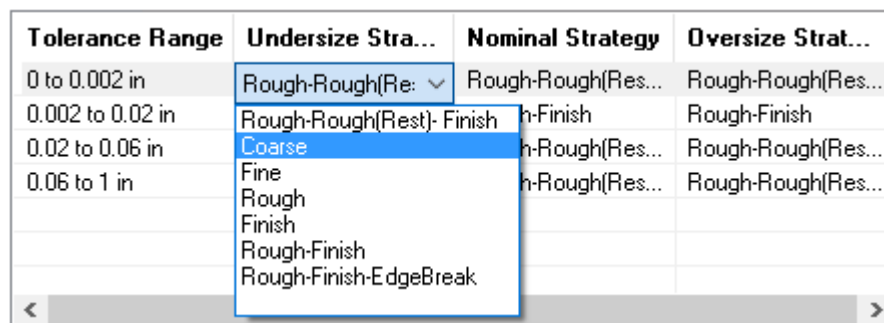
Strategies assigned are the same as the Default Strategy

Strategies assigned to the New Tolerance Ranges

Step 3: Reassigning Strategies for Tolerance Ranges

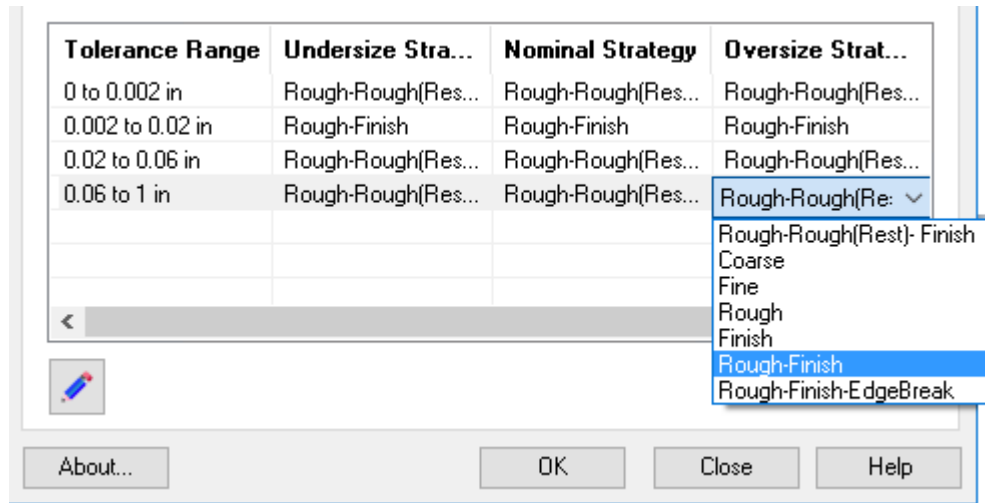
In this step, some of the machining strategies assigned to the Tolerance Ranges for the rectangular Pocket feature will be reassigned.

- In the *Tolerance Based Conditions* grid, highlight the first range viz. **0.0 to 0.002 in**.
- For this selected entry, click on the **Undersize Strategy** field and select **Coarse** from the dropdown list.



Reassigning the Undersize Strategy for a selected Tolerance Range

- iii. Next, highlight the last range viz. **0.6 to 1 in**. For this selected entry, click on the **Oversize Strategy** field and select **Rough-Finish** from the dropdown list.



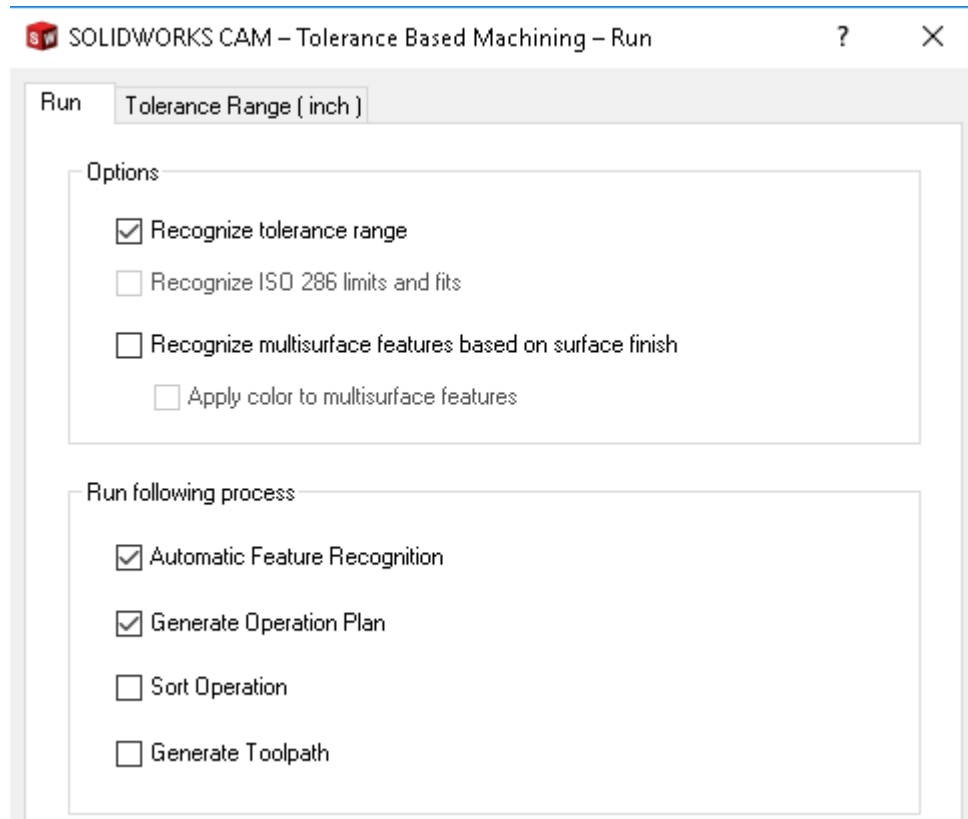
Reassigning the Oversize Strategy for a selected Tolerance Range

- iv. The desired machining strategies have now been assigned to all the Tolerance Ranges for the Rectangular Pocket feature. Click on the OK button to close the **Tolerance Based Machining – Settings** dialog box.


Step 4: Execute the ‘Run Tolerance Based Machining’ Command

- i. On the **SOLIDWORKS CAM TBM Command Manager**, click on the **Run Tolerance Based Machining** command.
- ii. Executing this command opens the **Tolerance Based Machining – Run** dialog box. The **Run tab** provides checkbox options to select the processes to be executed.
- iii. In this tutorial, machining strategies are to be assigned to 2.5 Axis Mill Features based on Tolerance Ranges. Hence, place a check in the checkbox labelled **Recognize tolerance range**.

Note: **Placing a check in the Recognize tolerance range checkbox activates the Tolerance Range tab. How to interpret the information in the Tolerance Range tab is explained in the next section: [Interpreting the information in the Tolerance Range tab](#).**



Run tab of Tolerance Based Machining – Run dialog box

- iv. The options to machine Hole and Circular Boss features based on ISO 286 Strategy Conditions is not applicable in when IPS units are used. Hence the **Recognize ISO 286 limits and fits** option is greyed out.
- v. The solid model part under consideration in this tutorial does not have any multisurface features. Hence the option **Recognize multisurface features based on surface finish** will not be checked.
- vi. The **Recognize tolerance range** option to assign machining strategies to 2.5 Axis Mill Features will work only if 2.5 Axis Mill Features for the solid part model under consideration have been recognized and listed in the *SOLIDWORKS CAM Feature tree*. If the case of this tutorial part, the features have not been recognized and listed in the *SOLIDWORKS CAM Feature tree*. Use any one of the following options to recognize the 2.5 Axis Mill features:
 - a. Place a check in the **Automatic Feature Recognition** checkbox within the *Run* tab. (This will ensure that 2.5 Axis Mill features are recognized first and then machining strategies are assigned based on the tolerance ranges defined for the recognized features.)
 - b. Switch to the *Tolerance Range* tab within this dialog box and click on the **Extract Machinable Features**  command button at the bottom left corner of the tab. (Executing this command will

immediately recognize all the 2.5 Axis Mill Features on the solid part and list them in the *SOLIDWORKS CAM Feature tree*.)

For the purposes of this tutorial, the first option will be exercised viz. the *Automatic Feature Recognition* checkbox within the *Run* tab will be checked.

- vii. In the *Run* tab, you can optionally choose to generate the operations based on the machining strategies that will be assigned to the features listed in the *SOLIDWORKS CAM Feature tree* by placing a check in the *Generate Operation Plan* checkbox.
- viii. Similarly, you can place checks in the **Sort Operation** and **Generate Toolpath** check boxes if required. In this tutorial, these checkboxes won't be checked.
- ix. Click on the **OK** button to close this dialog box and execute the command.


Note: Whenever the *Run Tolerance Based Machining* command is executed, the *Run* tab of the *Tolerance Based Machining – Run* dialog box will retain the settings for the checkboxes from the previous execution of the command.

- x. It will take a few seconds for the executed command to recognize the features and the matching tolerance ranges. Observe the *SOLIDWORKS CAM Feature tree*. All the 2.5 Axis Mill features on the solid part model have been successfully recognized and the correct machining strategies have been assigned.

Analyzing how the Machining Strategies have been assigned

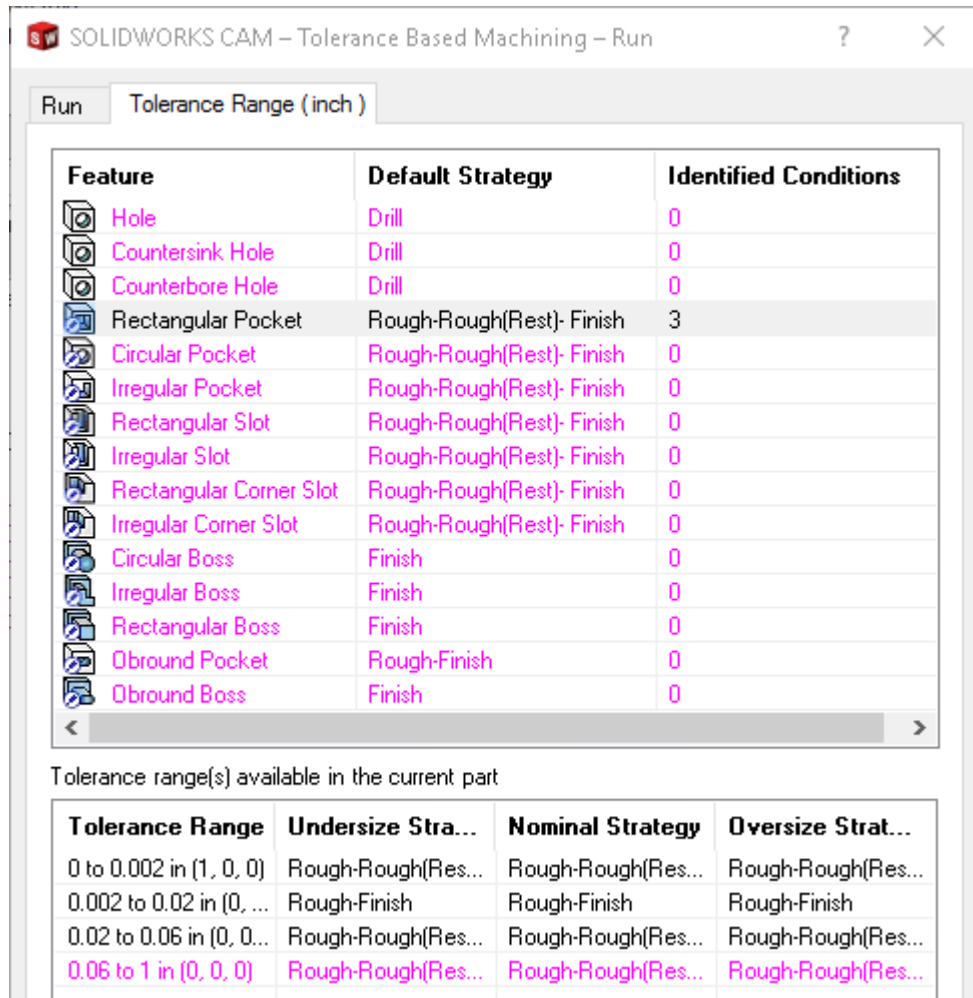
Let us now analyze the logic behind how the machining strategies were assigned to the four 2.5 Axis Mill Features recognized on the solid part model under consideration.

On the **SOLIDWORKS CAM TBM Command Manager**, once again click on the

Run Tolerance Based Machining  command to view the **Tolerance Based Machining – Run** dialog box. In the **Run** tab of this dialog box, observe that checkboxes that were checked during the previous execution of the command have been retained.

Interpreting the information in the Tolerance Range tab

Click on the **Tolerance Range (inch)** tab within this dialog box. This tab is displayed in the **Tolerance Based Machining – Run** dialog box only when the **Recognize tolerance range** checkbox in the **Run** tab is checked.



Tolerance Range tab in Tolerance Based Machining – Run dialog box

The contents of this tab is almost identical to the contents of the [Tolerance Range tab in the Tolerance Based Machining – Settings dialog box](#) but with a key difference. The **Tolerance Range** tab in the **Tolerance Based Machining-Run** dialog box provides additional information on the number of matches found between the **Tolerance Based Conditions** listed in this tab and the feature instances on the part being machined.

How to interpret the information in this tab is best understood via an illustrated example. Using this tutorial as an example, the explanation of the information displayed in this tab is given below.

Table of Features and Default strategies

Observe the table of listed features and default strategies. The last column in the table viz. **Identified Conditions** indicates whether matching tolerance ranges were found or not for the particular feature type on the part model.

- If one or more particular matching *Tolerance Based Conditions* have been identified for 2.5 Axis Mill feature recognized on the solid part, then the corresponding entry for that feature in this table will be in

black color font. The **Identified Conditions** field will indicate the number of matching Tolerance Based Conditions.

For this tutorial part, only the entry for the *Rectangular Pocket* feature will be displayed in black font color within this tab as three matching tolerance based conditions have been identified for the rectangular pocket features present on the solid part.

- For all 2.5 axis features that have not been recognized on the solid part, their respective entries will be displayed in **magenta** color.
- For 2.5 Axis features that have been recognized on the part and listed in the Feature tree,
 - If no matching tolerance based conditions are identified, then their respective entries will be displayed in **magenta** color
 - If tolerance based conditions have not been defined at all, then their respective entries will be displayed in **magenta** color
 - If MBD/PMI data has not been assigned for those particular feature types on the solid model part, then their respective entries will be displayed in **magenta** color.

In the solid model used in this tutorial, no MBD/PMI data was assigned to its Counterbore hole feature. Hence, entry for the *Counterbore hole* feature in the table is displayed in **magenta** color. The default strategy of **Drill** was assigned to this feature once the feature was recognized.

- The **Identified Conditions** field for all Feature entries displayed in **magenta** color will display the number '0' indicating that no matching tolerance based conditions were found.

'Tolerance range(s) available in the current part' grid

As observed in the previous section, the solid part in this tutorial has three rectangular pocket features for which machining strategies were assigned on the basis on matching tolerance based conditions. Highlight the **Rectangular Pocket** entry in the *table of features and default strategies*. Observe the corresponding **Tolerance range(s) available in the current part** at the bottom of the tab.

- If a tolerance based condition listed in this grid matches with one or more feature instances of the selected feature type on the solid part, then its entry will be displayed in **black** color font within the grid.
- If a tolerance based condition listed in this grid doesn't match with any feature instance of the selected feature type, then its entry will be displayed in **magenta** color font within the grid.

Tolerance range(s) available in the current part

Tolerance Range	Undersize Stra...	Nominal Strategy	Oversize Strat...
0 to 0.002 in (1, 0, 0)	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.002 to 0.02 in (0, ...	Rough-Finish	Rough-Finish	Rough-Finish
0.02 to 0.06 in (0, 0...	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.06 to 1 in (0, 0, 0)	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Finish

Tolerance range(s) available in the current part

Interpreting the Integers within parentheses adjacent to Tolerance Range field

Adjacent to the *Tolerance Range* value are three integer numbers separated by commas and contained within parentheses. For the selected feature type in the *Tolerance Range* tab, these numbers indicate the following:

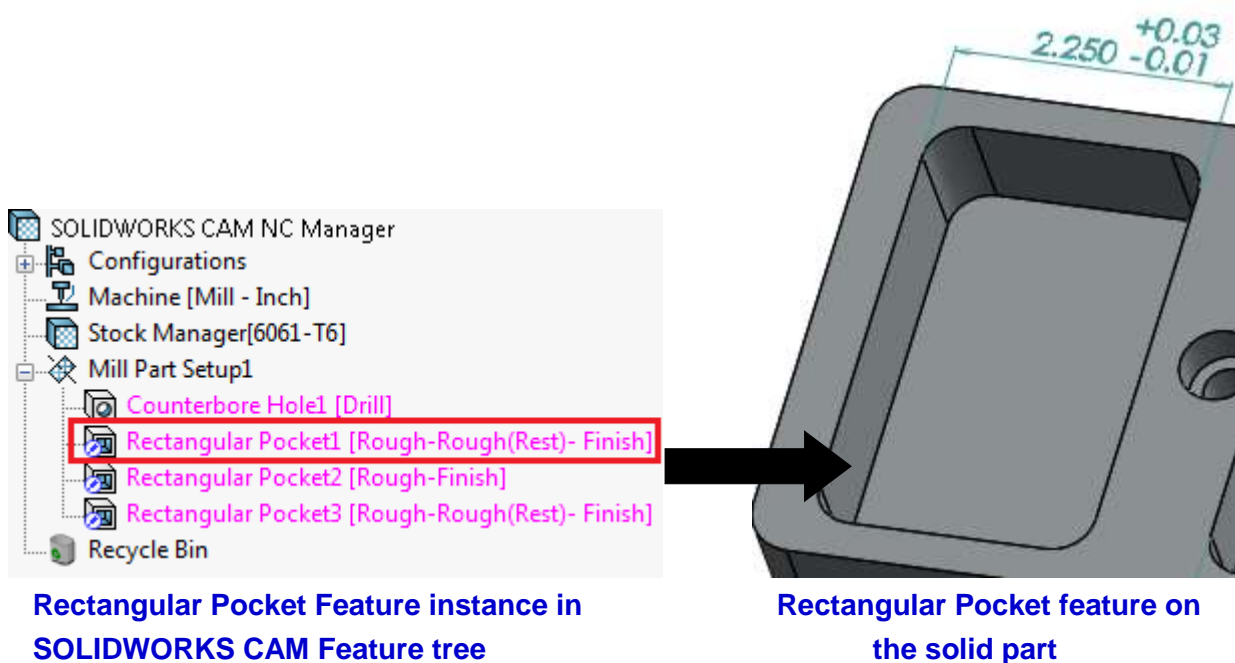
- The first number in the bracket indicates the number of feature instances present in the Feature tree which fulfills the Tolerance Based Condition such that the [Undersize Strategy](#) for machining will be applied.
- The second number in the bracket indicates the number of feature instances present in the Feature tree which fulfills the Tolerance Based Condition such that the [Nominal Strategy](#) for machining will be applied.
- The third number in the bracket indicates the number of feature instances present in the Feature tree which fulfills the Tolerance Based Condition such that the [Oversize Strategy](#) for machining will be applied.

For tolerance based conditions listed in **black** color font within this grid, at least one of the integers within the parentheses will have a non-zero value indicating a match with one or more feature instances.

Computing Mean Tolerance Values for the Features on the part

In this section, we will understand the mechanism behind how machining strategies were assigned to the feature instances on the part for which dimensions and tolerance limits were defined using MBD/PMI data.

Computing Tolerance Range and Mean Tolerance Value for the first Rectangular Pocket instance



For the above rectangular pocket feature, the dimension is 2.25 inches, the upper tolerance limit is 0.03 inches and the lower tolerance limit is -0.01 inches.

The **tolerance range** is difference between upper and lower limits of the feature. In this case, it would be $[0.03 - (-0.01)]$ which is equal to **0.04 inches**.

The **mean tolerance value** is the sum of the upper and lower tolerance limits of the feature divided by 2. In this case, it would be $[(-0.01 + 0.03)/2]$ which is equal to **0.01 inches**.

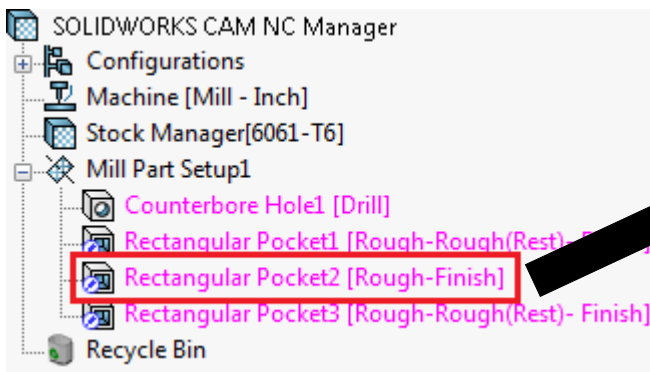
Based on this information, the tolerance range of 0.04 inches computed for this *Rectangular Pocket* feature instance falls within the tolerance range defined for the third *Tolerance Based Condition* entry. As the mean tolerance value is a positive value, the [Oversize Strategy](#) assigned to this tolerance based condition will be assigned to the Feature instance when the TBM command to assign machining strategies based on tolerance ranges is executed.

Tolerance range(s) available in the current part

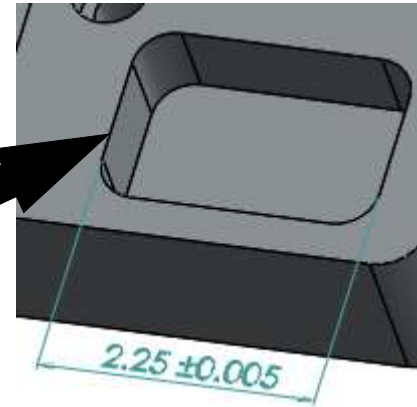
Tolerance Range	Undersize Stra...	Nominal Strategy	Oversize Strat...
0 to 0.002 in (1, 0, 0)	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.002 to 0.02 in (0, ...	Rough-Finish	Rough-Finish	Rough-Finish
0.02 to 0.06 in (0, 0...	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.06 to 1 in (0, 0, 0)	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Finish

Matching Tolerance Based Condition for first Rectangular Pocket Instance

Computing Tolerance Range and Mean Tolerance Value for the second Rectangular Pocket instance



Rectangular Pocket Feature instance in SOLIDWORKS CAM Feature tree



Rectangular Pocket feature on the solid part

For the above rectangular pocket feature, the dimension is 2.25 inches, the upper tolerance limit is 0.005 inches and the lower tolerance limit is -0.005 inches.

The **tolerance range** is difference between upper and lower limits of the feature. In this case, it would be $[0.005 - (-0.005)]$ which is equal to **0.01 inches**.

The **mean tolerance value** is the sum of the upper and lower tolerance limits of the feature divided by 2. In this case, it would be $[(-0.005 + 0.005)/2]$ which is equal to **0 inches**.

Based on this information, the tolerance range of **0.01 inches** computed for this Rectangular Pocket feature instance falls within the tolerance range defined for the second Tolerance Based Condition entry. As the mean tolerance value is zero, the [Nominal Strategy](#) assigned to this tolerance based condition will be assigned to the Feature instance when the TBM command to assign machining strategies based on tolerance ranges is executed.

Tolerance range(s) available in the current part

Tolerance Range	Undersize Stra...	Nominal Strategy	Oversize Strat...
0 to 0.002 in (1, 0, 0)	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.002 to 0.02 in (0, ..	Rough-Finish	Rough-Finish	Rough-Finish
0.02 to 0.06 in (0, 0...	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.06 to 1 in (0, 0, 0)	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Finish

Matching Tolerance Based Condition for second Rectangular Pocket Instance

Computing Tolerance Range and Mean Tolerance Value for the third Rectangular Pocket instance



For the above rectangular pocket feature, the dimension is 1.5 inches, the upper tolerance limit is 0 inches and the lower tolerance limit is -0.001 inches.

The **tolerance range** is difference between upper and lower limits of the feature. In this case, it would be $[0 - (-0.001)]$ which is equal to **0.001 inches**.

The **mean tolerance value** is the sum of the upper and lower tolerance limits of the feature divided by 2. In this case, it would be $[(-0.001 + 0)/2]$ which is equal to **-0.0005 inches**.

Based on this information, the tolerance range of 0.001 inches computed for this Rectangular Pocket feature instance falls within the tolerance range defined for the first *Tolerance Based Condition* entry. As the mean tolerance value is a negative value, the [Undersize Strategy](#) assigned to this tolerance based condition will be assigned to the Feature instance when the TBM command to assign machining strategies based on tolerance ranges is executed.


Tolerance range(s) available in the current part

Tolerance Range	Undersize Stra...	Nominal Strategy	Oversize Strat...
0 to 0.002 in (1, 0, 0)	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.002 to 0.02 in (0, ...	Rough-Finish	Rough-Finish	Rough-Finish
0.02 to 0.06 in (0, 0...	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Rough(Res...
0.06 to 1 in (0, 0, 0)	Rough-Rough(Res...	Rough-Rough(Res...	Rough-Finish

Matching Tolerance Based Condition for third Rectangular Pocket Instance

Reassigning the Machining Strategies

After the execution of the **Run Tolerance Based Machining** command, if you want to change the machining strategies assigned to the feature instances based on their tolerance ranges, you can do so by changing the strategies in the *Tolerance Range* tab of the **Tolerance Based Machining – Run** dialog box.

Execute the **Run Tolerance Based Machining**  command on the *SOLIDWORKS CAM TBM Command Manager* to view the **Tolerance Based Machining – Run** dialog box. Click on the *Tolerance Range* tab and select the desired *Feature type*. The corresponding tolerance based conditions will be displayed in the *Tolerance range(s) available in the current part grid* at the bottom of the tab.

To change an assigned strategy, click on the corresponding *Undersize Strategy*, *Nominal Strategy* or *Oversize Strategy* field and assign the desired machining strategy from the dropdown list. Switch to the *Run* tab of the dialog box and click on the *OK* button. The machining strategies for feature instances with matching tolerance based conditions will be reassigned based on the new settings.

4. ASSIGNING MACHINING STRATEGIES BASED ON ISO 286

What is ISO 286?


ISO 286 is an internationally used standard that provides standard tolerance classes and limit deviations for Holes and Circular boss features based on their nominal dimensions. It helps the manufacturer to machine the parts with specified limit deviations as given by the engineer(s).

How does ISO 286 fit with SOLIDWORKS CAM TBM?

SOLIDWORKS CAM TBM provides a functionality wherein you can choose to machine *Hole* and *Circular Boss* features present on a 3D part model based on *ISO 286 Strategy Conditions* (where machining strategies are assigned based on Size Ranges and Tolerances values as defined in ISO 286).


It is provided in the form of the **ISO 286** tab in the **Tolerance Based Machining – Settings** dialog box.

In order to machine *Hole* and *Circular Boss* features using this functionality, ensure that the following conditions are fulfilled:

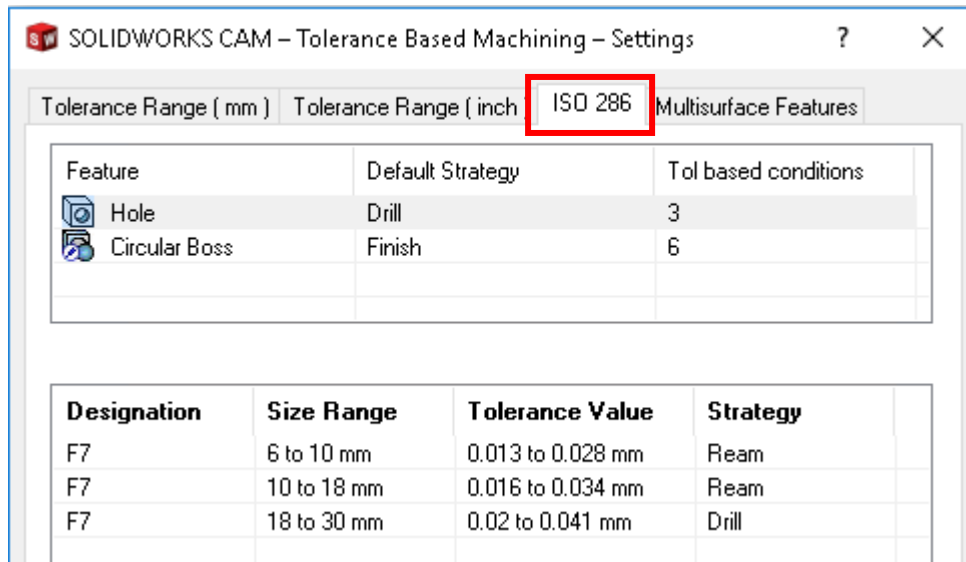
- i. [MBD/PMI data](#) is assigned for the *Hole* and *Circular Boss* features present on the part model to be machined.
- ii. The Unit system selected in SOLIDWORKS is a **Metric** based one (preferably **MMGS** units). [This is necessary as ISO 286 Standard is not supported for IPS unit system.]
- iii. The necessary ISO 286 Strategy Conditions are defined in the **ISO 286** tab of the **Tolerance Based Machining – Settings** dialog box for *Hole* and/or *Circular Boss* features.
- iv. When you execute the [Run Tolerance Based Machining command](#) , the **Recognize ISO 286 limits and fits** option in the **Run** tab of **Tolerance Based Machining – Run** dialog box is checked.

An illustrative tutorial explaining how to machine holes using the ISO 286 standard is provided in the next chapter.

ISO 286 tab of Tolerance Based Machining – Settings dialog box

When you click on the *Tolerance Based Machining – Settings* command button  on the SOLIDWORKS CAM TBM Command Manager, the **Tolerance Based Machining – Settings** dialog box is displayed. Use the **ISO 286** tab of this

dialog box to define *ISO 286 Strategy Conditions* for *Hole* and *Circular Boss* features.



ISO 286 tab of Tolerance Based Machining - Settings dialog box

Table of Features and Default Strategies in the ISO 286 tab

Feature

The *Feature* column lists the two feature types covered by *ISO 286* viz. *Hole* and *Circular Boss* features.

- If *ISO 286 Strategy Conditions* have not been defined for a particular feature type, then the font color for that particular feature type's entry within this tab will be **magenta** in color.
- If *ISO 286 Strategy Conditions* have been defined for a particular feature type, then the font color for that particular feature type's listing within this tab will be **black** in color.

Default Strategy

For every feature type listed in the *ISO 286* tab, the corresponding entry in the *Default Strategy* field indicates the default machining strategy assigned for that feature in the Technology Database.

On executing the [Run Tolerance Based Machining command](#), the *Default Strategy* will be applied to the specific instances of *Hole* and *Circular Boss* features present on the solid part in the following cases:

- *ISO 286 Strategy Conditions* have not been defined at all in the *ISO 286* tab for the *Hole* or *Circular Boss* feature. (Such a feature can easily be identified by the **magenta** font color of its entry within the *ISO 286* tab.)
- The feature instances do not fulfill any of the *ISO 286 Strategy Conditions* defined in the *ISO 286* tab.

No. of ISO 286 Strategy Conditions

For the *Hole* and **Circular Boss** feature types listed in the *ISO 286* tab, the corresponding entry in the **No. of ISO 286 Strategy Conditions** field indicates the number of *ISO 286 Strategy Conditions* defined for that particular feature type.

ISO Strategy Conditions grid

When the *Hole* or *Circular Boss* feature is selected in the *ISO 286* tab, the corresponding **ISO 286 Strategy Conditions** will be displayed below the table. Use this grid to view/edit the machining strategies for the selected feature type.

For the selected feature, this grid lists all those *ISO 286 Designations*, *Size Ranges* and corresponding *Tolerance values* for which machining strategies have been assigned.

The number of entries (rows) in this grid is equal to the number indicated in the [No. of ISO 286 Strategy Based Conditions](#) field for the selected feature.

Designation

This field indicates the designation for *ISO 286 Tolerance limits*. Within this grid, this is a read-only value.

- *ISO Hole Tolerance limits* are designated with upper case letters.
- *ISO Shaft (Circular Boss) Tolerance limits* are designated with lower case letters.

Size Range

This field indicates the *Size Range* (in millimeters) within which the nominal dimensions of the feature instance being machined must fall in order for the *ISO 286 Strategy Condition* to be applied. Within this grid, this is a read-only value.

Note: Using the **SOLIDWORKS CAM TBM plug-in**, you can define **ISO 286 Strategy Conditions** only for those **Hole** and **Circular Boss** features whose dimensions fall within the size range of 3 mm to 400 mm.

Tolerance Values

This field indicates the *Tolerance Range* (in millimeters) associated with the defined *Size Range* as per the *ISO 286* standard. Within this grid, this is a read-only value.


Strategy

This field indicates the machining strategy that will be applied to the feature being machined if it fulfills the *ISO 286 Strategy Condition*. You can also assign user-defined strategies from this dropdown list provided they

have been added to the Technology Database for the selected feature type.

When you click on the specific *Strategy* field, a dropdown list of all machining strategies defined in the Technology Database for that particular feature type will be displayed. You can reassign the *Strategy* by selecting the desired from this dropdown list. The changes will be saved when you click the *OK* button of the *Tolerance Based Machining - Settings* dialog box.

Add ISO 286 Strategy Conditions button

Clicking on the **Add ISO 286 Strategy Conditions** button  at the bottom left corner of the tab opens the **Add ISO 286 Strategy Conditions** dialog box. Use this dialog box to [add ISO 286 based strategy conditions](#).

Steps to add/edit/delete ISO 286 based Strategy Conditions

Editing an ISO Strategy Based Condition/ Reassigning Machining Strategies

For an ISO 286 Strategy Condition listed in the *ISO 286 Strategy Condition* grid, only the machining strategy assigned in the *Strategy* field can be edited as all the other parameters are read-only values.

When you click on the specific *Strategy* field, a dropdown list of all machining strategies defined in the Technology Database for that particular feature type will be displayed. You can reassign the *Strategy* by selecting the desired from this dropdown list. The changes will be saved when you click the *OK* button of the *Tolerance Based Machining - Settings* dialog box.

Deleting an ISO 286 Strategy Based Condition


In the [ISO Strategy Conditions grid](#), delete any unwanted entries. To delete an *ISO 286 Strategy Condition* listed in this grid, select the entry using a left mouse click and press the **Delete** button on the keyboard.

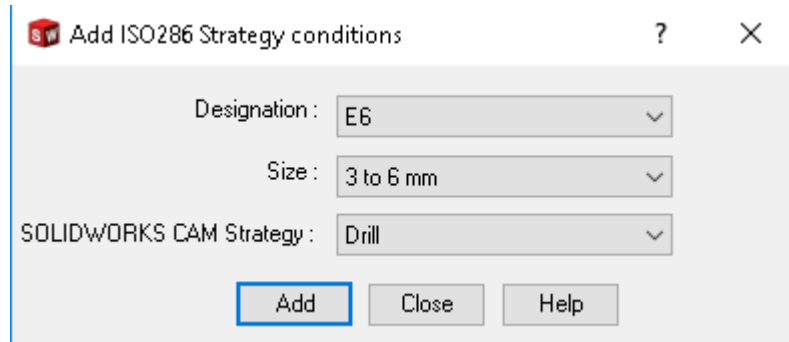
Adding an ISO 286 Strategy Condition in the ISO 286 tab

Following are the steps to add an *ISO 286 Strategy Condition* to the [ISO 286 Strategy Conditions grid](#) in the [ISO 286 tab](#) for a *Hole* or *Circular Boss* feature:

1. In the [ISO 286 tab](#) of the *Tolerance Based Machining – Settings* dialog box, select the feature (Hole or Circular Boss) in the [table of Features](#)

& [Default Strategies](#) for which the *ISO 286 Strategy Condition* is to be added.

2. Click on the *Add ISO 286 Strategy Conditions* button command  at the bottom left corner of the tab.
3. The **Add ISO 286 Strategy Conditions dialog box** will be displayed.



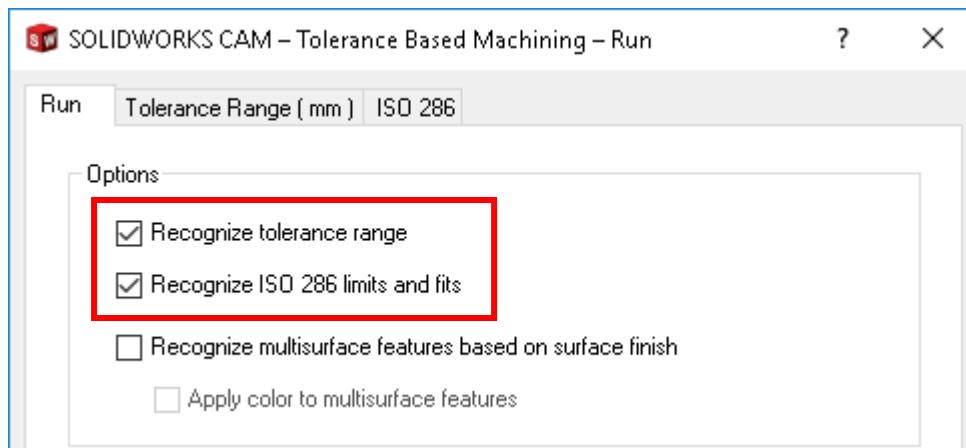
Add ISO 286 Strategy Conditions dialog box

4. Using the parameters in this dialog box, select settings for the new strategy condition as follows.
 - i. Select the **Designation**, **Size** and **Strategy** using the respective dropdown list to define an ISO 286 Strategy Condition.
 - ii. Click on the **Add** button of this dialog box to add the strategy condition to the [ISO 286 Strategy Conditions grid](#).
 - iii. After addition of the new strategy condition, the *Add ISO 286 Strategy Conditions* dialog box will still remain open, thus allowing you to add entries for new ISO 286 Strategy Conditions without the need to reopen the dialog box multiple times.
 - iv. Once you have added all the desired ISO Strategy Conditions, click on the **Close** button to exit this dialog box.
 - v. The [ISO 286 Strategy Conditions grid](#) will be populated with the customized Strategy conditions that you have defined.
 - vi. Click on the OK button to apply these changes/additions and close the *Tolerance Based Machining - Settings* dialog box.

Strategy applied when both Tolerance Based Conditions & ISO 286 Strategy Conditions are defined

For Hole and/or Circular Boss features, if Tolerance based conditions are defined in the **Tolerance Range (mm) tab** and *ISO 286 Strategy Conditions* are defined in the **ISO 286 tab** of **Tolerance Based Machining – Settings dialog box**, then the machining strategy that would be applied to the hole/circular boss feature instances listed in the SOLIDWORKSCAM Feature tree on executing

the [Run Tolerance Based Machining command](#) will depend on the settings in the **Run tab of Tolerance Based Machining – Run dialog box**.

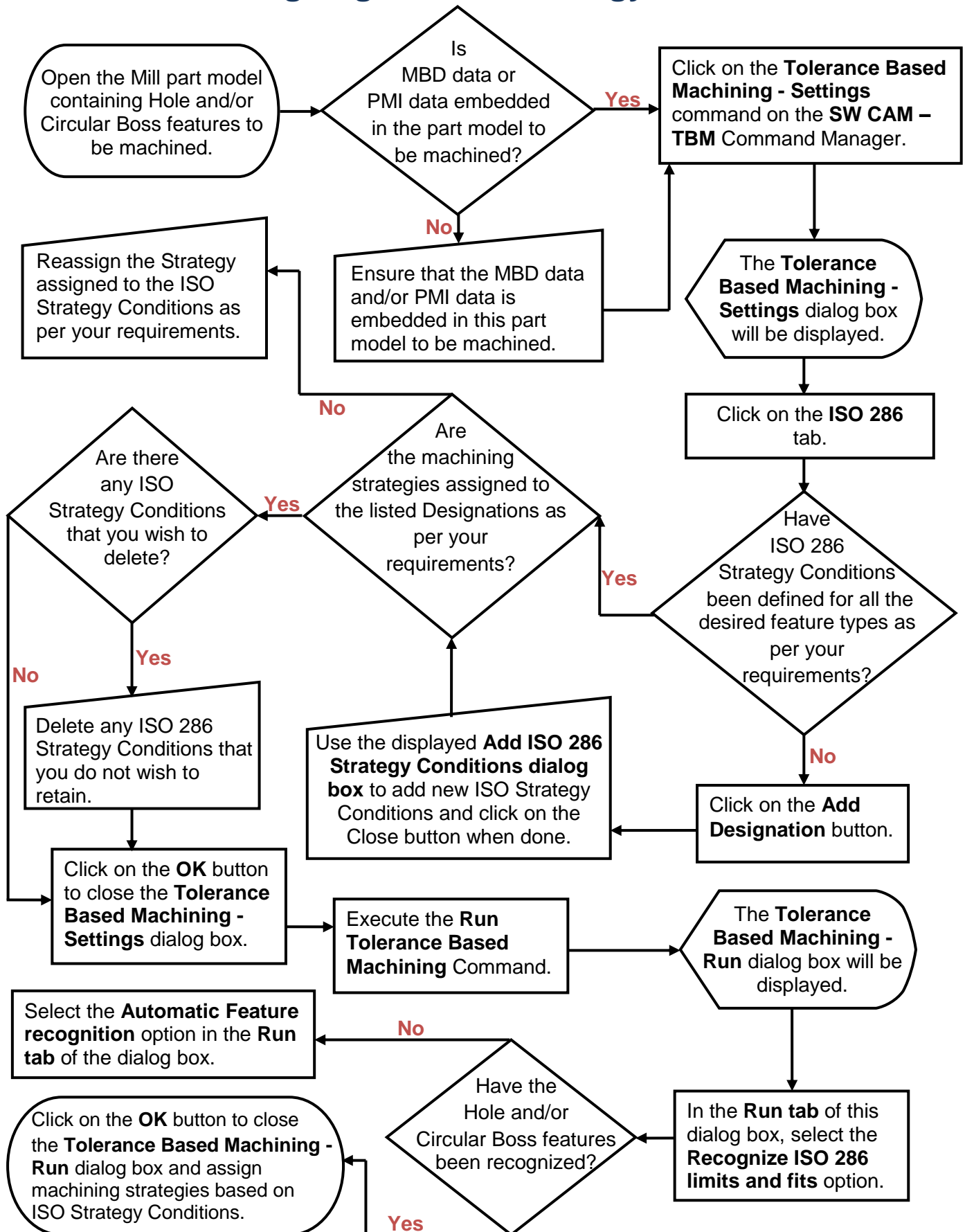


Options within the Run tab that affect the Machining Strategies assigned to Hole and Circular Boss Features

- If the *Recognize tolerance range* option is checked but *Recognize ISO limits and fits* option is not checked, then the instances of the *Hole* and *Circular Boss* features will be machined as per the Tolerance based conditions defined for these features in the *Tolerance Range (mm)* tab.
- If the *Recognize ISO limits and fits* option is checked but the *Recognize tolerance range* option is not checked, then instances of the *Hole* and *Circular Boss* features will be machined as per the *ISO 286 Strategy Conditions* defined for these features in the *ISO 286 tab*. If the size of the feature instance to be machined doesn't match the limits and fits defined for any one of the ISO Strategy Conditions in the *ISO 286 tab*, then the [default machining strategy](#) (as defined in the Technology Database) will be applied.
- If both the *Recognize tolerance range* and *Recognize ISO limits and fits* options are checked, then instances of the *Hole* and *Circular Boss* features will be machined as per the *ISO 286 Strategy Conditions* defined for these features in the *ISO 286 tab*. However, if the size of the feature instance to be machined doesn't match the limits and fits defined for any one of the ISO Strategy Conditions in the *ISO 286 tab*, then the feature will be machined as per the *Tolerance based conditions* defined in the *Tolerance Range (mm)* tab for that feature type. If a matching Tolerance Based Condition is not found, then the [default machining strategy](#) (as defined in the Technology Database) will be applied to the feature instance.

Given on the next page is a flowchart that summarizes the concepts explained in this chapter viz. how to assign ISO 286 Strategy Conditions in the *ISO 286 tab* of the *Tolerance Based Machining – Settings* dialog box.

Flowchart: Assigning ISO 286 Strategy Conditions



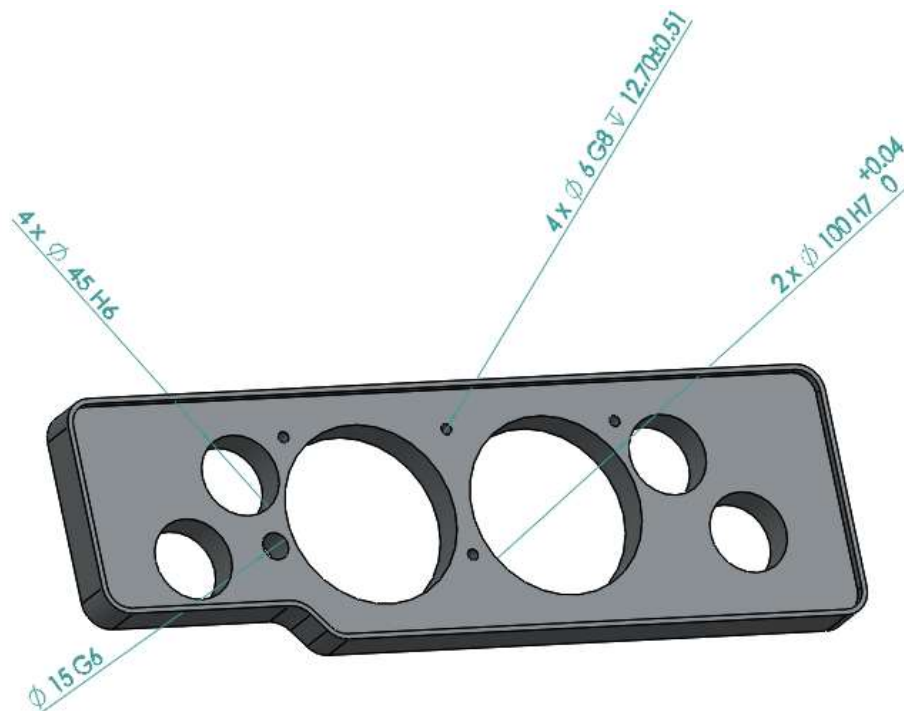
5. TUTORIAL 2: MACHINING A PART BASED ON ISO 286 STRATEGY CONDITIONS

This tutorial illustrates how to machine a mill part (which has multiple instances of the hole feature and contains embedded MBD/PMI data) using the *SOLIDWORKS CAM TBM* functionality to assign machining strategies based on *ISO 286 Strategy Conditions*.

Before proceeding with this tutorial, ensure that you have understood the concepts explained in [Chapter 4](#) of this document.


Step 1: Open the tutorial part model

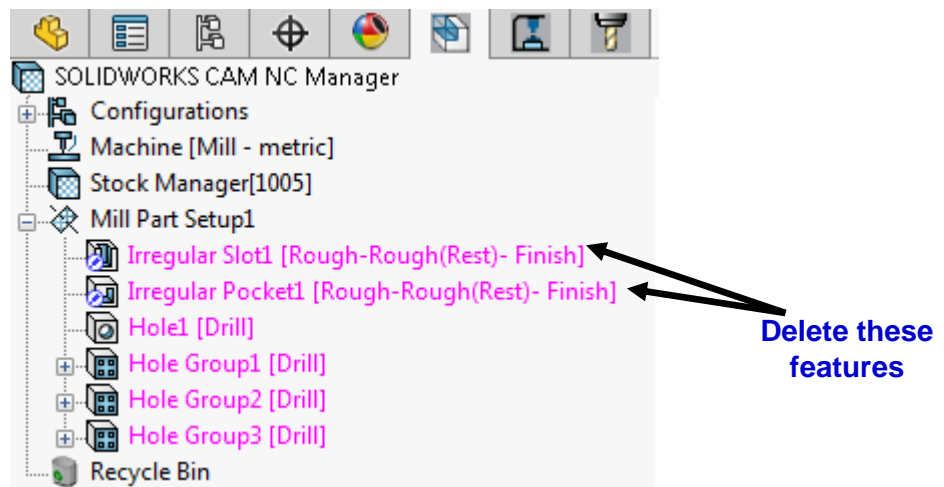
- i. Launch *SOLIDWORKS CAM* application as an Add-In within the *SOLIDWORKS* application.
- ii. Click on the *File* menu of *SOLIDWORKS* application and select the *Open* menu option.
- iii. The *File Open* dialog box will be displayed. Browse to the following folder location:
C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2018\CAM Examples\TBM_SampleParts\Tutorials Parts
- iv. Select the ***TBM_Tutorial_2.SLDPRT*** file and click on the *Open* button.
- v. The selected tutorial solid model part will be displayed in the graphics area of the *SOLIDWORKS* application.



TBM_Tutorial_2.SLDPRT

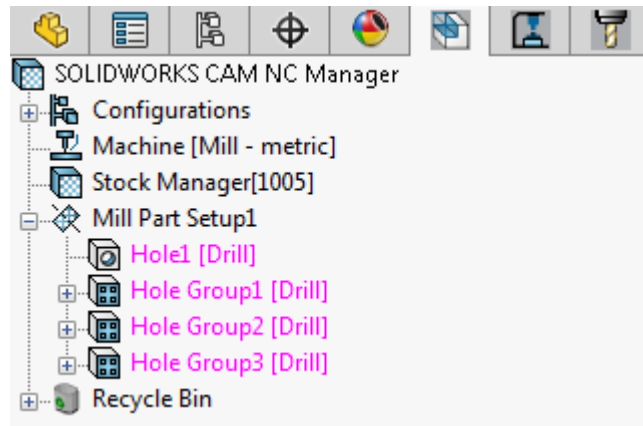
Step 2: Execute the Extract Machinable Features Command

- i. Ensure that the Unit System selected for SOLIDWORKS is **MMGS**. (The ISO 286 functionality in SOLIDWORKS CAM TBM plug-in is not available in IPS Unit System.)
- ii. Click on the **Extract Machinable Features** command  on the SOLIDWORKS CAM Command Manager.
- iii. All 2.5 Axis Mill Features present on the tutorial part will be recognized and listed in the **SOLIDWORKS CAM Feature tree**. Observe that an irregular slot, an irregular pocket, a hole and three *Hole* group features have been identified and listed in the **Feature tree**. The machining strategies assigned to these features are indicated in square brackets adjacent to the feature name. These strategies have been assigned on the basis of the default strategies assigned to these features in the *Technology Database*.



Features listed in the SOLIDWORKS CAM Feature tree

- iv. In this tutorial, only the hole features will be machined based on machining strategies assigned as per ISO 286 Strategy Conditions. Hence, delete the *Irregular Slot* and *Irregular Pocket* features from the SOLIDWORKS CAM Feature tree.



SOLIDWORKS CAM Feature after deletion of Irregular Slot and Irregular Pocket features

- v. The tutorial part has a multiple hole feature instances (1 hole feature and three hole groups). MBD/PMI data has been assigned to the *Hole1* feature as well as one instance of the hole feature in each of the hole group features. Given in the table on the next page is a table with the corresponding details.



Name of Feature	ISO 286 Designation	Diameter of the Hole Instance	Number of instances of the Hole Feature on the part
Hole1	G6	15 mm	1
Hole Group1	H6	45 mm	4
Hole Group2	H7	100 mm	2
Hole Group3	G8	6 mm	4

Details of the Hole Feature Instances present in the Tutorial part

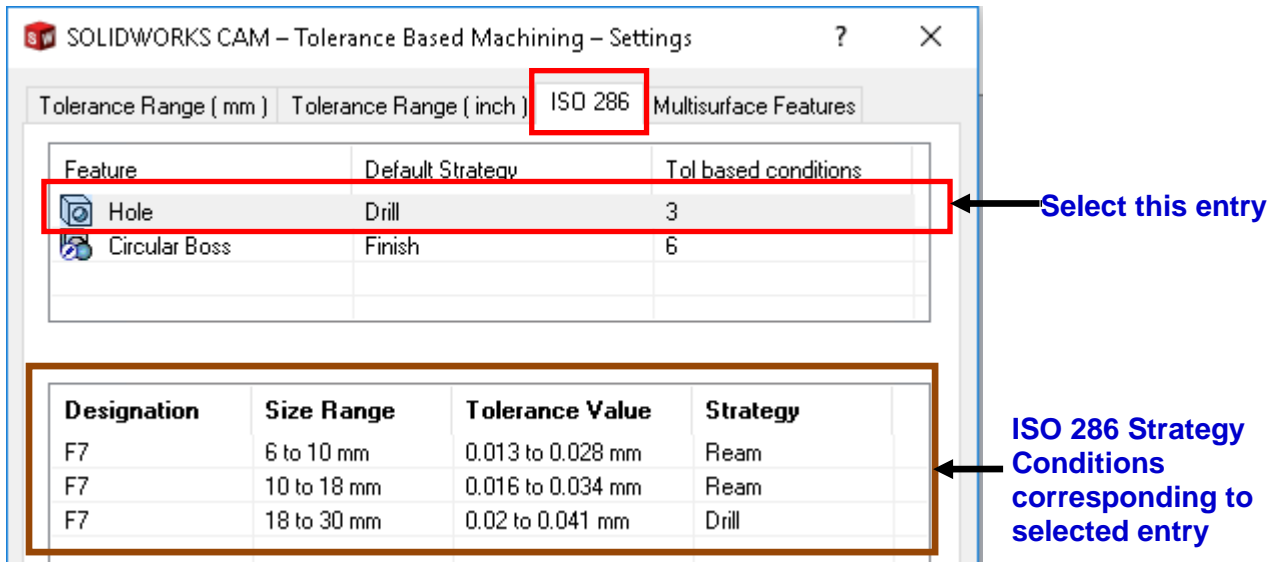
In this tutorial, using the SOLIDWORKS CAM TBM plug-in, the machining strategies assigned to all the hole feature instances will be reassigned based on *ISO 286 Strategy Conditions*.

Step 3: Open the ISO 286 tab

In this step, we will use the parameters within the **ISO 286 tab** of the *Tolerance Based Machining - Settings* dialog box to add/edit ISO 286 Strategy Conditions for Hole feature.

- i. On the *SOLIDWORKS CAM Command Manager*, click on the **Tolerance Based Machining** command button .
- ii. The **SOLIDWORKS CAM TBM Command Manager** will be activated. Click on the **Tolerance Based Machining - Settings**  command on this *Command Manager*.
- iii. The **Tolerance Based Machining – Settings** dialog box will be displayed. Click on the **ISO 286** tab.

- iv. The solid part model under consideration in this tutorial has multiple instances of hole features (and no circular boss features or any other 2.5 Axis Mill Features). Hence, in the [table of Features and Default Strategies](#), click on the entry for **Hole** in order to select it.
- v. The *ISO 286 Strategy Conditions* associated with the Hole feature will be displayed in the **ISO 286 Strategy Conditions** grid.



ISO 286 tab in Tolerance Based Machining – Settings Dialog box


Step 4: Assign ISO 286 Strategy Conditions

In this step, new ISO 286 Strategy Conditions will be added to the *ISO 286 Strategy Conditions grid* for the Hole Feature. Following are the steps:

- i. In the **ISO 286** tab of the **Tolerance Based Machining – Settings** dialog box, click on the on the entry for **Hole** in order to select it.
- ii. The *ISO 286 Strategy Conditions* associated with the Hole feature will be displayed in the **ISO 286 Strategy Conditions** grid.
- iii. Based on the MBD/PMI data defined for the *Hole* feature instances on the tutorial part, the *ISO 286 Designations* and machining strategies to be assigned to these hole features are as follows:

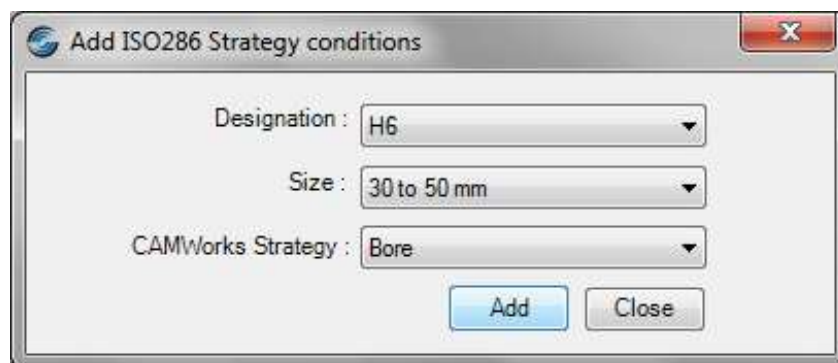
Name of Feature	ISO 286 Designation	Diameter of the Hole Instance	Strategy to be Assigned for Machining the Hole feature instance
Hole1	G6	15 mm	Drill
Hole Group1	H7	100 mm	Bore
Hole Group2	H6	45 mm	Bore
Hole Group3	G8	6 mm	Drill

Machining Strategies to assigned to Hole feature instances on the tutorial part

- iv. The **ISO 286 Strategy Conditions** grid does not list any of these *ISO 286 Designations*. Hence, these entries need to be added. To do so, following are the steps:
- Click on the **Add Designation** command button  at the bottom left corner of the tab.
 - The **Add ISO 286 Strategy Conditions** dialog box will be displayed. In the **Designation** dropdown list of this dialog box, select **G8**.
 - In the **Size** dropdown list, select **6 to 10 mm**. (The size range that is selected must be one within which the diameter of the Hole feature instance will fall into, resulting in a match.)
 - In the **SOLIDWORKS CAM Strategy** dropdown list, select **Drill**.
 - Click on the **Add** button within the dialog box. The entry will be added to the **ISO 286 Strategy Conditions** grid but the **Add ISO 286 Strategy Conditions** dialog box will remain open.
 - Select the following within this dialog box to add the next entry:

<i>Designation</i>	→ G6
<i>Size</i>	→ 10 to 18 mm
<i>SOLIDWORKS CAM Strategy</i>	→ Drill
 - Click on the **Add** button.
 - Select the following within this dialog box:

<i>Designation</i>	→ H6
<i>Size</i>	→ 30 to 50 mm
<i>SOLIDWORKS CAM Strategy</i>	→ Bore
 - Click on the **Add** button.

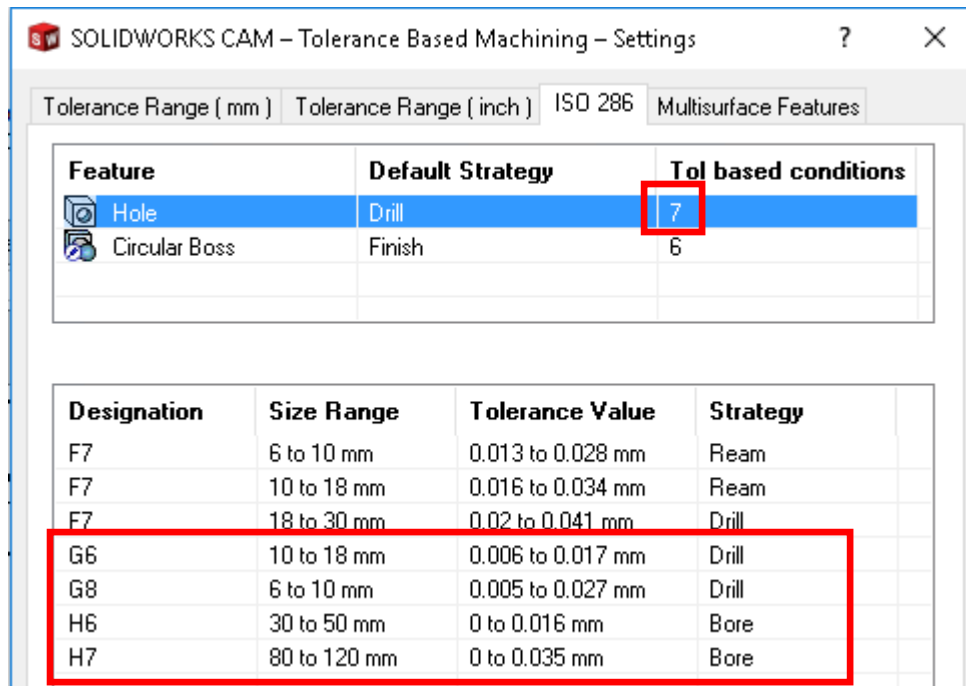


Add ISO 286 Strategy Conditions Dialog box

- Select the following within this dialog box to add the next entry:

<i>Designation</i>	→ H7
<i>Size</i>	→ 80 to 120 mm
<i>SOLIDWORKS CAM Strategy</i>	→ Bore
- Click on the **Add** button.
- All required **ISO 286 Designations** have now been added. Click on the **Close** button to close this dialog box.

- v. Observe the **ISO 286 Strategy Conditions** grid within the **ISO 286** tab. The new entries have been added. Observe that the **No. of Tol Based Conditions** field for the **Hole** entry too has been updated to reflect the increased number of **ISO 286 Strategy Conditions** defined for the Hole feature.



New designations added to the ISO 286 Strategy Conditions grid of ISO 286 tab

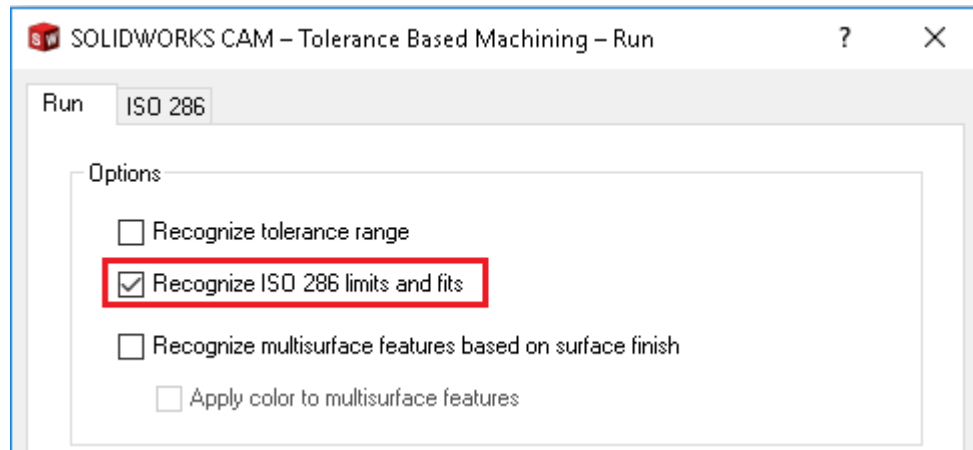
- vi. Click on the **OK** button of the **Tolerance Based Machining – Settings** dialog box to apply the changes and close the dialog box.

Step 5: Execute the Run Tolerance Based Machining Command

Selecting the Options in Tolerance Based Machining –Run dialog box

- i. On the **SOLIDWORKS CAM TBM Command Manager**, click on the **Run Tolerance Based Machining** command.
- ii. The **Tolerance Based Machining – Run** dialog box will be displayed. As this dialog box retains the settings from the previous time the **Run** command was executed, the checkbox options within the **Run tab** of this dialog box will accordingly be either in a checked or unchecked state.
- iii. Ensure that the **Recognize ISO 286 limits and fits** checkbox option is checked. Selecting this option is necessary to assign machining strategies to hole feature instances based on **ISO 286 Strategy Conditions**.

- iv. In this tutorial, no other 2.5 Axis Mill features other than hole features are being machined. The hole features present will be machined based on ISO 286 Strategy Conditions. Hence, ensure that the **Recognize tolerance range** checkbox option is unchecked as it is irrelevant to this tutorial (due to absence of other 2.5 Axis Mill Features on the tutorial part).
- v. The tutorial part does not contain any multisurface features. Hence, ensure that the **Recognize multisurface features based on surface finish** checkbox option is unchecked.



Select the 'Recognize ISO 286 limits and fits' option in the Run tab

Interpreting the information in the ISO 286 tab of TBM-Run dialog box

The **ISO 286** tab will be added to the *Tolerance Based Machining – Run* dialog box when the **Recognize ISO 286 limits and fits** option is checked. Click on this **ISO 286** tab.

The contents of this tab is almost identical to the contents of the [ISO 286 tab in the Tolerance Based Machining – Settings dialog box](#) but with a key difference. The **ISO 286** tab in the *Tolerance Based Machining- Run* dialog box provides additional information on the number of matches found between the **ISO Strategy Conditions** listed in this tab and the feature instances on the part being machined.

How to interpret the information in this tab is best understood via an illustrated example. Using this tutorial as an example, the explanation of the information displayed in this tab is given below.

Table of Features & Default Strategies

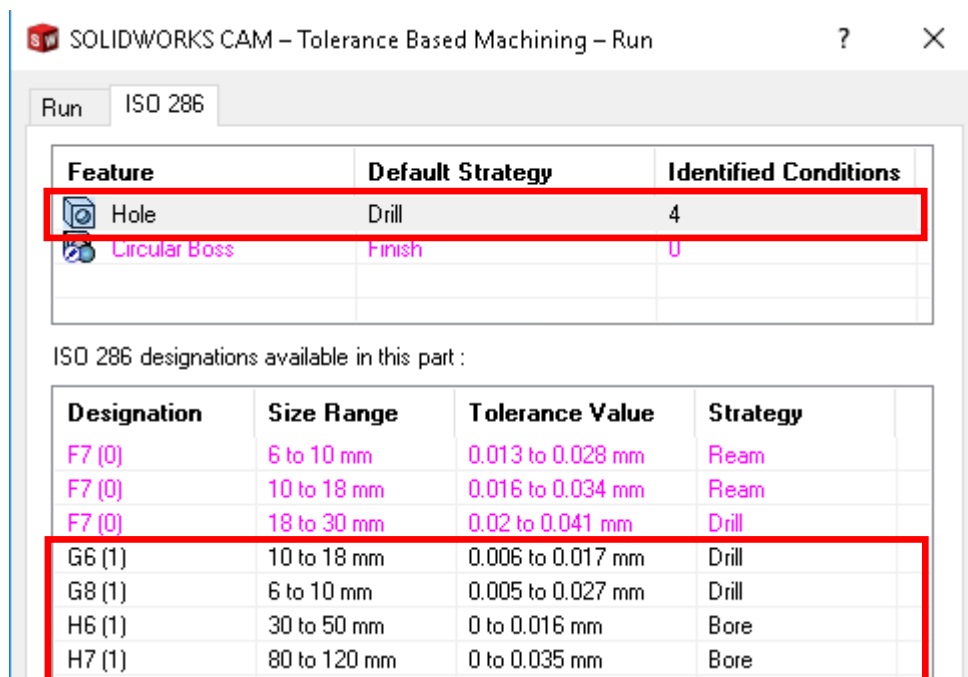
The **Hole** feature entry in this tab is displayed in **black** color font while the Circular Boss feature entry is in **magenta** color font.

- The **magenta** color font for the *Circular Boss* feature entry indicates any one of the following:

- None of the *ISO 286 Strategy Conditions* defined for the Circular Boss feature match with the Circular Boss feature instances on the tutorial part.
- The solid part does not have any Circular Boss feature instances. (This is the case for this tutorial.)

Consequently, the **Identified Conditions** field will display the number '0' indicating that no matches were found. All entries in the corresponding **ISO 286 Strategy Conditions** grid too will be displayed in **magenta** color font indicating that no matches were found.

- The **black** color font for the *Hole* feature entry indicates that one or more of the *ISO 286 Strategy Conditions* defined for the Hole feature match with the hole feature instances on the part. The **Identified Conditions** field indicates the number of matches found.



ISO 286 tab in the Tolerance Based Machining – Run dialog box

ISO 286 Strategy Conditions Grid

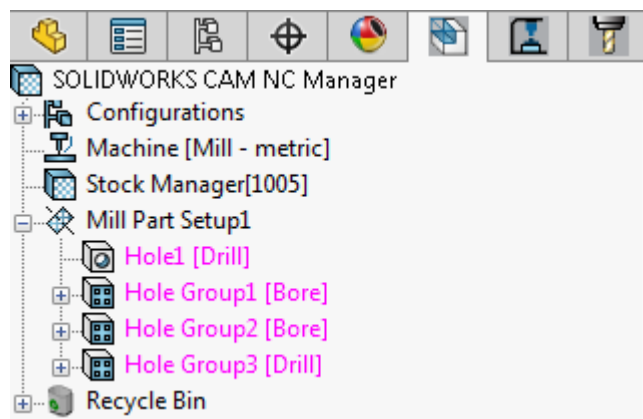
Observe the **ISO 286 Strategy Conditions** grid. The entries for **G6**, **G8**, **H6** and **H7** Designations in this grid are displayed in **black** color font. Other entries in this grid are in **magenta** color font.

- For *ISO 286 Strategy Condition* entries with **magenta** color font, the **magenta** color indicates that no matches with any of the hole feature instances present of the tutorial part were found. The number '0' in parentheses adjacent to the *Designation* name in the *Designation* field indicates the same viz. zero matches.
- For *ISO 286 Strategy Condition* entries with black color font, the **black** color font indicates that for each of these ISO 286 Strategy Condition entries,

one or more matches with the hole feature instances on the tutorial part have been found. The number of matches is indicated in parentheses in the *Designation* field adjacent to the *Designation* name. The machining strategies associated with these *ISO Strategy Condition* entries will be assigned to the hole feature instances when you click the **OK** button in the **Run** tab of this dialog box.

Executing the Run-TBM Command

- i. Switch to the **Run** tab of this dialog box.
- ii. In the **Run** tab, ensure only the **Recognize ISO 286 limits and fits** option is checked.
- iii. Click on the **OK** button to close the dialog box and execute the command.
- iv. Observe the **SOLIDWORKS CAM Feature tree**. The machining strategies for the Hole feature/hole groups have been reassigned as per the **ISO 286 Strategy Conditions**.



Machining Strategies reassigned for Hole Features in the SOLIDWORKS CAM Feature tree


Operations can now be generated for these features by executing the **Generate Operation Plan** command. Once operations are generated, generate toolpaths using the **Generate Toolpaths** command.

6. RECOGNIZING MULTISURFACE FEATURES BASED ON SURFACE FINISH

Recognizing Multisurface features using SOLIDWORKS CAM TBM


If *Surface Finish* properties and Roughness values are assigned to the various surfaces of a 3D part model using the **Annotations** functionality of SOLIDWORKS, then SOLIDWORKS CAM TBM provides a functionality wherein multisurface features can be recognized from such surfaces based on their specific Surface Finish or Roughness values. Additionally, you can choose to assign specific machining strategies to these multisurface features recognized on the basis of their *Surface Finish* values. This SOLIDWORKS CAM TBM functionality is provided in the form of **Multisurface Features** tab in the **Tolerance Based Machining – Settings** dialog box.

In order to do recognize multisurface features using SOLIDWORKS CAM TBM, ensure that the following conditions are fulfilled:

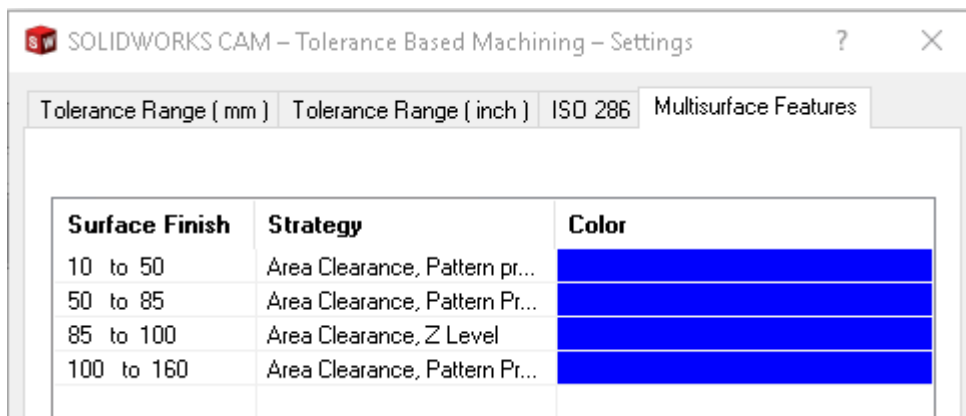
- i. *Surface Finish* properties and Roughness values are assigned to the various surfaces of a 3D part model using the **Annotations** functionality of SOLIDWORKS.
- ii. The necessary *Surface Finish* Ranges and corresponding Machining Strategies to be assigned are defined in the **Multisurface Features** tab of the **Tolerance Based Machining – Settings** dialog box. Optionally, you can assign the color with which the multisurface surface feature should be highlighted on the solid model in the graphics area once it is recognized.
- iii. When you execute the [Run Tolerance Based Machining command](#) , the **Recognize ISO 286 limits and fits** option in the **Run** tab of **Tolerance Based Machining – Run** dialog box is checked.

An illustrative tutorial explaining how to recognize multisurface features based on their Surface Finish and Roughness values using the SOLIDWORKS CAM TBM Plug-in is provided in the [next chapter](#).

Multisurface Features tab of Tolerance Based Machining – Settings dialog box

When you click on the *Tolerance Based Machining – Settings* command button  on the SOLIDWORKS CAM TBM Command Manager, the **Tolerance Based Machining – Settings** dialog box is displayed. Use the **Multisurface Features** tab of this dialog box to:

- i. Define the **Surface Finish** ranges based on which multisurface features present on a solid part model will be recognized (i.e. surfaces for which *Surface Finish* properties and Roughness values have been assigned using the **Annotations** functionality of *SOLIDWORKS*)
- ii. Indicate the machining strategies to be assigned to the multisurface features which fall within a particular **Surface Finish** range
- iii. Assign the color with which the multisurface surface feature should be highlighted on the solid model in the graphics area once it is recognized



Multisurface Features tab in Tolerance Based Machining – Settings Dialog Box

Parameters in the Multisurface Features tab

Surface Finish

This column indicates the Surface Finish range defined for each Strategy Condition. For each range, the lower and upper limit of the Surface Finish range is displayed.

You can edit/add *Surface Finish* ranges by using the [Range dialog box](#) displayed on clicking on the [Edit Surface Finish](#) button at the bottom left corner of the tab.

Strategy

This field indicates the machining strategy to be applied to the multisurface feature which falls within the *Surface Finish* range defined for the *Strategy* condition.

When a new entry (i.e. a new *Surface Finish* range) is added in this tab, the default machining strategy, as defined in the Technology Database, will be assigned in the corresponding *Strategy* field.


You can reassign this strategy by simply clicking on the *Strategy* field of a specific entry. Clicking on the *Strategy* field displays a dropdown list of all machining strategies defined in the Technology Database for multisurface features, including user-defined strategies. Select the desired strategy from

this dropdown list to change the strategy assigned. The changes will be saved when you click the **OK** button of the dialog box.

Color

This field indicates the color with which the multisurface feature present on the 3D part model will be highlighted in the graphics area on execution of the *Run Tolerance Based Machining* command when the *Surface Finish* value assigned to its surface falls within one of the **Surface Finish** ranges defined in the *Multisurface Features* tab.

In order for such multisurface features to be highlighted in the graphics area, the **Apply color to multisurface features** option in the *Run* tab of the *Tolerance Based machining – Run* dialog box must be checked.

By default, the color that will be used to highlight recognized multisurface features in the graphics area will be blue. Use the **Edit Color** button  at the bottom left corner of this tab to reassign the color with which the multisurface feature will be highlighted.

Edit Surface Finish



Clicking on this button opens the displays the [Range dialog box](#). Use this dialog box to add/edit the Surface Finish Ranges defined for strategy conditions in this tab.

Edit Color button



Clicking on this button opens *Color* dialog box. Use this dialog box to reassign the color associated with a *Surface Finish* range entry listed in the *Multisurface Features* tab. This is the color which will be used to highlight the multisurface feature in the graphics area when its associated **Surface Finish** value falls within one of the *Surface Finish* ranges listed in the *Multisurface Features* tab.

For details, refer: [Steps to change the color associated with the Strategy condition for a Multisurface Feature](#).

Thumbnail view of the selected color

When you select any entry listed in the **Multisurface Features** tab, the entire entry would be highlighted, including the [Color field](#). This makes it impossible to view the assigned color in the [Color field](#). In such a scenario, the purpose of the Thumbnail view at the bottom left corner of this tab is to display the color associated with selected entry.

Range dialog box to add Surface Finish ranges in Multisurface Features tab

The **Range** dialog box is displayed when you click on the [Edit Surface Finish](#)



button at the bottom left corner of the **Multisurface Features** tab. This dialog box is used to add/edit the **Surface Finish** Ranges of the Strategy condition entries in this tab.



How to use this dialog box is best understood with an illustrative example.

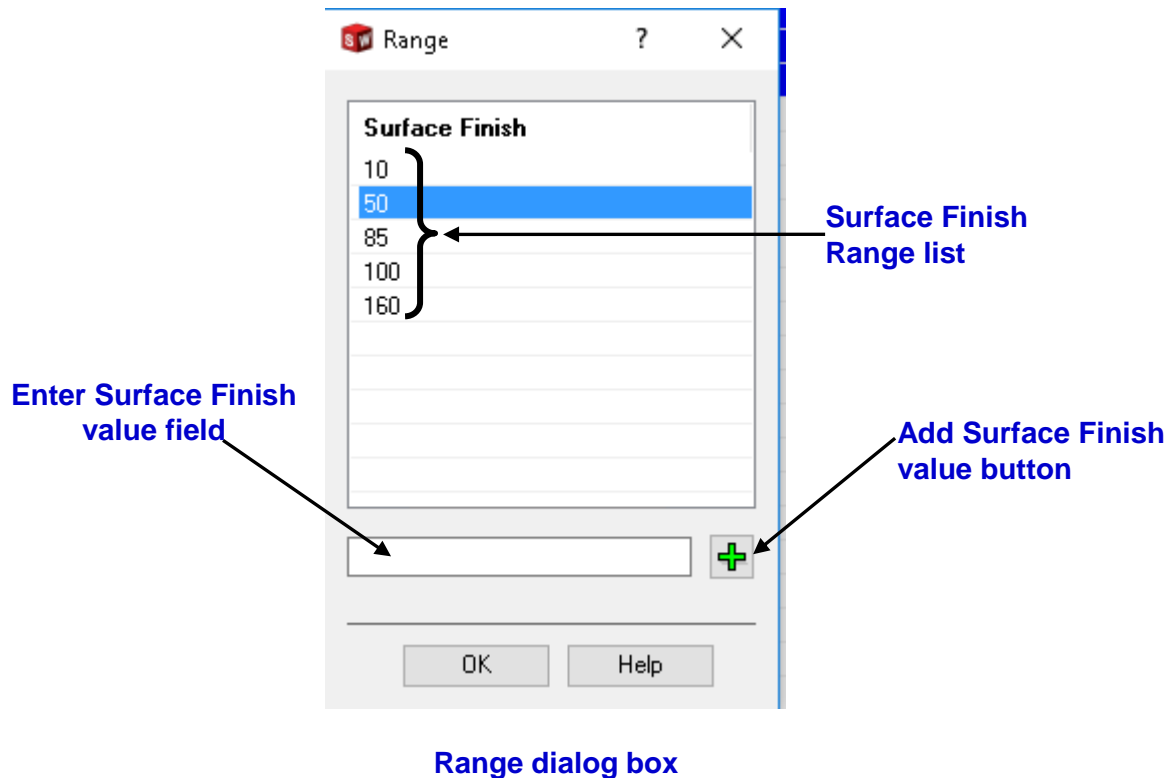
Example illustrating how to add Surface Finish Ranges using Range Dialog box

Consider that you want to define the following four Surface Finish ranges in the *Multisurface Features* tab of the *Tolerance Based Machining - Settings* dialog box:

	Lower Limit of Surface Finish Range	Upper Limit of Surface Finish Range
Range 1:	10	50
Range 2:	50	85
Range 3:	85	100
Range 4:	100	160

Following are the steps to define these surface finish ranges in the *Multisurface Features* tab:

- i. Open the Range dialog box by clicking on the [Edit Surface Finish](#) button  at the bottom left corner of the *Multisurface Features* tab.
- ii. Input each unique **Surface Finish** value in the **Enter Surface Finish** Value field. (The first value is 10.)
- iii. After entering one Surface Finish value, click on the **Add Surface Finish Value** button .
- iv. The Surface Finish value you have input will be listed in the **Surface Finish list** of this dialog box. Repeat *Steps ii.* and *iii.* for all remaining unique *Surface Finish* values that comprise the range. (In this example, there are five unique Surface Finish values viz. 10, 50, 85, 100 and 160.) The *Surface Finish* list will automatically sort and display the *Surface Finish* values in ascending order of value.
- v. In case you accidentally enter an incorrect Surface Finish value, you can delete its entry from the **Surface Finish** list by selecting the entry with a single mouse-click and pressing the *Delete* button.
- vi. After all *Surface Finish* values have been input, click on the **OK** button in the **Range** dialog box.




- vii. The **Range** dialog box will close and the Surface Finish ranges based on values that were defined in the **Range** dialog box will be listed in ascending order in the **Multisurface Features** tab of the *Tolerance Based Machining – Settings* tab.

Adding/Editing/Deleting the entries in the Multisurface Features tab

Editing the entries in the Multisurface tab

For existing entries listed in the Multisurface features tab, only the assigned Strategy or Color can be edited.

Steps to change the color associated with the Strategy condition for a Multisurface Feature

1. Using left mouse click, highlight the entry in the **Multisurface Features** tab for which you wish to change the color.
2. At the bottom left corner of this tab, click on the *Edit Color* button .
3. The **Color** dialog box will be displayed. Select the color of your choice and press the **OK** button.
4. The selected color will be visible in the [thumbnail view](#) to the right of the **Edit Color** button. When you shift the focus from the selected strategy

condition by clicking somewhere else within this tab, the **Color** field for the selected strategy condition will display the newly updated color.

Reassigning a Strategy


You can reassign this strategy by simply clicking on the *Strategy* field of that specific entry. Clicking on the *Strategy* field displays a dropdown list of all machining strategies defined in the Technology Database for multisurface features, including user-defined strategies. Select the desired strategy from this dropdown list to change the strategy assigned. The changes will be saved when you click the **OK** button of the dialog box.

Deleting entries in the Multisurface Features tab

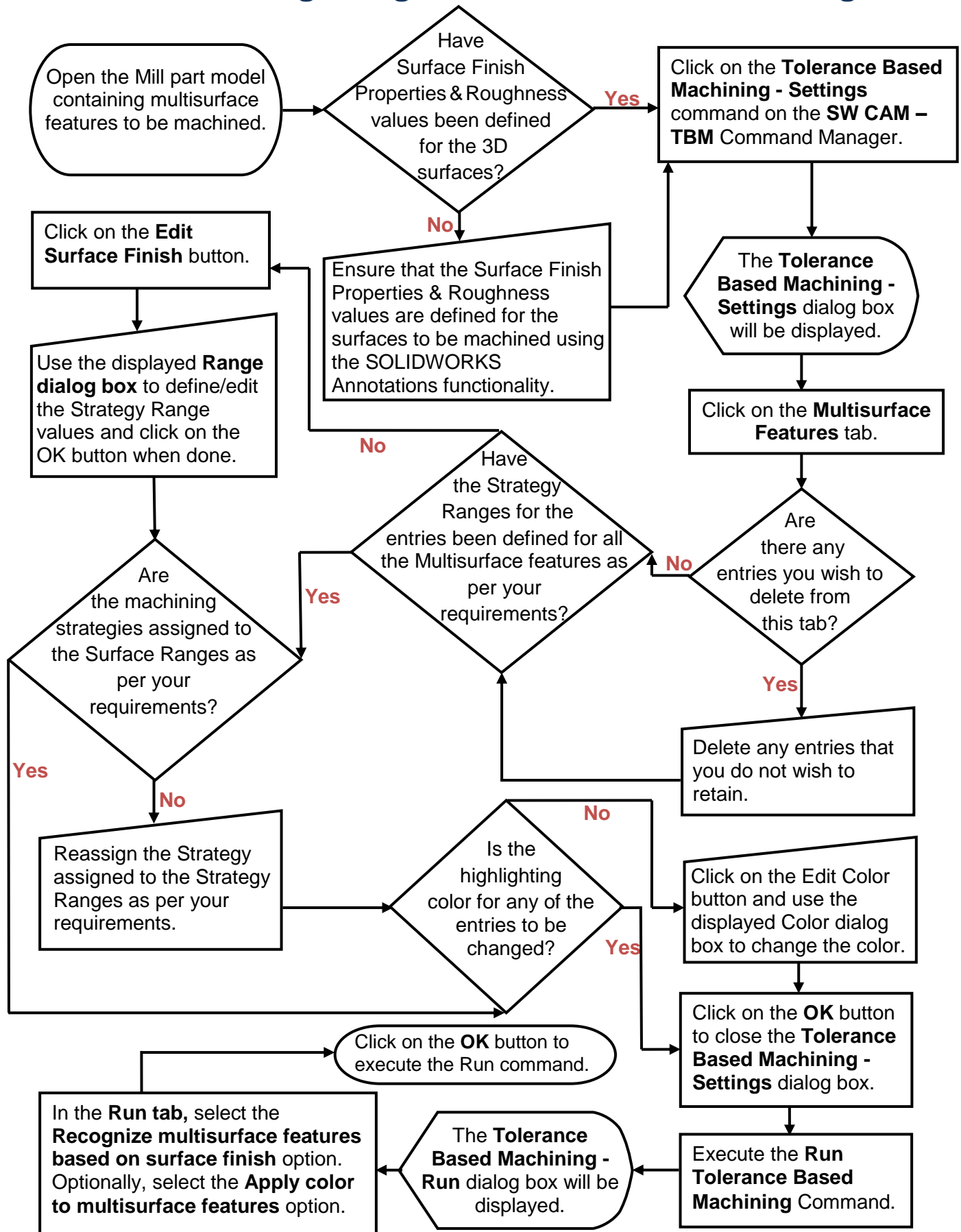
To delete an entry listed within the grid of the *Multisurface Features* tab, highlight the entry and click on the **Delete** button on the keyboard.

Adding new entries in the Multisurface Features tab

Following are the steps to add a new entry in the *Multisurface Features* tab:

- i. Add the Strategy Range for the new entry. [For detailed steps, refer: [Range dialog box to add Surface Finish ranges in Multisurface Features tab.](#)]
- ii. The default machining strategy for a multisurface feature (as defined in the Technology Database) will be listed in the **Strategy** field. [Reassign the machining strategy](#) as required.
- iii. Optionally, use the [Edit Color button](#)  to reassign the color with which the multisurface feature, if recognized, will be highlighted in the graphics area. [For details, refer: [Steps to change the color associated with the Strategy condition for a Multisurface Feature.](#)]
- iv. Once the required changes are completed, click on the **OK** button to close the dialog box.

Flowchart: Recognizing Multisurface Features using TBM

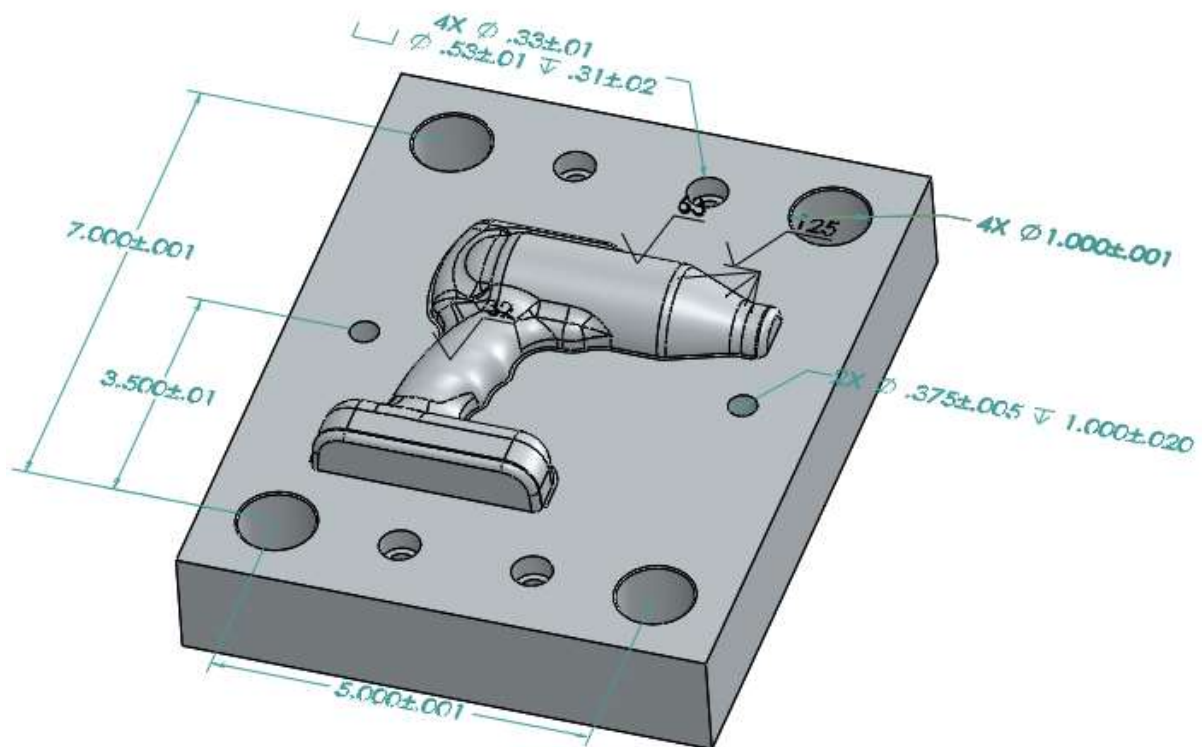


7. TUTORIAL 3: RECOGNIZING MULTISURFACE FEATURES BASED ON SURFACE FINISH

This tutorial illustrates how to recognize multisurface features on a mill part (whose surfaces have been assigned Surface Finish properties and Roughness values using the **Annotations** functionality of *SOLIDWORKS*) using the *SOLIDWORKS CAM TBM* plug-in.

Step 1: Open the tutorial part model



- Launch *SOLIDWORKS CAM* application as an Add-In within the *SOLIDWORKS* application.
- Click on the *File* menu of *SOLIDWORKS* application and select the *Open* menu option.
- The *File Open* dialog box will be displayed. Browse to the following folder location:
C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2018\CAM Examples\TBM_SampleParts\Tutorials Parts
- Select the ***TBM_Tutorial_3.SLDPRT*** file and click on the *Open* button.

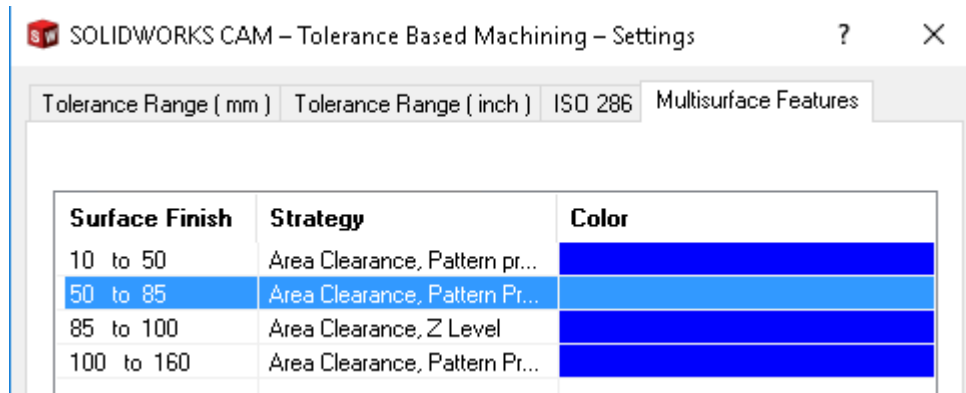


TBM_Tutorial_3.SLDPRT

Observe that this solid model has multiple surfaces and three hole group features.

Step 2: Assign Settings in Multisurface Features dialog box



- On the *SOLIDWORKS CAM Command Manager*, click on the **Tolerance Based Machining** command button .
- The **SOLIDWORKS CAM TBM Command Manager** will be activated. Click on the **Tolerance Based Machining - Settings**  command on this *Command Manager*.
- The **Tolerance Based Machining – Settings** dialog box will be displayed. Click on the **Multisurface Features** tab.
- In this tutorial, we will delete all the existing entries in the *Multisurface Features* tab and replace it with new entries and associated machining strategies. (The default installation of **SOLIDWORKS CAM TBM** has four entries in this tab.)
- To delete an entry in this tab, select the entry with the left click of the mouse and click on the **Delete** button on the keyboard. Use this method to delete all the entries.

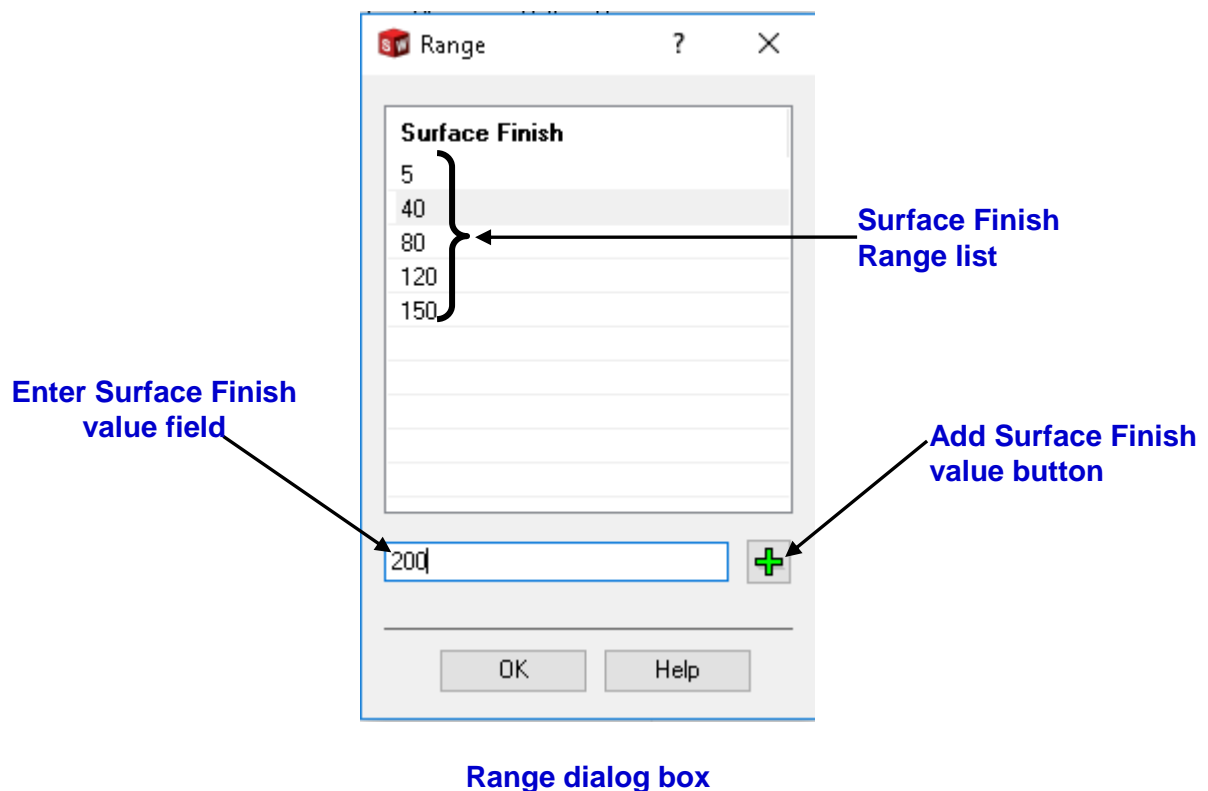


Highlight an entry within the tab and press 'Delete' button to delete the entry

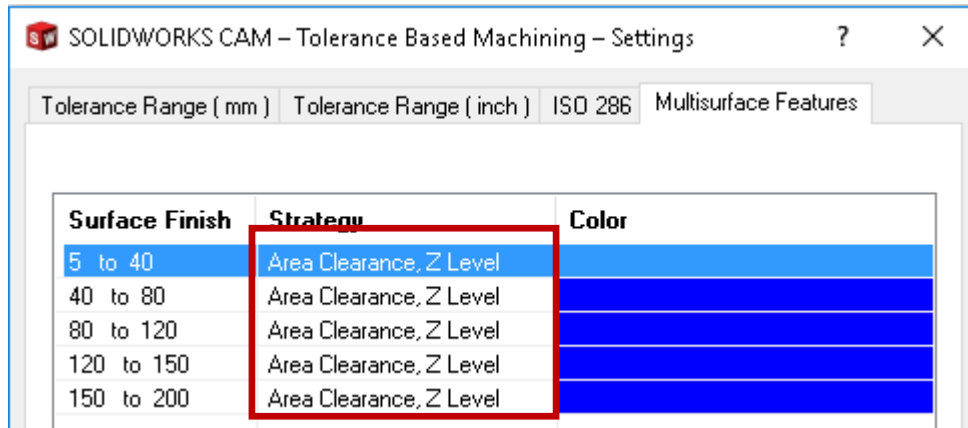
- Once all the entries are deleted, the next step is to add new entries. Surface ranges have to be defined and machining strategies have to be assigned to these new entries. The new surface ranges to be defined and the corresponding machining strategies to be assigned are given in the below table.

	Lower Limit of Surface Finish Range	Upper Limit of Surface Finish Range	Machining Strategy to be assigned
Range 1:	05	40	Fine
Range 2:	40	80	Area Clearance, Z Level
Range 3:	80	120	Area Clearance, Constant Stepover
Range 4:	120	150	Area clearance, Pattern Project
Range 5:	150	200	Coarse

- vii. Following are the steps to define the Surface Ranges.
- Open the **Range** dialog box by clicking on the *Edit Surface Finish* button  at the bottom left corner of the *Multisurface Features* tab.
 - Click within the **Enter Surface Finish Value** field to shift focus on this field. (The cursor must blink within this field.)
 - The Surface Ranges to be entered have six unique values (viz. 05, 40, 80, 120, 150, 200). Input any one of these unique **Surface Finish** values in the **Enter Surface Finish Value** field.
 - After entering one Surface Finish value, click on the **Add Surface Finish Value** button .
 - The Surface Finish value you have input will be listed in the **Surface Finish list** of this dialog box. Repeat Steps b, c and d for all remaining unique *Surface Finish* values that comprise the range. The *Surface Finish* list will automatically sort and display the *Surface Finish* values in ascending order of value.
 - In case you accidentally enter an incorrect Surface Finish value, you can delete its entry from the **Surface Finish** list by selecting the entry with a single mouse-click and pressing the *Delete* button.
 - After all *Surface Finish* values have been input, click on the **OK** button in the **Range** dialog box.

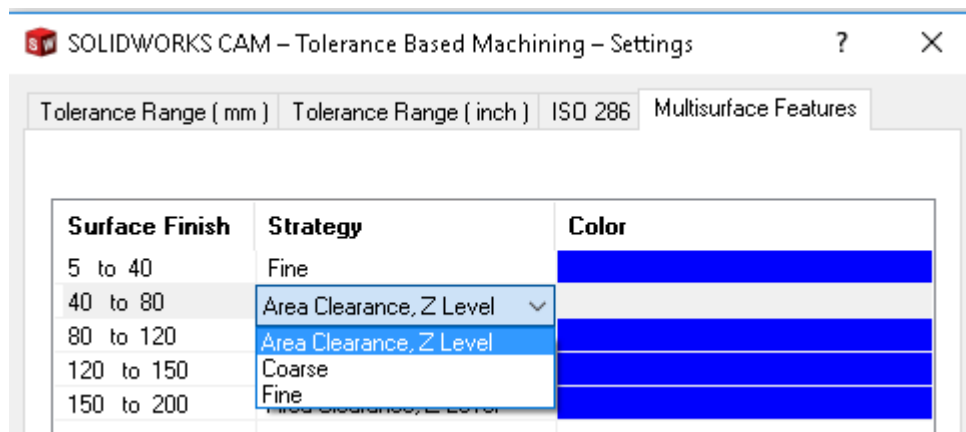


- viii. The user interface will revert to the **Multisurface Features** tab. Observe that the newly input Surface Ranges are listed within this tab. The machining strategy assigned to each of these new entries is the default machining strategy for multisurface features in the Technology database.



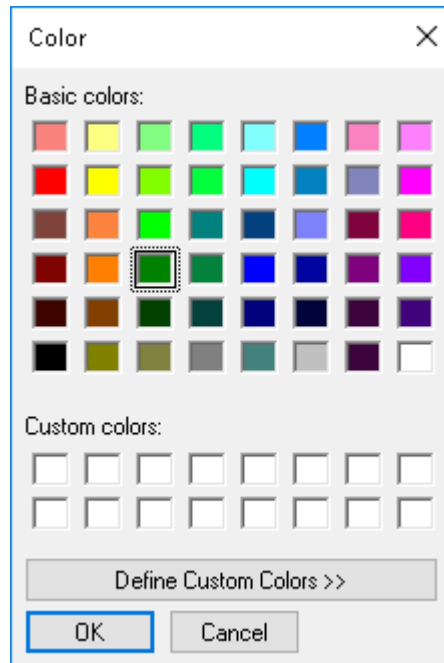
Default Machining Strategy from TechDB assigned to all new entries

- ix. To reassign the strategy, click on the **Strategy** field of a specific entry and select the desired strategy from the dropdown list.



Reassigning Machining Strategy for a Strategy Range

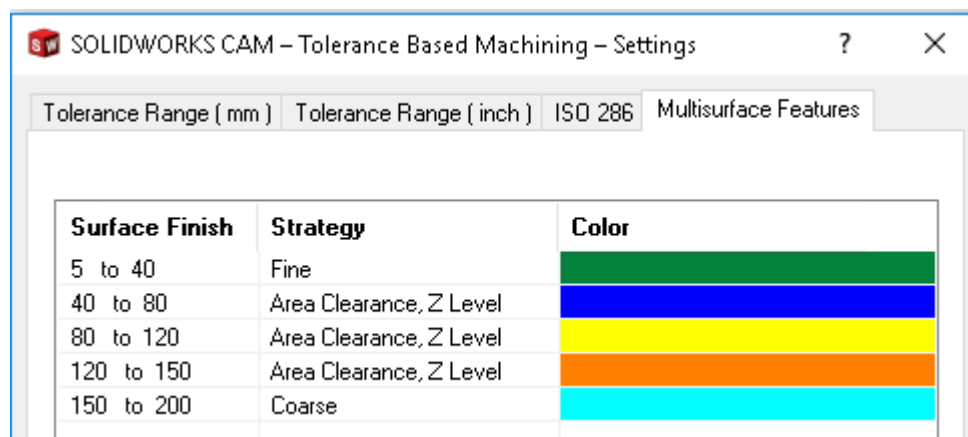
- x. Assign machining strategies for each of the newly defined Surface ranges as per the table given in [Step vi](#).
- xi. The next step is to assign the color which will be used to highlight the multisurface features on the solid model. (The multisurface features can be thus highlighted only once they are recognized based on their with Surface Finish values.) By default, the color for all the entries is **Blue**. To assign a different color, following are the steps:
- Highlight the specific entry in the **Multisurface Features** tab and click on the **Edit Color** button at the bottom left corner of the tab
 - The **Color** dialog box will be displayed. Pick the desired color and click on the **OK** button to close the dialog box.



Color Dialog Box

- c. Repeat this step for all the entries so that each entry has a different color in order to enable easy identification of multisurface features once they are recognized.


Whenever an entry is selected in the Multisurface Features tab, the corresponding color will be displayed in a thumbnail view at the bottom left corner of the tab.

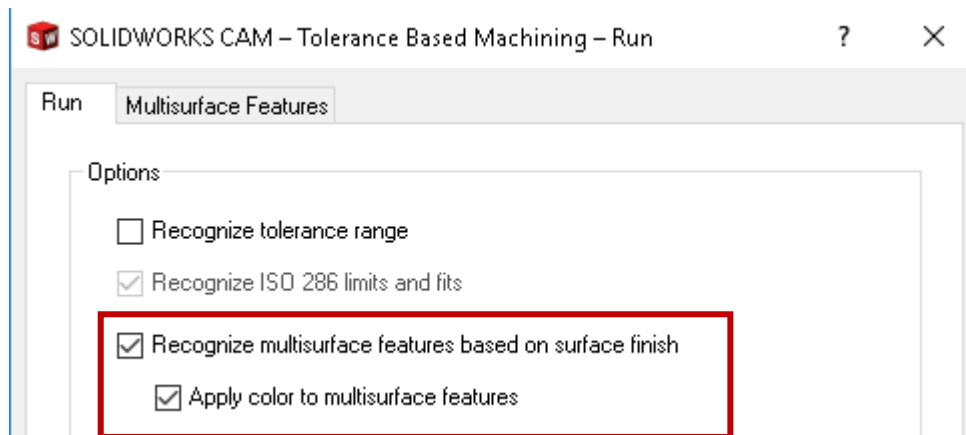


Multisurface Features tab after assigning different colors for each of the entries

- xii. Click on the OK button to close the dialog box.

Step 3: Execute the Run Tolerance Based Machining Command

- i. On the **SOLIDWORKS CAM TBM Command Manager**, click on the **Run Tolerance Based Machining**  command.
- ii. The **Tolerance Based Machining – Run** dialog box will be displayed. As this dialog box retains the settings from the previous time the **Run** command was executed, the checkbox options within the **Run tab** of this dialog box will accordingly be either in a checked or unchecked state.
- iii. Ensure that the **Recognize multisurface features based on surface finish** checkbox option is checked. Selecting this option is necessary to recognize multisurface features and assign machining strategies based on their *Surface Finish* values.
- iv. Placing a check in the *Recognize multisurface features based on surface finish* checkbox option also enables the checkbox labelled **Apply color to multisurface features** beneath it. In this tutorial, the multisurface features that will be recognized on the basis of their surface finish values will be highlighted on the solid part in the graphics area using colors specified in the *Multisurface Features* tab of the **Tolerance Based Machining – Settings** dialog box. To enable such colored highlighting, place a check in this checkbox.



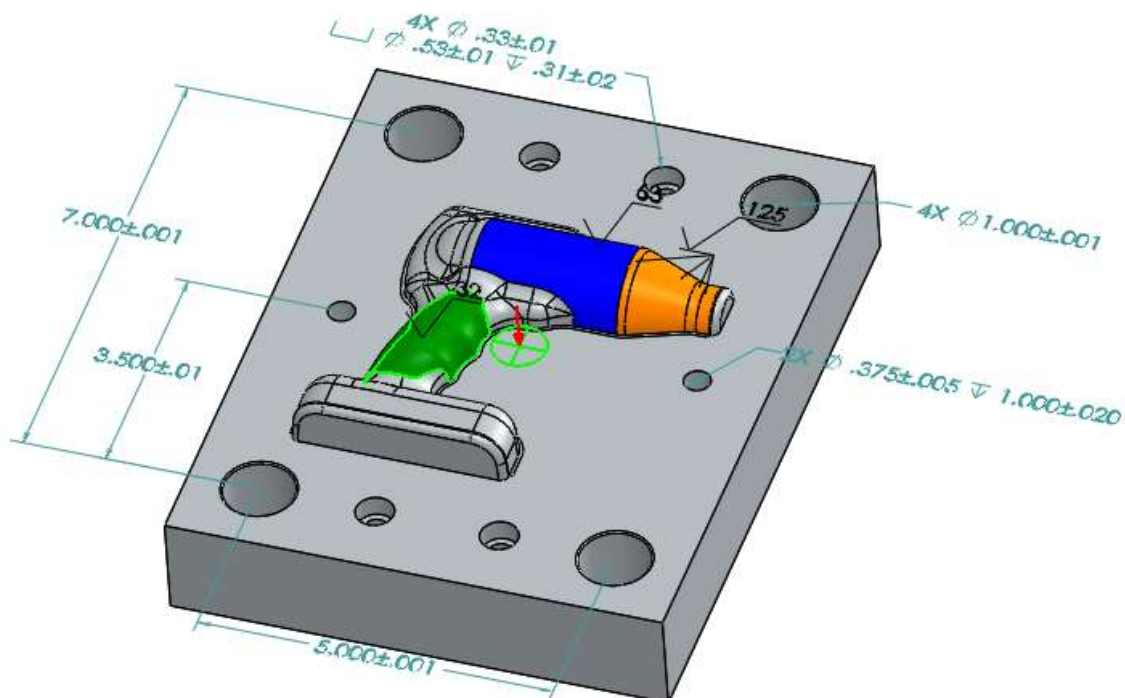
Options to be selected in Run tab of Tolerance Based Machining – Run dialog box

- v. Observe that the *Multisurface Features* tab is added to this dialog box on selecting the *Recognize multisurface features based on surface finish* checkbox option. Click on this tab.
- vi. The parameters within this tab are identical to the parameters in the *Multisurface Features* tab of **TBM – Settings** dialog box. However, this tab provides additional information with respect to the solid part to be machined.



Multisurface Features tab in TBM – Run dialog box

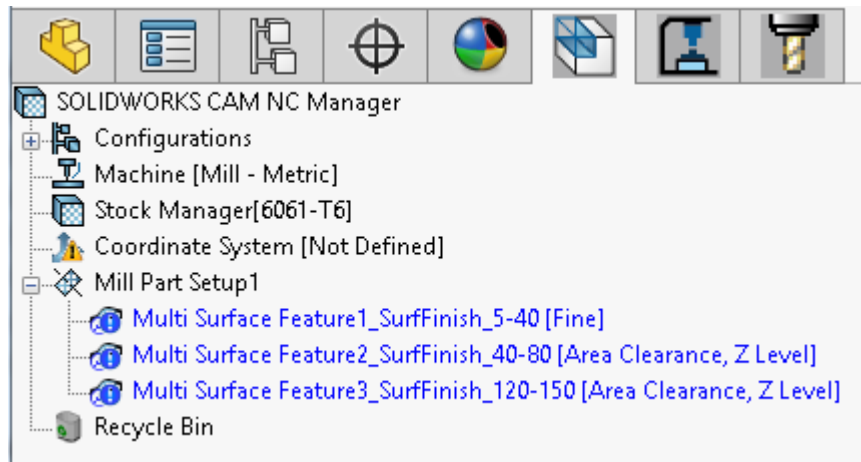
- vii. Observe that three of the five entries within this tab are displayed in black color font while two of them are displayed in magenta color font. The black colored entries indicate that one or more matches with the part being machined has been found i.e. one or more multisurface features with surface finish values falling within that particular **Surface Finish** range will be identified on clicking the **OK** button in the **Run** tab of this dialog box.
- viii. Switch to the **Run** tab. Click on the **OK** button to execute multisurface feature recognition.
- ix. Observe the solid model. The three surface for which Surface Finish values had been defined have been recognized and highlighted with a specific color on the model.



Multisurface features recognized on the basis of their Surface Finish values

Note: The only reason other multisurface features have not been recognized is because no Surface Finish values were defined for those surfaces. SOLIDWORKS CAM TBM can recognize multisurface feature only if their Surface Finish values are defined using SOLIDWORKS *Annotations* functionality.

- x. Observe the *SOLIDWORKS CAM Feature tree*. The three recognized Multisurface features are listed in the tree. Their corresponding machining strategies (as defined in the **TBM – Settings** dialog box) are displayed adjacent to the multisurface feature item.



Recognized Multisurface features listed in the SOLIDWORKS CAM Feature tree

LEGAL NOTICES

© 1995-2018, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

The information and the software discussed in this document are subject to change without notice and are not commitments by Dassault Systemes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.

Patent Notices

- SOLIDWORKS® 3D mechanical CAD and/or Simulation software is protected by U.S. Patents 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940; 8,305,376; 8,581,902; 8,817,028; 8,910,078; 9,129,083; 9,153,072; 9,262,863; 9,465,894; 9,646,412; 9,870,436; 10,055,083; 10,073,600 and foreign patents, (e.g., EP 1,116,190 B1 and JP 3,517,643).
- eDrawings® software is protected by U.S. Patent 7,184,044; U.S. Patent 7,502,027; and Canadian Patent 2,318,706.
- U.S. and foreign patents pending.

Trademarks and Product Names for SOLIDWORKS Products and Services

SOLIDWORKS, 3D ContentCentral, 3D PartStream.NET, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.

CircuitWorks, FloXpress, PhotoView 360, and TolAnalyst are trademarks of DS SolidWorks.

FeatureWorks is a registered trademark of HCL Technologies Ltd.

SOLIDWORKS 2019, SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, SOLIDWORKS PDM Professional, SOLIDWORKS PDM Standard, SOLIDWORKS Simulation Standard, SOLIDWORKS Simulation Professional, SOLIDWORKS Simulation Premium, SOLIDWORKS Flow Simulation, SOLIDWORKS CAM, SOLIDWORKS Manage, eDrawings Viewer, eDrawings Professional, SOLIDWORKS Sustainability, SOLIDWORKS Plastics, SOLIDWORKS Electrical Schematic Standard, SOLIDWORKS Electrical Schematic Professional, SOLIDWORKS Electrical 3D, SOLIDWORKS Electrical Professional, CircuitWorks, SOLIDWORKS Composer, SOLIDWORKS Inspection, SOLIDWORKS MBD, SOLIDWORKS PCB powered by Altium, SOLIDWORKS PCB Connector powered by Altium, and SOLIDWORKS Visualize are product names of DS SolidWorks.

Other brand or product names are trademarks or registered trademarks of their respective Holders.

COMMERCIAL COMPUTER SOFTWARE – PROPRIETARY

The Software is a “commercial item” as that term is defined at 48 C.F.R. 2.101 (OCT 1995), consisting of “commercial computer software” and “commercial software documentation” as

such terms are used in 48 C.F.R. 12.212 (SEPT 1995) and is provided to the U.S. Government (a) for acquisition by or on behalf of civilian agencies, consistent with the policy set forth in 48 C.F.R. 12.212; or (b) for acquisition by or on behalf of units of the Department of Defense, consistent with the policies set forth in 48 C.F.R. 227.7202-1 (JUN 1995) and 227.7202-4 (JUN 1995).

In the event that you receive a request from any agency of the U.S. Government to provide Software with rights beyond those set forth above, you will notify DS SolidWorks of the scope of the request and DS SolidWorks will have five (5) business days to, in its sole discretion, accept or reject such request. Contractor/Manufacturer: Dassault Systemes SolidWorks Corporation, 175 Wyman Street, Waltham, Massachusetts 02451 USA.

Copyright Notices for SOLIDWORKS Standard, Premium, Professional, and Education Products

- Portions of this software © 1986-2018 Siemens Product Lifecycle Management Software Inc. All rights reserved.
- This work contains the following software owned by Siemens Industry Software Limited:
 - ▀ D-Cubed® 2D DCM © 2018. Siemens Industry Software Limited. All Rights Reserved.
 - ▀ D-Cubed® 3D DCM © 2018. Siemens Industry Software Limited. All Rights Reserved.
 - ▀ D-Cubed® PGM © 2018. Siemens Industry Software Limited. All Rights Reserved.
 - ▀ D-Cubed® CDM © 2018. Siemens Industry Software Limited. All Rights Reserved.
 - ▀ D-Cubed® AEM © 2018. Siemens Industry Software Limited. All Rights Reserved.
- Portions of this software © 1998-2018 HCL Technologies Ltd.
- Portions of this software incorporate PhysX™ by NVIDIA 2006-2010.
- Portions of this software © 2001-2018 Luxology, LLC. All rights reserved, patents pending.
- Portions of this software © 2007-2018 DriveWorks Ltd.
- © 2011, Microsoft Corporation. All rights reserved.
- Includes Adobe® PDF Library technology
Copyright 1984-2016 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents 5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382;
- Patents Pending.

Copyright Notices for SOLIDWORKS Simulation Products

- Portions of this software © 2008 Solversoft Corporation.
- PCGLSS © 1992-2017 Computational Applications and System Integration, Inc. All rights reserved.

Copyright Notices for SOLIDWORKS PDM Professional Product

- Outside In® Viewer Technology, © 1992-2012 Oracle
- © 2011, Microsoft Corporation. All rights reserved.

Copyright Notices for eDrawings Products

- Portions of this software © 2000-2014 Tech Soft 3D.
- Portions of this software © 1995-1998 Jean-Loup Gailly and Mark Adler.
- Portions of this software © 1998-2001 3Dconnexion.
- Portions of this software © 1998-2017 Open Design Alliance. All rights reserved.
The eDrawings® for Windows® software is based in part on the work of the Independent JPEG Group.
- Portions of eDrawings® for iPad® copyright © 1996-1999 Silicon Graphics Systems, Inc.



- Portions of eDrawings® for iPad® copyright © 2003 – 2005 Apple Computer Inc.

Copyright Notices for SOLIDWORKS PCB Products

- Portions of this software © 2017-2018 Altium Limited.

Copyright Notices for SOLIDWORKS Visualize Products

- NVIDIA GameWorks™ Technology provided under license from NVIDIA Corporation. Copyright © 2002-2015 NVIDIA Corporation. All rights reserved