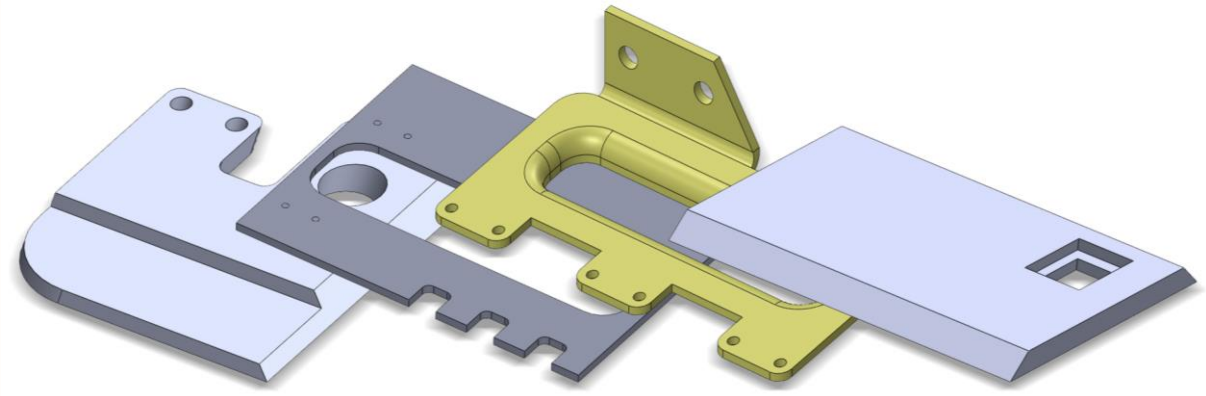
CAMWorks Nesting

A Geometric Product

CAMWorks Nesting 2015

User Guide & Tutorials



Disclaimer

Geometric Americas Inc. makes no warranties, either expressed or implied with respect to this manual. Geometric Americas Inc. reserves the right to revise and improve products as it sees fit, and to revise the specifications and information contained herein without prior notice. Due to continuing product development, specifications and capabilities described in this manual are subject to change without notice.

Trademarks

The following and other product names and corporate references herein are registered or proprietary trademarks of their respective owners.

CAMWorks Nesting is a registered trademark of Geometric Americas Inc.

SOLIDWORKS® is a registered trademark of Dassault Systèmes SolidWorks Corp.

Microsoft®, Windows®, Windows Vista®, Window 7® and Access® are registered trademarks of Microsoft Corporation in the United States and/or other countries.

All other brands and names are the property of their respective owners.

Geometric Ltd. is a subsidiary of Geometric Americas, Inc.

Product Name: **CAMWorks Nesting**
Version: **2015 SP 0.0**
License Version Date: **September 15, 2014**

TABLE OF CONTENTS

CAMWORKS NESTING: INTRODUCTION

What is CAMWorks Nesting?

CAMWorks Nesting, developed by **Geometric Americas, Inc.**, is an automatic, true-shape nesting program that easily creates fast and efficient nested layouts. It is seamlessly integrated within SOLIDWORKS®/CAMWorks Solids and allows nesting of flat or 3D solid or sheet metal parts and assemblies.

CAMWorks Nesting can be used to create efficient layouts of metal, wood or composite based materials, producing the maximum number of parts from a single piece of raw material within minutes.

Procuring CAMWorks Nesting

Geometric Americas, Inc. sells CAMWorks Nesting and related program modules through a worldwide network of **Value Added Resellers**.

➔ If you are an **existing user** of our other product named CAMWorks, you can contact your Reseller for CAMWorks Nesting.

➔ If you are a **first-time user** of CAMWorks Nesting, you can find your local CAMWorks Reseller on www.camworks.com

Note: **CAMWorks Nesting can be purchased only through CAMWorks Resellers. Though the CAMWorks Nesting installer can be downloaded from the CAMWorks website, the license required to run the CAMWorks Nesting application can be purchased only from an authorized Reseller.**

Why CAMWorks Nesting should be your preferred nesting program

CAMWorks Nesting has the following advantages that make it the ideal choice when it comes to choosing the nesting application to suit your needs:

- **Ease of Use:** Parts imported from other CAD applications or created in SOLIDWORKS as well as assemblies can be directly used as an input without the need to convert them to flat patterns.
- **Full Associativity with SOLIDWORKS:** Updates are tracked and flagged whenever the component is changed. Refresh rebuild the nest to reflect the updated designs.
- **SOLIDWORKS Compatible Output:** Provides the nested output as a new SOLIDWORKS assembly and retains the original part and assembly model files. The SOLIDWORKS nested assembly can then be used for further processing, such as toolpath and NC Code generation with CAMWorks or any other CAM software, if required.

- **Part Requirements automatically assigned**: Automatically nests multiple parts, based on material and thickness, within an assembly in a single run.

  This feature helps users eliminate manual efforts in segregating individual parts with the same material and thickness for a nesting operation.
- **Material Optimization:** The advanced true-shape automatic nesting algorithms reduce raw material consumption by providing optimized and compact layouts.
- **Nesting with multiple tool heads:** An optional feature to nest two or more identical nesting layouts using multiple tool heads is provided. This feature is useful for flame cutting applications.
- **Save Nested layout Output as DXF file:** An optional feature that allows users to save the nested layouts in the internationally accepted CAD data file format known as 'Drawing Exchange Format' *(.dxf)*, in addition to the existing assembly file format *(.sldasm)*.
- **Unfold Imported Sheet Metal Bodies:** Supports nesting of imported sheet metal part models containing bends. Using this 'Unfold Imported Bodies' dialog box, such sheet metal parts can be unfolded before executing the nesting job.

Receiving latest updates & Update Support Plan (USP)

Update Support Plan

Update Support Plan (USP): When you purchase CAMWorks Nesting through a Reseller, you will receive a permanent license required to run the CAMWorks Nesting application. In addition to this, you will also be enrolled in a CAMWorks Nesting Update Support Plan (USP) for a specific duration. Your Reseller will brief you about the USP when you purchase CAMWorks Nesting.

Being enrolled in the CAMWorks Nesting Update Support Plan has the following benefits:

➔ **Receiving updates:** It allows you to keep your CAMWorks Nesting application up-to-date with the new features and performance improvements of CAMWorks Nesting released in the form of Service Packs.

➔ **Technical Support:** You receive technical support for all your queries and doubts regarding CAMWorks Nesting.

Once your USP expires, you will no longer receive updates or support. Ensure that you repurchase an appropriate Update Support Plan from your CAMWorks Reseller to continue receiving technical support and updates.

Note: **The CAMWorks Nesting license you purchase from your Reseller will be perpetual in nature. However, the Update Support Plan has a fixed duration. You need to repurchase an Update Support Plan after your current plan expires.**

Technical Support

This manual has been designed to be as informative as possible. In case you still face problems related to installation, license activation or using CAMWorks Nesting, you can write back to us at:

support@camworks.com

We will get back to you with the solution to your query within two working days.

CAMWorks Nesting Installation & License Activation

A separate manual has been provided to acquaint you with the details of installation and License activation for CAMWorks Nesting.
This manual is available on the CAMWorks website in the Downloads section for CAMWorks Nesting-related downloads.
After you install CAMWorks Nesting, this manual can be accessed from ***Start>>All Programs>>CAMWorksNesting 201x>>Installation & License Activation Guide***.

CAMWorks Nesting Tutorials

The last section of this manual contains illustrated tutorials which will help you understand all the aspects of using CAMWorks Nesting for practical purposes.
Refer: CAMWorks Nesting Tutorials section.

ABOUT THESE TUTORIALS

Section 1: The first section '**Understanding the CAMWorks Nesting Fundamentals**' introduces the CAMWorks Nesting User Interface, working environment and the various Nesting parameters.

Section 2: The second section '**Initialization files of CAMWorks Nesting**' explains how to use the initialization files present in CAMWorks Nesting to define and edit default values, settings and populate the dropdown fields. It is highly recommended that you read this section in order to gain an understanding of how to customize the CAMWorks Nesting settings to meet your facility's requirements.

Section 3: An understanding of these basic elements is required before proceeding to the tutorials. The 12 tutorials given in this document will help you to learn how to use CAMWorks Nesting through a step by step hands-on tour of its features and functions. The tutorials are presented in order of increasing complexity, each building upon the knowledge gained from the previous tutorial.

Tutorial	Topic covered in the Tutorial
Tutorial 1 – Assembly Nesting	Nesting a Sheet Metal assembly.
Tutorial 2 – Single part, Single sheet Nesting	Nesting a Solid part.
Tutorial 3 – Single Part, Single sheet Nesting	Nesting a Sheet Metal Part.
Tutorial 4 – Nesting by Thickness	Nesting Parts of different thickness.
Tutorial 5 – Nest by material, Nest by Thickness	Nesting Parts of different material & thickness.
Tutorial 6 – Nesting with Multiple tool heads	Nesting Parts of identical material and thickness intended to be machined using a Machine with Multiple Tool Heads.
Tutorial 7 – Nesting of Imported Sheet Metal Parts	Nesting of Imported sheet metal parts with bends.
Tutorial 8 – Nesting of Imported Sheet Metal Parts with faulty surfaces	Nesting of Imported sheet metal parts.
Tutorial 9 – Assigning Assembly Quantities	Nesting an Assembly comprising sub-assemblies and parts.

Tutorial	Topic covered in the Tutorial
Tutorial 10 – Unfolding Sheet Metal Components Using 'Interactive Unfold' Command	Using 'Interactive Unfold" command to unfold sheet metal parts.
Tutorial 11 – The Stamp Feature Unfold Option	Using 'Stamp Feature Unfold' Command.
Tutorial 12 – Generating NC codes for Nested layouts using CAMWorks (I)	Generating NC codes using CAMWorks application for Nested layouts.
Tutorial 13 – Generating NC codes for Nested layouts using CAMWorks (II)	Generating NC codes using CAMWorks application for Nested layouts using functionality to link CAMWorks Nesting with CAMWorks.

Additional information is available in the **CAMWorks Nesting Context Based Help**. It is highly recommended that you read theses tutorials to gain a deeper and practical understanding of CAMWorks Nesting features and capabilities.

SECTION ONE

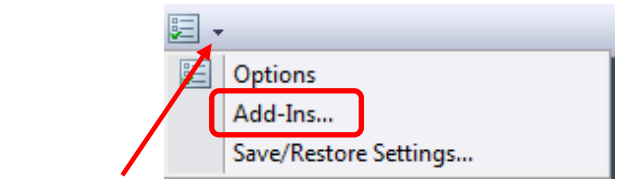
CAMWorks Nesting Fundamentals

UNDERSTANDING THE CAMWORKS NESTING FUNDAMENTALS

Basic Procedure of Nesting

Follow these general procedures to generate nested layouts using CAMWorks Nesting.

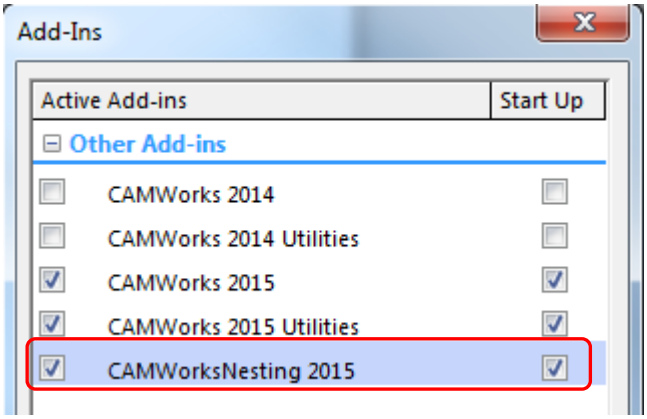
1. Open *SOLIDWORKS/CAMWorks Solids*. Click on the dropdown button of the *CAMWorks Options* in the CAMWorks Menu Bar and select *Add-Ins*.



CAMWorks Options icon

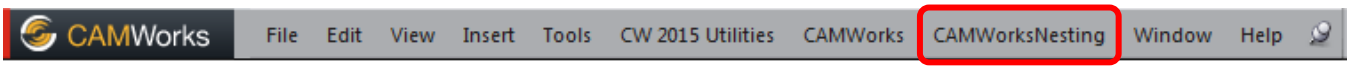
Selecting CAMWorks Add-Ins

2. Load the CAMWorks Nesting Add-In.



Selecting the CAMWorksNesting Add-In

3. The CAMWorks Nesting Menu will be added to the SOLIDWORKS/CAMWorks Solids menu bar.



CAMWorksNesting Menu added to CAMWorks Menu bar

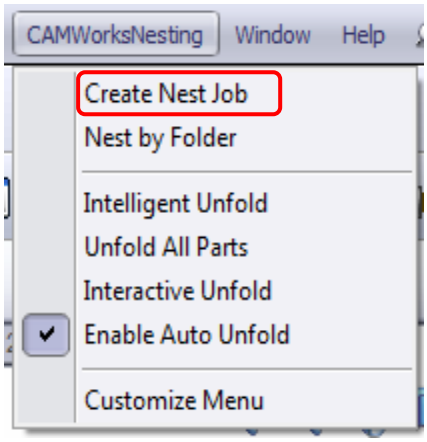
4. For **Single-part nesting**:
   a. Model or open a sheet metal part/ solid part model in SOLIDWORKS. For example, open the part *'Tutorial_1a'* located in the following folder.

      Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples/Tutorials\Assemblies\Tutorial1

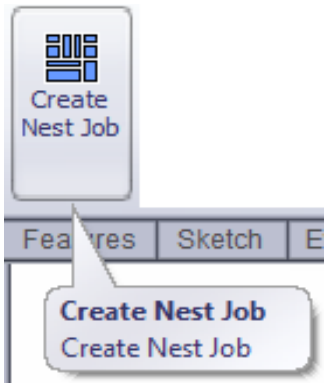
   b. Select *Create Nesting Job* from the CAMWorksNesting menu bar.

      OR

      Click on the *Create Nest Job* button on the CAMWorks Nesting Ribbon Bar.

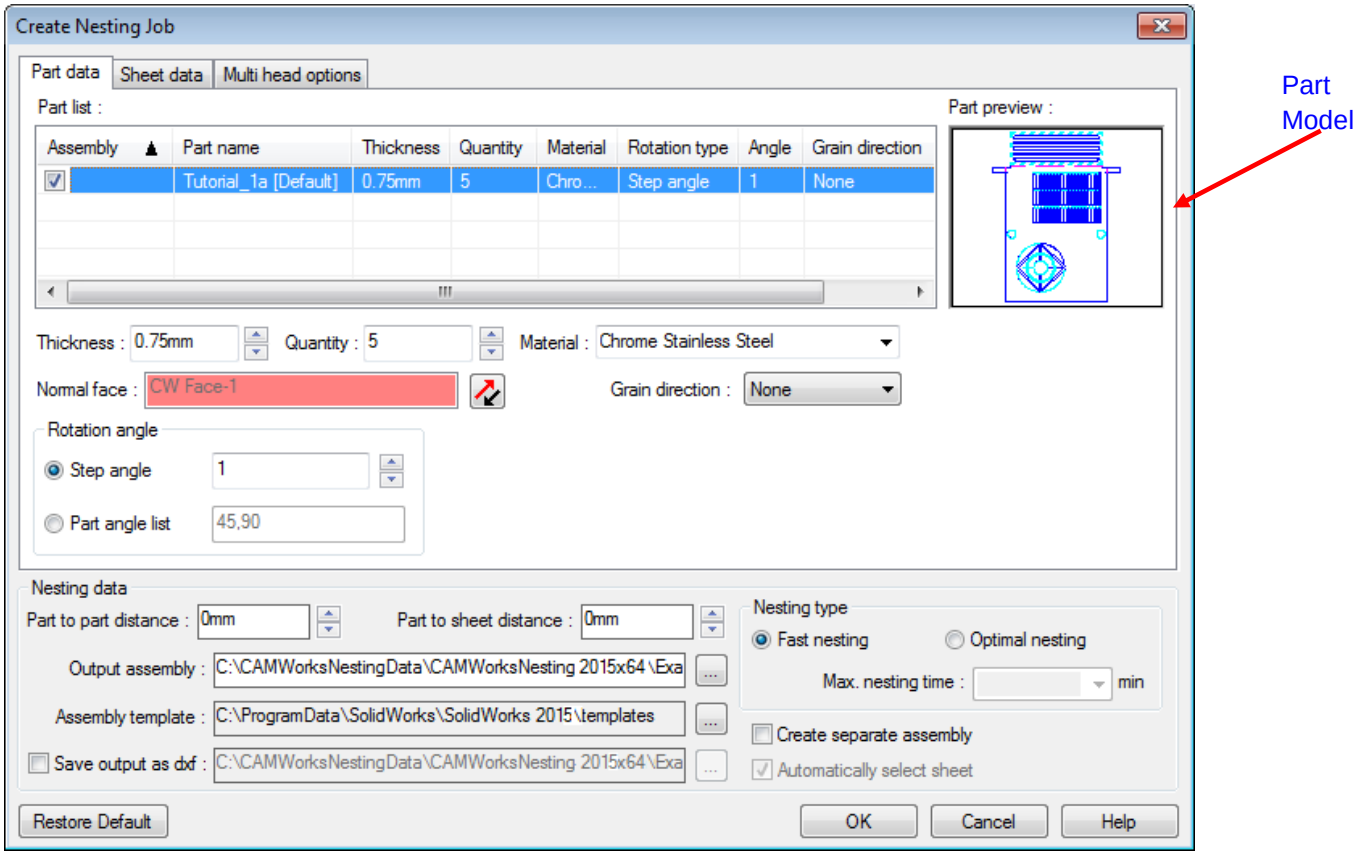


CAMWorksNesting Menu



CAMWorks Nesting Ribbon Bar

c. The *Create Nesting Job* dialog box is displayed.



The CAMWorks Nesting 'Create Nesting Job' dialog box

5. For **Assembly Nesting**:
   a. Model or open a sheet metal/solid part/solid assembly in SOLIDWORKS.
   b. Select *Create Nesting Job* from the CAMWorksNesting menu bar.
   c. The *Create Nesting Job* dialog box is displayed.
6. For **Multi-part Nesting → Use *Nest by Folder***
   a. If an assembly model is not available already and if the parts to be nested are available in a folder then select *Nest by Folder* from the CAMWorksNesting menu.
   b. Browse to the folder containing the parts to be nested. Click *OK*.
   c. The parts to be nested will be displayed in the SOLIDWORKS Graphical User Interface.
   d. The *Create Nesting Job* dialog box is displayed. All the parts are listed under the *Part Data* tab of this dialog box along with the part parameters.
7. In all the three cases viz. Single Part Nesting, Assembly Nesting and Multi-part Nesting, if the part or assembly contains an imported sheet metal part with bends, then the ***Unfold Imported Bodies*** dialog box will be displayed prior to the *Create Nesting Job* dialog box. Use this dialog box to select the imported sheet metal parts to be unfolded and to assign parameters related to unfolding. Once you make the required selection and assign parameters, click *OK*. The *Create Nesting Job* dialog box will be displayed.
8. In the *Create Nesting Job* dialog box, under the *Part Data* tab, modify or assign the Part controller parameters for the part(s) as required. These parameters include thickness, material, grain direction, quantity, Step angle, Normal Face Selection as required. These parameters are discussed in detail in the section **Part Parameters**.
9. Under the *Sheet data* tab, select the required sheet size(s). Modify or assign the sheet parameters such as sheet name, sheet thickness, sheet material, sheet quantity, grain direction, sheet length and width. These parameters are discussed in detail in the section **Sheet Parameters**.
10. If you wish to nest the part(s)/assembly using multiple tool heads, use the *Multi head options* tab to assign the associated parameters such as the machine name, number of tool heads, to be used, rail direction, tool head distance and multi-tool head nesting type. These parameters are discussed in detail in the section **Multi Head Options parameters**.
11. In the *Nesting Data* group box:
    a. Assign appropriate values to the parameters of *Part-to-Part distance* and *Part-to-Sheet distance*.

b. Use the browse button  to assign a new location for the output file rather than the default location given in the Output Assembly File field.

c. CAMWorks Nesting always saves the nested layouts generated after the execution of a nesting job in the assembly file format *(.sldasm)*. To optionally save the nested layouts in the *.dxf* format, check the *Save output as dxf* checkbox. Use the browse button  to assign the folder location where the .dxf files are to be saved.

d. *Fast Nesting* and *Optimal Nesting* indicate the two different sets of algorithms used to implement Nesting. Select the option that best suits your requirements. Time Constraint can be applied to *Optimal Nesting* if required.

   The parameters in the *Nesting Data* group box are discussed in detail in the section **Nesting Data Parameters**.

12. After all the parameters are set, click *OK* to execute the Nesting Job.

    This sets into motion the process to generate a nested layout. Generating the layout might take some time depending on the complexity of the part.

    Two files will be generated during the nesting process, namely a text file and an assembly file.

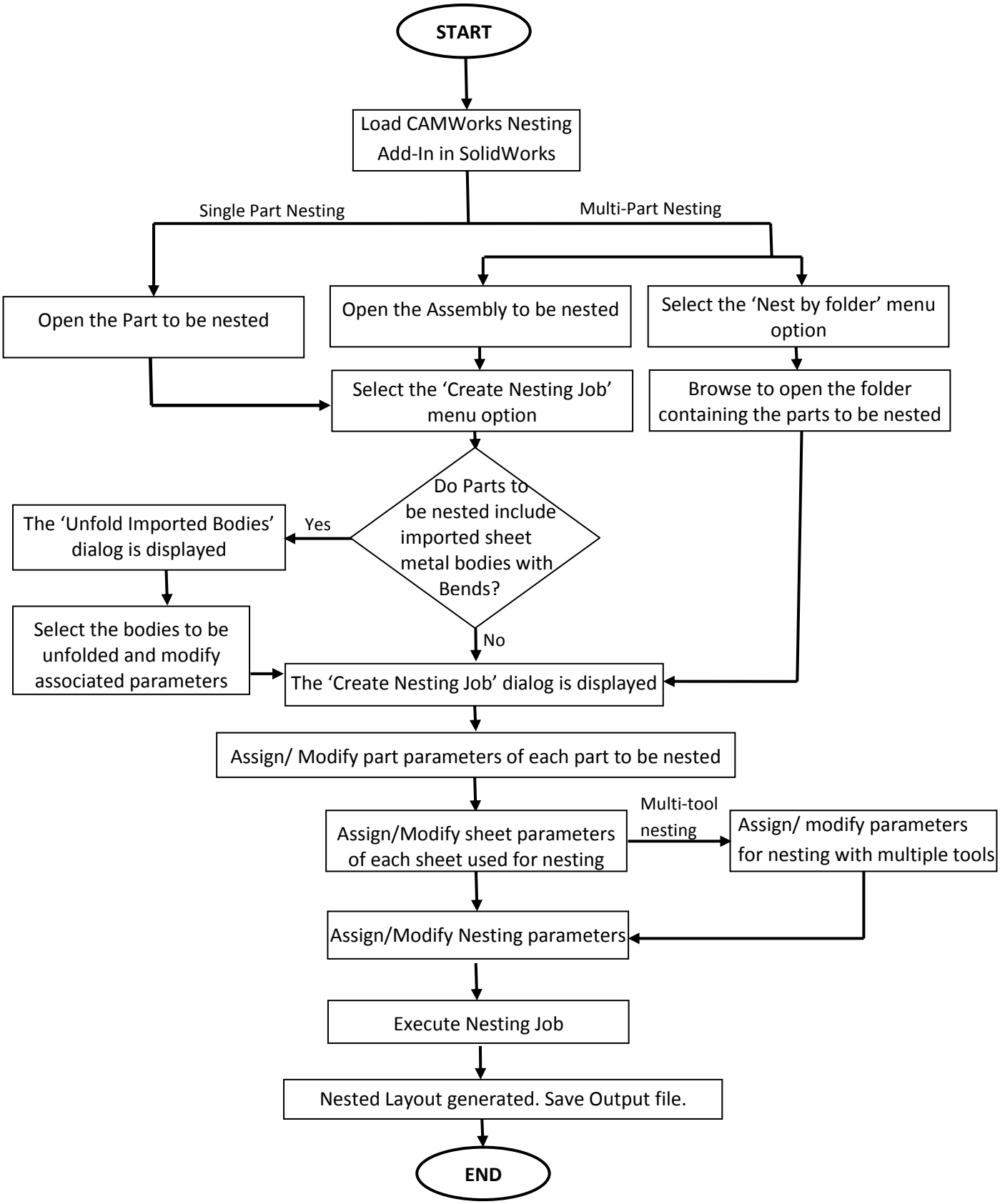
    After the Nesting process is completed, CAMWorks Nesting will display a message indicating the location of the text file containing the **summary of the Nest Results**. Click *OK* to close the message. The Text file will be displayed.

    The **Nested layout assembly** will be displayed in the Graphics area. Both these files are saved in the location indicated by *Output Assembly File* path stated in the *Apply Nesting* dialog box.

Note: **The assembly file format (.sldasm) is the standard file format in which the nested layouts are generated. If the Save output as dxf option is used, then the nested layouts will be generated in two file formats: *.sldasm* & *.dxf* and saved in the specified folder locations.**

Flowchart on Basic Procedure to Implement Nesting

A **flow chart** of the basic procedure to implement nesting using CAMWorks Nesting is given on the next page.



Steps to generate Nesting Layout in CAMWorks Nesting

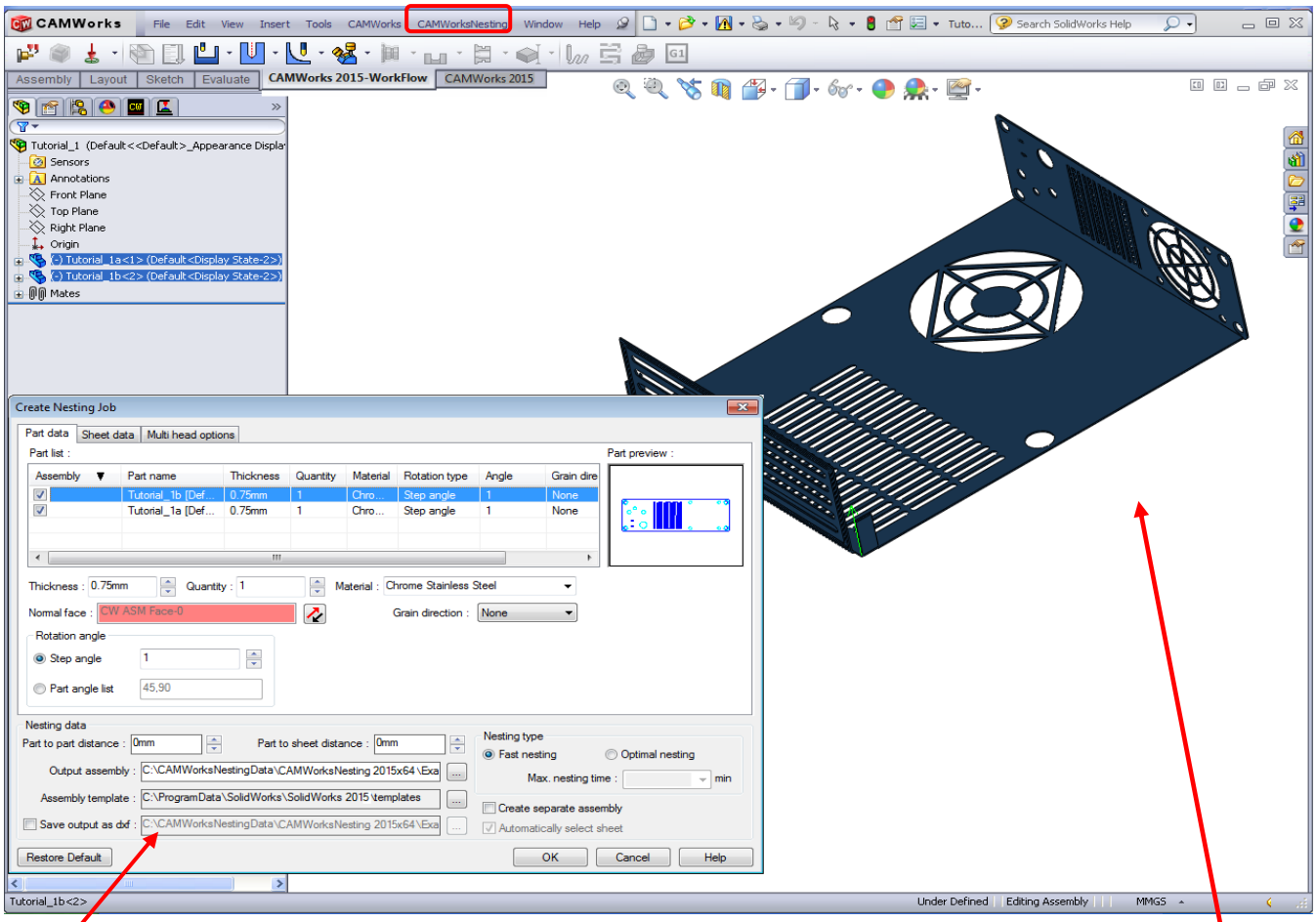
The Part Model/Assembly

In CAMWorks Nesting, your part model is a solid created with SOLIDWORKS or imported into SOLIDWORKS from another CAD system via an IGES, STEP, Parasolid, SAT or other neutral translators. A part can contain multiple bodies.
Similarly, an assembly is a group of parts created with SOLIDWORKS or imported into SOLIDWORKS from another CAD system. An assembly can contain multi-body parts. Assemblies with parts having multiple configurations are supported.

The tutorials in this manual use existing SOLIDWORKS Parts installed with CAMWorks Nesting.

For example, open the assembly ***Tutorial_1.sldasm*** located in the following folder of your CAMWorks Nesting installation folder.
Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies\Tutorial1

Select *Create Nesting Job* from the CAMWorksNesting menu.



The CAMWorks Nesting Graphical User Interface

Create Nesting Job

Part Model

Defining Part Parameters

The *Create Nesting Job* dialog box is used to set the part, sheet and nesting parameters for Single Part nesting as well as Multi-Part nesting.

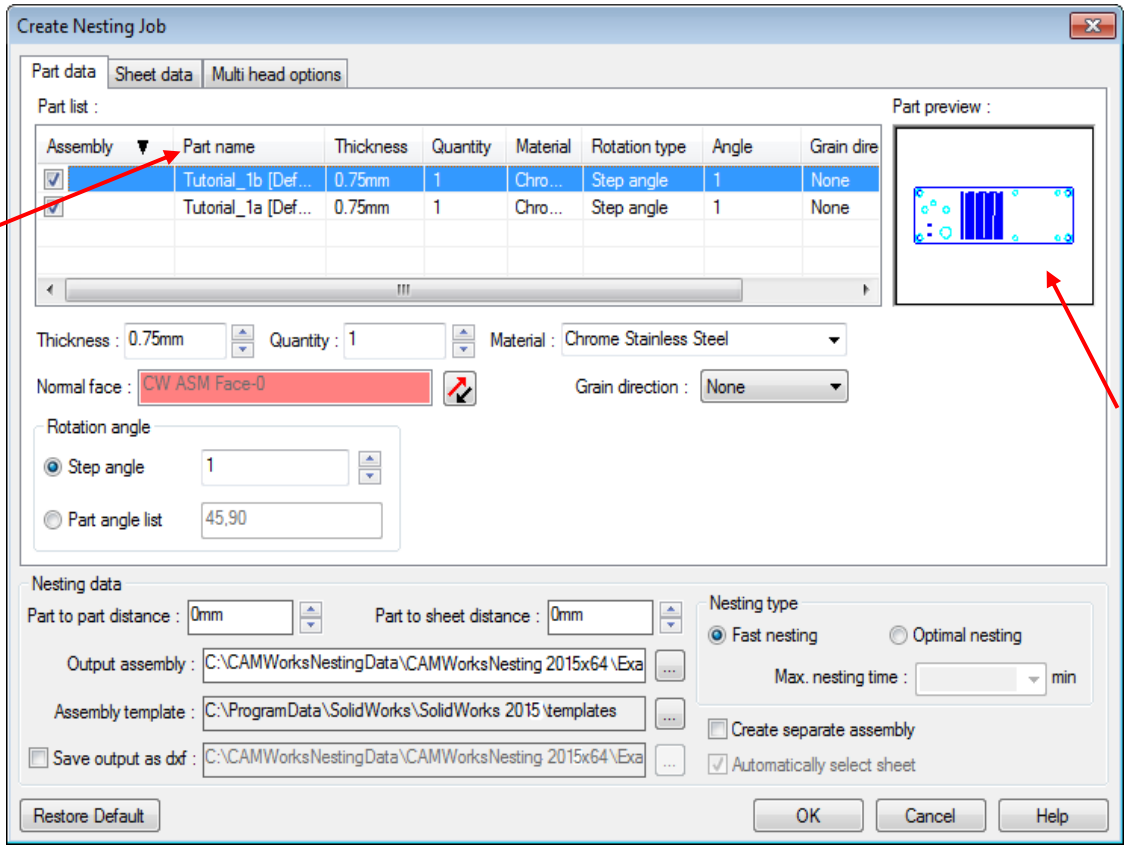
The Part data Tab

Use the *Part Data* tab of the *Create Nesting Job* dialog box to view and edit the part related parameters.

The Part Data tab is the default tab displayed when the *Create Nesting Job* dialog box is displayed.

The below data fields are available and can be edited in the *Part Data* tab:

- Part List
- Thickness
- Material
- Rotation Angle
- Grain Direction
- Normal Face Selection
- Quantity



Part name and parameters of the part to be nested are displayed in Part List

The preview of the part to be nested is displayed here

The Part Data Tab

Part List

The part parameters of Part name, Thickness, Quantity, Material, Rotation Type, Rotation Angle, Grain direction are displayed in the **Part List**. The Part

parameters of **thickness** and **material** are extracted from the solid model part and displayed in the **Part List.**

All the above parameters except the Part name can be edited directly in the Part List. Alternatively, use the various Part parameter fields given below the Part List grid to edit the parameters.

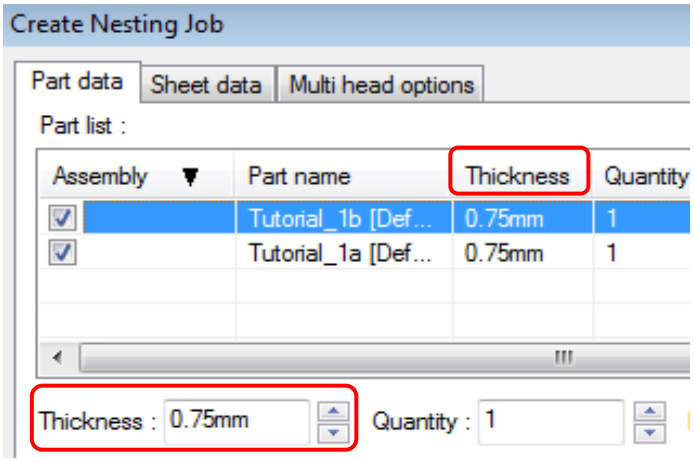
To edit the part parameter fields, highlight the part to be edited in the part list and double-click on the required field. Edit the values as required.

Thickness

CAMWorks Nesting extracts the part parameter of Thickness from the Solid Part and displays it in the Thickness field as default thickness for the part.

To assign a desired thickness, enter the thickness value in the Thickness field.

The Thickness field in the Part List as well as Thickness field below the Part List grid can be used to edit the value.



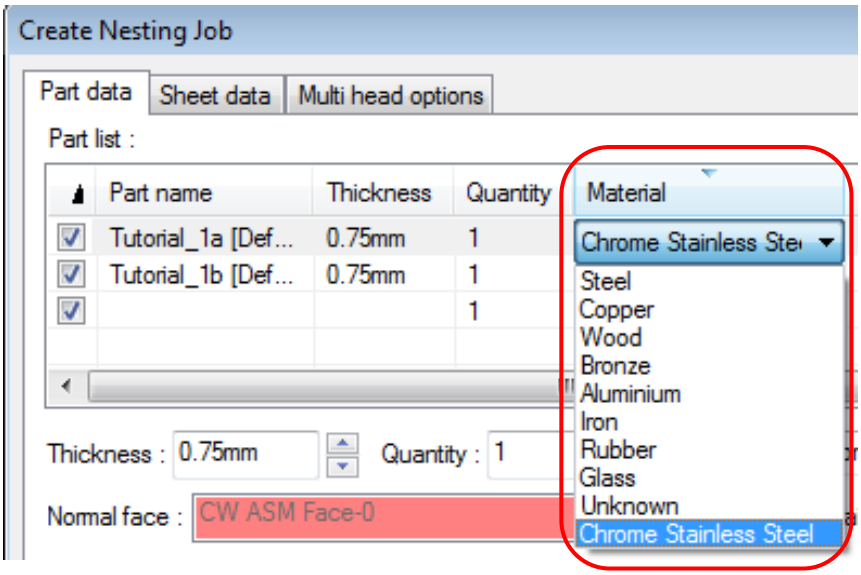
Assigning thickness value to a part

Material

CAMWorks Nesting extracts Material related information from the solid part in the SOLIDWORKS environment and displays it in the 'Material' field as default material for the part.

If the material value is not extracted from the 3D model then, CAMWorks Nesting assigns a default value. This default value will be the first material listed in the material dropdown list.

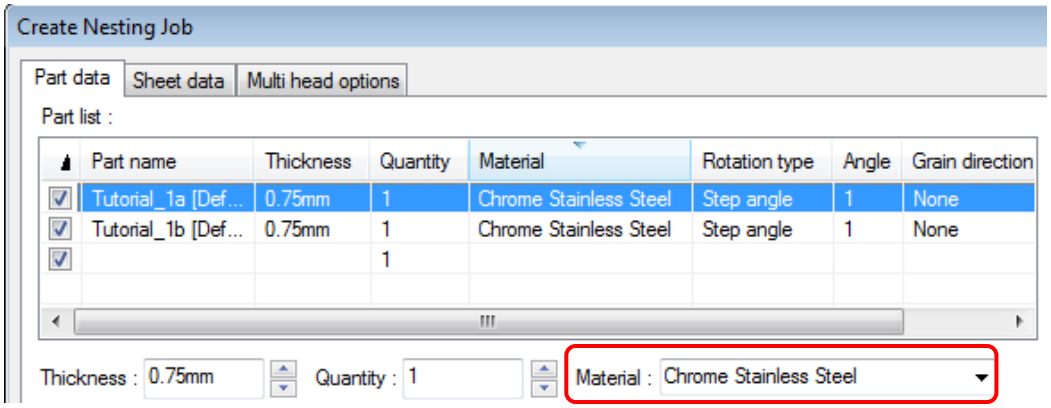
To assign a material from the material dropdown list, select the desired material



Assigning Material from dropdown list in Material Column

from the dropdown list in the material column of the Part list. Alternatively, use the Material dropdown list given below the Part List grid.

A user-defined material (a material which is not part of the material dropdown list) can also be assigned to a part. However, such a user-defined material cannot be assigned to a part using the Part List.



Enter user-defined material name in the field given below the Part list. It will then update the Material Column of Part List

To assign a user defined material, following are the steps:

a) Select the Part (for which material is to be changed) in the Part List.

b) Enter the Material name into the Material combo box given below the Part List.

c) Shift the focus from this field by pressing the tab button. Observe that the new material assigned is reflected the Part List.

Note: **You must customize the material dropdown list to populate it with the materials used at your facility. For details, read: Viewing/ Editing the Material dropdown list.**

Rotation Angle

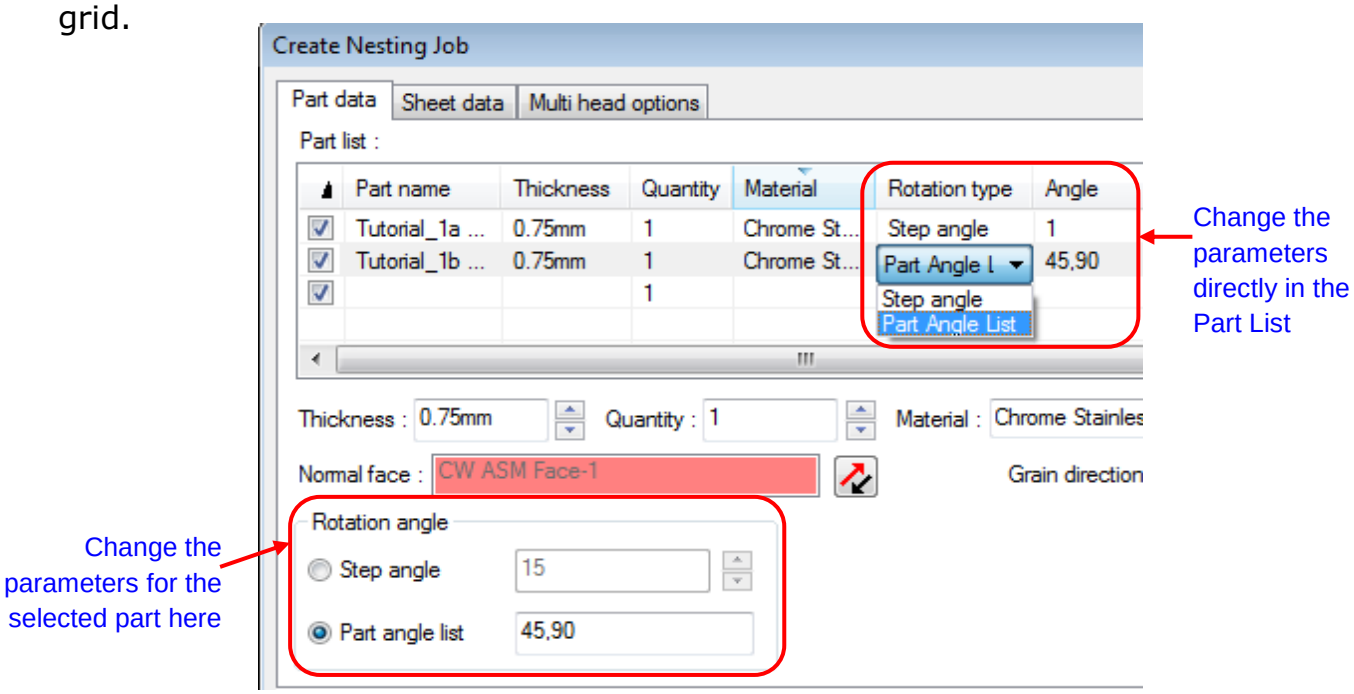
CAMWorks Nesting provides two options for applying the rotation control for a part:

Step Angle: This is the angle that specifies the step in which the part is tried for nesting.
For example, if the Step angle provided for the part is 90 degrees, then that part will be tried in 90, 180, 270 and 360 degrees. The default step angle is 1^0.
If step angle is 15^0, then nesting of the part will be tried in 15, 30, 45, 60, 75, 90, 105, 120, 135, 150, 165 …… up to 360 degrees.

Angle List: This is the second way in which the rotation control can be applied on a part. The set of angles need to be input in a list. CAMWorks Nesting will then try nesting the parts only for the specified set of angles. To specify an Angle List in

the Create Nesting Job dialog box, separate the angles by a comma. For example, to specify angles of 10, 20, 30, 60 and 90 degrees, you must enter '10, 20, 30, 60, 90' in the Part Angle List field.

The Rotation type and the subsequent Part angle can be set in the Part List. Alternatively, you can highlight the part(s) in Part List for which this parameter is to be changed. Then change the Rotation Type and part angle for the highlighted part in the **Rotation Angle** group box below the Part list grid.



Assigning Rotation angle to the part

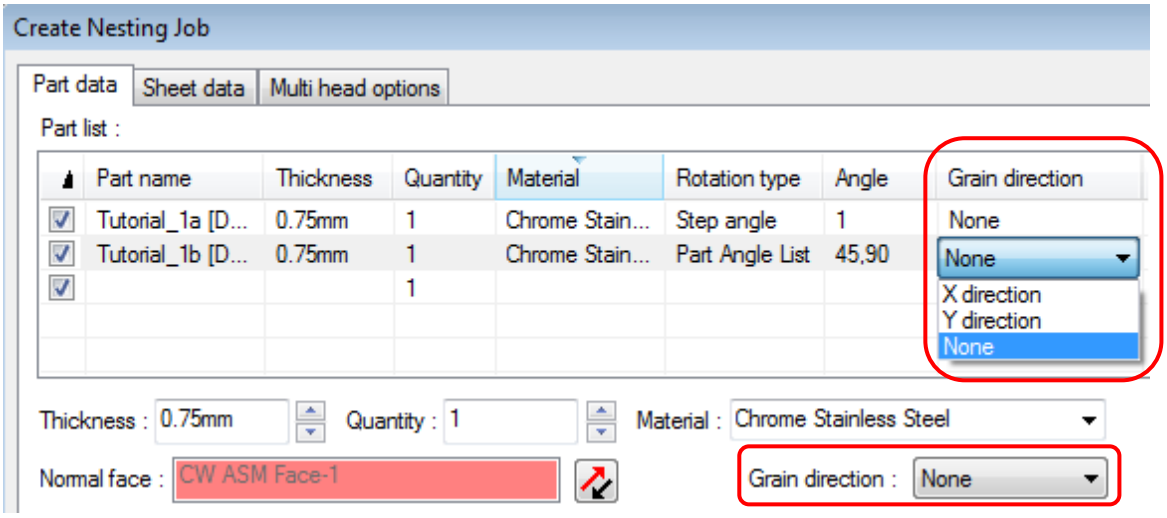
Grain Direction

To ensure accuracy and avoid defects during the subsequent mechanical operations like bending, it is necessary to cut critical parts, such that they have pre-defined and proper grain direction.

The Grain direction field is a drop down list from which you must choose any one option. The options are:

- X direction
- Y direction
- None (default option for both part and sheet)

The Grain Direction for a part can be set directly in the *Part list*. Alternatively, you can highlight the part(s) in *Part list* for which this parameter is to be changed. Then change the *Grain direction* for the highlighted part using the Grain direction dropdown list below the *Part list* grid.



Assigning the grain direction for the part to be nested

Normal Face

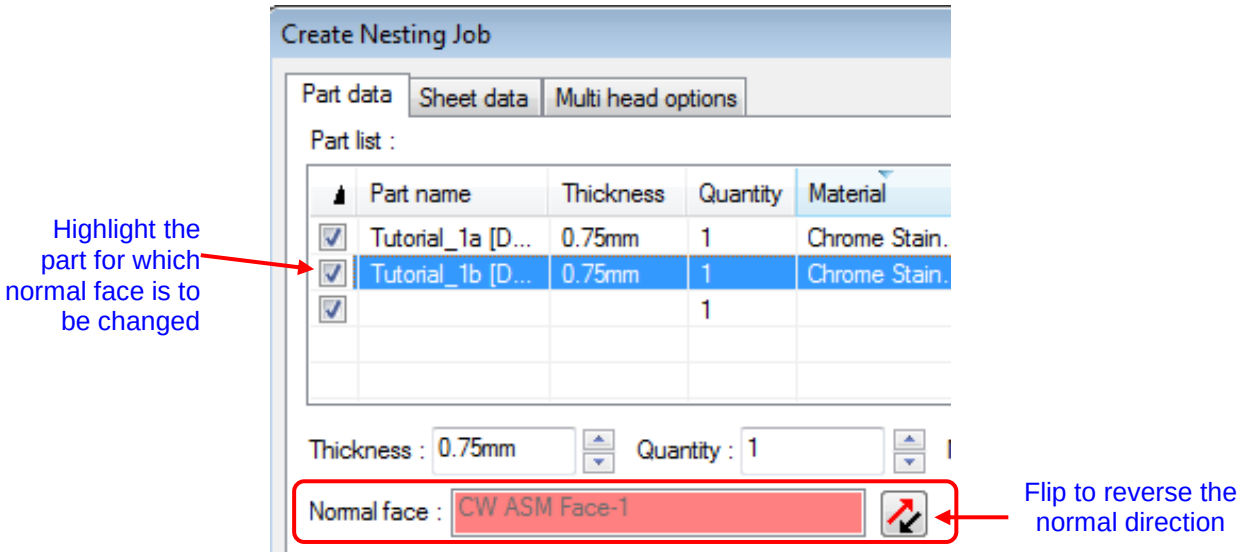
The Normal Face selection is used to select the part face to be used as a normal plane for generating silhouette profile of the part to be nested.

By default, CAMWorks Nesting uses the part face with the maximum number of features as a normal plane to generate a silhouette profile. However, for certain solid/imported parts, a need may arise to select another face of the part for various reasons such as ease of machining, single setup machining etc. For such situations, CAMWorks Nesting provides the option to manually define the Normal face.

The Normal Face selection cannot be executed in the Part List grid.

To change the Normal face direction of a part:

- Highlight the required part in the Part List grid.
- The sheet metal/solid part to be nested is highlighted in the Graphical User Interface of SOLIDWORKS. In this interface, click on the face of the part that you wish to assign as the normal face.
- The Normal face field will now list the selected face.
- Click  to reverse the normal direction.



Normal Face Selection

Quantity

The Part *Quantity* field indicates the number of instances of the part to be nested. CAMWorks Nesting assigns a default quantity to all parts listed in the Part List.

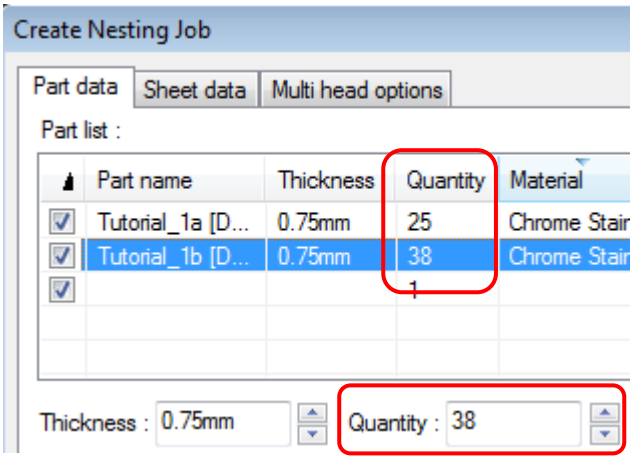
Use the *Quantity* field to assign the number of instances of the part to be nested.

The quantity for a part can be set directly in the Part List.

Alternatively, you can highlight the part(s) in Part List for which this parameter is to be changed. Then change the quantity for the highlighted part using the Quantity field below the Part list grid. You can use spin control to increase or decrease the Quantity value. Spin control increases the value in steps of +1 and decreases it in steps of -1.

Note:

For assemblies, default quantity assigned for a part is equal to the number of instances of the part in the assembly. For single part nesting, the default value assigned is based on the value defined in the DefaultValues.ini file. You can edit the default Quantity to be assigned for parts. For details, read: Defining default Part quantity.



Assigning Quantity to the parts to be nested

Note: **If the feature for assigning assembly quantities is enabled, then the Quantity column will also display values in the Quantity column for the assembly to be nested as well as its constituent sub-assemblies, if any. Editing the quantity of the assembly or its constituent sub-assemblies updates the quantity of its constituent parts automatically. For more detail, refer: Tutorial 9**

Defining Sheet Parameters

The *Create Nesting Job* dialog box is used to set the part, sheet and nesting parameters for Single Part nesting as well as Multi-Part nesting.

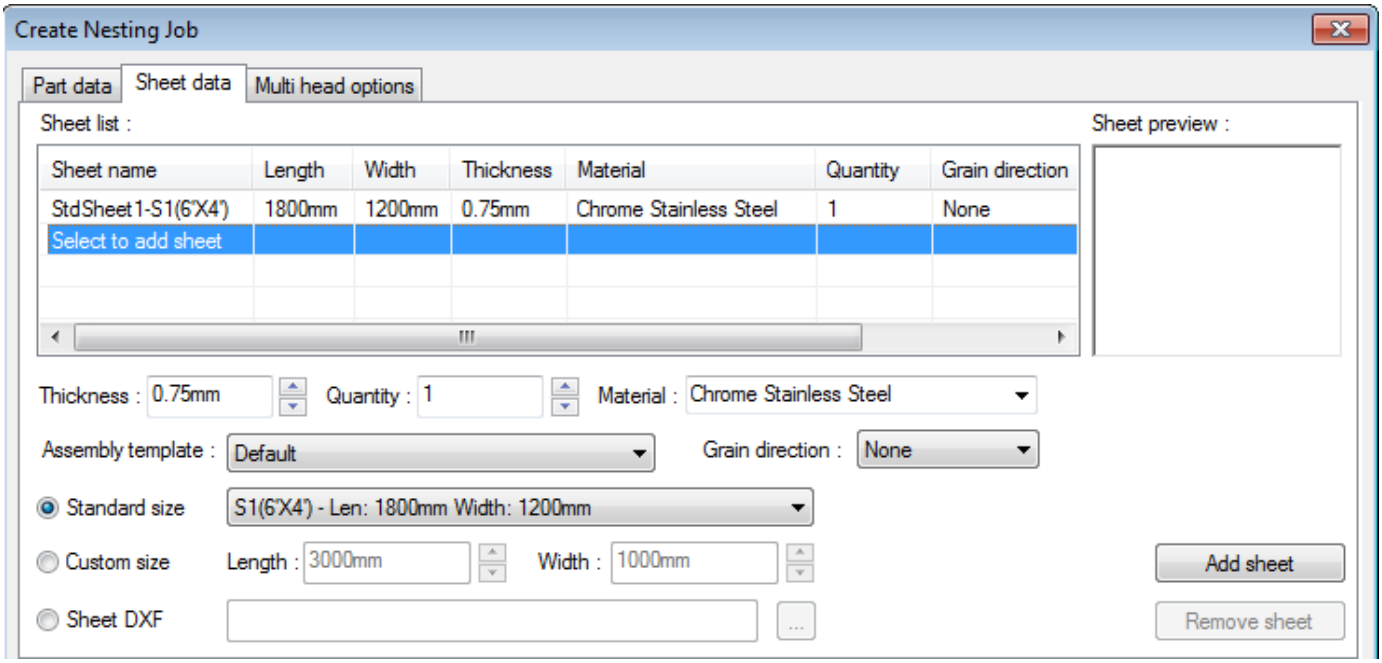
The Sheet data Tab

Use the *Sheet Data* tab of the *Create Nesting Job* dialog box to add sheet(s) and set the parameters for sheet(s) in which parts will be nested.

The *Part Data* tab is the default tab displayed when the *Create Nesting Job* dialog box is displayed. Click on the *Sheet Data* tab to view Sheet related data fields.

The below data fields are available and can be edited in the *Sheet Data* tab:

- Sheet List
- Thickness
- Material
- Quantity
- Grain Direction
- Sheet Size
- Length
- Width



Sheet Data tab of 'Create Nesting Job' dialog box

Sheet List

The Sheet List is populated by adding sheets using the *Add Sheet* button. Parts defined in the *Part Data* tab will be nested only on the sheet(s) listed in the *Sheet List*.

The sheet parameters of Sheet name, Length, Width, Thickness, Quantity, Material and Grain direction are displayed in the **Sheet List**.

Of these parameters, *Thickness*, *Quantity*, *Material* and *Grain direction* parameters can be edited directly in the *Sheet List* grid after insertion. This is true for all sheet sizes. The parameters of Sheet Name, Length and Width can be edited only for Sheet of type *Custom Size*.

To edit the sheet parameter fields, highlight the sheet to be edited in the *Sheet List* and double-click on the required field to edit the values.

Thickness

Some intelligence is added in CAMWorks Nesting such that it ensures all sheets with relevant **materials and thicknesses** are available for nesting all the parts in the assembly. CAMWorks Nesting automatically extracts the thickness and material of the first part in the part list and assigns these as the default value of the first sheet. CAMWorks Nesting automatically checks for different material and thickness if any in the part list and assigns these as the default value of the second sheet and so on till all required sheets with relevant material and thickness are added.

For a sheet inserted in the *Sheet List*, the thickness field displays the sheet's thickness.

Use the thickness field given below the sheet list to set the thickness value before adding the sheet to the *Sheet List*. Once the sheet is added to the *Sheet List*, the thickness value can be edited directly within the *Sheet List* in the respective field.

Material

For a sheet inserted in the Sheet list grid, the *Material* field displays the material the sheet is made of.

Use the Material field given below the sheet list to set the Material type before adding the sheet to the *Sheet List*. Once the sheet is added to the *Sheet List*, the material type can be edited directly within the *Sheet List* in the respective field.

Quantity

The *Quantity* field indicates the number of sheets available. Use the *Quantity* field given below the sheet list to set the *Quantity* value before adding the sheet to the Sheet List. Once the sheet is added to the *Sheet List*, the *Quantity* field can be edited directly within the *Sheet List* in the respective field.
The default quantity assigned is based on the value defined in the DefaultValues.ini file. You can edit the default *Quantity* to be assigned for sheets. For details, read: **Defining default Sheet Quantity**.

Grain Direction

Grain direction can be set for a sheet just like it is set for parts. This field is a drop down list from which you must choose an option. The options are:

- X direction
- Y direction
- None *(default option for both part and sheet)*

Use the *Grain Direction* field given below the sheet list to set the G*rain Direction* before adding the sheet to the *Sheet List*. Once the sheet is added to the *Sheet List*, the *Grain Direction* field can be edited directly within the *Sheet List* in the respective field.

Assigning Grain directions

The Grain Direction which you can assign to a particular sheet is dependent on the Grain Direction of the Parts which will be nested within that sheet. The allowed relationship between the *Grain Direction* of the part(s) and sheet is given in the following table:

Grain Direction of Part	Allowed Grain Direction for Sheet	Description
X	X or Y but not None	If a part has grain direction “X”, then at least one of its corresponding sheets should have either “X” or “Y” but not “None” as its grain.
Y	X or Y but not None	If a part has grain direction “Y”, then at least one of its corresponding sheets should have either “X” or “Y” but not “None” as its grain.
None	X or Y or None	If a part has grain direction “None”, then the corresponding sheets can have either “X” or “Y” or “None” as its grain direction.

Add Sheet & Remove Sheet buttons

Use the *Add Sheet* button to add a sheet to the Sheet List after setting its parameters.

To remove a sheet from the Sheet List, select the sheet to be deleted in the Sheet List and click *Remove Sheet*.

Sheet Size

The Sheet List grid will initially be empty when you click on the *Sheet Data* tab of the *Create Nesting Job* dialog box. The size of the sheet in which the part(s) will be nested needs to be defined and then added to the *Sheet List*. CAMWorks Nesting provides three options with respect to sheet size:

Standard Size: The *Standard Size* dropdown box lists all the standard sheet sizes listed in the StandardSheets.ini file. This option is best exercised if you have defined the standard sheet sizes used at your facility in the **StandardSheets-INCH.ini** or **StandardSheets-MM.ini** file.

Custom Size: The *Custom Size* option is best used when adding a non-standard size rectangular sheet or adding a custom sheet size.
The default dimensions (length and breadth) for the Custom sheet can be defined in the **DefaultValues.ini** file. These default values will be displayed in the *Sheet Data* tab when you select the *Custom Size* option to execute a new nesting job.

Sheet DXF: Only rectangular sheets can be defined using *Standard Size* and *Custom Size*. The *Sheet DXF* option is best used when you want to use a non- rectangular sheet or remnant sheet. In order to nest parts using such a non-rectangular or

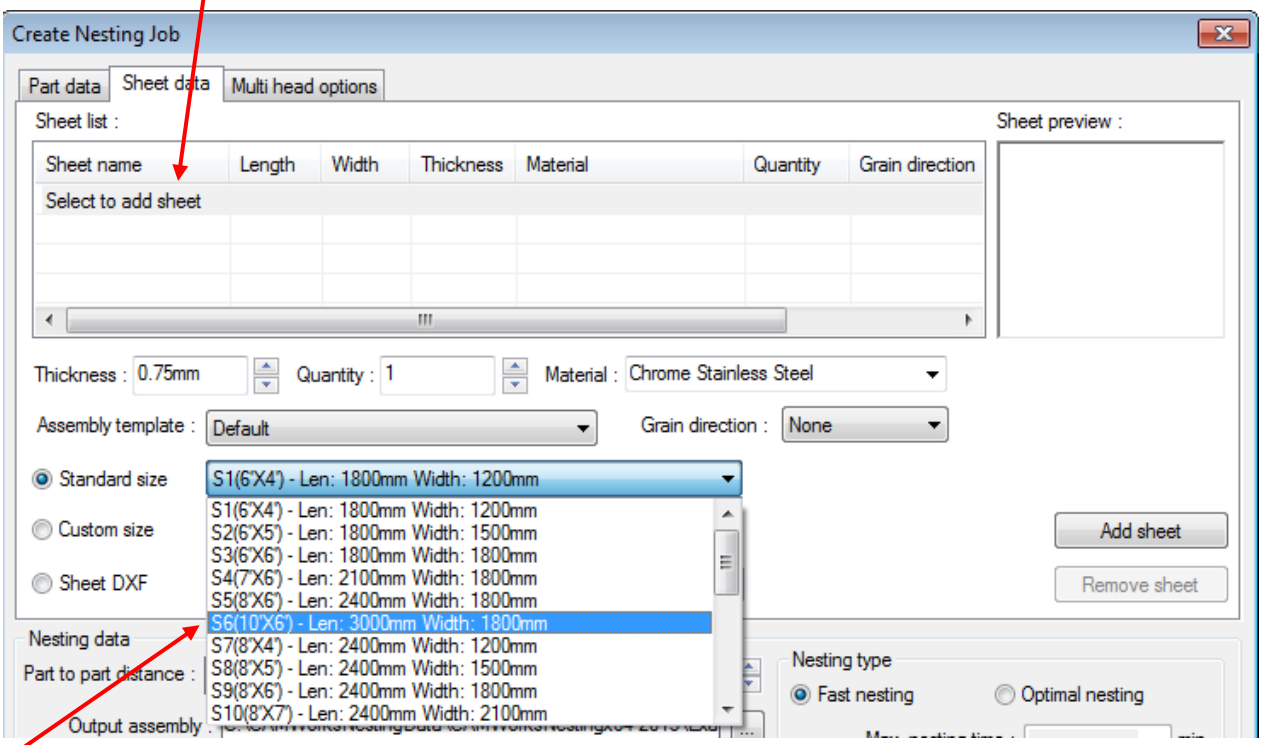
remnant sheet, the sheet should be saved in CAD graphic image file format called *Drawing Exchange format* (.dxf).

Defining Standard Sheet

To add a standard size sheet, following are the steps:

- In the *Sheet data* tab, the row indicating *Select to add sheet* in the Sheet list would have been selected by default, if it is not selected then click on the row indicating *Select to add sheet* in the Sheet list.
- To define the sheet size, select the option *Standard Size*.
- In the *Standard Size* dropdown list, select the required sheet size.
- Set the parameters of thickness, quantity, material and grain direction as required.
- Click on the *Add Sheet* button.
- The sheet is now added to the sheet list.

Standard Sheet is added to Sheet List after executing



Select the Standard size from the dropdown list.

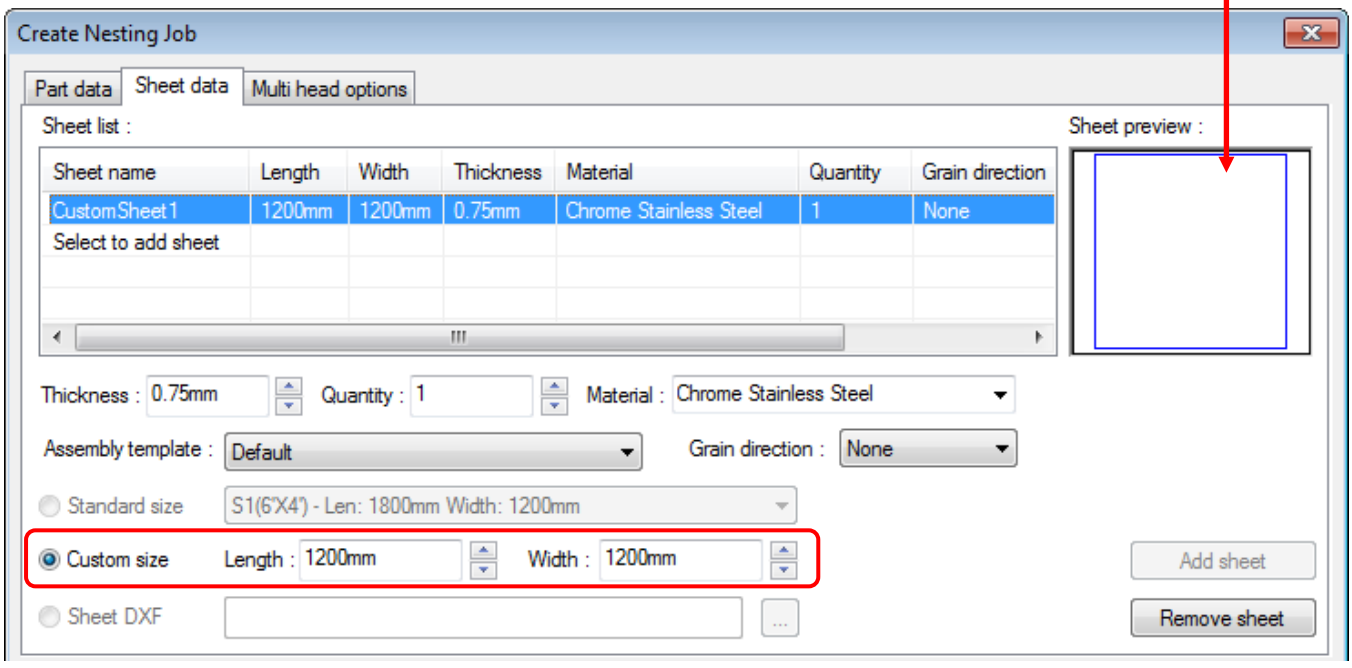
Adding Standard Size Sheet to Sheet List

Defining Custom Size Sheet

To define a *Custom size* sheet, following are the steps:

- In the *Sheet data* tab, the row indicating *Select to add sheet* in the Sheet list would have been selected by default, if it is not selected then click on the row indicating *Select to add sheet* in the Sheet list.
- In the Sheet data tab, select the option *Custom Size*.
- The **Length and Width fields** are activated on selecting this option. Default values are displayed in these fields. (These values are defined in the DefaultValues.ini file.) Edit the *Length* and *Width* fields to assign the required values. You can use spin control to increase or decrease the *Length* and *Width* values.
- Set *Thickness*, *Material* and *Grain Direction* values.
- Set the *Quantity* of the Sheet.
- Click on the *Add Sheet* button.
- The sheet is now added to the sheet list.

The preview of the sheet highlighted in the Sheet List is displayed here



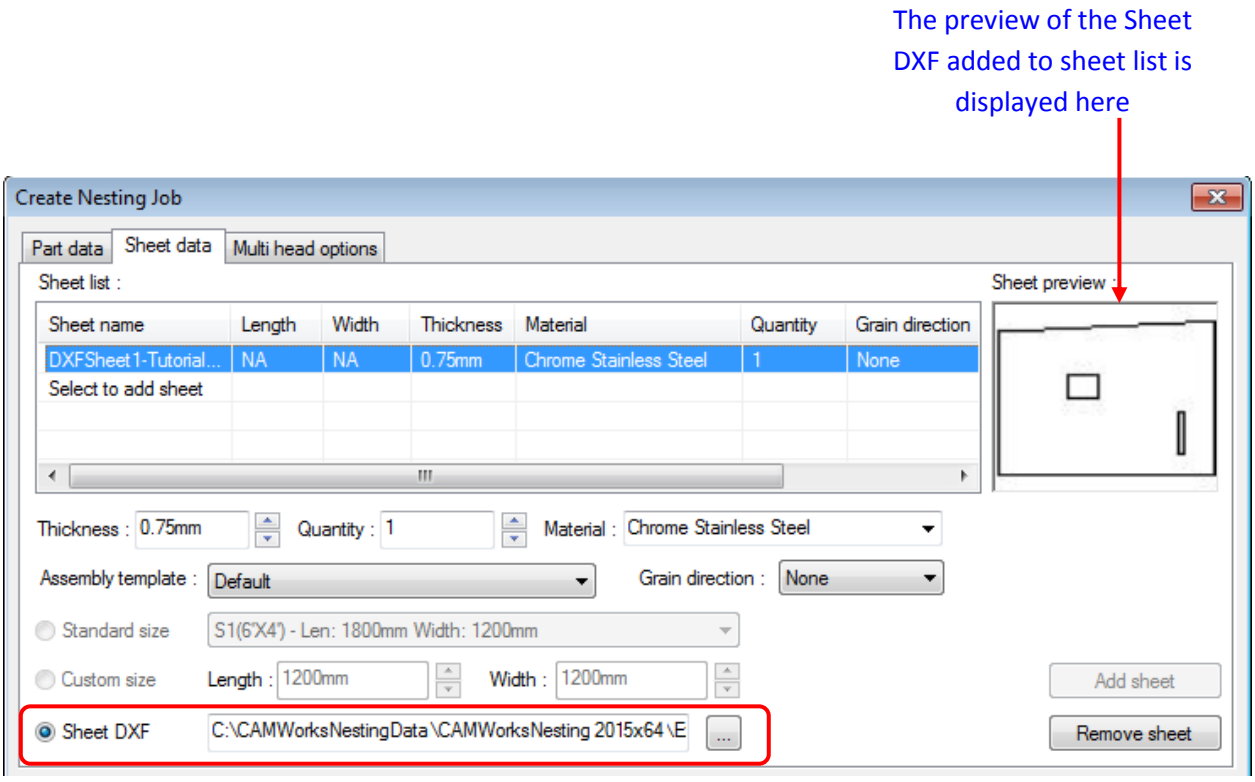
Adding Custom Size Sheet to Sheet List

Defining Sheet DXF

To use a sheet saved in DXF *(Drawing Exchange Format)* file format,

- In the *Sheet data* tab, the row indicating *Select to add sheet* in the Sheet list would have been selected by default, if it is not selected then click on the row indicating *Select to add sheet* in the Sheet list.
- In the *Sheet data* tab, select the option *Sheet DXF*.

- Click on the ... button to browse to the folder location where the .DXF file is located.
- Set *Thickness*, *Material* and *Grain Direction* values.
- Click on the *Add Sheet* button.
- The Sheet saved in .dxf format will be added to the sheet list.
- A thumbnail view of the shape of the sheet will be displayed in the *Sheet Preview*.



Adding a Sheet stored in .dxf file format

Note: **After a sheet is added to the Sheet list, its parameters can be edited either in Sheet List grid or in the respective fields below the Sheet list grid similar to part data editing.**

Defining Multi Head Options Parameters

The *Create Nesting Job* dialog box is used to set the part, sheet and nesting parameters for Single Part nesting as well as Multi-Part nesting.

For flame cutting applications, CAMWorks Nesting provides an optional functionality known as *Multiple Tool Head Nesting*. This functionality allows you to nest two or more identical layouts simultaneously using multiple tool heads. Machines which support nesting using multiple tool heads are known as *Multi tool head machines*.

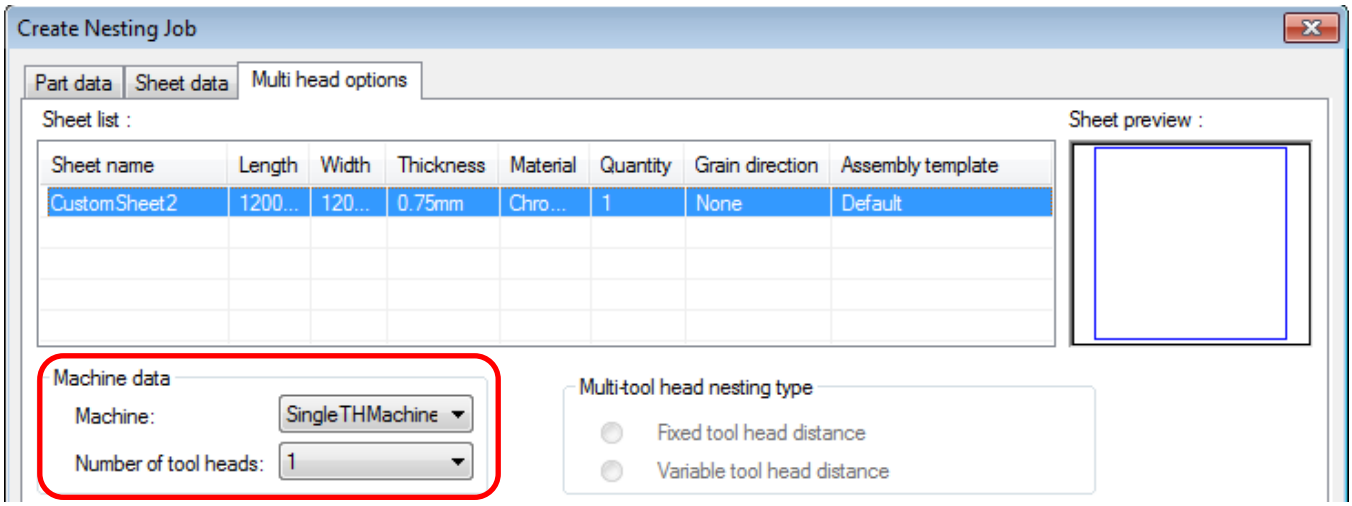
The tab named *Multi head options* in the *Create Nesting Job* dialog box allows you to define/edit parameters related to nesting with multiple tool heads.

How the functionality of nesting with multiple tool heads works

When nesting layout(s) are to be generated using multiple tool heads, CAMWorks Nesting will first attempt to nest the parts using the user-specified number of tool heads. Suppose this tool heads number is 'n'. If a best-fit layout is achievable, CAMWorks Nesting will generate 'n' identical nesting layouts on the sheet.
If a best-fit nesting layout is not achievable with this number, then CAMWorks Nesting will try to nest using 'n-1' number of tool heads. If this number too fails, then it will try to nest using 'n-2' tool heads and so on until finally nesting with a single tool head.

The Default Settings in the Multi Head Options tab

Click on the Multi Head Options tab of the *Create Nesting Job* dialog box.



The Multi Head Options tab (as typically seen with default settings)

Observe the Machine Data group box. The default machine displayed in the Machine field is SingleTHMachine. The *Number of tool heads* for this machine is '1'.

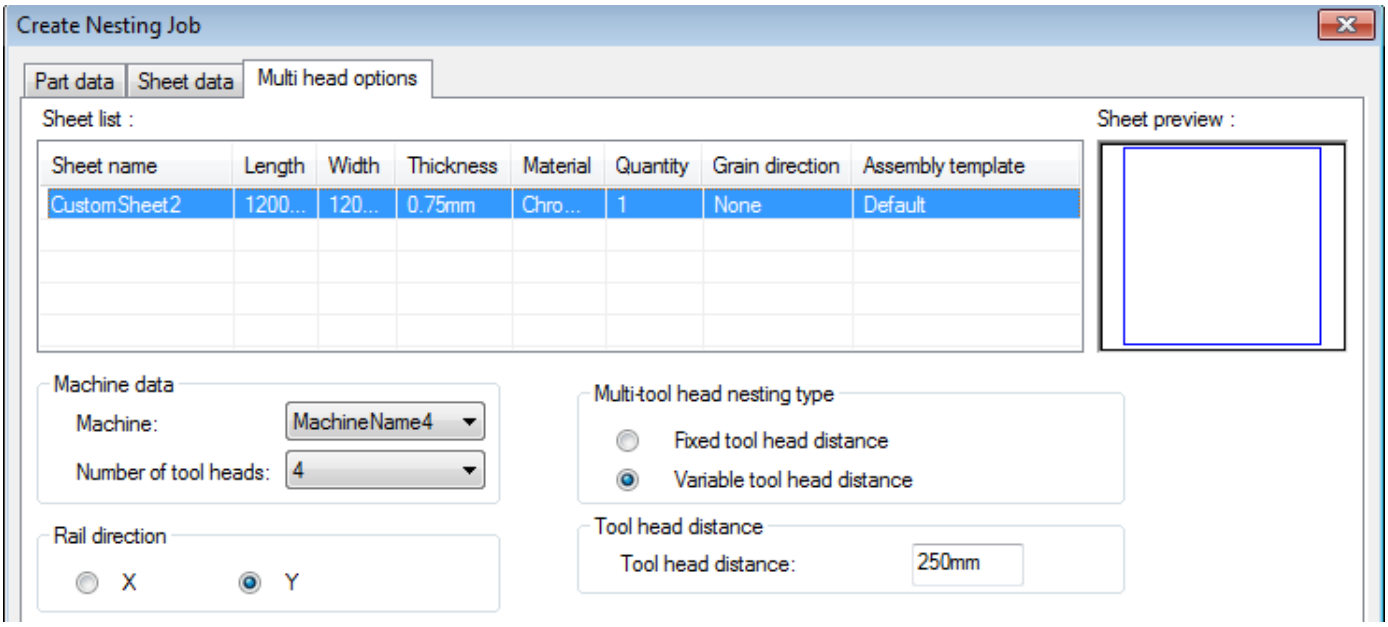
SingleTHMachine (Single Tool Head Machine)

SingleTHMachine is representative of machines with a single tool head. Such machines are usually used for the nesting process. When this machine is selected in the Multi Head Options tab, the functionality of nesting with multiple tool heads will be disabled for the current nesting job. All the parameters fields related to Multiple Tool Head Nesting in the tab are disabled, indicating that Multiple Tool Head Nesting functionality is disabled.

Enabling Multi Tool Head Machining

To generate nested layouts using a machine with multiple tool heads instead of the default Single Tool Head machine, you need to select a machine with multiple tool heads in the *Machine* field of the *Multi Head Options* tab.

For nesting with Multiple Tool Heads, select a machine other than *SingleTHMachine* in the *Machine* dropdown list. The number of tool heads possessed by the selected machine is listed in the *Number of tool heads* field. If the machine has two or more tool heads, nesting with Multiple Tool Heads will be enabled for the current nesting job.

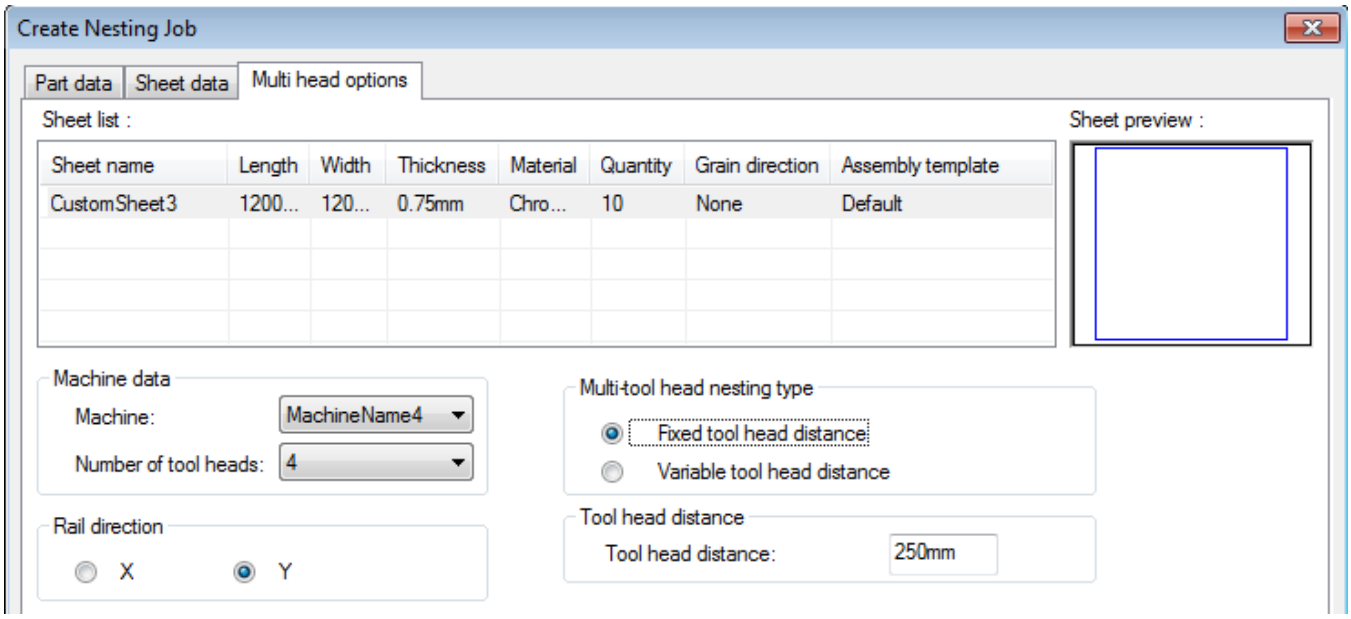


The Multi Head Options tab (as typically displayed when *Multiple Tool Head Nesting* is active)

Parameters and Data fields in the Multi Head Options tab

This tab provides an interactive interface to view/edit the parameters related to nesting with multiple tool heads. The following data fields are available in the *Multi head options* tab:

- Sheet List
- Machine name
- Number of Tool Heads
- Multi-tool nesting type option
- Rail Direction
- Tool head distance



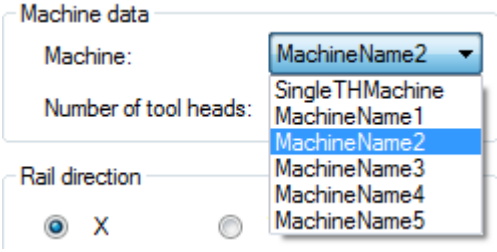
Multi head options tab in the Create Nesting Job dialog box

Sheet List

The Sheet List is populated by the sheets listed in the *Sheet data* tab of the *Create Nesting Job* dialog box. Unlike the Sheet List grid in the Sheet Data tab, this grid is read-only. For nesting with multiple tool heads, a separate set of parameters needs to be defined for each individual sheet listed in the *Sheet List*.
To define these parameters, highlight the desired sheet in the Sheet List. The default values of these parameters will be displayed below the *Sheet List* grid in their corresponding fields. Proceed to edit the parameters as desired.

Machine (machine name)

The default machine displayed in the *Machine* field is the ***SingleTHMachine***. When this machine (representative of Single Tool Head Machines) is selected, nesting with multiple tool heads is disabled. All the other parameter fields in the *Multi Head Options* tab which are related to Multiple Tool Head Nesting will be disabled.



Machine data dropdown list

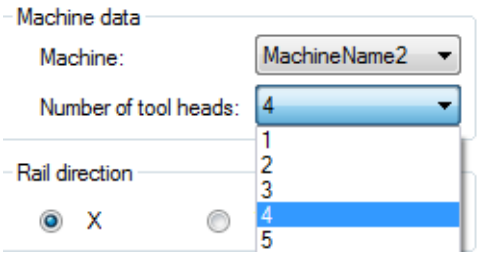
All the other machines listed in the *Machine* dropdown list support Nesting with Multiple Tool Heads. To enable nesting with multiple tool heads, select the desired machine (other than *SingleTHMachine*) from the dropdown list.

When such a machine is selected, the default parameter values associated with that machine are displayed in the *Multi head options* tab. These default parameter values for each machine are defined in the Machine.ini file.

The *Machine* dropdown list is populated with the Machine names listed in the Machine.ini file. Before executing a nesting job using multiple tool heads,

ensure that you customize the Machine.ini file to suit your nesting job requirements.

Number of Tool Heads

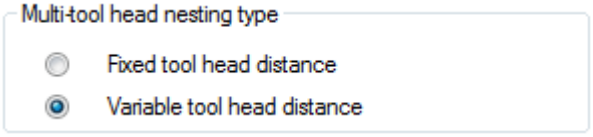


Number of tool heads dropdown list

For every machine listed in the *Machine* dropdown list, the maximum number of permissible tool heads is defined in the Machine.ini file. When you select a particular machine from the *Machine* dropdown list, the maximum permissible number of tool heads for that machine is displayed in the *Number of tool heads* dropdown list.

Based on your nesting requirements, you have the option of choosing any number ranging from 1 to this maximum number from the *Number of tool heads* dropdown list.

Multi-tool head nesting type



Selection the multi-tool head nesting type option

You can choose any one of the following *Multi-tool head nesting type* option:

a. **Fixed tool head distance**: When this option is chosen, the distance between the tool heads is fixed to the minimum tool head distance.

b. **Variable tool head distance**: When this option is chosen, the distance between tool heads can vary but will be greater than the minimum tool head distance.

When you select a particular machine from the *Machine* dropdown list, the default *Multi-tool head nesting type* option for that machine is displayed in the *Multi Head Options* tab. This default option is assigned in the Machine.ini file. You can change this default option to suit your nesting job requirements.

Rail direction

Rail direction is defined as the direction the master tool head follows while cutting. It can be either horizontal (X) or vertical (Y). When the rail direction is horizontal, the slave tool heads are either to the top or to the bottom of the master tool head. When the rail direction is vertical, slave tool heads lie either to the left or to the right of the master tool head.

The tool heads are arranged along the height of the sheet when the rail direction is 'X' and along the length of the sheet if the rail direction is 'Y'.

When you select a particular machine from the *Machine* dropdown list, the default *Rail direction* for that particular machine is displayed with the *Multi head options* tab. This default value is defined in the Machine.ini file. You can change the *Rail direction* to suit your nesting job requirements.

Tool head distance
The *Tool head distance* value indicates minimum tool head distance to be used for a nesting job involving multiple tool heads.
When you select a particular machine from the *Machine* dropdown list, the default *Tool head distance* value for that particular machine is displayed with the *Multi head options* tab. This default value is defined in the Machine.ini file.

Editing the default settings for the Multi Head Options tab

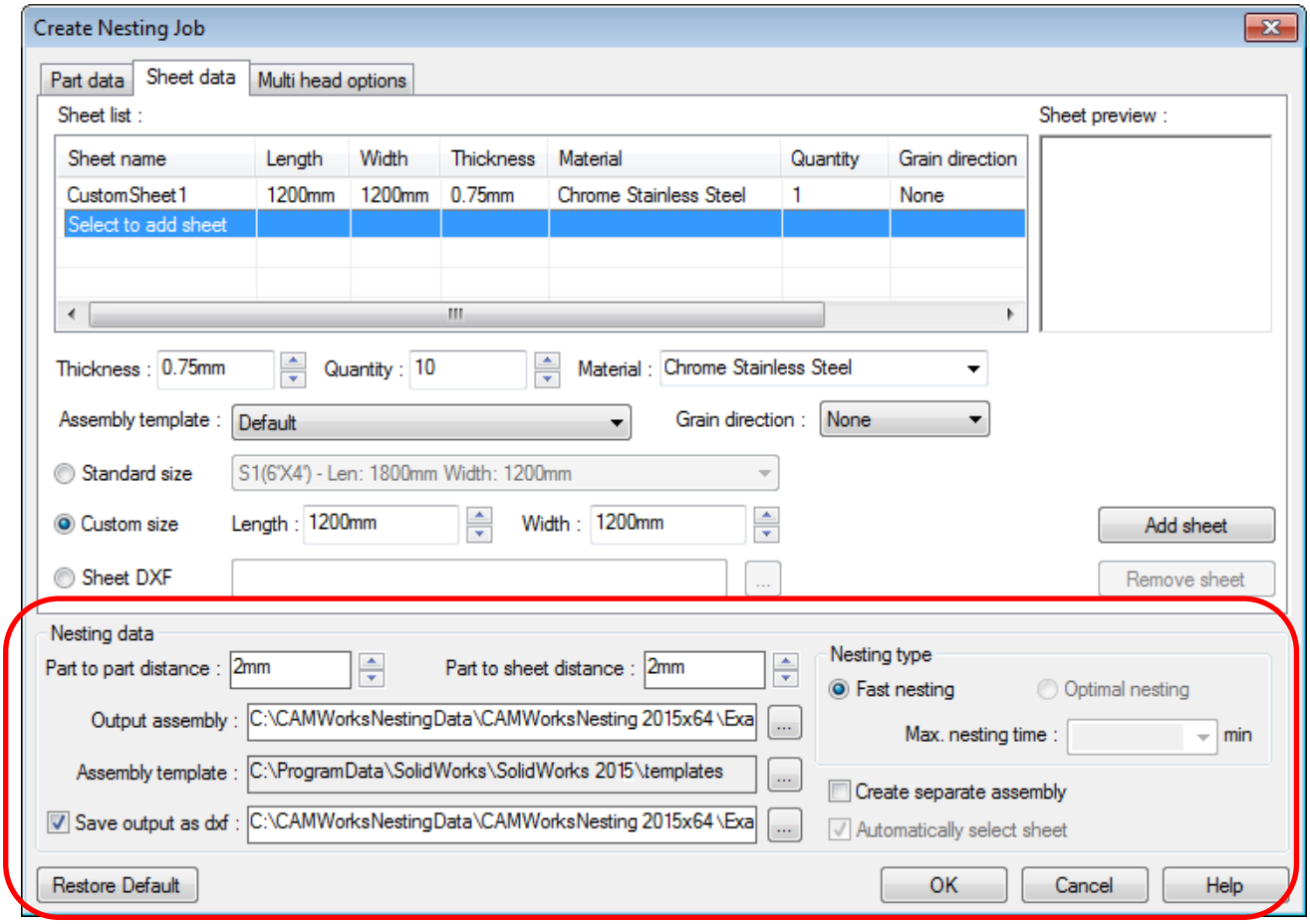
The nesting specific information and default parameter values for machines which support nesting with multiple tool heads is defined in the Machine.ini file located in the *CAMWorksNestingData**CAMWorksNesting* 201x*Lang**English* folder.

If you intend to make use of the feature of nesting with multiple tool heads, then the *Machine.ini* file can be customized depending on your requirements. For a detailed understanding of how to customize this file, read the section on Machine.ini.

Define Nesting Data Parameters

The *Nesting Data* group box in the bottom area of the *Create Nesting Job* dialog box is used to set the following nesting parameters:

- Part to Part Distance
- Part to Sheet Distance
- Output Assembly File path
- Assembly template path
- Save output as dxf
- 'Automatically Select Sheet' option
- 'Create separate assembly' option
- Nesting Type
- Nesting Time (in minutes)



Defining Nesting Parameter values

Part to Part Distance

The Part to Part Distance indicates the distance to be maintained between two nested parts in the sheet. The default value is '0'. Assign the required value by entering it in the field.

Part to Sheet Distance

The Part to Sheet Distance indicates the distance to be maintained between a part and the edge of the sheet. The default value is zero. Assign the required value by entering it in the field.

Output Assembly file

The folder location specified in the field *Output Assembly File* indicates the location where the generated Assembly of nested parts will be saved. The default location is always the same folder location as the part(s)/assembly to be nested. Click on the ... button to select an alternate location to save the Assembly file(s) that will generated after the successful execution of a nesting job.

Assembly Template Path

An assembly templates (*.*asmdot*) is a template document that includes user-defined parameters and customized options which forms the basis for new assemblies.

Use the  button given next to the Assembly template field to browse to the location where the desired assembly template is saved. In CAMWorks Nesting, the default assembly template loaded is the one defined in the Default Templates section of the SOLIDWORKS Systems Options.

Save output as dxf

The CAD data file format called *Drawing Exchange format* (*.dxf*) is an international standard which enables data interoperability between AutoCAD and other programs.

The *Save output as dxf* checkbox option is a feature which allows you to save the nested layouts which are generated after executing a nesting job in the **.dxf** format.

Use the  button given next to this field to browse to the folder location where the .dxf files are to be saved. This field which is used to specify the folder location is enabled only when the *Save output as dxf* checkbox is checked.

If the *Save output as dxf* option is used when multiple nested layouts are generated (saved either as separate configurations or as separate assembly files), then a separate .dxf file will be created for each nested layout that is output and saved in the specified folder location.

When the *Save output as dxf* checkbox is not checked, then the nested layouts generated after executing a nesting job will be saved only in the assembly file format *(.sldasm)*.

Note: **The assembly file format (.sldasm) is the standard file format in which the nested layouts are generated. If the Save output as dxf option is used, then the nested layouts will be generated in two file formats: .sldasm & .dxf and saved in the specified folder locations.**

Automatically Select Sheet

For a nesting job containing multiple parts and sheet types, it is difficult for the user to select the best sheet type or best sequence of sheets in order to obtain best yield based on the sheet utilization factor. Since it is very important to predict and procure the inventory in correct numbers, an Inventory Forecasting Module (IFM) which forecasts the optimum sheet inventory is necessary.

The Inventory Forecasting Module operates within CAMWorks Nesting in the form of **Automatically Select Sheet** option.

Automatic Sheet Selection supports two methods:

1. **Unique Sheet Forecaster**

   If this method is selected, the feature would select one best sheet among the set of sheets considered, depending upon overall utilization obtained. After knowing which sheet type is the best for that particular nesting order, the user can place an order for that much quantity of the sheet type. This will help in reducing the sheet variety – thus reducing the time required for machine specific sheet settings.

2. **Combinatorial Sheet Forecaster**

   If this method is selected, the feature selects a combination of sheet types from the set of sheets available in the Sheet list, depending on the overall utilization obtained.

Setting the Forecaster Method in DefaultValues.ini

At any given point of time, only one of the above mentioned Forecasting methods can be used. This setting is available in the DefaultValues.ini file. The default method is *Combinatorial Sheet Forecaster*.

Create Separate Assembly

The *Create separate assembly* option is available under the Nesting Data group box in the *Create Nesting Job* dialog box.

When multiple nesting layouts are generated after the execution of a nesting job, CAMWorks Nesting lists all the nested layouts under the Configurations Manager Tab of SOLIDWORKS. These nested layouts are saved as a part of a single assembly file (**.sldasm*).

If you wish to generate separate assemblies for each such nested layout generated, the select the *Create Separate Assembly* option. After executing a nesting job, all the nested layouts will then be saved as separate assemblies in the destination folder specified in the *Output Assembly* file location.

Nesting Type

Fast Nesting and *Optimal Nesting* indicate the two different sets of algorithms used to implement Nesting.

Fast Nesting: This method should be used when nesting quickly is more important than optimal sheet utilization.

Optimal Nesting: This method focuses on optimal utilization of the sheet by running multiple algorithms and chooses the best result in terms of utilization. It is the default setting for CAMWorks Nesting.

Max Nesting time

Time constraints can be applied to optimal nesting. The *Max. nesting time* field is enabled when the *Nesting type* is set to *Optimal Nesting*. Use this field to enter nesting time limit in minutes. This will restrict the maximum

allowable time for nesting to the set value. The default value for max nesting time field is *No constraint*, which indicates that a full optimal nesting will be run without any time constraints.

Note: **Optimal Nesting option is not available for a nesting job that uses multiple tool heads. In other words, the Optimal Nesting option is disabled when the functionality of nesting with multiple tool heads is enabled.**

Generating the Nested Layout

After setting the part, sheet and nesting parameters, click *OK*.

This sets into motion the process to generate a nested layout. Generating the layout might take some time depending on the complexity of the part.

Two files will be generated during the nesting process, namely a text file and an assembly file.

After the Nesting process is completed, CAMWorks Nesting will display a message indicating the location of the text file containing the **summary of the Nest Results**. Click *OK* to close the message. The Text file will be displayed.

The **Nested layout assembly** will be displayed in the Graphics area. Both these files are saved in the location indicated *Output Assembly* file path stated in the *Apply Nesting* dialog box.

How the Nested layouts generated are saved within SOLIDWORKS

Once the nesting process using the CAMWorks Nesting application is completed, the nested layout(s) generated will always be saved as a SOLIDWORKS assembly file (***.sldasm***). Depending on various factors such as thickness and/or material part of part, number of sheets, grain direction, etc., either one or multiple Nested layouts will be generated.

- **When only one nested layout is generated:**
  - The will be saved as a SOLIDWORKS Assembly file comprising of nested parts.
  - The sheet dimensions will be saved as a SOLIDWORKS sketch.
- **When multiple nested layouts are generated:**
  - These nested layouts will be saved as a SOLIDWORKS Assembly file comprising of assemblies. Each assembly is a nested layout comprising of nested parts.
  - The sheet dimensions for each sheet will be saved as a SOLIDWORKS sketch.

Once the nested layout(s) are generated, each nested layout assembly (sheet layout containing nested parts) will be listed in the *SOLIDWORKS Configurations Manager*.

TUTORIAL 1 - NESTING AN ASSEMBLY

This tutorial is intended to give you a preview of how nesting is done for a simple assembly file comprising sheet metal parts. The sheet metal parts will be nested using a single tool head.

Generating a nested layout as explained in this tutorial will help you understand better the concepts explained in the next section: *Initialization Files of CAMWorks Nesting*.

Step 1: Open the Assembly

1. Load the CAMWorks Nesting Add-In in SOLIDWORKS.
2. Open the assembly file ***Tutorial_1.sldasm*** located in the ...*CAMWorksNestingData**Examples**Tutorials**Assemblies**Tutorial1* folder of your CAMWorks Nesting installation folder. *(**Refer page 19**)*

Step 2: Assign Nesting Parameters

1. Select *Create Nesting Job* from the CAMWorksNesting menu bar.

   OR

   Click on the *Create Nest Job* button on the CAMWorks Nesting Ribbon Bar.
2. The *Create Nesting Job* dialog box is displayed. Click on the Multi Head Options tab and ensure that *SingleTHMachine* is the machine listed in the *Machine* field.

Part Data Tab

1. Click on *Part Data* tab. The solid parts *Tutorial_1a.sldprt* and *Tutorial_1b*.sldprt which comprise the assembly are listed in the Part List along with their part parameters.
2. In the **Quantity** column of the *Part List*, set the quantity of *Tutorial_1a.sldprt* to **25** and the quantity of *Tutorial_1b.sldprt* to **38**.Leave all other default part parameter settings as it is. *(**Refer page 25**)*
3. Assign a **Step Angle** of 90 degrees to both the parts *Tutorial_1a.sldprt* and *Tutorial_1b.sldprt*.

Sheet Data Tab

1. Click on the *Sheet Data* tab. Observe that the **thickness and material** fields display values identical to those of the first part in the part list.
2. To add a sheet, following are the steps:
   a. Select *Custom Size*. The *Length* and *Width* fields will be activated and will display default values as defined in the DefaultValues.ini file.

b. Assign a **length of 1200mm** and a **width of 1200 mm**. Set the sheet *Quantity* to '1'.

c. Click *Add Sheet* to add the sheet to the Sheet List. *(**Refer page 31**)*

Nesting Parameters

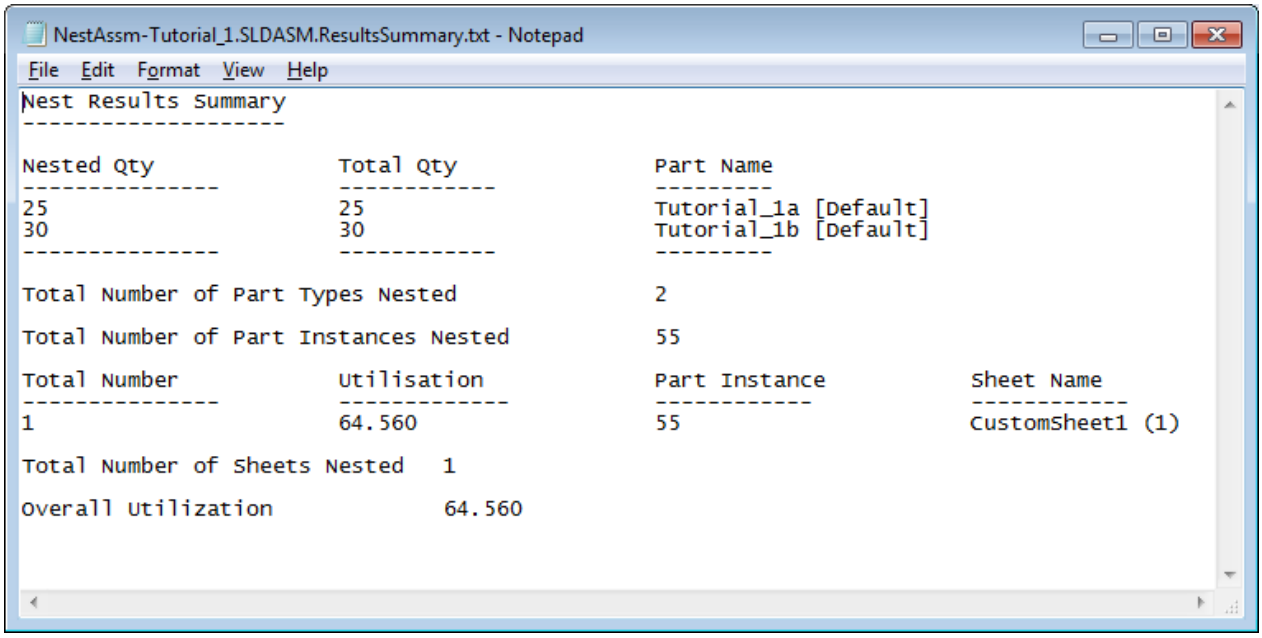
1. In the Nesting Parameters group box, set a *Part to Part distance* of 2 mm is set and a *Part to Sheet distance* of 3mm.
2. Click on the ... button to select the location to save the output Assembly file and Summary text file.
3. Set the Nesting method to *Optimal nesting*.
4. Click *OK* to execute the Nesting Process.

Step 3: Generating the Nested Layout

Summary Text

The Summary text file indicates that the prescribed quantities for all the parts have been nested within the sheet. This summary text file is saved in the same location as the assembly that was nested.

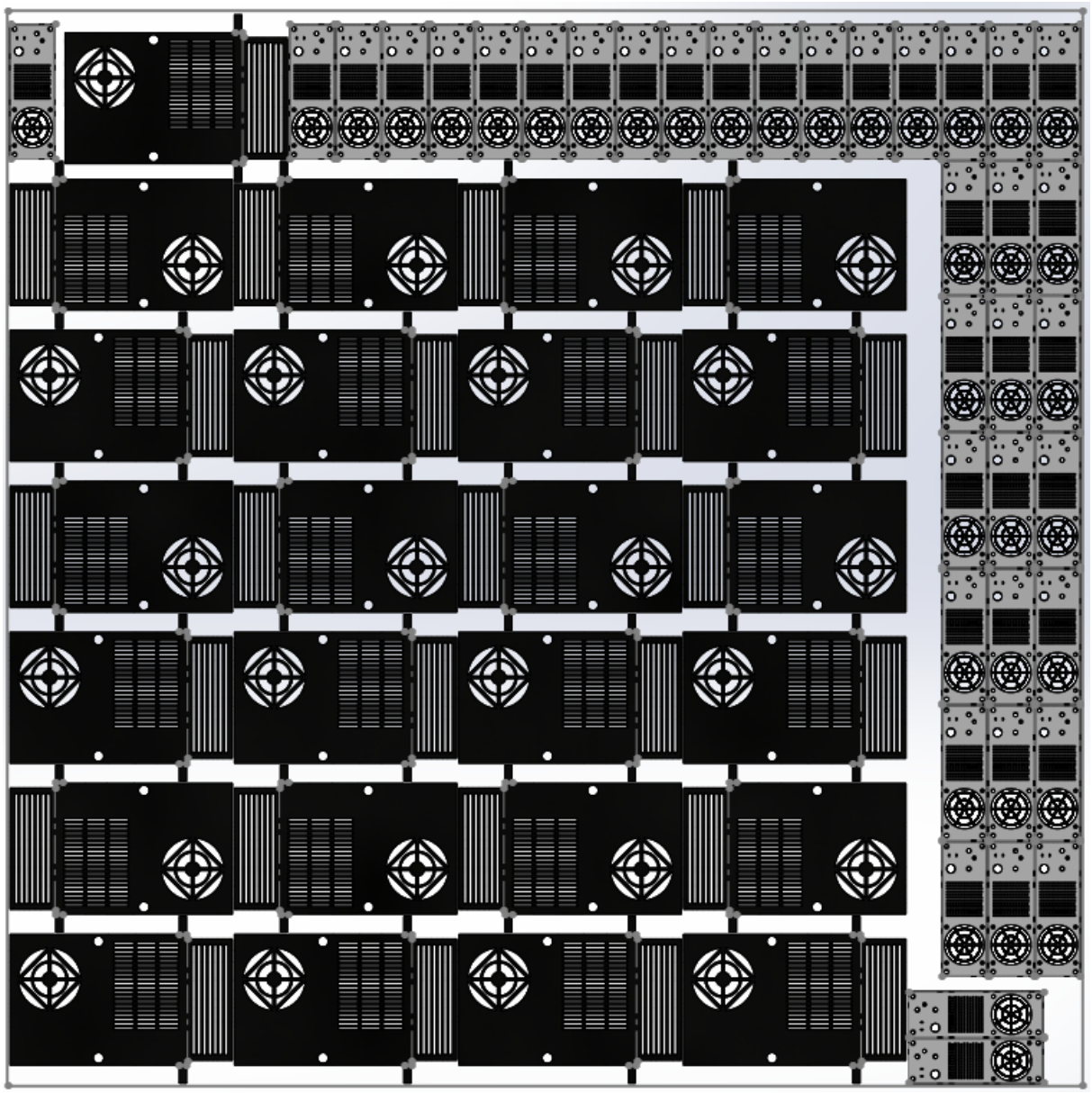
This nested assembly will be saved with the prefix *'NestAssm-'* and suffix *'.ResultsSummary'* added to the name of the assembly file that was nested.



Nesting Results Summary Text File

Nested Assembly

The nested layout assembly generated after executing the nesting job and the Summary Results file are stored in the same location as the assembly file that was nested. This nested assembly will be saved with the prefix **'NestAssm-'** added to the name of the assembly file that was nested. The resultant nested layout is given on the next page.



Nested Layout of Assembly achieved with Fast Nesting

SECTION TWO

Initialization Files

INITIALIZATION FILES OF CAMWORKS NESTING

CAMWorks Nesting provides the facility to define and edit default values, settings and populate the dropdown fields.

These values can be defined, set or edited in the initialization files (files with *.ini* extension) present in the CAMWorks Nesting installation folder.

Location of the CAMWorks Nesting Initialization files

All these initialization files are located in the CAMWorks Nesting installation folder. A typical installation path will be:

Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Lang\English

Backing up the Initialization Files

Every time you install a fresh version or Service Pack of CAMWorks Nesting, the initialization files will be overwritten. Hence, if you have customized the settings in the various initialization files, we recommend that you take a back-up of the initialization files before installing a fresh version of CAMWorks Nesting.

Configuration settings available in Initialization files

Following are the various settings that can be controlled from the CAMWorks Nesting Initialization files.

Setting	Initialization File
Define and store information on the standard sheet sizes	StandardSheets-INCH.ini / StandardSheets-MM.ini
Populating the Materials dropdown list	Materials.ini
Defining default part quantity	DefaultValues.ini
Defining default sheet thickness and material	DefaultValues.ini
Defining default dimensions for Custom sheet	DefaultValues.ini
Defining default values for *Part to part distance* and *Part to sheet distance*	DefaultValues.ini
Enabling/disabling the option of Preferential hole filling	DefaultValues.ini

Defining the default state of the *Create Separate Assembly* option	DefaultValues.ini
Defining the default state of the *Automatically Select Sheet* option	DefaultValues.ini
Defining default state of the *'Save output as dxf'* option	DefaultValues.ini
Enabling/disabling the functionality to link CAMWorks Nesting to CAMWorks	DefaultValues.ini
Enabling/Disabling the Display of the Message shown before linking CAMWorks Nesting with CAMWorks	DefaultValues.ini
Enabling/disabling the option of flattening sheet metal parts before generating nested layouts	DefaultValues.ini
Setting the Inventory Forecast Method to be used	DefaultValues.ini
Enabling/disabling the display of Assembly quantities in the *Create Nesting Job* dialog box	DefaultValues.ini
Setting the Stamp Feature Unfold Option for sheet metal parts	DefaultValues.ini
Enabling/disabling the option of nesting with multiple tool heads	Machine.ini
Enabling the interactive dialog box to edit parameters associated with nesting with multiple tool heads	Machine.ini

Note: **It is highly recommended that you read this section in order to gain an understanding of how to customize the CAMWorks Nesting settings to meet your facility's requirements.**

StandardsSheets-INCH.ini & StandardsSheets-MM.ini

CAMWorks Nesting provides the option to define and store information of standard sheet sizes used at your facility. This provides the benefit of automating repetitive operations and saves time during the nesting procedure by eliminating the need to define sheet dimensions.

➔ To define the length and width of a sheet in IPS units, use the *StandardSheets-INCH.ini* file
➔ To define the length and width of a sheet in MMGS units, use the *StandardSheets-MM.ini* file

Viewing the StandardSheets-INCH.ini/ StandardsSheets-MM.ini file:

1. Go to the CAMWorks Nesting installation folder. (A typical installation path will be...*Drive:\CAMWorksNestingData\ CAMWorksNesting 201x\Lang\English*)
2. Open the file named ***StandardSheets-INCH.ini*** or ***StandardSheets-MM.ini***.
3. The fifth line of both these files indicates the **sheet count**. For example, if 15 standard sheets have been defined, then the sheet count is set to 15.

Adding sheets to the StandardSheets-INCH.ini/ StandardsSheets-MM.ini file:

1. The parameters defined in for a standard sheet include its name, length, width.
2. Increment the Sheet count by 1 every time you add a sheet.
3. For example, suppose the StandardSheets-MM.ini file has 24 standard sheets defined. To add a 25th sheet with name *S25*, length of 3500 mm and width of 2200 mm, increment the sheet count by one and set it to 25. The format for defining this new sheet in the StandardSheets-MM.ini file is as follows:

```
[Sheet25]
Name = S25
Length = 3500
Width = 2200
```

Adding a Standard Sheet in the StandardSheets-MM.ini file

Editing parameters in StandardSheets-INCH.ini/ StandardsSheets-MM.ini file

1. Once defined, the parameters of a sheet defined in the *StandardSheets-INCH.ini* and *StandardSheets-MM.ini* can be changed as and when the need arises.
2. Consider this example: Suppose a sheet named *S23(12' X8')* with a Length of 144 inches and width of 96 inches is defined in the StandardSheets-INCH.ini file. To change this sheet's name to *Std_Sheet23 (15' x10')* with a length of 180 inches and width of 120 inches, the following changes need to be done in the *StandardSheets-INCH.ini* file.

Sheet Parameters before changing	Sheet Parameters after changing
[Sheet23] Name = S23(12' x8') Length = 144 Width = 96	[Sheet23] Name = Std_Sheet_23(15' x10') Length = 180 Width = 120

Editing Parameters of a Standard Sheet in StandardSheets-INCH.ini

Material.ini

The Material dropdown list available in the Pat Data tab allows you to assign the material of the part(s) to be nested.

CAMWorks Nesting provides the option to populate the Material dropdown list and thus store standard materials used at your facility. This provides the benefit of making the material selection easier by enabling you to select desired material from the dropdown list instead of manually typing it in.

Location

This file is located in the CAMWorks Nesting installation folder.
A typical installation path will be:
Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Lang\English

Viewing/Editing the Material dropdown list

1. Open the *Material.ini* file located in the *CAMWorks Nesting* installation folder.
2. The second line of this file indicates the number of materials already listed. For example, if 10 materials are defined in the file, then the *MaterialCount* is set to '10'.
3. To add more materials to the list, enter the name of the new material in the same syntax as those already listed.
4. Increment the Material Count by 1 every time you add a material.
5. For example, to add a fifth material to a *Material list* containing four materials, set the Material count to 5 and add the new material at the bottom of the list:

```
[Material]
MaterialCount = 5
Material1 = Steel
Material2 = Copper
Material3 = Iron
Material4 = Bronze
Material5 = Aluminium
```

Typical syntax of the Material.ini file

Note: **CAMWorks Nesting extracts the part parameter of Material from the Solid Part and displays it in the Material field as default material of the part. When the material cannot be extracted from the 3D model, CAMWorks Nesting assigns the first material listed in the Material.ini file as the default material for the part.**

DefaultValues.ini

This file is used to assign default values of part, sheet and nesting data parameters. Certain default settings which cannot be set in the *Create Nesting Job* dialog box are set from the *DefaultValues.ini* file.

Location

This file is located in the *CAMWorks Nesting* installation folder.

A typical installation path will be:

Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Lang\English

Assigning default Part Quantity

The default value for part's quantity can be assigned and edited in the *DefaultValues.ini* file.

Steps to edit default part quantity in the DefaultValues.ini file

1. Open the file named DefaultValues.ini located in the CAMWorks Nesting installation folder.
2. Observe the section named [PartData]. The default quantity is defined in this section.
3. Edit the quantity value to apply a new default value for quantity of the parts to be nested. For example, to assign a new default quantity of '6', the value in the *DefaultValues.ini* should be changed as follows:

```
[PartData]
;Quantity only Integers, real values are not supported
Quantity = 6
```

Typical syntax for setting Part quantity in DefaultValues.ini

Defining default Sheet Thickness and Quantity

- ➔ The DefaultValues.ini file can be used to define the default sheet thickness for sheets of type *Custom Size* and *Sheet DXF*.
- ➔ The default sheet *Quantity* for all sheet types is defined in *DefaultValues.ini*.

Steps to edit default sheet thickness and quantity in DefaultValues.ini file

1. Open the file named *DefaultValues.ini* located in the *CAMWorks Nesting* installation folder.
2. Observe the section named [SheetData]. The default quantity (line number 2) and thickness (line no. 3 and 4) for a sheet are defined in this section.

3. The default quantity is always defined as an integer. Edit the quantity value to apply a new default value for quantity of the sheets.
4. The thickness value is defined in both millimeters (MMGS units) and inches (IPS units). Edit these values to change the default value of thickness.
5. For example, to assign a new default thickness of 4mm in MMGS units, 0.1 inch in IPS units and a quantity of '2', the values in the DefaultValues.ini will be changed as follows:

```
[SheetData]
;Quantity only Integers, real values are not supported
Quantity = 2
Thickness-MM = 4.000000
Thickness-Inch = 0.1
```

Typical syntax for setting Sheet Thickness & quantity in DefaultValues.ini

Defining default dimensions for Custom Sheet

➔ For sheets of type *Standard Size*, the dimensions for each sheet is defined either in the StandardSheets-INCH.ini or StandardSheets-MM.ini file based on the unit system used.

➔ For sheets of type *Custom Size*, the dimensions are to be input by the user. The default dimensions displayed on selection of sheet type *Custom Size* is defined in the DefaultValues.ini file. Since the dimensions are defined either in the MMGS or IPS unit system, the default values too are defined in MMGS and IPS units separately.

Editing the Default Custom Size Dimensions in the DefaultValues.ini file

1. Open the file named *DefaultValues.ini* located in the CAMWorks Nesting installation folder.
2. Observe the section named [CustomSheet]. The default Length and Width to be displayed, when sheet of type *Custom Size* is selected, is displayed in this section.
3. The length and width is defined in both MMGS and IPS units. Edit these values to change the default Length and Width values.
4. For example, to assign a new default length of 2500mm and a Width of 800 mm (MMGS units) and a length of 120 inches and width of 72 inches (IPS units), the values under [CustomSheet] in the *DefaultValues.ini* should be changed as follows:

```
[CustomSheet]
Length-MM = 2500
Width-MM = 800
Length-Inch = 120
Width-Inch = 72
```

Typical syntax for setting default dimensions for custom size sheet in DefaultValues.ini

Assigning default Part to Part Distance & Part to Sheet Distance

The default values to be displayed for *Part to part distance* and *Part to sheet distance* in the *Nesting Data* Group Box are defined in the *DefaultValues.ini* file.

Editing the Part to Part distance and Part to Sheet Distance

1. Open the *DefaultValues.ini* file located in the CAMWorks Nesting installation folder.
2. Observe the section named [NestingData]. The default values for *Part to part distance* and *Part to sheet distance* are defined here.
3. The *Part to part distance* and *Part to sheet distance* are defined in both MMGS and IPS units. Edit these values to change the default values.
4. For example, consider that a *Part to Part Distance* of 3mm and a *Part to Sheet Distance* of 2mm is to be assigned in the new default values in the MMGS units. Similarly, a *Part to Part Distance* of 0.125 inch and a *Part to Sheet Distance* of 0.25 inch is to be assigned in the new default values in the MMGS units.
5. To apply these changes, the values under [NestingData] in the *DefaultValues.ini* file should be changed as follows:

```
[NestingData]
PartToPartDistance-MM = 3
PartToPartDistance-Inch = 0.1250000
PartToSheetDistance-MM = 2
PartToSheetDistance-Inch = 0.250000
```

Typical syntax for setting default Part to Part and Part to Sheet distance

Enabling/Disabling the Preferential Hole Filling Functionality

The 'Preferential hole filling' functionality enables a smaller part to be nested in the holes of larger parts during the nesting process resulting in higher sheet utilization and minimal scrap.

Setting the *PreferHoleFilling* flag in the *DefaultValues.ini* file to '1' enables this functionality while setting it to '0' will disable the functionality.

```
[NestingData]
;Options for PreferHoleFilling: 0 : No, 1 : Yes
PreferHoleFilling = 1
```

Settings in DefaultValues.ini file to enable Preferential Hole Filling

Note: **The option of Preferential Hole Filling cannot be set from within the Create Nesting Job dialog box. You need to assign your preferred settings in the DefaultValues.ini file for this option.**

This functionality is illustrated in Tutorial 5.

Assigning default state for the *Create Separate Assembly* option

You can set the default option whether the *Create Separate Assembly* checkbox in the *Nesting Data* group box is to remain checked/ unchecked when you open the *Create Nesting Job* dialog box.

Setting the ***CreateSeparateAssembly*** flag in the *DefaultValues.ini* file to '0' leaves this checkbox unchecked while setting it to '1' will place a check in this checkbox.

```
[NestingData]
;Options for CreateSeparateAssembly: 0 : No, 1 : Yes
CreateSeparateAssembly = 0
```

Setting for the 'Create Separate Assembly' option in DefaultValues.ini file

Assigning default state for the *Automatically Select sheet* option

You can set the default option whether the *Automatically select sheet* checkbox in the *Nesting Data* group box is to remain checked/ unchecked when you open the *Create Nesting Job* dialog box.

Setting the ***AutomaticallySelectSheet*** flag in the *DefaultValues.ini* file to '0' leaves this checkbox unchecked while setting it to '1' will place a check in this checkbox.

```
[NestingData]
;Options for Tick/Un-tick AutomaticallySelectSheet: 0:No, 1:Yes
AutomaticallySelectSheet = 1
```

Settings for the 'Automatically Select Sheet' option in DefaultValues.ini file

Enabling/Disabling the option of Flattening Sheet Metal Parts

With respect to nesting of native sheet metal parts, you can choose whether the nested layout is to be computed based on the flattened or non-flattened sheet metal parts.

Setting the ***FlattenSheetMetalPart*** flag in the *DefaultValues.ini* file to '1' enables computation of the nested layout based on the flattened (unfolded) sheet metal parts. Setting this flag to '0' will lead to computation of the nested layout based on non-flattened sheet metal parts.

Note that this setting applies only to native sheet metal parts and not imported sheet metal parts.

```
[NestingData]
;Options for FlattenSheetMetalPart: 0 : No, 1 : Yes
FlattenSheetMetalPart = 1
```

Settings in DefaultValues.ini file to enable flattening of sheet metal parts for nesting

Note: **The option of flattening or not flattening sheet metal parts for computation of the nested layout cannot be set within the 'Create Nesting Job' dialog box. You need to assign the settings in the *DefaultValues.ini* file for this option.**

This functionality is illustrated in Tutorial 3 and Tutorial 6.

For information on flattening imported sheet metal parts, refer Tutorial 7 & Tutorial 8.

Enabling/Disabling the option of utilizing 'Fix Component' or 'Mate-Lock' feature

The nested layout generated after a nesting job is an assembly of parts. Sometimes, the parts may get accidentally repositioned from their original position in the nested assembly due to human error, thus disturbing the nested layout.

The '**Fix component**' feature within SOLIDWORKS/CAMWorks Solids prevents the movement of parts within an assembly for which this feature is enabled. Similarly, the '**Mate-lock**' feature of SOLIDWORKS/CAMWorks Solids too serves the same purpose.

CAMWorks Nesting provides a setting in the DefaultValues.ini file wherein the 'Fix component' or 'Mate-Lock' feature of SOLIDWORKS/ CAMWorks Solids can be activated by default for all the parts in nested layout(s) generated after the execution of a nesting job.

When the ***FixComponent*** flag in the *DefaultValues.ini* file is set to '1', the 'Fix component' feature of SOLIDWORKS/CAMWorks Solids is activated as default setting. However, the default setting of this flag at the time of installation is '0', indicating that both the 'Fix Component' and 'Mate-Lock' features are inactive.

```
[NestingData]
;Option for FixComponent: 0 : No, 1 : Yes, 2: Mate - Lock
FixComponent = 1
```

Settings in DefaultValues.ini file to activate the 'Fix Component' Feature

When the ***FixComponent*** flag in the *DefaultValues.ini* file is set to '2', the 'Mate-Lock' feature of SOLIDWORKS/CAMWorks Solids is activated as default setting.

```
[NestingData]
;Option for FixComponent: 0 : No, 1 : Yes, 2: Mate - Lock
FixComponent = 2
```

Settings in DefaultValues.ini file to activate the 'Mate-Lock' Feature of SolidWorks

Note: **The settings for the 'Fix Component' or 'Mate-Lock' Feature cannot be changed within the Create Nesting Job dialog box. You can only change the settings in the *DefaultValues.ini* file.**

Assigning the default Inventory Forecasting method

The Inventory Forecasting Module operates within CAMWorks Nesting in the form of *Automatically select sheet* option. CAMWorks Nesting supports two inventory forecasting methods viz. *Unique Sheet Forecaster* and *Combinatorial Sheet Forecaster*.

The forecasting method to be used is defined in the *DefaultVales.ini* file using the ***IFMType*** flag. Setting this flag to '1' enables the *Unique Sheet Forecaster* method while setting it to '2' enables the *Combinatorial Sheet Forecaster* method.

```
[NestingData]
;Option for IFMType: 1: For UNIQUE_SHEET_FORECASTER,
;2: For COMBINATORIAL_SHEET_FORECASTER
IFMType = 2
```

Settings in DefaultValues.ini file to assign the Inventory Forecasting method to be used

Note: **The Inventory Forecasting Method (IFM) used cannot be changed within the Create Nesting Job dialog box. You need to change the settings in the DefaultValues.ini file to change the method for inventory forecasting.**

Assigning default state for the 'Save Output as dxf' option

You can set the default option whether the *Save output as dxf* checkbox in the *Nesting Data* group box should remain checked/ unchecked when you open the *Create Nesting Job* dialog box.

In the [NestingData] section of the *DefaultValues.ini* file, setting the ***DxfFile*** flag to '0' leaves this checkbox unchecked while setting it to '1' places a check in this checkbox.

```
[NestingData]
Option for creating DXF file: 1 : Yes, 0 : No
DxfFile = 1
```

Settings for the 'Save output as dxf' option in DefaultValues.ini file

Enabling/disabling the functionality to add nested Parts to CAMWorks Part Manager

An option is provided within CAMWorks Nesting that allows you to automatically link the nested layout output of CAMWorks Nesting as the input for CAMWorks. This automation saves considerable time by reducing the steps required to generate the NC code.

For more details, read: **Functionality to link CAMWorks Nesting with CAMWorks**

In the section named [AddPartsToCW] of the *DefaultValues.ini* file, setting the flag named *AddPartstoCWManager* to '1' enables the functionality and '0' disables this functionality.

By default, this flag is set to '1'. Thus, the functionality is enabled by default.

```
[AddPartsToCW]
AddPartstoCWManager=1   :Yes,  0 : No
AddPartstoCWManager = 1
```

Settings for enabling the functionality to link CAMWorks Nesting with CAMWorks

This functionality is illustrated in Tutorial 13.

Enabling/disabling the Display of the Message shown before linking CAMWorks Nesting with CAMWorks

An option provide the how to enable/disable the display of the message before linking CAMWorks Nesting with CAMWorks. The setting for enabling/disabling the display of this message is controlled from the DefaultValues.ini file.

In the section named [AddPartsToCW] of the *DefaultValues.ini* file, setting the flag named *DontShowAddPartsToCWWarning* to '1' enables the functionality and '0' disables this functionality.

By default, this flag is set to '1'. This setting enables the display of the warning message but the warning message will be displayed in the CAMWorks Nesting UI only if the flag named 'AddPartstoCWManager' is set to '1' in the DefaultValues.ini file.

```
[AddPartsToCW]
DontShowAddPartsToCWWarning=1 : Show msg, 0 : Don't show msg
DontShowAddPartsToCWWarning=1
```

Settings for enabling the functionality to display the warning message

This functionality is illustrated in Tutorial 13.

Enabling/disabling the feature of 'Assigning Assembly Quantities'

This setting comes into effect if you are nesting an assembly. It is used to set the default state whether Quantity column for the Assembly (and its constituent sub-assemblies, if any) are to be displayed in the Part Data tab of the *Create Nesting Job* dialog box.

In the [AssemblyData] section of the *DefaultValues.ini* file, setting the ***ShowAssemblyQuantity*** flag to '1' enables the display of Assembly name and Quantity in the *Part List* grid of the *Part Data* tab in the *Create Nesting Job* d0 ialog box. The parts comprising the assembly and its sub-assemblies too will be listed in the Part Grid. This is the default setting.
When this flag is set to '0', the Assembly name and Quantity columns will not be displayed in the *Part Data* tab of the *Create Nesting Job* dialog box. Only the parts comprising the assembly to be nested will be listed in the dialog box.

```
[AssemblyData]
;Options for ShowAssemblyQuantity; 0 : Only Part Data,
1: Both Assembly and Part Data
ShowAssemblyQuantity = 1
```

Settings to enable display of Assembly Quantities in the Part Data tab of the 'Create Nesting Job' dialog box

This functionality is illustrated in Tutorial 9.

Settings for the Stamp Feature Unfold Option

The setting to control the behavior of the stamp feature display on a sheet metal part after the part is unfolded can be assigned only from the *DefaultValues.ini* file.

There are three available settings to control the behavior of the stamp feature after the part is unfolded. This setting is controlled by the flag named ***StampFeatureUnfoldingOption*** in the [Unfold_Options] section of the *DefaultValues.ini* file.

Following are the values that can be assigned to this flag to control the stamp feature behavior:

i. **0** : Assigning the value '0' ensures that the stamp feature is **retained** after the part is unfolded. (This is the default setting at the time of installation.)

ii. **1** : When the value '1' is assigned to this flag, the stamp feature is **patched** with a flat planar surface after the part is unfolded.

iii. **2** : When the value '2' is assigned to this flag, the stamp feature is **ignored** after the part is unfolded. The area covered by the stamp feature is replaced with a hole.

```
[Unfold_Options]
;StampFeatureUnfoldingOption=0 : Retain, 1 : Patch, 2 : Ignore
StampFeatureUnfoldingOption=0
```

Settings for the Stamp Feature Unfold Option for sheet metal parts

This functionality is illustrated in Tutorial 11.

Machine.ini

For flame cutting applications, CAMWorks Nesting provides an optional functionality known as *Multiple Tool Head Nesting*. This functionality allows you to nest two or more identical layouts simultaneously using multiple tool heads. Machines which support nesting using multiple tool heads are known as *Multi tool head machines*.

The nesting specific information and default values of parameters for such machines are configured in the Machine.ini file.

➔ If your plan to make use of the *Nesting with Multiple Tool Heads* feature, then the Machine.ini file needs to be customized depending on the requirements at your machining facility.

➔ If you do not plan to use the feature of *Nesting with Multiple Tool Heads*, then you can either disable the feature or leave the default settings untouched.

Configuration settings in the Machine.ini file

The following settings are configured from the Machine.ini file:

- Enabling/ disabling the functionality for nesting with multiple tool heads
- Enabling/Disabling the display of the Multi Head Options tab in the Create Nesting Job dialog box
- The number (count) and names of machines which support the functionality for nesting with multiple tool heads
- Default values of parameters associated with nesting with multiple tool heads for such machines

Location

The Machine.ini file is located in the CAMWorks Nesting installation folder.
A typical installation path will be:
Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Lang\English

Enabling/Disabling the option of Nesting with Multiple Tool heads

1. Open the file named *Machine.ini* located in the CAMWorks Nesting installation folder. Observe the first section named [MultiHeadData].

```
[MultiHeadData]
;Multi head flag; set to True(1) for multi tool head nesting, or else set to False(0)
MultiHeadFlag = 1
```

Syntax for enabling the option for nesting with multiple tools heads

2. The second line under this section contains the **MultiHeadFlag**. The flag named *MultiHeadFlag* is used to enable/disable the functionality of nesting with multiple tools.
   a. When the *MultiHeadFlag* is set to '1', the option of nesting with multiple tool heads will be activated. This is the default setting at the time of installation.
   b. When the *MultiHeadFlag* is set to '0', the option of nesting with multiple tool heads will be disabled. No interactive dialog box (the *Multi Head Options* tab in the *Create Nesting Job* dialog box) to view /edit parameters associated with Multiple Tool Head Nesting will be displayed. The settings of *ShowMultiHeadDialog* flag will be immaterial since the function is inactive.

Note: **To disable the feature of Nesting with Multiple Tool Heads, set the MultiHeadFlag in the Machine.ini file to '0'.**

Enabling/disabling the display of the *Multi Head Options* tab in the *Create Nesting Job* dialog box

The *Multi Head Options* tab in the *Create Nesting Job* dialog box is the interactive interface that allows you to view/edit the parameters related to Nesting with Multiple Tool Heads. The setting to enable/disable the display of this tab is controlled from the Machine.ini file.

1. Open the file named *Machine.ini* located in the CAMWorks Nesting installation folder. Observe the first section named *[MultiHeadData]*.

```
[MultiHeadData]
;Multi head flag; set to True(1) for multi tool head nesting, or else set to False(0)
MultiHeadFlag = 1
;Value to indicate whether Multi head api needs to be shown or not : Yes : 1, No , 0
ShowMultiHeadDialog = 1
```

Syntax for enabling the display of the Multi Head Option tab

2. The fourth line under this section contains the **ShowMultiHeadDialog** flag. When *MultiHeadFlag* is set to '1', the feature of Nesting with Multiple Tool Heads is activated. The *ShowMultiHeadDialog* flag is used to set the option whether a nesting job will be executed interactively using user-specified parameters or with default parameter values assigned to the Default machine in the *Machine.ini* file. This is explained as follows:
   - ➔ If the *ShowMultiHeadDialog* is also set to '1', then the *Multi head Options* tab is displayed in the *Create Nesting Job* dialog box. This tab allows you to view/edit the parameters for nesting with multiple tool heads. This is the default setting at the time of installation.
   - ➔ If both the *MultiHeadFlag* and *ShowMultiHeadDialog* are set to '0', then the feature of Nesting with Multiple Tool Heads will be disabled and the *Multi head Options* tab will not be displayed in the *Create Nesting Job* dialog box.
   - ➔ If the *MultiHeadFlag* is set to '1' and the *ShowMultiHeadDialog* is set to '0', then the feature of Nesting with Multiple Tool Heads will be active but *Multi head Options* tab will not be displayed in the *Create Nesting Job* dialog box. Consequently, the next nesting job executed will use the default parameter values assigned to the Default machine in the *Machine.ini* file and complete the nesting process.

Note: The ShowMultiHeadDialog flag controls whether the Multi Head Options tab in the Create Nesting Job dialog box will be displayed or not.

Defining the Machines which support nesting with multiple tools

For machines at your facility which support nesting with multiple tools, you need to define their names and the number of such machines. You also need to specify the default machine to be used when multiple machines are present.

In the *Machine.ini* file, these machine names, machine count and default machine to be used are defined in the **[DefaultMachine]** section.

Assigning the Machine Count, machine names and default machine

1. Observe the section named *[DefaultMachine]* of the *Machine.ini* located in the CAMWorks Nesting installation folder.

```
[DefaultMachine]
;Machine# where # is a number greater than 0 and less than the value of "Machine Count"
MachineCount = 5
Machine1 = SingleTHMachine
Machine2 = MachineName1
Machine3 = MachineName2
Machine4 = MachineName3
Machine5 = MachineName4
;Default machine name: # where # is a number between 0 and MachineCount
DefaultMachineName = Machine1
```

Typical syntax for defining the machine names, machine count and default machine

2. The machine count, name of the machines and default machine to be used are defined here.
   - **MachineCount**: The integer value assigned to this setting indicates the number of machines which support the 'nesting with multiple tools' functionality. The machine count has to necessarily be an integer value greater than zero. Increment the *MachineCount* by 1 every time you add a machine.

     For example, if you have three machines at your facility, the machine count will be '3'.
   - **Machine# = <machine name>**: This setting indicates names of the machines which support the *nesting with multiple tools* functionality. Machine# denotes the machine number. # is a number greater than zero and less than/equal to the *MachineCount* value. The *<machine name>* is an alphanumeric text string that represents the machine name. The defined machine names form a list of machines.

     In the *Multi head options* tab of the *Create Nesting Job* dialog box (the interactive dialog box to edit the multiple tool head related

parameters), the *Machine* dropdown list is populated by the Machines listed in this setting.

- **DefaultMachineName:** This setting is used to indicate the default machine from the list of machine(s) defined. When *MultiHeadFlag* is set to '1' and *ShowMultiHeadDialog* is set to '0', the nesting job will be executed using default parameters of the machine assigned in this setting.

  Example: Consider that you have three machines which support the *Nesting with multiple tools* functionality. First machine is named *SUN360*, second is named *RAK100* and the third *MARS99*. The first machine is to be assigned as the default machine. Then the settings under *[DefaultMachine]* section in the *Machine.ini* file should be as follows:

  MachineCount = 3
  Machine1 = SUN360
  Machine2 = RAK100
  Machine3 = MARS99
  DefaultMachineName = SUN360

The Default Machine Configuration

Observe the *[DefaultMachine]* section of the Machine.ini file. Machine1 (*SingleTHMachine*) is assigned as the default machine. This machine contains only a single tool head and thus represents machines used to execute nesting jobs using a single tool head.

If majority of your nesting jobs are done using single tool head machines, then it is highly recommended you leave Machine1 (*SingleTHMachine*) set as the default machine. Using *SingleTHMachine* as the default machine ensures that all nesting job are executed considering a single tool head. This setting effectively keeps the feature of Nesting with multiple tool heads inactive unless another machine is manually chosen by the user in the *Multi Head Options* tab of the *Create Nesting Job* dialog box.

Note: **If a majority of your nesting jobs are executed with single tool head machines, then it is highly recommended that you do not change the machine (SingleTHMachine) assigned as the Default Machine in the Machine.ini file.**

Defining default parameter values for Machines which support nesting with multiple tools

Default values need to be assigned to the parameters associated with nesting with multiple tools for the machine(s) at your facility.

For every machine listed in the [DefaultMachine] section, these default parameter values is set individually in the **[Machine#]** section. (*# refers to Machine number*)

Assigning default values to parameters associated with nesting with multiple tool heads

1. Open the file named *Machine.ini* located in the CAMWorks Nesting installation folder.
2. Observe any of the sections named *[Machine#]*. (*# refers to Machine number*)
3. For every machine that was listed in the *[DefaultMachine]* section, the default parameters associated with nesting with multiple tool heads are assigned here. For every listed machine, a separate [Machine#] section with default parameter values needs to be created.

   As an example, the parameter values for *[Machine2]* are given below:

```
[Machine2]
;Maximum number of tool heads
MaxNoToolHeads = 5
;Rail direction: X or Y
RailDirection = X
;Tool head distance
ToolHeadDistance-MM = 500
ToolHeadDistance-INCH = 20
;Multi head nesting type: Fixed : 1 , Variable: 2
MultiToolHeadNestingType = 1
```

Typical syntax for assigning default values to parameters associated with nesting with multiple tools

4. These parameters are explained below:
   - **MaxNoToolHeads**: Indicates the maximum number of tool heads available for the machine.

     For example, if the *MaxNoToolHeads* is 4 for a particular machine, then in the *Multi Head Options* tab of the *Create Nesting Job* dialog box, the *Number of tool heads* dropdown list will be populated with integer values in the range of 1 to 4. You can assign the number of tool heads as any number from 1 to 4 by selecting it from the dropdown list.
   - **Rail Direction:** You can assign the default rail direction as 'X' or 'Y'. This default rail direction will be displayed in the *Multi Head*

Options tab of the *Create Nesting Job* dialog box. The default option can be changed within this dialog box.

- **ToolHeadDistance-MM & ToolHeadDistance-INCH:** This value indicates the default minimum tool head distance to be used for nesting with multiple tool heads.

  When the MMGS units are used, CAMWorks Nesting will display the value assigned to *ToolHeadDistance-MM* as the default Tool head distance in the *Multi Head Options* tab of the *Create Nesting Job* dialog box. This default value displayed can be edited within the dialog box.

  When the IPS units are used, CAMWorks Nesting will display the value assigned to *ToolHeadDistance-INCH* as the default Tool head distance in the *Multi Head Options* tab of the *Create Nesting Job* dialog box. This default value displayed can be edited within the dialog box.

- **MultiToolHeadNestingType:** You can choose between *Fixed tool head distance* and *Variable tool head distanc*e for the multi-tool head nesting type. The default multi-tool head nesting type option will be displayed in the *Multi Head Options* tab of the *Create Nesting Job* dialog box in the respective parameter fields. The default option can be changed within the dialog box.

  a. **Fixed tool head distance**: Assign '1' to *MultiToolHeadNestingType* to indicate *Fixed tool head distance* as the default multi-tool head nesting type.

  b. **Variable tool head distance**: Assign '2' to *MultiToolHeadNestingType* to indicate *Variable tool head distance* as the default multi-Tool head nesting type.

Adding a new machine in the Machine.ini file

The following example illustrates how to add a new machine to the list of machines in the *Machine.ini* file.

Example: Consider that three machines named 'SUN360', 'RAK100' and 'MARS99' are already listed in the Machine.ini file with 'SUN360' assigned as the default machine. You wish to add a new machine with the name 'SKY444' to this list and assign it as the default machine. The default parameters to be assigned to this machine are as follows:

- ➔ Max number of tool heads: **6**
- ➔ Default Rail Direction: **Y**
- ➔ Tool Head Distance (in millimeters): **125mm**
- ➔ Tool Head Distance (in inches): **5 inches**
- ➔ Default multi-tool head nesting type: **Variable tool head distance**

Solution:

1. Open the *Machine.ini* file.
2. Go the section named *[DefaultMachine]* and make the following changes:

```
[DefaultMachine]
MachineCount = 3
Machine1 = SUN360
Machine2 = RAK100
Machine3 = MARS99
DefaultMachineName = SUN360
```

[DefaultMachine] before changes

```
[DefaultMachine]
MachineCount = 4
Machine1 = SUN360
Machine2 = RAK100
Machine3 = MARS99
Machine4 = SKY444
DefaultMachineName = SKY444
```

[DefaultMachine] after changes

3. Next, after the *[Machine3]* section in the Machine.ini file, add a new section named *[Machine4]* with the following values assigned to its parameters:

```
[Machine4]
;Maximum number of tool heads
MaxNoToolHeads = 6
;Rail direction: X or Y
RailDirection = Y
;Tool head distance
ToolHeadDistance-MM = 125
ToolHeadDistance-INCH = 5
;Multi head nesting type: Fixed : 1, Variable : 2
MultiToolHeadNestingType = 2
```

Assigning default parameter values to the new machine added in the Machine.ini file

4. Save the changes made to *Machine.ini* file. The new machine will now be added to the list of machines. In the *Multi Head Options* tab of the *Create Nesting Job* dialog box, this machine will be available in the dropdown list of available machines.

SECTION THREE

CAMWorks Nesting Tutorials

TUTORIAL 2- SINGLE PART, SINGLE SHEET NESTING FOR A SOLID PART

Introduction

This tutorial explains how to nest a solid part in a sheet layout. You will also learn how to nest the part using CAMWorks Nesting commands that automatically nest multiple instances of the part on a pre-defined sheet and generates a best fit resulting in high sheet utilization and minimal scrap.

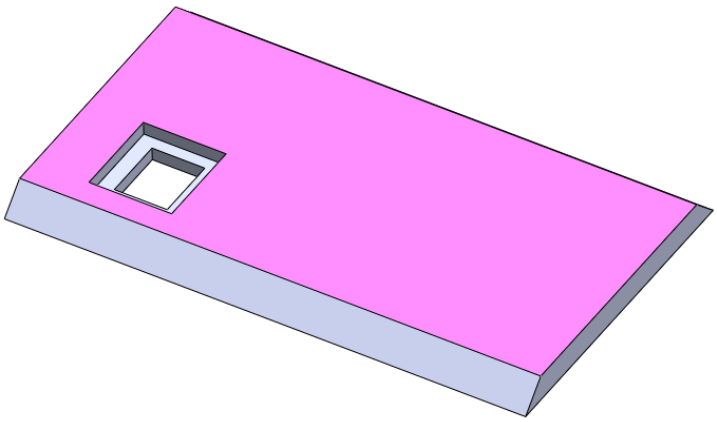
Topic covered in this Tutorial:

- Selecting the part to be nested
- Setting part parameters such as thickness, quantity, material, grain direction and rotation angle.
- Defining sheet size of type *Standard Size*
- Selecting the *Normal* Direction
- Selecting the output assembly file

STEP 1: Open the Part

1. Load the *CAMWorks Nesting Add-In* in SOLIDWORKS or CAMWorks Solids.
2. Open the part file **Tutorial_2.sldprt** in the following folder location.

   Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Parts



Tutorial_2.sldprt

STEP 2: Define the Part Parameters

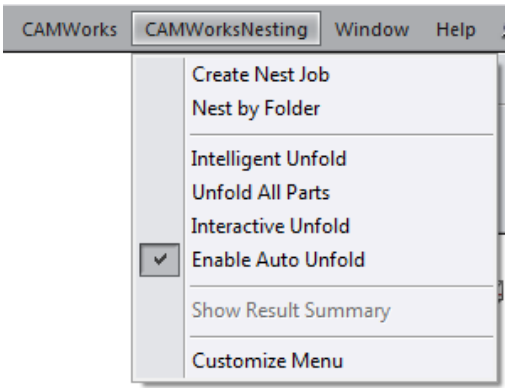
1. Select *Create Nest Job* from the CAMWorksNesting menu bar.

   OR

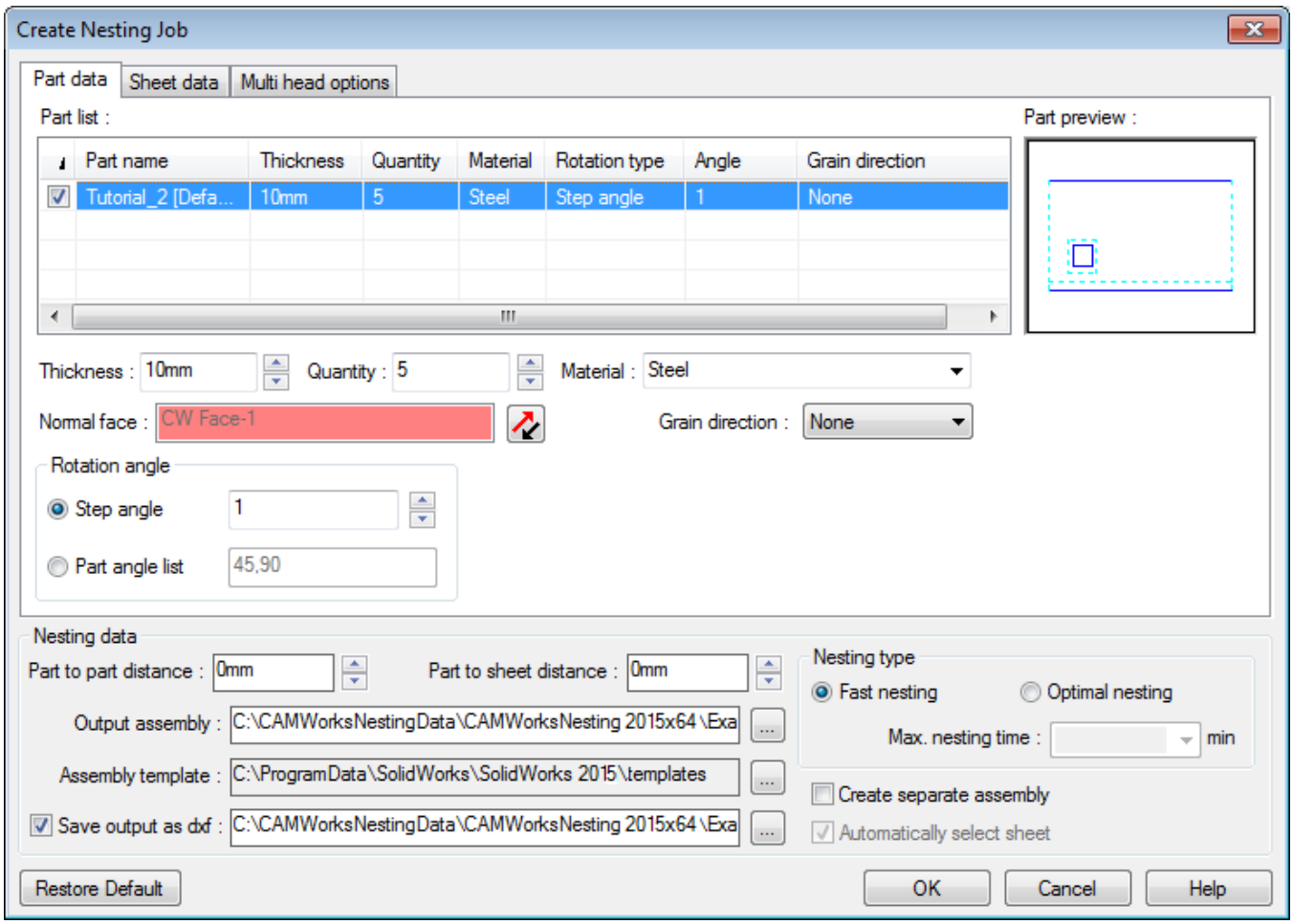
   Click on the *Create Nesting Job* button on the CAMWorks Nesting Ribbon Bar.

2. The *Create Nesting Job* dialog box is displayed. Use the *Part data* tab of this dialog box to set the parameters for the part.

3. The solid part *Tutorial_2.sldprt* is listed in the *Part List* along with its nesting parameters.



Select *Create Nesting Job* in the CAMWorksNesting menu



The Part Data tab of the Create Nesting Job dialog box

Assign the following values to the following Part Parameters:

a) Thickness: CAMWorks Nesting extracts the part parameter of Thickness from the Solid Part and displays it in the Thickness field

as default thickness for the part. The thickness of the part, as extracted from the solid part, is displayed as 10 mm.

b) Material: CAMWorks Nesting extracts the material info from the Solid Part and displays it in the material field. The material for this part, as extracted from the solid part, is Steel.

c) Quantity: The default quantity value is displayed in the Quantity field *(As per default value defined in the DefaultValues.ini file)*. Double click on the 'Quantity' field in the Part list. Set the Part Quantity to '100'.

d) Angle: Double on the *Angle* column of the *Part List*. Edit and assign an angle of 90 degrees.

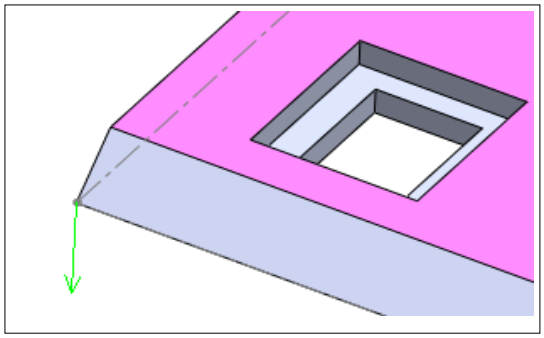
e) Grain Direction: Leave the *Grain direction* set to *None*.

f) Normal Face: By default, CAMWorks Nesting chooses the face with the largest number of features. So the bottom face of the solid part is chosen by default. The normal direction is indicated by an arrow in the graphics area. To chose the top face (indicated by pink color) as the normal face, do either of the following:

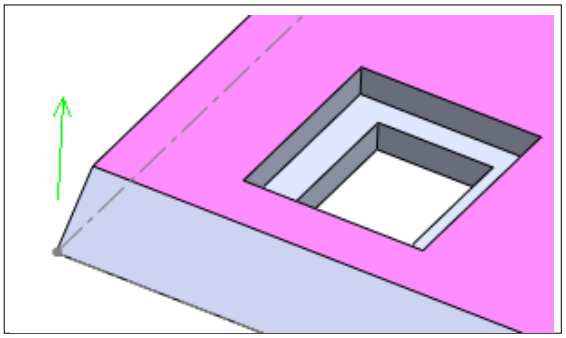
i. Click on the *Reverse* button 

ii. In the graphics area, click on the top most face of the solid part (face in pink color)

Observe that the arrow indicating the normal direction changes accordingly.



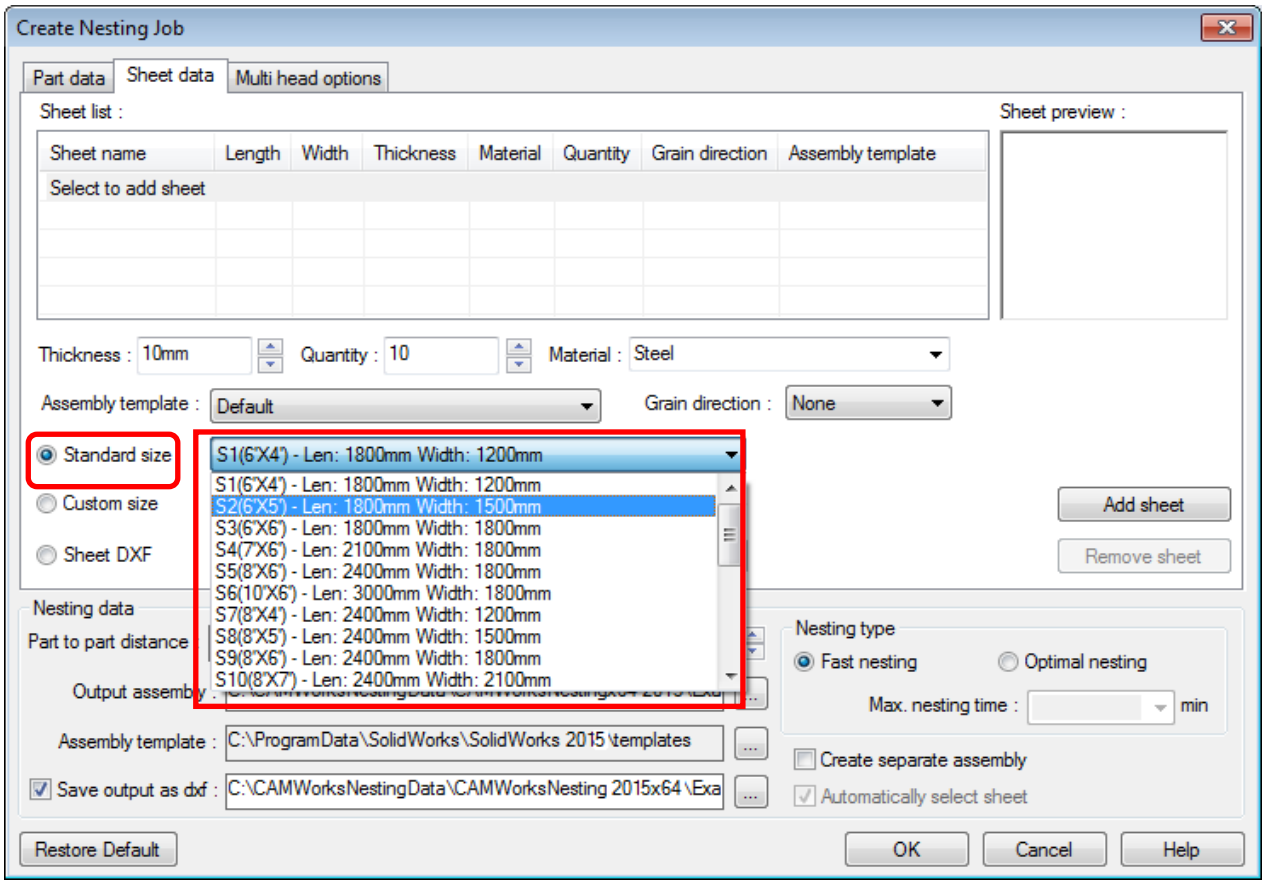
Normal direction when bottom face is selected



Normal direction when top face is selected

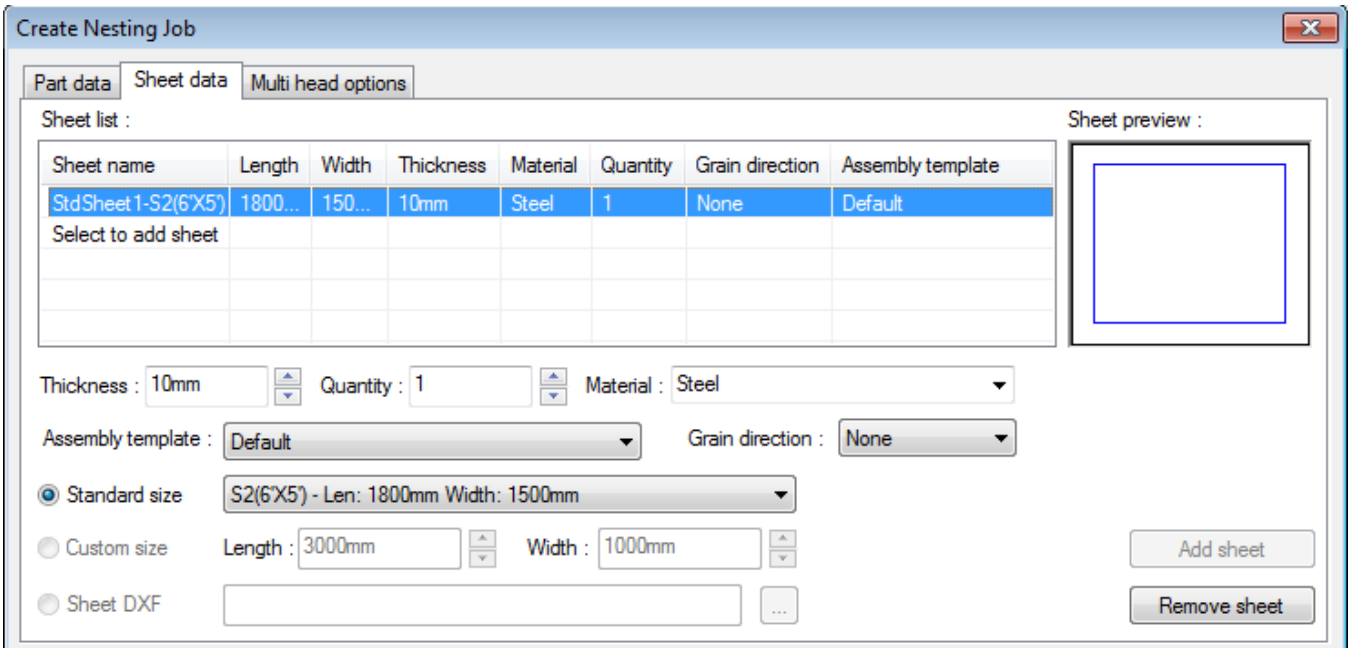
STEP 3: Define the Sheet Parameters & adding a standard sheet

1. Click on the *Sheet Data* tab. Observe that the assigned thickness and material of the sheet are identical to those of the part to be nested.
2. Set the sheet *Quantity* to '1'.
3. In this tutorial, we will nest the part using a *Standard size* sheet. Click on the Standard size dropdown list. Observe the Standard Sheet sizes defined in the *Standard size* dropdown list.



Standard size sheets dropdown list

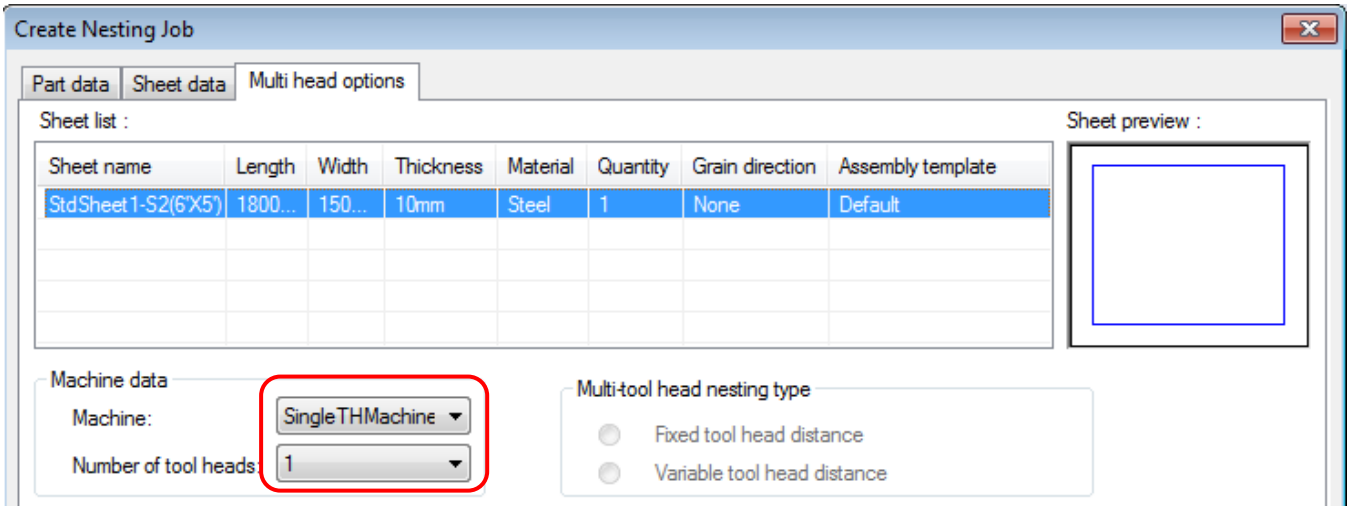
4. Observe that the standard sizes defined in the *Standard Sheets.ini* file are listed in the dropdown list. In this example, we will choose the second sheet displayed in the list. (with Length = 1800 mm & Width = 1500 mm)
5. Click *Add Sheet* button. The sheet is added to the *Sheet list*.



Selected sheet added to Sheet list

STEP 4: Selecting a machine with Single Tool Head for the Nesting Process

1. Click on the *Multi Head Options* tab.
2. In the *Machine Data* group box, ensure that the Machine selected is *SingleTHMachine*. The *Number of tool heads* for this machine should be '1'.

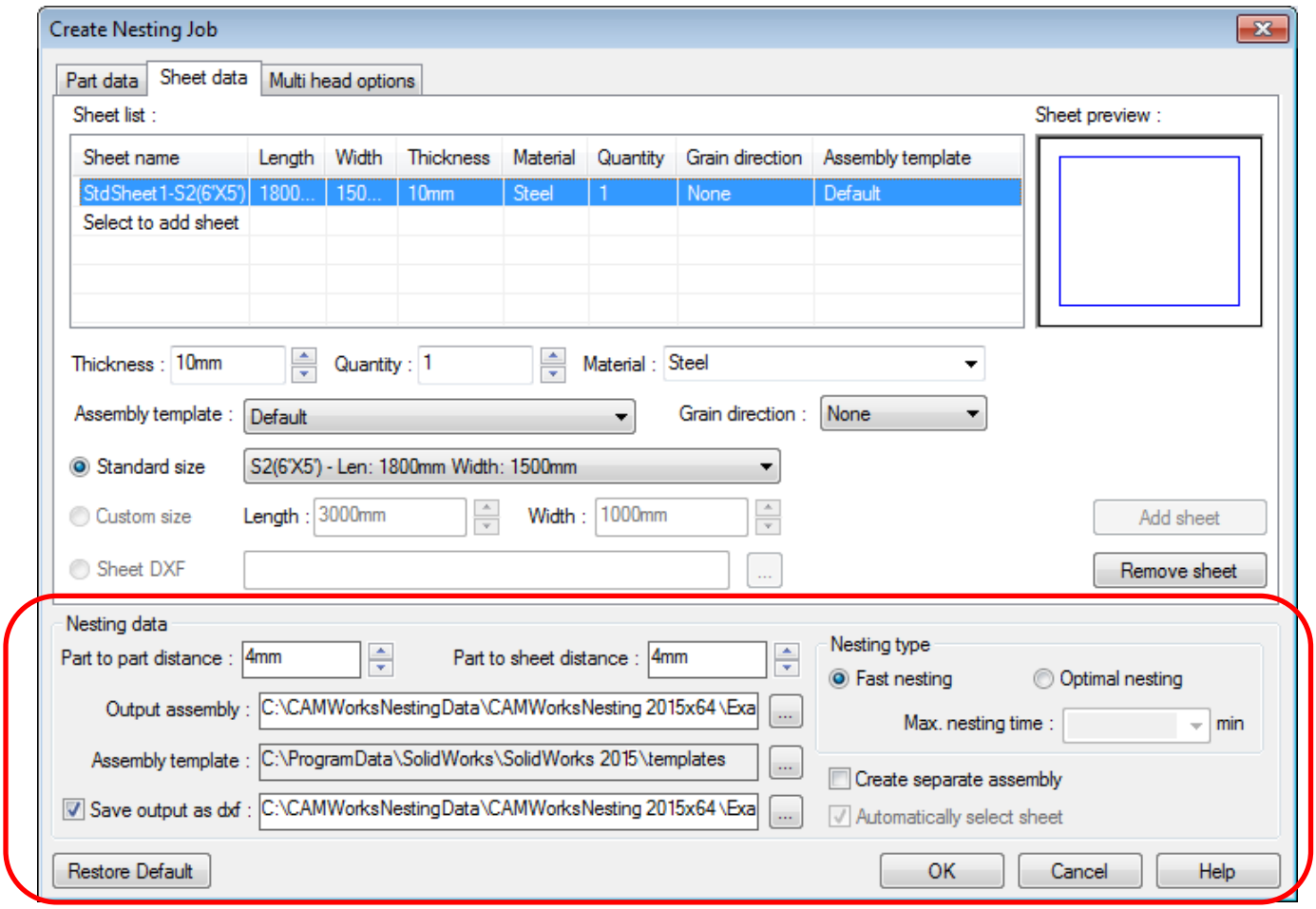


Selecting SingleTHMachine as the machine in the Multi Head Options tab

Selecting *SingleTHMachine* as the machine ensures the nesting job is executed considering a single tool head and not multiple tool heads.

STEP 5: Define Nesting Parameters

1. Observe the *Nesting Data* Group Box. For this tutorial, set a *Part to Part distance* of 4 mm and a *Part to Sheet distance* of 4mm.
2. Use the ... button to specify the folder location where the Output Assembly file and Summary text file will be saved.
3. Leave the checkbox *Save output as dxf* checked. Use the ... button next to this checkbox to assign the folder location where the nested layouts generated will be saved in the *.dxf* file format.
4. Set the Nesting method to *Fast Nesting*.



STEP 6: Generating the Nested Layout

1. After setting the part, sheet and nesting parameters, click *OK*. This sets into motion the process to generate a nested layout.
2. After the Nesting process is completed, CAMWorks Nesting will display a message indicating the location of the text file containing the **summary of the Nest Results**. Click *OK* to close the message. The Text file will be displayed.
3. The **Nested layout assembly** will be displayed in the Graphics area. Both the summary file and the assembly files are saved in the location indicated Output Assembly File path stated in the *Create Nesting Job* dialog box.
4. Browse to the folder location specified for saving the nested layouts in the .dxf format. Observe that the nested layout in .dxf file format is saved in the folder.

In this tutorial, we will observe the 3 nesting results:

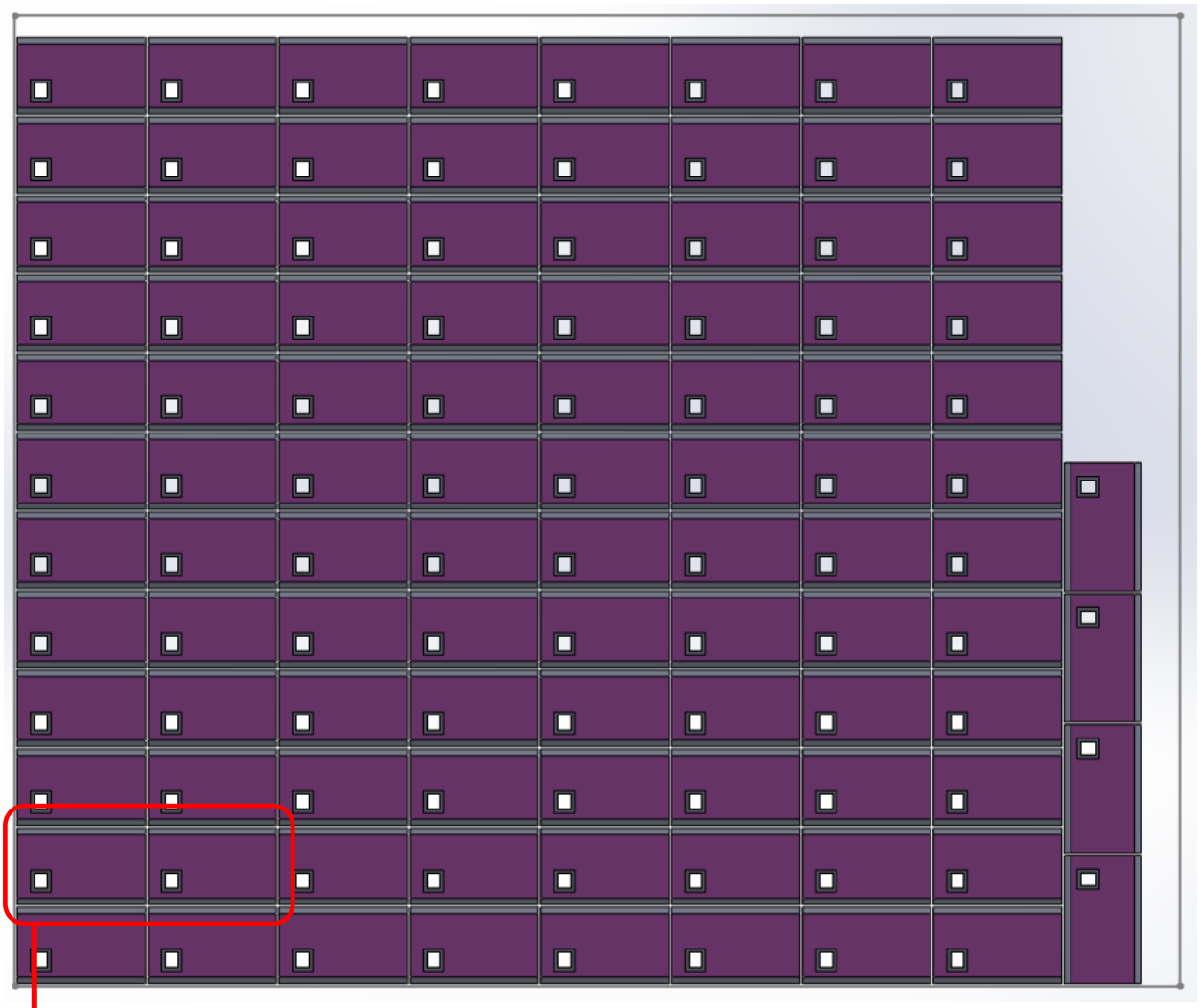
i. Nesting layout generated when top face of the part is chosen as normal face

ii. Nesting layout generated when bottom face of the part is chosen as normal face

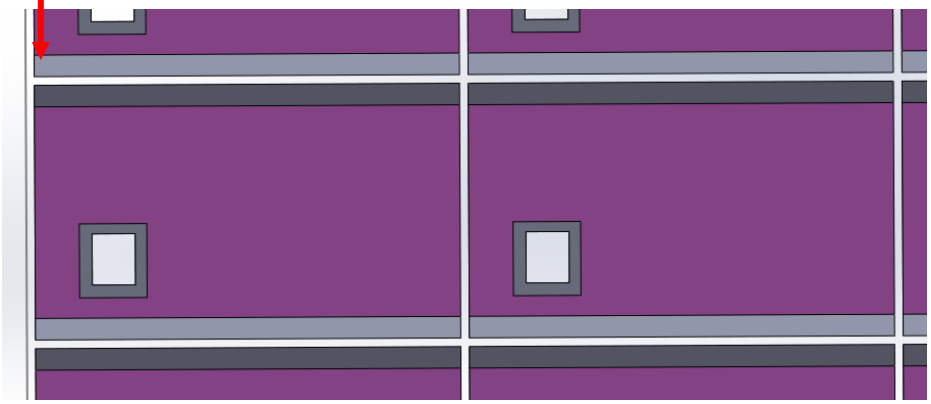
iii. Nesting layout when Grain direction is set for part and the sheet.

Result A

Follow all the above steps and view the Nested layout. Observe that all the 100 instances of the part (specified quantity) have been nested.



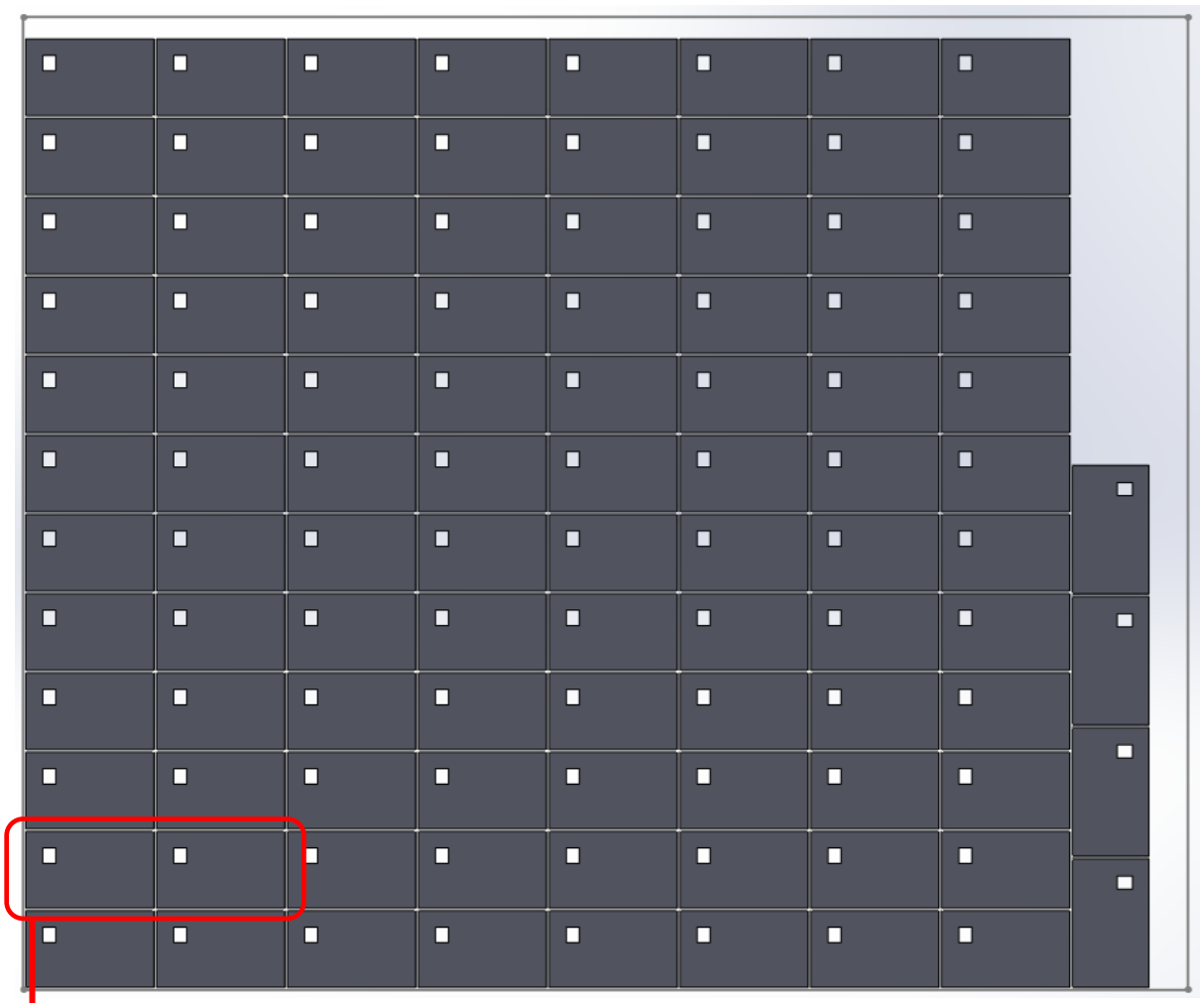
Result A: Nest Result obtained with the top face of the part chosen as normal face



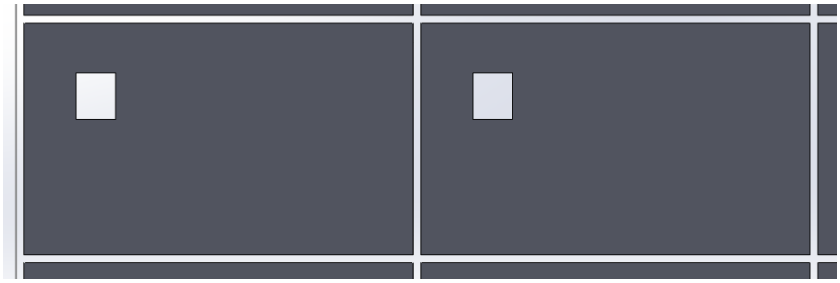
Close up view of the Nesting Layout

Result B

Repeat all the steps listed in this tutorial without changing the default Normal direction (Step2-4-f). To set the previous normal direction, select the bottom face (gray-colored face) of the part in the graphics area when the *Create Nesting Job* dialog box is displayed and the *Part Data* tab is the active tab. View the nesting layout. Observe that all the 100 instances of the part (specified quantity was 100) have been nested.



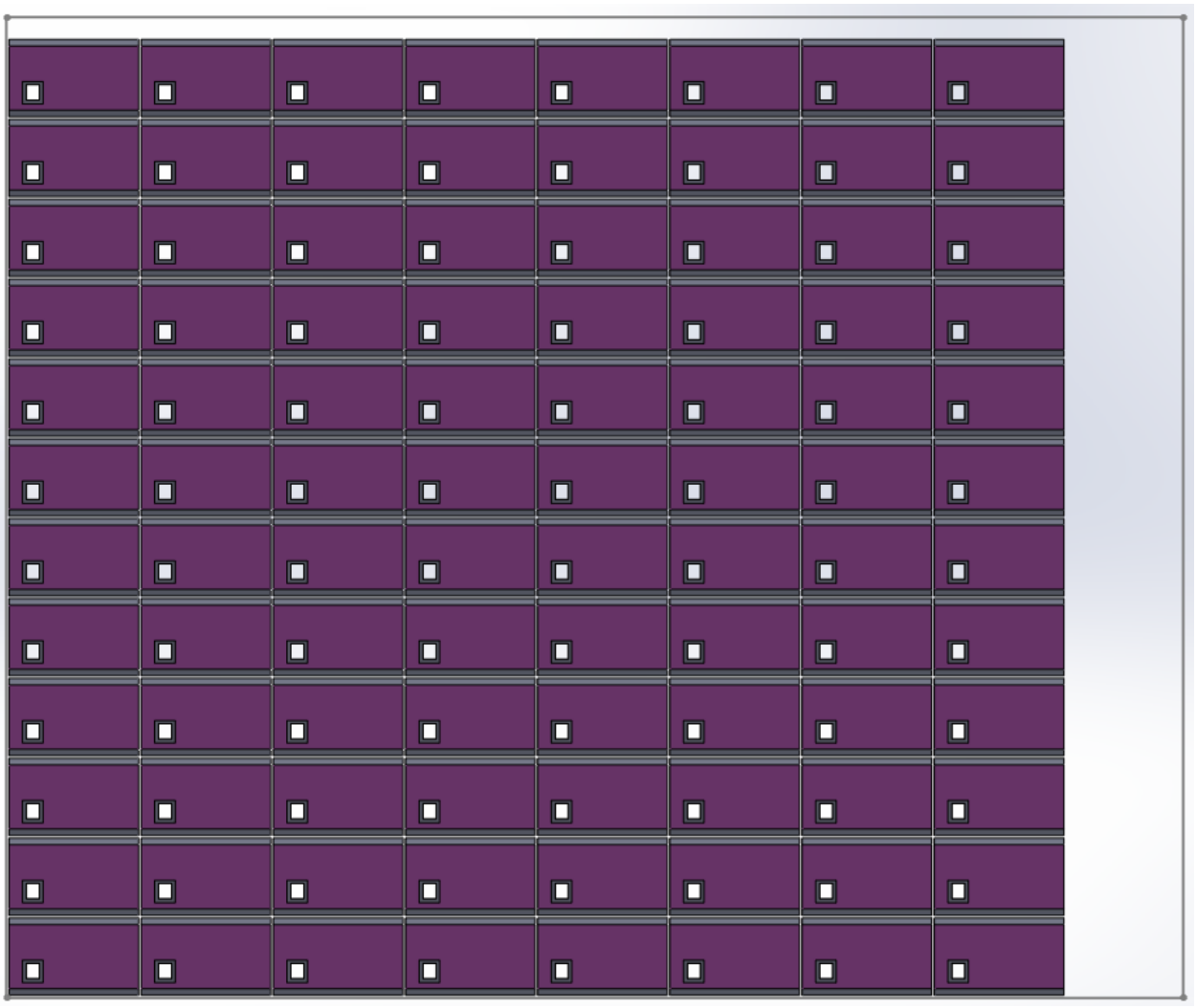
Result B: Nest Result obtained with the bottom face of the part chosen as normal face



Close up view of the Nesting Layout

Result C

Repeat all the steps listed in this tutorial. However, this time, in Step 2-4-e, set the Grain Direction of the Part to X direction. In the *Sheet Data* tab, set the Grain Direction of the Sheet to X direction. Execute Nesting. View the nesting layout. All the parts are nested along the specified grain direction. Observe that only 96 instances of the part are nested while the quantity specified was 100. The same result will be obtained if the Grain direction of both the part and sheet are set to Y direction.



Result C: Nest Result obtained with the top face of the part chosen as normal

TUTORIAL 3 – SINGLE PART, SINGLE SHEET NESTING FOR SHEET METAL PART

Introduction

This tutorial explains how to nest a sheet metal part in a sheet layout. You will also learn how to nest the part using CAMWorks Nesting commands that automatically nests multiple instances of the part on a pre-defined sheet and generates a best fit resulting in high sheet utilization and minimal scrap.

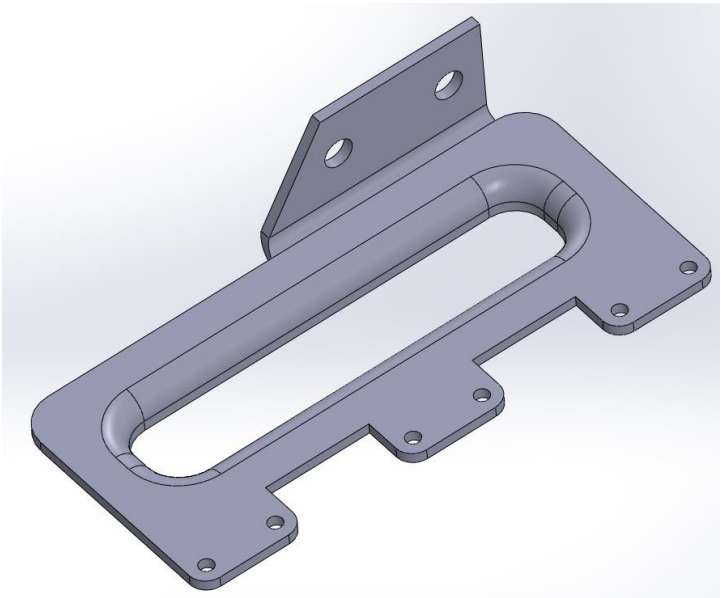
Topic covered in this Tutorial:

- Selecting the sheet metal part to be nested
- Setting user-defined material for the part.
- Setting the Angle List
- Defining sheet size of type 'Custom Size'

STEP 1: Open the Part

1. Load the *CAMWorks Nesting Add-In* in SOLIDWORKS/CAMWorks Solids.
2. Open the part file **Tutorial_3.sldprt** in the following folder location.

   Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Parts



Tutorial_3.sldprt

STEP 2: Change in Configuration File settings

Enabling the option to Flatten Sheet Metal Part

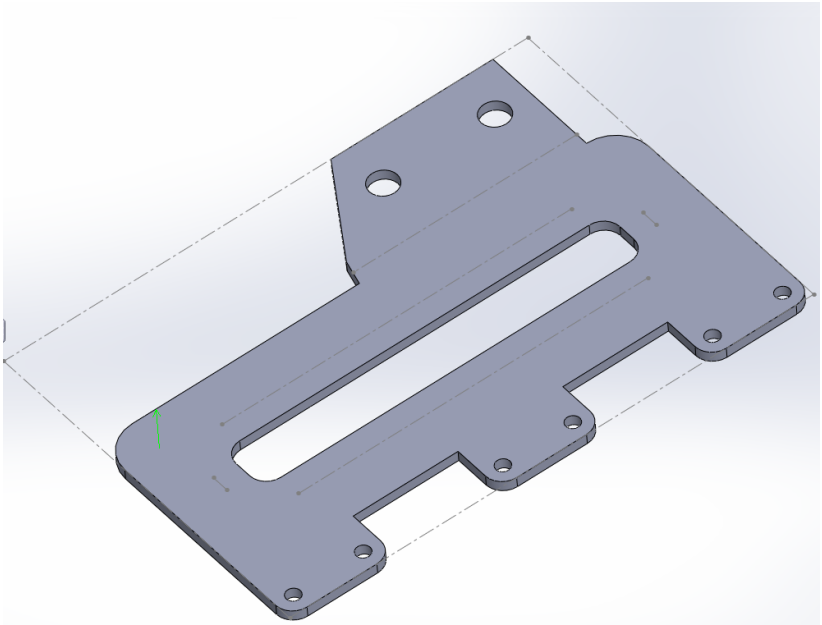
In this tutorial, you will nest the sheet metal part based on its dimensions after flattening. The default settings configured in the *DefaultValues.ini* file ensure that sheet metal parts are flattened before the nesting job is executed. If you are unsure about the settings, open the *DefaultValues.ini* file and set the *FlattenSheetMetalPart* flag to '1' in order to activate the option of flattening.

Enabling the Fix Component Feature of SOLIDWORKS

In the configuration file DefaultValues.ini (located within the CAMWorks Nesting Installation folder), ensure that the flag **FixComponent** under [*NestingData*] section is set to '1'.
This setting enables the SOLIDWORKS Fix Component feature which will ensure that after the Nested layouts are generated, the parts in the Nested layout assembly do not get accidentally repositioned.

STEP 3: Define the Part Parameters

1. Select 'Create Nest Job' from the CAMWorksNesting menu bar.
2. The Create Nesting Job dialog box opens. Observe that the sheet metal part Tutorial_3.sldprt displayed in the graphics area is automatically flattened.



Tutorial_3.sldprt after flattening

3. In the Part Data tab, set the following nesting parameters:

a) **Thickness:** The thickness of the sheet metal part, as extracted from the solid part is 3mm. In this tutorial, no changes are made to the thickness.

b) **Material:** Since Material related information is not defined for this sheet metal part, CAMWorks Nesting will display the first material (Steel) in the Material drop down list as the default material. In this tutorial, we will assign a material 'Chrome Steel' which is not part of the Material Dropdown list. To assign 'Chrome Steel' as the material, following are the steps:

i. In the Part List, highlight the part for which material is to be assigned.

ii. In the Material combo box (located below the Part List), enter the material name as 'Chrome Steel'.

iii. Shift focus. Observe that the Material of the part is updated in the Part List.

c) **Quantity:** Set the Part Quantity to '125'.

d) **Angle:** Set a step angle of 90^0.

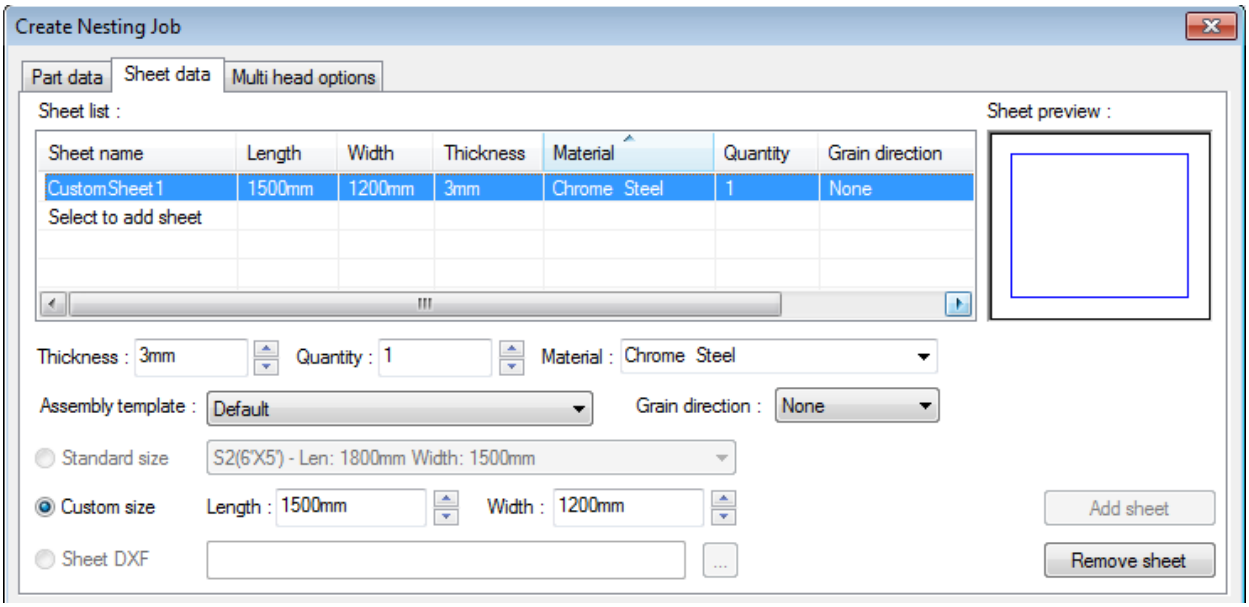
e) **Grain Direction:** Leave the Grain direction is set to 'None'.

f) **Normal Face:** No changes are made to the default normal face selection.

STEP 4: Defining a 'Custom' size sheet

1. Click on the Sheet Data tab. Observe that the assigned thickness and material of the sheet are identical to those of the part to be nested.
2. In this tutorial, a custom sheet will be used to nest the parts:
   a) Select 'Custom Size'. The Length and Width fields will be activated and will display default values as defined in the DefaultValues.ini file.
   b) Assign a length of 1500mm and a width of 1200 mm.
   c) Set the Sheet quantity to 1.
   d) Some intelligence is added in CAMWorks Nesting such that it ensures the sheets with relevant **material and thickness** is available for nesting the part. CAMWorks Nesting automatically extracts the thickness and material of the first part in the part list and assigns these as the default value of the first sheet. Observe that the material field displays 'Chrome Steel' and thickness field displays 3 mm.
   e) Click *Add Sheet* to add the sheet to the *Sheet List*.
   f) In the sheet List, the Sheet name of the added Custom Sheet can be changed as required by double-clicking on the sheet name in the sheet list.

g) Sheet name in the Sheet Name column of the Sheet List. Assign a new name Custom(1500'X1200')



Defining a Custom Sheet

STEP 5: Selecting a machine with Single Tool Head for the Nesting Process

1. Click on the *Multi Head Options* tab.
2. In the *Machine Data* group box, ensure that the Machine selected is *SingleTHMachine*. The *Number of tool heads* for this machine should be '1'.

   Selecting *SingleTHMachine* as the machine ensures the nesting job is executed considering a single tool head and not multiple tool heads.

STEP 6: Define Nesting Parameters

1. In the Nesting Data Group Box, set a *Part to Part distance* of 3 mm and a *Part to Sheet distance* of 2 mm.
2. Use the ... button next to the *Output assembly* field to specify where the nested layout and the Summary text file that are generated will be saved.

STEP 7: Generating the Nested Layout

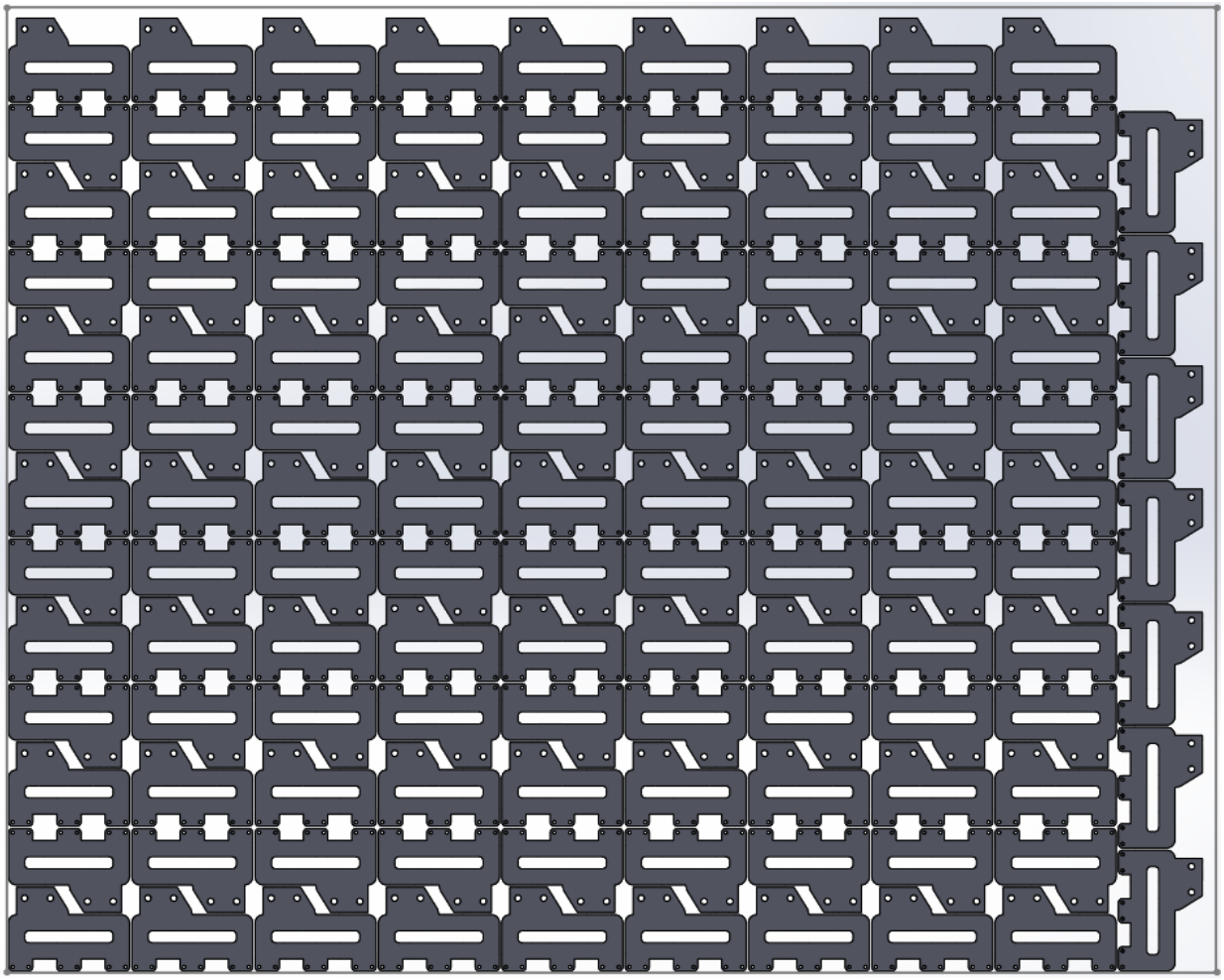
a. Select *Fast Nesting* as the Nesting method. Click *OK*.

b. Read the Results Summary text file. It indicates that 124 instances of the part required are nested.

NestAssm-Tutorial_3.SLDASM.ResultsSummary.txt - Notepad

File Edit Format View Help

```
Nest Results Summary
--------------------

Nested Qty              Total Qty              Part Name
---------------         ------------           ---------
124                     125                    Tutorial_3 [Default]
---------------         ------------           ---------

Total Number of Part Types Nested              1

Total Number of Part Instances Nested          124

Total Number            Utilisation            Part Instance          Sheet Name
---------------         -------------          ------------           ------------
1                       66.600                 124                    CustomSheet1 (1)

Total Number of Sheets Nested   1

Overall Utilization             66.600
```



Nesting layout obtained for Tutorial_3.sldprt

TUTORIAL 4 – NESTING OF MULTIPLE PARTS BASED ON THICKNESS

Introduction

This tutorial explains how to nest multiple solid parts of varying thicknesses. You will observe how CAMWorks Nesting generates a multiple layout based on the part thickness in a single Nesting job.

Topic covered in this Tutorial:

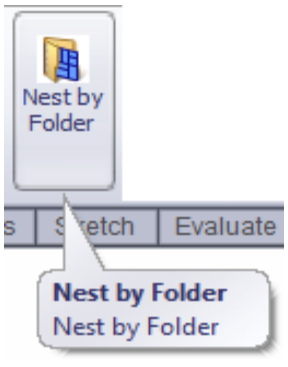
- Using the *Nest by Folder* option
- Assembly Nesting of multiple parts
- Selectively nest few parts in the Part List
- Defining a sheet using a DXF file
- Nesting multiple parts of varying thickness on sheets of corresponding thickness
- Nesting of multi-body parts and assemblies containing multi-body parts.

STEP 1: Using 'Nest by Folder' to open the Assembly

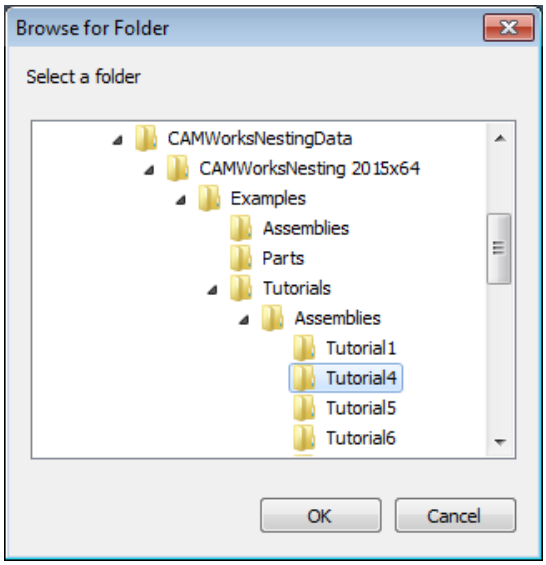
1. Select ***Nest by Folder*** option in the CAMWorks Nesting menu bar.

   OR

   Click on the Nest by Folder button on the CAMWorks Nesting Ribbon Bar.



CAMWorks Nesting Ribbon Bar



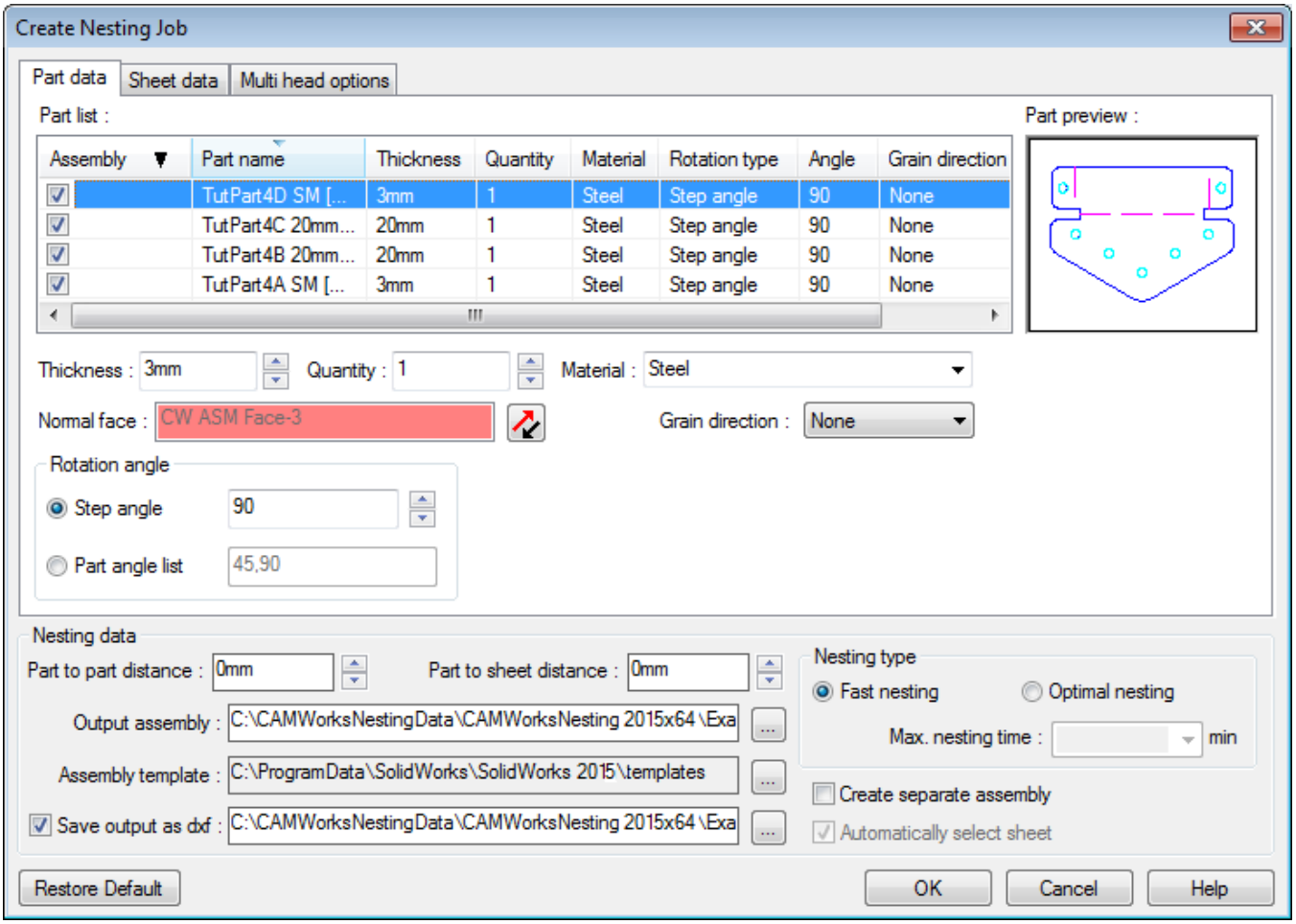
'Browse' for Folder dialog box

2. The *Browse for folder* dialog box opens. Browse to the folder named *Tutorial4* in the following folder location.

   Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies\Tutorial4

3. CAMWorks Nesting opens all the parts contained in the folder as an assembly in the SOLIDWORKS Graphics area.
4. The *Create Nesting Job* dialog box is displayed. All the parts present in the folder are listed in the *Part List* of the *Part data* tab.

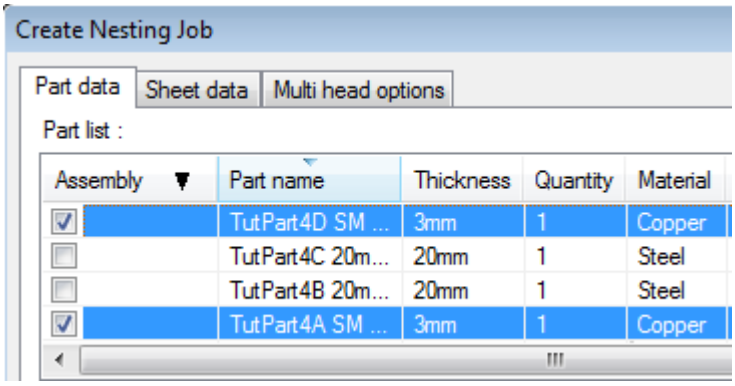
STEP 2: Define the Part Parameters



'Nest by Folder' parts listed in the Part List

Selectively Nesting Parts

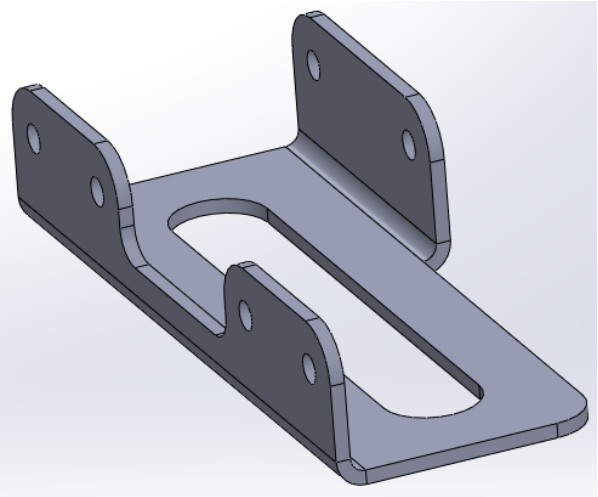
In the Part data tab, observe the Part name column of the Part List. Every listed part has a checkbox to its right which is selected. Such a selected checkbox indicates that the associated part will be taken up for Nesting during the Nesting process. To selectively nest only certain parts in the Part list,



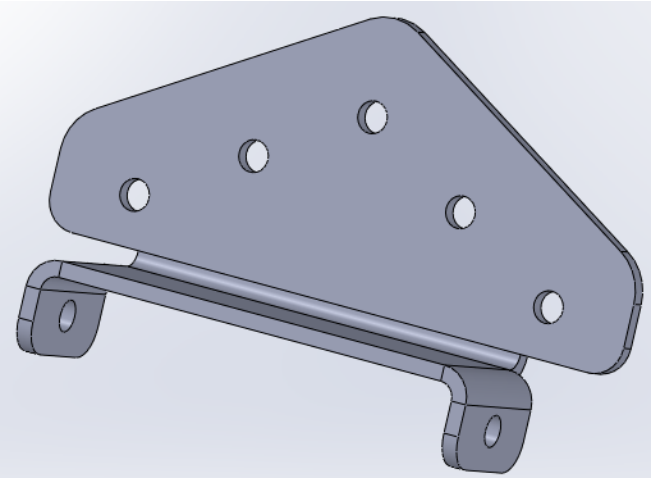
Deselecting parts which are not be nested

deselect the checkbox of those parts which you do not want to nest.

In this tutorial, we will initially nest only the parts 'TutPart4A SM' and 'TutPart4D SM'. Both of these are sheet metal parts. Hence, in the Part list, uncheck the checkboxes given against the parts 'TutPart4B SM' and 'TutPart4C SM'.



TutPart4A SM



TutPart4D SM

Material

Since Material related information is not defined for this sheet metal part, CAMWorks Nesting will display the first material in the Material drop down list as the default material (Steel). In this tutorial, we will assign a material *Copper* to all the parts. This material is listed in the Material Dropdown list. Select TutPart4A and TutPart4D by pressing ctrl key and assign *Copper* from the Material dropdown list.

Normal Face

No changes are made to the default normal face selection for any of the parts.

Grain Direction

Leave the Grain direction set to *None*.

Step Angle

Assign a Step Angle of 90^0 to all the parts.

Quantity

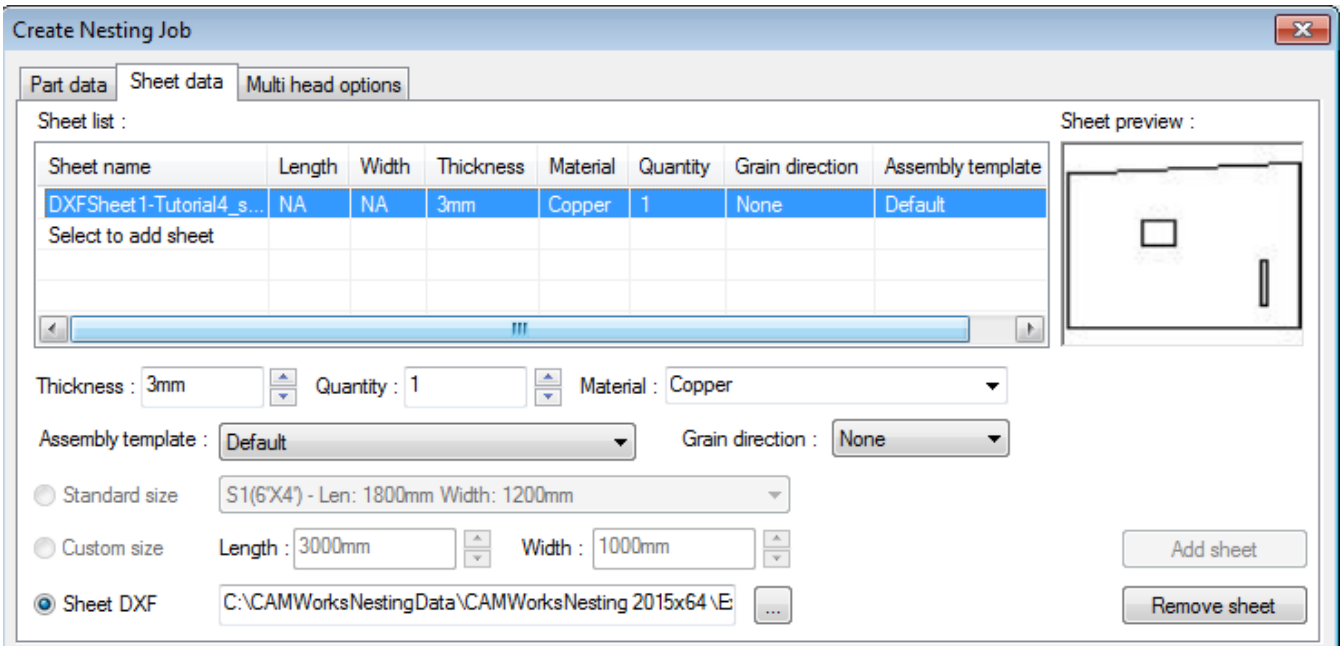
Assign a quantity of '62' to both the parts to be nested.

STEP 3: Adding a sheet of using 'DXF' file.

In previous tutorials, we learned how to add Standard size and Custom size sheets. In this tutorial, we will use a file in .dxf format to define the sheet.

Following are the steps to define a sheet using a file in .dxf format:

1. Under the Sheet data tab, select the option *Sheet DXF*.
2. This activates the field used to indicate the path of the DXF file. Click on the ... button to browse to the folder containing the DXF file.
3. Select the .dxf format file named 'Tutorial4_sheet.dxf' from following location.
   Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\ Sheets
4. CAMWorks Nesting populates the thickness and material field for each prospective sheet to be added to the sheet list based on the serial order of the parts listed in the part tab. Hence, by default, the Thickness field and the Material field will display the values of the first selected part listed in the part list. In this tutorial, the thickness and material of the first part (3mm and Copper respectively) will be displayed as default values.
5. Assign Sheet quantity as '1' and Grain direction as *None*.
6. Click *Add sheet* to add the file in .dxf format to the Sheet List.



DXF file added to sheet list

7. The file in .dxf format is added to the Sheet List. The Sheet preview indicates that this sheet is a remnant (remainder) sheet.

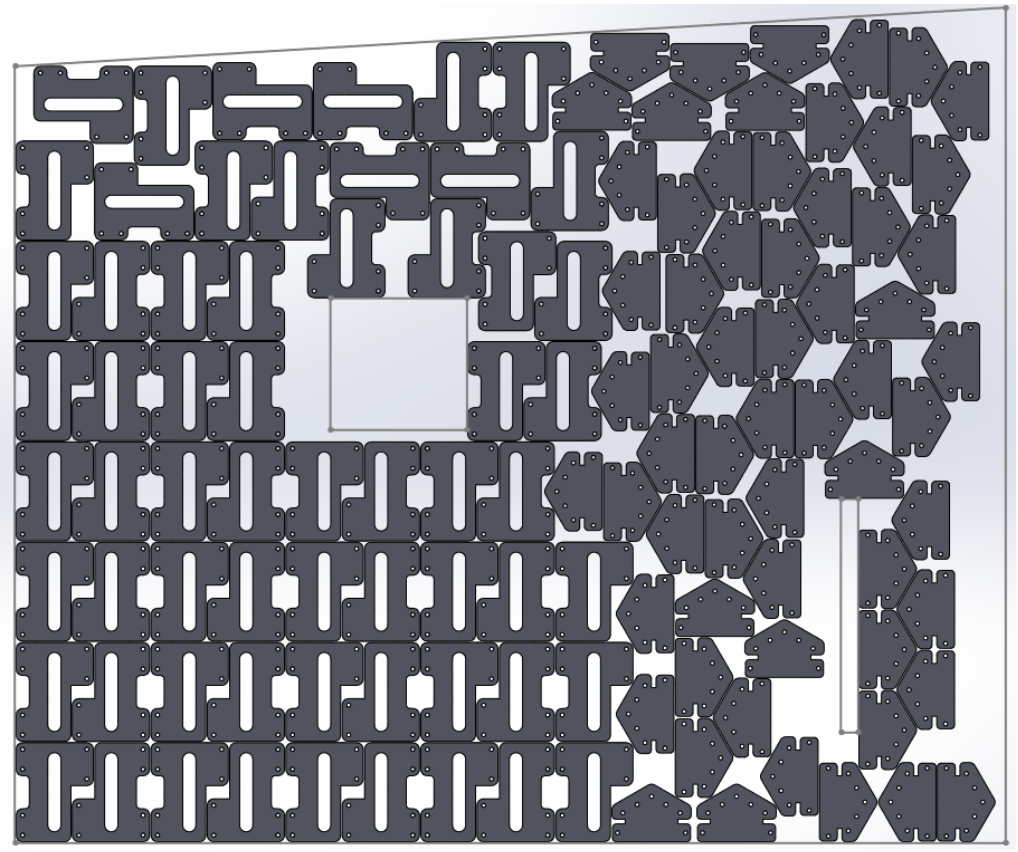
STEP 4: Selecting a machine with Single Tool Head for the Nesting Process

1. Click on the *Multi Head Options* tab.
2. In the *Machine Data* group box, ensure that the Machine selected is *SingleTHMachine*. The *Number of tool heads* for this machine should be '1'.

   Selecting *SingleTHMachine* as the machine ensures the nesting job is executed considering a single tool head and not multiple tool heads.

STEP 5: Define Nesting Parameters

1. In the Nesting Data Group Box, set a Part to Part distance of 2mm and a Part to sheet distance of 2mm.
2. Under Nesting type, select *Fast Nesting* as the Nesting method.
3. Use the  button next to the *Output assembly* field to specify where the nested layouts and the Summary text file that are generated will be saved.
4. Click *OK* to execute the nesting process.
5. Observe the Nested layout. The assigned quantities of both parts have been nested.

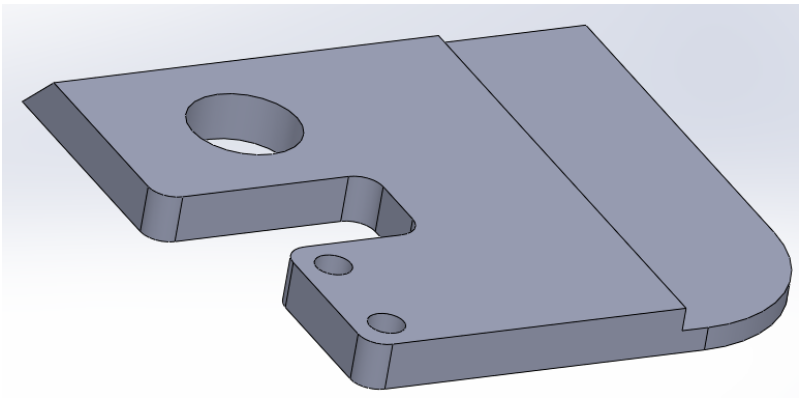


Nested Layout in the DXF sheet

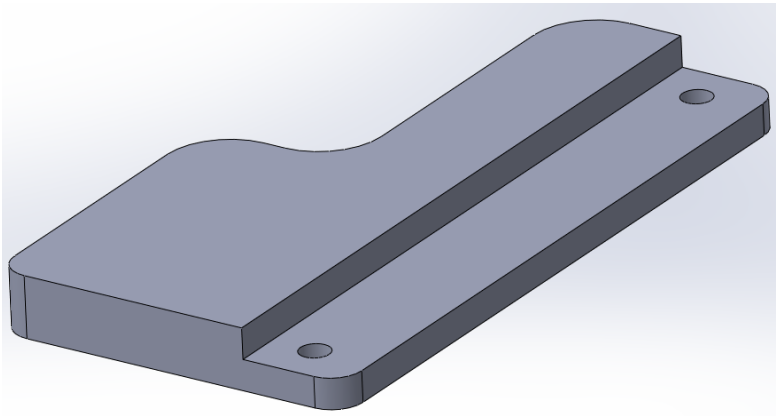
STEP 6: Nesting all the Parts in the Assembly

In the following section, we will learn how to nest parts of varying thickness in a single nesting job.

1. Close the generated assembly file. The four parts are still displayed in the SOLIDWORKS Graphics area. Select *Create Nesting Job* from the CAMWorksNesting menu. The *Create Nesting Job* dialog box is displayed.
2. Observe the Part list. As observed in Step 2 of this tutorial, two of the parts viz. 'TutPart4A SM [Default]' and 'TutPart4D SM [Default]' are sheet metal parts of 3mm thickness each. The other two parts, 'TutPart4B SM [Default]' and 'TutPart4C SM [Default]' are solid parts of thickness 20 mm each.



TutPart4B SM.sldprt



TutPart4C SM.sldprt

3. Double click on thickness header in the part list; it automatically arranges the parts based on thickness as shown below. Set the following quantities for the parts displayed in the Part list:

Part Name	Part Quantity	Thickness
TutPart4B 20mm.SLDPRT	30	20 mm
TutPart4C 20mm.SLDPRT	20	20 mm
TutPart4A SM.SLDPRT	64	3 mm
TutPart4D SM.SLDPRT	122	3 mm

4. Set the material, grain direction, normal face and angle with the values as given in Step 2.
5. All the parts now have the same material but, as observed in the above table, two parts have a thickness of 20mm while the other two have a thickness of 3 mm. Hence, at least two sheets with a thickness of 20mm and 3mm respectively will be required to nest these parts. In this tutorial, we will use two standard sheets of size 'S1 (6' X 4')', each assigned the appropriate thickness to nest these parts.

Adding a standard sheet

Following are the steps to add a standard sheet for this tutorial:

i. Click on the *Sheet Data* tab. In the Sheet list, click on *Select to add sheet*.

ii. To add a standard sheet, select the 'S1 (6' X 4') – Len: 1800 mm Width: 1200 mm' sheet from the Standard Size dropdown List.

iii. CAMWorks Nesting populates the thickness and material field for each sheet to be added to the sheet list based on the serial order of the parts listed in the part tab. Hence, by default, the Thickness field and the Material field will display the values of the first part listed in the part list. In this tutorial, the thickness and material of the first part is 3mm and Copper respectively. In case this value is not displayed in the fields, assign the appropriate values.

iv. In the *Quantity* field, assign a quantity of '1'.

v. Leave the *Grain Direction* set to *None*.

vi. Click *Add Sheet* to add the sheet to the Sheet list.

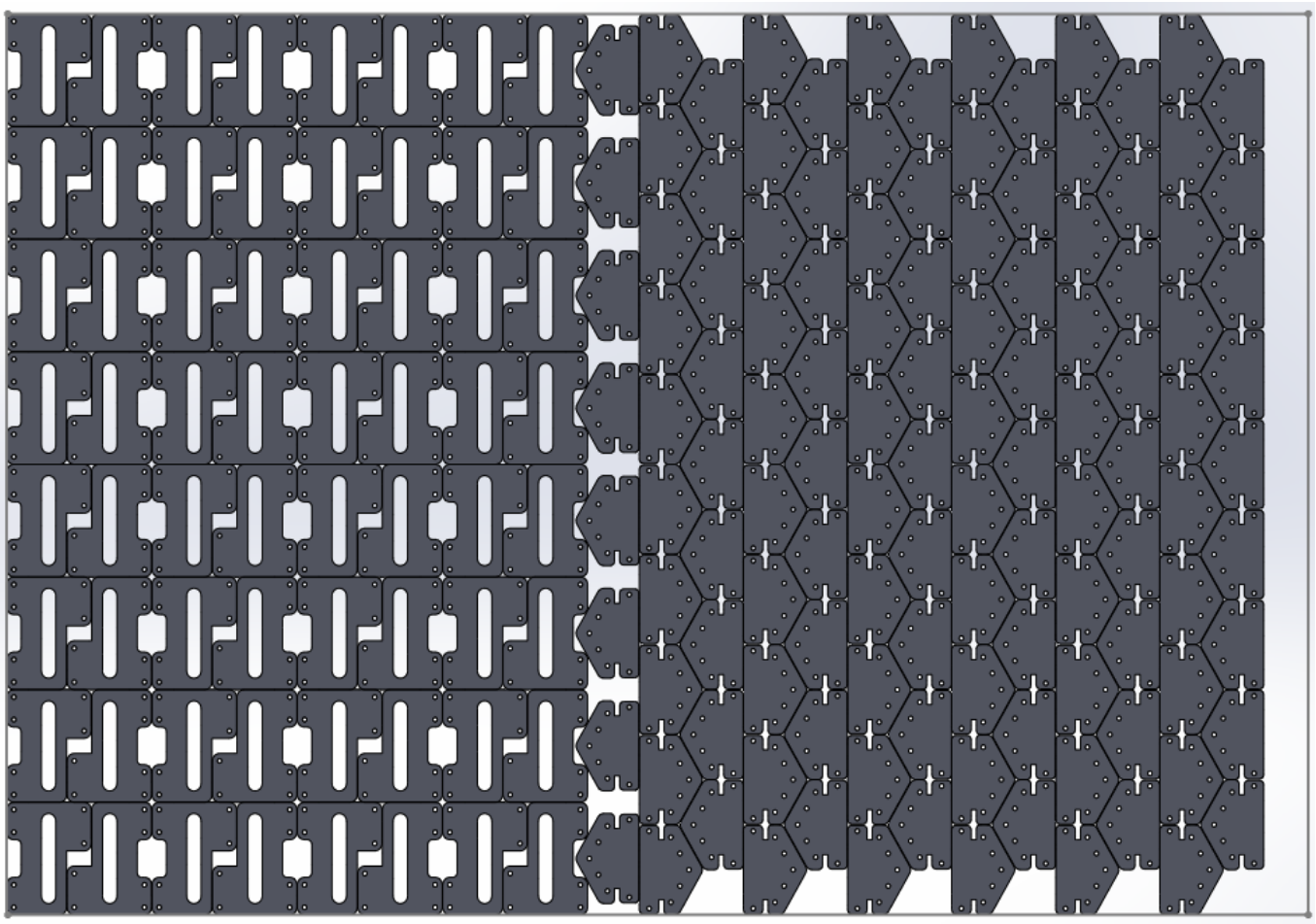
vii. The standard sheet is added to the sheet list. Click on *Select to add sheet* in the sheet list.

viii. Repeat step 2.

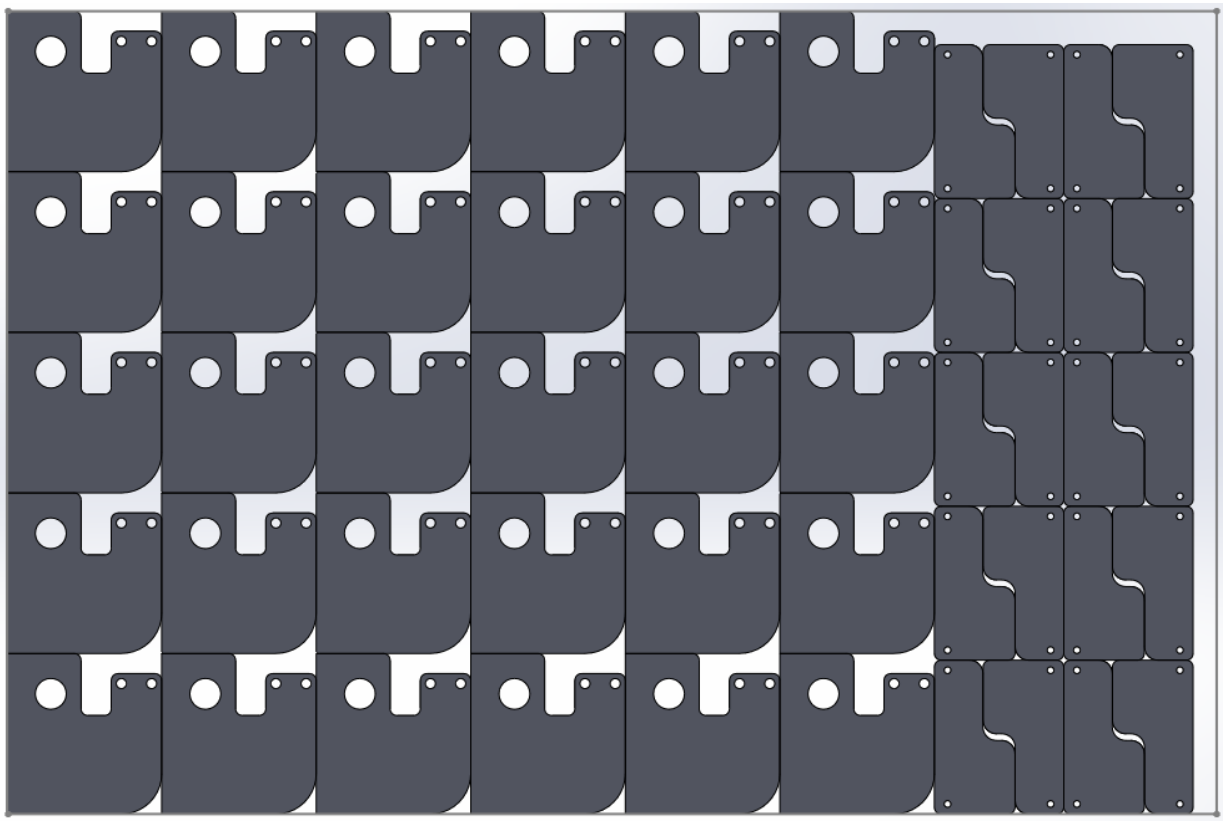
ix. This time, as per the principle explained in step iii, the thickness and Material field will display values of the next part in the part list which has either its thickness or material or both different from the previous part. Thus, thickness field will display a value of 20mm and material field will display 'Copper'.

x. Repeat step iv, v and vi to add the sheet.

6. In the Nesting data group box, leave the Part to part distance and Part to sheet distance set to '0mm'. Specify the location for the output assembly file and Summary text file using the ... button next to the Output Assembly File field.
7. Click *OK* to execute the Nesting process.
8. View the Summary text file. All the parts have been nested as per their assigned quantities.



Nested Layout of TutPart4A SM.SLDPRT and TutPart4D SM.SLDPRT



Nested Layout of TutPart4B SM.SLDPRT and TutPart4C SM.SLDPRT

Nesting of multi-body parts

CAMWorks Nesting supports nesting of multi-body parts and assemblies containing multi-body parts. However, additional steps must be executed in order to nest such a part or assembly.
CAMWorks Nesting processes the multi-body part before it can be nested. In order to nest such a part, CAMWorks Nesting creates and saves each body contained in the multi-body part as a new part. It then proceeds to create an assembly comprising these newly created parts. This newly created assembly becomes the active document considered for the nesting process.

Steps to nest a Multi-body Part

i. Model or open a sheet metal part/ solid part model in SOLIDWORKS/ CAMWorks Solids.
ii. Select *Create Nesting Job* from the CAMWorksNesting menu bar.
iii. CAMWorks Nesting will check the part for multiple bodies.
iv. If the part has multiple bodies, you will be prompted with a message box stating that each body of the part will be saved as a new part and that a new assembly will be created for this multi body part with each body as a separate component. Click *OK* to continue.

v. If you agree to proceed, a new part will be created for each body and will be stored in a new folder located inside the folder containing the parent part (original part with multiple bodies).

vi. Suppose the name of the parent part is *PartName*. Then the new folder will be named as *PartName_WithoutMultiBodyParts*. If a folder with such a name already exists, then the newly created folder will be named 'PartName_WithoutMultipleBodyParts1' and so forth. The new part made out of the first body of the parent part will be named as *PartName_1*; the second body will be named *PartName_2* and so forth. A new assembly named *Assembly.SLDASM* comprising these new parts will be created and saved in the newly created folder.

vii. If the folder which contains the parent part does not have write permissions, you will be prompted to choose a folder location to save the newly created parts and to input the name of the new assembly to be created. The parts created out of the parent part with multiple bodies will be saved inside the folder specified by you.

viii. The new assembly comprising these parts will be saved inside the same folder with the name input by you.

ix. This new assembly comprising parts created out of the parent part will now become the active document considered for nesting process. The single body parts are listed under the Part Data tab of the Create Nesting Job dialog box.

x. Complete the nesting process for this assembly by following the general steps explained in Tutorial 4.

Nesting of assemblies containing multi-body parts

CAMWorks Nesting supports nesting of multi-body parts and assemblies containing multi-body parts. However, additional steps must be executed in order to nest such a part or assembly.
CAMWorks Nesting processes the assembly containing multi-body part(s) before it can be nested. Before nesting an assembly, CAMWorks Nesting checks the assembly for parts containing multiple bodies. If multi-body parts are found, CAMWorks Nesting will create a new part out of each body of the multi-body part(s). After this action, either a new assembly containing parts with single bodies will be created or the existing assembly will be modified to with the multiple body part(s) being replaced with the newly created parts. The action executed is based on the choice input by you. The newly created assembly or modified existing assembly becomes the active document considered for the nesting process.

Steps to nest an Assembly containing Multi-body Parts

i. Model or open the Assembly to be nested in SOLIDWORKS/CAMWorks Solids.

ii. Select 'Create Nesting Job' from the CAMWorksNesting menu bar.

iii. CAMWorks Nesting will check the Assembly for parts with multiple bodies.

iv. On detecting part(s) with multiple bodies in the assembly, you will be prompted with a message box stating that each body of the part will be saved as a new part and that either a new assembly will be created or the existing assembly will be modified. Click *Yes* to create a new assembly else click *No* to modify the existing assembly.

v. If you click *Yes*, a new assembly containing all parts with single bodies will be created. If you click *No*, the existing assembly will be modified with the multi-body part being replaced with single body parts. (In either assembly, the multi-body part will be removed). Note that in case of modifying the existing assembly, the sub-assemblies (if there are any) will be removed and all parts will have the existing assembly as their immediate parent.

vi. Suppose the name of the existing assembly to be nested is *XYZ.sldasm* and it contains two multi-body parts, say 'X' and 'Y' and a single body part named 'Z'. Then CAMWorks Nesting creates new parts out the multi-body parts and either generates the new assembly or modifies the existing assembly in the following manner:

- A new folder named *XYZ_WithoutMultiBodyParts* is created within the folder where the existing assembly is located.
- The new parts created out of the multiple bodies of part 'X' will be named *X_1*, *X_2* and so on and these parts will be saved in this *XYZ_WithoutMultiBodyParts* folder.
- Similarly, the new parts created out of the multiple bodies of part 'Y' will be named *Y_1*, *Y_2* and so on and these parts will also be saved in the same folder.
- The single body part named 'Z' too will be copied into this newly created folder.
- If you selected *Yes* (i.e. you chose to create a new assembly with single body parts), then this newly created assembly will be named *Assembly.sldasm* and this file too will be saved in the *XYZ_WithoutMultiBodyParts* folder. This new assembly file will comprise of all new parts (X_1, X_2 etc.; Y_1,Y_2, etc.) created out the original multi-body parts as well as the single-body parts (Z).
- If you selected *No* (i.e. you chose to modify the existing assembly [XYZ.sldasm]), then the existing assembly will be modified to now contain parts saved within the *XYZ_WithoutMultiBodyParts* folder. Effectively, the original multi-body parts will be replaced with their corresponding parts created out of the multiple bodies.

vii. Thus the newly created assembly or modified existing assembly containing single body parts will become the active document considered for nesting process. The single body parts are listed under the Part Data tab of the Create Nesting Job dialog box.

viii. Complete the nesting process for this assembly by following the general steps explained in Tutorial 4.

TUTORIAL 5 – NEST BY MATERIAL, NEST BY THICKNESS

Introduction

This tutorial explains how to nest multiple solid parts of varying thickness and materials. You will observe how CAMWorks Nesting generates a multiple layout based on the part material and thickness and performs *Preferential hole filling*.

Topic covered in this Tutorial:

- Nesting multiple parts of varying thickness and material
- Preferential hole filling
- Viewing the nested layouts on multiple sheets

Preferential Hole Filling

In this tutorial, we will explore preferential hole filling. In one of the sheet layouts, you will observe how a smaller part can be nested in the holes of larger parts resulting in higher sheet utilization and minimal scrap.

STEP 1: Enabling 'Preferential Hole Filling' functionality

Since the feature of Preferential Hole Filling will be used in this tutorial, it is imperative that this feature be enabled. The default settings configured in the *DefaultValues.ini* file are configured to keep this feature enabled for all the nesting jobs. If you are unsure about the settings, open the *DefaultValues.ini* file and set the *PreferHoleFilling* flag to '1' in order to enable the *'Preferential Hole Filling'* feature.

STEP 2: Using 'Nest by Folder' to open the Assembly

1. Select ***Nest by Folder*** option in the CAMWorks Nesting menu bar.

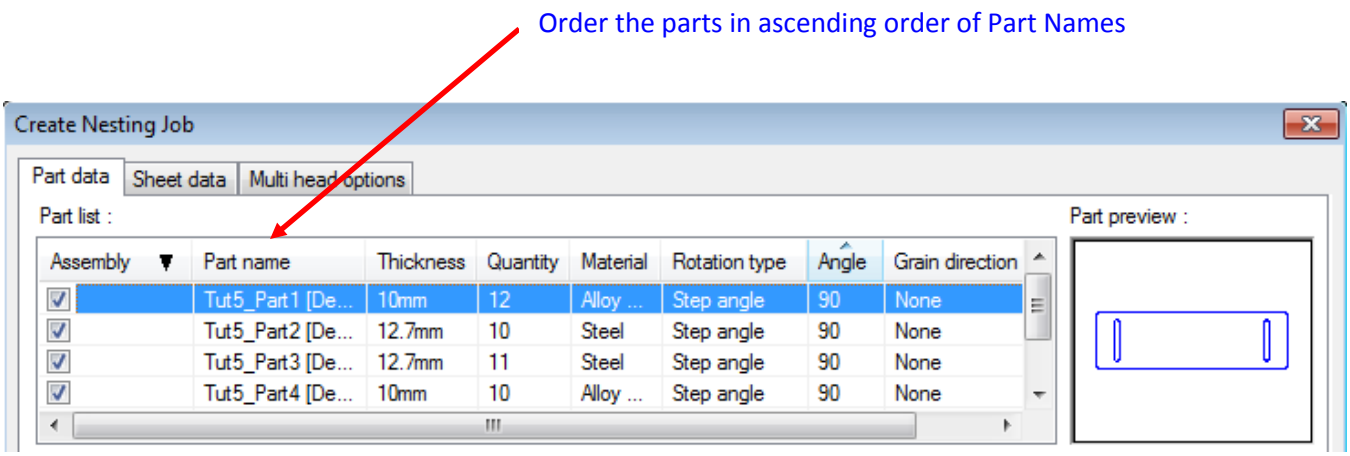
   OR

   Click on the *Nest by Folder* button on the CAMWorks Nesting Ribbon Bar.
2. The *Browse for folder* dialog box opens. Browse to the folder named *Tutorial5* in the following folder location.

   Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies

   CAMWorks Nesting opens all the parts contained in the folder as an assembly in the SOLIDWORKS Graphics area.
3. The *Create Nesting Job* dialog box is displayed. All the parts present in the assembly are listed in the *Part List* of the *Part data* tab.

   In the *Part list*, click on the column heading *Part Name* to sort the data in ascending order from A to Z.



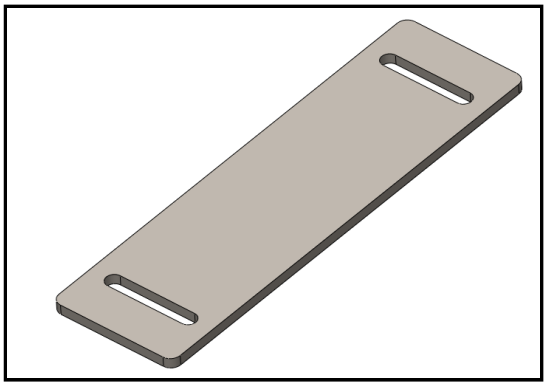
'Nest by Folder' parts listed in the Part List

STEP 3: Define the Part Parameters

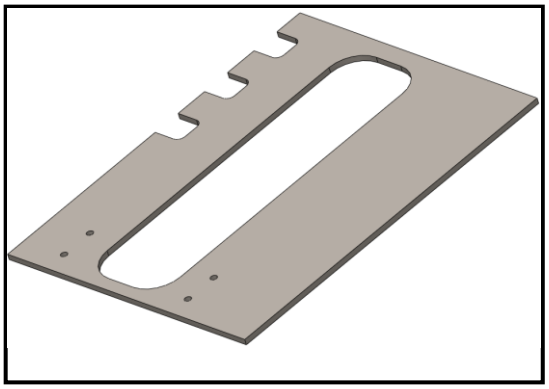
Thickness & Material of the parts

The thickness and material of the solid parts extracted from the solid models is displayed in the Part List.

- The part named 'Tut5_Part1' and 'Tut5_Part4' have the same material 'Alloy Steel (SS)' and thickness (10 mm).

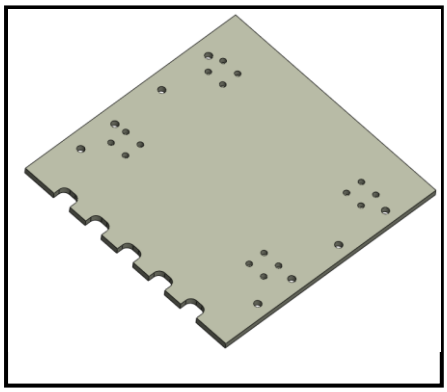


Tut5_Part1.sldprt

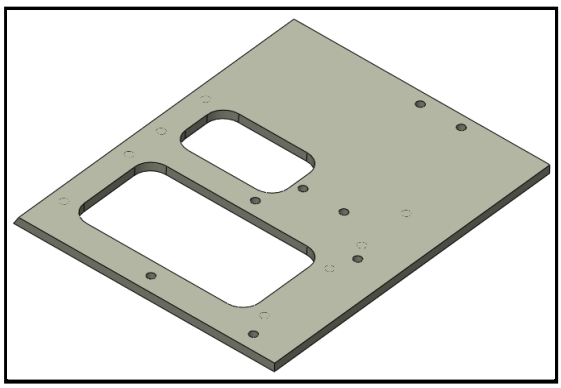


Tut5_Part4.sldprt

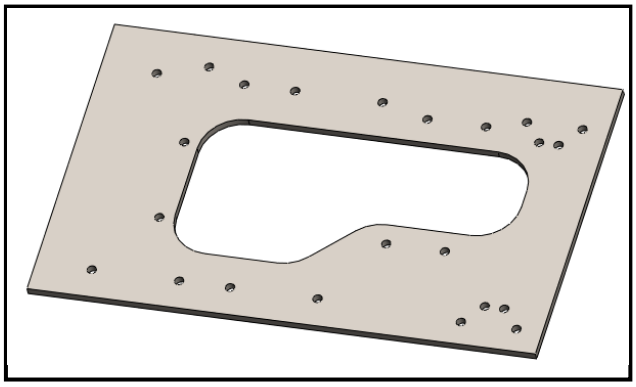
- The parts 'Tut5_Part2', 'Tut5_Part3', 'Tut5_Part5' and 'Tut5_Part6' have identical material 'Steel' and thickness (12.7 mm).



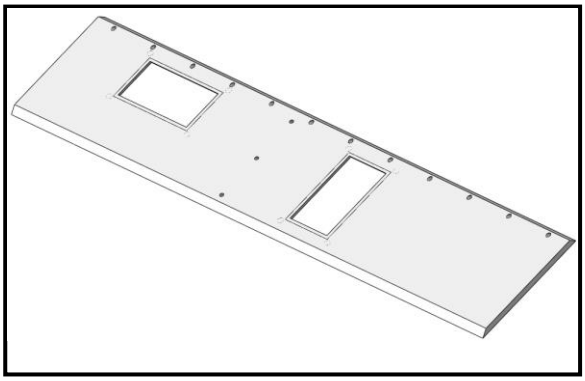
Tut5_Part2.sldprt



Tut5_Part3.sldprt



Tut5_Part5.sldprt



Tut5_Part6.sldprt

Only parts with identical material and thickness can be nested within the same sheet. Based on the above observation, it is clear that 2 different sheets need to be defined to generate nested layouts. Each such sheet nests parts having the same material and thickness.

Normal Face

No changes are made to the default normal face selection for any of the parts.

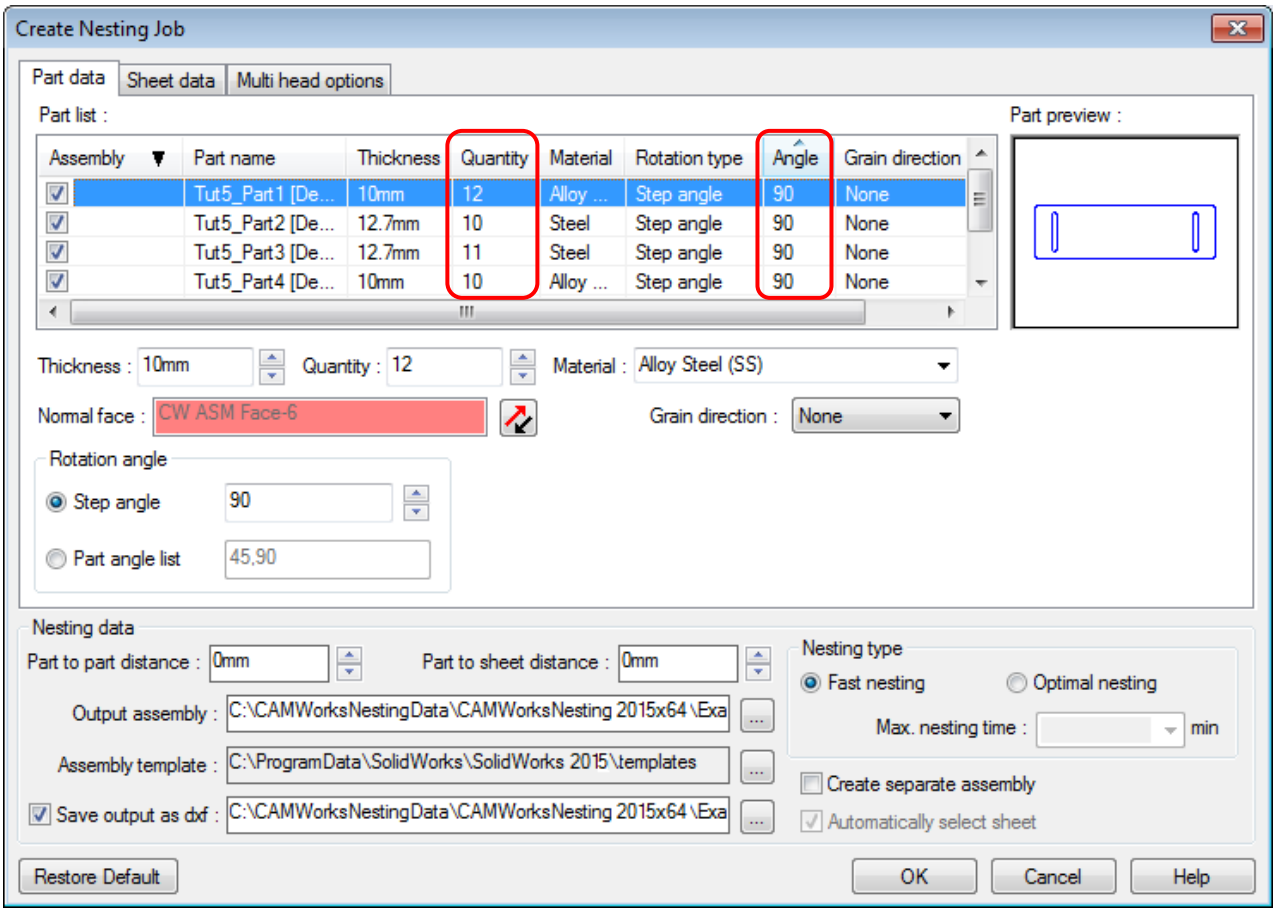
Grain Direction

Leave the Grain direction set to *None* for all the parts.

Step Angle & Quantity

Set the following quantities for the parts:

Part Name	Step Angle to be assigned	Quantity to be assigned
Tut5_Part1	90^0	12
Tut5_Part2	90^0	10
Tut5_Part3	90^0	11
Tut5_Part4	90^0	10
Tut5_Part5	90^0	10
Tut5_Part6	90^0	9



Setting appropriate Part angle and Quantity for the parts

STEP 4: Defining sheet parameters

To nest all the six parts in the part list, three different sheets of varying thickness and material need to be added to the sheet list.

Adding Standard Sheet

Since the parts 'Tut5_Part1.sldprt' and 'Tut5_Part4.sldprt' have identical material [Alloy Steel (SS)] and thickness (10 mm), they can be nested within the same sheet.

To add a standard sheet to nest these parts, following are the steps:

1. Click on the *Sheet Data* tab. In the Sheet list, click on *Select to add sheet*.
2. By default, the thickness of the first part listed in the Part list is 10mm. In case this value is not displayed in the thickness field, assign a 10mm value.
3. By default, the material of the first part listed in the Part list is 'Alloy Steel (SS)'. In case this value is not displayed in the material field, type the material name into the field.

4. To add a standard sheet, select the 'S13 (10' X 4') – Len: 3000 mm Width: 1200 mm' sheet from the Standard Size dropdown List.
5. In the Quantity field, assign a quantity of '1'.
6. Click *Add Sheet* to add the sheet to the Sheet list.

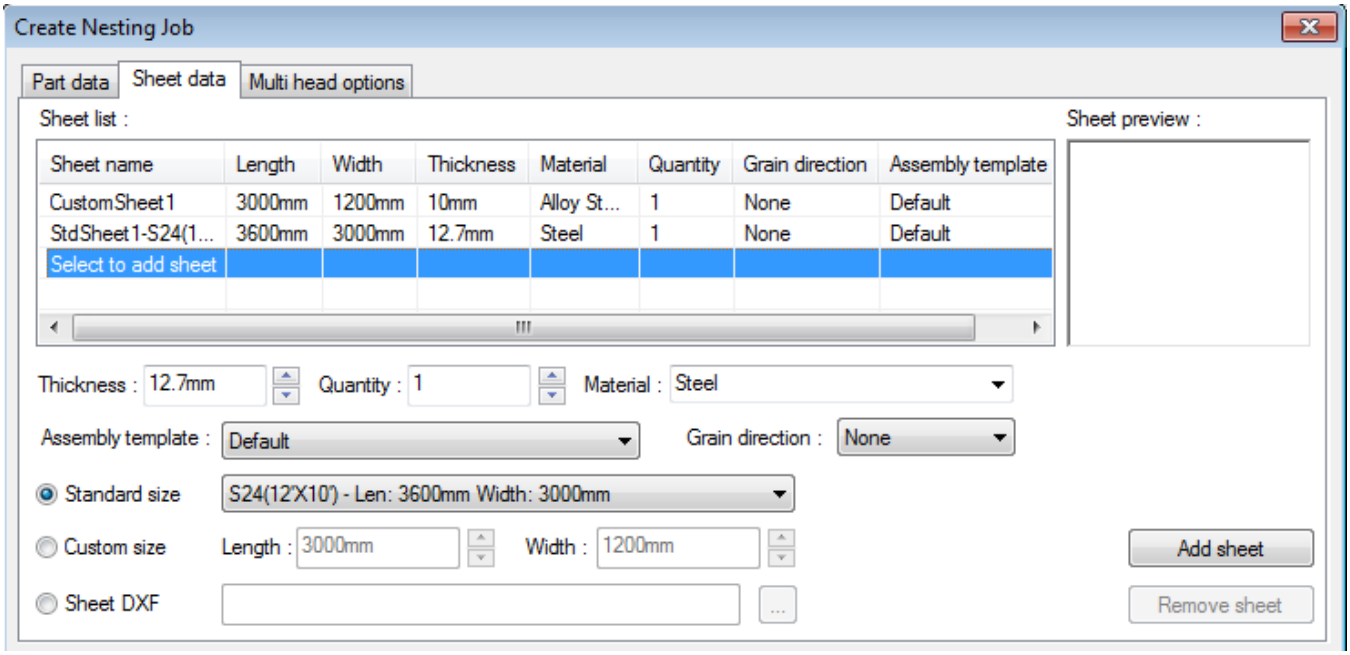
Adding Standard Sheet 2

Next, the parts 'Tut5_Part2', 'Tut5_Part3', 'Tut5_Part5' and 'Tut5_Part6' have identical material [Steel] and thickness [12.7mm]. They can be nested on the same sheet.

Follow the same steps i. to vi. given above to add the standard sheet to nest these parts. However, in step ii, choose the standard sheet of size 'S24 (12' X 10') – Len: 3600 mm Width: 3000 mm'.

In step iii, a thickness of 12.7 mm needs to be assigned to the sheet. Observe that CAMWorks Nesting already displays 12.7 mm as the default thickness.

In step iv, you need to assign the material of the sheet as *Steel*. Observe that CAMWorks Nesting already now displays this material in the material field.



Adding multiple sheets of varying thickness and material to the sheet list

STEP 5: Selecting a machine with Single Tool Head for the Nesting Process

1. Click on the *Multi Head Options* tab.
2. In the *Machine Data* group box, ensure that the Machine selected is *SingleTHMachine*. The *Number of tool heads* for this machine should be '1'.

   Selecting *SingleTHMachine* as the machine ensures the nesting job is executed considering a single tool head and not multiple tool heads.

STEP 6: Define Nesting Parameters

1. In the Nesting Data Group Box, set a *Part to Part distance* of 5 mm and a *Part to Sheet distance* of 5 mm.
2. Select *Fast Nesting* as the Nesting method.
3. Click *OK* to execute the nesting process.
4. Leave the checkbox *Save output as dxf* checked. Use the  button next to the *Output assembly* field to assign the folder location where the nested layouts will be saved in the dxf file format.

Step 7: Generating the Nested Layout

The nested layouts are generated in two file formats:

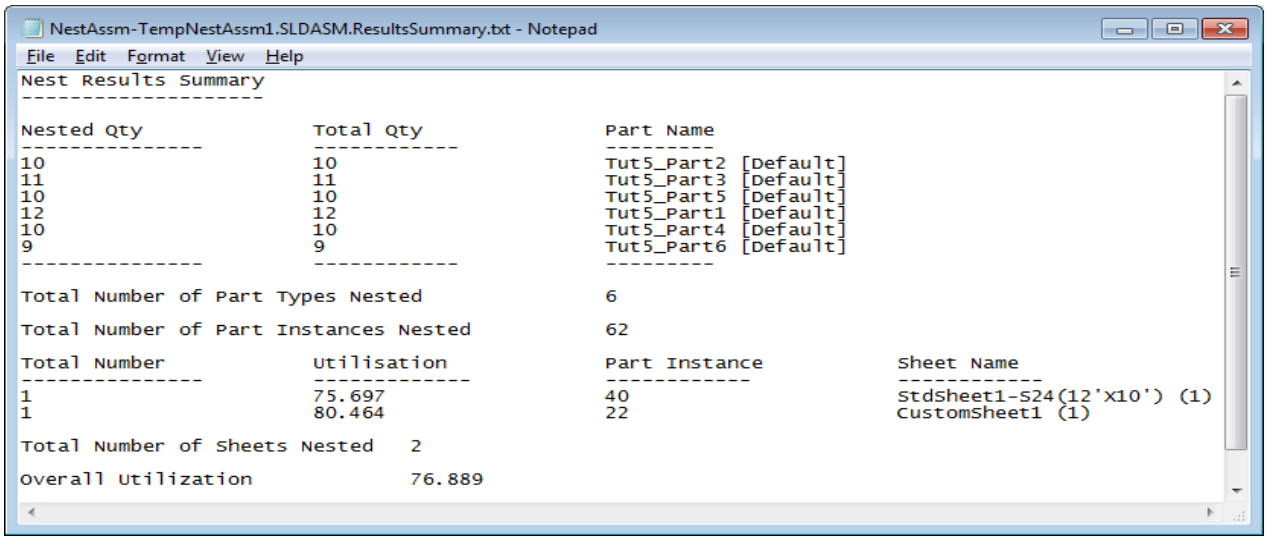
- ➔ Assembly file format (.sldasm)
- ➔ Drawing Exchange Format (.dxf)

Saving Files in the .dxf format

Browse to the folder location assigned for saving the nested layouts in .dxf format. Since two nested layouts were generated, observe that two separate files have been saved in the .dxf format in this folder.

Summary File

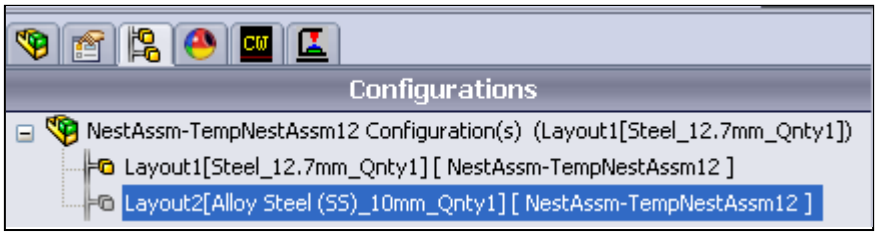
The Summary text file indicates that all the parts have been nested. Observe that the smaller parts have been nested in the holes of the larger parts.



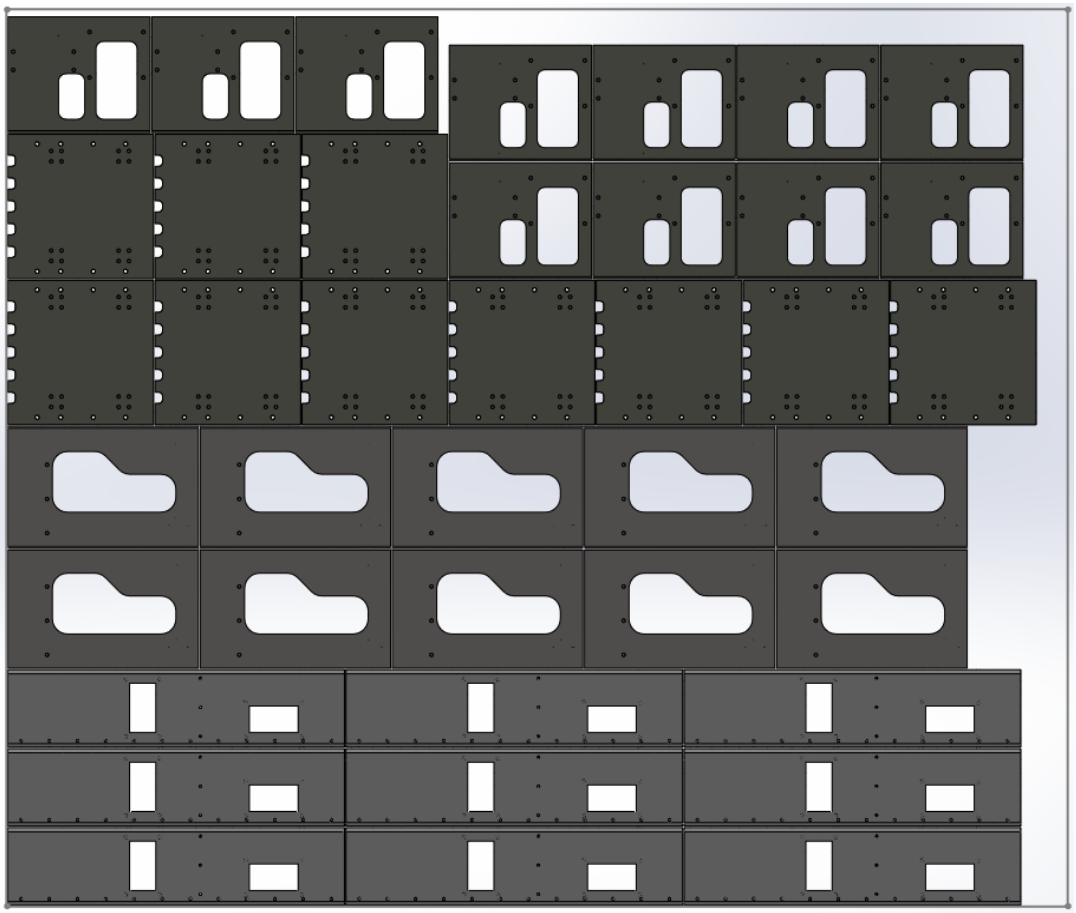
Summary Text File

Viewing the Nested Layouts

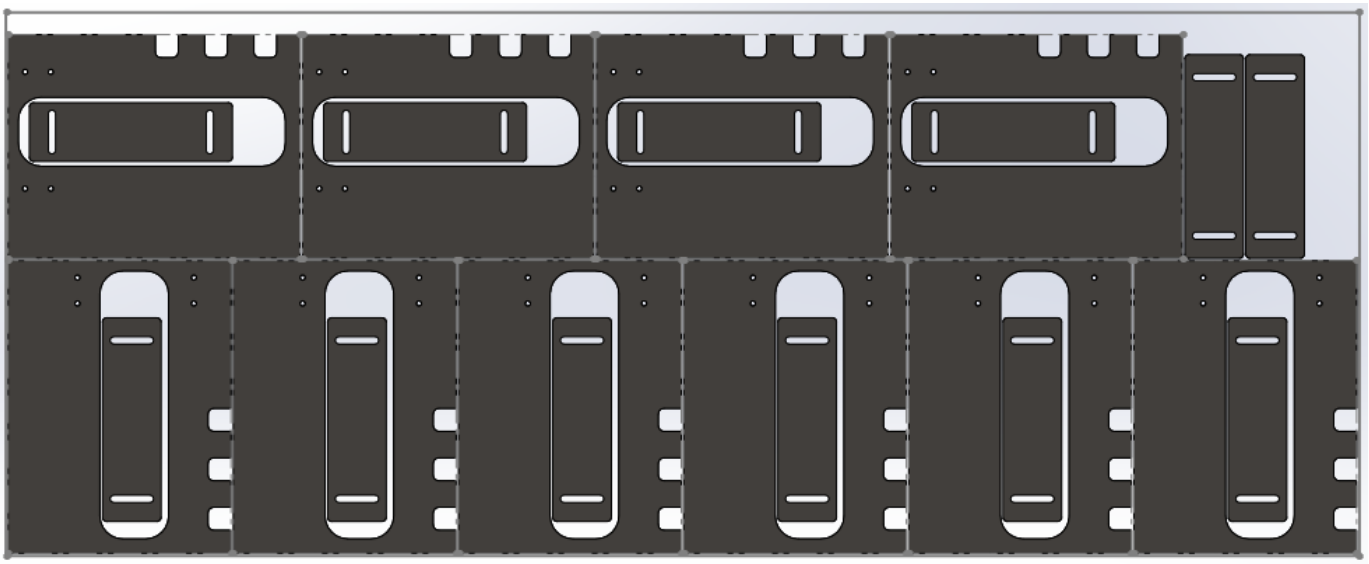
Use the SOLIDWORKS/CAMWorks Solids Configurations tree to view the Nested layouts generated.



SOLIDWORKS Configuration Tree



Nesting layout (Tut5_Part2.sldprt, Tut5_Part3.sldprt, Tut5_Part5.sldprt and Tut5_Part6.sldprt)



Nesting layout with preferential hole filling (Tut5_Part1.sldprt & Tut5_Part4.sldprt)

TUTORIAL 6 – NESTING WITH MULTIPLE TOOL HEADS

Introduction

This tutorial explains how to nest multiple solid parts of the same thickness and material in two or more identical layouts on a sheet simultaneously by using multiple tool heads.

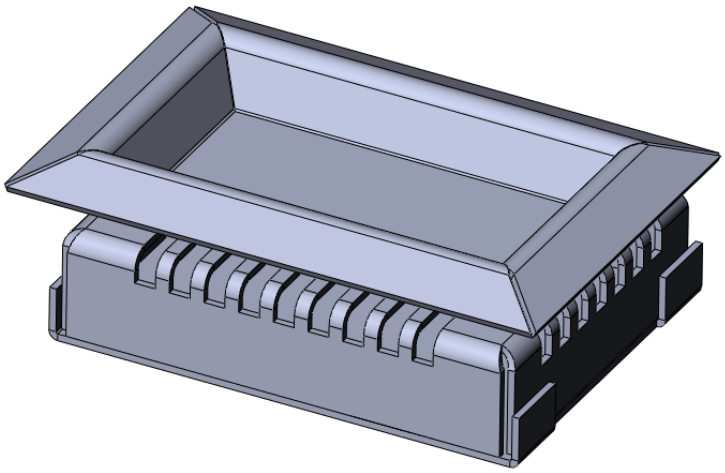
Topic covered in this Tutorial:

- Activating the functionality of nesting with multiple tool heads.
- Nesting parts within a sheet using multiple tool heads to create identical nested regions.

STEP 1: Open the Assembly

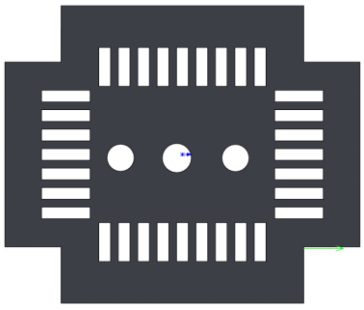
Open the assembly file **Tutorial_6_Multi_Tool.sldasm** in the folllwoing folder location.

Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies\Tutorial6



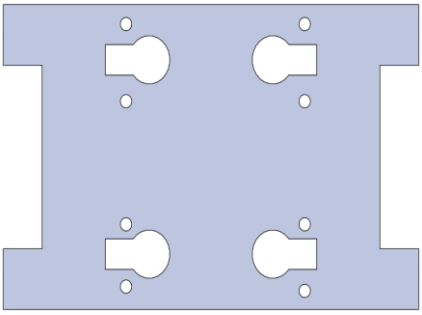
Tutorial_6_Multi_Tool.sldasm

This assembly is made up of three parts.



Tutorial_6_Part1.sldprt

Tutorial_6_Part2.sldprt



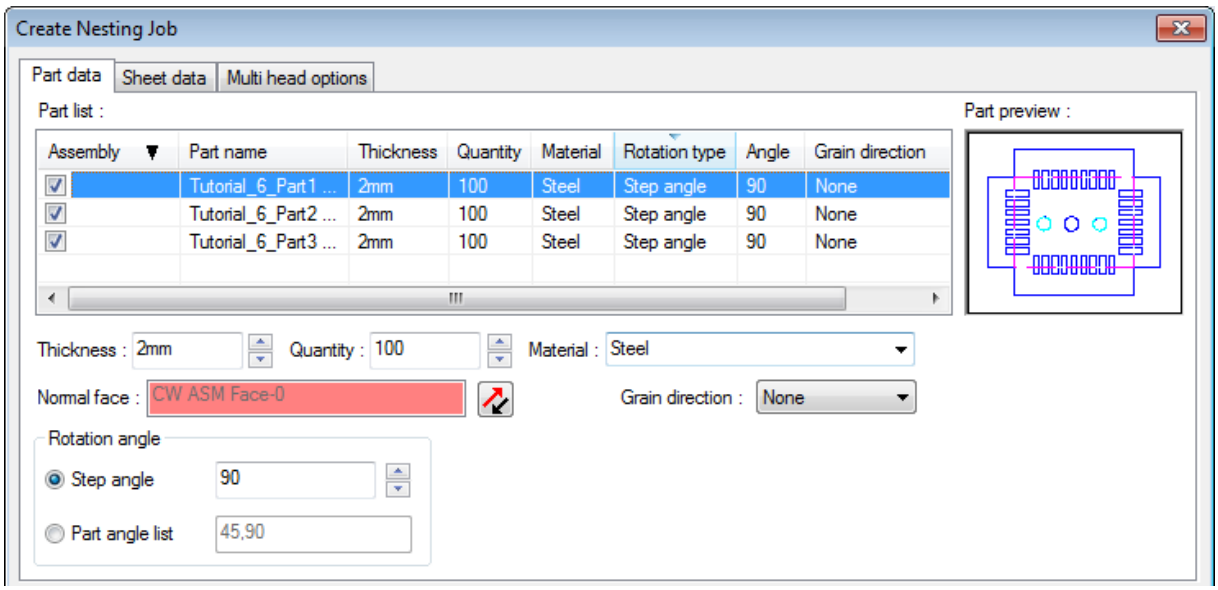
Tutorial_6_Part3.sldprt

STEP 2: Enabling the option of flattening the sheet metal parts

In this tutorial, you will nest the sheet metal part based on its dimensions after flattening. The default settings configured in the *DefaultValues.ini* file ensure that sheet metal parts are flattened before the nesting job is executed. If you are unsure about the settings, open the *DefaultValues.ini* file and set the *FlattenSheetMetalPart* flag to '1' in order to activate the option of flattening.

STEP 3: Define the Part Parameters

1. Select *Create Nesting Job* from the *CAMWorks Nesting* menu. All the parts which constitute the assembly are listed in the *Part List* of the *Part data* tab of this dialog box.



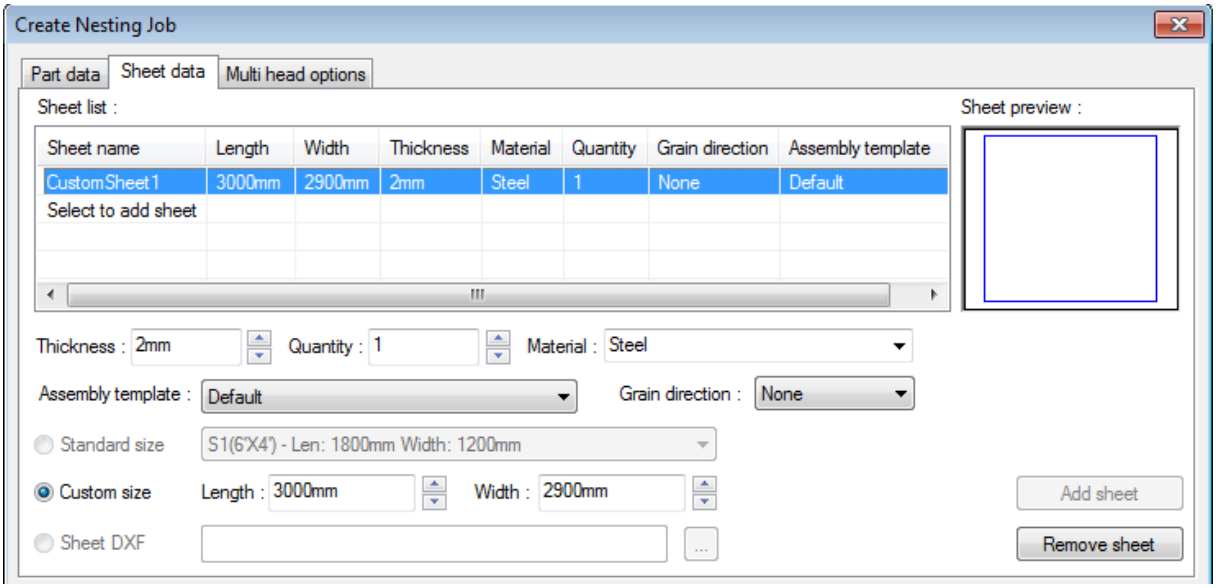
Defining the Part Parameters

2. Observe the *Thickness* and the *Material* of all the three parts are identical. These default values will remain unchanged. Identical thickness and material will enable nesting of these parts in the same sheet.
3. Assign the *Quantity* '100' to all the three parts.
4. Assign a *Step Angle* of 90 degrees to all the three parts.
5. Assign the material as 'Steel' for all the parts.
6. Leave the *Grain Direction* set to *None* for all the three parts.

STEP 4: Define the Sheet Parameters

In this exercise, you will use a custom sheet with a length of 3000mm and width of 2900mm to nest the parts.

1. Click on the *Sheet Data* tab of the *Create Nesting Job* dialog box.



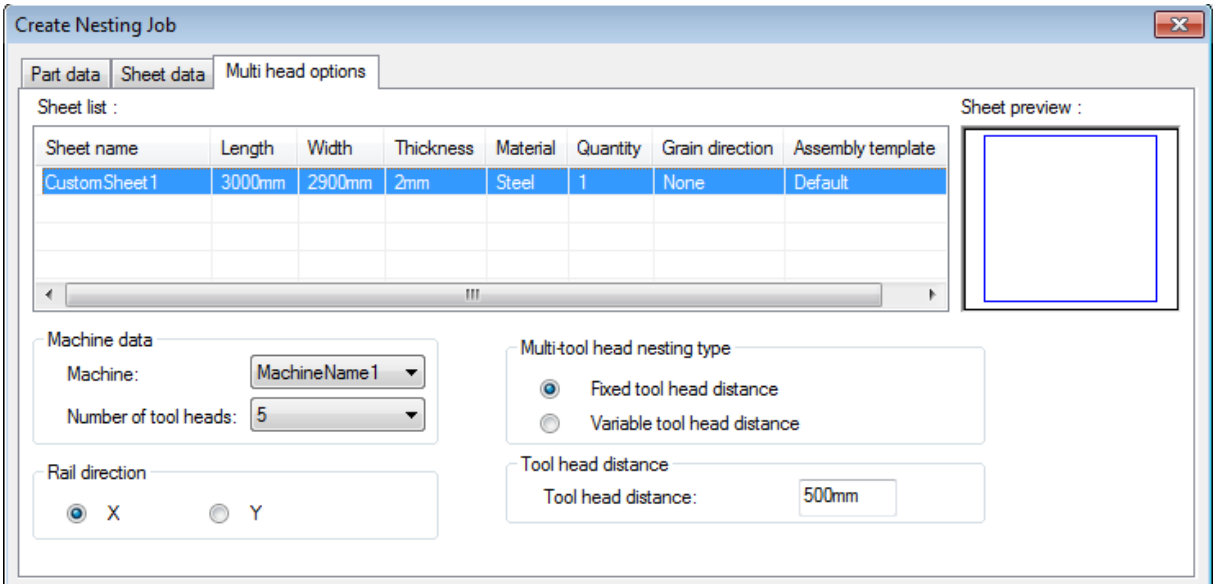
Defining the Sheet Parameters for Custom size sheet

2. In the Sheet list, click on *Select to add sheet*.
3. By default, the thickness of the first part given in the *Part list* is given as the Thickness field (2mm). Leave this parameter value as it is.
4. By default, the material of the first part listed in the *Part list* is given in the Material field (Steel). Leave this parameter value as it is.
5. In the Quantity field, assign a quantity of '1'.
6. Leave the *Grain Direction* set to *None* and the *Assembly Template* set to *Default*.
7. To add a custom size sheet, select the *Custom size* option. Assign a *Length* of 3000mm and a *Width* of 2900mm.
8. Click *Add Sheet* to add the sheet to the *Sheet list*.

STEP 5: Define the Multi head options parameters

To nest using multiple tool heads, it is necessary to assign appropriate values to the parameters associated with nesting using multiple tool heads. The *Multi head options* tab of the *Create nesting Job* dialog box allows you assign/edit these parameters.

1. Click on the *Multi head options* tab of the *Create Nesting Job* dialog box.



Defining the Multi head options parameters for nesting with multiple tool heads

2. The *Sheet list* in this dialog box lists the *Custom size* sheet added in the *Sheet Data* tab. The parameters associated with nesting using multiple tool heads have to be defined separately for each sheet listed in the Sheet list.
3. Highlight the lone sheet listed in the *Sheet list*.
4. In the Machine dropdown list, select *MachineName1* for the machine.

   In case your *Machine* list has already been customized to suit your facility's requirements, then *MachineName1* will not be listed. To proceed with the tutorial, you can do one of the following:

   i. Create a dummy machine named *MachineName1* with associated parameters in the Machine.ini file so that the machine is listed here in this list. This is explained in the section *Adding a new machine to the Machine.ini file*.

   ii. Select another machine from the *Machine* list which has at least 5 tool heads. All the other parameters can be edited to suit the requirements of this tutorial before the nesting job is executed.
5. The default values for the parameters associated with *MachineName1* will be displayed. (These default values are defined in the *Machine.ini* file.)

   The default values associated with the parameters are:

   a. Number of Tool heads: 5
   b. Rail Direction: X
   c. Multi-tool head nesting type: Fixed tool head distance
   d. Tool head distance: 500mm

   In case you selected a machine other than *MachineName1*, edit the parameters to assign them the values/options given above.

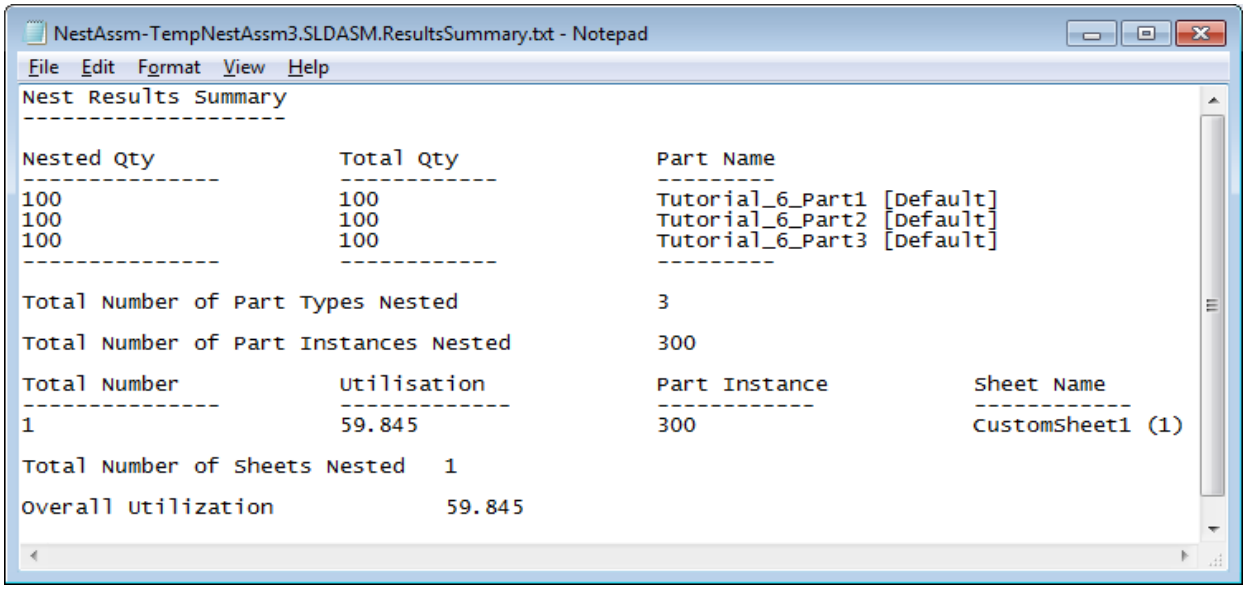
6. In this tutorial, the nesting with multiple tools will be executed using the default parameter values associated with the machine *MachineName1*.

STEP 6: Define Nesting Parameters

1. In the Nesting Data Group Box, set a *Part to Part distance* of 10 mm and a *Part to Sheet distance* of 10 mm.
2. *Fast Nesting* is the default *Nesting type*. Note that *Optimal Nesting* option has been disabled.
3. Use the  button next to the *Output assembly* field to specify where the nested layout assembly file and the Summary text file that are generated will be saved.
4. Leave the *Create separate assembly* checkbox unchecked.
5. Click *OK* to execute the nesting process.

Step 7: Generating the Nested Layout

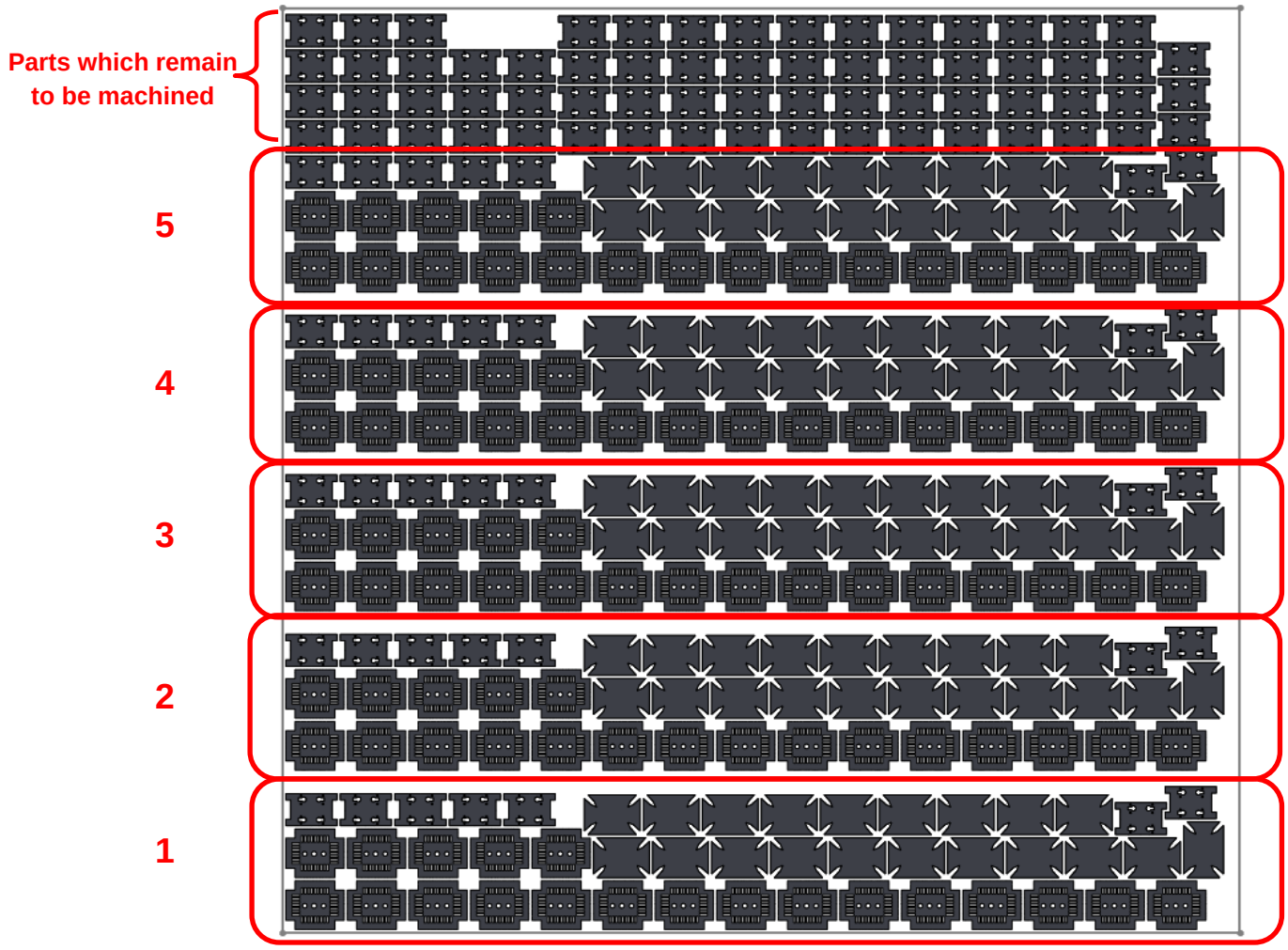
The Summary text file indicates that the prescribed quantities for all the parts have been nested within the sheet.



Nesting Results Summary Text file

The nested layout generated after executing the nesting job is given on the next page.

Observe the nested layout. The five tool heads used create 5 identical nested layouts in the 'X' direction.



Nesting layout with 5 identical regions created using 5 tool heads

20 instances of the first part *(Tutorial_6_Part1.sldprt)*, 20 instances of the second part *(Tutorial_6_Part2.sldprt)* and 5 instances of the third part *(Tutorial_6_Part3.sldprt)* are nested in each identical nesting region.

Thus, 100 instances of the first two parts are nested within the 5 identical regions. Only 25 instances of the third part are nested within the identical regions. The remaining 75 instances can be nested in the remnant sheet left after nesting the 5 regions.

TUTORIAL 7 – NESTING IMPORTED SHEET METAL COMPONENTS WITH BENDS

Introduction

This tutorial explains how to unfold imported 3D sheet metal components with bends using various menu options available within CAMWorks Nesting.

Topic covered in this Tutorial:

- Using the 'Unfold Imported Bodies' dialog box
- Using the 'Enable Auto Unfold' option
- Using the 'Intelligent Unfold' command
- Using the 'Unfold All Parts' command

The functionality of Unfolding Sheet Metal Parts

CAMWorks Nesting supports nesting of imported part models. If the sheet metal parts to be nested contain bends, then these parts should ideally be unfolded before the nesting job is executed.

Most native sheet metal parts (parts created using the solid modeler of SOLIDWORKS or CAMWorks Solids) as well as imported sheet metal parts can be unfolded using the in-built functionality of the Solid Modeler itself. However, sheet metal parts with complex architectures can sometimes be flattened/unfolded incorrectly.

CAMWorks Nesting provides a more robust functionality for unfolding both imported and native sheet metal parts. Sheet metal parts that are unfolded incorrectly by the Solid Modeler can be unfolded correctly using this functionality.

The 'Unfold Imported Bodies' dialog box

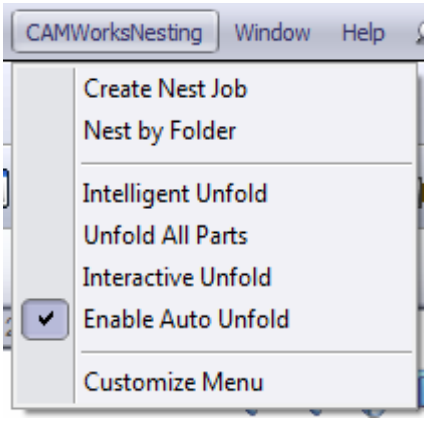
The CAMWorks Nesting functionality for unfolding both native and imported sheet metal parts is provided in the form of *'Unfold Imported Bodies'* dialog box. This dialog box allows you to select sheet metal parts with bends to be unfolded and assign parameters associated with unfolding sheet metal parts.

Note: **The functionality of Unfolding is meant for sheet metal parts only. If the part(s)/ assembly to be nested contain imported solid parts, then these imported solid parts too will be displayed in the 'Unfold Imported Bodies' dialog box. Ensure that you unselect the solid parts from the list of parts to be unfolded.**

Commands to Invoke 'Unfold Imported Bodies' dialog box

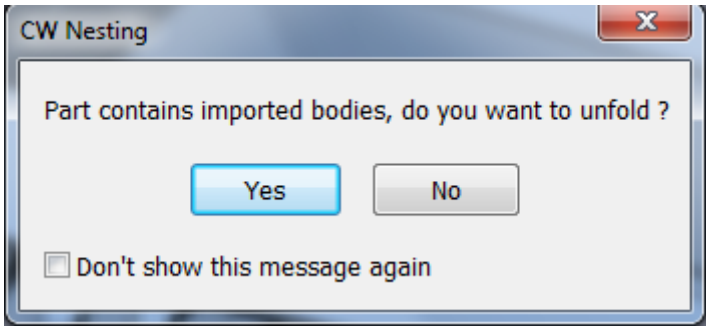
1. The 'Enable Auto Unfold' Option

Click on the CAMWorksNesting menu. Observe that the *Enable Auto Unfold* option is checked by default.



'Enable Auto Unfold' option in CAMWorksNesting menu

- When the *Enable Auto Unfold* option is checked and the *Create Nesting Job* command is executed for part(s)/assembly containing one or more imported bodies, CAMWorks Nesting will automatically display the *Unfold Imported Bodies* dialog box. Use this dialog box to select the bodies to be unfolded and set the parameters associated with unfolding.
- When the *Enable Auto Unfold* option is not checked and the *Create Nesting Job* command is executed for a part(s)/assembly containing imported bodies, CAMWorks Nesting will display a warning message stating that the part contains imported bodies and whether you wish to unfold these imported bodies.



CAMWorks Nesting message displayed when part or assembly contains imported sheet metal bodies with folds

➔ If the part(s)/assembly to be nested comprises of one or more sheet metal bodies with bends, then click ***Yes*** to unfold the

imported sheet metal parts. The *Unfold Imported bodies* dialog box will be displayed. All the imported parts will be listed in the dialog box. Note that native sheet metal parts with bends, if present in the part(s)/assembly o be nested, will not be listed in this dialog box. These native parts will either be unfolded/remain folded based on settings for Flattening sheet metal parts in the DefaultValues.ini.

➔ If the part(s)/ assembly to be nested comprises of only solid parts [i.e. it does not have a single sheet metal part with bend(s)], then click ***No***. (CAMWorks Nesting does not support unfolding of solid parts.) The 'Create Nesting Job' dialog box will be displayed.

➔ If you select the ***Don't show this message again*** option, CAMWorks Nesting will remember your preference.

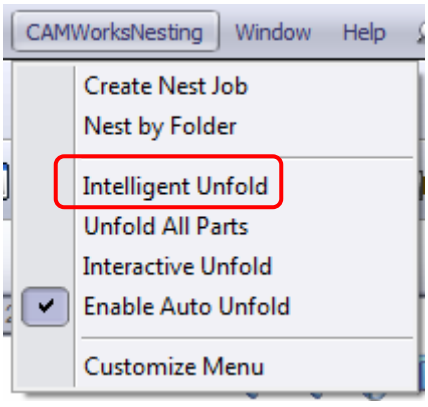
2. The 'Intelligent Unfold' Command

Function

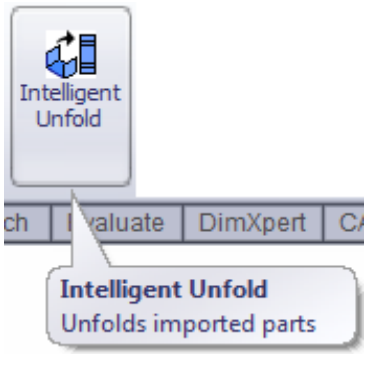
The Solid Modeler (SOLIDWORKS/CAMWorks Solids) can only unfold native sheet metal parts with bends. Imported sheet metal parts with bends cannot be unfolded using Solid Modeler functionality. The '*Intelligent Unfold*' command is ideal for nesting parts/assemblies comprising imported sheet metal part(s) with bends.

Command Execution

1. Clicking on the 'Intelligent unfold' command in the CAMWorksNesting menu opens the *Unfold Imported Bodies* dialog box.
2. Alternatively, clicking on the *Intelligent Unfold* button on the CAMWorks Nesting Ribbon Bar also opens this dialog box.



'Intelligent Unfold' command in CAMWorksNesting menu



'Intelligent Unfold' button in the CAMWorks Nesting Ribbon Bar

How it works

When the *Unfold Imported Bodies* dialog box is opened using the *Intelligent Unfold* command, only imported parts will be listed. Any native

part present will not be listed. If there are any imported solid bodies listed in the grid, ensure that you unselect them to avoid unfolding. Use this dialog box to set parameters associated with flattening the parts before nesting.

Note: **If you open the 'Unfold Imported Bodies' dialog box using either the 'Intelligent Unfold' or 'Create Nesting Job' command, then only imported sheet metal parts with bends will be listed in the dialog box. Native sheet metal parts with bends, if present, will not be listed.**

When you click 'OK' button of this dialog box:

1. The imported sheet metal parts selected for unfolding will be unfolded based on user-defined parameters input in this dialog box.
2. Native sheet metal bodies with bends, if present in the parts/assembly, will either be unfolded/remain folded based on settings for Flattening sheet metal parts in the DefaultValues.ini. The Solid Modeler's functionality for unfolding will be applied for unfolding the native sheet metal parts.

Next Step

After unfolding the imported parts using this dialog box, you can click on the 'Create Nesting Job' menu option to proceed with the nesting process.

3. The 'Unfold All Parts' Command

Function

The Solid Modeler (SOLIDWORKS/CAMWorks Solids) has in-built functionality for unfolding native sheet metal parts with bends. CAMWorks Nesting too contains an in-built functionality to unfold both native and imported sheet metal parts.

However, certain native sheet metal parts can sometimes be incorrectly unfolded by the Solid Modeler. Such parts can then be alternatively unfolded using the CAMWorks Nesting functionality.

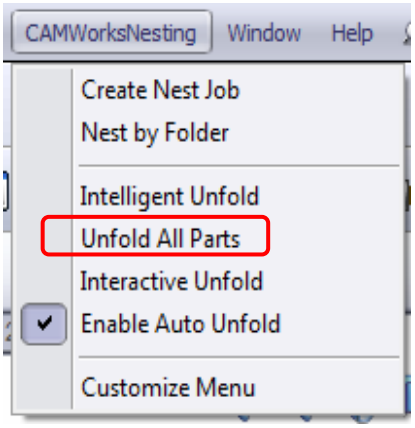
The '*Unfold All Parts*' command is ideal when you wish to unfold all sheet metal parts with bends (both native and imported) using the CAMWorks Nesting functionality of unfolding.

When this command is executed:

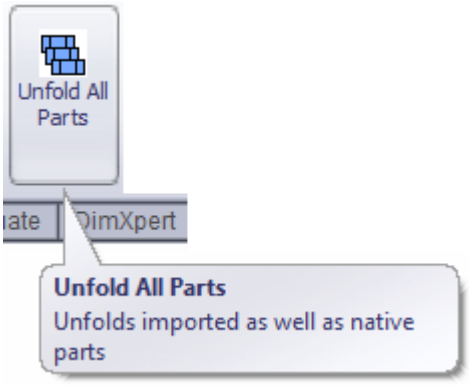
1. The imported sheet metal parts will be unfolded using the CAMWorks Nesting functionality of unfolding.
2. For native sheet metal parts, you can select/unselect the native sheet metal parts to be unfolded using this functionality. Any unselected native part will then be unfolded using the Solid Modeler functionality of unfolding.

Command Execution

1. Clicking on the '*Unfold All Parts*' command in the CAMWorksNesting menu opens the *Unfold Imported Bodies* dialog box.
2. Alternatively, clicking on the *Unfold All Parts* button on the CAMWorks Nesting Ribbon Bar also opens this dialog box.



'Unfold All Parts' command in CAMWorksNesting menu



'Unfold All Parts' button in the CAMWorks Nesting Ribbon Bar

How it works

When the *Unfold Imported Bodies* dialog box is opened using the *Unfold All Parts* command, both imported parts as well as native parts will be listed within the dialog box.

- If there are any imported solid bodies listed in the grid, ensure that you unselect them to avoid unfolding.
- If there are any native sheet metal parts that you do not wish to unfold using the CAMWorks Nesting functionality for unfolding, ensure that you unselect such parts. Use this dialog box to set parameters associated with unfolding the sheet parts before nesting.

Note: **If you open the 'Unfold Imported Bodies' dialog box using the 'Unfold All Parts' command, then both imported sheet metal parts with bends as well as native sheet metal parts with bends will be listed in the dialog box.**

When you click 'OK' button of this dialog box:

1. All the imported sheet metal parts selected for unfolding will be unfolded based on user-defined parameters input in this dialog box.
2. All native sheet metal parts listed in the dialog box which were selected for unfolding will be unfolded using the CAMWorks Nesting functionality for unfolding.
3. If any native sheet metal part listed in the dialog box was deselected from unfolding, then that part will be either be unfolded/remain folded based on settings for Flattening sheet metal parts in the

DefaultValues.ini. If unfolded, then the Solid Modeler's functionality for unfolding will be applied for unfolding the native sheet metal parts.

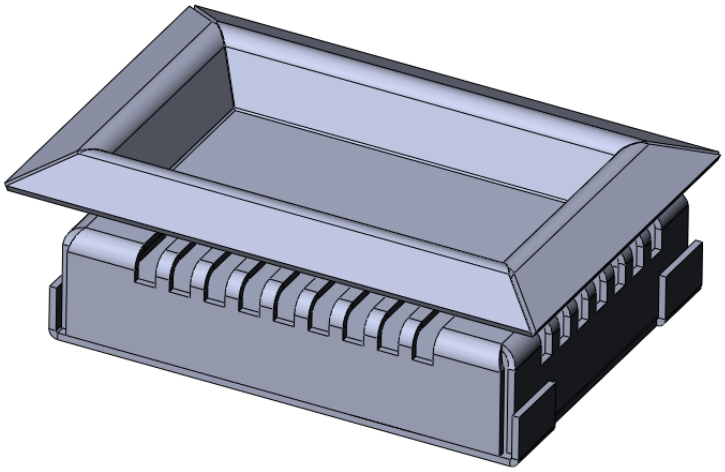
Next Step

After unfolding the sheet metal parts using this dialog box, you can execute the 'Create Nesting Job' command to proceed with the nesting process.

STEP 1: Open the Assembly

Open the assembly file **Tutorial7_Unfold_Assembly.sldasm** in the following folder location.

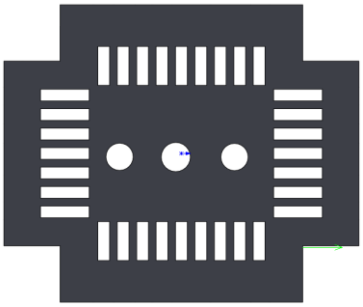
Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies\Tutorial7



Tutorial7_Unfold_Assembly.sldasm

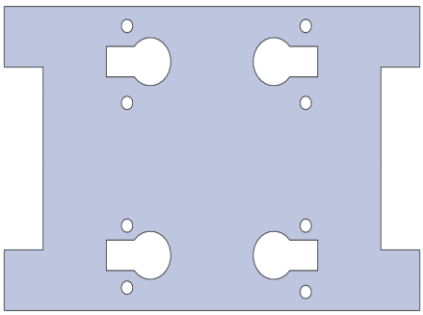
This assembly comprises of three sheet metal parts:

i. Tutorial_7a_Native.sldprt *(native part)*
ii. Tutorial_7b_Imported.sldprt *(imported part)*
iii. Tutorial_7c_Imported.sldprt *(imported part)*



Tutorial_7a_Native.sldprt

Tutorial_7b_Imported.sldprt



Tutorial_7c_Imported.sldprt

STEP 2: Unfolding the Parts with bends

1. In the CAMWorksNesting menu, ensure that there is no check placed against the *Enable Auto Unfold* option.
2. Click on the menu item *Create Nesting Job* command in the CAMWorksNesting menu.

   OR

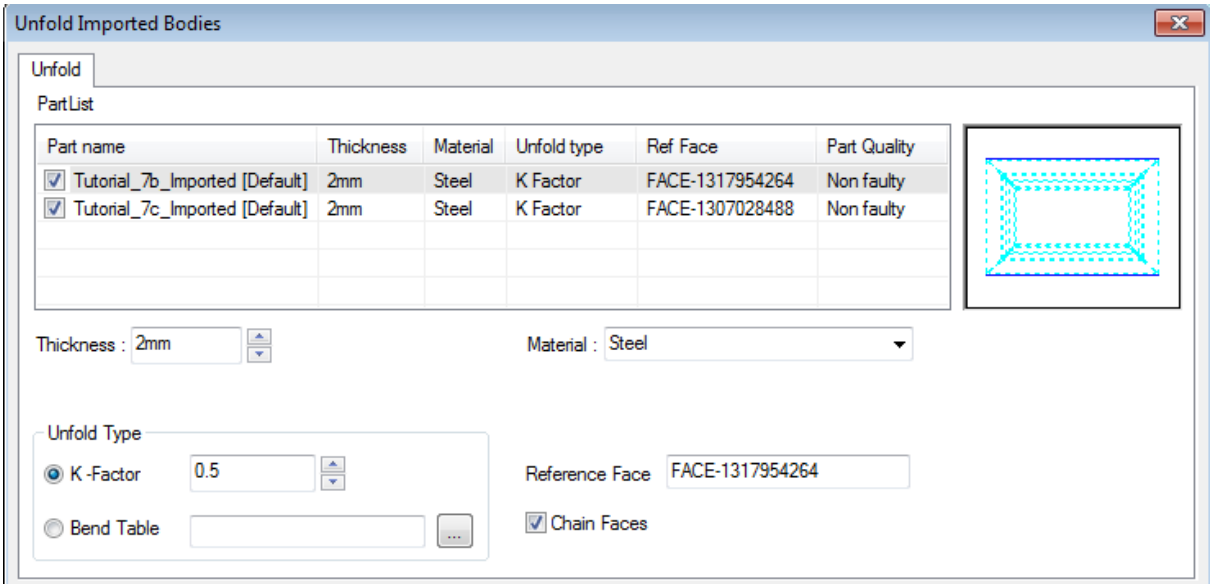
   Click on the *Create Nest Job* button in the CAMWorks Nesting Ribbon Bar.
3. Since the assembly to be nested contains imported sheet metal parts with bends, CAMWorks Nesting will display a message stating that the part contains imported bodies and whether you want to unfold these bodies. Click *Yes*.

Note: **If the parts/assembly to be nested contains only native sheet metal parts with bends, then this message won't be displayed. Instead, CAMWorks Nesting will directly display the 'Create Nesting Job' dialog box. The native parts will be unfolded/remain folded depending upon the settings in the DefaultValues.ini file.**

4. The *Unfold Imported Bodies* dialog box is displayed. This dialog box is used to facilitate the unfolding of sheet metal parts and associated parameters before proceeding with a nesting job. All the imported parts comprising this assembly are listed *Unfold* tab of this dialog box. In this case, the sheet metal parts ***Tutorial_7b_Imported.sldprt*** and ***Tutorial_7c_Imported.sldprt*** will be listed.
5. Observe that the sheet metal part named ***Tutorial_7a_Native.sldprt*** is not listed in this grid. This is because *Unfold Imported Bodies* dialog box is displayed using the 'Intelligent Unfold' command lists only imported parts. This native part will either be automatically unfolded or remain unfolded depending upon the option settings for Flattening sheet metal parts in the DefaultValues.ini. If unfolded, then the Solid Modeler functionality for unfolding will be used.

Note: **Native sheet metal parts will be listed in the *Unfold Imported Bodies* dialog box only if the 'Unfold all Parts' command is used to invoke this dialog box. Native parts selected for unfolding in the *Unfold Imported Bodies* dialog box will be unfolded using the CAMWorks Nesting functionality for unfolding instead of the default Solid Modeler functionality.**

6. In the Part List grid, remove the check ☑ from the checkboxes of those imported parts that you do not wish to unfold. In this tutorial, you will unfold both the imported parts. Hence, do not remove the check from the check boxes.



The 'Unfold Imported Bodies' dialog box

4. Assign the following values to the Unfold Parameters:
   i. **Thickness & Material:** CAMWorks Nesting extracts the part parameter of Thickness and Material from the Solid Part and displays it in the Thickness and Material fields respectively as default thickness and material for the part. The thickness of the part, as extracted from the solid part, is displayed as **2mm** and the material is 'steel'. Leave these values as they are.
   ii. Unfold Type:
      a) **K-factor:** CAMWorks Nesting displays K-factor value **0.5** as default for unfold of the part, you can change the K-factor.
      b) **Bend Table:** Optionally you can also select the bend table.

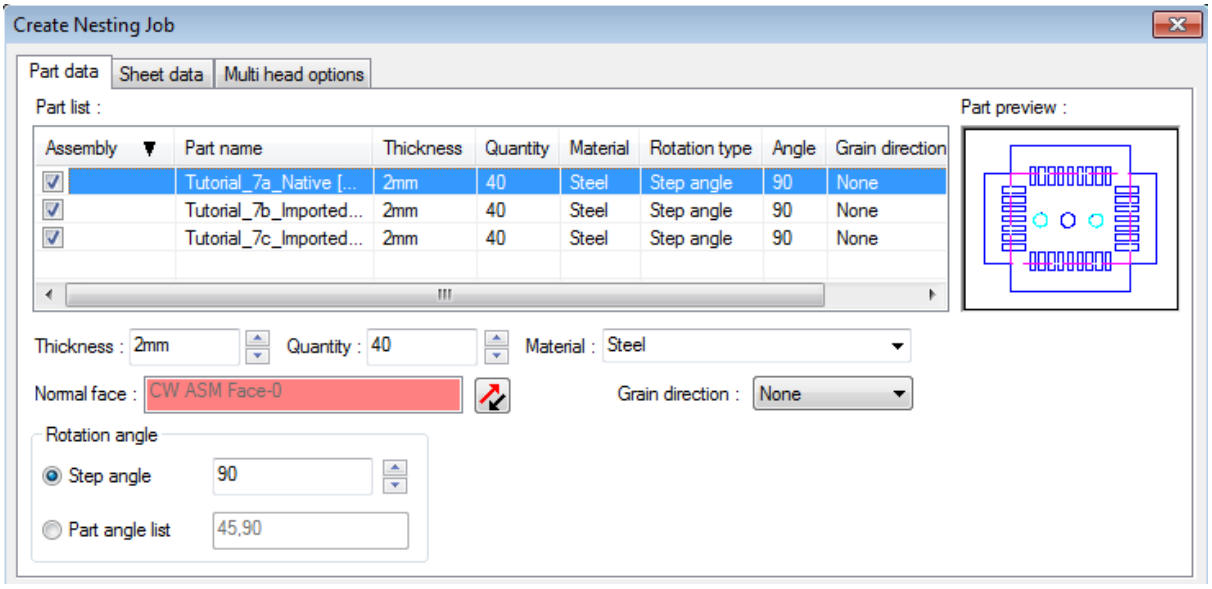
      In this tutorial, we will use the default value of the K-factor for both the sheet metal parts.
   iii. **Reference face:** By default, CAMWorks Nesting chooses the face with the largest area as the *Reference Face*. In this tutorial, the bottom face of the solid part is chosen by default. The normal direction is indicated by an arrow in the graphics area.
5. Click the *OK* button of this dialog box to unfold the imported parts. The unfolded parts are displayed in the graphics area of SOLIDWORKS/CAMWorks Solids.
6. The *Create Nesting Job* dialog box is now displayed.

STEP 3: Defining the Part, Sheet & Nesting Parameters

Part Data Tab

In the *Part data* tab of the *Create Nesting Job* dialog box, assign the following values to the parameters:

1. **Thickness & Material:** Observe that the Thickness and the Material of all the three parts are identical. These default values will remain unchanged. Identical thickness and material will enable nesting of these parts in the same sheet.
2. **Quantity:** Assign the Quantity of **40** to all the three parts.
3. **Step Angle:** Assign a Step Angle of **90 degrees** to all the three parts.
4. **Grain Direction:** Leave the Grain direction set to *None* for all the three parts.



Defining the Individual Part Parameters in the 'Create Nesting Job' dialog box

Sheet Data Tab

In this tutorial, we will use a standard sheet of size S3 [Length = 1800mm; Width = 1800mm] to nest all the three parts.

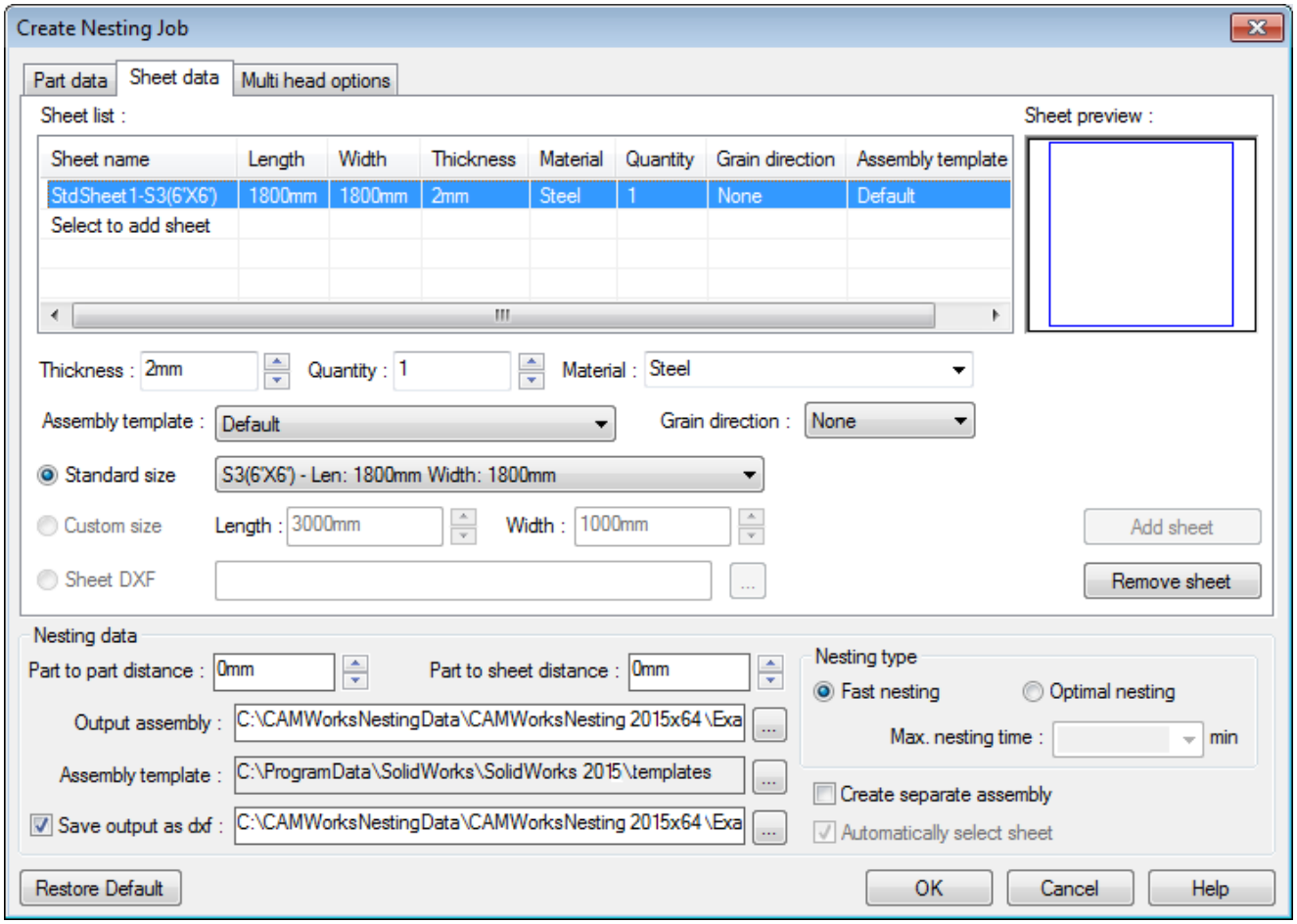
Click on the *Sheet data* tab of the *Create Nesting Job* dialog box and assign the following values to the parameters:

1. In the *Sheet list* grid, click on *Select to add sheet*.
2. **Thickness & Material:** By default, the thickness and material of the first part given in the *Part list* grid of the *Part Data* tab is assigned as the thickness and material in the Thickness (2mm) and Material (Steel) fields respectively. Leave these parameter values as it is.
3. **Quantity:** In the Quantity field, assign a quantity of **'1'**.
4. **Grain Direction:** Leave the Grain direction set to *None*.
5. **Assembly Template:** Leave the Assembly Template set to *Default*.
6. **Adding a Standard Sheet:** To add a Standard size sheet,

i. Select the *Standard size* option.

ii. In the *Standard size* dropdown list, select S3 (6” X 6”)

7. Click on the *Add sheet* button to add the sheet to the Sheet List.



Defining the Sheet data and Nesting Data Parameters in the Create Nesting Job dialog box

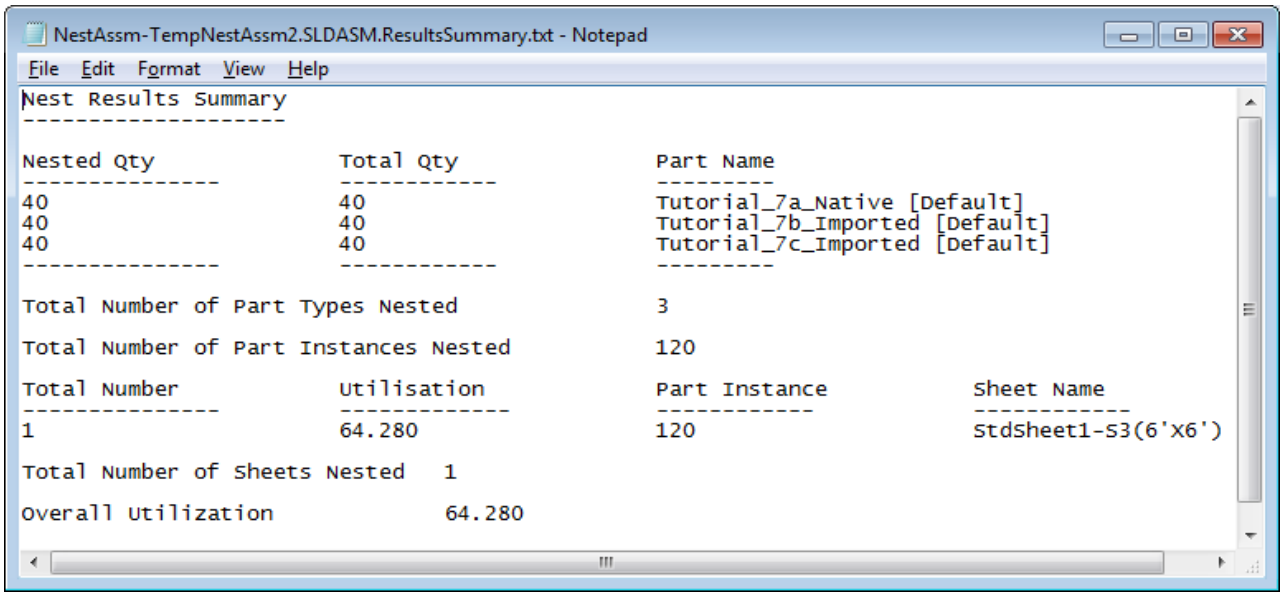
Nesting Parameters

In the Nesting Data group box, assign the following parameters:

1. Set a *Part to Part distance* of 10 mm and a *Part to Sheet distance* of 10 mm.
2. *Fast Nesting* is the default *Nesting type*.
3. Use the ... button next to the *Output assembly* field to specify where the nested layout assembly and the Summary text file that are generated will be saved.
4. Leave the *Create separate assembly* checkbox unchecked.
5. Click *OK* to execute the nesting process.

Step 4: Generating the Nested Layout

The Summary text file indicates that the prescribed quantities for all the parts have been nested within the sheet.



NestAssm-TempNestAssm2.SLDASM.ResultsSummary.txt - Notepad

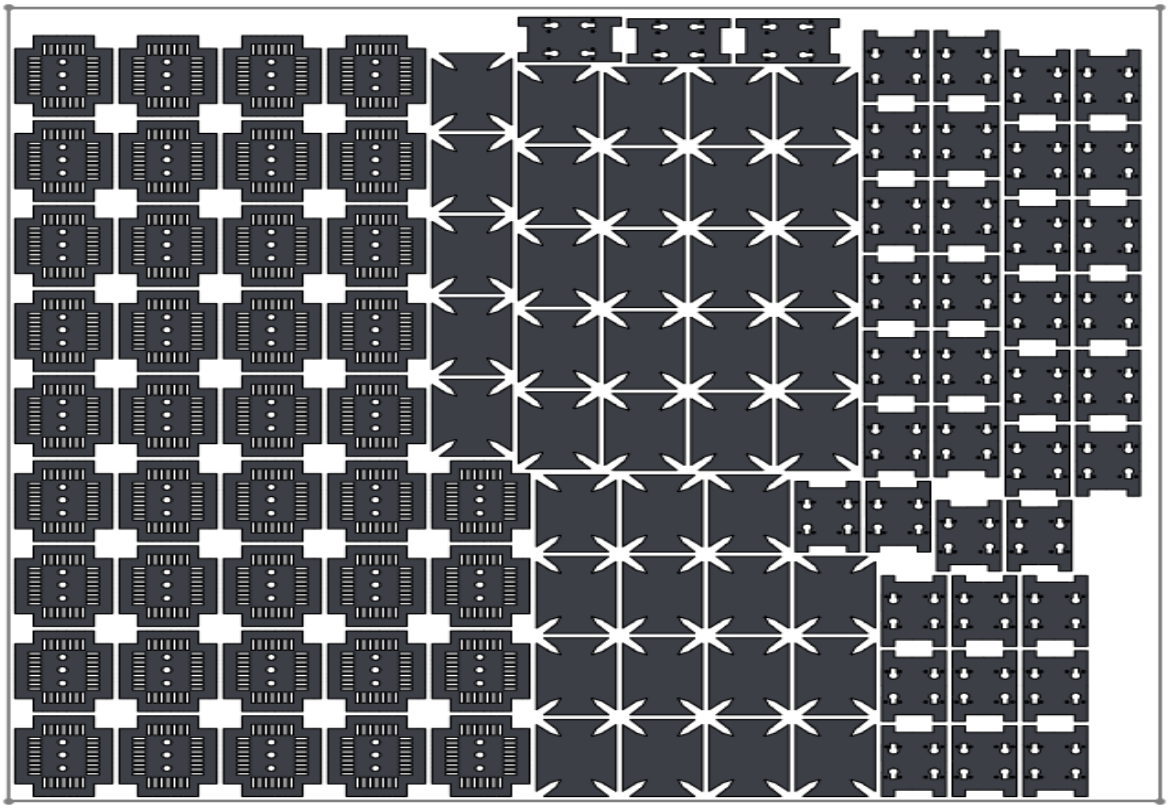
File Edit Format View Help

```
Nest Results Summary
--------------------

Nested Qty              Total Qty              Part Name
---------------         ------------           ---------
40                      40                     Tutorial_7a_Native [Default]
40                      40                     Tutorial_7b_Imported [Default]
40                      40                     Tutorial_7c_Imported [Default]
---------------         ------------           ---------

Total Number of Part Types Nested              3

Total Number of Part Instances Nested          120

Total Number            Utilisation            Part Instance          Sheet Name
---------------         -------------          ------------           ------------
1                       64.280                 120                    StdSheet1-S3(6'X6')

Total Number of Sheets Nested   1

Overall Utilization             64.280
```

Nesting Results Summary Text file

The nested layout assembly generated after executing the nesting job and the Summary Results file are stored in the same location as the parts to be nested.

All 40 instances of the each sheet metal part are nested within the same sheet.



Nesting layout generated for all the three sheet metal parts comprising the assembly

TUTORIAL 8 – UNFOLDING IMPORTED 3D SHEET METAL COMPONENTS WITH FAULTY SURFACES

Introduction

The previous tutorial (Tutorial 7) explained how to unfold imported 3D sheet metal components using the 'Unfold Imported Bodies' dialog box before nesting such parts.
However, CAMWorks Nesting cannot fully unfold imported sheet metal parts if such parts have faulty bodies or surfaces. Tutorial 8 explores how to nest imported sheet metal parts containing faulty bodies or surfaces.
It is recommended that you go through the concepts explained in Tutorial 7 before commencing with this tutorial.

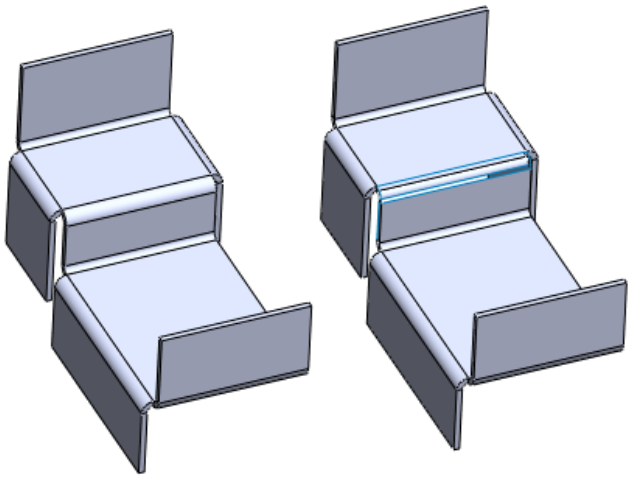
Topic covered in this Tutorial:

- Selective Unfolding of faulty parts
- Using the 'Unfold Imported Bodies' dialog box to unfold imported sheet metal parts with faults

STEP 1: Open the Assembly

Open the assembly file **Tutorial8_Unfold_Assembly.sldasm** in the following folder location.

Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies\Tutorial8



Tutorial8_Unfold_Assembly.sldasm

This assembly comprises of two identical sheet metal parts:

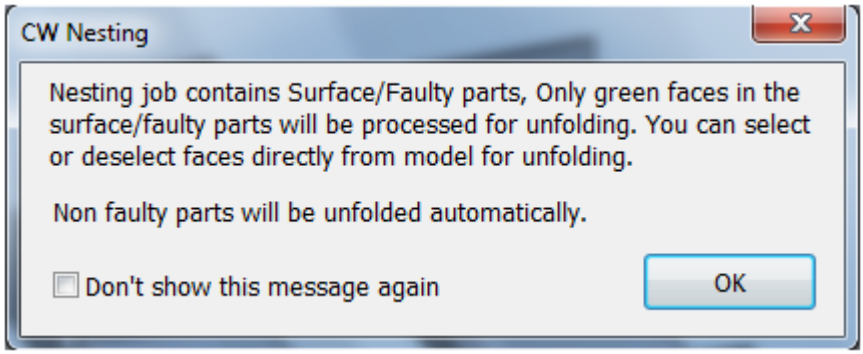
i. Tutorial_8a_Faulty.sldprt *(imported part)*

ii. Tutorial_8b_Non-Faulty.sldprt *(imported part)*

Though these two parts are identical, one of the parts has a faulty body.

STEP 2: Executing the 'Intelligent Unfold' command

1. In the CAMWorksNesting menu, click on the menu item *Intelligent Unfold*.
2. Since one of the parts comprising this assembly has a faulty body, CAMWorks Nesting will display a message stating that the assembly contains faulty parts/surfaces and that you will need to select/deselect faces directly on the model for unfolding.



CAMWorks Nesting message indicating the presence of faulty imported parts with bends

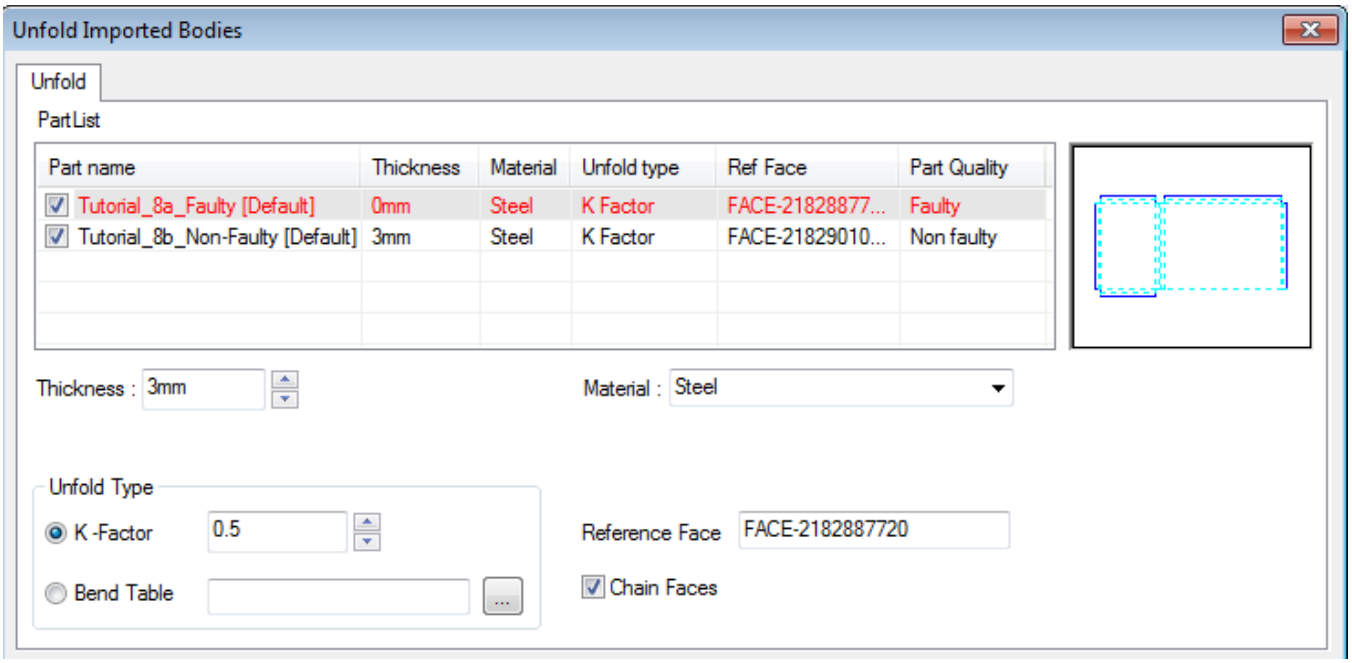
3. Click *OK* to close this dialog box and proceed with nesting.
   If you don't wish to see this error message again in future nesting jobs, select *Don't show this message again* before you click OK.
   If you click *CANCEL* by selecting the close button on the top right hand corner of this message, then the job will not proceed for nesting.
4. The *Unfold Imported Bodies* dialog box is displayed. All the imported parts with bends that constitute this assembly are listed *Unfold* tab of this dialog box. The faulty sheet metal part is highlighted in red colored font while the non-faulty part will be displayed in the default black colored font.

STEP 3: Selective unfolding of imported parts

As explained in Step 2, the 'Unfold Imported Bodies' dialog box is used to facilitate the unfolding of imported sheet metal parts and associated parameters before proceeding with a nesting job.

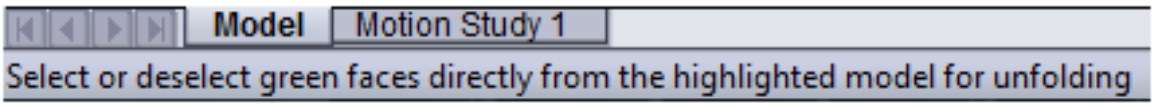
Following are the steps to open the 'Unfold Imported Bodies' dialog box to unfold the sheet metal parts and assign associated parameters:

1. Observe the Part Quality column in the Unfolded Imported Bodies dialog box. This column indicates which imported parts are faulty and which are non-faulty.



The 'Unfold Imported Bodies' dialog box

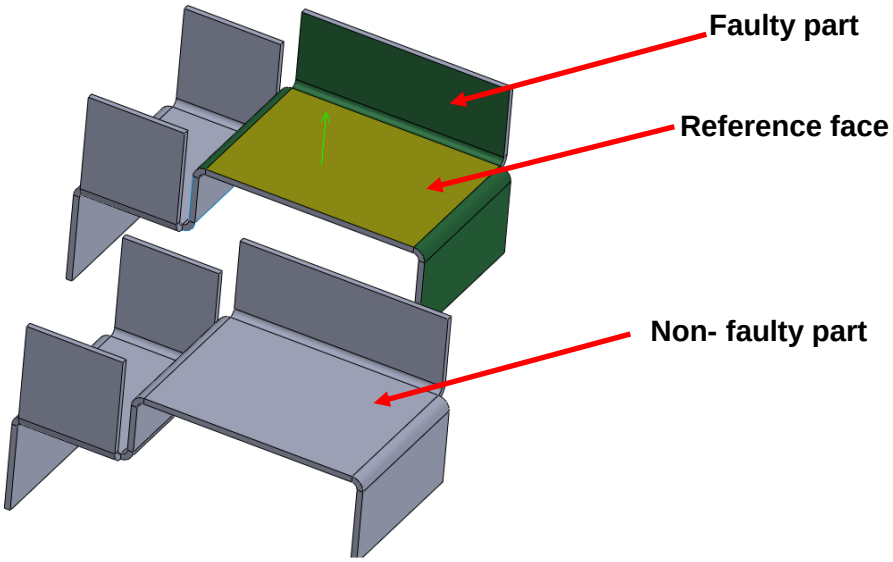
2. Highlight the Faulty part in the *Part List* grid of this dialog box. Observe that a Status message is displayed in the bottom left hand corner of SOLIDWORKS/CAMWorks Solids.



Status message displayed when a faulty part is highlighted in the 'Unfold Imported Bodies' dialog box

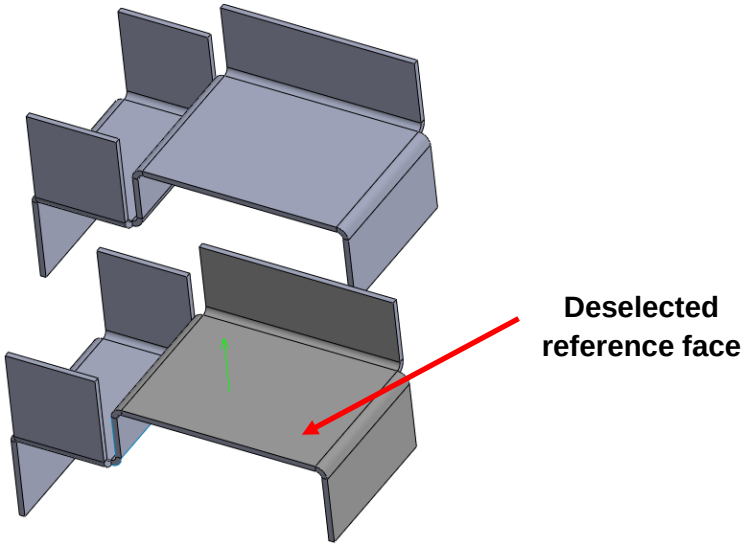
3. In the graphics area, observe the faulty part. Notice that the certain bent edges of the faulty part are highlighted in green. Rotate the assembly so that the bottom surfaces are visible in order to get a clear view.
4. In the graphics area, observe the faulty sheet metal part (*Tutorial_8a_Faulty.sldprt*). Notice that the certain bent edges of the faulty part are highlighted in green. All the faces tangentially connected to the reference face and will be highlighted in **dark green** colors on the faulty part. The default reference face for a part is always the surface

with the largest area. This reference face is highlighted in **light olive green** color.



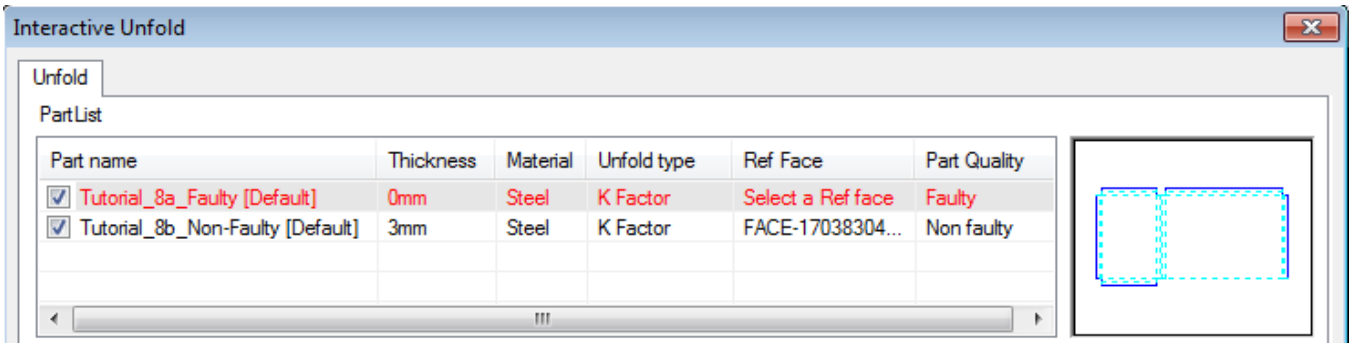
Tangentially connected faces highlighted in green on the faulty part

5. As indicated by the status message, only the highlighted green faces in the faulty part will be processed for unfolding while non-faulty part(s) will be unfolded automatically.
6. You can remove all the faces thus selected for unfolding by clicking on the reference face of the part in the graphics area. (The reference face is the one with the normal arrow and always highlighted in light olive green color). When you click on the highlighted reference face, all the selected faces, including the reference face, are discarded.



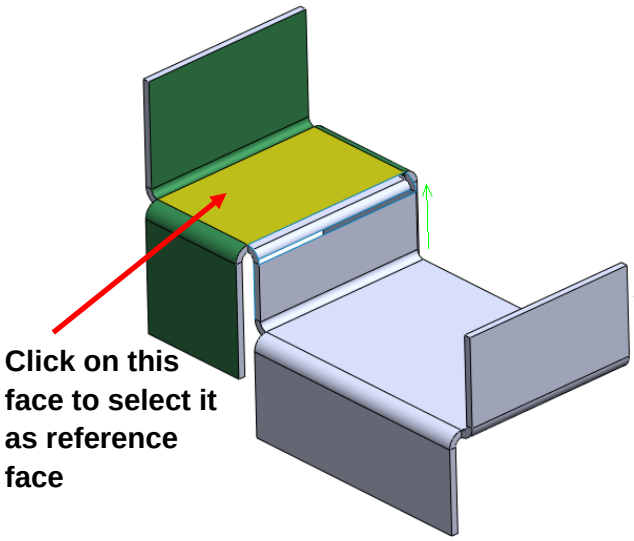
All highlighted green faces are deselected from unfolding when you click on the reference face

7. Observe the *Unfold Imported Bodies* dialog box. Since the faulty part no longer has a reference face, the *Reference Face* field in this dialog box displays a message prompting you to select a reference face on the part.



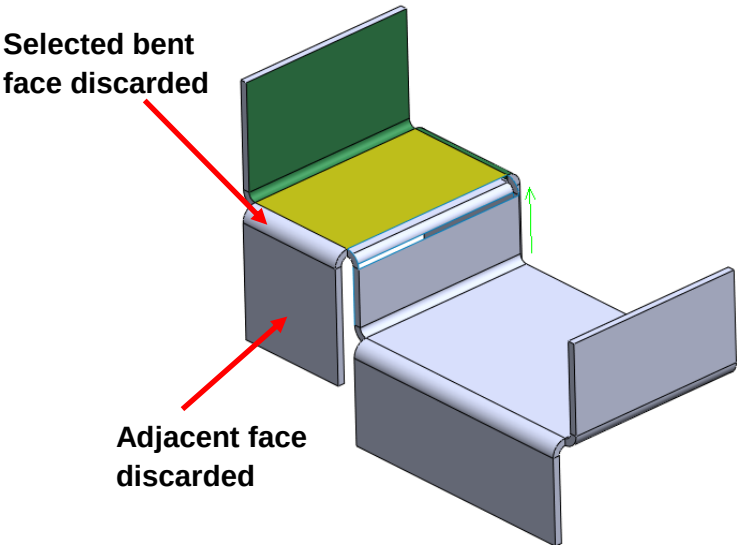
Message in Ref Face column prompting user to select a reference face

8. Rotate the part so that the top surfaces are once again visible. In the graphics area, pick the planar surface on the faulty part as shown in the image on the right. This planar surface will become the reference face (highlighted in light olive green color) for the faulty part. All the corresponding faces tangent to this reference face will be highlighted in green. Faces highlighted in green indicate that they will be unfolded.



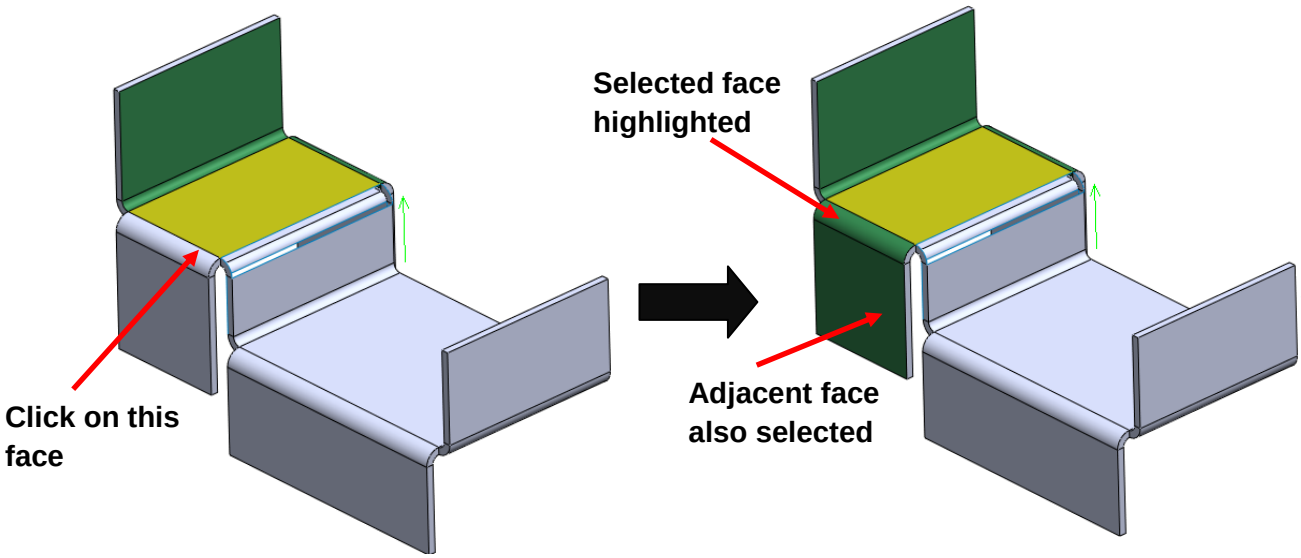
9. If you select one of the highlighted green surfaces, then all the highlighted green faces which are adjacent to the selected face and also disconnected from the reference face will be discarded from unfolding.

   For example, click on the bend adjacent to the reference face as shown in the image on the left. Observe that this bent surface as well as the planar face that was adjacent and connected to it was discarded.

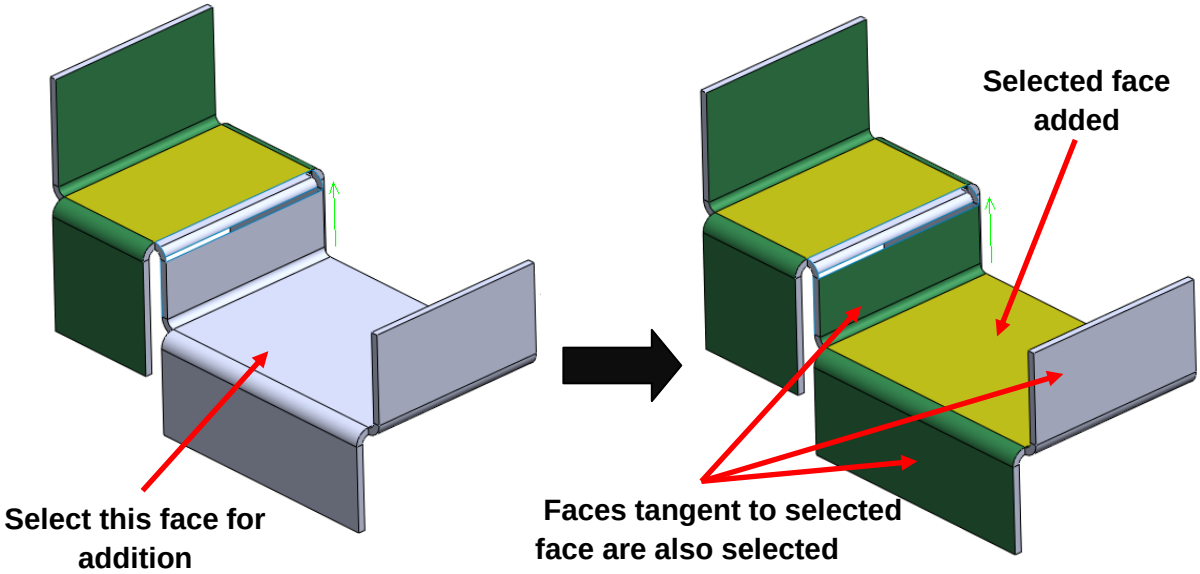


10. In the graphics area, click on the bent face of the faulty part that you deselected in the previous

step. Observe that this face and its adjacent face (which was disconnected from the reference face) are once again highlighted.



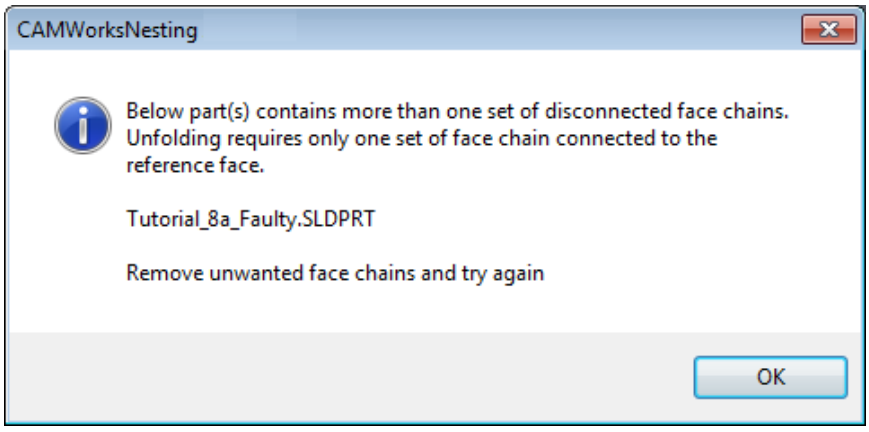
11. To add faces for unfolding to the set of green faces already highlighted, click on that desired face in the graphics area. This action also selects all faces tangent to the selected face for unfolding.



When you select a non-highlighted face for addition, all faces tangent to this face are also selected for unfolding

12. In the *Unfold Imported Bodies* dialog box, assign a thickness of 3mm to the faulty part.
13. Click the *OK* button of this dialog box to unfold the imported parts. CAMWorks Nesting will check the imported parts for the number of disconnected face chains. If the selected part still retains more than one disconnected face chain, then an error message will be displayed prompting you to remove unwanted face chains. The parts won't be

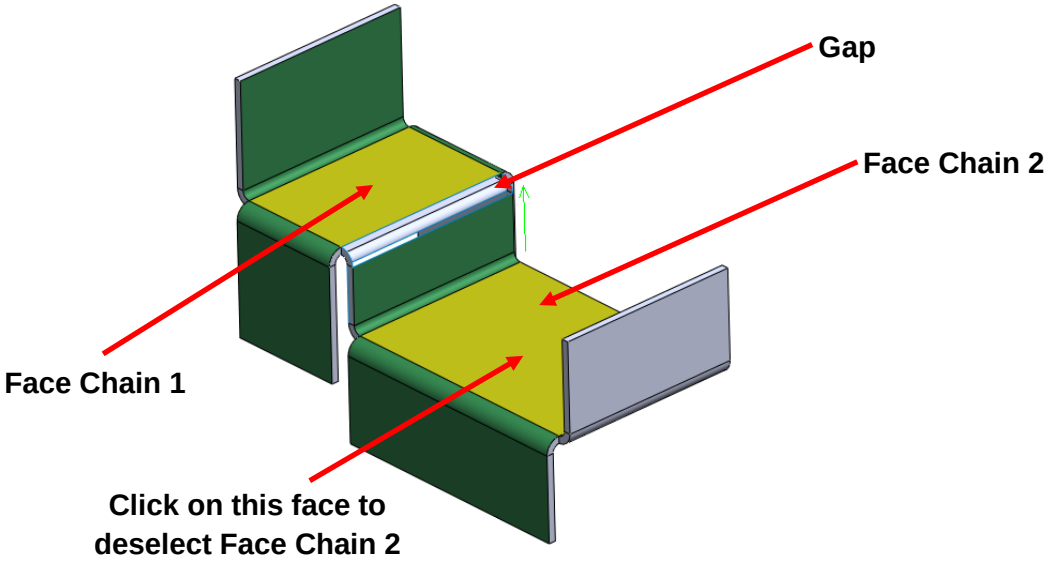
unfolded and the *Unfold Imported Bodies* dialog box will remain displayed.



Error message dialog box

14. This error message is displayed because two faces you selected as references faces for the faulty part do not form a single face chain. The presence of a gap creates two face chains. For imported sheet metal parts, CAMWorks Nesting can unfold only a single face chain. If multiple disconnected face chains are manually selected by you, then this error message regarding the presence of multiple face chains will be displayed and the nesting process will not proceed further. Observe the non-faulty part. There is only a single face chain in this part.

For imported sheet metal parts containing bends which are to be unfolded before nesting, CAMWorks Nesting can only unfold any one single face chain. Multiple face chains cannot be unfolded and thereby cannot be nested.



CAMWorks Nesting cannot unfold an imported sheet metal part with bends when multiple face chains are present

15. To proceed with nesting, you need to ensure that only one face chain is selected on the faulty part. Since there are two face chains present, deselect one of the face chains by clicking on face chain with the smaller reference area as shown in the above image.
16. Click on the *OK* button of the *Unfold Imported Bodies* dialog box. Observe that for the non-faulty imported part, its entire body is flattened. For the faulty part, only the faces highlighted in green are flattened. The *Create Nesting Job* dialog box is displayed.

STEP 4: Executing the Nesting Job

Part Data Tab

In the *Part data* tab of the *Create Nesting Job* dialog box, assign the following values to the parameters:

1. **Thickness & Material:** Observe that the Thickness and the Material of all the two parts are identical. These default values will remain unchanged. Identical thickness and material will enable nesting of these parts in the same sheet.
2. **Quantity:**

   Assign a Quantity of **80** to part named *Tutorial_8a_Faulty*.

   Assign a Quantity of **65** to part named *Tutorial_8b_Non-Faulty*.

3. **Step Angle:** Assign a Step Angle of **90 degrees** to both the parts.
4. **Grain Direction:** Leave the Grain direction set to *None* for both the parts.

Sheet Data Tab

In this tutorial, we will use a customized sheet [Length = 2200mm; Width = 2200mm] to nest both the imported parts.

Click on the *Sheet data* tab of the *Create Nesting Job* dialog box and assign the following values to the parameters:

1. In the *Sheet list* grid, click on *Select to add sheet*.
2. **Thickness & Material:** By default, the thickness and material of the first part given in the *Part list* grid of the *Part Data* tab is assigned as the thickness and material in the Thickness (3mm) and Material (Steel) fields respectively. Leave these parameter values as it is.
3. **Quantity:** In the Quantity field, assign a quantity of **'1'**.
4. **Grain Direction:** Leave the Grain direction set to *None*.
5. **Assembly Template:** Ensure that the Assembly Template is set to *Default*.
6. **Adding a Custom size Sheet:** To add a Custom size sheet,
   i. Assign a length of 2200mm in the Length field.
   ii. Assign a width of 2200mm in the Width field

7. Click on the *Add sheet* button to add the sheet to the Sheet List.

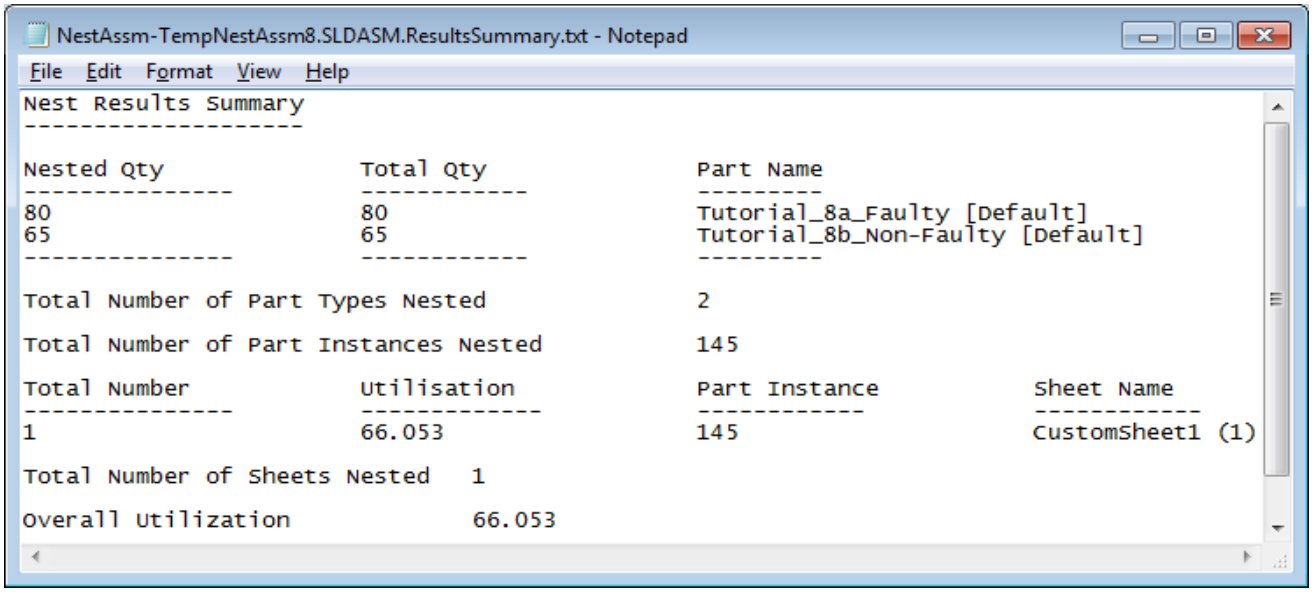
Nesting Parameters

In the Nesting Data group box, assign the following parameters:

1. Set a *Part to Part distance* of 10 mm and a *Part to Sheet distance* of 10 mm.
2. *Fast Nesting* is the default *Nesting type*.
3. Use the ... button next to the *Output assembly* field to specify where the generated nested layout assembly file and the Summary text file will be saved.
4. Leave the *Create separate assembly* checkbox unchecked.
5. Click *OK* to execute the nesting process.

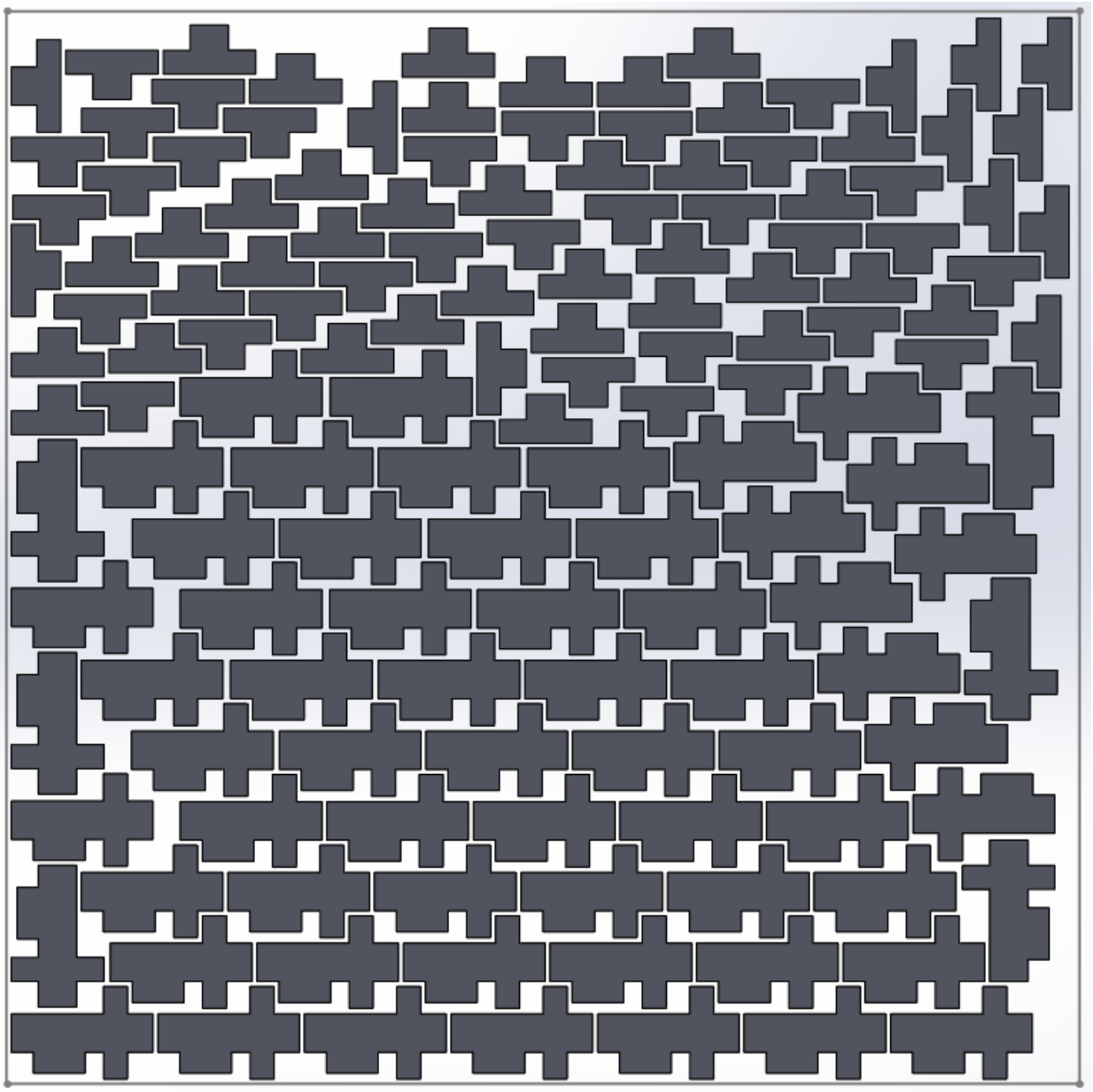
Step 5: Generating the Nested Layout

The Summary text file indicates that the prescribed quantities for all the parts have been nested within the sheet.



Nesting Results Summary Text file

The nested layout assembly generated after executing the nesting job and the Summary Results file are stored in the folder location specified in the *Output assembly* field of the *Nesting Parameters* group box in the *Create Nesting Job* dialog box.
The nested layout is given on the next page.



Nesting layout generated for the two sheet metal parts comprising the assembly

TUTORIAL 9 – ASSIGNING ASSEMBLY QUANTITIES

Introduction

CAMWorks Nesting provides a feature wherein, if an Assembly is to be nested, you can assign a quantity to the Assembly itself within the Part Data tab of the *Create Nesting Job* dialog box. Assigning the quantity to the assembly being nested automatically updates the quantities of its constituent parts. Thus, the need to assign quantity values to individual parts of the assembly is eliminated.
In this tutorial, you will explore how assigning quantity value to the assembly to be nested (or its constituent sub-assemblies) automatically updates the part quantity values.

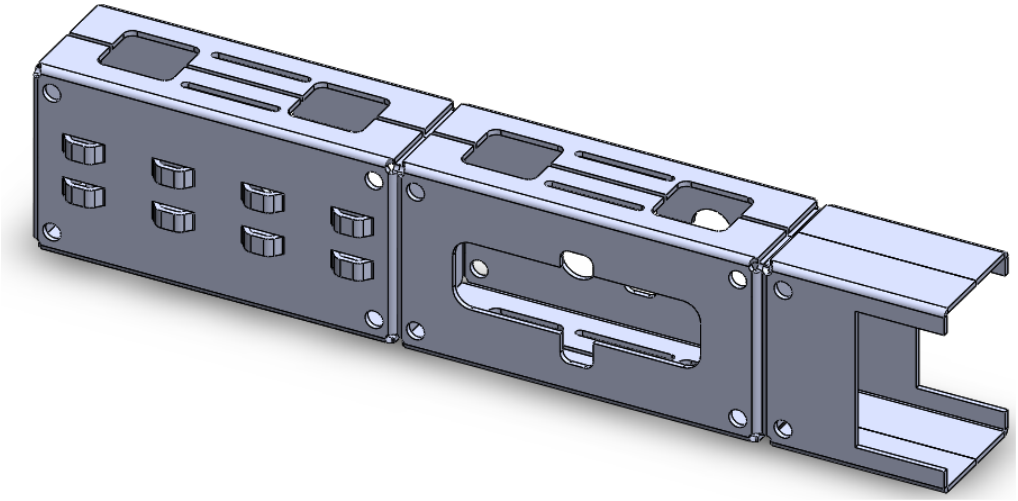
Topic covered in this Tutorial:

- Enabling the feature for Assigning Assembly Quantities
- Assigning Quantity value to the Assembly to be nested
- Assigning Quantity values to the sub-assemblies of the assembly
- Overwriting the quantities assigned automatically to the individual parts of an assembly with user-defined quantity values

STEP 1: Open the Assembly

Open the assembly file ***Parent Assembly.sldasm*** in the following folder location.

Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies\Tutorial9

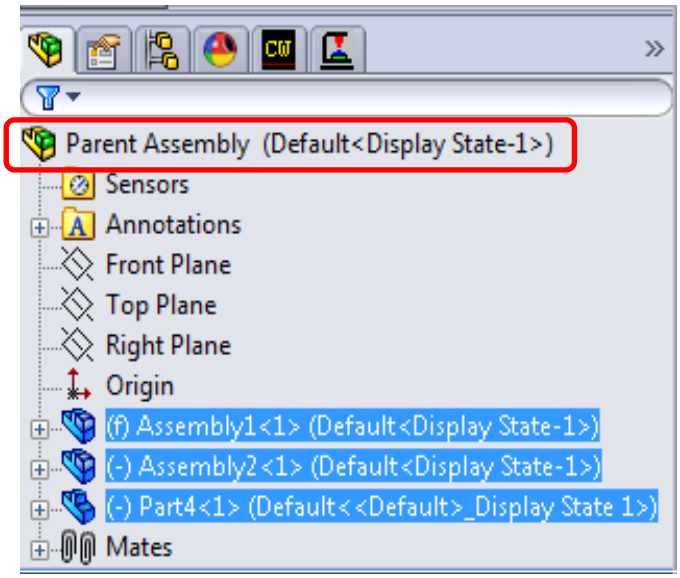


Parent Assembly.sldasm

Components of the Parent Assembly

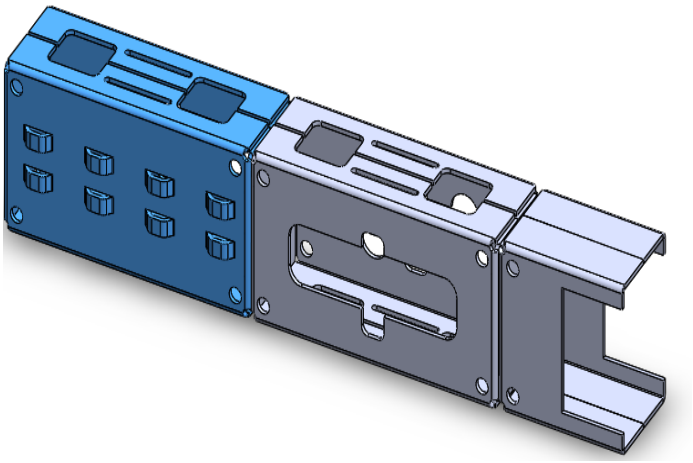
This **Parent Assembly** consists of:

i. Assembly1,
ii. Assembly2 and
iii. Part 4.

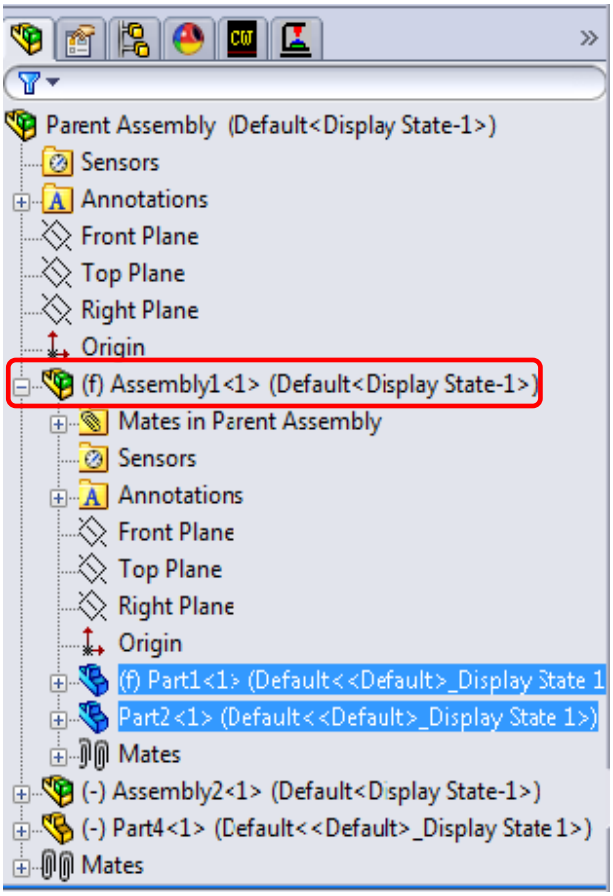


***Parent Assembly* in the FeatureManager Design tree**

Assembly 1 comprises ***Part1*** and ***Part2***.

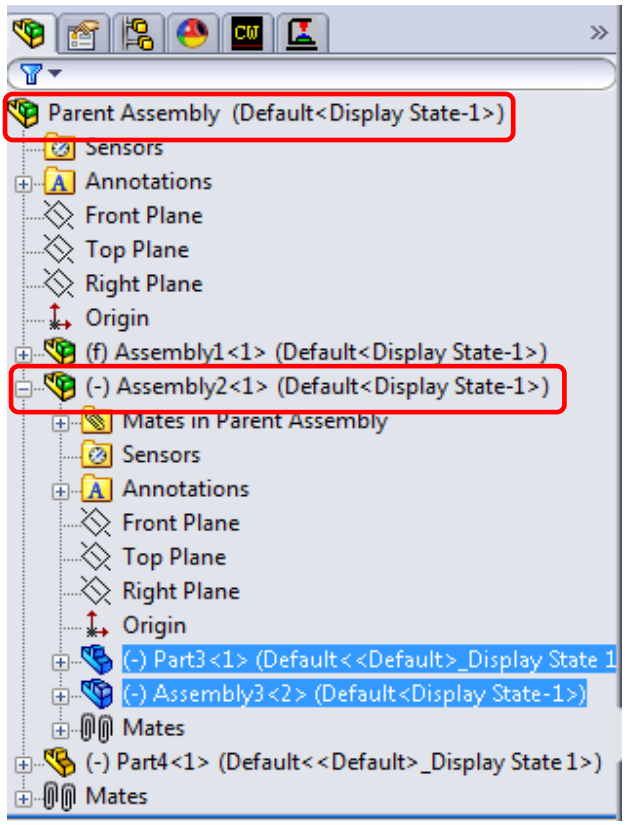


Components of *Assembly1* highlighted on *Parent Assembly*

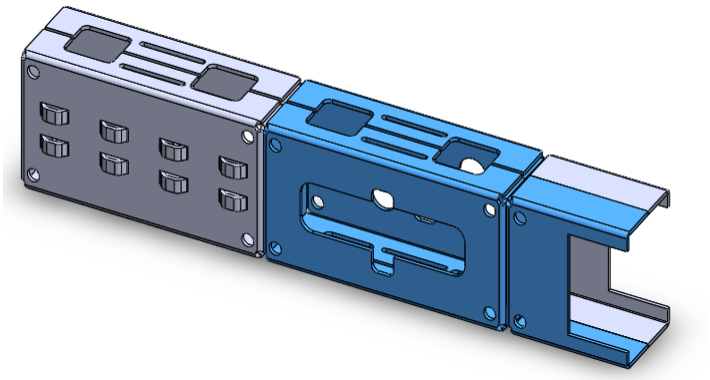


Assembly1 in FeatureManager Design tree

Assembly2 comprises ***Assembly3*** and ***Part 3*.**

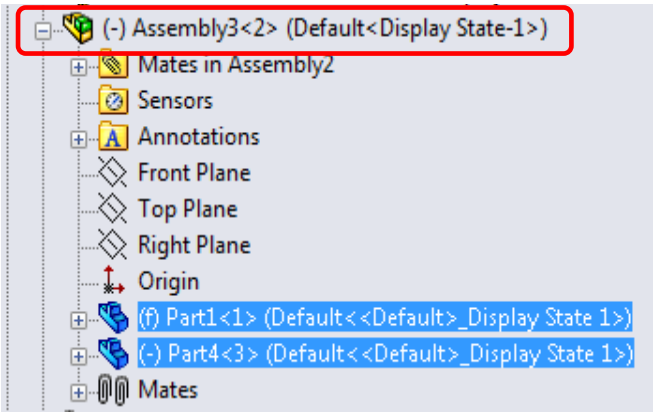


Assembly2 in FeatureManager Design tree

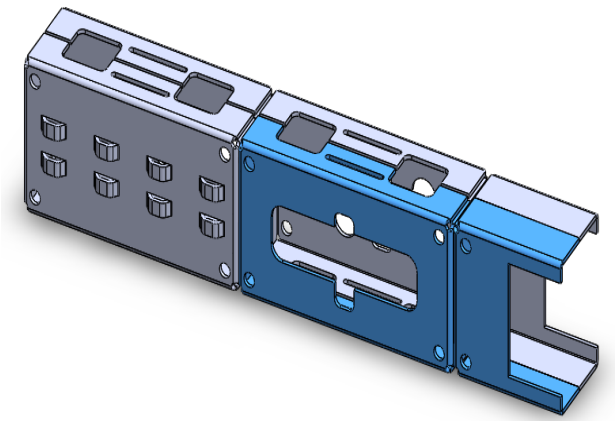


Components of *Assembly2* highlighted on *Parent Assembly*

Assembly3 (sub-assembly of *Assembly2*) comprises ***Part1*** and ***Part4***.

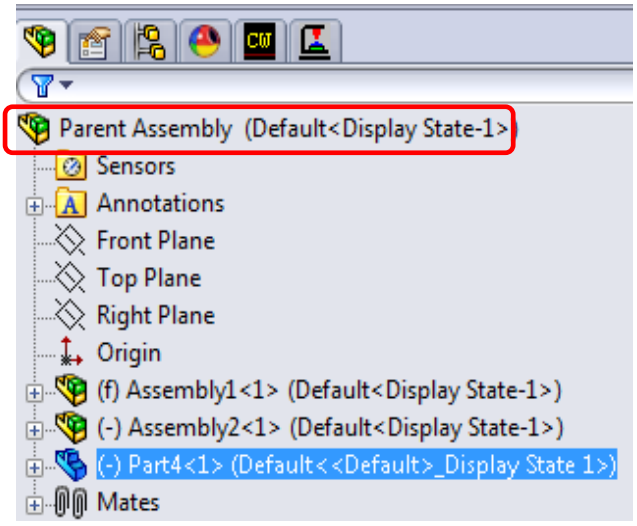


Assembly3 in FeatureManager Design tree

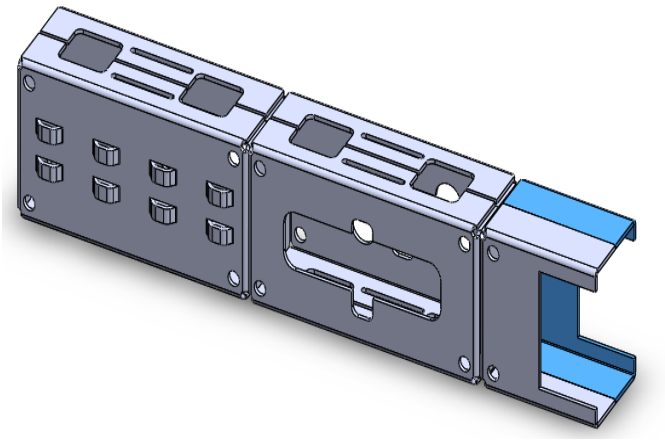


Components of *Assembly3* highlighted on *Parent Assembly*

Part4 is a component of Parent Assembly as stated previously.

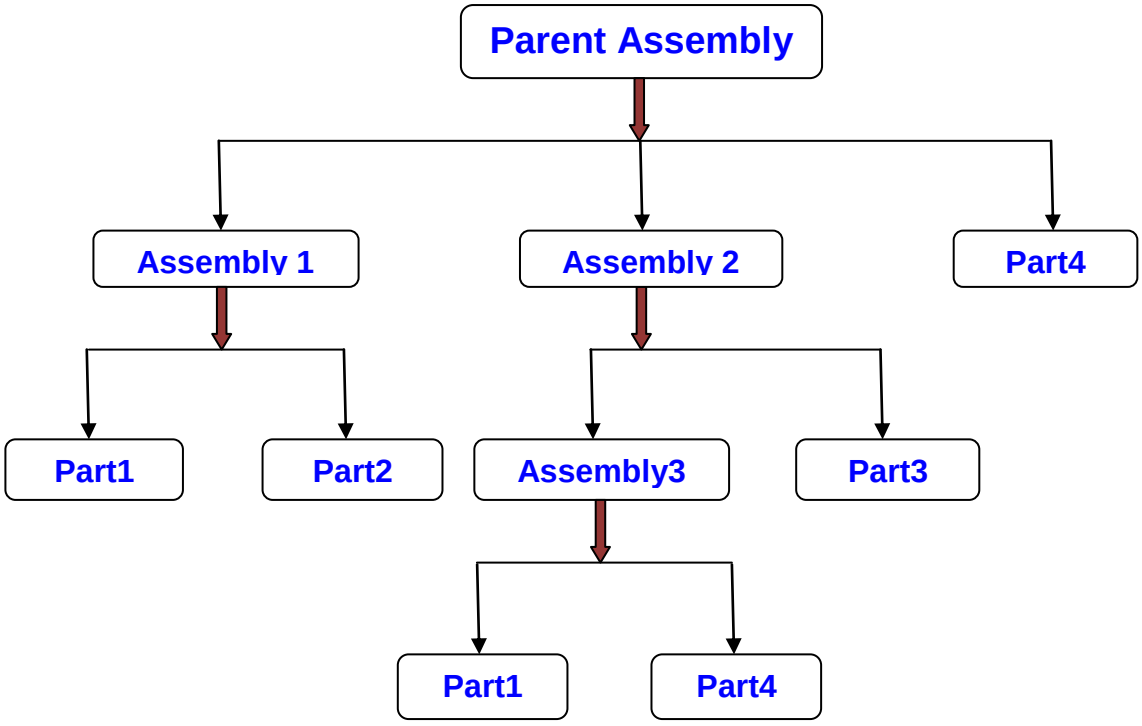


Part4 in FeatureManager Design tree



Part4* highlighted on *Parent Assembly

Thus, the Parent Assembly consists of the following sub-assemblies and parts:



STEP 2: Enabling the option of flattening the sheet metal parts

The assembly to be nested (*Parent Assembly.sldasm*) consists of sheet metal parts. These sheet metal parts need to be unfolded before executing the nesting job. To nest these sheet metal parts based on its dimensions after flattening, the option for flattening (unfolding) sheet metal parts needs to be configured in the *DefaultValues.ini* file. If you are unsure about the settings, open the *DefaultValues.ini* file and set the *FlattenSheetMetalPart* flag to '1' in order to activate the option of flattening.

STEP 3: Enabling the feature for Assigning Assembly Quantities

To assign quantity value to the assembly is to be nested (Parent Assembly.sldasm), the feature for assigning Assembly Quantities needs to be enabled in in the *DefaultValues.ini* file. If you are unsure about the settings, open the *DefaultValues.ini* file and set the *ShowAssemblyQuantity* flag to '1' in order to enable the feature of Assigning Assembly Quantities.

STEP 4: Open the 'Create Nesting Job' Dialog box

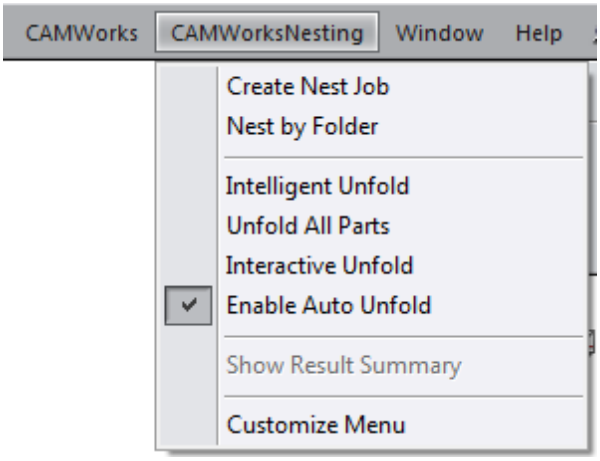
1. In the CAMWorksNesting menu bar, ensure that the *Enable Auto Unfold* option is checked.

   Activating this option ensures that sheet metal components to be nested are automatically unfolded based on their dimensions before executing the nesting job.

2. Select *Create Nesting Job* from the CAMWorksNesting menu bar.

   OR

   Click on the *Create Nesting Job* button on the CAMWorks Nesting Ribbon bar.

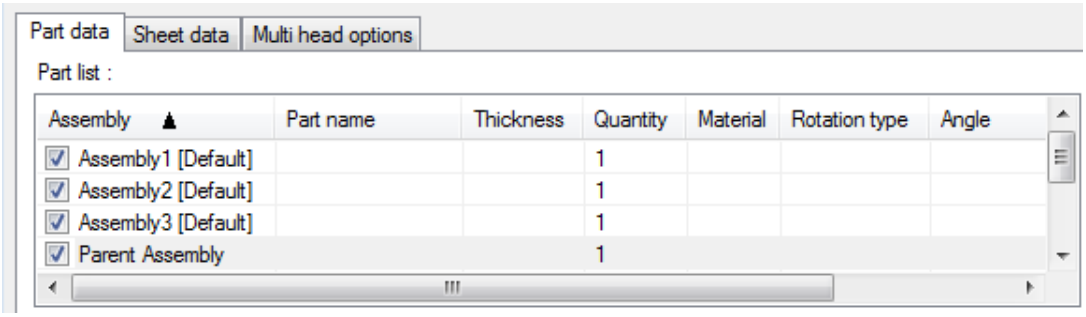


'Enable Auto Unfold' option checked in the CAMWorksNesting menu

3. The *Create Nesting Job* dialog box is displayed. Observe that the parent assembly (*Parent Assembly.sldasm*), its sub-assemblies as well as the parts comprising these sub-assemblies have been listed in the *Part Data* tab in the *Create Nesting Job* dialog box.

Assembly Column

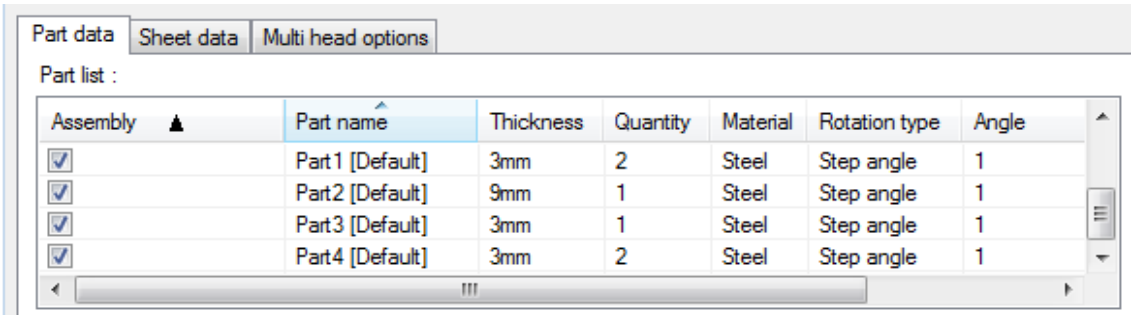
The Parent Assembly and its sub-assemblies are listed in the ***Assembly*** *column* of the Part List in alphabetical order. The quantity of the *Parent Assembly* and its sub-assemblies are listed in the ***Quantity column***.



Parent Assembly and sub-assemblies listed in the Part List grid of Part Data tab

Part name column

Use the vertical scroll bar to scroll down the Part List. Observe that the parts constituting the Parent assembly and sub-assemblies are listed in the *Part name* column along with associated parameters of *Thickness*, *Quantity*, *Material*, *Rotation type*, *Angle*, etc.



Parts constituting the Parent Assembly and sub-assemblies listed in the Part List

Quantity Column

Observe the Quantity column. The number of instances each sub-assembly and part appears in the Parent Assembly is listed. In the Quantity column, ensure that the Quantity for Parent Assembly is '1'. (If it isn't, double-click on this field in the *Part list* grid and assign a quantity of '1'.)

When the quantity for Parent assembly is '1', the quantity of its constituent sub-assemblies and parts will be as follows:

Name	Quantity
Parent Assembly	1
Assembly1	1
Assembly2	1
Assembly3	1
Part1	2
Part2	1
Part3	1
Part4	2

Step 5: Changing the quantity of the Parent assembly

1. In the *Part list* grid, double-click on the Quantity field for Parent Assembly and change the Quantity to '20'. Press the Tab button to shift the focus.

   Observe that the Quantity values of all the constituent sub-assemblies and parts were automatically updated to reflect new values based on the Parent Assembly.

Name	Original Quantity	New Quantity
Parent Assembly	1	20
Assembly1	1	20
Assembly2	1	20
Assembly3	1	20
Part1	2	40
Part2	1	20
Part3	1	20
Part4	2	20

Analysis

i. **Assembly1 and Assembly2:** Both *Assembly1* and *Assembly2* have one instance each in the Parent Assembly. Hence, when the quantity of the Parent Assembly is updated, the quantity of this sub-assembly too will be updated to the same value.

ii. **Assembly3:** *Assembly3* has one instance in *Assembly2* which in turn has one instance in the Parent Assembly. Hence, when the quantity of the Parent Assembly is updated, the quantity of this sub-assembly too will be updated to the same value.

iii. **Part1:** This part has one instance in *Assembly1* and another instance in *Assembly3*. Hence, two instances of *Part1* occur within the Parent Assembly. Hence, when the quantity of the Parent Assembly is updated, the quantity of *Part1* will be updated to double of the Parent Assembly's quantity value.

iv. **Part2:** This part has one instance in *Assembly1* which in turn has one instance in the Parent Assembly. Hence, when the quantity of the Parent Assembly is updated, the quantity of *Part2* will be updated to the same value as that of the Parent Assembly.

v. **Part3:** This part has one instance in *Assembly3* which has one instance in *Assembly2* and which in turn has one instance in the Parent Assembly. Hence, when the quantity of the Parent Assembly is updated, the quantity of *Part3* will be updated to the same value as that of the Parent Assembly.

vi. **Part4:** This part has one instance in *Assembly2* and another instance in the Parent Assembly. Hence, two instances of *Part4* occur within the Parent Assembly. Hence, when the quantity of the Parent Assembly is updated, the quantity of *Part4* will be updated to double of the Parent Assembly's quantity value.

Thus, if you change the quantity of the assembly to be nested, then the quantity of all its constituent sub-assemblies and parts will be automatically updated to values in sync with quantity of the assembly.

This feature eliminates the need to assign individual quantities to the constituent parts.

STEP 6: Changing the Quantity of a sub-assembly

1. In the *Part list* grid, double-click on the Quantity field for *Assembly2* and change the Quantity to '22'. Press the *Tab* button to shift the focus.

   Observe that the Quantity values for *Assembly3*, *Part1*, *Part3* and *Part4* were automatically updated to reflect new values based on the quantity of *Assembly2*.

Name	Original Quantity	New Quantity
Parent Assembly	20	20 (unchanged)
Assembly1	20	20 (unchanged)
Assembly2	20	22 (updated)
Assembly3	20	22 (updated)
Part1	40	42 (updated)
Part2	20	20 (unchanged)
Part3	20	22 (updated)
Part4	20	42 (updated)

Analysis

i. **Assembly1:** *Assembly1* is not a component of *Assembly2*. Hence, its Quantity value will remain unchanged.

ii. **Assembly3:** Within the Parent Assembly, the sub-assembly named *Assembly3* has only one instance - as component of *Assembly2*. Hence, when the quantity of the *Assembly2* is updated, the quantity of this sub-assembly too will be updated to the same value.

iii. **Part1:**

Part1 has 20 instances in *Assembly1* and 20 instances in *Assembly3* → 20 +20 = **40**.

Assembly1 is not a component of *Assembly2* and hence, the first 20 instances will remain unchanged.

However, *Assembly3* has once instance in *Assembly2*. Hence, when the quantity of *Assembly2* is updated from '20' to '22', the quantity of *Assembly3* will be updated to '22'. Since Part4 has one instance in Assembly3, the quantity component will be updated to '22'.

The updated Quantity value for *Part1* will be 20 instances in Assembly1 and 22 instances in Assembly3 → 20+22 = **42**.

iv. **Part2:** *Part2* is not a component of *Assembly2*. Hence, its Quantity value will remain unchanged at '**20**' instances.

v. **Part3:** This part has 20 instances in *Assembly3* which in turn has 20 instances in *Assembly2*. Hence, when the quantity of *Assembly2* is updated to '22', the quantity of *Part3* will be updated to the same value as that of the *Assembly2*.

Thus, the updated Quantity value for *Part3* will be **22** instances in *Assembly3*.

vi. **Part4:**

Part4 has 20 instances in the *Parent Assembly* and 20 instances in *Assembly2* → 20 +20 = **40**.

Since the Parent Assembly is not a component of *Assembly2*, the first 20 instances will remain unchanged.

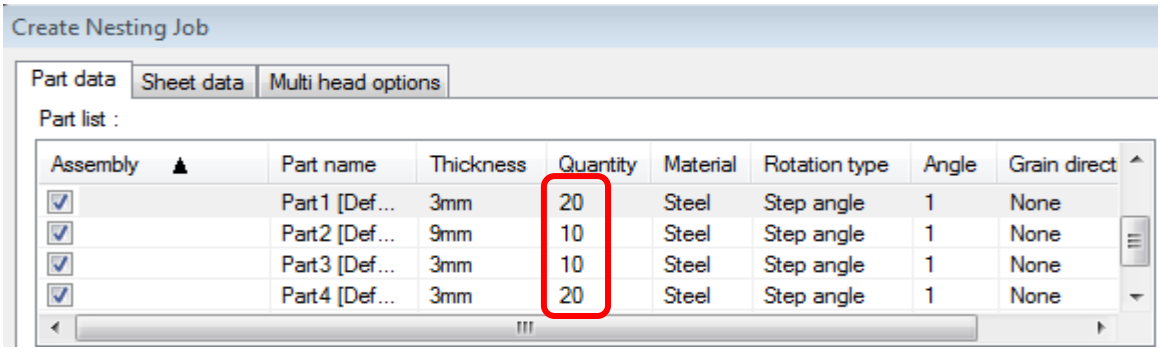
However, the next 20 instances are in Assembly2. Hence, when the quantity of *Assembly2* is updated from '20' to '22', the quantity of Part4 will also be updated to '22'.

The updated Quantity value for *Part4* will be 20 instances in Parent Assembly and 22 instances in Assembly2 → 20+22 = **42**.

Note: **If you change the quantity of a sub-assembly listed in the Part List grid of the Part Data tab, then only the quantities of its constituent sub-assemblies and parts will be updated. The quantities of parts/assemblies which are not a component of this sub-assembly will remain unchanged.**

STEP 7: Changing the Quantity of Parent Assembly

1. In the *Part list* grid, double-click on the *Quantity* field for Parent Assembly and change the quantity to '10'. Press the *Tab* button to shift the focus.
2. Observe that the quantity values of all the constituent sub-assemblies and parts were automatically updated to reflect new values based on the Parent Assembly.



Updated Result of the Parts

Name	Quantity
Parent Assembly	10
Assembly1	10
Assembly2	10
Assembly3	10
Part1	20
Part2	10
Part3	10
Part4	20

Note: **If you change the quantity of the Parent assembly, then the quantity of its constituent sub-assemblies and parts will be recalculated and automatically updated.**

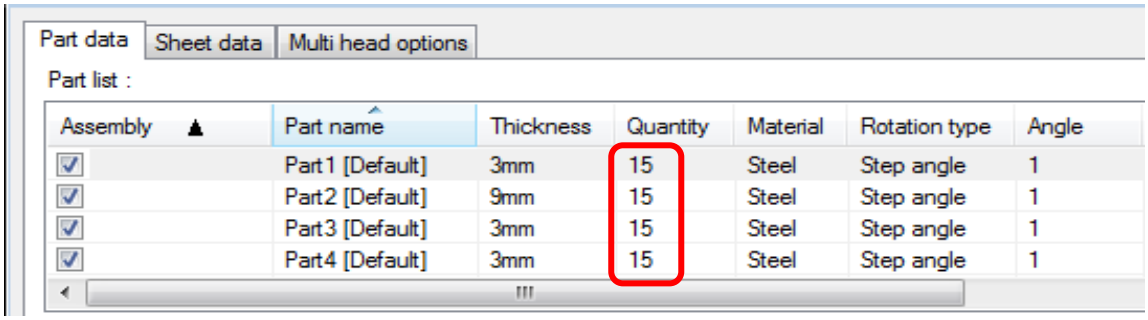
Step 8: Overwriting automatically assigned Quantity values for Parts with user-defined values

CAMWorks Nesting executes all nesting jobs based on the quantity of the parts in the Part Data tab (and not based on the quantity of the Assembly). This is the reason why, when you assign a quantity to an assembly in the Part Data tab, the quantity of its constituent parts are updated.

Even when the feature for Assigning Assembly Quantities is active, you can assign user-defined quantity values to the individual parts in the Part List grid at any point of time.

1. Double-click on the Quantity field for each of the four parts in the Part list grid and assign a quantity of '15' to each part.

   If you proceed further with the nesting job, the quantity considered will be based on these user-defined values of the parts.



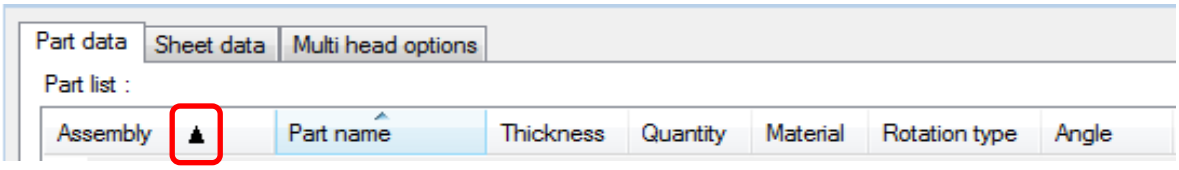
Parts constituting the Parent Assembly and sub-assemblies listed in the Part List

2. Assign a quantity of '10' to the Parent Assembly. Observe that the quantity of all the sub-assemblies and parts are updated to the values given in Step 7.

Note: **You can update the Part quantities with user-defined quantity values at any point of time even if the feature for assigning assembly quantities is active. However, your user-defined values will be overwritten with automatically assigned values if you once again assign quantity values to the assembly constituting the parts.**

Step 9: Deactivating the feature of assigning assembly quantities

1. Observe the Assembly column of the *Part list* grid of the *Part data* tab. The arrow mark pointing upwards indicates that the feature for Assigning Assembly Quantities is currently active.



Arrow mark in the Assembly column pointing upwards

If the feature for assigning assembly quantities is enabled in the DefaultValues.ini file, then, whenever you open the *Create Nesting Job* dialog box, the arrow mark in the Assembly column of the Part list grid will point upwards indicating that the feature is currently active.

Deactivating the feature only for the current nesting job

1. Left-click on the *Assembly column* heading. Observe that the arrow mark in the Assembly column now points downwards. The Part list no longer displays any assembly in the Assembly column. This column will be empty. Only parts constituting the assembly will be listed in the *Part list* grid. The arrow pointing downwards indicates the feature for assigning assembly quantities is currently inactive.

Arrow mark in the Assembly column pointing downwards

If you do not wish to use the feature of assigning assembly quantities for a particular nesting job, then deactivate this feature temporarily only the current job by having the arrow mark in the *Assembly column* point downwards.

Deactivating the feature for all nesting jobs

1. If you work primarily with parts rather than assemblies or if you prefer assigning user-defined quantities to the part of an assembly, you might consider disabling the feature of assigning assembly quantities.

   To disable the feature of assigning assembly quantities, set the *ShowAssemblyQuantity* flag in the *DefaultValues.ini* file to '0' and exit after saving the new settings.
2. With the feature disable, the next time you open the *Create Nesting Job* dialog box in order to nest an assembly, the *Assembly column* will not be displayed at all in the *Part data* tab. Only the parts comprising the assembly will be listed.
3. You can enable this feature again by setting the *ShowAssemblyQuantity* flag in the *DefaultValues.ini* file back to '1'.

Note: **When the feature for assigning assembly quantities is disabled, the Assembly column will not be displayed in the Part list grid of the Part data tab within the 'Create Nesting Job' dialog box.**

Complete the nesting job by assigning the remaining Part parameters, Sheet parameters and Nesting data parameters.

TUTORIAL 10 – UNFOLDING SHEET METAL COMPONENTS USING 'INTERACTIVE UNFOLD' COMMAND

Introduction

The 'Interactive Unfold' Command

The 'Interactive Unfold' command is a means to selectively unfold faces of parts with bends before nesting the parts. This is done by providing a reference face and selecting faces to unfold either automatically or manually with respect to the reference face. Both native as well as imported parts can be unfolded using this command.

Difference between the various Unfold commands

Given below is table of comparison highlighting the difference in the way a part is unfolded using the various Unfold commands.

	Intelligent Unfold Command	Unfold All Parts Command	Interactive Unfold Command	Create Nest Job Command
Faulty Native Part				
Non-Faulty Native Part				
Faulty Imported Part				
Non-Faulty Imported Part				

Legend:

	User can change the default reference face and define the chain of faces to unfold.
	User can change the default reference face but cannot define chain of faces to unfold.
	Command not applicable to native parts
	Opens the dialog box associated with 'Intelligent Unfold'.
	Part is unfolded using SOLIDWORKS functionality

The 'Chain Faces' option for Unfold commands

The ***Chain Faces*** is a checkbox option present in the Unfold dialog box of all Unfold commands.

- When this checkbox option is enabled, all faces connected tangentially to the reference face are automatically selected.

Chain Faces Option in Unfold dialog box

- When this checkbox option is disabled, faces connected tangentially to the reference face will not be automatically selected. You need to manually select each face to be unfolded.

If the *Chain Faces* option is enabled/disabled at any point of time during the unfolding process, then the changed settings for this option are applicable for all further selections. However, the faces already selected/deselected will not affected.

Topic covered in this Tutorial:

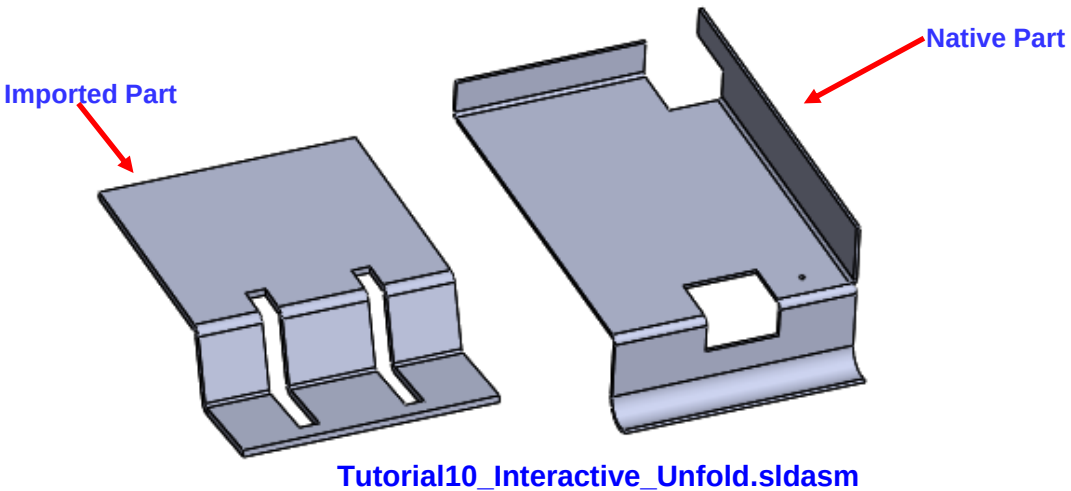
This tutorial explains how to selectively unfold imported and native 3D sheet metal parts with non-faulty bodies using the 'Interactive Unfold' command. The tutorial explores

- Using the 'Interactive Unfold' with the 'Chain Faces' option enabled
- Using the 'Interactive Unfold' with the 'Chain Faces' option disabled

STEP 1: Open the Assembly

Open the assembly file **Tutorial10_Interactive_Unfold.sldasm** in the following folder location.

Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies\Tutorial10



Tutorial10_Interactive_Unfold.sldasm

This assembly comprises of two different sheet metal parts:

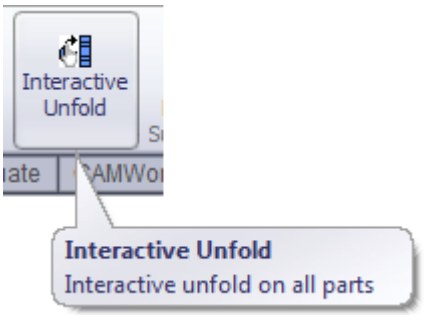
i. InteractiveUnfold_Native.sldprt (***Native Part***)

ii. InteractiveUnfold_Imported.sldprt (***Imported Part***)

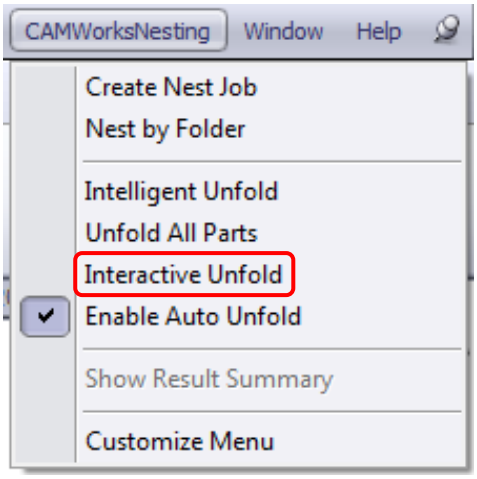
Both the parts have non-faulty bodies.

STEP 2: Executing the 'Interactive Unfold' command

1. Click on the *Interactive Unfold* command in the CAMWorksNesting menu. Alternatively, click on the *Interactive Unfold* button on the CAMWorks Nesting Ribbon Bar.

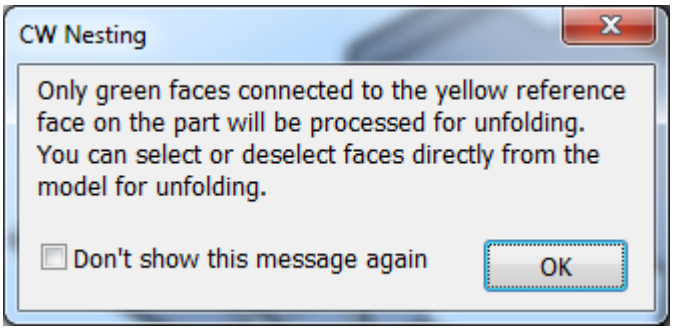


'Interactive Unfold' button in the CAMWorks Nesting Ribbon Bar



'Interactive Unfold' command in CAMWorksNesting menu

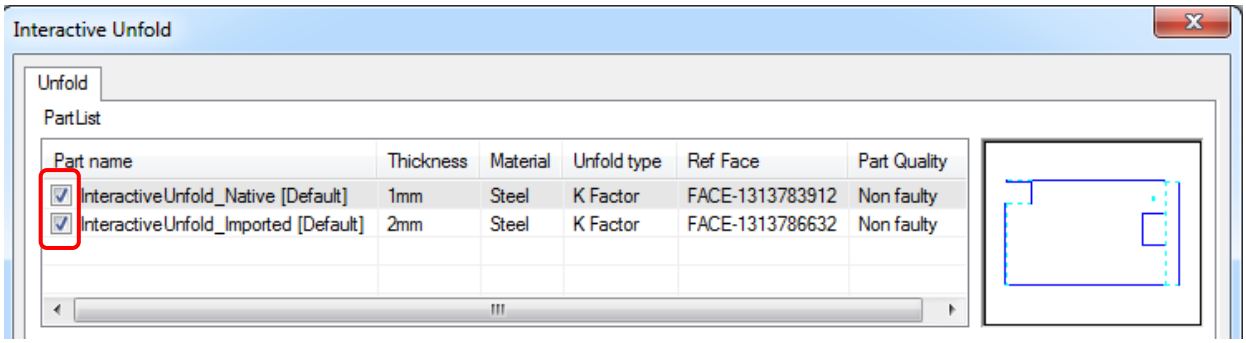
2. CAMWorks Nesting will display a message stating that you will need to select/deselect faces directly on the model for unfolding.
   - Click *OK* to close the message box and proceed with unfolding.
   - If you don't wish to see this warning message again in future nesting jobs, select the checkbox ***Don't show this message again*** before you click *OK*.



CAMWorks Nesting Message

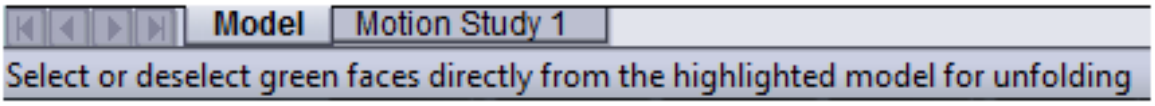
   - If you click *CANCEL* by selecting the close button on the top right hand corner of the message box, then the job will not proceed for nesting.

3. When you click the *OK* button in the message, the *Interactive Unfold* dialog box is displayed. All the parts comprising the assembly (both native and imported) are listed in the Part List grid of this dialog box.
   Since the assembly used in this tutorial consists of one imported part and one native part, both are listed in the *Part List* grid.
4. If you do not wish to unfold any particular part listed in the Part List grid, you can deselect the part from unfolding by unchecking the checkbox next to the part name in the *Part List* grid.
   In this tutorial, you will unfold both the listed parts. Hence, both the checkboxes indicating the parts to be unfolded will remain checked.



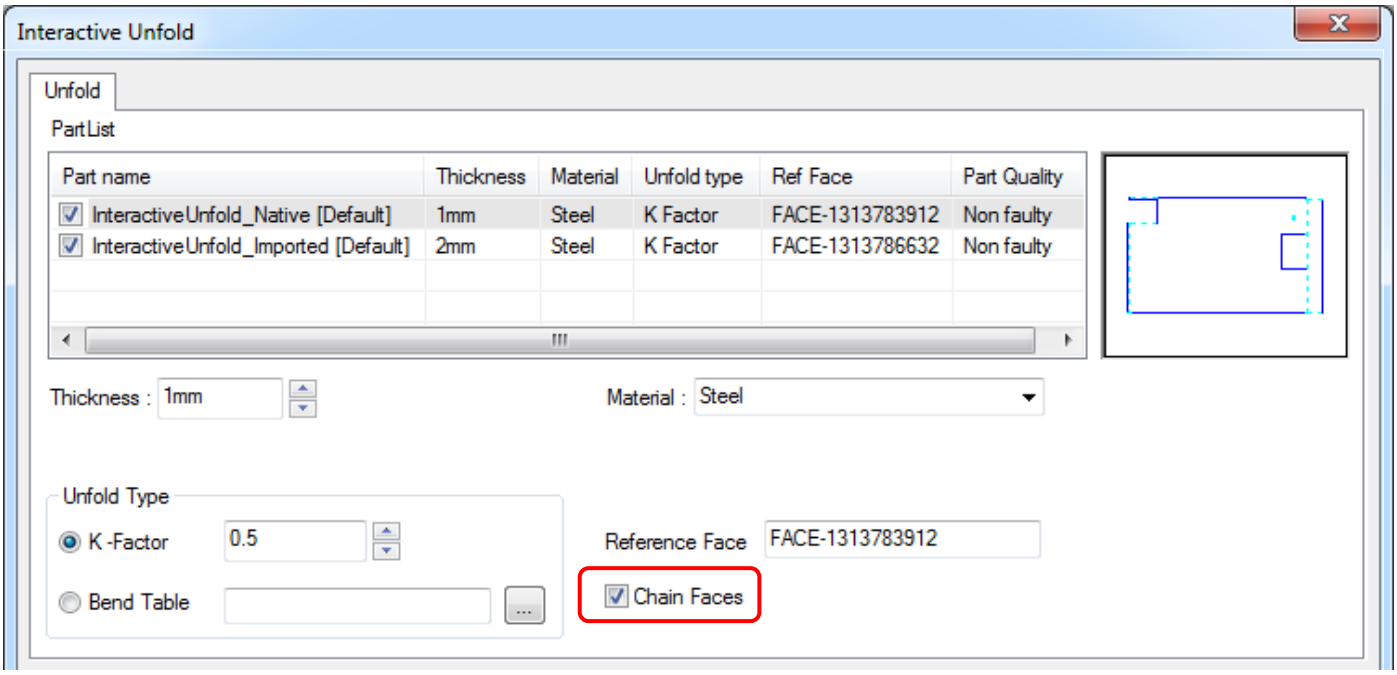
STEP 3: Selective Unfolding of Parts when 'Chain Faces' option is enabled

1. Highlight any part in the *Part List* grid of the *Interactive Unfold* dialog box. Observe that a Status message is displayed in the bottom left corner of the Status bar of SOLIDWORKS/CAMWorks Solids.



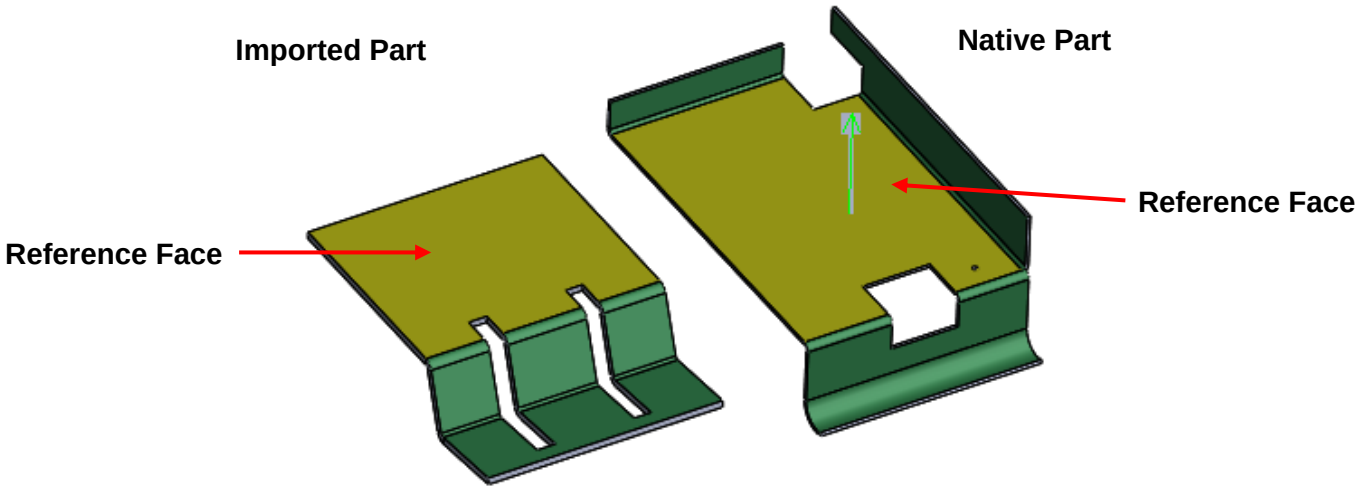
Status message displayed when a part is highlighted in the 'Interactive Unfold' dialog

2. In the *Interactive Unfold* dialog box, both the non-faulty imported and non-faulty native parts are listed in the *Part List* grid and will be processed through selective unfolding.



Interactive Unfold dialog box

3. In the graphics area, observe that a reference face (**yellow color**) and faces connected tangentially (**green color**) to the reference face are highlighted automatically on both the imported and native parts, thus indicating the faces to be unfolded.



Reference Faces and Faces tangentially connected to the reference face highlighted in Yellow and Green color respectively

Deselecting/ Selecting the faces to be unfolded

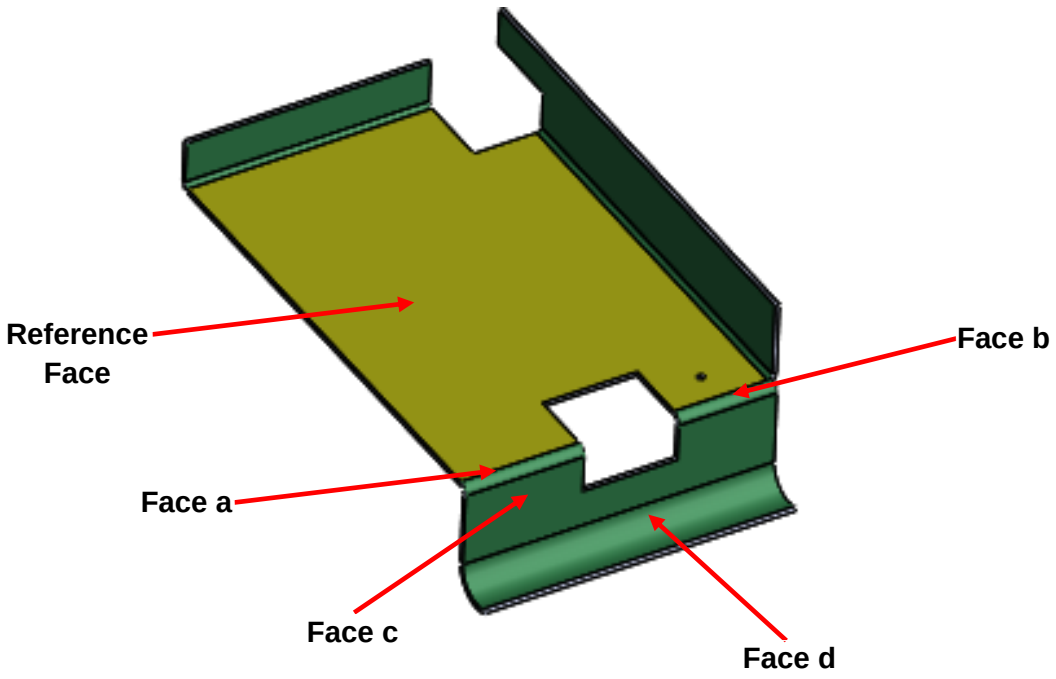
Faces of the part which are selected for unfolding are highlighted in **green color** when the *Interactive Unfold* dialog box is open. If you wish to deselect such a face from being unfolded, then you need to mouse click on that face of the part in the graphics area.

- **When you deselect a green-colored face, then all the faces which are adjacent to the deselected face but now disconnected from the reference face due to this deselection will also be deselected automatically.**
- **Similarly, when you select an unselected face, all the faces adjacent to it that now connected to the reference face due to the selection will be selected automatically. Their selection is indicated by green color highlights.**

This feature of automatic selection/deselection of faces is possible only when the *Chain faces* option is enabled.

In this section, we will explore the selection/deselection of the faces to be unfolded when the *Chain Faces* option is enabled.

Consider the native part of the assembly in this tutorial in its default selection state. For the purposes of illustration the various faces have been labeled as given below.



The following illustrations explain how to select/deselect faces to be unfolded.

Illustration 1:

i. On the native part, click on ***Face d*** as shown in the image on the right.

ii. Observe that only *Face d* is deselected. The only face adjacent to ***Face d*** is ***Face c***. However, ***Face c*** is not deselected and continues to be highlighted in green indicating its selection for unfolding. This is

because it continues to be connected to the Reference face via ***Face a*** and ***Face b***.

iii. Click on ***Face d*** again in order to select it.

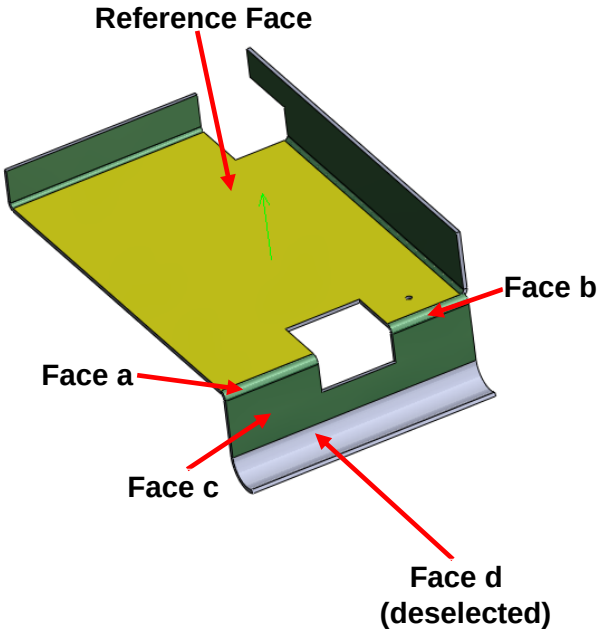


Illustration 2:

i. On the native part, click on ***Face c*** in order to deselect it.

ii. Observe that in addition to ***Face c***, the ***Face d*** (which is adjacent to ***Face c***) is also deselected. This is because the deselection of ***Face c*** causes ***Face d*** to become completely disconnected from the reference face.

iii. Click on ***Face c*** again in order to select it. The ***Face d*** will also be automatically selected.

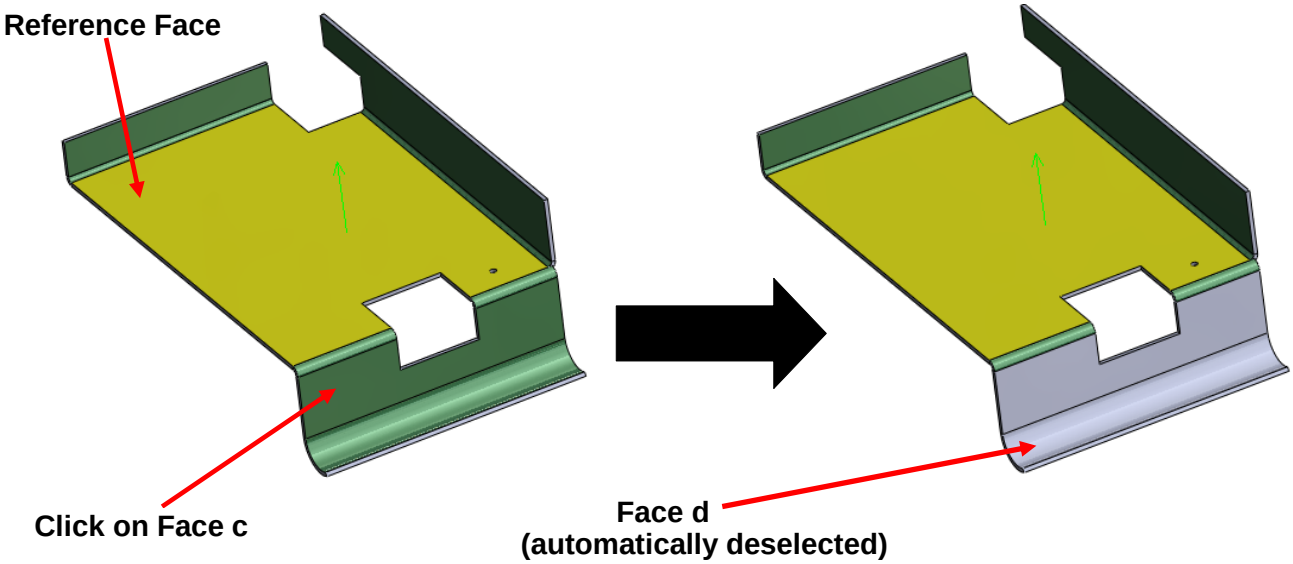


Illustration 3:

i. On the native part, click on ***Face a*** (the bent face on the left) in order to deselect it.

   Observe that ***Face c***, (which is adjacent to ***Face a***) is not deselected. This is because ***Face c*** continues to remain connected to the *Reference face* via ***Face b***.

ii. Now click on ***Face b*** in order to deselect it.

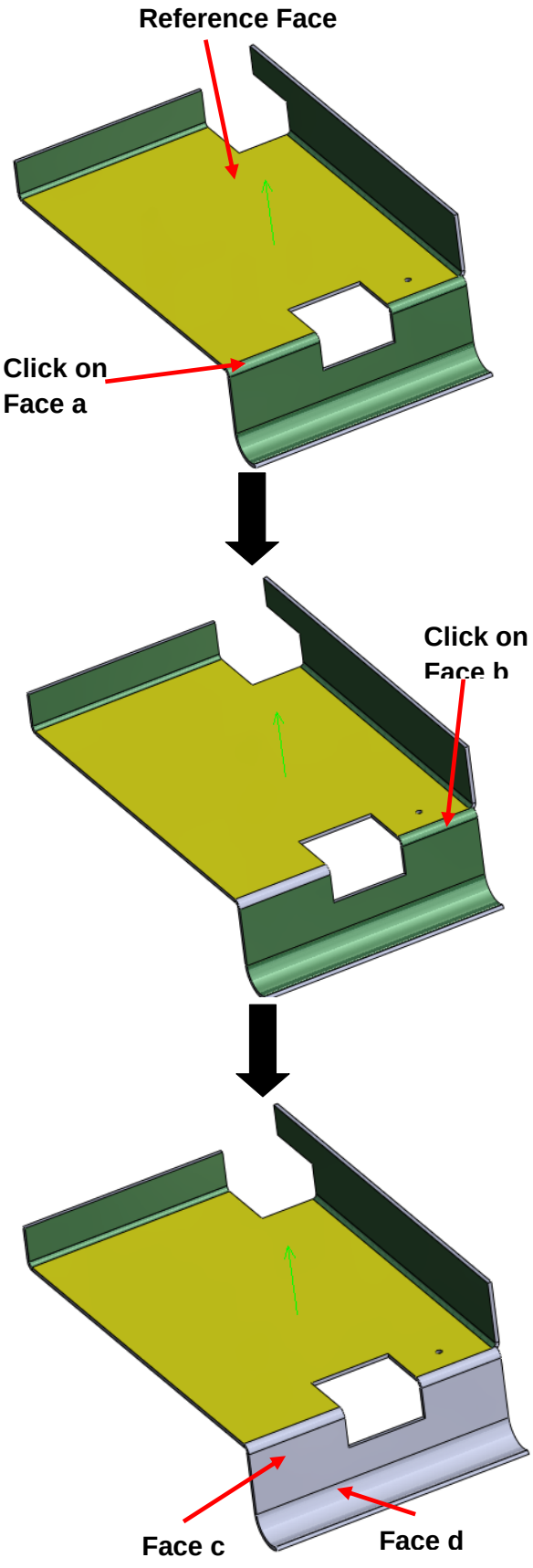
   Observe that, along with ***Face b***, ***Face c*** and ***Face d*** are also deselected.

   Face c is adjacent to both ***Face a*** and ***Face b*** and connected to the reference face via these two faces. When both ***Face a*** and ***Face b*** are deselected, ***Face c*** is no longer connected to the *Reference face* and hence is automatically deselected.

   Face d is adjacent to ***Face c***. When ***Face c*** is deselected, then ***Face d*** will no longer be connected to the reference face and is hence deselected.

iii. Now click on either ***Face a*** or ***Face b*** in order to select it once again.

   Observe that if you select ***Face a***, then ***Face b***, ***Face c*** and ***Face d*** are also automatically selected. Similarly, if you selected ***Face b***, then ***Face a***, ***Face c*** and ***Face d*** are also selected automatically for unfolding. This is because these adjacent faces are then once again connected to the *Reference face*.



Changing the Reference Face

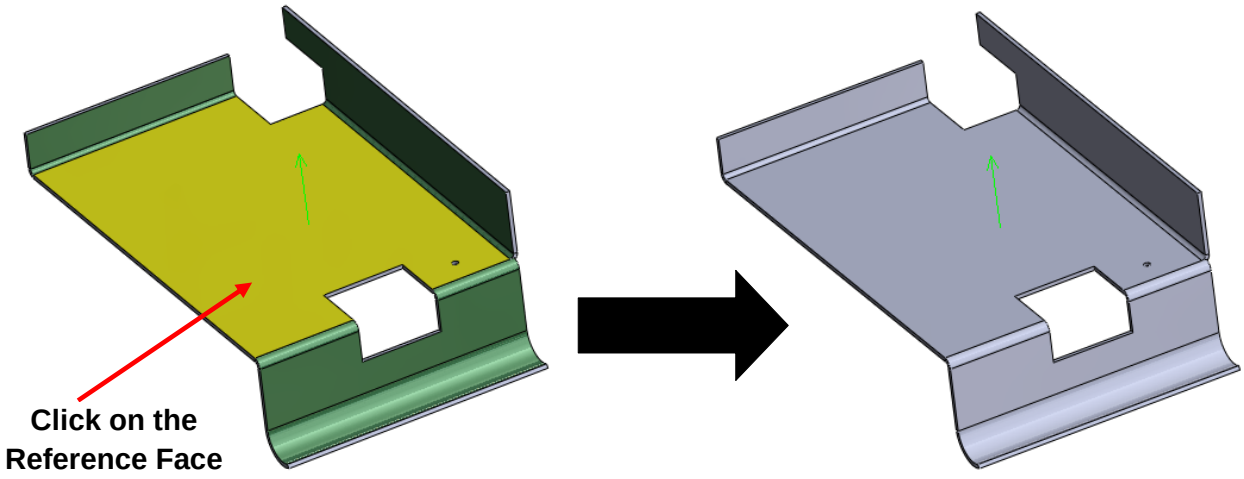
By default, the reference face is the face with the largest surface area (highlighted in yellow color when the *Interactive Unfold* dialog box is open). All faces tangential to the reference face are selected for unfolding. You can change the face selected as the reference face.

Deselecting the Reference Face

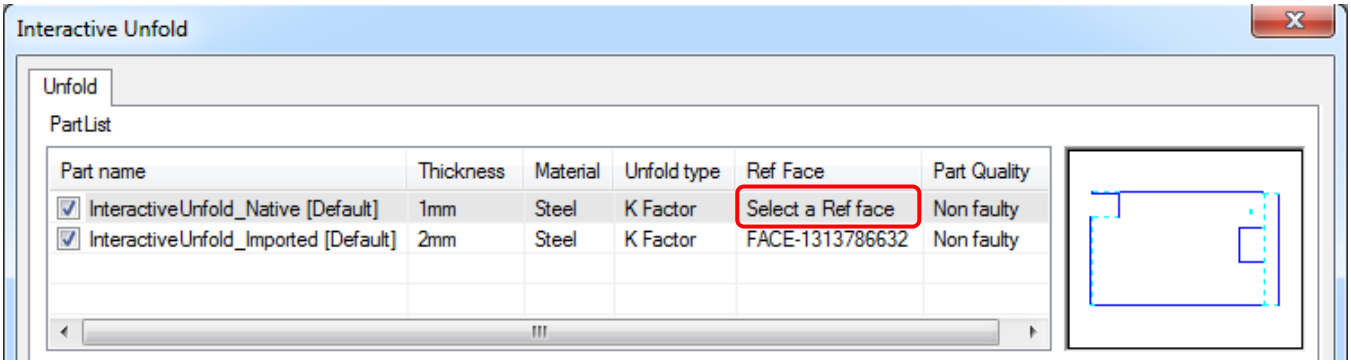
To deselect the reference face of a part, you need to click on the Reference face of the part in the graphics area.

1. In the graphics area, click on the *Reference face* (highlighted in yellow color) of the native part.

   Observe that the *Reference face* is deselected and all the faces selected for unfolding are also deselected.



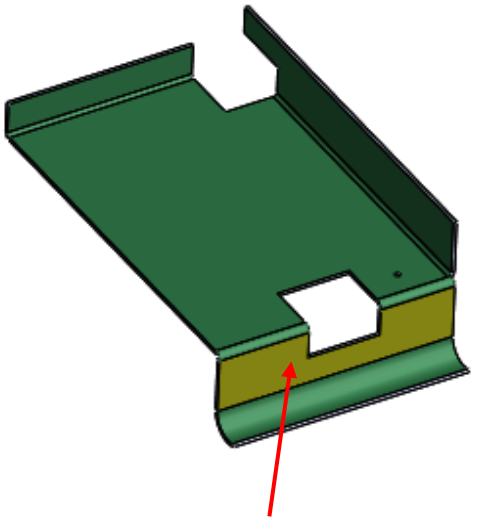
2. After deselecting the Reference face, observe the *Interactive Unfold* dialog box. Since the native part no longer has a reference face; the Reference Face field for the native part in this dialog box prompts you to select a reference face on the part by displaying the message "*Select a Ref Face*".



Message in Ref Face column prompting user to select a reference face

Selecting a Reference Face

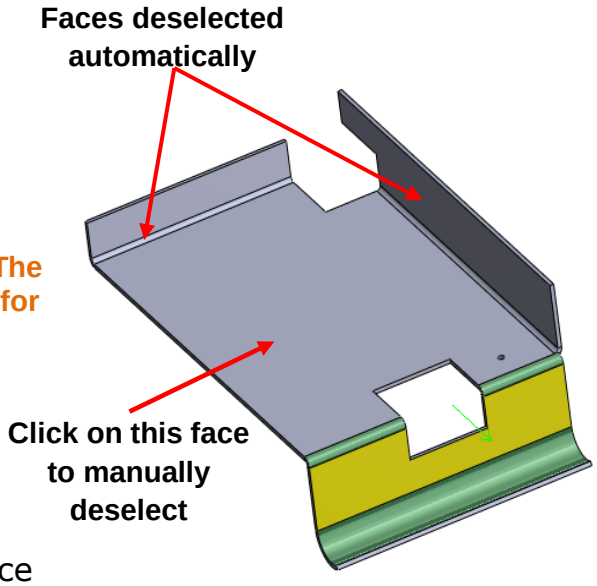
When no face is selected as the reference face for unfolding on a part, the 'Interactive Unfold' dialog box prompts you to select a face as the Reference face for the particular part. In order to select a face as the reference face, click on the desired face in the graphics area.



1. In the previous step, the reference face of the native part was deselected. Now, in the graphics area, pick the planar surface on the part as shown in the image on the right. This planar surface will become the reference face (highlighted in yellow color) for the native part. All the faces tangential to this new reference face will be highlighted in green thus indicating their selection for unfolding.

   Not that auto-selection of tangential faces for unfolding is enabled only when the 'Chain Faces' option is selected in the *Interactive Unfold* dialog box.

As long as the 'Chain Faces' option is checked, the principle of selecting/deselecting the highlighted green faces will apply i.e. If you deselect any highlighted green face, then all the highlighted green faces are adjacent to the deselected face and now disconnected from the reference face will be also be deselected. The vice versa principle applies to selecting faces for unfolding.



   For example, click on the highlighted green face with the largest surface area in order to deselect it. Observe that faces tangential to it which are now disconnected from the reference face due to the deselection are also deselected.

2. Once the faces to unfold are selected, you need to click on the *OK* button on the *Interactive Unfold* dialog box in order to unfold the part.

   The flattened part will be displayed automatically in the graphics area.

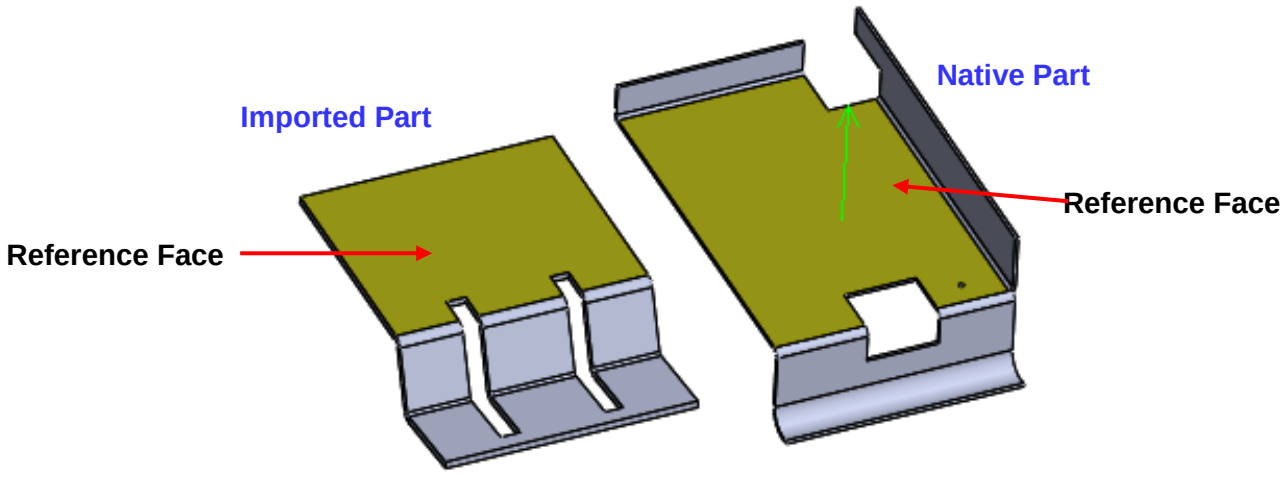
STEP 4: Selective Unfolding of Parts when 'Chain Faces' option is disabled

In this section of the tutorial, we will explore how to select references faces and faces to unfold when the Chain Features option is disabled.

1. If you have followed the instructions given in Step 3 of this tutorial, then close the assembly without saving any changes.
2. Reopen the assembly (Step 1) and execute the Interactive Unfold command (Step 2).

Disabling the Chain Faces option

3. In the graphics area, observe that faces are selected for unfolding (highlighted in **green** color) because the *Chain Faces* option is enabled.
4. In the displayed *Interactive Unfold* dialog box, uncheck the *Chain Faces* option.
5. After unchecking the option, close the *Interactive Unfold* dialog box by clicking on the *Cancel* button in the dialog box. This action is necessary for the in order to let the effect of the disabled *Chain Features* option to take place.
6. Open the *Interactive Unfold* dialog box again by executing the *Interactive Unfold* command on the CAMWorksNesting menu or CAMWorks Nesting Ribbon bar.
7. In the graphics area, observe that only reference face (in **yellow** color) will be selected for both the native and imported parts. The face with the largest surface area is selected as the reference face by default. You need to manually select the faces for unfolding.



Only Reference Faces are highlighted in Yellow color

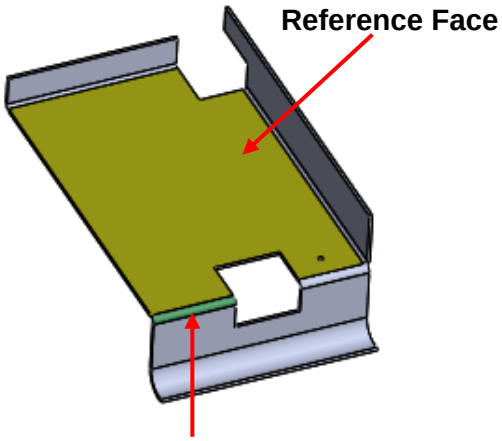
Selecting faces to unfold when Chain Faces option is disabled

To select a face for unfolding, the face should be either adjacent to the reference face or connected to the reference face via an already selected face. A face selected for unfolding is highlighted in green color in the graphics area.

Example:

i. In the graphics area, on the native part, click on the bent face of the native part as shown in the image on the right. This face then becomes highlighted in green.

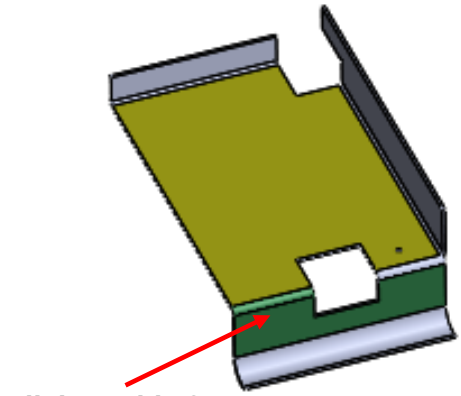
Observe that since this face is adjacent to the reference, it is selected for unfolding.



ii. Now click on the face which is tangential and adjacent to the newly selected face as shown in the image on the right.

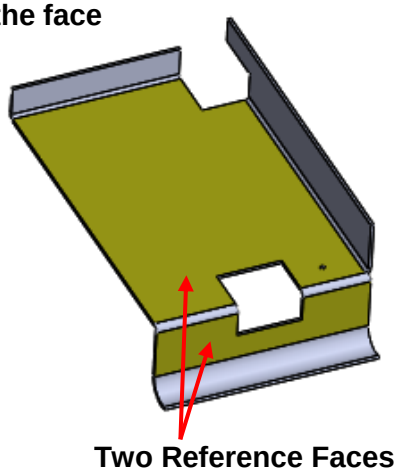
Observe that this face too is selected for unfolding though it is not adjacent to the reference face. This face is selected as a face to be unfolded because it is connected to the reference face via the previously selected face.

You can thus select all the desired faces for unfolding one by one by clicking on faces adjacent to the reference face or an already selected face.



iii. Now deselect the bent face selected for unfolding by clicking on that face. Observe that the face tangential to it (which was also selected for unfolding) has now become a reference face. This is indicated by the change in color from green to yellow.

The face changes to a reference face because the face connecting it to the original reference face is no longer selected.



Changing Reference face when Chain Faces option is disabled

To select a face as reference face when the *Chain Faces* option is disabled:

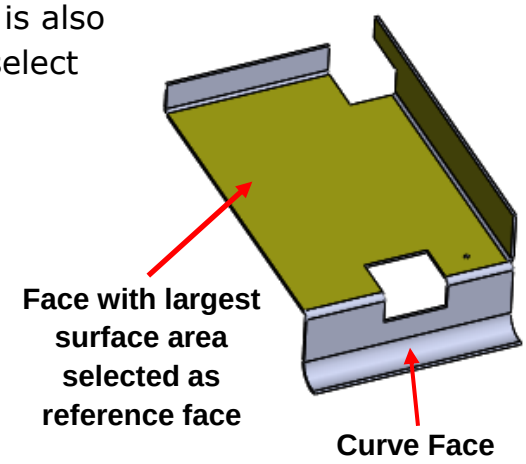
- The face should be planar in nature. Curved faces cannot be selected as a reference faces.
- If no face is selected as a reference face on the part, then any planar face can be selected as the Reference face.
- If one or more faces are already selected as reference faces, then any planar face which is neither adjacent to any of the reference faces nor connected to them via a selected face can be selected as a reference face.

A face selected as reference face will be highlighted in **yellow color** in the graphics area.

Illustration:

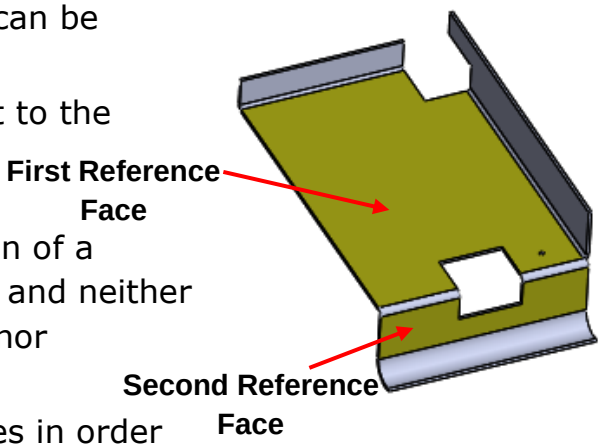
i. In the graphics area, deselect any faces that you may have already selected for unfolding by clicking on the face. If any face other than the face with the largest surface area is also selected as a reference face, then deselect that face by clicking on it.

ii. When only the face with the largest surface area remains selected as the reference face, then click on the curved face. This curved face is neither adjacent to the reference face nor connected to it via a selected face.



Observe that you cannot select the curved face as a reference face. This is because only planar surfaces can be selected as reference faces.

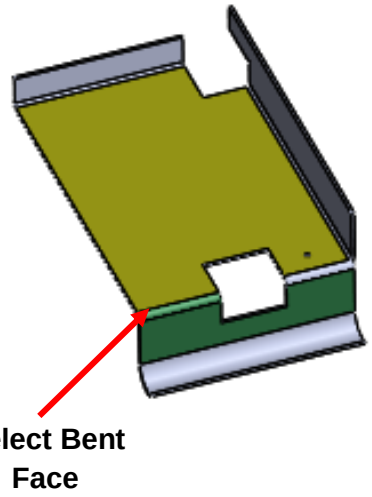
iii. Now click on the planar face adjacent to the curved face. This face gets selected as a reference face since it fulfills all the conditions for selection of a reference face. It is planar in nature, and neither adjacent to the other reference face nor connected to it via a selected face.



iv. Now click on any one of the bent faces in order to connect the first and second reference face.

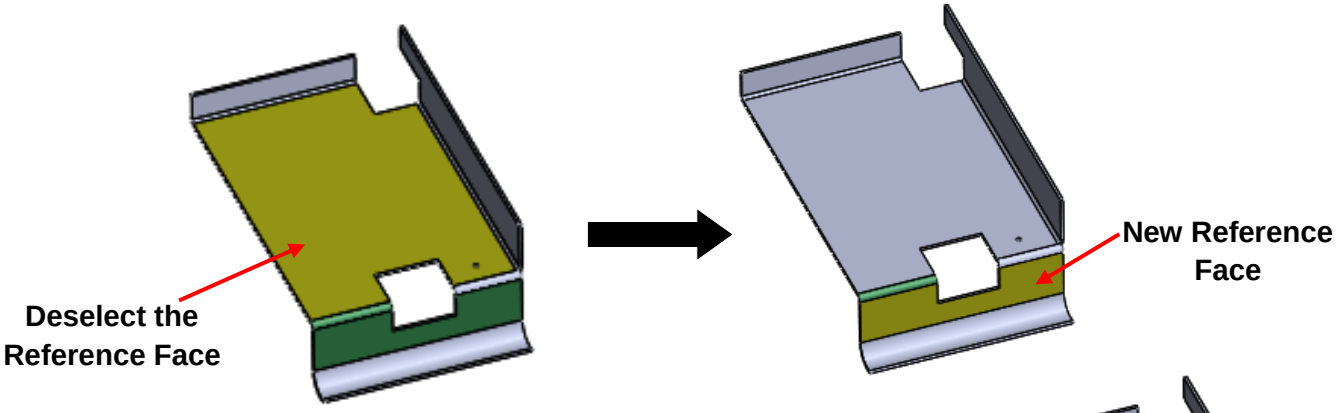
Observe that second reference face that was selected in the previous step now becomes a selected face (indicated by the change in color of

the face from yellow to green). This is because the moment the bent face is selected, a face chain is formed and the second reference becomes connected to the original reference face via the bent face.



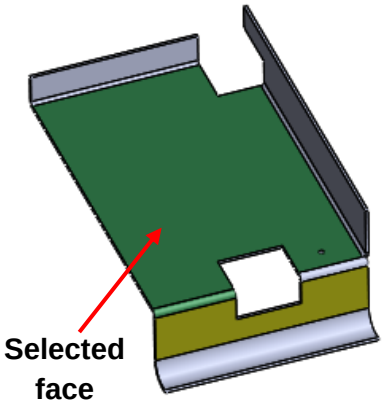
v. Now click on the original reference face (the face with the largest surface area) in order to deselect it. Observe that the planar face highlighted in green now once again becomes the reference face.

This is because whenever you deselect a reference connected to other face chains, then the tangentially connected planar face will be selected as a new reference face (selection indicated by change in color of the face from green to yellow).



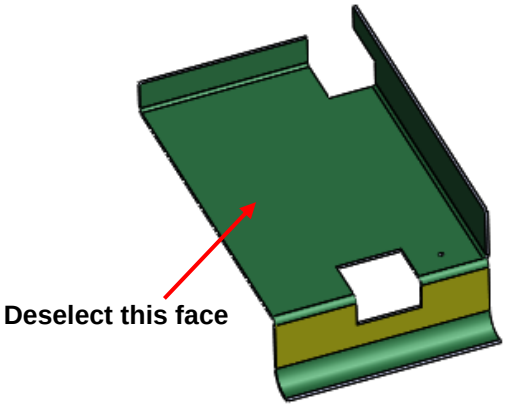
Now click on the deselected reference face (face with largest surface area) once again in order to select it.

Observe that this now selected as a selected face for unfold since it is connected to the current reference face via another selected face.



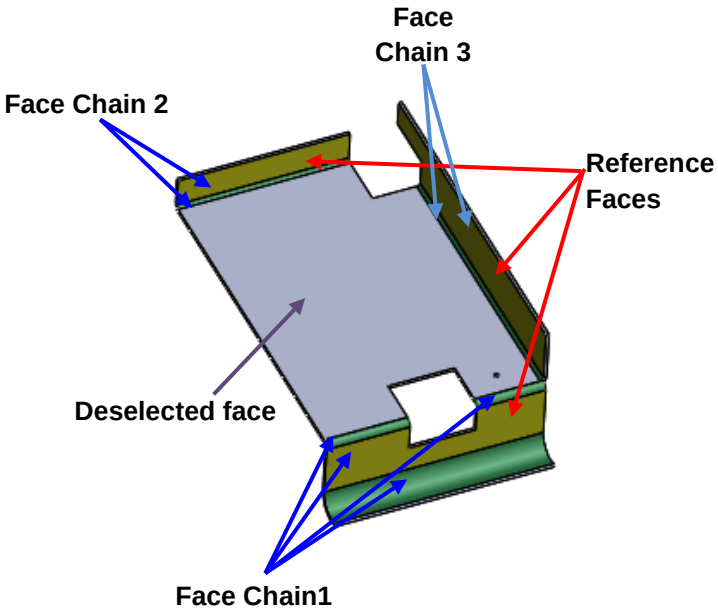
vi. Now click on the all the remaining faces in order to select them for unfolding. As you select each face, they will be highlighted in green color thus indicating their selection.

vii. Now once again click on the central planar surface (face with the largest area) in order to deselect it.



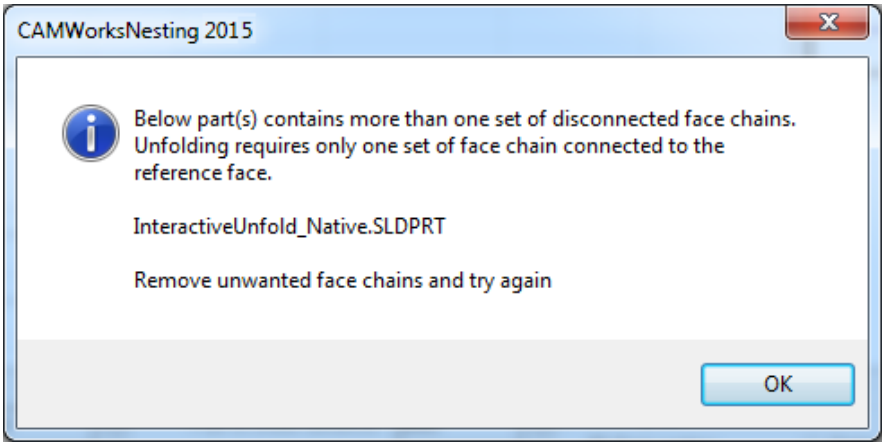
Observe that four faces (two bent faces and two planar faces) are now no longer connected to the reference face due to the deselection of this face. These disconnected faces will now form separate face chains. In each face chain, the planar surface will become the reference face.

As seen in the image on the right, three face chains will be formed with each containing a reference face.



Note: **In the newly formed face chain, if there is no planar surface available then such chains will be rejected from the unfolding process automatically.**

viii. Click the *OK* button of *Interactive Unfold* dialog box to unfold the parts. Before unfolding, CAMWorks Nesting will check the parts to be unfolded for the number of disconnected face chains. If a part selected for unfolding retains more than one disconnected face chain, then an error message will be displayed prompting you to remove unwanted face chains. The parts won't be unfolded and the *Interactive Unfold* dialog box remains open.

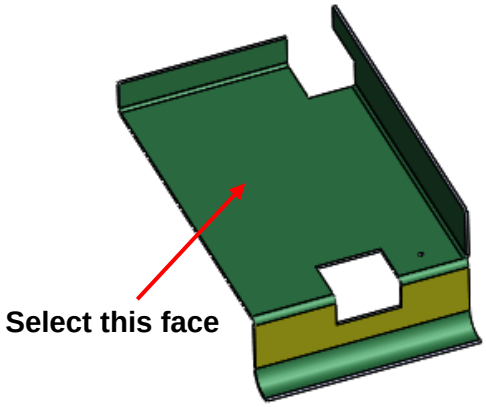


Error message dialog box

To proceed with nesting, you need to ensure that only one face chain is selected on the each part to be unfolded.

Note: **On non-faulty parts, the Reference face and the selected faces form a single face chain. CAMWorks Nesting can only unfold a single face chain when an Unfold command is executed. Multiple face chains cannot be unfolded.**

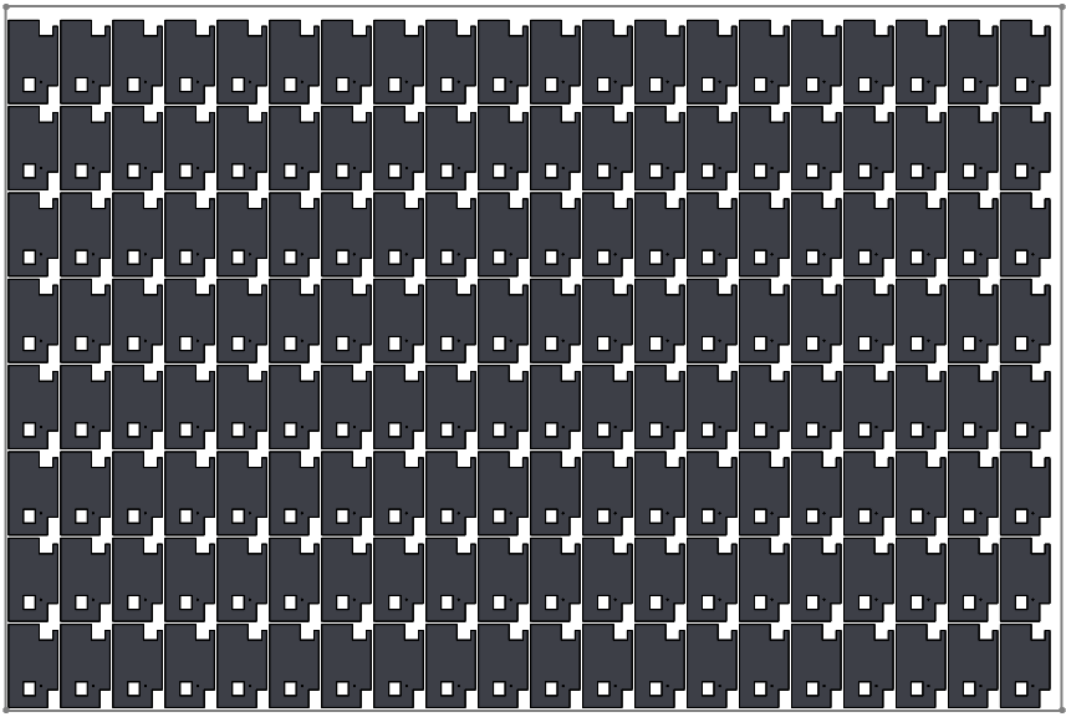
ix. In the graphics area, click on the central face (face with the largest surface area) in order to select it. Selecting this face for unfolding connects all the face chains.



x. Click the *OK* button of *Interactive Unfold* dialog box to unfold the parts.

Observe that unfolding is successful for both the imported part as well as the native part.

xi. Once unfolded, use the *Create Nesting Job* command to nest these parts.



Sample Nesting Layout for native part after unfolding

TUTORIAL 11 – THE STAMP FEATURE UNFOLD OPTION

Introduction

If a sheet metal part to be nested contains stamp features, an option is provided within CAMWorks Nesting to control the display of these stamp features after the part is unfolded using one of the unfold commands. The setting to control the behavior of the stamp features display can be assigned only from the *DefaultValues.ini* file. It cannot be set from the '*Create Nesting Job*' dialog box.

Topic covered in this Tutorial:

In this tutorial, you will explore how the settings in the *DefaultValues.ini* file affect the behaviour of the stamp features present on sheet metal part when the sheet metal part is unfolded.

Assigning Stamp Feature Unfold Option settings in DefaultValues.ini

There are three available settings to control the behavior of the stamp features before nesting the sheet metal part. These settings are controlled from the *DefaultValues.ini* file.

1. Open the file named *DefaultValues.ini* located in the CAMWorks Nesting Installation folder. (*Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Lang\English*)
2. In the [Unfold_Options] section, observe the flag named '*StampFeatureUnfoldingOption*'. This is the flag used to control the behavior of stamp features after the part is unfolded. Following are the settings:
   i. **0**: Assigning the value '0' ensures that the stamp feature is **retained** after the part is unfolded. (This is the default setting at the time of installation.)
   ii. **1**: When the value '1' is assigned to this flag, the stamp feature is **patched** with a flat planar surface after the part is unfolded.
   iii. **2**: When the value '2' is assigned to this flag, the stamp feature is **ignored** after the part is unfolded. The area covered by the stamp feature is replaced with a hole.
3. Once you make any changes to the settings in the *Defaultvalues.ini* file, save the changes and close the file.

Stamp Feature Unfold Option settings for Native parts & Imported Parts

After a sheet metal part with stamp feature(s) is unfolded, the resultant display of the stamp feature based on settings in the *DefaultValues.ini* file depends on whether the unfolded part is a native sheet metal part or imported sheet metal part. In the case of native parts, it also depends on the type of command used to unfold the part.

Given below is table indicating the relation between the various unfold commands and the applicability of the *Stamp Feature Unfold Option* for native parts and imported parts.

<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Type of Part</th></tr>
<tr><th>Native Part</th><th>Imported Part</th></tr>
<tr><td rowspan="4">Type of Unfold Command</td><td>‘Intelligent Unfold’ command</td><td>This command is not applicable to native parts.</td><td rowspan="4">Stamp Feature Unfold Option settings in DefaultValues.ini file applied when part is unfolded using this command.</td></tr>
<tr><td>‘Unfold All Parts’ command</td><td rowspan="2">Stamp Feature Unfold Option settings in DefaultValues.ini file applied when part is unfolded using this command.</td></tr>
<tr><td>‘Interactive Unfold’ command</td></tr>
<tr><td>‘Create Nest Job’ command</td><td>If this command is executed directly without using any other unfold command, then the stamp features will be always retained irrespective of the settings in the DefaultValues.ini file.</td></tr>
</table>

This tutorial is divided into two sections:

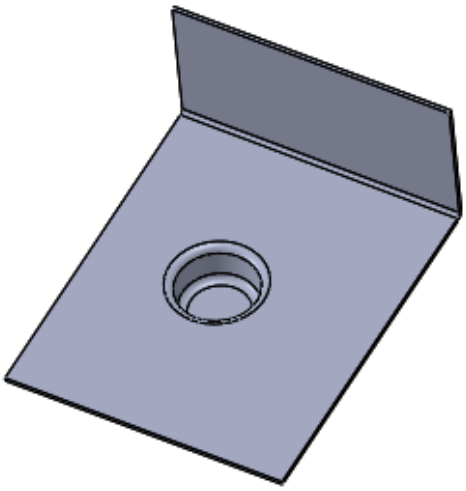
Part 1: Stamp Feature Unfold Options for Native Sheet Metal Parts

Part 2: Stamp Feature Unfold Options for Imported Sheet Metal Parts

Part 1: Stamp Feature Unfold Options for Native Sheet Metal Parts

Step 1: Open the Part

1. Launch CAMWorks Nesting as an Add-In in the SOLIDWORKS or CAMWorks Solids environment.
2. Open the part file ***Tutorial_11a_native.sldprt*** located in the following folder: *Drive:\CAMWorksNestingData\CAMWorksNesting201x\Examples\Tutorials\Parts*
3. Observe that this is a native part with a stamp feature.



Tutorial_11a_native.sldprt

Step 2: Executing the Unfold Command

For the settings of the Stamp Feature Unfold Option in *DefaultValues.ini* to take effect, you need to first unfold the part. To unfold the part you can use the unfold commands available on the CAMWorks Nesting Ribbon bar as well as the CAMWorksNesting menu. Following are the unfold commands you can use to unfold a native part:

1. **The 'Unfold All Parts' command** 

   When you execute this command, the *Unfold All Parts* dialog box will be displayed. Click the *OK* button to unfold the part.

2. **The 'Interactive Unfold' command** 

   When you execute this command, the *Interactive Unfold* dialog box will be displayed. Click the *OK* button to unfold the part.

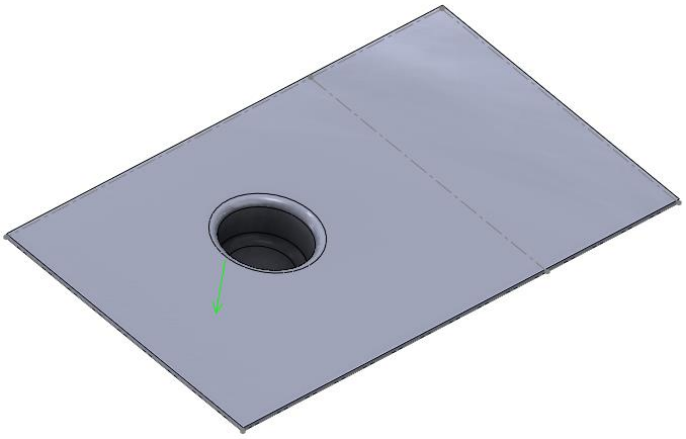
3. **The 'Create Nest Job' command** 

   When you execute this command, the native part will be auto-unfolded before the *Create Nesting Job* dialog box is displayed. (The native sheet metal part will be unfolded only if the flag named *FlattenSheetMetalPart* in the *DefaultValues.ini* file is set to '1'. This is the default setting)

Note: **The 'Intelligent Unfold' command cannot be used to unfold the native parts. This command is applicable only for imported parts.**

Step 3: Retaining the stamp feature

1. Open the *DefaultValues.ini* file and set the *StampFeatureUnfoldingOption* flag to '**0**'. This setting ensures that the stamp feature is retained after the part unfolding process.
2. When you unfold the native part using any one of the commands mentioned in Step 2, the stamp feature will be retained.



Result of the Retained Stamp Option

Step 4: Patching the stamp feature

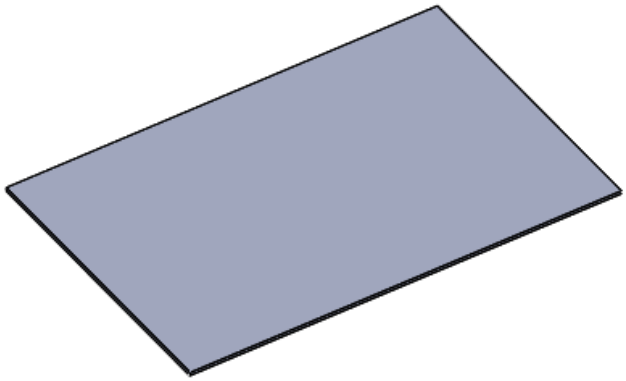
You will now set the stamp feature unfolding option to patch the stamp feature after unfolding.

1. Open the *DefaultValues.ini* file and set the *StampFeatureUnfoldingOption* flag to '**1**'. This setting ensures that the stamp feature is patched after the part unfolding process.
2. Save the changes and close the *DefaultValues.ini* file.
3. Unfold the part using either the '*Unfold All Parts*' or '*Interactive Unfold*' command.

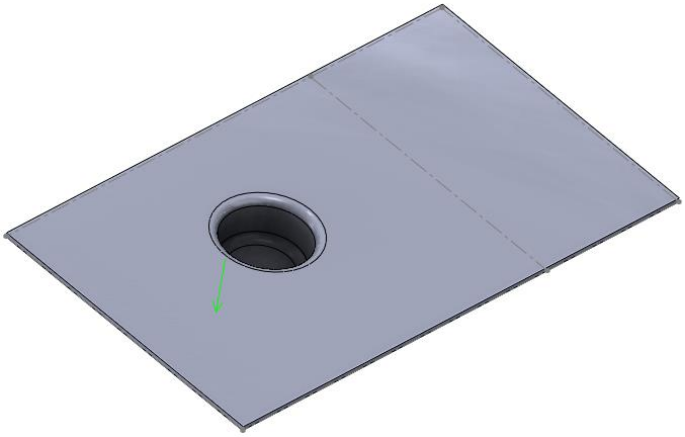
   Observe that the stamp feature is patched (replaced) with a planar surface after the part is unfolded.
4. Now close the part (without saving the changes) and reopen the part in SOLIDWORKS/CAMWorks Solids. Directly execute the 'Create Nest Job' command.

   Observe that the stamp feature is retained instead of being patched.

   This indicates that the setting for the Stamp Feature Unfold Option in the *DefaultValues.ini* was not applied to the part.



Result of the Patch Stamp Feature option when the 'Unfold All Parts' or 'Interactive Unfold' command is executed



Result of the Patch Stamp Feature option when the 'Create Nest Job' command is executed

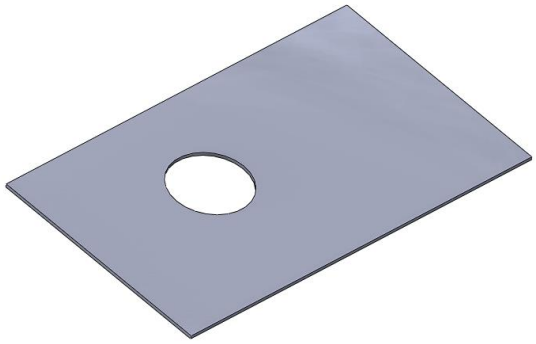
Note: **If you unfold a native part directly using the 'Create Nest Job' command, then the Stamp features present on the part will always be retained on the part after it is unfolded irrespective of the settings for 'Stamp Feature Unfold Option' in the DefaultValues.ini file. This behavior results because CAMWorks Nesting uses SOLIDWORKS functionality to flatten the native sheet metal parts instead of CAMWorks Nesting functionalities.**

Step 5: Ignoring the stamp feature

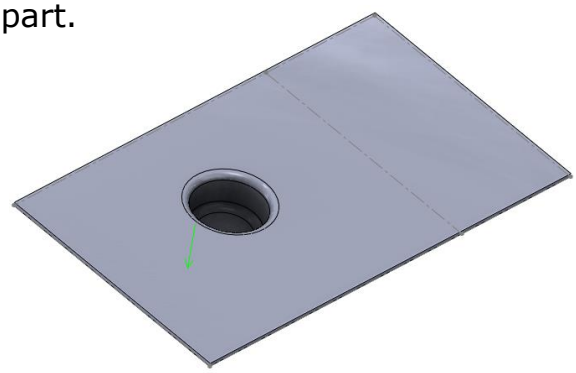
You will now set the stamp feature unfolding option to ignore the stamp feature after unfolding process.

1. Open the *DefaultValues.ini* file and set the *StampFeatureUnfoldingOption* flag to '**2**'. This setting ensures that the stamp feature is ignored after the unfolding the part.
2. Save the changes and close the *DefaultValues.ini* file.
3. Unfold the part using either the '*Unfold All Parts*' or '*Interactive Unfold*' command. Observe that the stamp feature is ignored. The area covered by the stamp feature is replaced with a hole.
4. Now close the part (without saving the changes) and reopen the part in SOLIDWORKS/CAMWorks Solids. Directly execute the '*Create Nest Job*' command. Observe that the stamp feature is retained instead of being ignored.

   This indicates that the setting for the *Stamp Feature Unfold Option* in the *DefaultValues.ini* was not applied to the part.



Result of the Ignore Stamp Feature option when the 'Unfold All Parts' or 'Interactive Unfold' command is executed



Result of the Ignore Stamp Feature option when the 'Create Nest Job' command is executed

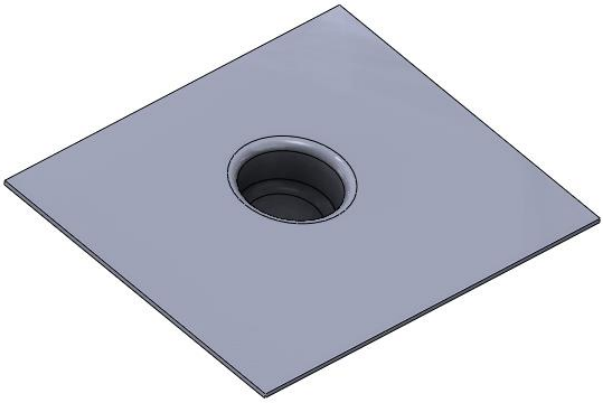
Once the part is unfolded, proceed to nest the part using the *Create Nesting Job* dialog box.

Step 6: Behaviour in native parts without bends

1. Open the part file ***Tutorial_11c_native.sldprt*** located in the following folder:

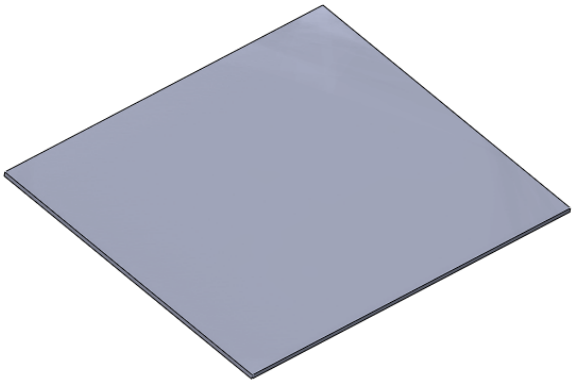
   Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Parts

   Observe that this is an imported part without bends. The part has a stamp feature.

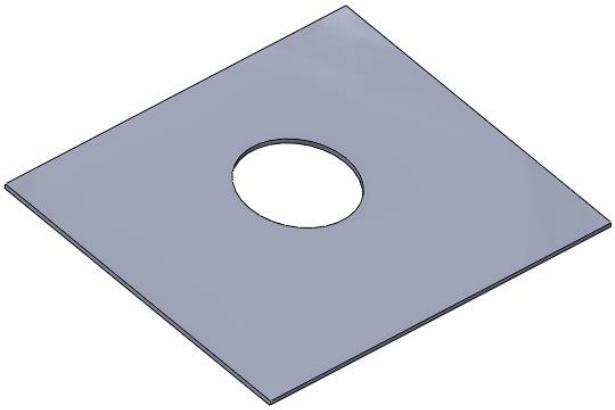


Tutorial_11c_native.sldprt

2. Execute Step 3, Step 4 and Step 5 and once again observe that the behaviour of the stamp feature changes as per the settings in the *DefaultValues.ini* file only when the '*Unfold All Parts*' or the '*Interactive Unfold*' command is executed. If the part is unfolded directly using the '*Create Nest job*' command, then the setting for Stamp Feature Unfold Option will not take effect and the stamp feature will be retained.



Result of the Patch Stamp Feature option when the 'Unfold All Parts' or 'Interactive Unfold' command is executed



Result of the Ignore Stamp Feature option when the 'Unfold All Parts' or 'Interactive Unfold' command is executed

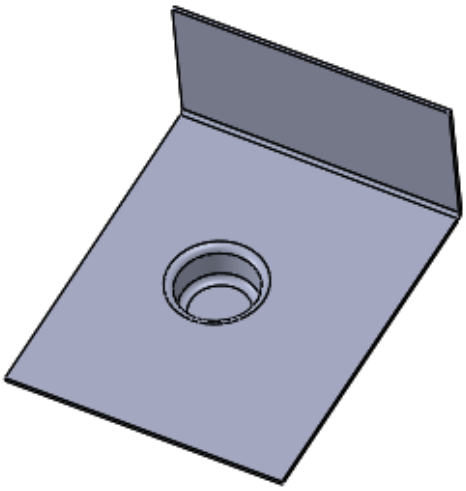
Note: **The settings for the Stamp Feature Unfold Option in the DefaultValues.ini file is applicable for a native part only if you first unfold the part using either the 'Unfold All Parts' or the 'Interactive Unfold' command.**

For a native part (with or without bends), the stamp feature is always retained on the part if you directly execute the 'Create Nest Job' command without first executing the 'Interactive Unfold' or 'Unfold All Parts' command.

Part 2: Stamp Feature Unfold Option for Imported Sheet Metal Parts

Step 1: Open the Part

1. Launch CAMWorks Nesting as an Add-In in the SOLIDWORKS or CAMWorks Solids environment.
2. Open the part file ***Tutorial_11b_imported.sldprt*** located in the following folder: *Drive:\CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Parts*
3. Observe that this is an imported part with bends. The part has a stamp feature.



Tutorial_11b_imported.sldprt

Step 2: Executing the Unfold Command

For the settings of the Stamp Feature Unfold Option in *DefaultValues.ini* to take effect, you need to first unfold the part. To unfold an imported part, you can use any one of the following commands available on the CAMWorks Nesting Ribbon bar as well as the CAMWorksNesting menu.

1. **The 'Intelligent Unfold' command** 

   When you execute this command, the *Unfold Imported Bodies* dialog box is displayed. Click the *OK* button to unfold the part.

2. **The 'Unfold All Parts' command** 

   When you execute this command, the *Unfold All Parts* dialog box is displayed. Click the *OK* button to unfold the part.

3. **The 'Interactive Unfold' command** 

   When you execute this command, the *Interactive Unfold* dialog box is displayed. Click the *OK* button to unfold the part.

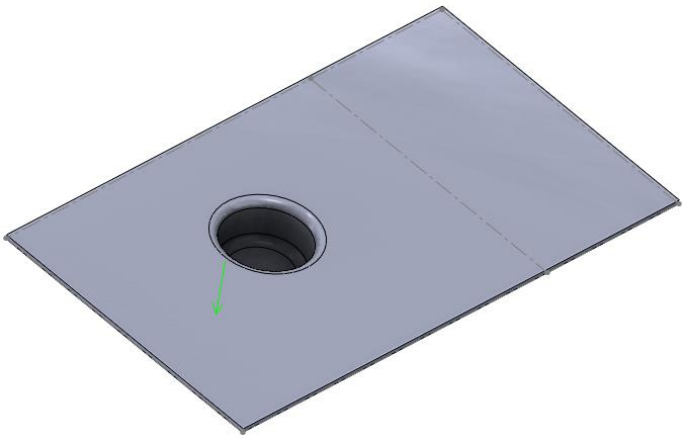
4. **The 'Create Nest Job' command** 

   When you directly execute this command for imported parts/assembly, CAMWorks Nesting will display a message indicating that the part/assembly contains imported parts and whether you wish to unfold the parts before proceeding with the nesting process. Click *Yes*. The *Unfold All Parts* dialog box will be displayed. Click *OK* button in this dialog box to unfold the parts. If you click the *Cancel* button, then the parts will neither be unfolded nor will the settings for the Stamp Feature Unfold Options be applied to the parts.

Step 3: Retaining the stamp feature

You will now set the stamp feature unfolding option to retain the stamp feature after unfolding the part.

1. Open the *DefaultValues.ini* file and set the *StampFeatureUnfoldingOption* flag to '**0**'. This setting ensures that the stamp feature is retained after the part unfolding process.
2. Save the changes and close the *DefaultValues.ini* file.
3. Unfold the part using any one of the commands as listed in Step 2. Observe that the stamp feature is retained after the part is unfolded.



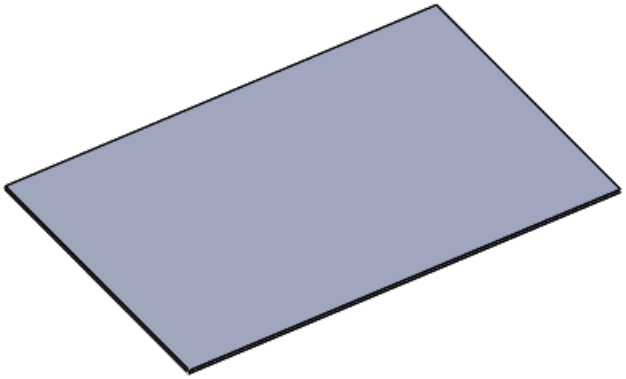
Result of the Retained Stamp Option

Step 4: Patching the stamp feature

You will now set the Stamp Feature Unfold Option to patch the stamp feature after unfolding the part.

1. Open the *DefaultValues.ini* file and set the *StampFeatureUnfoldingOption* flag to '**1**'. This setting ensures that the stamp feature is patched after the part is unfolded.
2. Save the changes and close the *DefaultValues.ini* file.
3. Unfold the part using any one of the commands as listed in Step 2.

   Observe that the stamp feature is patched (replaced) with a planar surface after the part is unfolded.



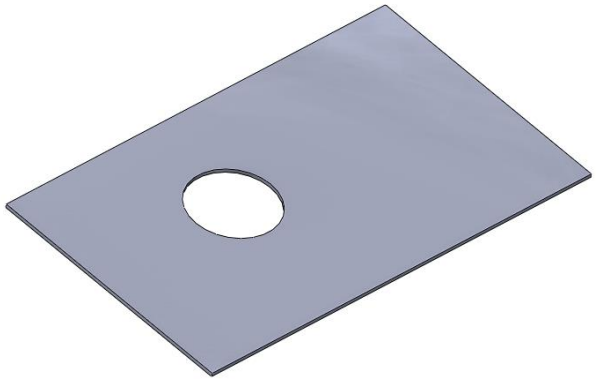
Result of the Patch Stamp Feature option after the part is unfolded

Step 5: Ignoring the stamp feature

You will now set the Stamp Feature Unfold Option to ignore the stamp feature after unfolding.

1. Open the *DefaultValues.ini* file and set the *StampFeatureUnfoldingOption* flag to '**2**'. This setting ensures that the stamp feature is ignored after the unfolding the part.

2. Save the changes and close the *DefaultValues.ini* file.
3. Unfold the part using any one of the commands listed in Step 2. Observe that the stamp feature is ignored. The area covered by the stamp feature is replaced with a hole.



Result of the Ignore Stamp Feature option after the part is unfolded

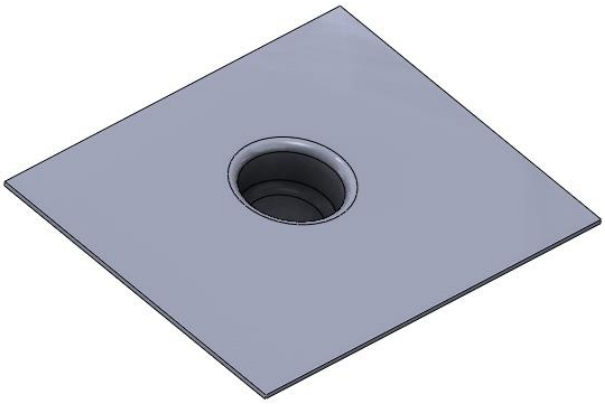
Note: **If you unfold an imported part containing stamp features before nesting the part, then the stamp feature will be retained, patched or ignored based on 'StampFeatureUnfoldOption' flag settings in the DefaultValues.ini file.**

Once the part is unfolded, proceed to nest the part using the *Create Nesting Job* dialog box.

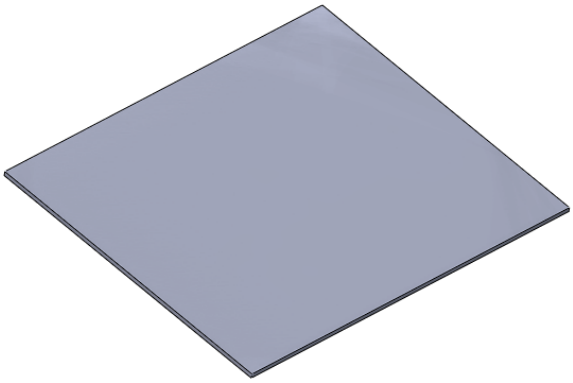
Step 6: Behaviour in imported parts without bends

1. Open the part file ***Tutorial_11d_imported.sldprt*** located in the following folder: *Drive:\CAMWorksNestingData\CAMWorks Nesting 201x\Examples\Tutorials\Parts*

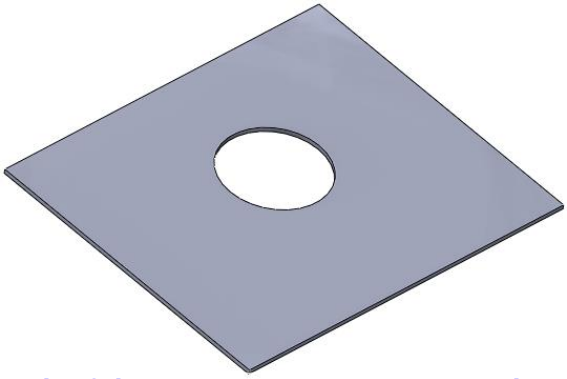
   Observe that this is an imported part without bends. The part has a stamp feature.
2. Execute Step 3, Step 4 and Step 5 and observe that the behaviour of the stamp feature changes as per the settings in the *DefaultValues.ini* file.



Tutorial_11d_imported.sldprt



Result of the Patch Stamp Feature option after the imported part is unfolded



Result of the Ignore Stamp Feature option after the part is unfolded

Note: If you want the settings for the Stamp Feature Unfold Option in the DefaultValues.ini file to be applied to an imported sheet metal part without any bends, then unfold the part using any one of the unfold commands.

TUTORIAL 12 – GENERATING NC CODES FOR NESTED LAYOUTS USING CAMWORKS (I)

How the Nested layouts generated are saved within SOLIDWORKS

Once the Nesting process using the CAMWorks Nesting application is completed, the nested layout(s) generated will always be saved as a SOLIDWORKS assembly file (***.sldasm***). Depending on various factors such as thickness and/or material part of part, number of sheets, grain direction, etc., either one or multiple Nested layouts will be generated.

- If only one nested layout is generated, then it will be saved as a SOLIDWORKS Assembly file comprising of nested parts. The sheet dimensions will be saved as a SOLIDWORKS sketch.
- If multiple nested layouts are generated, then these nested layouts will be saved as a SOLIDWORKS Assembly file comprising of assemblies. Each assembly is a nested layout comprising of nested parts. The sheet dimensions for each sheet will be saved as a SOLIDWORKS sketch.

Once the nested layout(s) are generated, each nested layout assembly (sheet layout containing nested parts) will be listed in the *SOLIDWORKS Configurations Manager*.

Relation between CAMWorks Nesting and CAMWorks

Both CAMWorks Nesting and CAMWorks, developed by ***Geometric Americas, Inc.*** are applications which are fully integrated with the CAD application of SOLIDWORKS/CAMWorks Solids. While *CAMWorks Nesting* is a Nesting application, *CAMWorks* is a highly-intelligent CAM application used for generating NC codes.

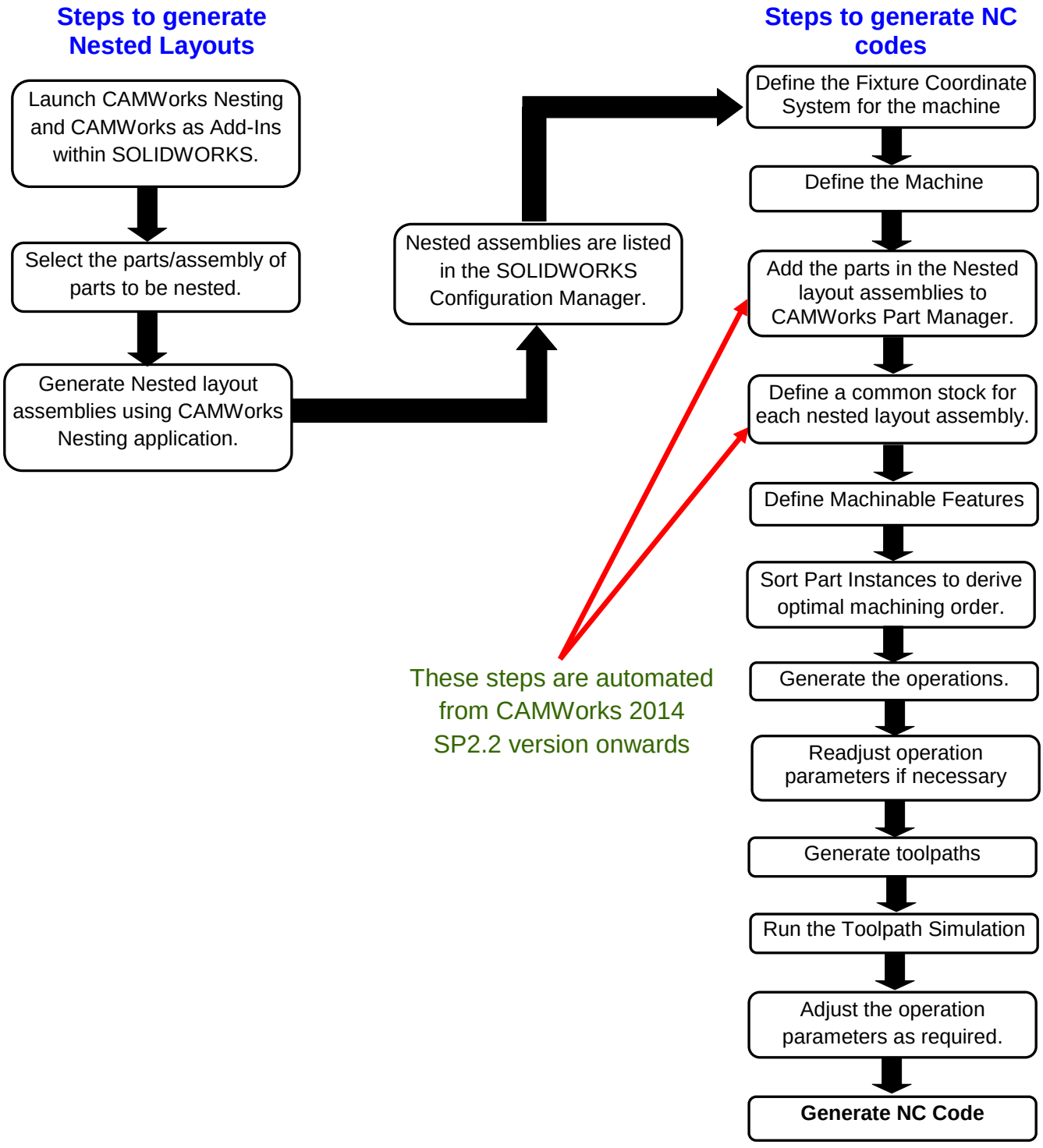
After generating the nested layout with the CAMWorks Nesting application, the next step would ideally be to generate NC codes for the nested layouts. To generate NC codes, a CAM application needs to be used. Generating NC codes using the CAMWorks application is easier than using other CAM application since:

- Both CAMWorks Nesting and CAMWorks have been developed by the same entity viz. *Geometric Americas, Inc.*
- Both these applications are fully integrated with SOLIDWORKS.
- Both these applications work with the same file types viz. file types that are compatible with SOLIDWORKS.
- A new functionality is provided from *CAMWorks Nesting 2014 SP1* version onwards that automatically links the nested layout output of CAMWorks Nesting as the input for CAMWorks, thereby reducing the number of steps required for generating NC codes. (This functionality is discussed in the next tutorial in detail).

In this tutorial and the next tutorial, you will learn how to generate NC codes for the nested layouts using the CAMWorks application.

Steps to generate NC codes for Nested layouts

To generate NC codes for nested layout assemblies, a number of steps are involved. Following are the steps for generating NC codes for Nested layouts using CAMWorks:



Flowchart illustrating how to generate NC codes for Nested Layout assemblies using CAMWorks

Generating the nested layout assembly

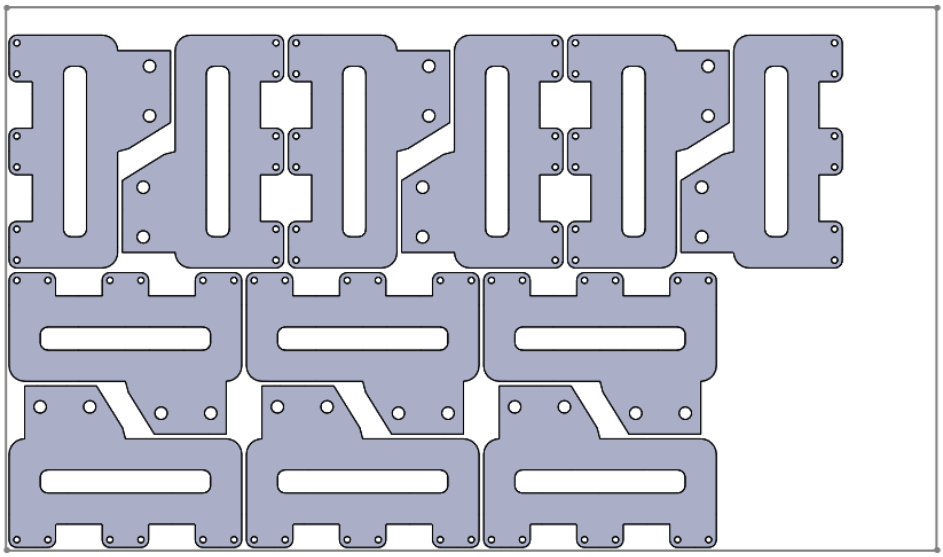
In this tutorial, you will generate NC codes for the nested layout generated in Tutorial 3.

1. Launch *CAMWorks Nesting* as an Add-In in the SOLIDWORKS environment.
2. To generate the nested layout, do any one of the following:

a. Direct open the nested layout assembly file:

Open the assembly file ***Tutorial_12_Nested_Layout.sldasm*** located in the following folder:

Drive:*CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies/Tutorial_12*



Nested Layout

b. Generate the nested layout assembly:

i. Open the part file ***Tutorial_3.sldprt*** located in the following folder:

Drive:*CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Parts*

ii. Follow the steps mentioned in the Tutorial 3 of this document to generate the nested layout. However, while executing the Tutorial 3, minor changes are required in *Step 3* and *Step 4* as follows:

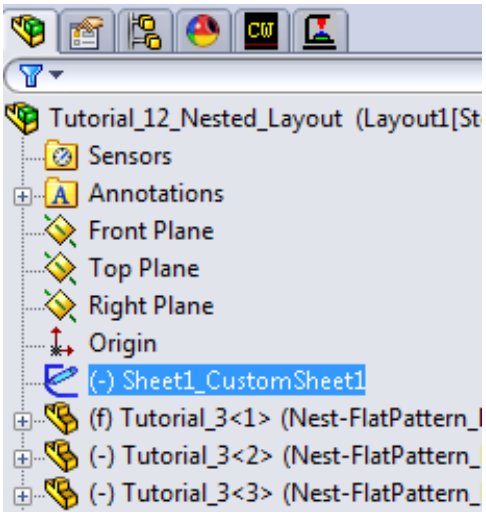
- Under the *Step 3*: Define the Part Parameters; change the assigned quantity for parts from 125 to **12**.
- Under the *Step 4*: Defining a 'Custom' size sheet, change the assigned length to **600mm** and a width to **350mm**.
- Execute all the other steps as it is mentioned in the *Tutorial 3* to generate the nested layout as shown below.

iii. The generated nested layout obtained from *Tutorial 3* will be used as input for CAMWorks.

3. In the SOLIDWORKS left hand side panel, click on the SOLIDWORKS ***FeatureManager Design Tree***.

FeatureManager Design Tree

4. Observe that a sketch (*Sheet1_CustomSheet1*) representing the dimensions of the Custom sheet (in which the parts are nested) is listed in this tree.



Sketch representing dimensions of Custom sheet

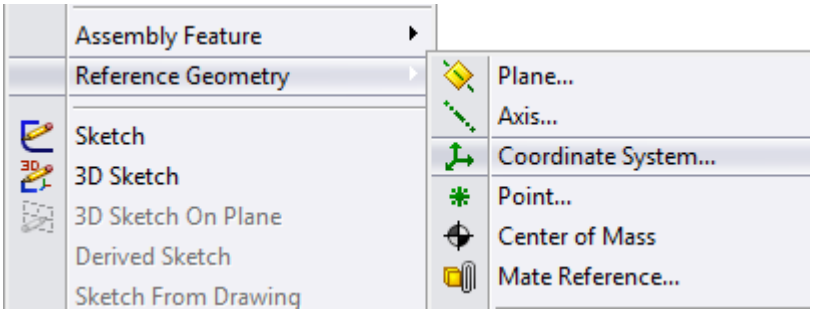
Step 1: Define the Fixture Coordinates

The Fixture Coordinate System defines the "home point" or main zero position on the machine. It defines the default G-code origin, defines the XYZ machining directions and acts as a reference point, if subroutines are used. This coordinate system needs to be defined in the SOLIDWORKS ***FeatureManager Design Tree***.

Steps to set the Fixture Coordinates System

1. If necessary, rotate and zoom the nested layout assembly in the graphics area to clearly view the position where you desire to assign the coordinate system.
2. Click the *Insert* menu on the SOLIDWORKS menu bar.
3. From the dropdown menu, select *Reference Geometry* and then select the *Coordinate System* from the cascading context menu.

   The *Coordinate System* dialog box is displayed.



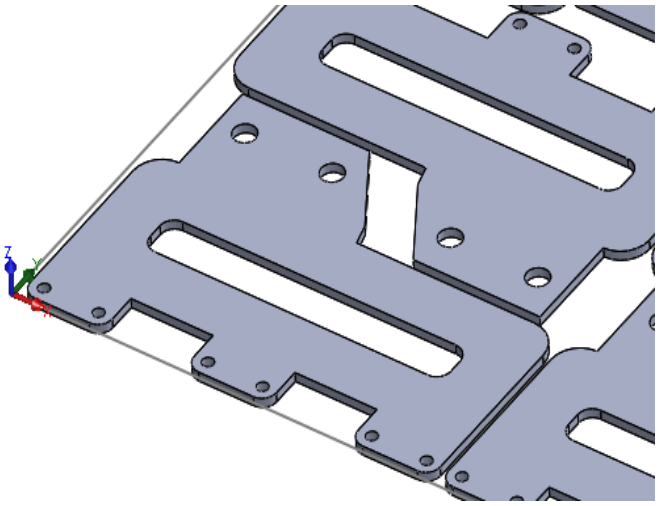
Selecting 'Coordinate System' from cascading menu

4. In the graphics area, click on the *Coordinate System origin*.

   This action will display the selected coordinate system origin in the field of *Selection* group box.

5. The XYZ machining direction should be same as displayed in the image on the right. If necessary, click on the Reverse Axis Direction button to obtain the correct machining direction.
6. Click the *OK* button to save the changes and close the dialog box.

   The defined coordinated system is listed under the *FeatureManager* Design tree.



XYZ machining direction

Step 2: Define the Machine

Before you machine the Nested layout assembly, you need to define the Machine that will be used to machine the assembly.

CAMWorks Feature Tree

1. In the SOLIDWORKS left hand side panel, click on the *CAMWorks Feature Tree* tab. (Note that this tab will be visible only if CAMWorks is loaded as an Add-In within SOLIDWORKS)

   When the CAMWorks Feature tree is displayed, it initially lists *Configurations*, *Machine*, *Part Manager* and *Recycle Bin* items.
2. Double-click on the Machine item (*Machine [Mill - metric]* in this case) to open the *Machine* dialog box.
3. The Machine tab of the Machine dialog box is displayed. This tab allows you to select the machine that the assembly will be machined on. By default, either the *Mill - metric* or *Mill - inch* will be already selected.

   If you wish to select any other Mill machine or a user-defined Machine definition, then highlight it in the Available Machines list and click the Select button.
4. Click on the *Tool Crib* tab, ensure that ***Tool Crib 1(metric)*** is selected.

   To select an alternative tool crib, select the desired tool crib In the Available tool cribs list box and click on the Select button.
5. Click on the *Post Processor* tab. This tab allows you to select a post processor for generating NC codes or for generating enhanced CL files that can be used by external third party post processing programs.

   By default, the sample post processor ***M3AXIS-TUTORIAL*** is selected. For this tutorial, this default post processor will be used.

   If you wish to use another post processor or a customized post processor provided to you by your CAMWorks Reseller, then highlight the desired post processor in the Available list and click the Select button. If the post processor

is not listed, then click on the Browse button to navigate to the folder where the post processor file is located.

6. Click on the *Setup* tab. This tab allows you to set the Fixture Coordinate System for the machine.
   - Since a 2.5 Axis/ 3 Axis Mill Machine will be used to machine the assembly, Indexing will remain set to ***None***.
   - In the *CNC comp options* group box, ensure that the *Calculate safe CNC comp toolpath* option is checked.
   - In the *Fixture Coordinate system* group box, highlight ***Coordinate Sytem1*** in the *Coordinate systems* list box. This action will display the highlighted entity in the *Selected entity* list box.
7. Click *OK* to apply the changes and close the *Machine* dialog box.

Step 3: Addition of nested Parts to Part Manager

The parts that are to be machined must be identified to CAMWorks by adding them to the *Part Manager* item in the CAMWorks Feature tree.

The Assembly document (**.sldasm*) contains different part model documents. In addition to the parts that are going to be machined, the document might contain clamps, fixture of machine components which are included to assist in the layout of the parts and shop documentation. To help CAMWorks identify the components of the assembly file to be machined, the parts that are to be machined must be added to the *Part Manager*.

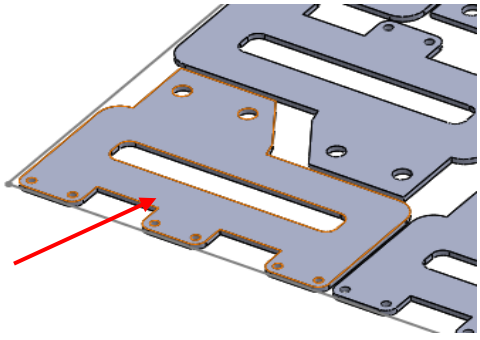
- When machining multiple instances of the same part, you must add all instances to the Part Manager.
- Feature recognition will only run once for each unique part name. Automatic and interactive features will be referenced automatically at all other part instances.

Following are the steps to add parts to the Part Manager:

1. Double click *Part Manager* item in the CAMWorks Feature tree.

   The *Manage Parts* dialog box is displayed.
2. Select the part in the left corner of the assembly as shown in the image on the right.

   For each unique part in the assembly, the first instance that you select is called the **seed part**. When an action is performed on the seed part, the same action will be applied to every other instance of that part in the assembly.



Select the left corner part of Assembly

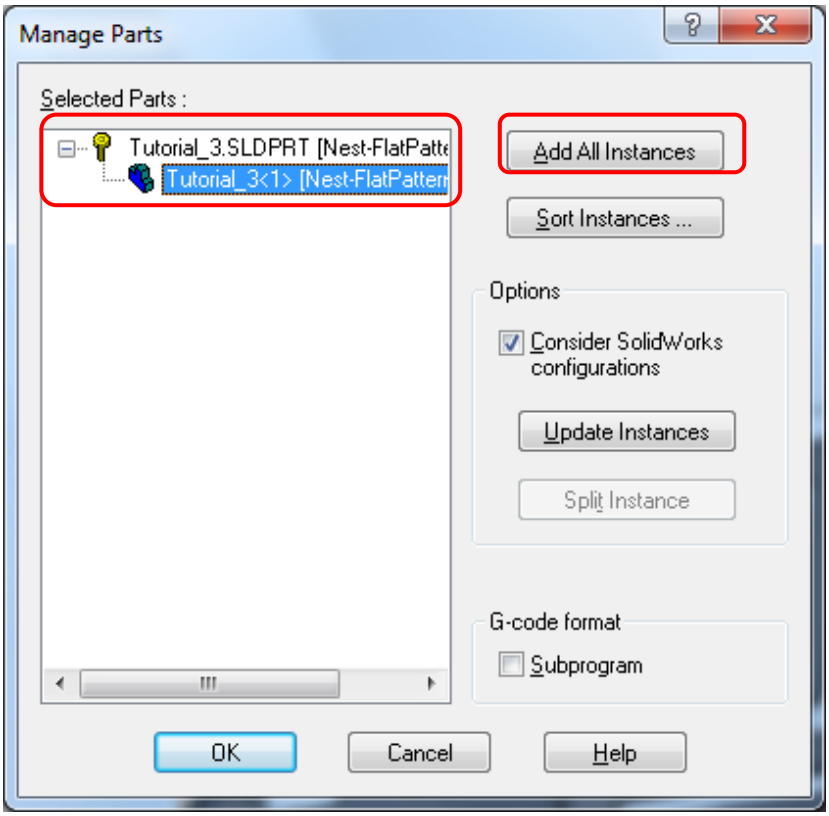
3. Highlight the part in the *Selected Parts* list and click the ***Add All Instances*** button. The parts are listed in the order they are in the file.

   OR

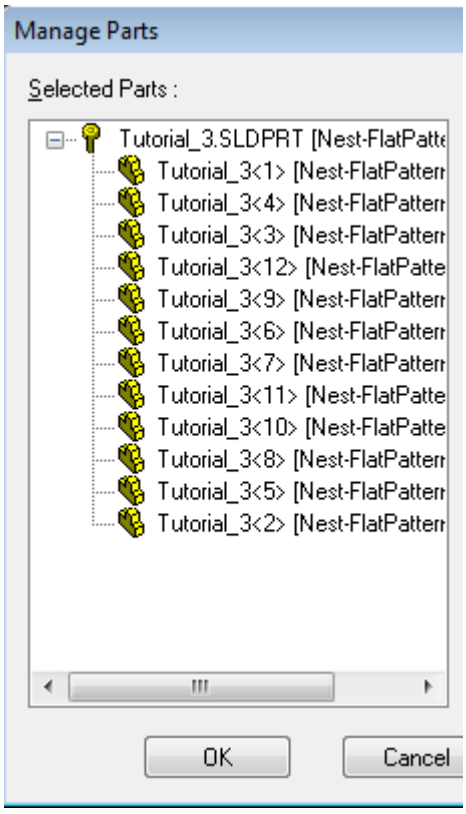
   You can also pick the parts individually in the graphics area or in the ***SOLIDWORKS FeatureManager Design Tree***.

   Part instances can be added at any time. You can select only one instance of a part (the seed part) to work on first and then add other instances later. Any features, operations and toolpaths that have been generated for the seed part are automatically transferred to instances of the same part when they are added in the Manage Parts dialog box.

4. Later in this tutorial, you use the *Sort Instances* function to change the machining order.
5. Click *OK* to exit the *Manage Parts* dialog box. In the CAMWorks Feature tree, under the ***Part Manager*** item, observe that:
   - The part name is listed under the ***Part Manager*** item.
   - A Feature Manager is created for each unique part. In this tutorial, since only one unique part is machined, only one *Feature Manager* item is created. (It will be used to define the Mill Part Setups and machinable features associated with the seed part.)
   - For each unique part, all the instances are listed under the ***Instances*** item. You can re-order and/or delete the part instances in the tree.



Manage Parts Dialog Box



List of Parts

Step 4: Define the Stock

When you add parts in the *Manage Parts* dialog box, a default Stock is created for each part based on a 0.00 bounding box offset (cuboid with the minimum required dimensions from which the part can be machined). The *Stock Manager* dialog box allows you to customize the stock associated with the parts.

In this tutorial, all the default individual stocks of type *Bounding Box* created for each part will be replaced a common stock. All the parts will be machined from this common stock.

1. Double click *Stock Manager* in the CAMWorks Feature tree.

   OR

   Right click *Stock Manager* item in the CAMWorks Feature tree and select *Edit Definition* on the context menu.

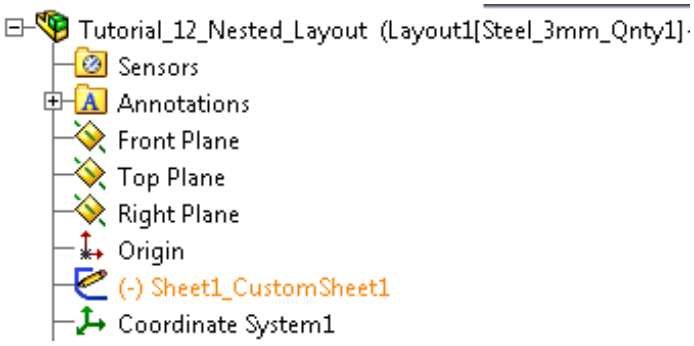
   The *Stock Manager* dialog box is displayed. This dialog box allows you to modify the existing default stock or create new stock for single parts or define common stock for multiple parts. Observe that the default stock is *Bounding Box* with zero offsets.

2. Under Stock Type, select ***Extruded Sketch***.

3. Pick the rectangular sketch representing the sheet in the graphics area.

   OR

   In the top left corner of the graphics area, expand the SOLIDWORKS tree (*Tutorial_12_Nested_Layout*) and select the sketch ***Sheet1_CustomSheet1***.

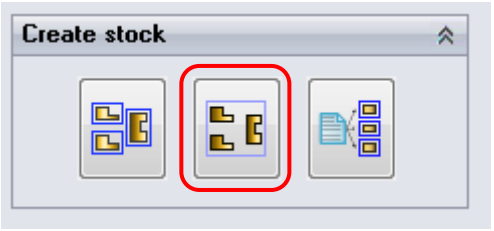
   

   Highlight Sheet1_CustomSheet1 in the tree

   This action will select this sketch.

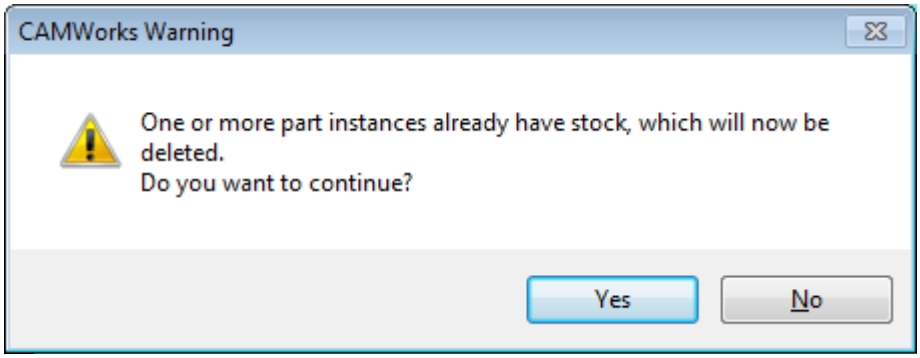
4. In the *Depth* field, set the Depth to **3mm**.

5. Scroll down the *Stock Manager* dialog box and in the *Create Stock* group box, click the *Common* button.

   Click 'Common' button

   CAMWorks will display the warning message stating that parts instances already have stock which will then be deleted.



CAMWorks Warning Message

6. Click *Yes* to delete the individual stocks for the parts and replace them with a common stock.
7. Click *OK* to close the *Stock Manager* dialog box.

Step 5: Defining Machinable Features

The next step is to automatically extract the machinable features using the ***Automatic Feature Recognition (AFR)*** technology available in CAMWorks. The machinable features extracted all applicable to all instances of the part.

At the Mill Part Setup level, features can be inserted interactively using the *New 2.5 Axis Feature* or *New Multi Surface Feature* or *New Part Perimeter Feature* commands. Such an insertion of features is known as ***Interactive Feature Recognition***.

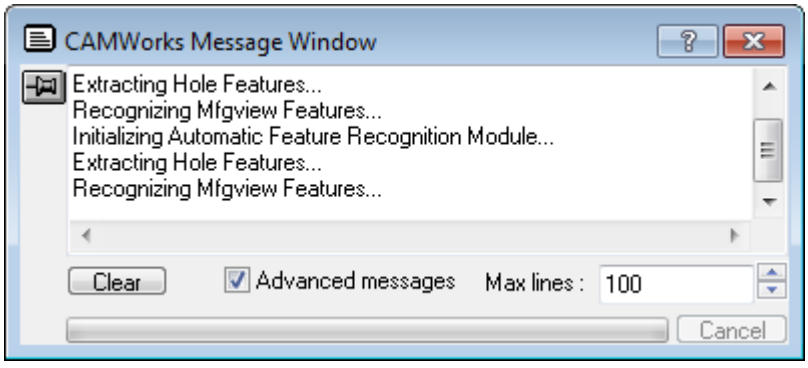
For each unique part, the Machinable Features recognized are added under the *Feature Manager* item of the *CAMWorks Feature Tree*. The features (both automatically recognized or interactively inserted) for the seed part are automatically copied to all other part instances defined in the *Part Manager*.

Extracting Machinable Feature using AFR

1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* in the CAMWorks Feature tree and select *Extract Machinable Features* command on the context menu.
2. The *CAMWorks Message Window* is displayed. This window is displayed automatically to report the

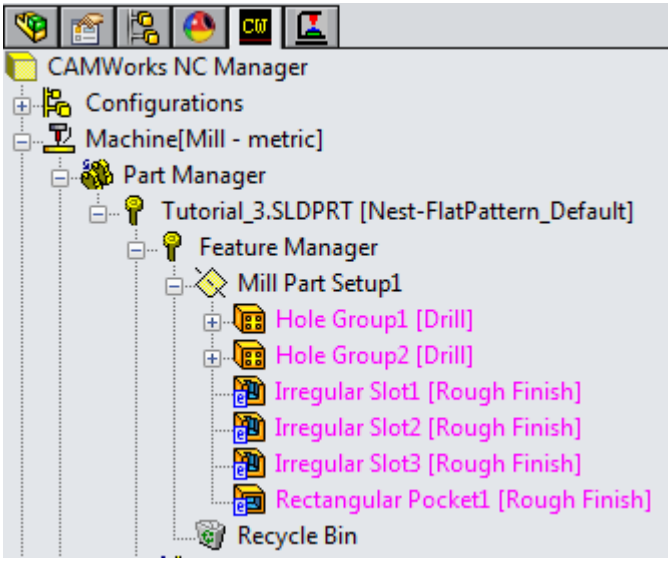


CAMWorks Message Window

progress of the current process. Close this Message Window.

3. On execution of the *Extract Machinable Features* command, CAMWorks generates the Mill part Setup and the machinable features. The items are displayed in the CAMWorks Feature tree under *Part Manager>>Feature Manager*.
4. Expand the *Feature Manager* item in the CAMWorks Feature tree by clicking on the **+** sign next to it.

   The *Feature Manager* lists the Mill Part Setup and machinable features that were automatically recognized using AFR.



List of Machinable Features recognized using AFR

Interactively Inserting Features

The Part Perimeter feature was not recognized automatically using AFR. Hence, this feature will be inserted interactively.

1. Right click *Mill Part Setup1* under the *Feature Manager* and select *New Part Perimeter Feature* on the context menu.

   The *New Perimeter Feature* dialog box is displayed.
2. Within this dialog box, change the Feature type to *Boss*.
3. Click *OK* to close the dialog box.

   The *Perimeter Boss* feature is added to the list of features under *Mill Part Setup1.*

   All features listed under *Mill Part Setup1* are added to the seed part and also to every instance of the part.

When you recognize features by Automatic Feature Recognition (AFR) or Interactive Feature Recognition (IFR), the features listed in the CAMWorks Feature tree will display in Magenta color (by default) till you generate operations for these features. Once a valid operation is generated, the color of the corresponding feature item will change Black color (by default) indicating successful generation of the operation(s).

If operations could not be generated for a feature (because the feature conditions have not been defined in the Technology Database for that particular feature type), then the feature will continue to display in the initial color (Magenta color), thus indicating that they have no operations defined.
You can set these colors on the Display tab in the CAMWorks Options dialog box.

Step 6: Sorting Part Instances

When part instances are automatically added or manually added using the *Add All Instances* button, the instances need not necessarily be listed in the best machining order. CAMWorks provides options for sorting part instances to be processed in a more efficient order.

Following are the steps to sort Part Instances:

1. Under *Setup1* in the CAMWorks Feature Tree, expand all the listed feature items by clicking on the ⊞ plus sign next to them.

   The order in which the part instances are listed under each feature is the machining order for that feature. By default, for all features, the parts are in the order they appear in the *Part Manager*. You can change the order globally for all features or for individual features.

   In this tutorial, you will set the machining order for all the features globally.

2. Double click *Part Manager* in the CAMWorks Feature Tree.
3. Click the *Sort Instances* button in the *Manage Parts* dialog box.
4. The *Sort Instances* dialog box is displayed. This dialog box provides automatic or manual options for sorting the part instances for features in the Setup.

   The Part Manager instances option automatically sorts part instances for all features in the Setup based on the user-defined order of instances listed in the tree under the Part Manager. To set the order using this option, expand the Part Manager and Instances items, then drag and drop the part instances.

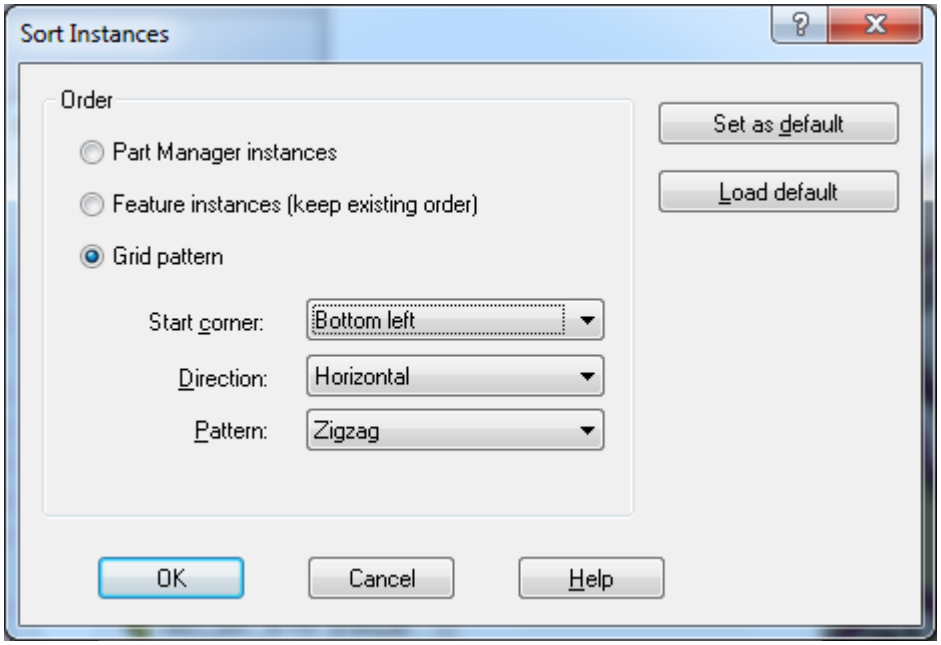
   - The ***Part Manager instances*** option automatically sorts part instances for all features in the Setup based on the user-defined order of instances listed in the tree under the *Part Manager*. To set the order using this option, expand the *Part Manager* and Instances items, then drag and drop the part instances.
   - ***Grid pattern*** automatically sorts part instances for all features in the Setup based on the start corner, processing direction and process order.
   - The ***Feature instances*** option allows you to manually reorder the part instances listed under each feature in the Setup. To set the order using this option, expand a feature in the Setup, then use drag and drop to move the part instances.

You can use one of the automatic methods, then if necessary, select the Feature instances option and make changes to the part order for individual features.

5. Select the *Grid pattern* option.

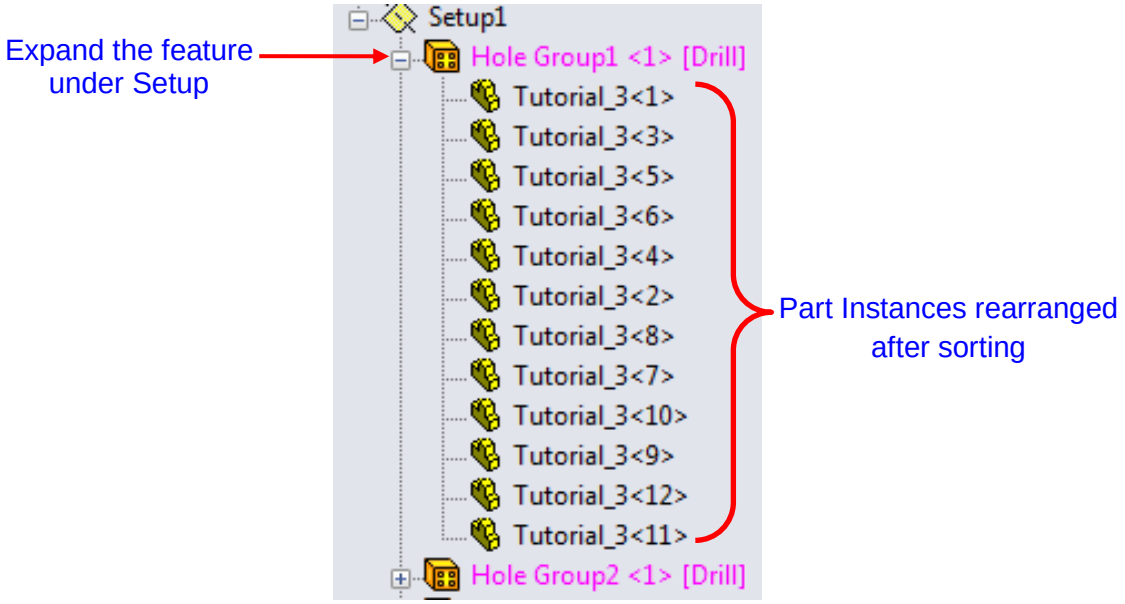
   When you will select the *Grid pattern* option, the order will change for the part instances under every feature in the Setup.

6. Select the following Grid options:
   - Start corner= *Bottom left*
   - Direction= *Horizontal*
   - Pattern= *Zigzag*



Sort Instances Dialog Box

7. Click *OK* to close the *Manage Parts* dialog box.
8. Click the (⊞) plus sign next to any feature listed under *Setup1.* Observe any changes in the order of the part instances.



Part Instances rearranged after executing Sort Instances command

Step 7: Generating the Operation Plan

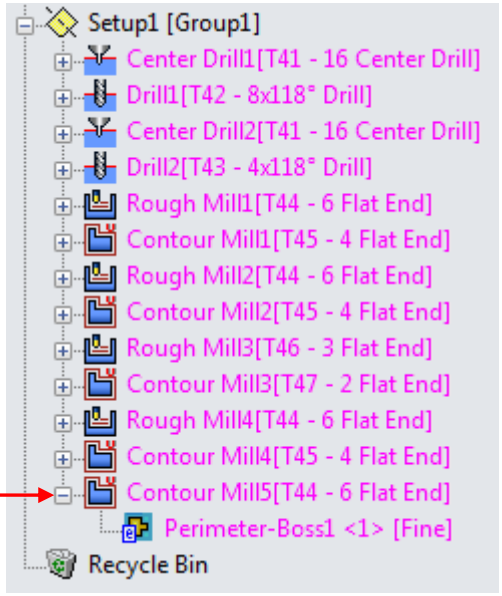
An Operation Plan contains information on how each machinable feature is to be machined and how the NC code will be output. When *Generate Operation Plan* command is executed, operations for each machinable feature are created automatically based on information in the TechDB. The operations generated are listed in the *CAMWorks Operation tree*.

To execute this command:

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click the *CAMWorks NC Manager* item of the CAMWorks Feature tree and select *Generate Operation Plan* on the context menu.
2. On execution of this command, CAMWorks switches to the CAMWorks Operation tree. All the operations generated are listed under *Setup1* in the CAMWorks Operation tree.



Expand an operation to view the feature for which it has been generated

Generated Operations listed in CAMWorks Operation Tree

When Operations are generated or interactively inserted, they will be displayed in Magenta color (by default) in the CAMWorks Operation Tree till you generate toolpaths for these operations. Once the toolpath is generated, the color of the corresponding operation will change to Black color (by default) indicating successful generation of the toolpath.

3. Click on the (⊞) plus sign next to an operation indicates the feature for which the operation has been generated. One or more operations may be generated for each machinable feature. For example, click on the ⊞ sign next to *Contour Mill5* operation. This operation has been generated for the *Perimeter Boss* feature.

Step 8: Adjusting Operation Parameters

While generating the nested layout, the ***Part-to-part distance*** was set to **3 mm** and the ***Part-to-sheet distance*** was set to **2 mm**.

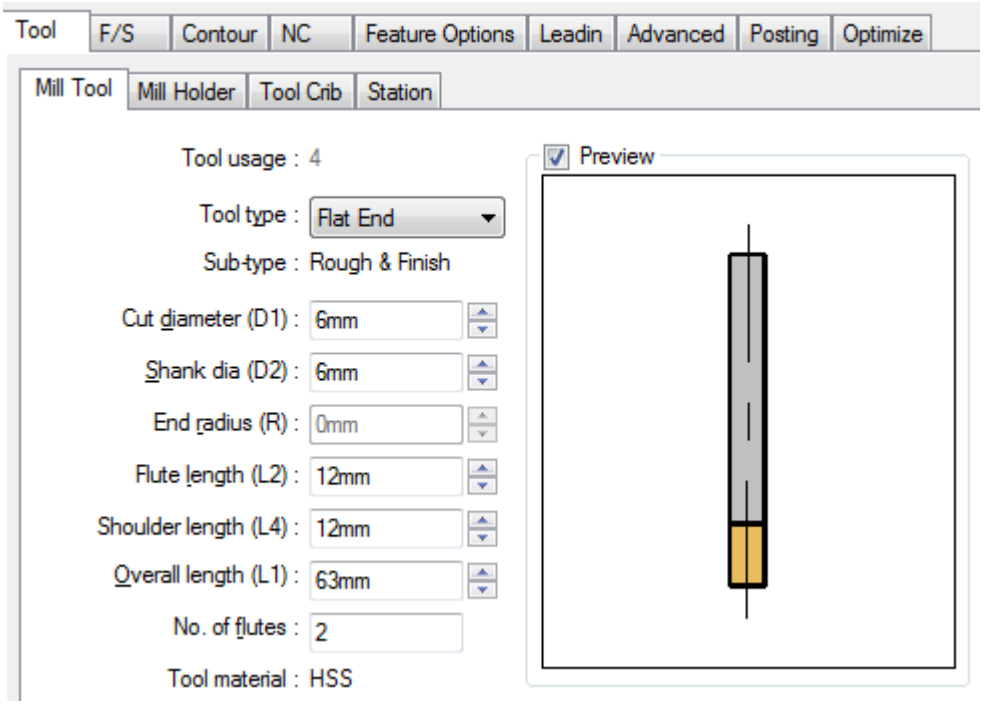
The ***Contour Mill5*** operation generated for Perimeter Boss feature is used to machine the perimeter of the part and thereby separate it from the common stock. Since the *Part-to-part distance* is **3 mm** and the *Part-to-sheet distance* is **2 mm**, the *Flat End Mill tool* used for machining the *Contour Mill5* operation should not exceed 2mm in diameter else it might end up gouging the part.

1. In the Operation tree, double click *Contour Mill5* operation.

   OR

   Right click *Contour Mill5* and select *Edit Definition* on the context menu.

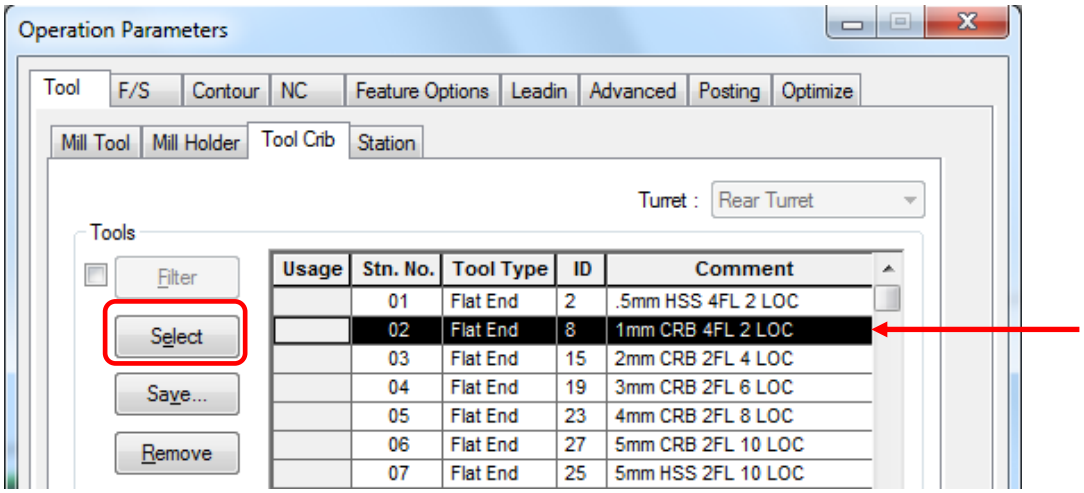
   The *Operation Parameters* dialog box is displayed.
2. Click on the *Tool* tab and select the *Mill Tool* Page.



Mill Tool Page under Tool Tab of Operation Parameters dialog box

Observe that the diameter of the tool currently selected for this operation is 6mm. This tool will gouge the part. Hence, another tool needs to be selected for machining this operation.

3. Under the *Tool* tab, click on the *Tool Crib* page.
4. In the displayed tool crib, highlight the **1mm** diameter *Flat End* Mill Tool within the list of displayed tools.
5. Click the *Select* button. This action will assign the highlighted tool as the tool to be used for machining this operation.

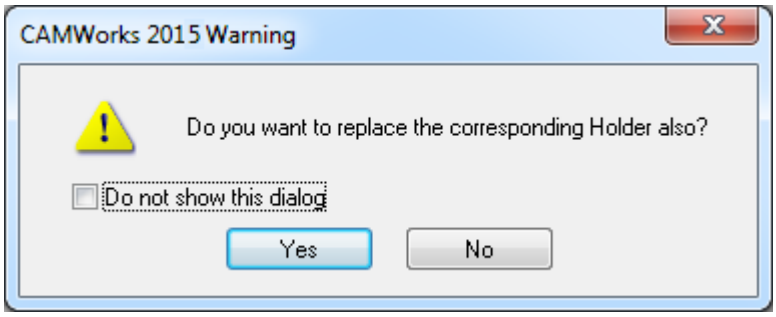


Highlight a 1mm Flat End Mill Tool in the Tool Crib and click the Select button

6. CAMWorks will display a warning message indicating whether you wish to replace the corresponding holder also. Click *Yes* to replace the corresponding holder.

   The *Mill Tool* page is now displayed. It displays the parameters of the selected tool.

7. Click *OK* to apply the changes and close the *Operation Parameters* dialog box.



CAMWorks Warning Message

Step 9: Defining G-code Program Zero Location

Toolpaths can be output relative to the Part Setup origin or a global Setup origin. In this tutorial, you will use the Part Setup origin. The Part Setup origin specifies only the toolpath zero point, not the XYZ machining direction. The machining direction is based on the Fixture Coordinate System. When machining multiple instances of the same part, the origin is defined relative to the first (seed) part and referenced for all other instances of the same part.

1. Double click *Setup1* in the CAMWorks Operation tree.

   The *Setup Parameters* dialog box is displayed.

2. On the *Origin* tab, make sure *Part Setup origin* is selected for the Output origin.

   Note that when Setup origin is selected, you can specify the origin using several methods.

3. Click on the *Offset* tab.

   The order of the parts on this page affects only the assignment of the offsets, not the machining order.

4. In the Sort by group box, select *Grid pattern*.

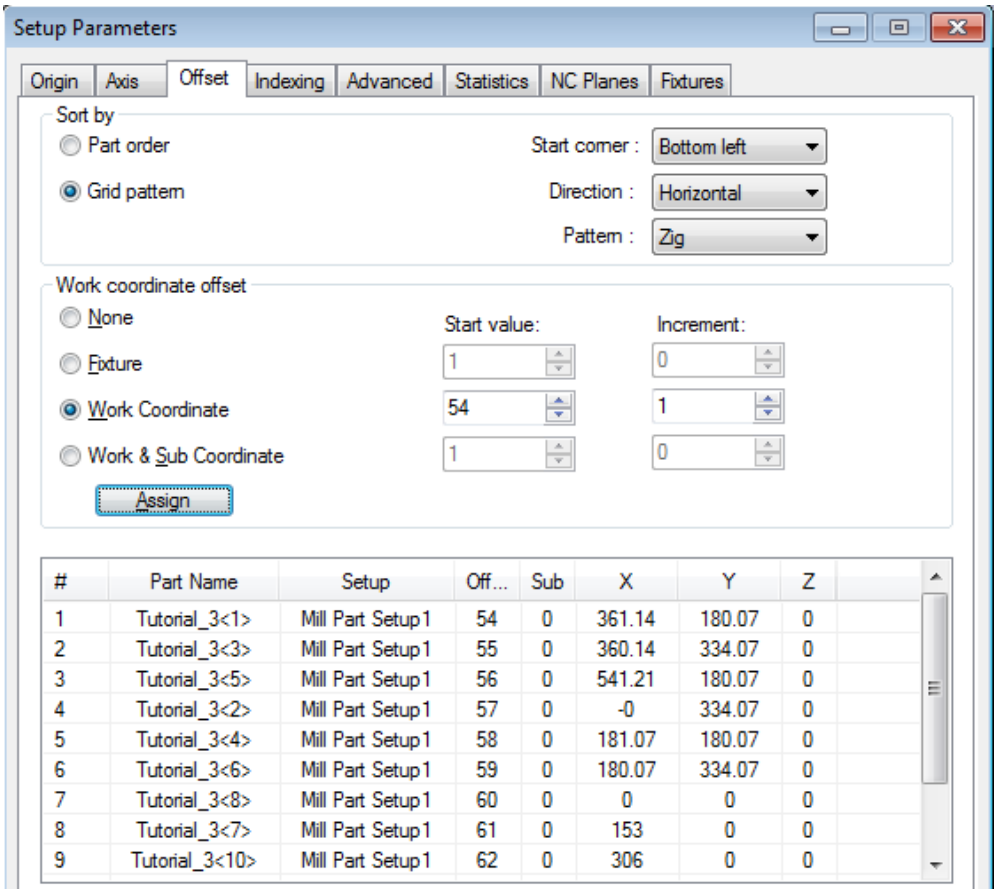
   When you pick this option, the parts in the table are automatically reordered based on the current settings for Start corner, Direction and Pattern.

5. Set the Grid pattern parameters to the same settings you used when sorting part instances for the machining order (Step 7-Point 6):

   - Start corner = *Bottom left* (specifies which part, based on a grid layout, will be assigned the register equal to the Start Value)
   - Direction = *Horizontal* (relative to the Start corner part, the Direction defines which part will be assigned the next offset register value)
   - Pattern = *Zigzag* (defines the order the offsets are assigned)

     Notice that the part order is updated in the table. You can specify a programmable coordinate offset and assign an offset to each part.

6. Set the Work coordinate offset to *Work Coordinate*. This option will output G54, G55, etc.
7. Set the *Start* value to **54** and the *Increment* to **1**.
8. For the *Start* value, specify only the numerical value of the offset and not the G-code prefix.
9. Click the *Assign* button of the Work Coordinate offset group box. The numbers update in the Offset and Sub columns in the table.
10. Click *OK* to close the Setup Parameters dialog box.



Setup Parameters Dialog Box

Note: **Changing the machining order does not automatically change the offset assignments. If you want the offset order to correspond to the machining order, you need to sort the parts and reassign the offsets on the Offset tab.**

Step 10: Generating Toolpaths and Sorting Operations

Operations are generated for machinable features and listed in the CAMWorks Operation tree in the same order as the corresponding features in the CAMWorks Feature tree. This sequence is not necessarily the ideal machining sequence. Operations can be sorted in order to reduce machining time.

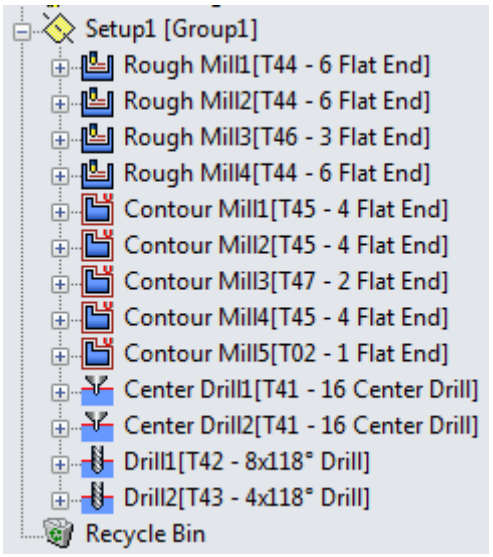
In this step, all the operations listed in the CAMWorks Operation tree will be sorted in order to create a logical machining sequence.

1. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click *Setup1* in the Operation tree and select *Generate Toolpath* on the context menu.

   On executing the *Generate Toolpath* command, CAMWorks calculates the toolpaths for each operation in the Setup. The font color of all the listed operations in the Operation tree changes from magenta to black. This change in color indicates that toolpaths were successfully generated.



Updated list of operations after executing the Sort Command

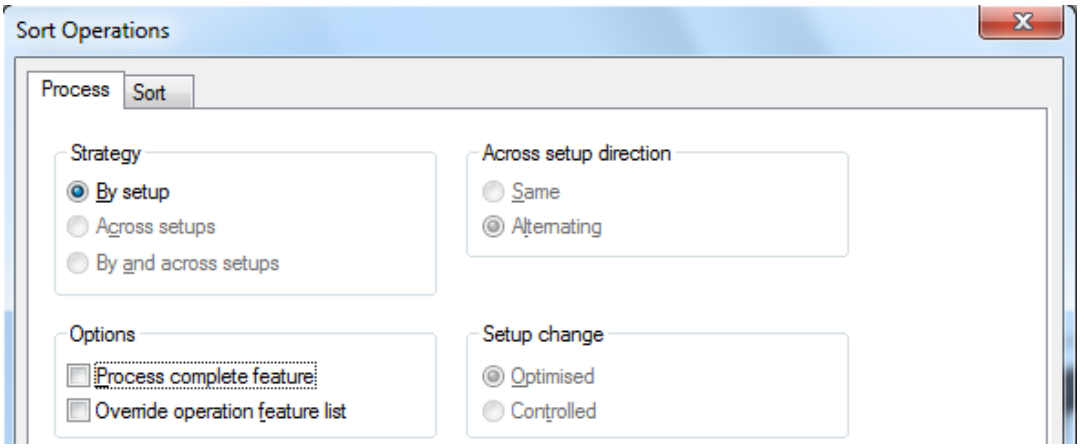
   Note: **If an operation displays in a magenta color instead of black even after the Generate Toolpath command has been executed, then it indicates that the toolpath has not been generated. This might occur in one of the following situations:**

   i. **When you insert a new operation interactively;**

   ii. **When you insert a new feature interactively and then generate operations for the new feature**

   iii. **When CAMWorks cannot generate the toolpath for an operation because of an error in the toolpath algorithm or when a parameter is not correct.**

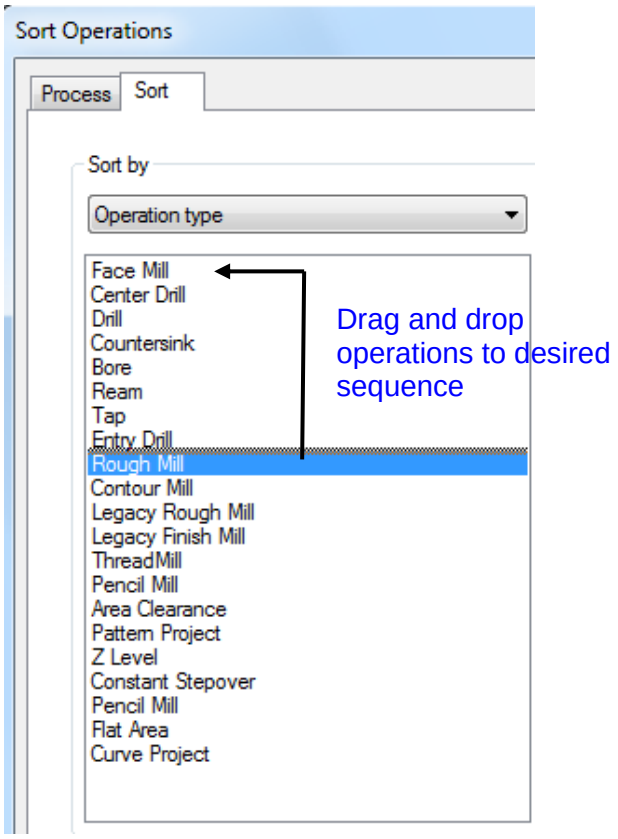
2. Right click *Setup1* in the CAMWorks Operation tree and select *Sort Operations* on the context menu.
3. On the *Process* tab, remove the check mark from the *Process complete feature* option.



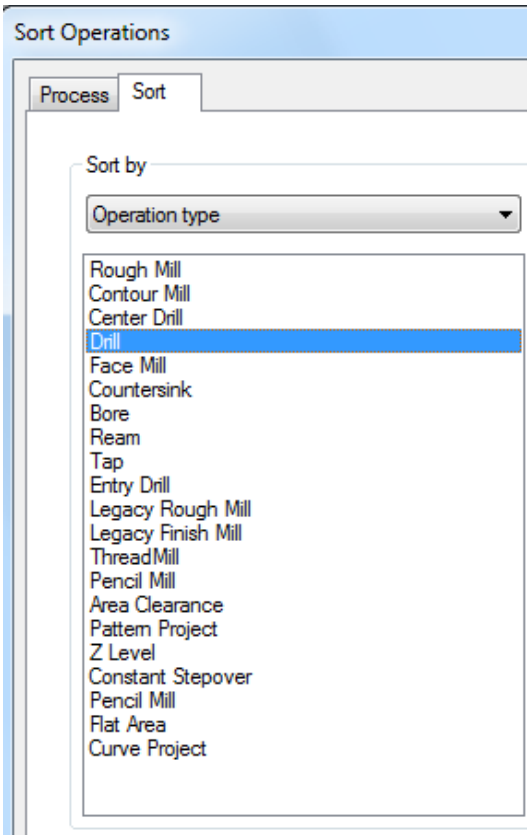
Remove the check mark from the 'Process complete feature' option

4. Click on the *Sort* tab.

5. In the ***Sort by Operation Type*** group box, drag and drop operations so that *Rough Mill* is at the top of the list, followed by *Contour Mill*, *Center Drill*, and *Drill*.



Operations before sorting

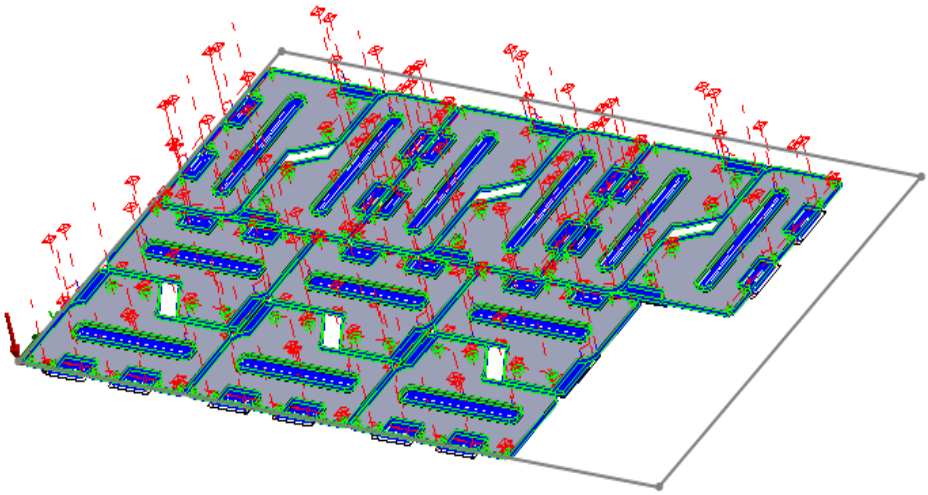


Operations after sorting

6. Click *Apply* button and confirm that the tree view updates to sort the operations according to this order. If it updates as expected, then click *OK*.

   The operations under *Setup1* are sorted based in the order on the Sort tab.

7. Left click any operation in the CAMWorks Operation tree. That operation will be highlighted in the Operation tree.
   - The toolpath for that highlighted operation will be displayed in the graphics area. As you highlight each operation in the tree, the toolpaths for that corresponding operation will be displayed.
   - Turning operation parameters can be edited and the operation can be renamed, moved, suppressed, deleted, etc. after toolpaths have been generated. These commands are available in the RMB context menu.
   - If you make any changes, the toolpaths must be updated by executing the *Generate Toolpath* command again at the Setup level.
8. Hold down the *Shift key* and select the first and last operation in the tree. This action selects all the operations. The toolpaths for all the operations will be displayed on the part showing the centerline of the toolpath.



Toolpaths for all the operations displayed on the part when all the operations are selected in the Operation tree

Step 11: Simulate Toolpaths

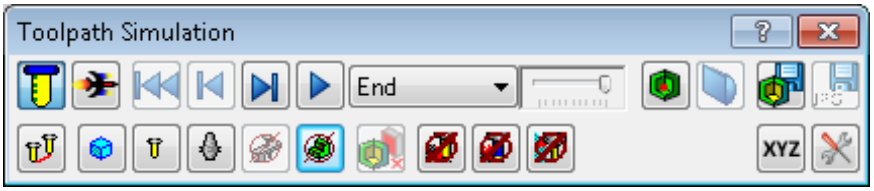
CAMWorks provides the ability to simulate the toolpaths showing the tool movement and the resulting shape of the part.

1. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click on *Setup1* in the Operation tree and select *Simulate Toolpath* on the context menu.

   The Toolpath Simulation toolbar is displayed.

   Toolpath Simulation toolbar

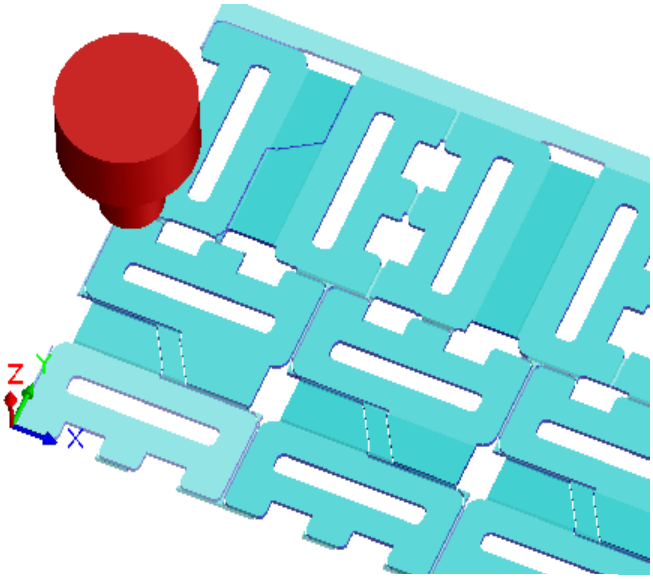
   When you click on the display control buttons of the Toolpath Simulation toolbar, the available settings associated with that button are displayed in a dropdown list.

2. Set the following display options:
   - Stock: *Translucent display*
   - Tool: *Shaded display*
   - Tool Holder: *Shaded display*

3. Click the *Run* button.

   The simulation is run with the tool displayed during simulation.

4.  Use the Simulation Speed Control slider to control the speed of the Simulation.
5.  To pause the simulation while it is running, click on the *Pause* button. When you click  *Run* button again, the Simulation will continue from the point where it was paused.
6.  Click the *Close* button in the upper right corner of the Simulation toolbar to exit the simulation mode and return to the SOLIDWORKS display.



Toolpath Simulation

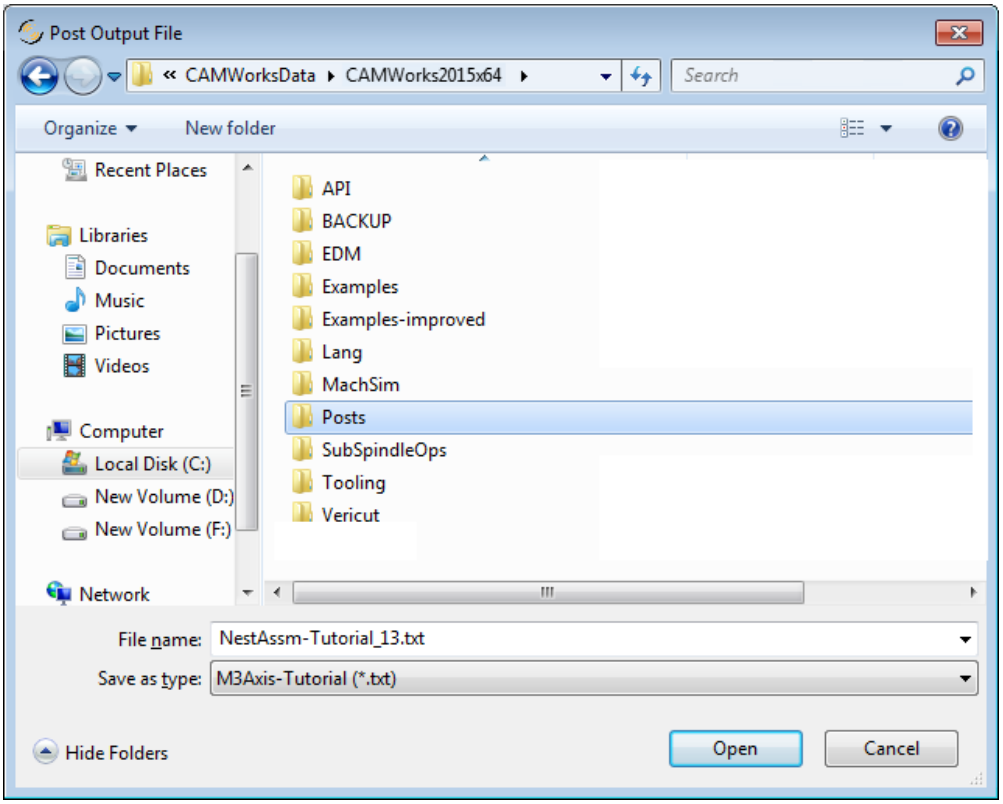
Step 12: Post Processing Toolpaths

Post processing is the final step in generating the NC program file. When you use a CAMWorks internal post processor, this step translates generalized toolpath and operation information into NC code for a specific machine tool controller. CAMWorks creates NC code for each toolpath in the order the operation appears in the CAMWorks Operation tree. When you post process a part, CAMWorks creates two files: the NC program and the Setup Sheet. These are text files that you can read, edit and print using a word processor or text editor.

In this tutorial, you will post process all the operations and generate the NC program:

1.  Click the *Post Process* button on the CAMWorks Command Manager.
   OR
   Right click on the *CAMWorks NC Manager* in the Operation tree and select *Post Process* on the context menu.
   The *Post Output File* dialog box is displayed so that you can save the NC program file.
   Typically, the NC program and Setup Sheet files are stored in the folder that contained the last part that was opened. If you want these files in another location, you can change the folder location.

Note: **If the Post Process command is grayed out on the CAMWorks Command Manager or on any context menu, make sure that you have selected a post processor and generated the toolpaths. Refer Step 11 in this tutorial.**



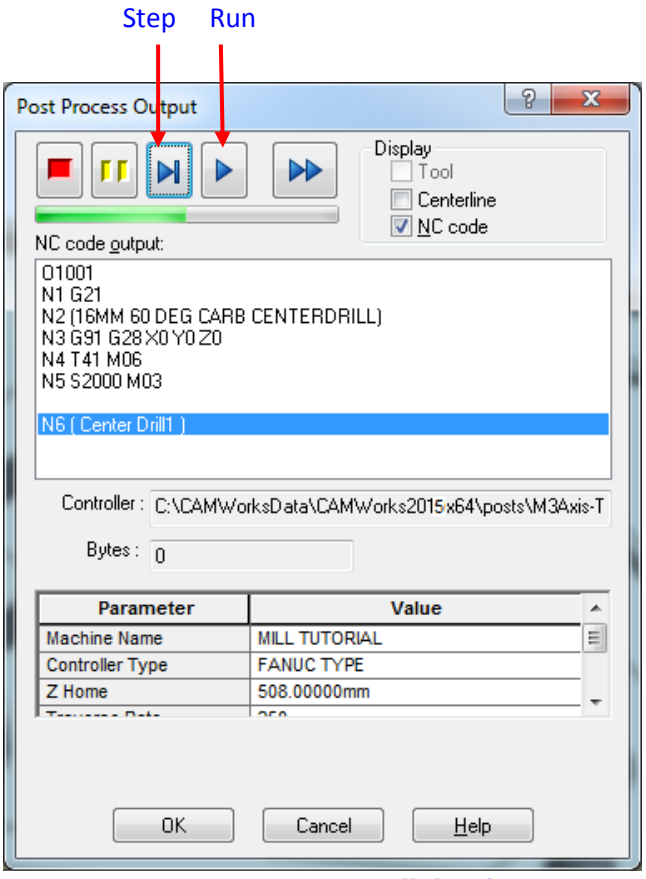
Post Output File dialog box

2. In the *Post Output File* dialog box, click the down arrow to the right of the *Save as type* box.
   CAMWorks provides a list of commonly used extensions that you can select. For this exercise, use the *.txt* extension.

   Note: **If you want change the default extension from .txt to one of the ones in the list or if you want a different file name extension for NC program files, you can edit or create a .pinf file and specify the new extension. For more information on making these changes, see the online Help.**

3. In the *File name* textbox, type the suitable file name, and then click *Save* button.
4. The *Post Process Output* dialog box is displayed. Click the *Step* button  on the control bar at the top.

   CAMWorks starts to generate the



Post process Output dialog box

NC program and the first line of NC code displays in the NC code output view box. The post processing mode is set to post process one line of code at a time (Step mode).

5. Click the *Step* button. The next line of NC code is displayed.
6. Click the *Run* button  . Post processing continues until it is completed.
7. When the post processing is finished, view the code using the vertical scroll bar.
8. Click *OK* to close the dialog box.

Note: **To understand the complete the process from defining the machine and extracting the machinable features to simulating toolpath and generating the NC code for nested parts using the CAMWorks, refer the Mill Assemblies Tutorial of CAMWorks. To locate the tutorial, select the *Start menu* on the Windows taskbar and follow the path:**

All Programs>>CAMWorks201x>>Manuals>>Mill Assemblies Tutorial

TUTORIAL 13 – GENERATING NC CODES FOR NESTED LAYOUTS USING CAMWORKS (II)

Topics covered in this Tutorial:

- Functionality to link CAMWorks Nesting with CAMWorks
- Tutorial illustrating how to generate NC Codes for Nested Layouts using CAMWorks

The previous tutorial explained how to generate NC codes using the CAMWorks application.

If you are using ***CAMWorks Nesting 2014 SP1*** or a later version in conjunction with ***CAMWorks 2014 SP2.2*** or a later version, then a new functionality that links the CAMWorks Nesting output as the input for CAMWorks comes into effect.

In this tutorial, you will learn how to generate NC codes for nested layouts using CAMWorks when the functionality to link CAMWorks Nesting with CAMWorks is enabled.

Functionality to link CAMWorks Nesting with CAMWorks

A new functionality provided with ***CAMWorks Nesting 2014 SP1*** version allows you to automatically link the nested layout output of CAMWorks Nesting as the input for CAMWorks.

When this functionality is enabled, the nested layouts output generated using CAMWorks Nesting will be automatically fed as the input assembly for CAMWorks. This is achieved by automatically listing all the nested parts in the *CAMWorks Part Manager* and auto-defining the common stock for the nested assemblies. This automation saves considerable time by reducing the steps required to generate the NC code.

The new functionality introduced in CAMWorks Nesting 2014 SP1 version links the CAMWorks Nesting application with the CAMWorks application. This linking is achieved by:

- **Automatic addition of nested parts in nested layouts to CAMWorks Part Manager**
- **Automatic definition of the stock (from which the parts will be machined) in the CAMWorks Stock Manager.**

Pre-requisites for using this functionality

The functionality to link the CAMWorks Nesting application with CAMWorks application will work if and only if all the below conditions are fulfilled:

- The *CAMWorks Nesting* version should be ***CAMWorks Nesting 2014 SP1*** or a later version.
- The *CAMWorks* version should be ***CAMWorks 2014 SP2.2*** or later version.
- Both *CAMWorks Nesting* and *CAMWorks* should be loaded as Add-Ins in SOLIDWORKS.

- The functionality should be enabled in the *DefaultValues.ini* Configuration file. (By default, it is enabled.)

Advantages of this functionality

In CAMWorks Nesting versions prior to the 2014 SP1.0 version, after the nested layouts were generated, users had to manually add instances of the parts present in each nested layout to the *CAMWorks Part Manager*. The settings for the common stock too had to be manually defined. These steps could be time-consuming.

From *CAMWorks Nesting 2014 SP1* version, the steps for adding parts to the *CAMWorks Part Manager* and defining the stock can be automated using this new functionality.
Refer the flowchart in the previous tutorial to gain an understanding of the steps involved in generating NC codes and the steps that are automated when this functionality is used.

Enabling the functionality

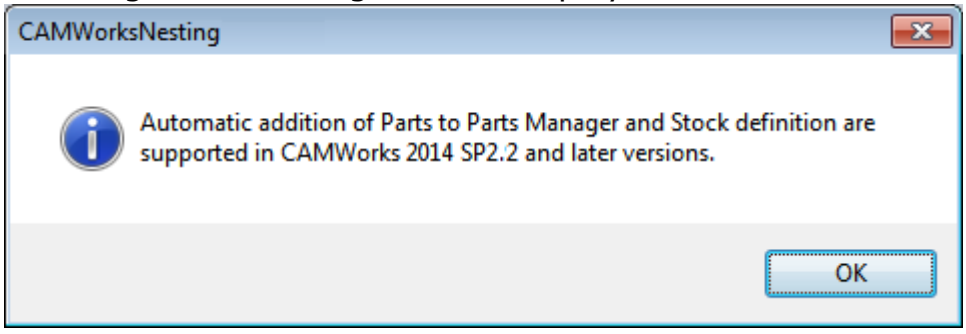
The option to link CAMWorks Nesting with CAMWorks is controlled from the CAMWorks Nesting configuration file DefaultValues.ini through the flag "***AddPartstoCWManager***". The functionality is enabled when the flag is set to "1" and disabled when the flag is set to "0". By default, this option is enabled.

For more details, read: Enabling/disabling the functionality to add nested parts to the CAMWorks Part Manager.

How the functionality works

When enabled, the functionality works in the following manner:

1. After nested layouts are generated using CAMWorks Nesting, each nested layout will be listed in the *SOLIDWORKS Configurations Manager*.
2. The CAMWorks Nesting application will then check for the presence of the CAMWorks Add-In.
   a. If the CAMWorks Add-In is not dete cted, then this functionality will not work.
   b. If the CAMWorks Add-In is detected, but it is a version lower than *CAMWorks 2014 SP2.2*, then the functionality will not work and the following error message will be displayed:



CAMWorks Warning Message

c. If the CAMWorks Add-In is detected and it is *CAMWorks 2014 SP2.2* or a higher version, then this functionality will come into effect. The *CAMWorks Feature Tree* tab will be populated in the following manner:
   i. If multiple nested layouts are generated after the nesting process, then each nested layout assembly will be listed under the *Configurations* item. If only one nested layout is generated, then it won' be listed under *Configurations* item.
   ii. The parts (instances) present in each nested layout will be automatically listed in the *CAMWorks Part Manager*. (Users can delete unwanted parts listed in the *Part Manager* using the *Delete* option.)
   iii. In the *CAMWorks Stock Manager*, a common stock of type *Extruded Sketch* will be automatically defined for the parts present in each nested layout.

Automatic Definition of Stock in CAMWorks Stock Manager

For each nested layout asembly input into CAMWorks, a common stock of type *Extruded Sketch* will be defined in the *CAMWorks Stock Manager*. The perimeter of the Sheet used to nest the parts will be used as the sketch for extruding. If Sheet sketch is not available (for example in cases where is sheet was defined from a **.dxf* file), then the stock of type 'Bounding box' will be used.

When the functionality to link CAMWorks Nesting with CAMWorks is enabled, then for each nested layout, the automatic definition of the stock in the CAMWorks Stock Manager will have the following properties:

- **The stock created will be a common stock from which all the nested parts will be machined.**
- **The stock type will be *Extruded Sketch* where the dimensions (length and breadth of the cuboid stock will be derived from a sketch).**
- **The sketch picked for defining this stock will be the sketch representing the dimensions of the sheet in which the parts are nested. (The sketch representing the dimensions of the sheet will be listed in the SOLIDWORKS *FeatureManger Design Tree* after the nested layouts are generated.)**
- **The height of the stock will be equivalent to the thickness of the parts.**

Tutorial illustrating Generating of NC codes for Nested Layouts

This tutorial is divided into two sections:

- Section I illustrates how to generate Nested layouts for the example parts using the *CAMWorks Nesting* application.
- Section II illustrates how to generate NC codes for the Nested layouts.

Section I: Generating Nested layouts

In this section, you will nest an assembly comprising two native sheet metal parts of different thicknesses.

1. In the configuration file DefaultValues.ini (located within the CAMWorks Nesting Installation folder), ensure that the flag **FixComponent** under [*NestingData*] section is set to '1'.

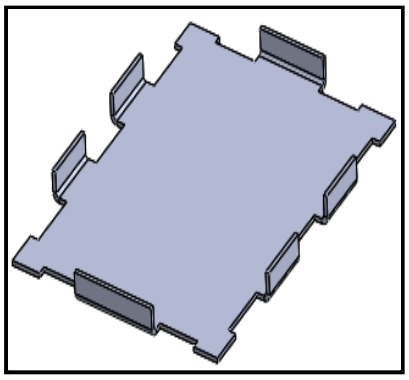
   This setting will ensure that after the Nested layouts are generated, the parts in the Nested layout assembly do not get accidentally repositioned.

2. Ensure that both *CAMWorks Nesting 2014* and *CAMWorks 2014* are loaded as Add-Ins within SOLIDWORKS.

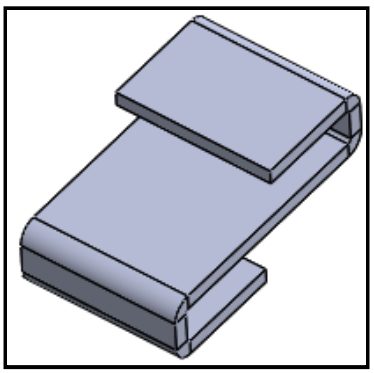
3. Open the assembly file named ***Tutorial_13.sldasm*** located in the following folder location:

   Drive:*CAMWorksNestingData\CAMWorksNesting 201x\Examples\Tutorials\Assemblies\Tutorial_13*

   This assembly file comprises of the following two sheet metal parts.



Tutorial_13_a.sldprt



Tutorial_13_b.sldprt

4. Select the *Create Nest Job* command on the CAMWorks Nesting Ribbon bar. OR

   In the SOLIDWORKS menu bar, select ***CAMWorksNesting>>Create Nest Job***.

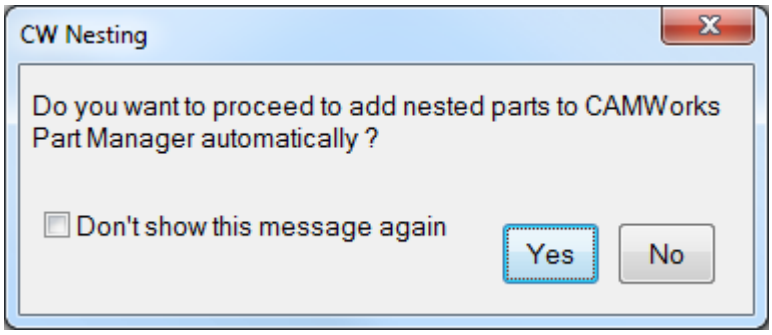
5. The *Create Nesting Job* dialog box is displayed. In the *Part Data* tab, assign the following values to the various parameters:

Part	Thickness	Quantity	Material	Step Angle	Grain Direction
Tutorial_13_a.sldprt	1 mm	12	Steel	90^0	None
Tutorial_13_b.sldprt	3 mm	12	Steel	90^0	None

6. Since the two parts to be nested have different thicknesses, two different sheets of varying thicknesses corresponding to the parts need to be defined. In the *Sheet Data* tab, add two *Custom size* sheets with the following dimensions and values assigned to the parameters:

Sheet Name	Length (mm)	Width (mm)	Thickness	Material	Quantity	Grain Direction	Assembly Template
CustomSheet1	500	200	1 mm	Steel	1	None	Default
CustomSheet2	250	250	3 mm	Steel	1	None	Default

7. Within the *Multi head options* tab, in the *Machine Data* group box, ensure that the Machine selected is *SingleTHMachine* (a machine with a single tool head). The *Number of tool heads* for the machine will display 1.
8. In the Nesting Data group box:
   - Assign a *Part to part distance* of **5mm** and *Part to sheet distance* of **5mm**.
   - Select *Fast Nesting* as the Nesting Type.
   - If you wish to save the output nested assembly in a folder location other than the default location, then specify the location in the *Output Assembly* field by using the *Browse* button.
   - Ensure that the *Save output as dxf* option is unchecked.
9. Click the *OK* button to execute the Nesting command.
10. CAMWorks will display the warning message
11. CAMWorks Nesting will display a warning message prompting whether you wish to add the nested parts to CAMWorks Part Manager automatically.

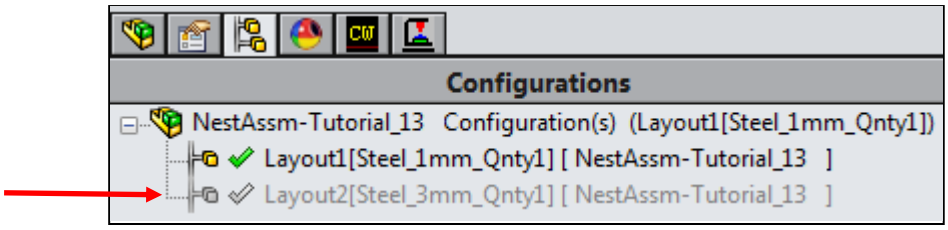


CAMWorks Nesting Warning Message

12. Two nested layouts will be generated. These nested layouts will be saved as a SOLIDWORKS Assembly file (**.sldasm*).
13. The *Summary text* file indicates that all the parts have been nested. Close this text file.
14. In the SOLIDWORKS left hand side panel, click on the SOLIDWORKS *ConfigurationsManager* tab. Observe that the nested layouts are listed under this tab.

    You can switch the nested layout assembly currently displayed in the graphics area by double-clicking on the desired nested layout assembly listed under this tab.

Double-click on *Layout 2* to display it in the graphics area. (The first nested layout is usually displayed in the graphics area by default.)



SOLIDWORKS ConfigurationsManager Tree

15. Click on the SOLIDWORKS *FeatureManager Design Tree* tab. Observe that the sketches for the three sheets are listed in this tab. These sketches will be used to define the common stock (of type *Extruded Sketch*) for each nested layout assembly.

Section II: Generating NC codes using CAMWorks

Step 1: Defining the Fixture Coordinate System for the Machine

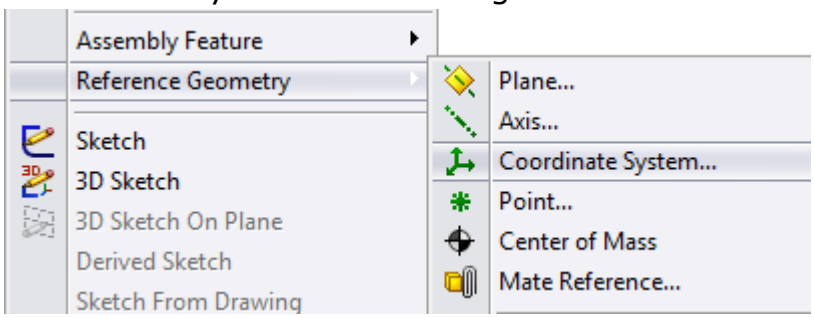
The Fixture Coordinate System defines the "home point" or "main zero" position on the machine. It defines the default G-code origin, defines the XYZ machining directions and acts as a reference point, if subroutines are used. This coordinate system needs to be defined in the *SOLIDWORKS* ***FeatureManager Design Tree***.

Following are the steps to define the Fixture Coordinate System:

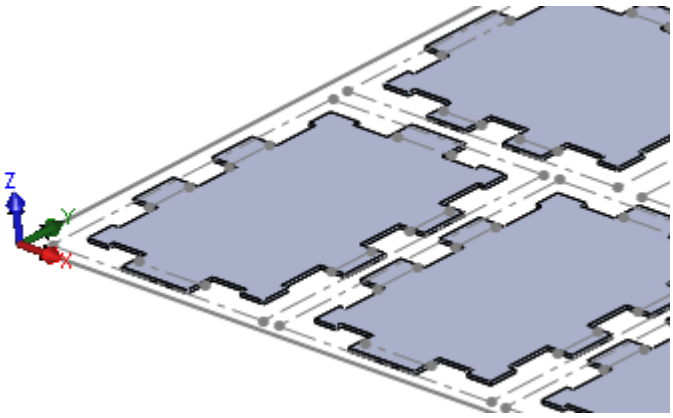
1. If necessary, rotate and zoom the nested layout assembly in the graphics area to clearly view the position where you desire to assign the coordinate system.
2. Click the ***Insert*** menu on the SOLIDWORKS menu bar.
3. From the dropdown menu, select *Reference Geometry* and then select the *Coordinate System* from the cascading menu.

   The *Coordinate System* dialog box is displayed.
4. In the graphics area, click on the point you wish to assign as the *Coordinate System origin*.

   This action will display the selected coordinate system origin in the *Selection* field of the dialog box.



Selecting 'Coordinate System' from cascading menu



XYZ machining direction

5. The XYZ machining direction should be same as displayed in the image on the right. If necessary, click on the ***Reverse Axis Direction*** button to obtain the correct machining direction.
6. Click the *OK* button to save the changes and close the dialog box.

   The defined coordinated system is listed under the *FeatureManager Design Tree*.

Step 2: Defining the Machine

Before you machine the Nested layout assemblies, you need to define the Machine that will be used to machine the assembly.

1. In the SOLIDWORKS left hand side panel, click on the *CAMWorks Feature Tree* tab. (Note that this tab will be visible only if CAMWorks is loaded as an Add-In within SOLIDWORKS)
2. Double-click on the *Machine item* (***Machine [Mill - metric]*** in this case) to open the *Machine* dialog box.
3. The Machine tab of the Machine dialog box is displayed. This tab allows you to select the machine that the assembly will be machine on. By default, either the *Mill - metric* or *Mill - inch* will be already selected.

   If you wish to select any other Mill machine or a user-defined Machine definition, then highlight it in the Available Machines list and click the Select button.
4. Click on the *Tool Crib* tab, ensure that ***Tool crib1 (metric)*** is selected.

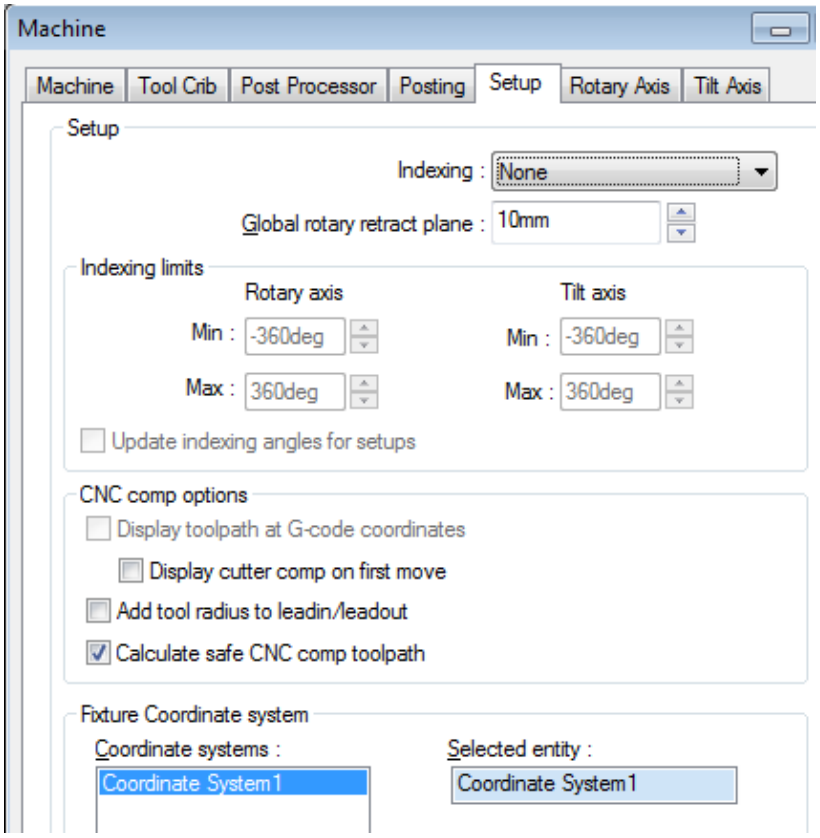
   To select an alternative tool crib, select the desired tool crib In the *Available tool cribs* list box and click on the Select button.
5. Click on the *Post Processor* tab. This tab allows you to select a post processor for generating NC codes or for generating enhanced CL files that can be used by external third party post processing programs.

   By default, the sample post processor ***M3AXIS-TUTORIAL*** is selected. For this tutorial, this default post processor will be used.

   If you wish to use another post processor or a customized post processor provided to you by your CAMWorks Reseller, then highlight the desired post processor in the Available list and click the Select button. If the post processor is not listed, then click on the Browse button to navigate to the folder where the post processor file is located.
6. Click on the *Setup* tab. This tab allows you to set the Fixture Coordinate System for the machine.
   - Since a 2.5 Axis/ 3 Axis Mill Machine will be used to machine the assembly, Indexing will remain set to ***None***.
   - In the *CNC comp options* group box, ensure that the *Calculate safe CNC comp toolpath* option is checked.
   - In the *Fixture Coordinate system* group box, highlight ***Coordinate System1*** in the *Coordinate systems* list box.

     This highlighted entity will be displayed in the *Selected entity* list box.

7. Click *OK* to apply the changes and close the *Machine* dialog box.
8. In the *CAMWorks Feature Tree*, under the *Configurations* item, double-click *Layout 2* to display it in the graphics area. Click *Yes* within the *Warning Message* dialog box displayed.
9. Double-click on the ***Machine item*** (***Machine [Mill - metric]*** in this case) to open the *Machine* dialog box.
10. Click on the *Setup* tab. This tab allows you to set the Fixture Coordinate System for the machine.
11. In the Fixture Coordinate System group box, highlight ***Coordinate System1*** in the *Coordinate systems* list box. This highlighted entity will be displayed in the ***Selected entity*** list box.
12. Click *OK* to apply the changes and close the *Machine* dialog box.



Setup tab of Machine Dialog Box

Step 3: Verifying the Addition of Parts in the CAMWorks Part Manager

With the new functionality to link CAMWorks Nesting and CAMWorks, for each Nested layout, the nested parts will be automatically added to the *CAMWorks Part Manager*.

In this step, you will verify the automatic addition of parts to the *CAMWorks Part Manager* for each nested layout.

1. In the SOLIDWORKS left hand side panel, click on the *CAMWorks Feature Tree* tab.
2. Within this tab, click on the (⊞) plus sign to expand the *Configurations* item. Observe that the two nested layout assemblies generated are listed under *Configurations*. (Note that if only one nested layout is generated after the nesting process, then it will not be listed under *Configurations*.)
3. Expand the *Part Manager* item (if not already expanded) by clicking on the (⊞) plus sign to its left. Under *Part Manager*:
   - The part name (*Tutorial_13_a.sldprt)* is listed under the *Part Manager*.

- A *Feature Manager*, which is created for each part, is used to define the Mill Part Setups and machinable features associated to the seed part.
- For each unique part, all the instances are listed under the *Instances* item. Observe that all 12 instances of the part in the nested layout assembly are listed.
- When you highlight an instance of the part listed under *Instances*, the corresponding part will be highlighted in the graphics area.
- To delete an instance of the part, highlight the instance of the part under *Instances* and press the *Delete* button.

4. Under the *Configurations* item, double-click *Layout 2*. Click *Yes* within the Warning Message dialog box displayed. The graphics area will now display the second nested layout assembly.

   (Alternatively, you can change the nested layout displayed in the graphics area by using the *SOLIDWORKS ConfigurationsManager* tab.)
5. Once again, expand the items listed under *Part Manager* and observe that all the instances of the part (*Tutorial_13_b.sldprt*) have been listed under *Part Manager*.

Step 4: Automatic Stock Definition

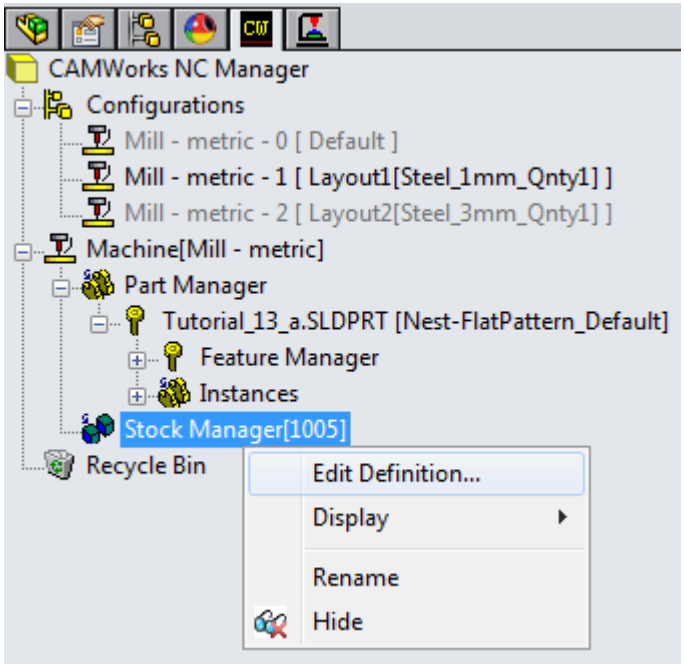
With the new functionality to link CAMWorks Nesting and CAMWorks, for each Nested layout, the stock definition will be automatically loaded in the *CAMWorks Stock Manager*.

The *Stock Manager* allows you to customize the stock associated with the parts. In this step, you will verify the stock definition that was automatically defined in the *CAMWorks Stock Manager*.

1. In the *CAMWorks Feature Tree*, under the *Configurations* item, double-click *Layout 1* to display it in the graphics area. Click *Yes* within the *Warning Message* dialog box displayed.
2. Double-click on the *Stock Manager* item in the CAMWorks Feature tree.

   OR

   Right click *Stock Manager* item in the CAMWorks Feature tree and select *Edit Definition* on the context menu.
3. The *Stock Manager* dialog box is displayed. Observe that:
   - In the *Stock Type* group box, the selected Stock Type is



Command to open the Stock Manager dialog box

Extruded Sketch.

- In the *Extruded Sketch* group box, the *Depth* of the stock is set to **1mm**. (This is equal to the thickness of the parts being nested).
- In the *Stock Size* group box, the dimensions of the stock in the X, Y and Z directions are indicated. The **X** (500 mm) and **Y** (200 mm) values correspond to the dimensions of the sheet used for creating the nested layout. The **Z** value (1mm) is equal to the thickness of the parts being nested
- In the *Create Stock* group box, the Stock form selected is ***Common*** Stock. (This selection indicates that all the parts in the nested layout assembly will be machined from a common stock.)

'Common' is selected in the Create Stock group box

4. Click *Cancel* to close the *Stock Manager* dialog box.
5. If required, you can similarly check the stock definition for the *Layout2*.

Step 5: Defining Machinable Features and Interactively Inserting Features

Extracting Machinable Features for Layout 1:

The next step is to interactively insert Boss features after extracting the machinable features using the ***Automatic Feature Recognition (AFR)*** technology available in CAMWorks. Machinable Features thus recognized are added under the *Feature Manager* item of the *CAMWorks Feature Tree*.

At the Mill Part Setup level, features can be inserted interactively using the ***New 2.5 Axis Feature*** or ***New Multi Surface Feature*** commands. CAMWorks automatically copies the features to every other instance of the part selected in the *Part Manager*.

When machining multiple instances of the same part, if you only want to create one instance of the feature, you can use the Assembly Feature command on the feature context menu to declare the feature an ***Assembly Feature***. By doing so, CAMWorks will not copy the feature to all instances of the part.

In this tutorial, you will discard the machinable features that were extracted automatically by executing the *Extract Machinable Features* command since the machinable features recognized are not suitable for sheet metal machining. Instead, you will interactively insert the Boss Features that define the perimeter of the part.

Following are the step to recognize features automatically:

1. Ensure that *Layout 1* is displayed in the graphics area.

2. Click the *Extract Machinable Features* button on *the CAMWorks Command Manager*.

   OR

   In the SOLIDWORKS menu bar, click on the *CAMWorks* menu and select *Extract Machinable Features* command.

   OR

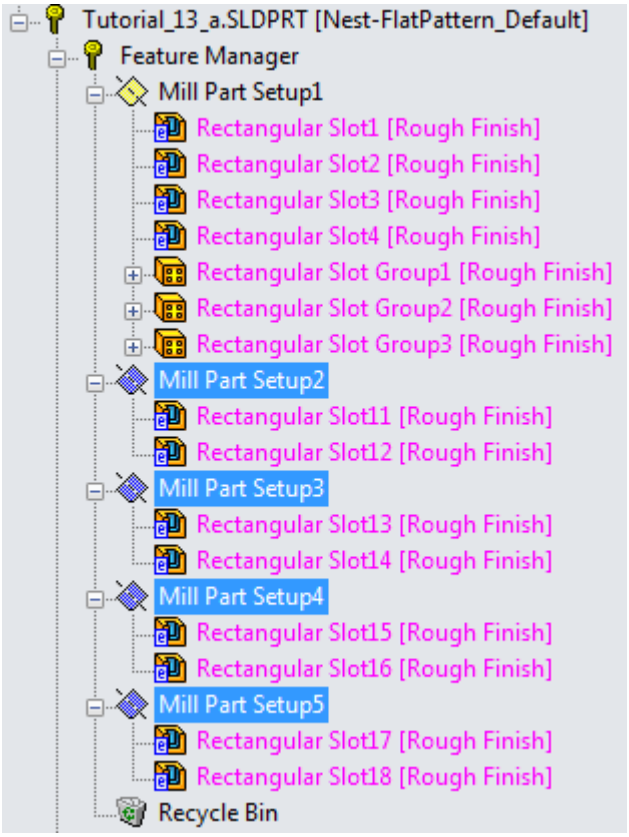
   Right click *CAMWorks NC Manager* in the CAMWorks Feature tree and select *Extract Machinable Features* on the context menu.
3. The *Message Window* is displayed. This window is displayed automatically to report the progress of the current process. Close this message window.
4. On execution of the *Extract Machinable Features* command, CAMWorks generates the Mill Part Setup and the machinable features. The items are displayed in the *CAMWorks Feature tree*.
5. Click the plus sign next to the *Feature Manager* in the Feature tree.

   The Feature Manager displays all the Mill Part Setups and machinable features that were created by *Automatic Feature Recognition*.

Deleting the Mill Part Setups and Features from Layout 1:

In this tutorial, you will have to delete Mill Part Setups and features and then insert the Boss Feature interactively.

1. Press the *Ctrl key* on the keyboard and left-click on *Mill Part Setup2, Mill Part Setup3, Mill Part Setup4* and *Mill Part Setup5* in the Feature Manager to highlight the items.
2. Right click on the *Mill Part Setup5* and select *Delete* on the context menu.
3. CAMWorks will display a warning message asking whether you are sure about deleting the Mill Part Setups and all dependent items. Click *Yes* to confirm to deletion.
4. The features are moved to the Recycle Bin. When a feature is deleted, it is automatically placed in the Recycle Bin, which is used to store machinable features that you do not intend to machine.

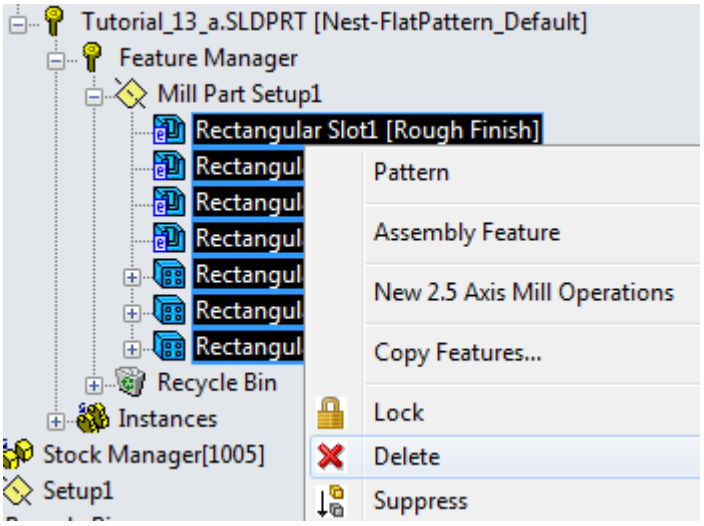


Selecting the Mill Part Setups to delete it

5. Click the (⊟) minus sign to the left of the Recycle Bin to collapse it.
6. Similarly, you will delete all the features extracted in the *Mill Part Setup1*.

7. Under *Mill Part Setup1*, click on the first feature *(i.e. Rectangular Slot 1)* and hold down the *Shift key* on the keyboard and then click on the last feature *(i.e. Rectangular Slot Group3)* to highlight all the items.
8. Right click and then select *Delete* on the context menu. Click *Yes* to confirm the deletion.

   The features are moved to the Recycle Bin.
9. Right click on the *Recycle Bin* under the Feature Manager and select the *Empty* on the context menu.



Deleting all the features from Mill Part Setup1

10. CAMWorks will display a warning message asking whether you are sure about emptying the Recycle Bin. Click *Yes* to confirm the process.

Interactively inserting Boss Feature in Layout 1:

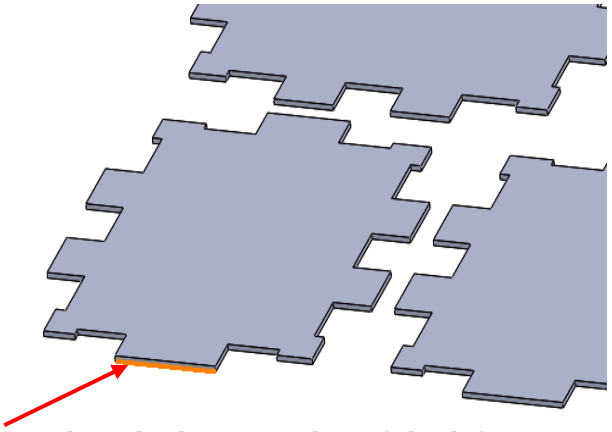
After deleting the unwanted features and Mill part Setups, you will now interactively insert the Boss Feature by using the *New 2.5 Axis Feature* command.

Following are the steps to insert the boss feature interactively:

1. Right click on the *Mill Part Setup1* in the *CAMWorks Feature tree* and select the *New 2.5 Axis Feature* command on the context menu.

   The *2.5 Axis Feature Wizard: Feature & Cross Section Definition* dialog box is displayed.
2. Select *Boss* as the Feature type from the dropdown list.
3. Highlight in the *Entities selected* field to set the focus.
4. Select the lower edge of the left corner part in the graphics area as shown in the image on right (*highlighted in orange*).

   This action will display the *Loop <1>* in the field of *Entities selected*.
5. Click *Next* to display the *2.5 Axis Feature Wizard: End Conditions* dialog box.



Select the bottom edge of the left corner part

6. Set the depth to **1mm**. (This is equal to the thickness of the parts being nested).
7. If required, remove the check from the check box next to the *Reverse direction* option in order to correct the direction.

8. Click *Finish* to insert the Irregular Boss feature under the *Mill Part Setup1*.
9. Click *Close* to exit the *2.5 Axis Feature Wizard* dialog box.

   Observe that the interactively inserted Boss feature is listed under *Setup1* at the bottom of the CAMWorks Feature Tree.

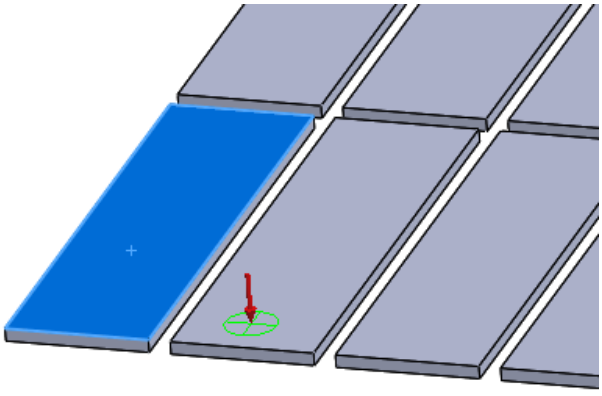
Interactively Inserting Mill Part Setup and Machinable Feature For Layout 2:

For Layout 2 in this tutorial, you will insert both the Mill Part Setup and Boss Feature interactively.

Following are the steps to interactively insert Mill Part Setup and machinable feature:

1. In the *CAMWorks Feature Tree*, under the *Configurations* item, double-click *Layout 2* to display it in the graphics area. Click *Yes* within the *Warning Message* dialog box displayed.
2. Right-click on the *Feature Manager* under the *Tutorial_13_b.sldprt* of CAMWorks Feature tree and select *New Mill part Setup* on the context menu.

   The *Mill Part Setup* dialog box is displayed.
3. Click on the left corner side part in the graphics area as shown in the image on right (*highlighted in blue*).

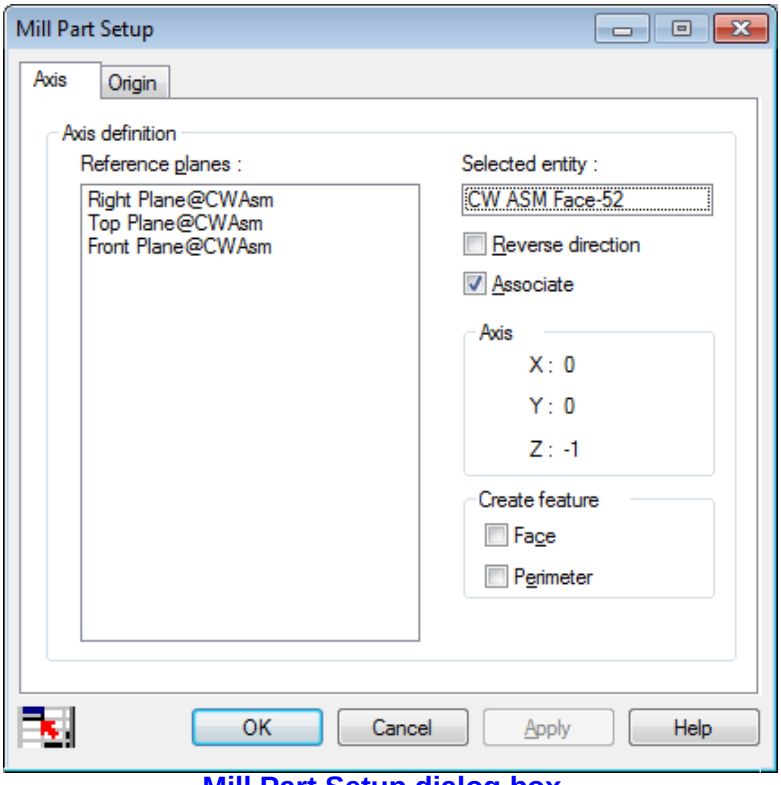


Select left corner part highlighted in blue

4. Make sure the direction (*indicated by red arrow*) is correct on the feature. If not, place a check in the check box next to the *Reverse direction* option.
5. Click *OK* to insert the setup and close the dialog box.

   Mill Part Setup1 is now listed under Feature Manager in the tree.
6. Right click on the *Mill Part Setup1* in the CAMWorks Feature tree and select the *New 2.5 Axis Feature* command on the context menu.

   The *2.5 Axis Feature Wizard: Feature & Cross Section Definition* dialog box is displayed.
7. Select the Feature type as *Boss* from the dropdown list.
8. Click within the *Entities selected*



Mill Part Setup dialog box

field to set the focus and select the left corner part in the graphics area.

This action will display the *Face <1>* in the field of Entities selected.

9. Click *Next* button to display the *2.5 Axis Feature Wizard: End Conditions* dialog box.
10. Set the depth to **3mm**. (This is equal to the thickness of the parts being nested).
11. If required, remove the check from the check box next to the *Reverse direction* option to correct the direction of the defined feature.
12. Click *Finish* to insert the Rectangular Boss feature under the *Mill Part Setup1*.
13. Click *Close* to exit the *2.5 Axis Feature Wizard* dialog box.

Step 6: Sorting Part Instances to Determine Machining Order

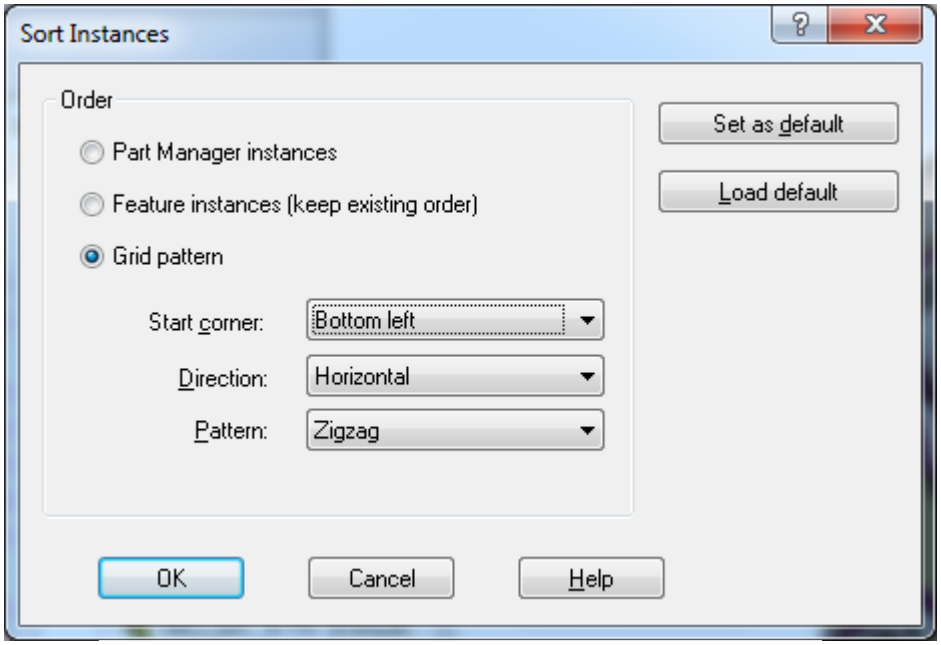
When part instances are automatically added or manually added using the *Add All Instances* button, the instances need not necessarily be listed in the best machining order. CAMWorks provides options for sorting part instances to be processed in a more efficient order.

The order in which the part instances are listed under the feature is the machining order for that feature. By default, the parts for all features are in the order they appear in the *Part Manager*. You can change the order globally for all features or for individual features.

Following are the steps to sort instances:

1. Ensure that *Layout 1* is displayed in the graphics area.
2. Under *Setup1* in the CAMWorks Feature tree, click on the (⊞) plus sign next to the feature to expand the feature items.
3. Double click *Part Manager* in the CAMWorks Feature tree.
4. Click the *Sort Instances* button in the Manage Parts dialog box.

   The *Sort Instances* dialog box is displayed.
5. Select the *Grid pattern* option.
6. Set the following options for Grid Pattern and then click the *OK* button.
   - Start corner = *Bottom left*
   - Direction = *Horizontal*
   - Pattern = *Zigzag*
7. Click *OK* to close the *Manage Parts* dialog box.
8. Click the (⊞) plus sign next to any feature listed under *Setup1* and click each part instance to view the machining order of the features in the graphics area.
9. In the *CAMWorks Feature Tree*, under the *Configurations* item, double-click *Layout 2* to display it in the graphics area. Click *Yes* within the *Warning Message* dialog box displayed.
10. Repeat the same steps from **2 to 8** for *Layout 2.*



Sort Instances Dialog Box

Step 7: Generating the Operation Plan and Adjusting Operation Parameters

When Generate Operation Plan command is executed, operations for machinable feature are created automatically based on information in the CAMWorks Technology Database.

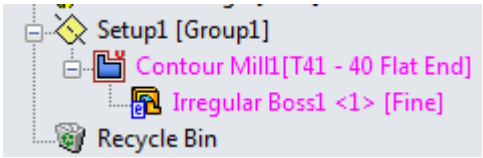
Generating Operation Plan for Layout 1:

1. Ensure that *Layout 1* is displayed in the graphics area.
2. Click the *Generate Operation Plan* button on the CAMWorks Command Manager. OR

   Right click the *CAMWorks NC Manager* of the Feature tree and select *Generate Operation Plan* on the context menu.

   In the Operation tree, the generated operation *Contour Mill1* is displayed under *Setup1*.

   Contour Mill1 operation is used for the Irregular Boss feature of the part.



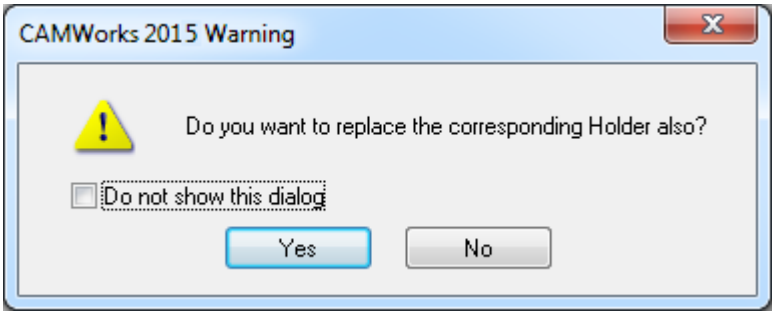
Generated Operation listed in Operation Tree

3. In the Operation tree, double click *Contour Mill1*. OR

   Right click *Contour Mill1* and select *Edit Definition* on the context menu.

   The Operation Parameters dialog will be displayed.
4. Under *Tool* tab, click on the *Mill tool* page. Observe that a tool of 40mm diameter is used to machine the Irregular boss feature. This irregular boss feature represents the perimeter of the part. Since the *Part to part distance* and *Part to Sheet distance* assigned before creating the nested layout was 5mm, selecting any tool with more than 5mm diameter will gouge the part. Hence, the tool used to machine this operation needs to be changed to a tool with a diameter 5mm or less.

5. Click on the *Tool* tab and select the *Tool Crib* page.
6. Highlight the Flat End tool with diameter of **3mm** within the list of displayed tools.
7. Click the *Select* button. This action will assign the highlighted tool as the tool to be used for machining this operation.
8. CAMWorks will display a warning message which prompts you to select whether the corresponding holder of the tool is also to be changed. Click *Yes* to replace the corresponding holder.



CAMWorks Warning Message

9. Click *OK* to apply the changes and close the Operation Parameters dialog box.

Generating Operation Plan for Layout 2:

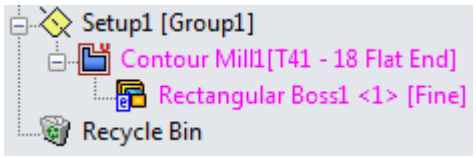
1. In the *CAMWorks Feature Tree*, under the *Configurations* item, double-click *Layout 2* to display it in the graphics area. Click *Yes* within the *Warning Message* dialog box displayed.
2. Click the *Generate Operation Plan* button on the CAMWorks Command Manager. OR

   Right click the *CAMWorks NC Manager* of the Feature tree and select *Generate Operation Plan* on the context menu.

   In the Operation tree, the generated *Contour Mill1* operation is listed under *Setup1*. *Contour Mill1* operation is used for the Rectangular Boss feature of the part. For this operation too, the tool used to machine the operation needs to be replaced with a tool of 5mm or less.
3. Double click *Contour Mill1* in the Operation tree. OR

   Right click *Contour Mill1* in the Operation tree and select *Edit Definition* on the context menu.

   The Operation Parameters dialog box is displayed.
4. Click on the *Tool* tab and select the *Tool Crib* page.
5. Highlight the Flat End tool with diameter 5mm or less. For this tutorial, select the **4mm** diameter Flat End mill within the list for this operation and then click the *Select* button.
6. CAMWorks will display a Warning message. Click *Yes* to replace the corresponding holder.
7. Click *OK* to apply the changes and close the Operation Parameters dialog box.



Generated Operation listed in Operation Tree

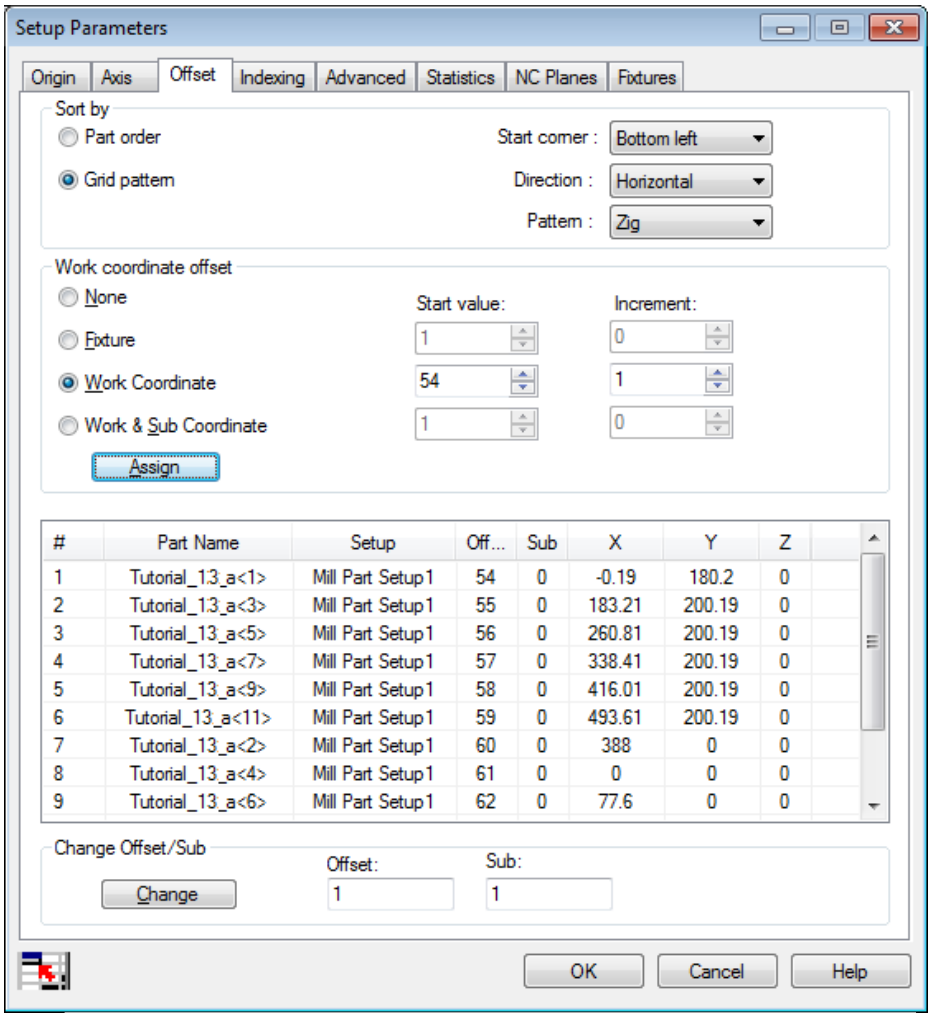
Step 8: Defining G-code Program Zero Location

1. Ensure that *Layout 1* is displayed in the graphics area.
2. Double click *Setup1* in the Operation tree.

   The *Setup Parameters* dialog box is displayed.
3. On the *Origin* tab, make sure *Part Setup origin* is selected for the Output origin.

   Note that when *Setup origin* is selected, you can specify the origin using several methods.
4. Click on the *Offset* tab.
5. In the *Sort by* group box, select *Grid pattern*.

   When you pick this option, the parts in the table are automatically reordered based on the current settings for *Start corner*, *Direction* and *Pattern*.
6. Set the Grid pattern parameters to the same settings you used when sorting part instances for the machining order:
   - Start corner= *Bottom left*
   - Direction= *Horizontal*
   - Pattern= *Zigzag*
7. Set the Work coordinate offset to *Work Coordinate*. This option will output G54, G55, etc.
8. Set the *Start* value to **54** and the Increment to **1**.
9. Click the *Assign* button of the Work Coordinate offset group box. Observe that the numbers are updated in the *Offset* and *Sub* columns in the table.
10. Click *OK* to close the Setup Parameters dialog box.
11. In the *CAMWorks Operation Tree*, under the *Configurations* item, double-click *Layout 2* to display it in the graphics area. Click *Yes* within the *Warning Message* dialog box displayed.
12. Repeat the same steps from **2 to 10** for *Layout 2.*



Setup Parameters Dialog Box (For Layout 1)

Step 9: Generating Toolpaths

CAMWorks calculates toolpaths using the operation parameters to define how to machine each machinable feature.

1. Ensure that *Layout 1* is displayed in the graphics area.
2. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click *Setup1* in the Operation tree and select *Generate Toolpath* on the context menu.

   On executing the *Generate Toolpath* command, the toolpath is generated for all the operations in the Setup.
3. Under the *Configurations* item, double-click *Layout 2* to display it in the graphics area. Click *Yes* within the *Warning Message* dialog box displayed.
4. Click the *Generate Toolpath* button on the CAMWorks Command Manager to generate the toolpath.

Step 10: Simulate Toolpaths

CAMWorks provides the ability to simulate the toolpaths showing the tool movement and the resulting shape of the part/assembly on machining the stock.

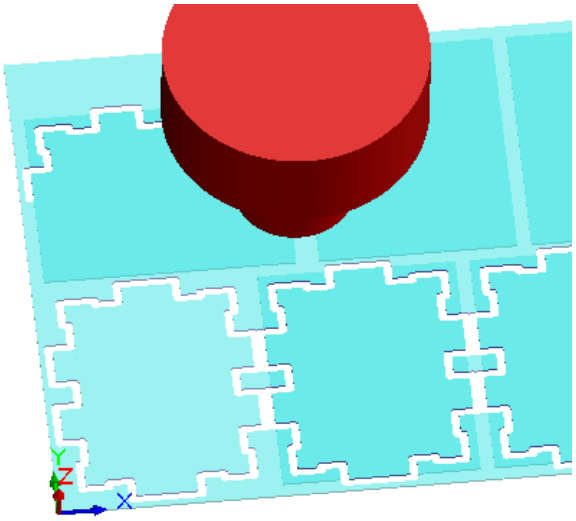
1. Ensure that *Layout 1* is displayed in the graphics area.
2. Click *the Simulate Toolpath* button on the CAMWorks Command Manager.

   OR

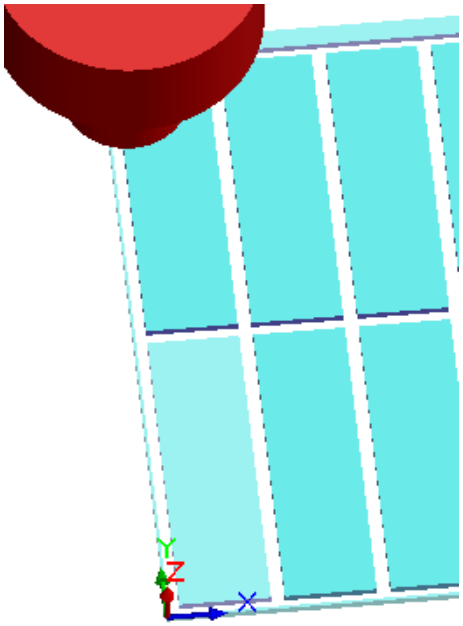
   Right click on *Setup1* in the operation tree and select *Simulate Toolpath* on the context menu.

   The *Toolpath Simulation* toolbar is displayed.
3. Set the following display options:
   - Stock: *Translucent display*
   - Tool: *Shaded display*
   - Tool Holder: *Shaded display*
4. Click the *Run* button.

   The simulation is run with the tool and holder displayed during simulation.
5. Click the *Close* button to exit the simulation mode and return to the SOLIDWORKS display.
6. To view the toolpath simulation for *Layout 2*, switch the display to *Layout2* in the graphics area using the *SOLIDWORKS Configuration Manager* or *Configurations* item in the *CAMWorks Feature Tree* and follow the same steps.



Toolpath Simulation for Layout 1



Toolpath Simulation for Layout 2

Step 11: Generate the NC code

Following are the steps to generate the NC program. Note that NC code needs to be generated separately for *Layout1* and *Layout2*.

1. Click the *Post Process* button on the CAMWorks Command Manager.

   OR

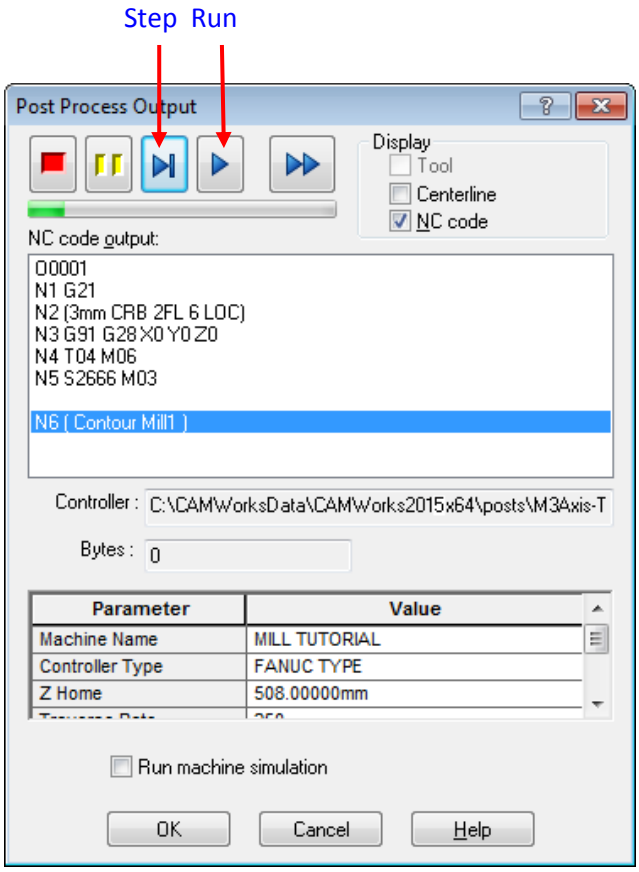
   Right click on the *CAMWorks NC Manager* in the Operation tree and select *Post Process* on the context menu.

   The *Post Output File* dialog box is displayed so that you can save the NC program file.

2. By default, NC files are stored into the folder that contained the last part model or assembly that was opened in SOLIDWORKS. If you want to save these files in another location, you can change the folder location.
3. In the *Post Output File* dialog box, type the suitable file name, and then click *Save* button.

   The *Post Process Output* dialog box is displayed.

4. Click the *Step* button on the control bar at the top of the dialog box.



Post Process Output dialog box

5. CAMWorks begins to generate the NC program and the first line of NC code displays in the NC code output view box.
6. Click the *Step* button again. The next line of NC code is displayed.
7. Click the *Run* button. Post processing continues until it is completed.

   When the post processing is finished, view the code using the vertical scroll bar.

8. Click *OK* to close the dialog box.
9. Repeat the steps 1 to 8 for *Layout2* in order to generate NC code for it.