The Complete Reference Guide

RhinoCAM-NEST 2026

Published: March 2026

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Quick Start

NEST Module 2026

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

Quick Start Guides for each RhinoCAM module are available in both PDF and Video format. Refer to the following information to access these guides:

How to Access the Quick Start Guide Documents

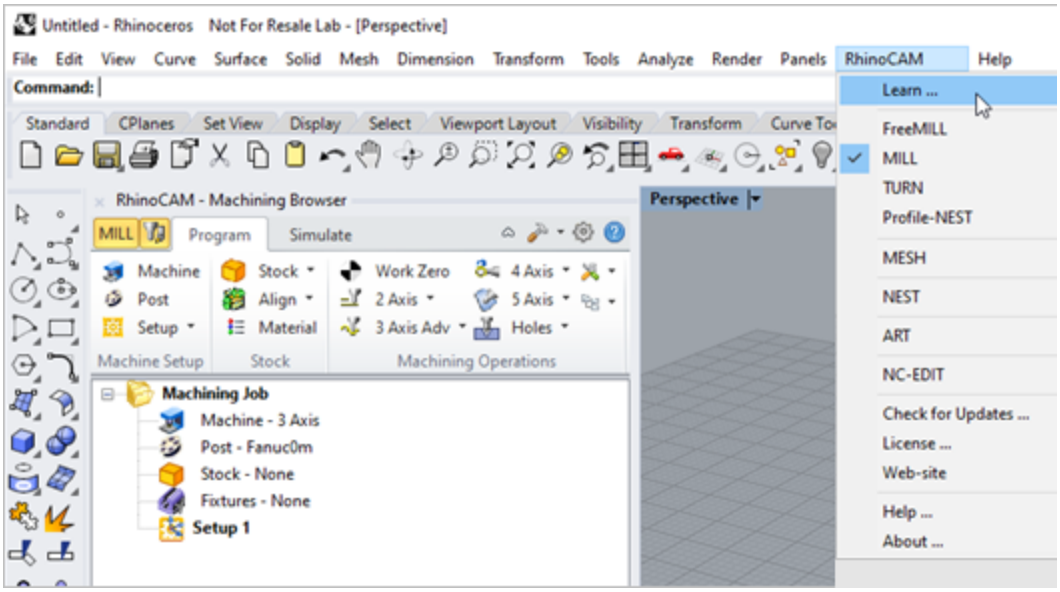
To help you quickly get started in working with each module, select one of the Help buttons located on the RhinoCAM Learning Resources dialog.

You will find:

- Quick Start Guides
- What's New documents
- Online Help links

The Quick Start Guides will help you step through an example tutorial which will illustrate how to use the module. To access the Learning Resources dialog:

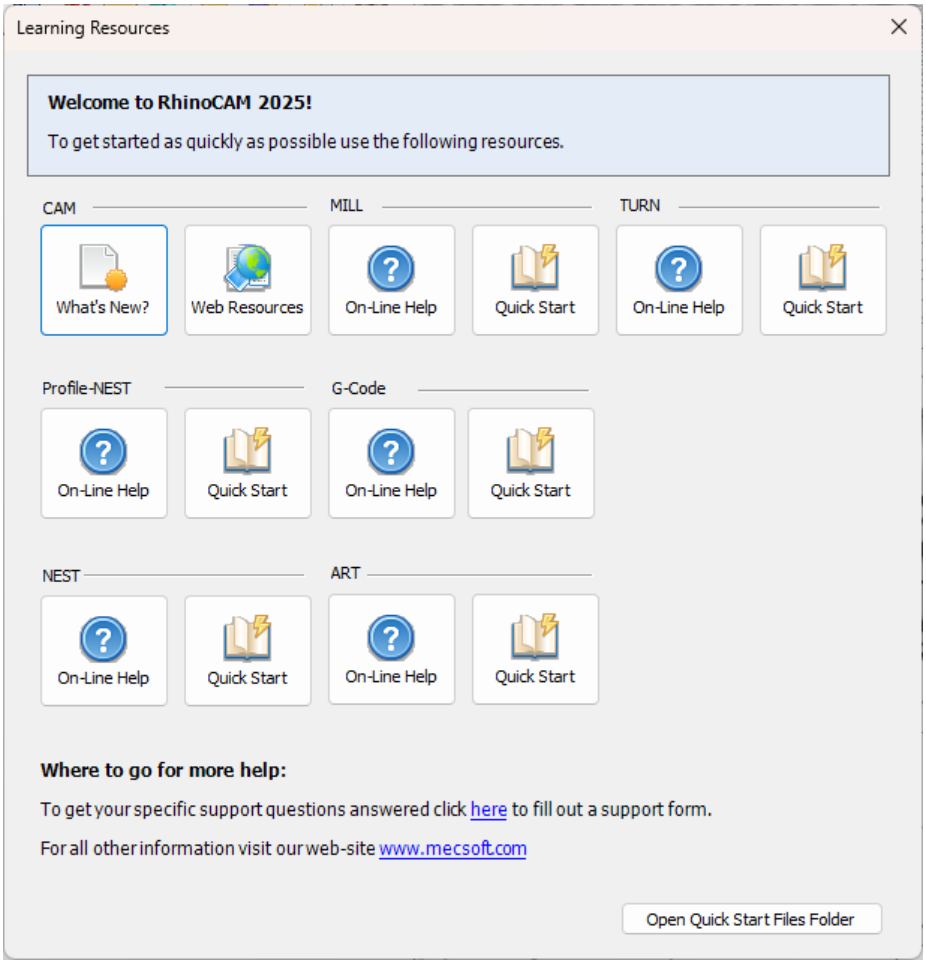
1. From the Rhino Main Menu, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in RhinoCAM

2. Select a document from the Learning Resources dialog to get started using the module of your choice.

You can also select the Open Quick Start Files Folder button located at the bottom of the dialog to open the Quick Start folder where the source files (start and completed versions) are located.



Learning Resources Dialog

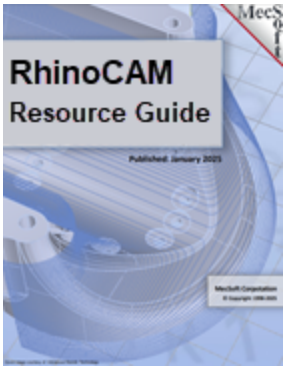
Related Topics

Find More Resources

Resource Guide

Download this PDF Guide for a list of the available RhinoCAM Resources.

2025 RhinoCAM Resource Guide



The 2026 RhinoCAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

RhinoCAM-NEST

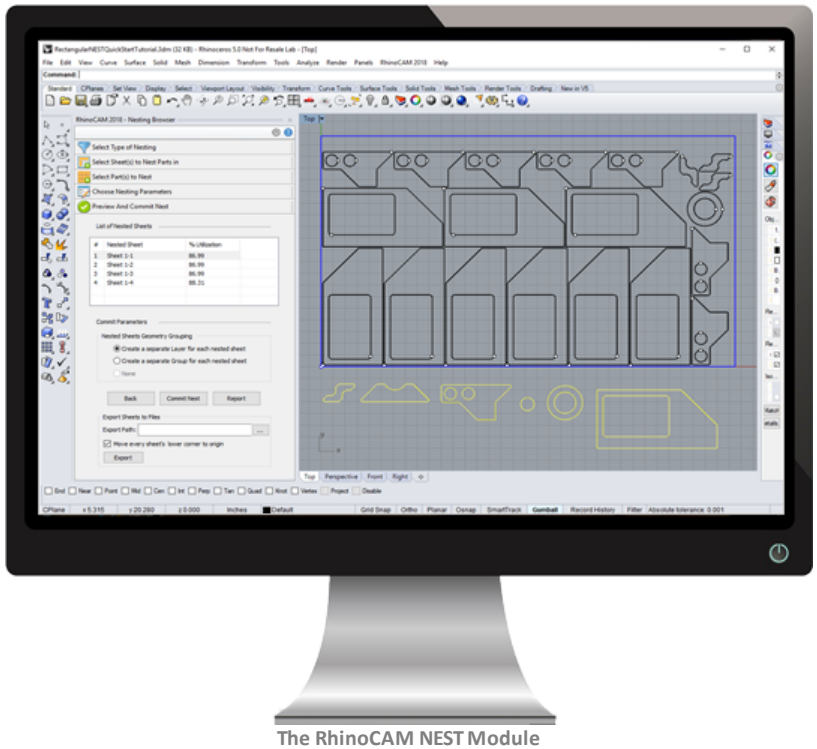
NEST Module 2026

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

Welcome to the on-line help system for the RhinoCAM NEST module included with your MecSoft CAM software. Nesting refers to the process of positioning and fitting 2D or 3D part shapes within flat raw material for the purpose of minimizing material scrap and to achieving efficient machining. In order to minimize the amount of scrap produced by this process, companies use nesting software. For purposes of brevity, RhinoCAM will be referred to as NEST in all subsequent references. Also, Rhino refers to both Rhinoceros 7 or Rhino 8.

Nesting is an add-on module for RhinoCAM that can nest parts by considering either their true shape or a rectangular form around each part shape. This product offers a simple and easy to use wizard driven interface to nest parts in preparation for CAM programming with RhinoCAM.



The RhinoCAM NEST Module

Related Topics

3.1 What are Sheets and Parts?

Sheets and parts can be created using CAD tools in Rhinoceros or can be imported as Dxf, Dwg, Rhino, IGES.

Sheets

Sheets are the 2D, closed geometric shapes that represent the raw stock material. These sheets could be rectangular or any arbitrary shape, but may not contain holes within their periphery.

Parts

This refers to shapes to be produced that are cut from the sheets. They are selected as 2D, closed shapes or 3D solids. Parts may have cutouts within their periphery.

Related Topics

What is Nesting?

3.2 What are the Nesting Types?

Here are descriptions of the two nesting types supported.

True Shape Nesting

The Nesting module uses the true shape of each part to fit the parts onto the sheet(s). This type of nesting supports rectangular and arbitrary shapes for the sheets, but does not support holes within any sheet.

Rectangular Nesting

The Nesting module determines a minimum rectangle around each part to be nested, then fits those rectangles within the sheet(s). This type of nesting supports only rectangular sheets.

Related Topics

What is Nesting?

Preparing Sheets & Parts for Nesting

Prior to creating a nest of some parts on one or more sheets, the 2D shapes of those sheets and parts should be brought together on the CAD display. This is referred to as staging. Below are some tips for staging the sheets and the parts.

Tips for Staging the Sheet(s)

- Shape: These 2D shapes should represent full-scale stock material, either full sheets or remnants. For True Shape Nesting, the 2D shapes can be irregular. For Rectangular Nesting, the 2D shapes should be rectangular.
- Placement: For better viewing, multiple sheets should not overlap or be placed on top of each other in the CAD display.
- Orientation: The sheets should be oriented with respect to the X,Y-axes of the CAD system as they would be oriented on the CNC machine (if applicable) and for Grain Control purposes (if applicable).
- Multiple Sheet Shapes: Nesting may be performed onto multiple dissimilar sheet shapes, if desired. Each of these dissimilar sheet shapes should be represented in the CAD display for the nesting process. But if nesting is to be done on multiple sheets that are identical, such as typical stock sizes, then only one 2D CAD shape for that sheet size needs to be represented in the CAD display. A 'Count' parameter will allow multiples of any of the sheets.

Tips for Staging the Part(s)

- Placement: The 2D or 3D shapes that represent the parts should be placed or staged outside of the sheets. The nesting process will fit the parts inside of the sheets. Where the parts are located or staged around the sheets is not important.

  Part shapes should not be staged within the cutouts or holes of other staged parts. Each part should be staged alongside other parts.
- Orientation: As staged, the part shapes should be oriented with respect to the X,Y axes of the CAD system primarily for the purpose of Grain Control (if applicable).

Related Topics

What is Nesting?

What are Sheets and Parts?

What are the Nesting Types?

How to use Nesting

Getting Started with the Nesting Browser

How to use Nesting

After successfully installing Rhinoceros (6.0 or 7.0) and RhinoCAM, launch RhinoCAM by selecting the Rhinoceros shortcut on your desktop or from the program menu by selecting Start > All Programs > Rhinoceros 6 (or Rhino 7).

Related Topics

What is Nesting?

What are Sheets and Parts?

What are the Nesting Types?

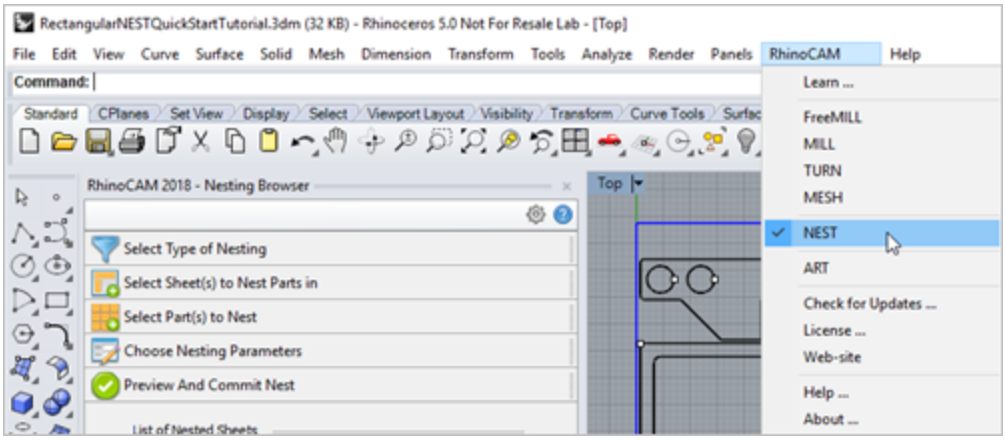
Getting Started with the Nesting Browser

Getting Started with the Nesting Browser

From the Rhino Main Menu Bar, select RhinoCAM and then NEST from the drop down menu..

Accessing NEST

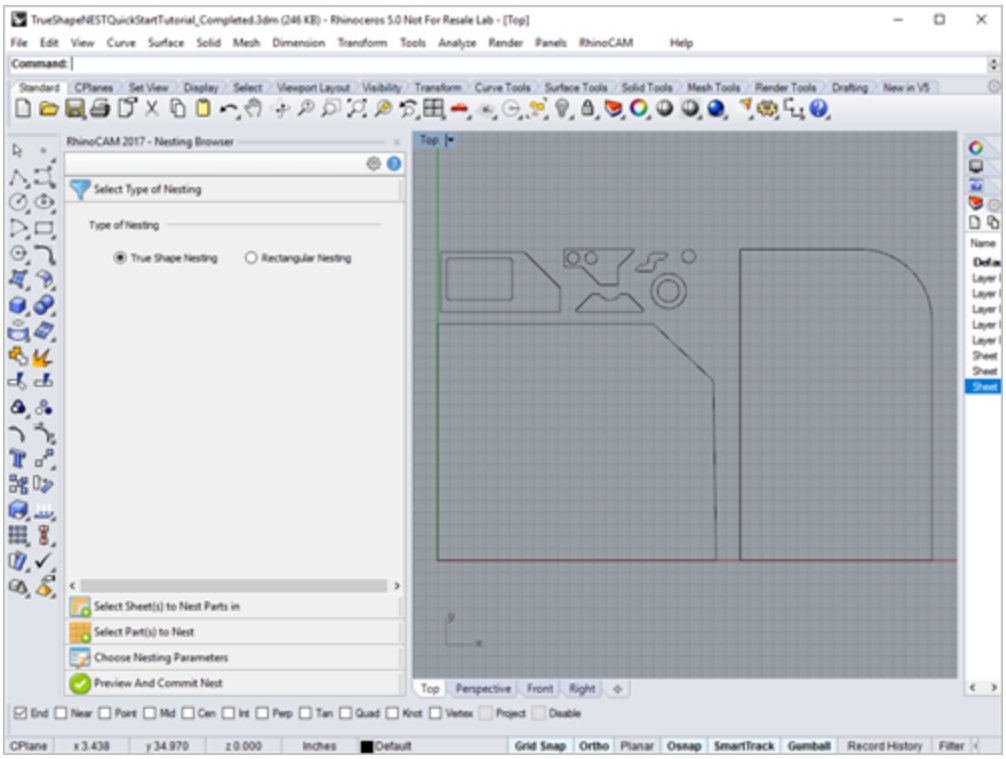
From the drop down menu bar select VisualCAM and then NEST.



From the drop down menu bar select RhinoCAM > Nesting Browser

The NEST Browser

Nesting Browser is now loaded and by default appears to the left of view port.



Nesting Browser is now loaded and by default appears to the left of view port.

Related Topics

[What is Nesting?]

[What are Sheets and Parts?]

[What are the Nesting Types?]

[How to use Nesting]

Nesting Browser Work Flow

Create parts to nest and sheet(s) in Rhinoceros using the CAD tools located on the geometry bar or Select File > Open from the menu bar and open part file.

The Nesting Browser uses a 5 step process to create 2D nesting of parts.

1. Select Type of Nesting
2. Select Sheet(s) to Nest Parts in
3. Select Part(s) to Nest
4. Choose Nesting Parameters
5. Preview and Commit Nest

These 5 steps are represented by 5 tabs (horizontal bars) stacked in the Nesting Browser. Start at the top tab and progress downward through the 5 steps to complete the nesting process. You can always return to a previous step to change parameters and information as you refine the nesting results.

Related Topics

Select Type of Nesting

Select Sheet(s) to Nest Parts in

Select Part(s) to Nest

Choose Nesting Parameters

Preview and Commit Nest

7.1 Nest Toolbar

The Nest Toolbar is located at the top of the Nesting Browser.

It provides the following icons:

Nesting Toolbar

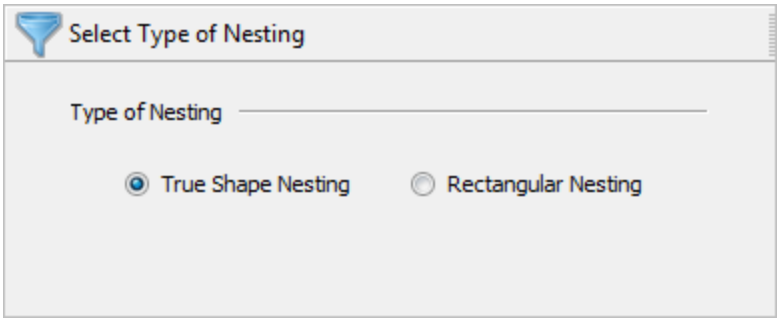
Displays the Nest Preferences dialog.

Displays the Nest Online Help dialog.

7.2 Select Type of Nesting

This tab allows you to choose between True Shape Nesting and Rectangular Nesting. Click on the radio button to choose the nesting type.

Nesting Browser, Select Type of Nesting

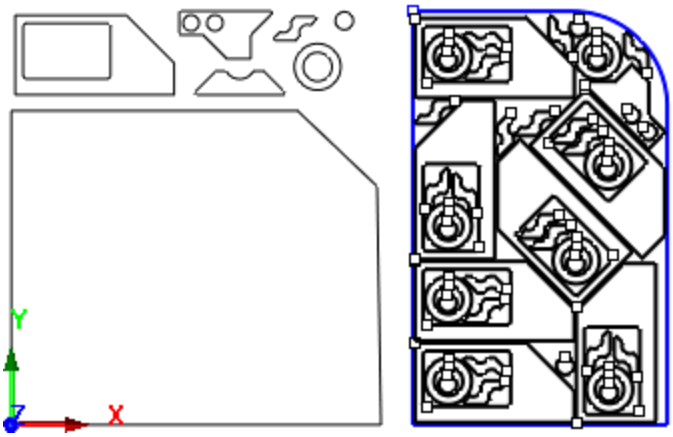


Functionality Comparison Table

Function	True Shape	Rectangular
Sheets		
Allowable stock sheet shape	2D Arbitrary	2D Rectangular
Allows holes in stock sheet	☐	☐
Allows nesting start corner and direction	☑	☐
Sheet thickness allowance	☑	☑
Allows grain direction control for sheet and part	☑	☑
Define sheets by parameters	☑	☑
Parts		
Allows 3D shapes as parts	☑	☑
Uses true shape for part nesting	☑	☐
Uses minimum rectangular shape for part nesting	☐	☑
Allows grain direction control for sheet and part	☑	☑
Allows part rotation by step angle	☑	☐
Allows for mirroring of parts	☑	☐
Allows nesting of part within part	☑	☐
Allows engraving and sign making (coincident regions)	☑	☑
Nest		
Utilization Reports	☑	☑
Minimum utilization control	☑	☑
Accuracy control	☑	☑
Auto Tagging Options	☑	☑
Nest Layout Controls	☑	☑
Nest Preview	☑	☑

True Shape Nesting

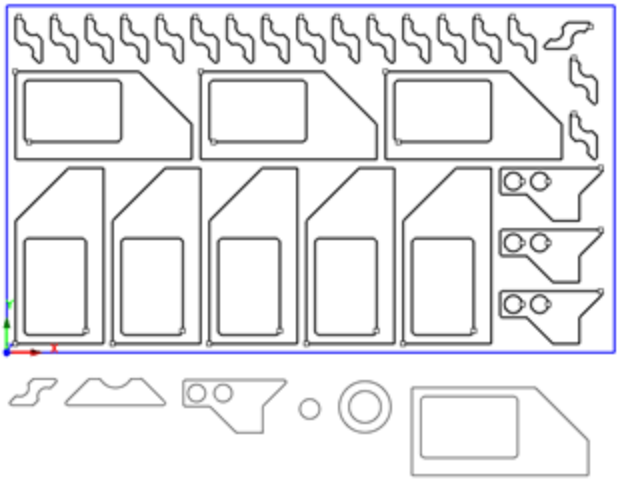
This method is useful when the actual geometric details of the part are taken into consideration while nesting. True shape nesting enables interlocking of parts, recognition of arbitrary shaped sheets.



True Shape Nesting Example

Rectangular Nesting

This method is useful for nesting of parts on rectangular sheets. This is ideal for those nesting mostly square and rectangular profiles. For all part shapes, an imaginary rectangle is drawn around the shape and then the rectangles are laid side-by-side.



Rectangular Nesting Example

Related Topics

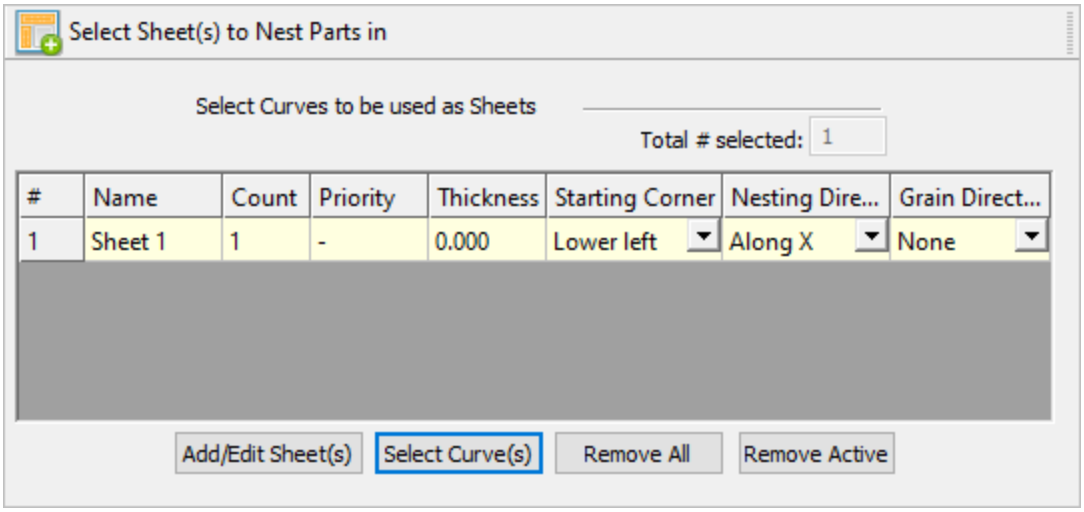
Nesting Browser work flow

7.3 Select Sheet(s) to Nest Parts in

This tab allows the definition of sheet(s) to nest parts. There are 4 buttons on this tab that provides control for the selection of geometry as sheets and for defining sheets from parameters.

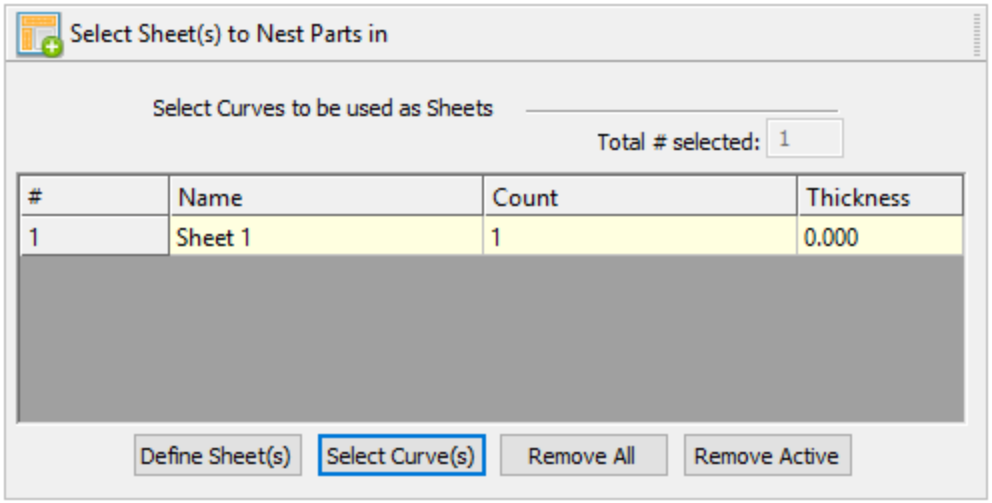
Once the sheet(s) are selected, you can specify sheet Count and Grain Direction for each sheet. For True Shape Nesting, you can also specify Starting Corner and Nesting Direction.

Select Sheet(s) to Nest Parts in tab, True Shape Nesting



Nesting Browser - Select Part(s) to Nest (TrueShape Nesting)

Select Sheet(s) to Nest Parts in tab, Rectangular Nesting



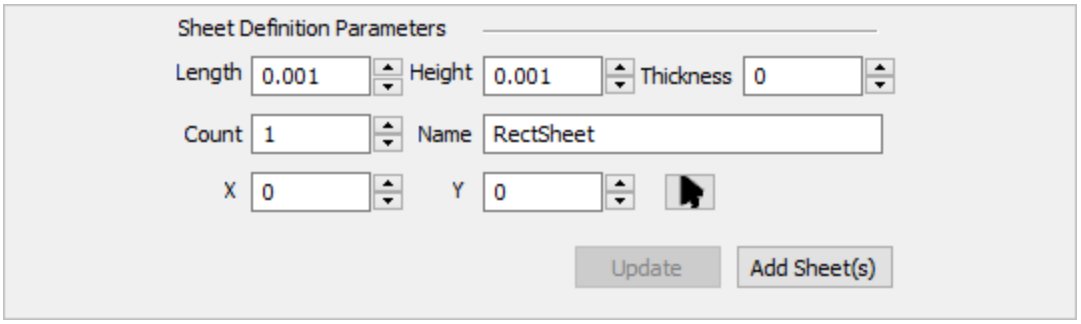
Select Sheet(s) to Nest Parts in tab, Rectangular Nesting

Total # Selected

This reports the total number of sheets that are defined in the table (not the count).

Define Sheet(s)

Select this button to display the Sheet Definition Parameters. This allows you to create a sheet by entering specific parameters. Then select the Add Sheet(s) button to add the sheet to the Sheets List.



- Sheet Thickness: If you plan to nest 2D curves, the sheet thickness MUST be set to zero or you will receive an error message when creating the nest.
- Sheet Thickness: If you plan to nest 3D solids, the sheet thickness must match the thickness of the part(s).
- IMPORTANT: Also note that if your 3D parts vary in thickness, then you must define at least one sheet for each thickness, even if the thicknesses vary by only a very small amount. You can refer to the Thickness Filter located on the Select Part(s) to Nest tab.

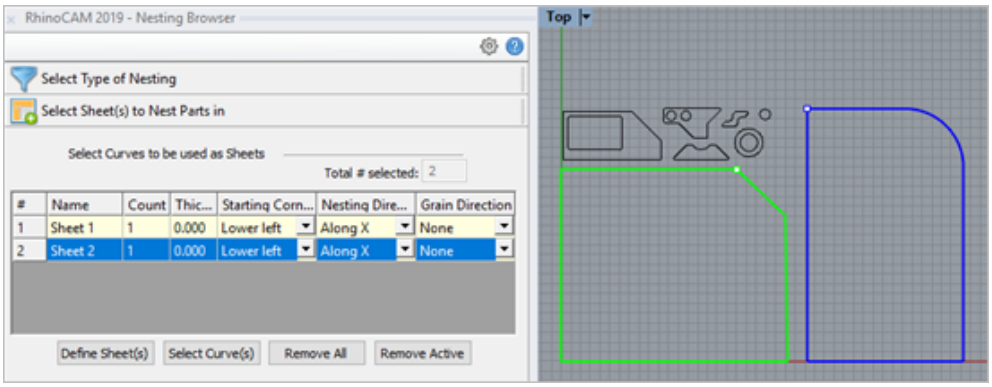
Select Curve(s) for Sheets

To select curves to be used as sheets, click the Select Curve(s) button. Select closed curves for sheets and press enter or click the right mouse button to add curves to the selection list. Open profiles are not supported as this may result in incorrect nests. Sheets are only periphery shapes and cannot contain holes or cutouts.

True shape nesting will use arbitrary shapes for sheets. Rectangular nesting will fit a rectangle around the curve shape that is selected for sheets and will use the rectangle as the sheet for nesting.

Example

Two or more closed curves can be selected as sheets when different sheet sizes are required to nest part. The selected curves are available under Select Sheet(s) to Nest Parts in tab in the nesting browser. The curves selected as sheets are displayed in blue color. Selecting a sheet from the browser shows the selection highlight of the geometry (in green color).



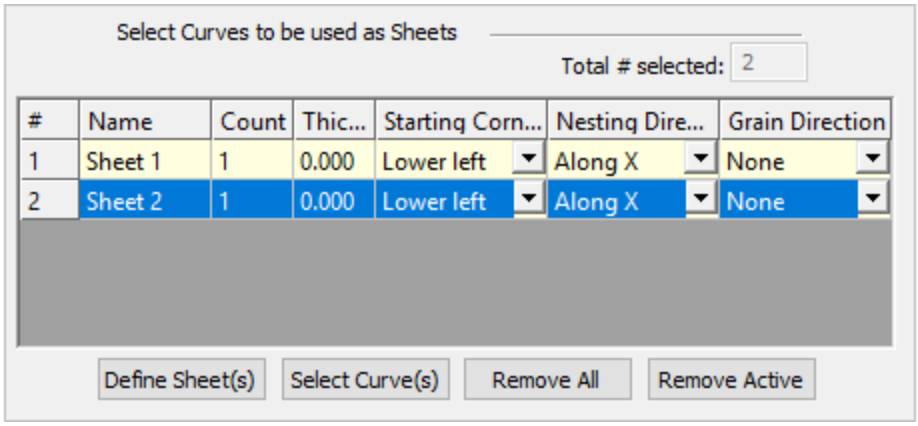
Select Curve(s) for Sheets

Remove All

Selecting Remove All removes all sheets from the selection list.

Remove Active

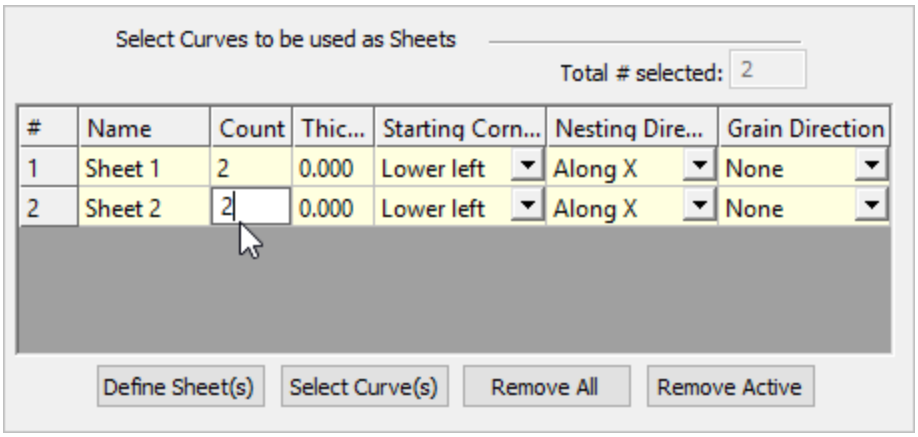
Selecting the Remove Active button removes the highlighted sheet(s) from the selection list. To remove a sheet, select it from the list and click Remove Active.



Remove Active for Sheets

Count for Sheets

Count defines the number of copies for each sheet. Count is set to 2 or more when parts need to be nested to multiple sheets of same size. By default, this value is set = 1.

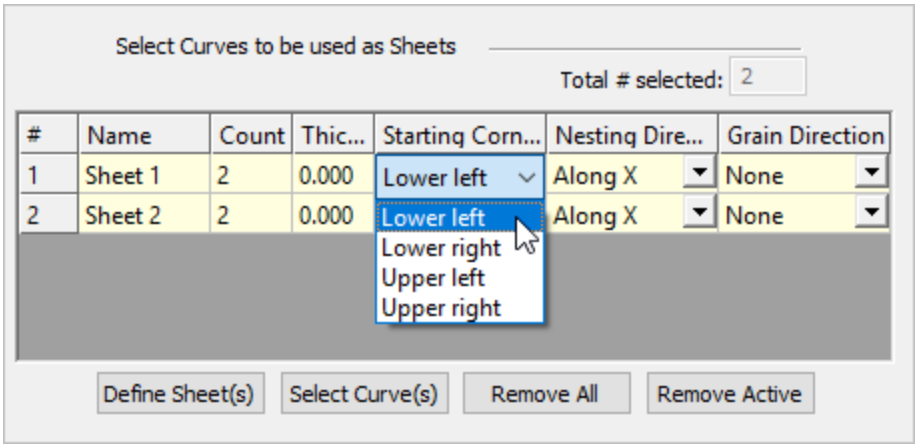


Count for Sheets

In the above example, count for Sheet1 is set = 2.

Starting Corner (True Shape Nesting)

This defines the corner of the sheet to start the nesting. The starting corner can be set to Lower left, Lower right, Upper left, or Upper right. Starting corner can be specified for each sheet.

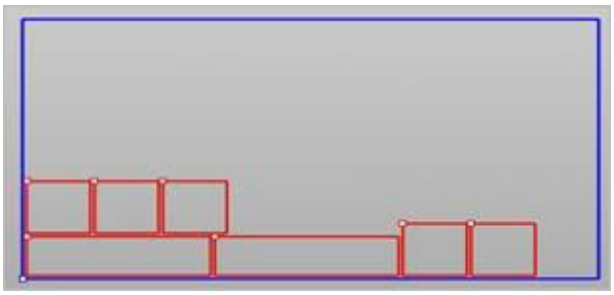


Starting Corner (True Shape Nesting)

By default Lower Left corner is used as starting corner.

Example

Example below shows Sheet Starting Corners for all 4 options.

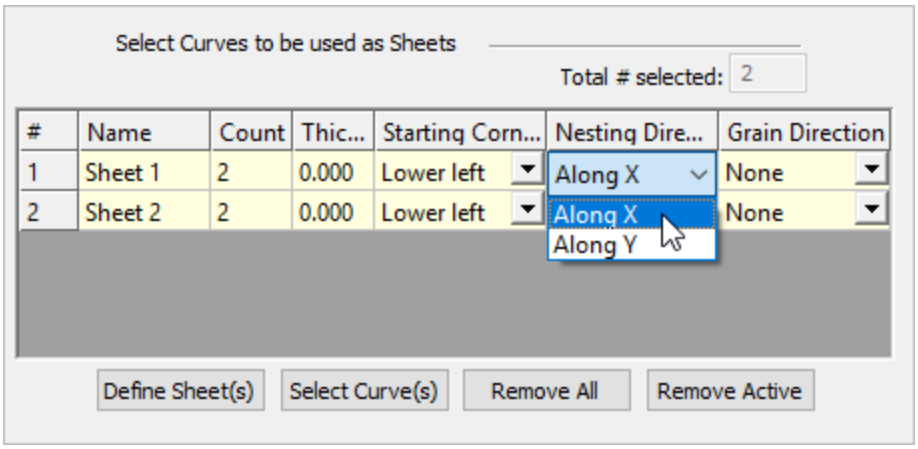


Sheet Starting Corners - Lower Left

Sheet Starting Corners - Lower Right

Nesting Direction (True Shape Nesting)

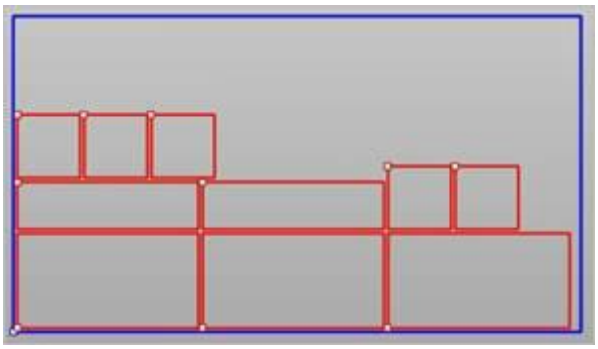
This defines the direction along which the sheet gets filled with parts. This could be either set to Along X or Along Y.



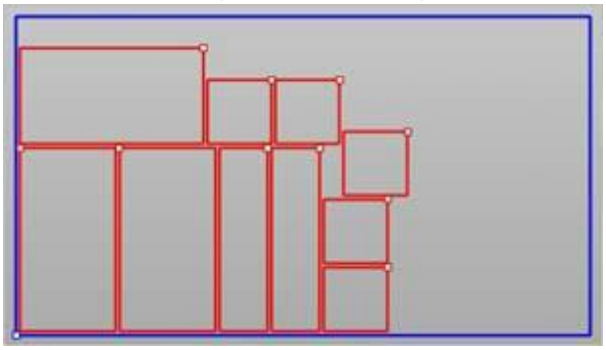
Nesting Direction (True Shape Nesting)

Example

By default Nesting Direction is set Along X.



Nesting Direction is set Along X



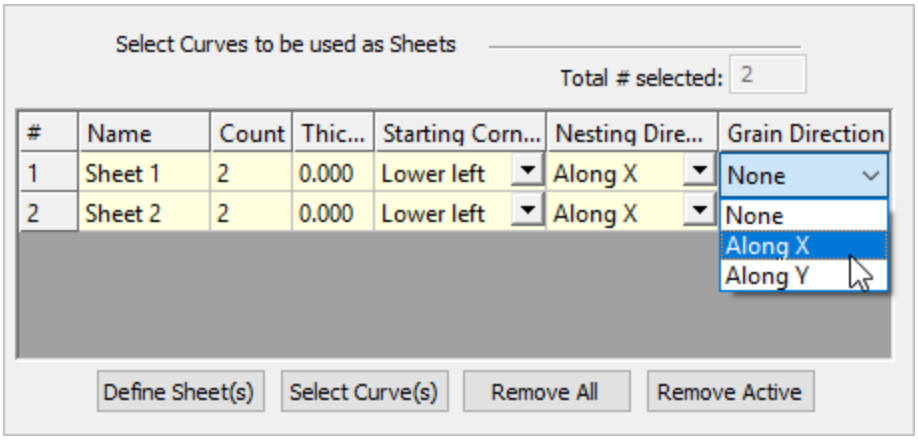
Nesting Direction is set Along Y

Nesting direction is only applicable for True Shape Nesting.

Grain Direction for Sheets

Rolled metal sheets have a property called grain direction, which affects the strength of the parts cut from that sheet. In order to ensure that a part does not fail/break during the subsequent mechanical operations like bending, it is necessary to cut critical parts, such that they align with the grain direction of the stock sheet material.

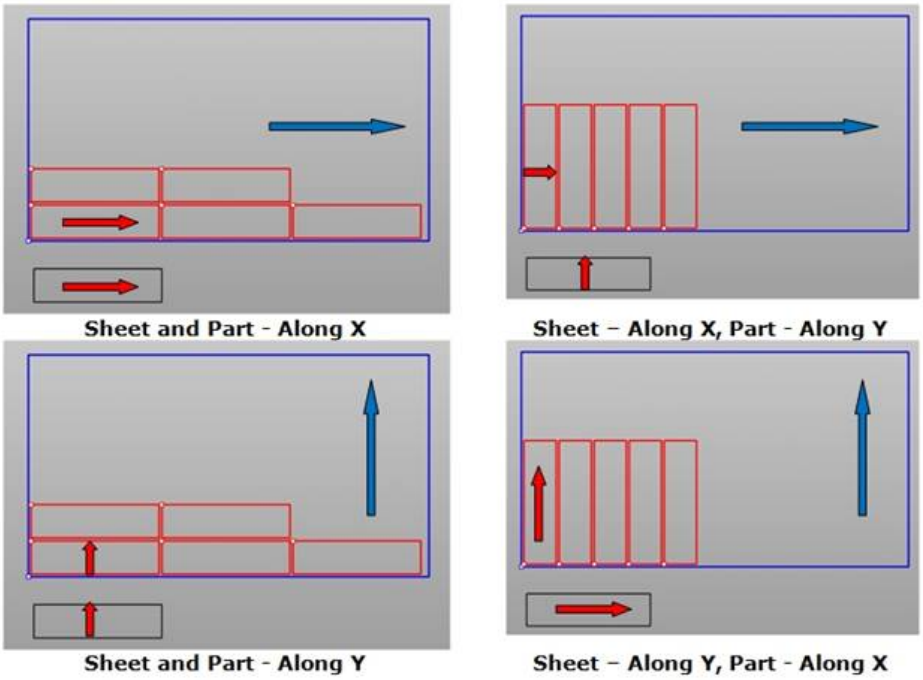
If Grain Direction is used in Nesting, it must be specified for both the stock sheets and the parts that need to be controlled. Grain Direction can be set to None, Along X, or Along Y. The default Grain Direction is set to None.



Grain Direction for Sheets

Example

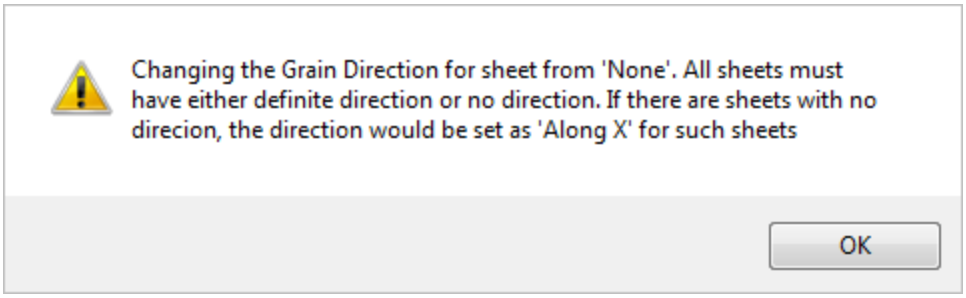
Nesting allows you to attach grain direction with each part and ensures that the grain directions of these parts are always along the grain directions of the corresponding sheets.



Grain Direction - Sheets and Parts

Notes

When using two or more sheets of different sizes, if grain direction is set for any of the sheets ‘Along X’ or ‘Along Y’, the grain direction for all other sheets with 'None' grain direction will be set as 'Along X'.



Related Topics

Nesting Browser work-flow

7.4 Select Part(s) to Nest

This tab allows the selection of Part(s) to be nested. There are 3 buttons on this tab that provide control for the selection of geometry as parts.

Things to Consider when Selecting Part Curves

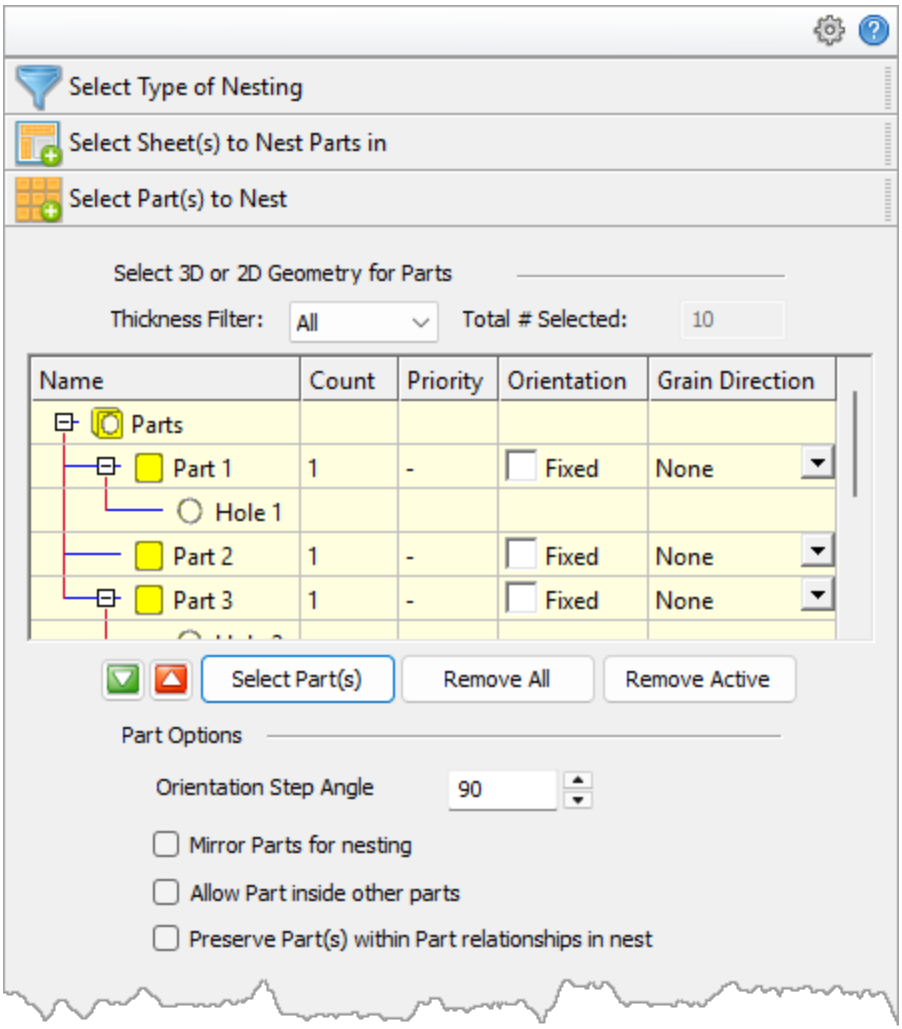
1. Nesting maintains the original Layer, Grouping and Color properties of curves when parts are nested. This means that if curves are grouped, they will stay grouped and maintain their original layer and color after being nested.
2. Holes may be located partially inside the part. This situation can occur in cabinet work when grooves sometimes extend past the part boundary.
3. IMPORTANT: If you plan to nest 2D curves, make sure the Sheet Thickness defined on the Select Sheets tab is set to 0 for all sheets. If the Sheet Thickness is greater than 0 and you are nesting curves, you will receive an error message to check your nest setting and try again.

Nesting Browser - Select Part(s) to Nest

Each selected shape will be shown in the Browser Parts List as either:

1. a Part periphery shown as Part x
2. an internal cutout shown as Hole x

Part parameters (Count, Priority, Orientation and Grain Direction) only apply to the periphery Part shapes.



Select Part(s) to Nest

Thickness Filter

Each sheet can be assigned a thickness. You can use the Thickness Filter to list only the parts on the sheets of the selected thickness.

Select Part(s)

This button will prompt you to select the parts to nest.

Remove All

Select this button first, then pick the curves on the screen that represent the Part shapes to be nested.

Remove Active

This button removes all selected Parts from the Parts List in the Nesting Browser.

Expand and Collapse the Parts List

Initially the Parts List shows all Parts and their subordinate Holes.

 Collapse Up:
The Parts List can be collapsed by selecting the red up-arrow icon, which will suppress the display of any Holes in the list. This Parts only list makes it easier to enter the Part parameters.

 Expand down:
The Parts List can be expanded by selecting the green down-arrow icon, which will then show all Parts and their subordinate Holes (if any).

Select Part(s)

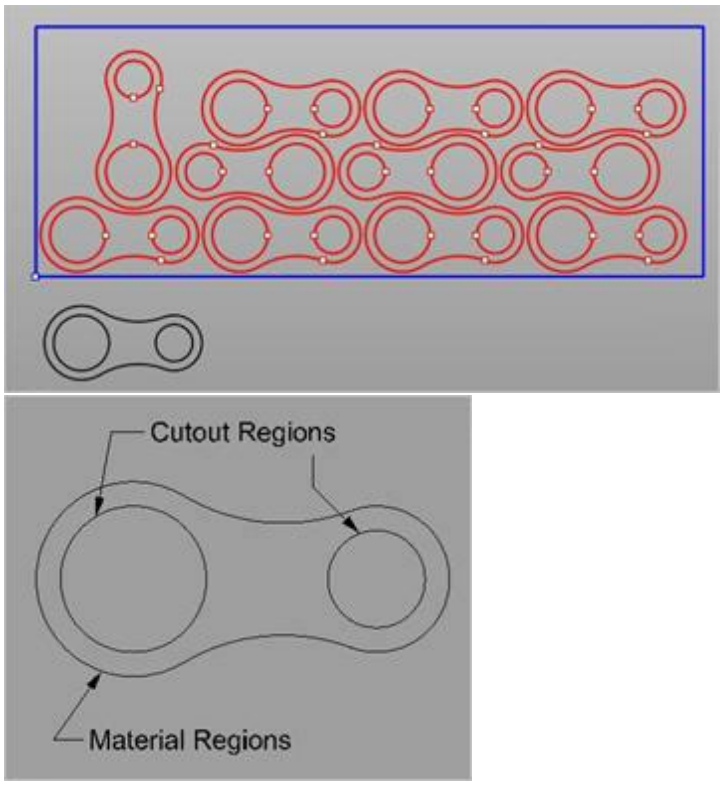
Click the Select Part(s) button to select curves to be used as Part(s). You may window-select closed 2D curves or 3D poly-surface solids or meshes for parts and press enter or click the right mouse button to add curves to the selection list. Selecting open profiles is not supported as this may result in incorrect nests. Parts to be nested may contain cutouts within their periphery. Both periphery curves and cutout curves should be selected as parts for the nesting process.

See Things to Consider when Selecting Part Curves above.

 Sheet Thickness: If you plan to nest 2D curves, the sheet thickness MUST be set to 0 or you will receive an error message when creating the nest.

Periphery Shapes and Cutout Shapes

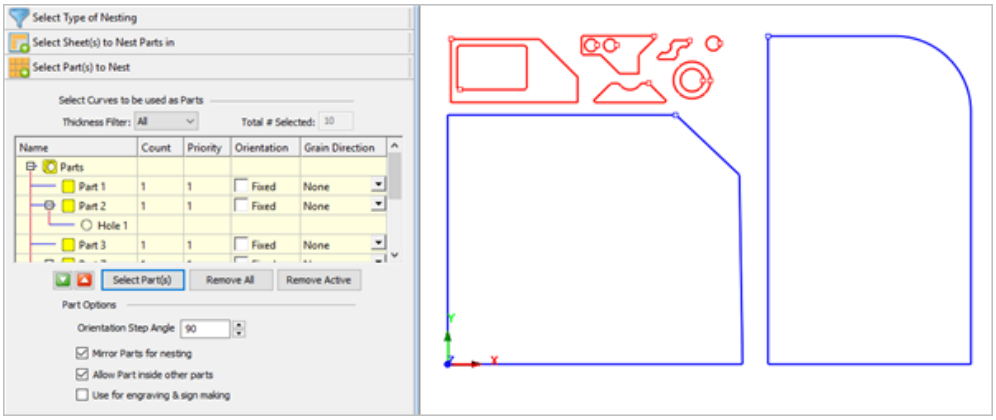
Prior to creating the nest, the Nesting software evaluates all closed curves that have been selected as parts. If it finds that some curves are within other curves, it determines whether each curve is an outer periphery Part shape (material region) or an inner Hole shape of the part. During nesting, each periphery part shape and its associated holes will be treated as a unit part for the purpose of nesting.



Periphery Shapes and Cutout Shapes

The Selected Parts

The selected parts are available under the Select Part(s) to Nest tab in the nesting browser. The geometry selected as parts are displayed in red color.

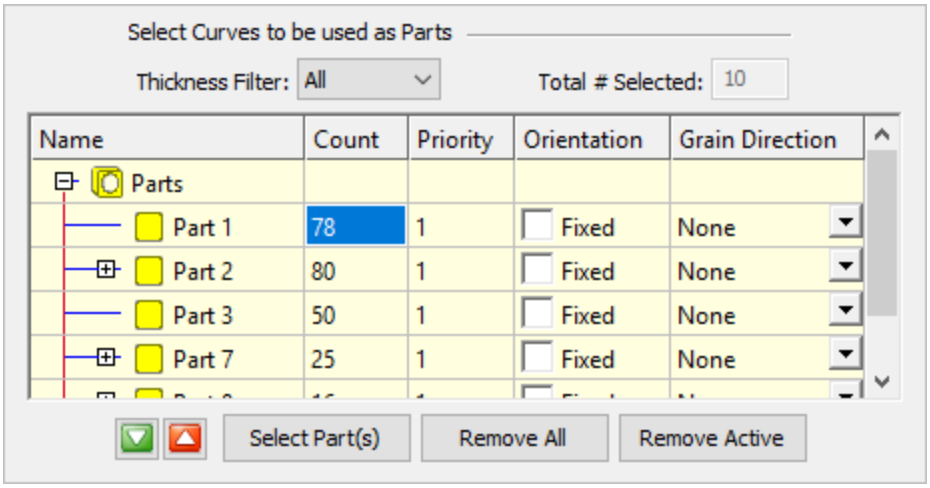


The selected curves are added to the Parts List

Selecting a Part or Hole from the browser shows the selection highlight of the geometry (in **green** color).

Count for Parts

Count defines the desired quantity for each part to be nested. The default value is set = 1.



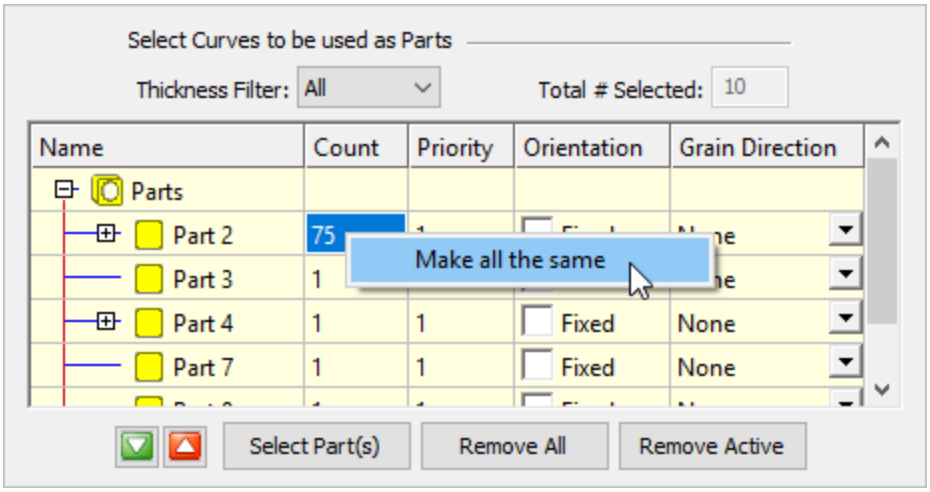
Count for Parts

In the above example, Count for Part1 = 78, Part2 = 80, etc.

Parts with Cutouts: As we can see in the Parts List, each exterior closed curve is defined as one Part. Any interior closed curves are defined automatically as Holes within each Part.

Make all the same

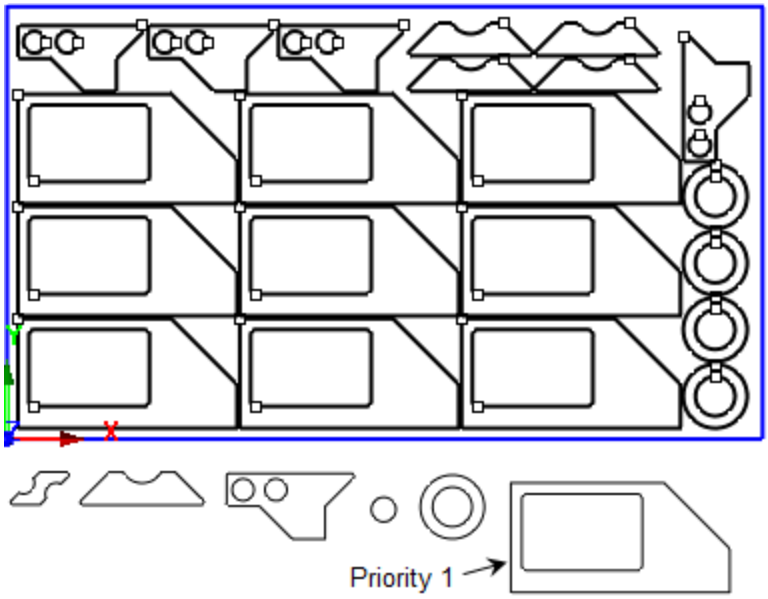
You can right-click on any part count and select Make all the same to automatically adjust all of the parts in the nest to the same count as that part.



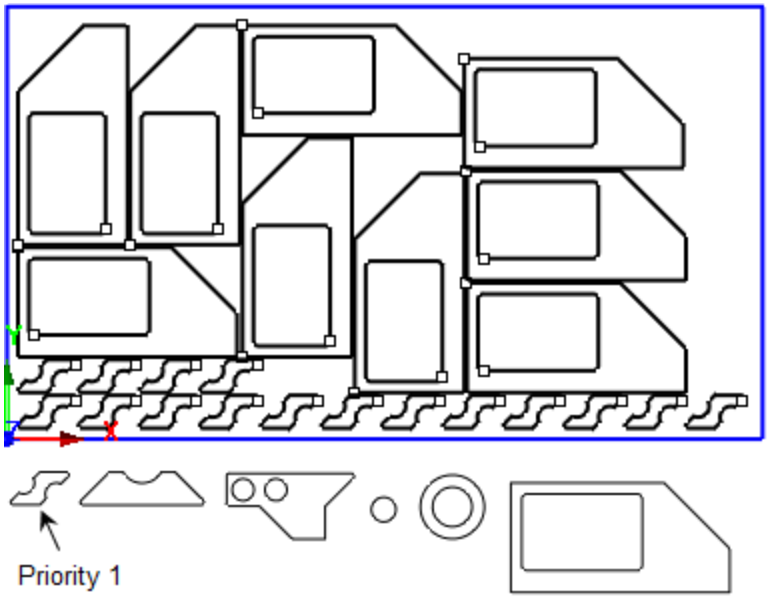
Make Part Count All the Same

Priority

You can set a priority level for each part in the nest. Higher priority parts will be nested first, beginning with sheet 1.



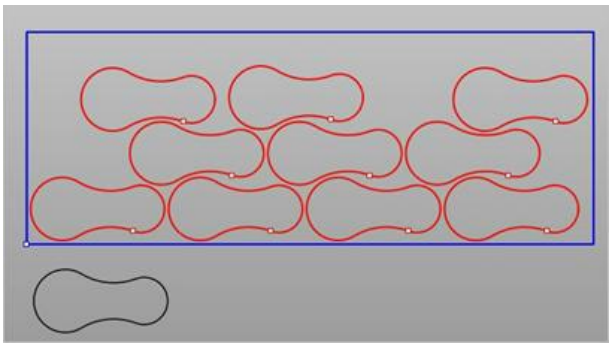
This part is set as Priority #1



Now this part is set as Priority #1

Orientation

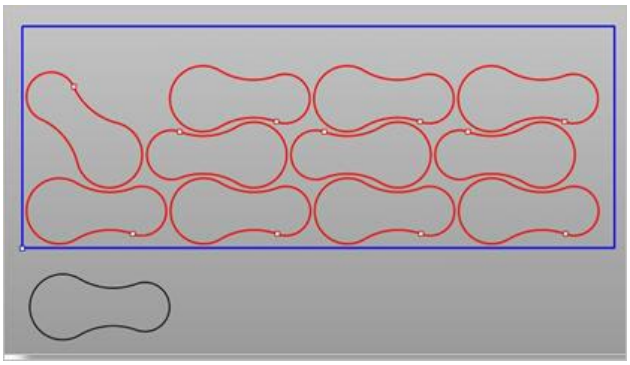
Selecting Fixed does not permit rotation of parts when nested in the sheet.



Fixed does not permit rotation of parts when nested in the sheet

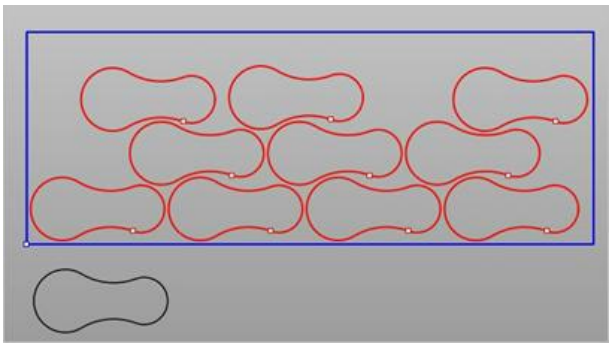
When using Fixed orientation, Grain Direction cannot be set for parts.

In this example, Orientation has been set = 45 to allow the parts to be rotated.



Orientation has been set = 45 to allow the parts to be rotated

Selecting Fixed does not permit rotation of parts when nested in the sheet.



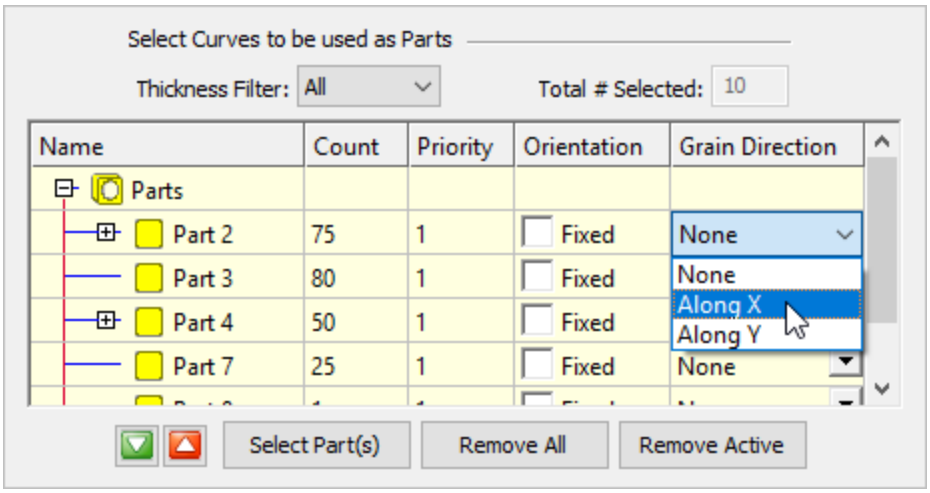
Fixed does not permit rotation of parts when nested in the sheet

When using Fixed orientation, Grain Direction cannot be set for parts.

Grain Direction for Parts

Refer to Grain Direction for Sheets. If Grain Direction has been specified for sheets, then it can be specified for any parts that require this control during nesting. Parts that have a Grain Direction specified will be aligned with the Grain Direction of the sheets.

Grain Direction can be set to None, Along X or Along Y.



Grain Direction for Parts

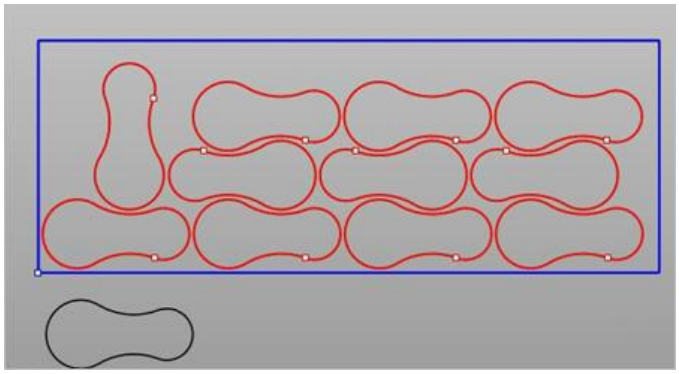
If Grain Direction for part(s) is set without setting grain direction for sheets, Nesting prompts you to set the Grain Direction for Sheets first before setting Grain Direction for Part(s).

Part Options

This provides control to set the following parameters:

Orientation Step Angle (True Shape Nesting)

This allows rotation of the parts to nest and can be defined by specifying orientation step angle. For example, specifying a Step Angle of 90 would allow rotation of all parts by a step increment of 90, which could be 90, 180 or 270 to fill the sheet optimally.



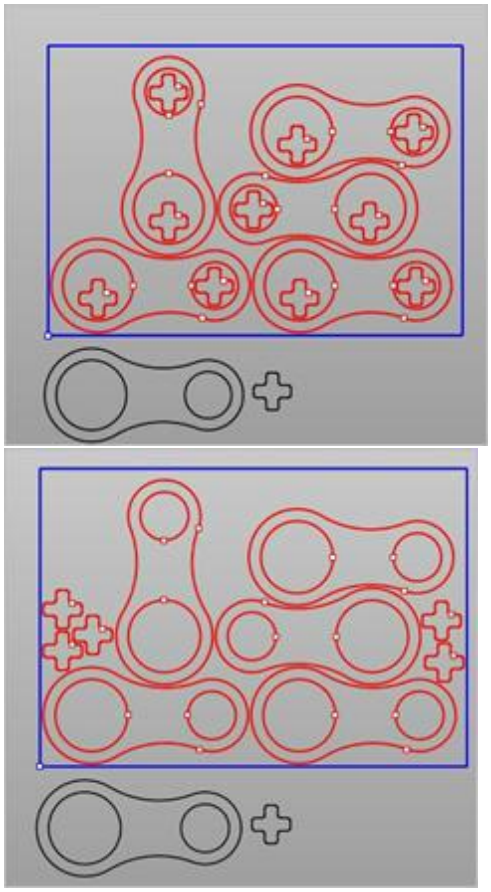
Orientation Step Angle (True Shape Nesting)

Mirror Parts for Nesting (True Shape Nesting)

Selecting this option mirrors the part for nesting.

Allow Part inside other parts (True Shape Nesting)

This allows nesting of parts inside the cutouts of other parts to maximize sheet utilization.



Orientation Step Angle, Mirror Parts for nesting and Allow Part inside other parts are only applicable for True Shape Nesting. These options are not available when nesting type is set to Rectangular.

Preserve part(s) within Part Relationships in Nest

Check this box to preserve the nested relationship of curves for nesting. This allows the ability to nest curves inside a hole as shown below. In this example, the outer rectangle would be the part and the exterior & interior of the letters would be treated as holes. This option also allows you to nest curves that overlap each other.

Use for engraving & sign making

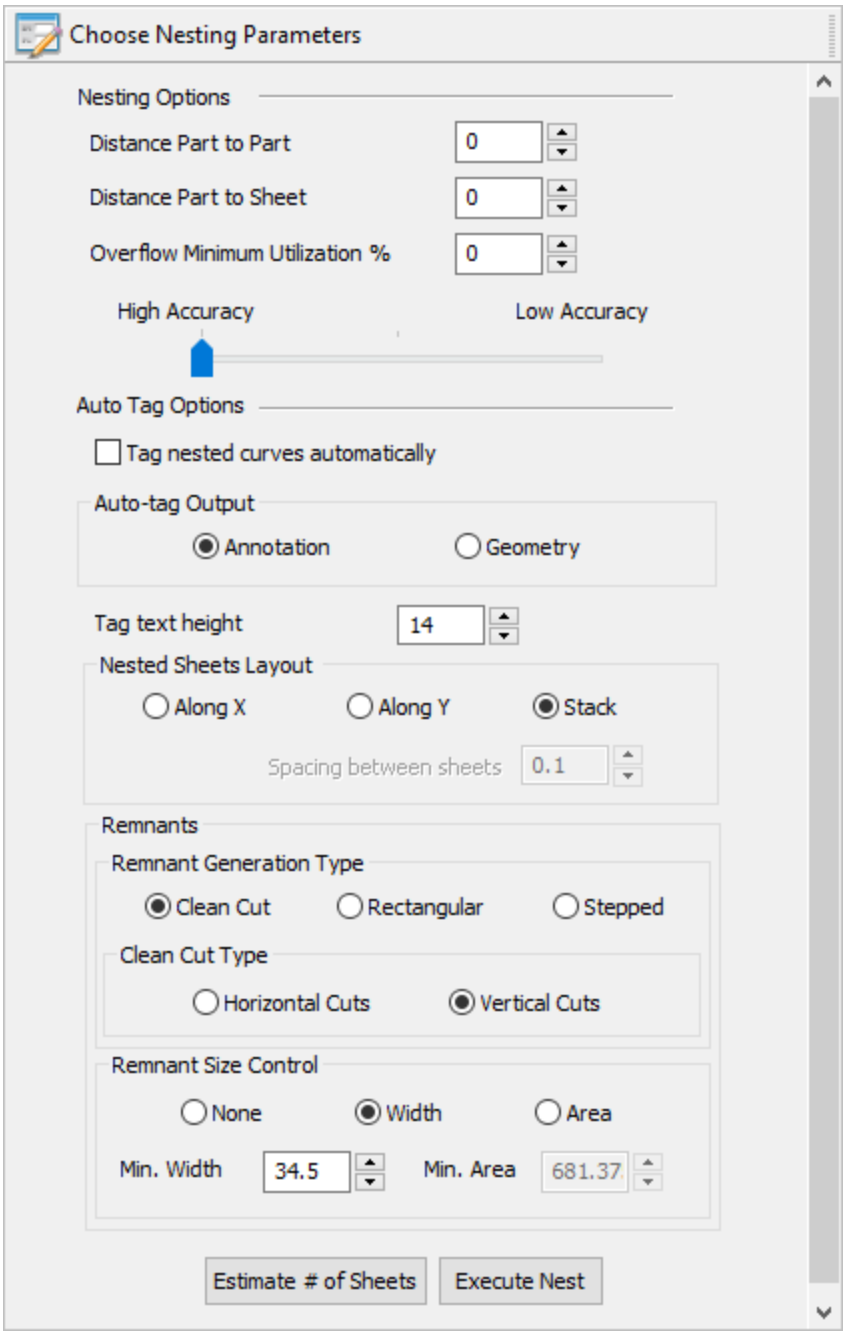
Related Topics

Nesting Browser work flow

7.5 Choose Nesting Parameters

This tab allows you to specify nesting parameters and lets you execute and preview the nest.

Nesting Browser - Choose Nesting Parameters tab



Nesting Browser - Choose Nesting Parameters tab

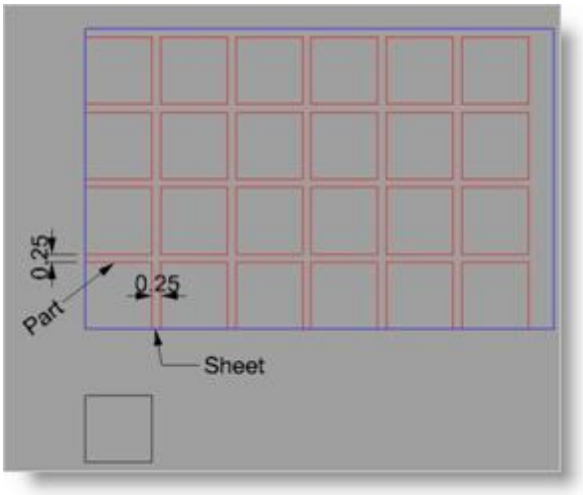
Nesting Options

Choose from the following nesting options:

Distance Part to Part

This parameter defines the minimum distance between each part within a sheet. In the example below, Distance Part to Part is set = 0.25. and Distance

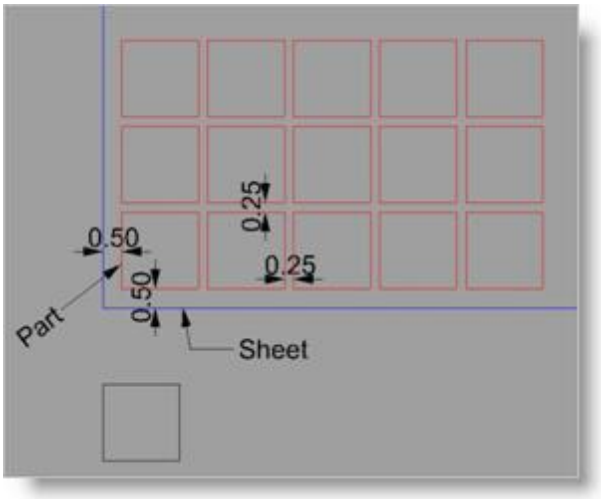
Part to Sheet= 0.



Distance Part to Part Example

Distance Part to Sheet

This parameter defines the minimum distance between parts to the edge of the sheet. In the example below, Distance Part to Part is set = 0.25. and Distance Part to Sheet = 0.50.



Distance Part to Sheet Example

Overflow Minimum Utilization %

When the Nesting system fits parts on a sheet of material, it calculates a percentage of utilization for that sheet, which is an indicator of how efficiently the sheet material is used.

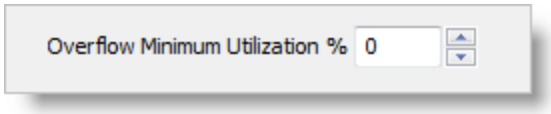
Overflow Minimum Utilization % defines the minimum percent of material utilization that is permitted on any sheet of the nest. If this parameter is set to a value, then the percentage of utilization of material for each sheet must

exceed this number. If the percentage of utilization for any sheet is below this number, then nesting will be suppressed for that sheet.

Overflow Minimum Utilization %: This parameter can be used to eliminate remnants on the last sheet used, since it is the last sheet that typically has the lowest percentage of utilization.

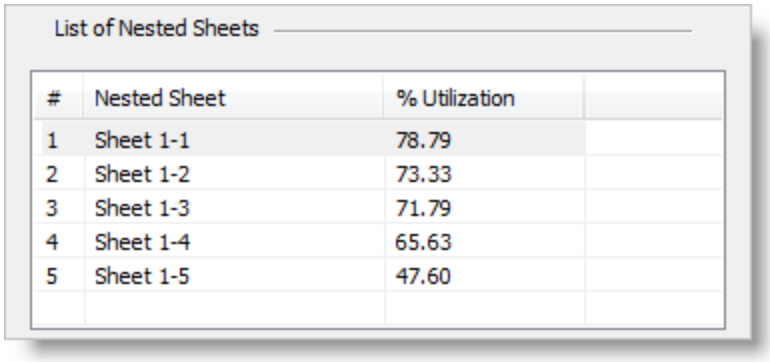
Example of Minimum Utilization = 0

An example the use of Overflow Minimum Utilization % is shown below. If the value is set to 0, then no restriction of the nesting will occur. In the example below, five sheets were utilized. The last sheet percentage utilization was determined to be 47.60 %.



Overflow Minimum Utilization % = 0

A setting of 0% will result in the following utilization report:

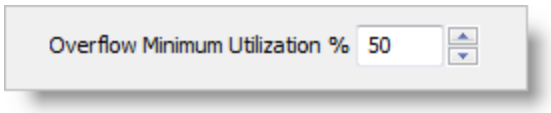


#	Nested Sheet	% Utilization
1	Sheet 1-1	78.79
2	Sheet 1-2	73.33
3	Sheet 1-3	71.79
4	Sheet 1-4	65.63
5	Sheet 1-5	47.60

List of Nested Sheets Results - Note Low Utilization of Sheet 5

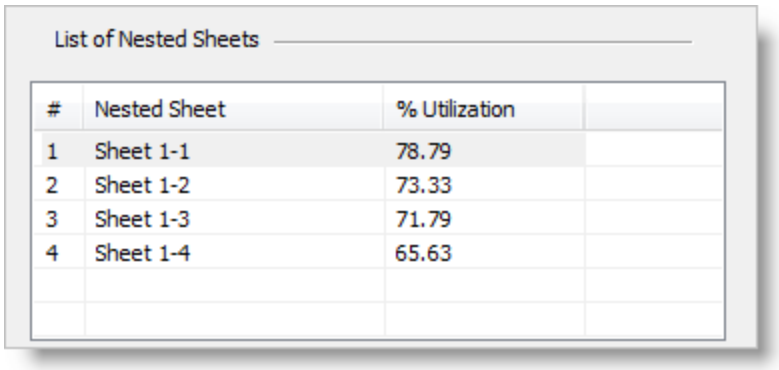
Example of Minimum Utilization = 50

Next we set Overflow Minimum Utilization % to 50. Note that the nesting of sheet 5 is eliminated because it does not meet or exceed the % Utilization.



Overflow Minimum Utilization % = 50

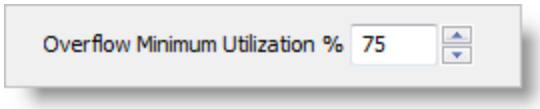
A setting of 50% will result in the following utilization report:



List of Nested Sheets Results - Note that Sheet 5 is Eliminated

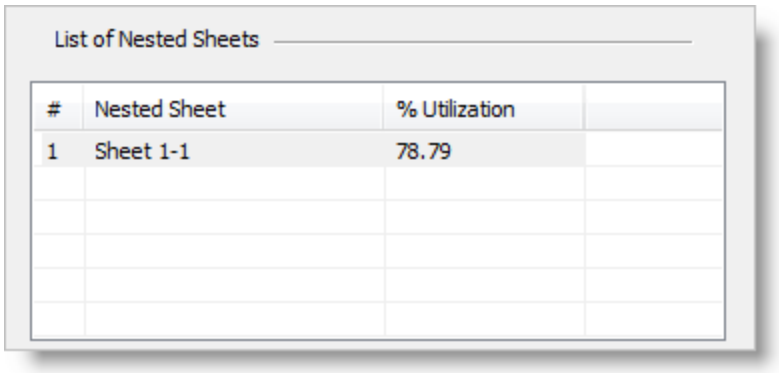
Example of Minimum Utilization = 75

Furthermore, if the Overflow Minimum Utilization % is set to 75, then nesting will be eliminated on all but the first sheet because their percentage of utilization is lower than 75% as shown below.



Overflow Minimum Utilization % = 75

A setting of 75% will result in the following utilization report:



List of Nested Sheets Results - Note that All but Sheet 1 is Eliminated

Nesting Accuracy

This defines the degree of accuracy of the nested pattern. A higher accuracy of the nest takes more time to compute the nest. The actual distance between the parts could be higher than the specified value and this is dependent on the level of accuracy defined. Accuracy can be set to High, Medium, or Low.

Auto Tag Options

The tagging option causes a number to be shown with each nested part when the nest is accepted or committed. The number is the sequence number assigned to the selected Part in the Parts List of the Nesting Browser.

Tag Nested Curves Automatically

By default, tagging is OFF, but may be activated by checking this box on the dialog.

Auto-Tag Output

There are two forms of tagging that can be generated, Annotation and Geometry. In both cases, the tag number is located with its origin at the center of the individual nested part in the nest.

Annotation

The tag number is shown on the screen as graphics display text only. The height of the annotation text is dynamically controlled by the Dimension Text Height.

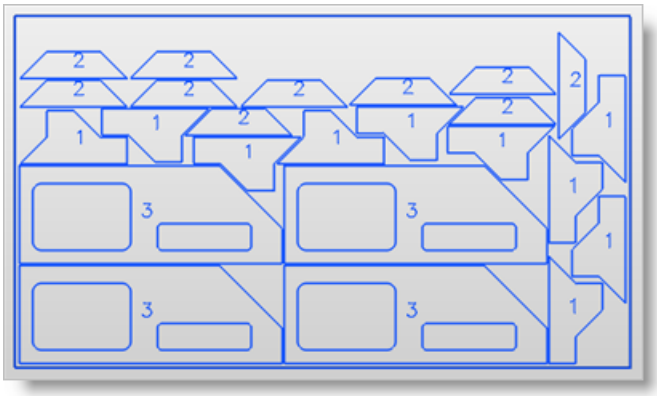
Geometry

The tag number is created as geometry curves, which are selectable as geometry.

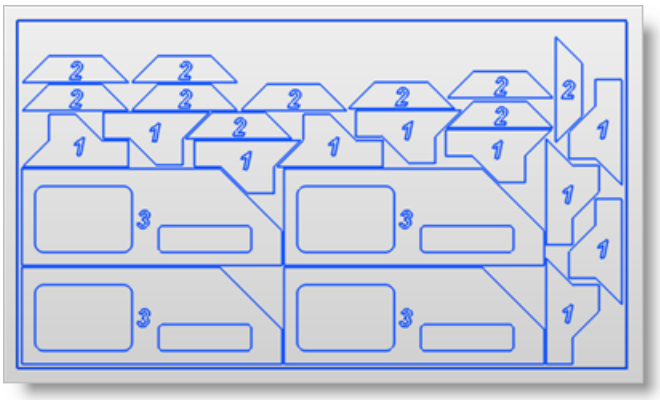
Tag Text Height

Enter the Text Height for tagging. The value entered in this height field is in the units of the part file, either Inches or Metric.

Example of Auto Tag Option



Annotation tagging with height of .75 inch (text is display only)



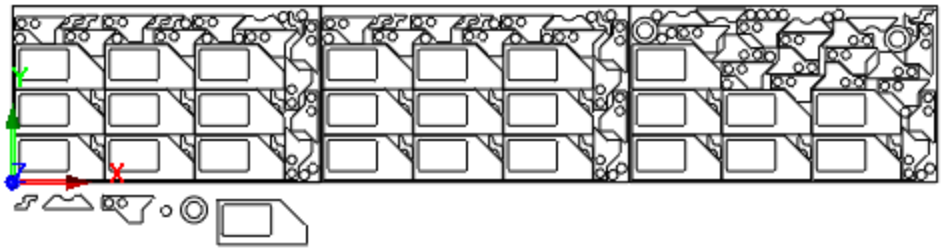
Geometry tagging with height of 1 inch (text is selectable)

Nested Sheet Layout

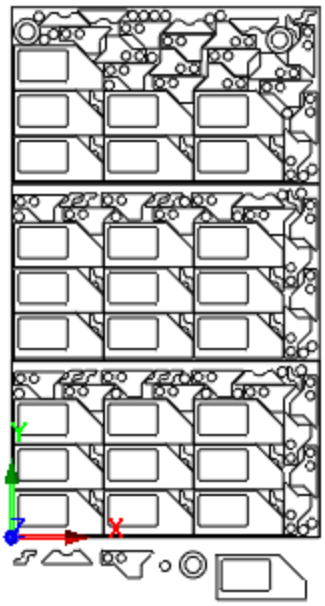
You can choose from the following nested sheet layout options. These options are enabled when you elect to create each sheet in a separate layer or group from the Commit Nest tab.

Along X / Along Y

Arranges all sheets along the X or Y direction depending on your selection.



Nest Layout = Along X



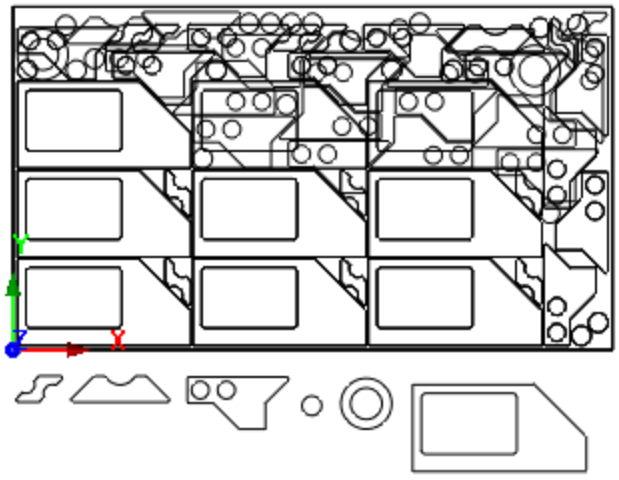
Nest Layout = Along Y

Force all parts to XY plane

Check this box and all nested geometry will be forced onto the XY Plane. 2D curves will lie on the XY Plane. For 3D solids, the base of the solids will lie on the XY Plane.

Stack

This option stacks the geometry. This option is disabled if None is selected from the Preview and Commit Nest tab under the Nested Sheets Geometry Grouping section.



Nest Layout = Stacked

Spacing between sheets

If Along X or Along Y is selected, enter the linear spacing between each sheet.

Remnant Controls

Remnant Generation Type

None
Remnant control is disabled.

Clean Cut
Select Clean Cut to limit the remnant stock to a "clean" vertical or horizontal cut and then select either Horizontal or Vertical.

Rectangular
Select Rectangular to limit the remnant stock to the remaining rectangular area.

Stepped
Select Stepped to limit the remnant stock to the remaining rectangular area in addition to the partial stepped area. This option allows for the greatest amount of remnant stock to be assigned to a sheet.

Remnant Size Control

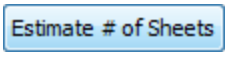
None
Select this option to accept any/all remnants

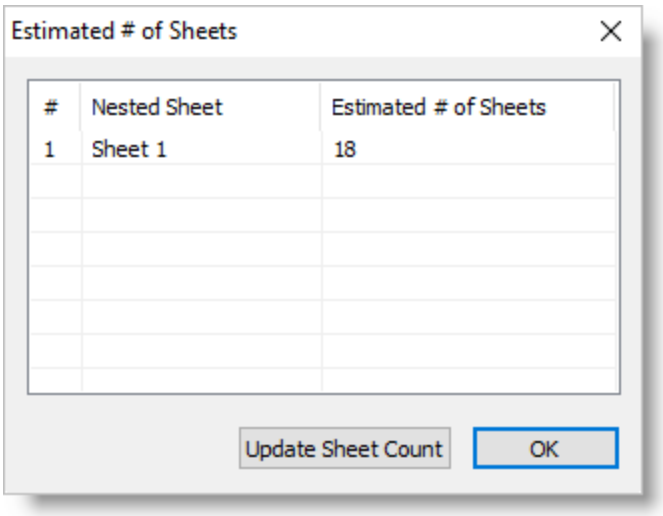
Width / Min Width
Select Width to limit remnants to a specific width dimension and then enter the dimension in the Min. Width field provided.

Area / Min Area
Select Area to limit remnants to a specific area measured in square units and then enter the area value in the Min. Area field provided.

Estimate # of Sheets

Selecting this button will produce an on-screen report showing how many of each of the sheets will be needed for the total nesting.





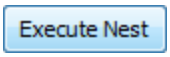
Dialog Box: Estimated # of Sheets

Update Sheet Count

Your report is listed here. If your total number of sheets is less than the Estimated # of Sheets listed in the report, you can select the Update Sheet Count button to adjust your sheet count automatically.

Execute Nest

Selecting this button computes the nest based on sheets, parts and nesting parameters specified. This does not display the resultant nested parts within the sheet. Selecting this button will move you to the Preview Nest tab automatically.



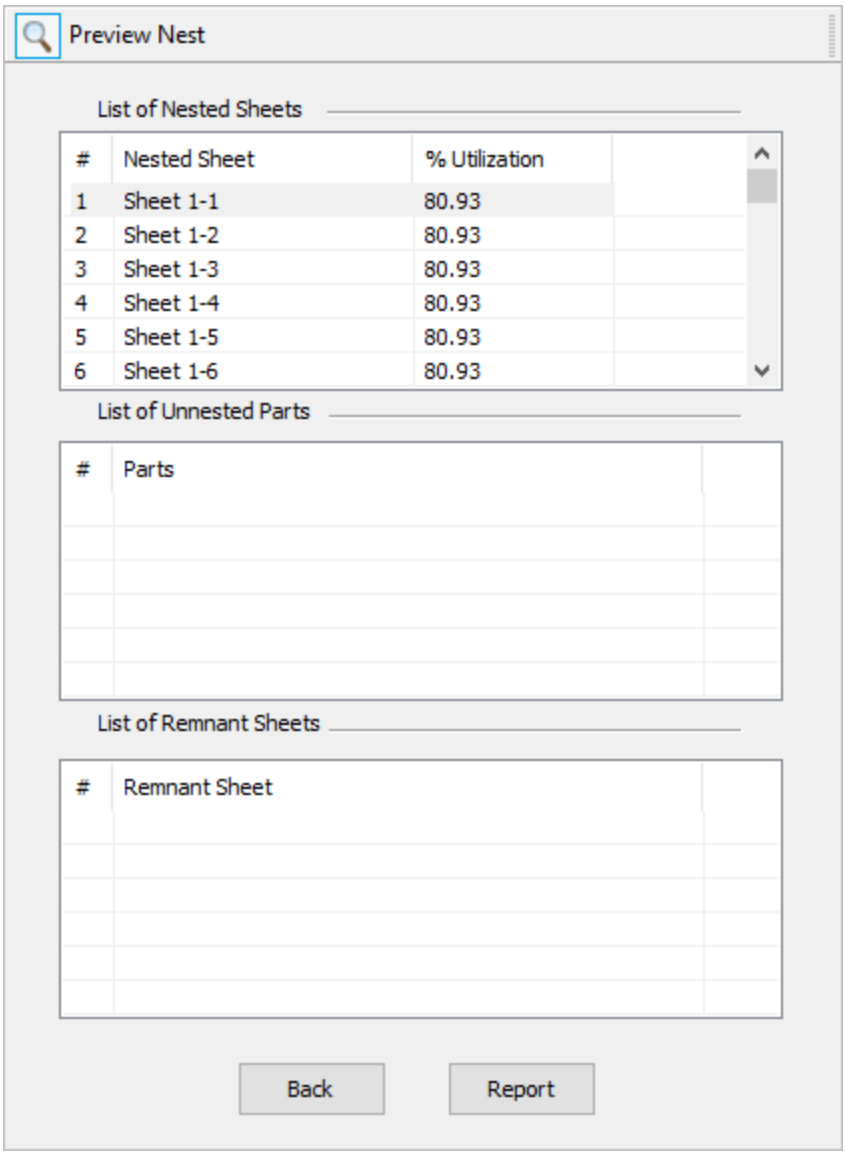
Related Topics

Nesting Browser work flow

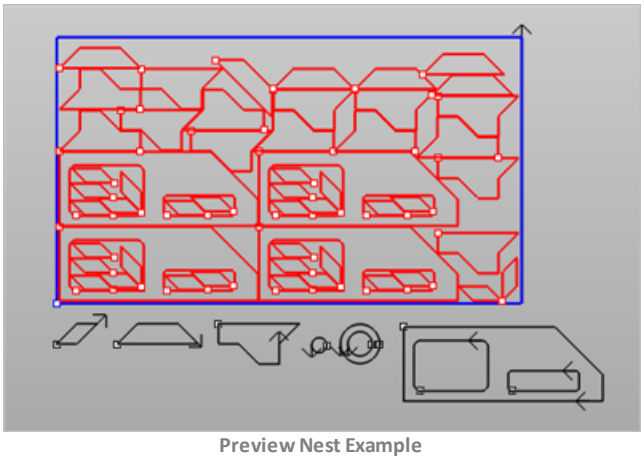
7.6 Preview Nest

Selecting Execute Nest from Choose Nesting Parameters tab automatically displays the resulting nest and switches to the Preview Nest tab. Select from the list of nested sheets to display that sheet. Any parts that were not nested are also listed here.

Nesting Browser - Preview Nest tab



Execute Nest Results



Preview Nest Example

List of Nested Sheets

True Shape Nesting - % Utilization
This is an efficiency calculation showing how much of the sheet area was used for true part area production.

Rectangular Nesting - % Utilization
This is calculated using the rectangular areas around each part, rather than the true shape of the part. This will not represent a true efficiency comparison between true part area and sheet area.

The non-utilized remainder of the sheet area would be remnant and/or scrap.

List of Unnested Parts

If for any reason a part could not be nested in any sheet, it will be listed here.

List of Remnant Sheets

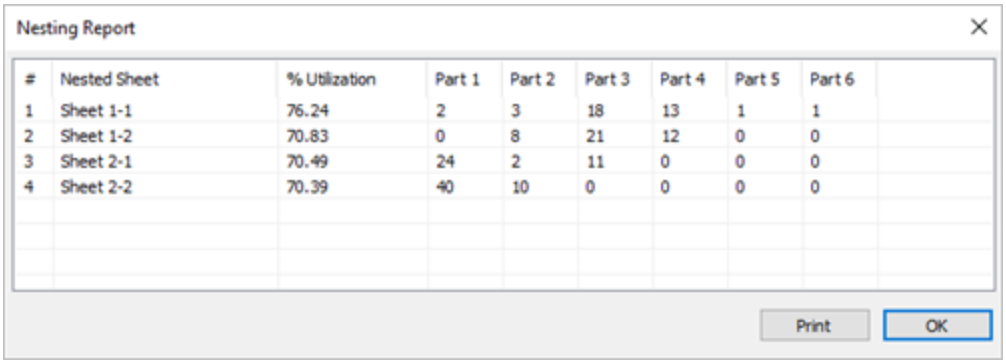
This list will include all Remnant Sheets if any, calculated using the Remnant Controls from the Choose Nesting Parameters tab.

Back

Returns you to the Choose Nesting Parameters tab.

Report

Displays a nesting report in a separate dialog.



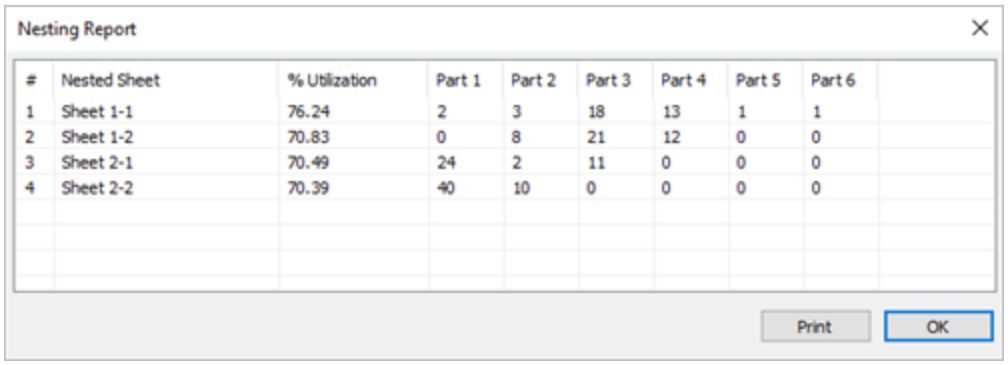
Nesting Report

Related Topics

Nesting Browser work flow

7.6.1 Nesting Report

Select Report from the Preview Nest tab of the Nesting Browser to display this Nesting Report dialog. It report provides information about the nest that you are previewing such as quantities and % Utilization for each nested sheet..

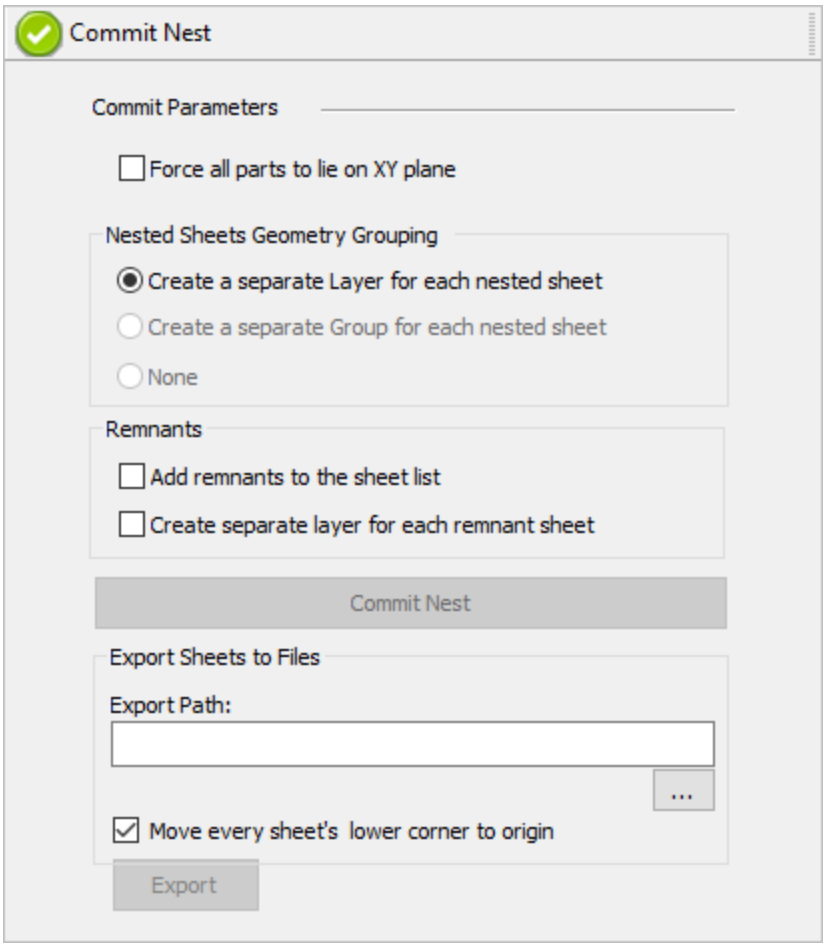


Nesting Report

7.7 Commit Nest

This tab displays a list of all sheets that have been specified, including multiples of the same size and sheets of other sizes. Selecting each sheet previews the nest for the selected sheet in the graphics area.

Nesting Browser - Commit Nest tab



Nesting Browser - Choose Nesting Parameters tab

Force all parts to lie on XY plane

Check this box and all nested geometry will be forced onto the XY Plane. 2D curves will lie on the XY Plane. For 3D solids, the base of the solids will lie on the XY Plane.

Nested Sheets Geometry Grouping

These options allow you to create the sheets and nested parts in separate groups. If one of these options is selected, each output nested sheet will be added to a separate group using the same naming conventions used for the layers. Select from one of the following options:

- Create a separate Layer for each nested sheet
- Create a separate Group for each nested sheet
- None

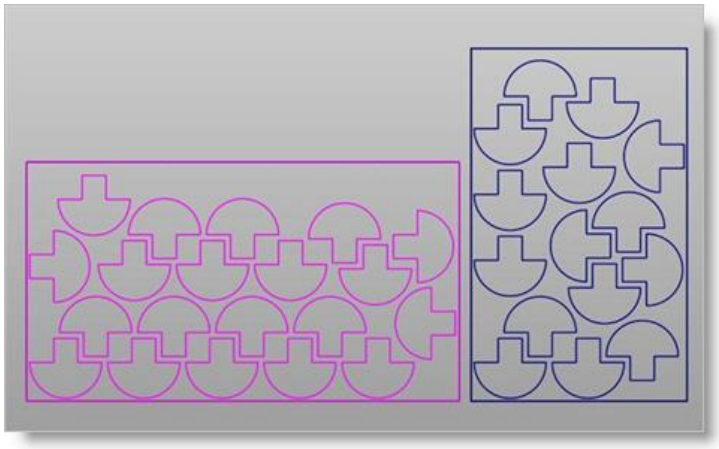
Remnants

These options allow you to control how remnant sheets are created during nesting. You can add any remnant sheet back into the sheets list so that they can be reused. You can also create a separate layer for each remnant sheet when the nest is committed.

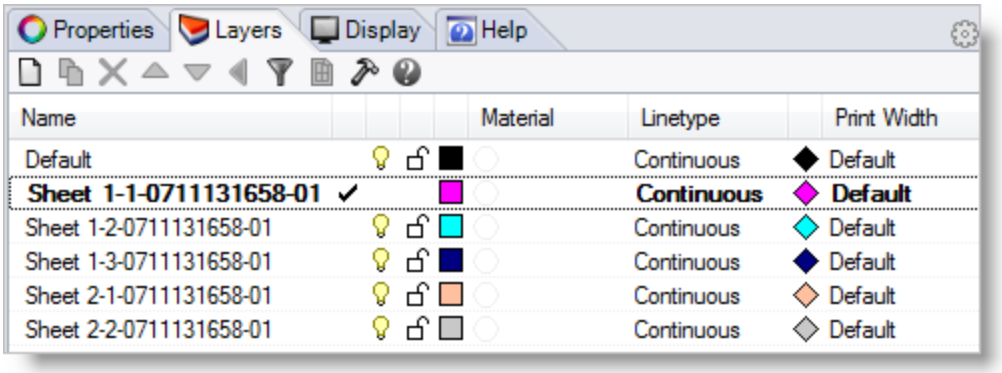
- Add remnants to the sheets list
- Create separate layer for each remnant sheet

Commit Nest

Selecting Commit Nest outputs the nest results to Rhinoceros.



Each sheet is output to a new layer assigned with a unique layer name.



Nesting Complete: At the end of this Nesting process, you can return to RhinoCAM and create Mops for machining the parts in the nest.

Export Sheets to Files

These options allow you to export each nested sheet to a separate drawing file.

Export Path

Use this to specify where you wish the exported sheet files to be located.

Move every sheet's lower corner to origin

Check this box to move the lower left corner of each sheet to the 0,0,0 Origin.

Export
Pick this button to export your sheets.

Related Topics

Nesting Browser work flow

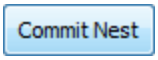
Cloning of Predefined Regions

The Machining Objects Browser located in the MILL module provides a Regions function that will clone an existing pre-defined region and propagate it to all identical geometry shapes. This allows such region characteristics as Start Point, Region Direction, and Bridge Points to be defined on one Part and then propagated to all identical parts in a Nest.

The following example describes the process to accomplish this:

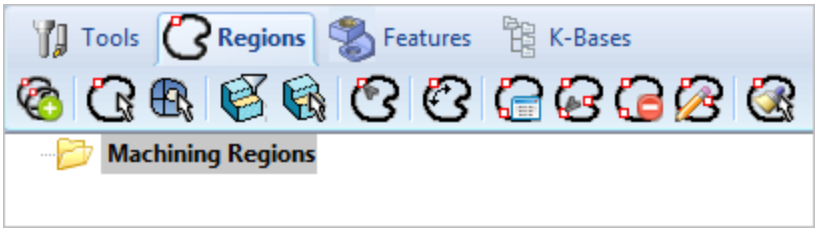
Step 1: Commit the Nest

The nest must be Committed so that the nest geometry exists on one or more layers of the CAD file. The Commit Nest button is located on the Preview and Commit Nest tab of the Nesting Browser.



Step 2: Select the Regions tab

Exit the NEST module and then activate the MILL module to access the Regions tab of the Machining Objects Browser.



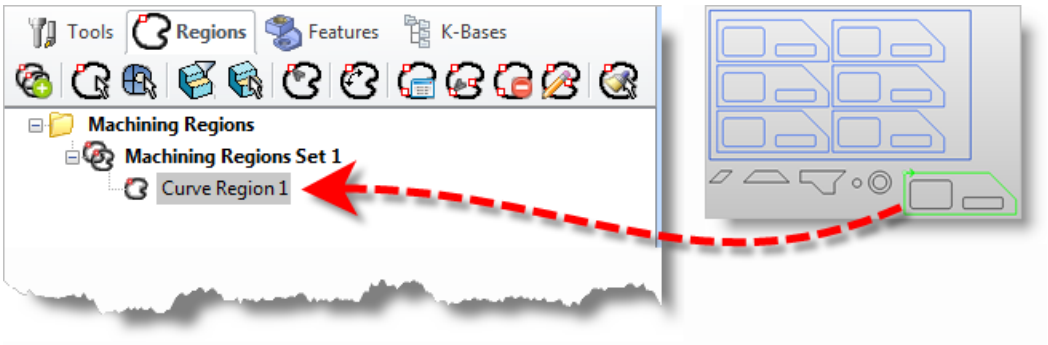
Regions tab of the Machining Objects Browser

Step 3: Create a Pre-defined Region

 Using the Select Curves icon (2nd from left), select the geometry shape of one of the parts where the Start Point, Region Direction, and/or Bridge Points need to be defined and propagated throughout the nest.

Accept the selection and notice that the region is created in the Browser list, such as Curve Region 1 shown below.

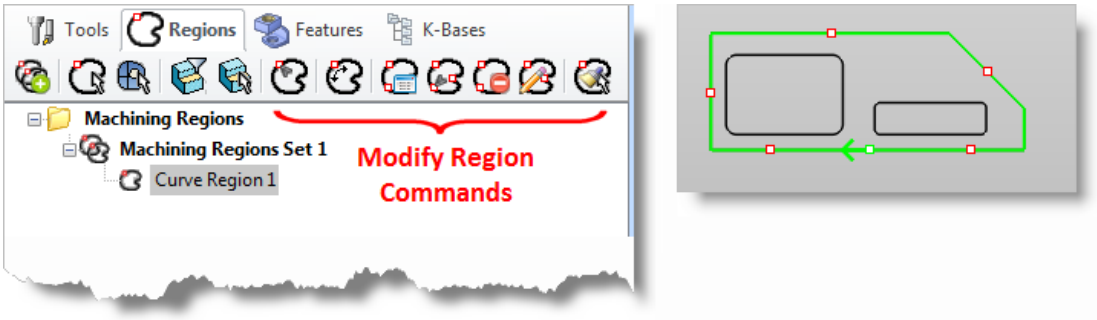
Note: In this example the staged part was used to create the original pre-defined Region, but any of the identical parts in the nest may also be used to create the original pre-defined Region in this step.



Curve Region Defined

Step 4: Add characteristics needed for that Pre-defined Region

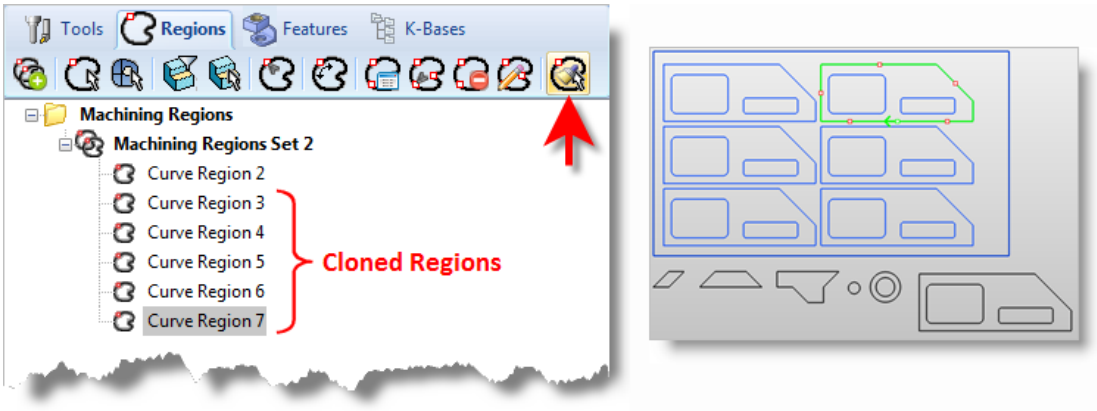
The Regions tab contains other commands to modify Regions. You can use these to modify the Start Point, change the Region Direction or add Bridge Points to that pre-defined region.



Locating the Modify Region Commands

Step 5: Clone the Pre-defined Region to all identical parts of the nest

When the pre-defined region is defined as desired and is highlighted in the Browser, select the Clone Selected Machining Regions icon (right-most). The system will find all other parts whose geometry is identical to the pre-defined Region and create a cloned pre-defined Region with the same characteristics on each those identical parts. See Notes below.



The Display of Cloned Regions

Note 1: Layers. The cloning process works only on visible geometry. Parts that are on invisible layers (multiple sheets) will not receive cloned Regions. Layer control is an important consideration in managing the cloned regions for machining purposes and ease of selection during the Machining Operations creation process.

Note 2: Duplication. If cloning is done multiple times using the same original pre-defined Region and the same target geometry (nested parts), then duplicate cloned Regions on each part will result.

Step 6: Clone additional Parts & Holes

Repeat Steps 3-5 for other Parts and Holes where needed.

Step 7: Cleanup

After all the pre-defined Curve Regions are created, go through the Curve Regions List and remove any Regions that are not needed, such as the Regions created on the original staged parts and any duplicate Regions.

Step 8: Modifications

If necessary, any of the resultant pre-defined Regions can be individually modified to meet machining needs, such as changing the Start Point and modifying Bridge Points.

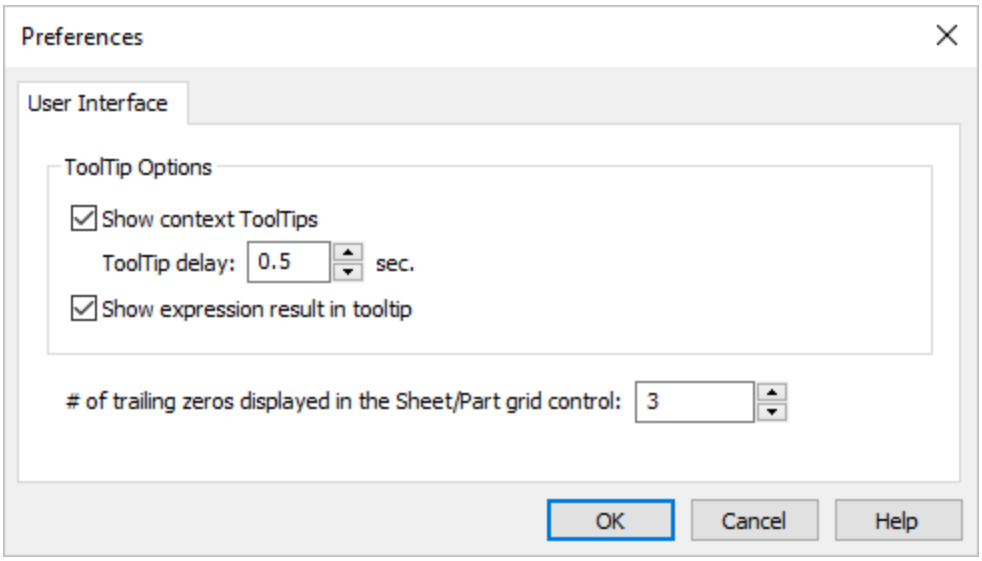
Related Topics

Nesting Browser work flow

NEST Preferences

Choose from the following NEST Preferences. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: Preferences



NEST Preferences

Preferences Icon

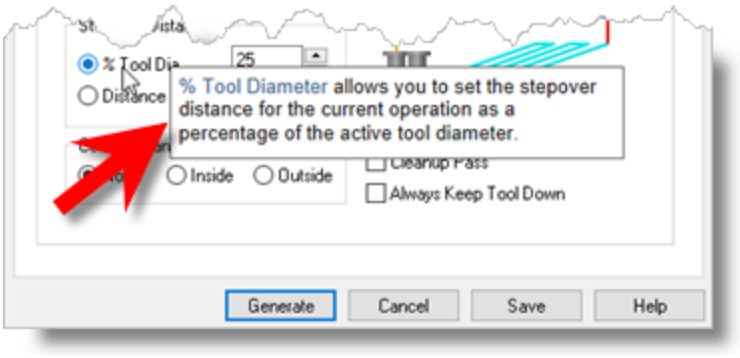
Locate the Nest Preferences Icon

Language

Use this to set the system language of the plugin's User Interface.

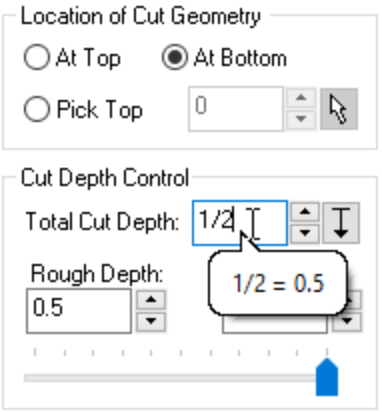
ToolTip Options

Show context ToolTips
Check this box to display Context ToolTips when the mouse moves over a parameter in a dialog. A definition of the parameter will pop-up automatically. **Note** that Context ToolTips may not be available for ALL dialogs. You can also set the ToolTip Delay in seconds. This is the amount of time it takes to display the Context ToolTip when the mouse activate it.



Show expressions results in tooltip

You can enter expressions in any dialog field that expects a numerical value and the value will be computed and entered automatically. Check this box to pop-up the results of any expressions in a ToolTip balloon. An example is shown below.



Show Expressions in ToolTip

of Trailing Zeros

This controls the number of trailing zeros displayed for numerical values such as Sheet Thickness.

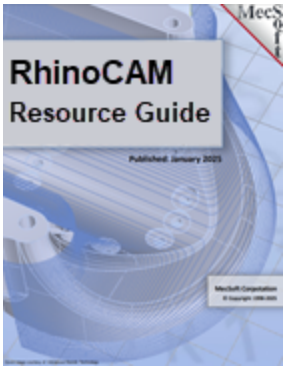
Related Topics

Nesting Browser work flow

More Resources

Download this PDF Guide for a list of the available RhinoCAM Resources.

2025 RhinoCAM Resource Guide



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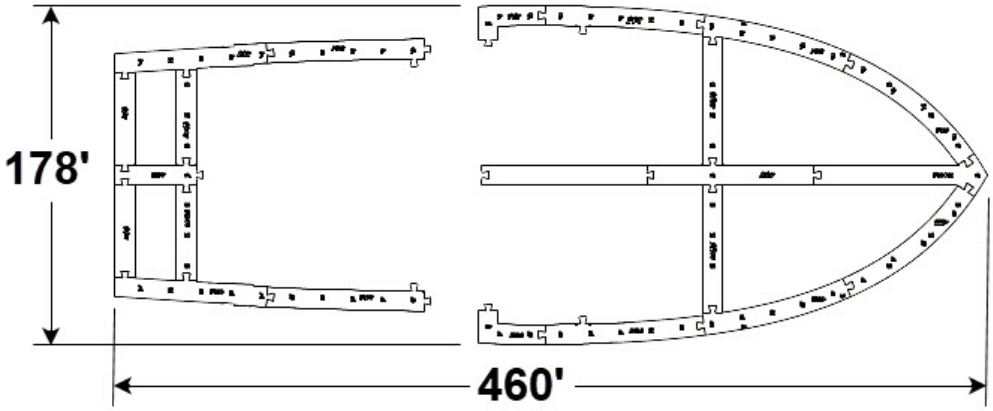
Nesting in Practice

The subject of this post is the Anjamar, a Custom Carolina 45, flybridge, featured at the Palm Beach Boat Show by Rhodes Yacht Design. The Anjamar is uniquely designed with tremendous thought given to styling performance and function with all the qualities of a custom sportfish powered with the simplicity of outboards. Anjamar was built to the highest of standards using modern technology right down to the carbon fiber bulkheads. Learn more about the full case study here.

The Custom Carolina 45

1. The Parts to NEST

The parts for this case study when assembled form the jig and fixtures upon which the Wheel House of the Custom Carolina 45 yacht is assembled. The assembly consists of 20 interlocking components that are each 2D line drawings that lie on the XY plane. You can see the assembly in the illustrations below.

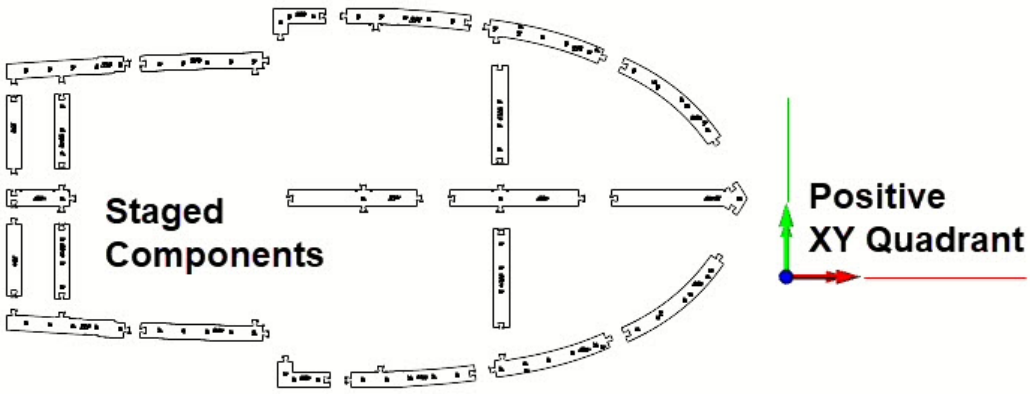


The components when assembled form the jig and fixtures upon which the Wheel House of the Custom Carolina 45 yacht is assembled.

2. Staging Components for Nesting

In preparation for nesting, each component is moved such that no component is touching or overlapping each other while remaining on the XY plane. Note that optionally, during the nesting process, each component can be projected onto the XY plane. All of the components are also moved outside of the default positive XY quadrant. This is done because the positive XY quadrant is where the actual nesting takes place.

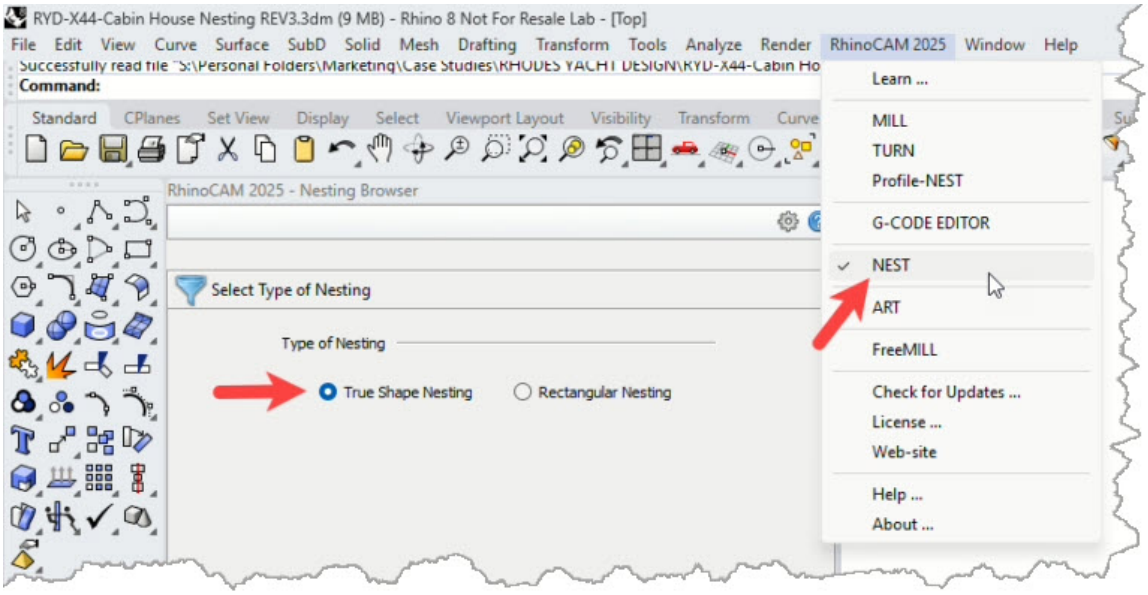
So now our assembly looks like this:



The part components to NEST are staged and ready to begin the NEST process

3. Loading NEST

NEST is a module of RhinoCAM (and VisualCADCAM). It is available to all users who are active Annual Maintenance Subscribers (AMS). To launch the NEST module, load the RhinoCAM plugin for Rhino. Then go to the RhinoCAM main menu and select NEST. This will display the NEST module browser as shown below.

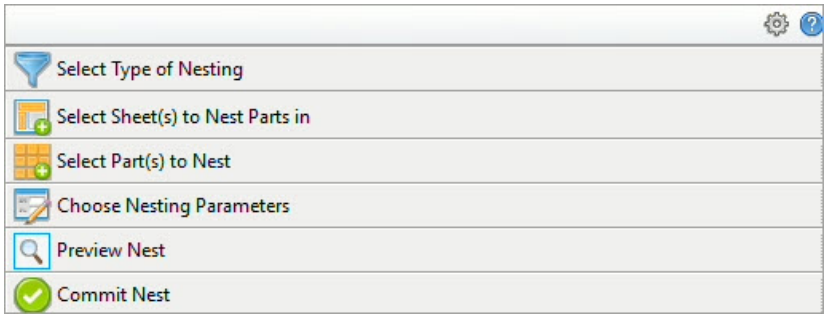


RhinoCAM NEST is loaded from the RhinoCAM Main Menu in Rhino

4. About the NEST Browser

When selecting NEST from the RhinoCAM Main Menu, the Nesting Browser by default will display on the left side of the Rhino display. You can move it by dragging and docking like any other Rhino

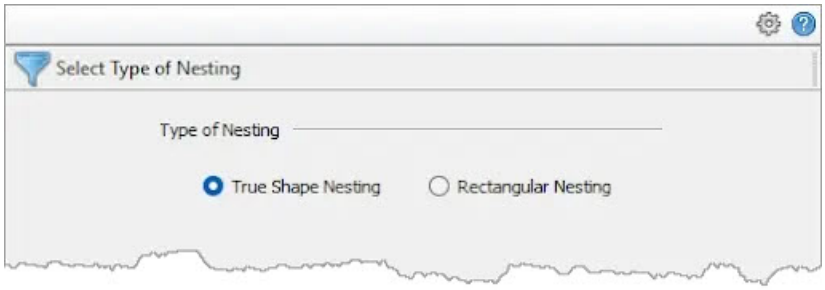
menu. The NEST Browser consists of a collection of Tabs that will walk you through the process of Nesting. These include the NEST type, the NEST sheets, the NEST parts, the NEST Parameters, the NEST Preview and finally to Commit the NEST.



The NEST Browser contains each tab required to create nested components onto a flat sheet.

5. The Select Type of Nesting Tab

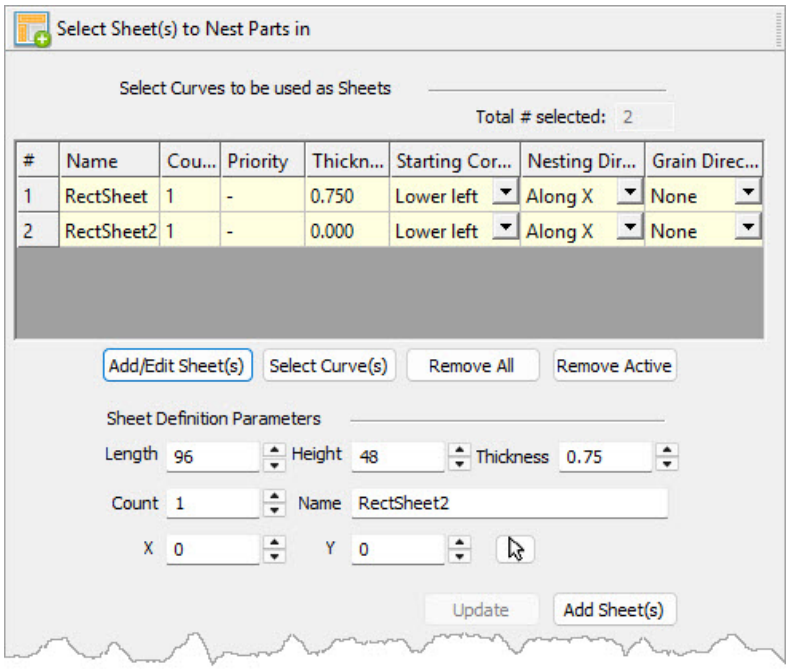
As shown in the image above the Select Type of Nesting tab allows you to select from two types of nesting, True Shape Nesting and Rectangular Nesting. Each is defined below and for this project, the True Shape Nesting option is selected.



- **True Shape Nesting:** This method is useful when the actual geometric details of the part are taken into consideration while nesting. True shape nesting enables interlocking of parts, recognition of arbitrary shaped sheets.
- **Rectangular Nesting:** This method is ideal for nesting mostly square and rectangular profiles. For all part shapes, an imaginary rectangle is drawn around the shape and then the rectangles are then nested.

6. Select Sheets to Nest Parts In

The next tab of the NEST Browser is where you define the size of each sheet and other sheet-related options in the NEST. As we mentioned above the nesting process occurs in the plane of the positive XY quadrant. As shown in the set of buttons below the sheets list, you can Add/Edit Sheet(s) or Select Curve(s). Fior the Select Curve(s) option you need to draw a rectangle whose bottom-left corner is located at the 0,0,0 location and whose sides extend in the direction of the positive X and Y axis quadrants.

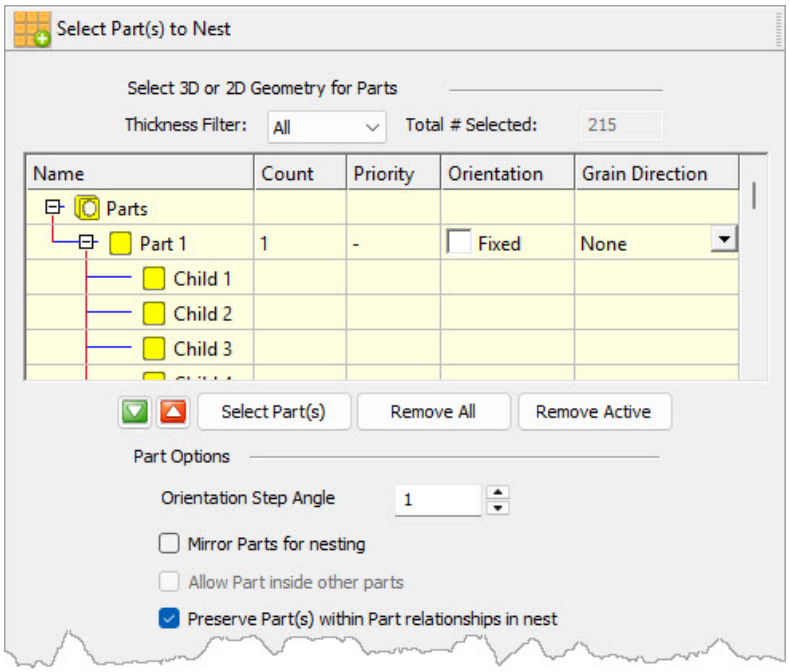


Here we see the Select Sheets to Nest Parts in tab of the Nesting Browser

For this project we are using the Add/Edit Sheet(s) button which then displays the additional fields to define the sheet parameters. At this point, we enter values in the Length, Width and Thickness fields, 96, 48 and 0.750 respectively. We will leave the Count set to 1 because we don't know yet how many sheets are needed and when we do (see step 8 below), it will update automatically! For the Name, and X, Y fields we will accept the default values.

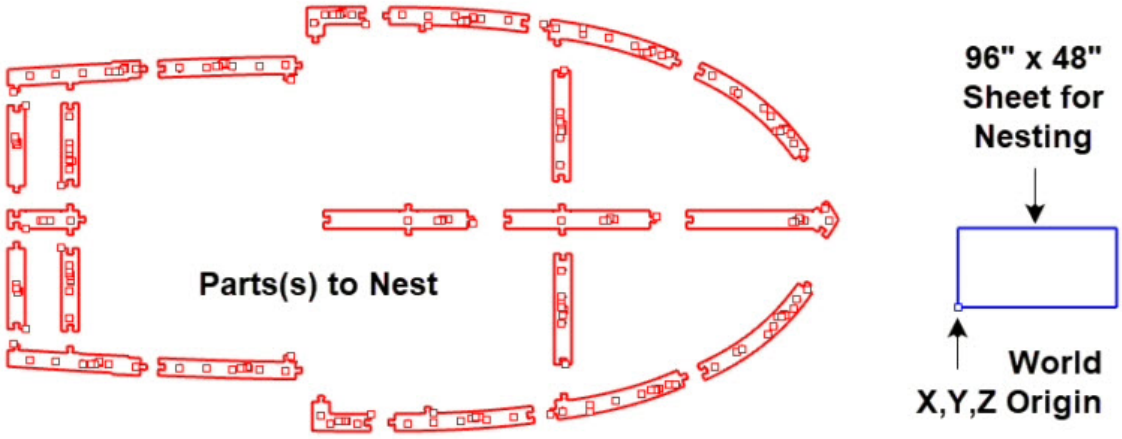
7. Select Part(s) to Nest Tab

This next tab is where we add the parts that we want nested onto the sheets. First, we will pick the Select Part(s) button. We are prompted to select geometry and we window select all of the parts that we have staged and then right-click or press Enter. When we do, we see that all of the parts get added to the table. You will notice from the list that each part has Child cutouts and each cutout is added to the list as a Child under the part they belong to.

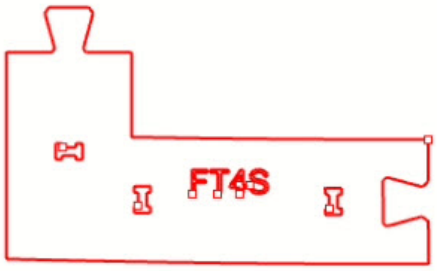


Here we see the Select Part(s) to Nest tab of the Nesting Browser

The Orientation Step Angle parameter under the Part Options section, is a constraint that is applied during nesting. It defines how many degrees of freedom each part can rotate in order to complete the nest. We want minimal constraints so we set it to 1. You will notice that each of our parts have interior cutouts so we want to check the box to Preserve Part(s) within Parts relationship. This will make sure that each part and its cutouts are preserved as one part. The parts Count, Priority, Orientation, and Grain Direction are left to their default values. The part to nest (in red) and the nested sheet (blue) are all shown in the illustration below.



Here we see the Parts to Nest in red and the Sheet to Nest Parts on in blue.



Here we see one of the parts in the nest.

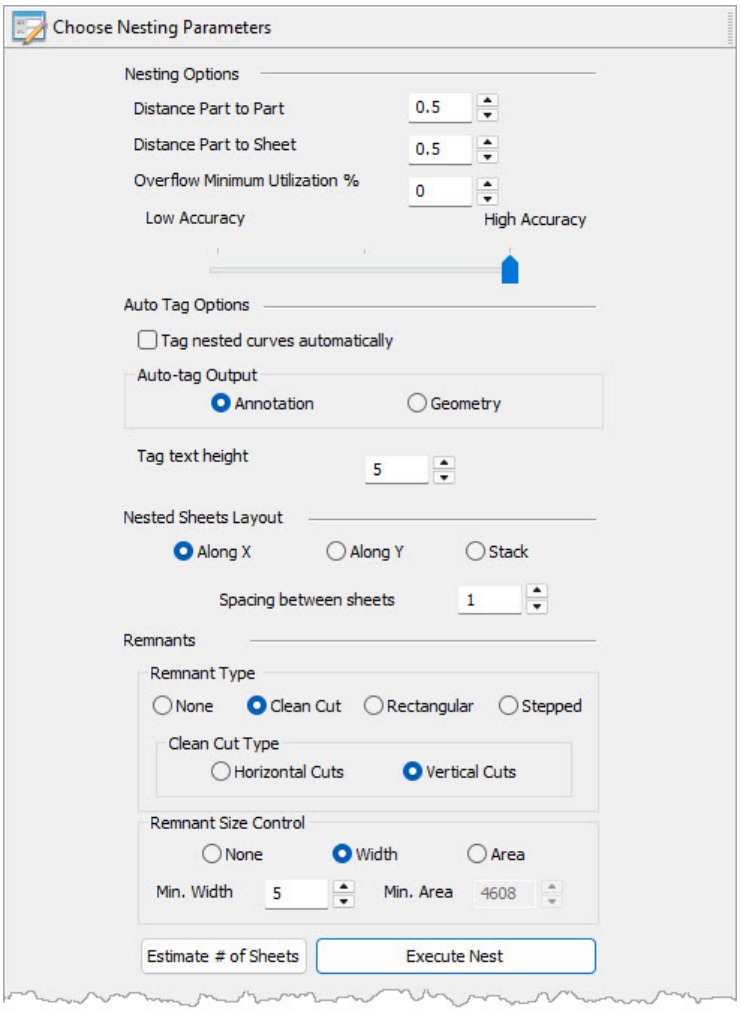
Notice that it has a closed perimeter curve and that it has a number of interior cutouts.

One of the parts is shown above. Notice that the perimeter of the part is a closed curve. This is required. Also notice that there are a total of 6 interior cutouts. Also notice that 4 of the cutouts are alpha-numeric characters. For this project this serves to identify the part in the final assembly. “Tagging” can also be done automatically during nesting.

8. Choose Nesting Parameters

Next up is the Choose Nesting Parameters tab that contains most of the parameters that you can adjust to influence the resulting nested sheet(s). We will list these parameters and their values below. Most are self-explanatory:

- Distance Part to Part: 0.5
- Distance Part to Sheet: 0.5
- Tag Height: default value (not used in this project)
- Nested Sheet Layout: Along X
- Spacing between sheets: 1
- Remnant type: Clean Cut (explained below)
- Remnant, Clean Cut Type: Vertical Cuts (Explained below)
- Remnant Size Control: Width (Explained below)
- Remnant Min. Width: 5 (Explained below)



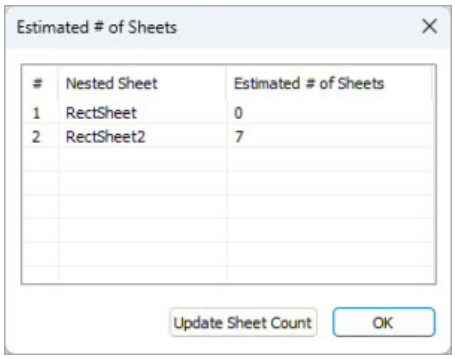
Here we see the Choose Nesting Parameters tab. Notice the section for Remnant Control.

9. Understanding Remnants

In many cases, there will be areas of each nested sheet that are not being used. This can be due to the shape of your parts, how many parts are in the nest and how each part is positioned. RhinoCAM NEST allows you to reuse these sheet remnants. The Remnants section of the Choose Nesting Parameters tab contains all of the options to control your remnant sheets. With Remnants enabled, the unused portions of each sheet can be reused in subsequent nesting jobs.

10. Estimating the Number of Sheets

When all of the nesting parameters are to our liking, we now pick the Estimate Number of Sheets button located at the bottom of the Choose Nesting Parameters tab. RhinoCAM-NEST will perform calculations to estimate the total number of nested sheets we will need. Also, the Estimate # of Sheets dialog below also allows us to automatically update the sheet count displayed on the Select Sheet(s) to Nest Parts in tab (shown in 5. above).



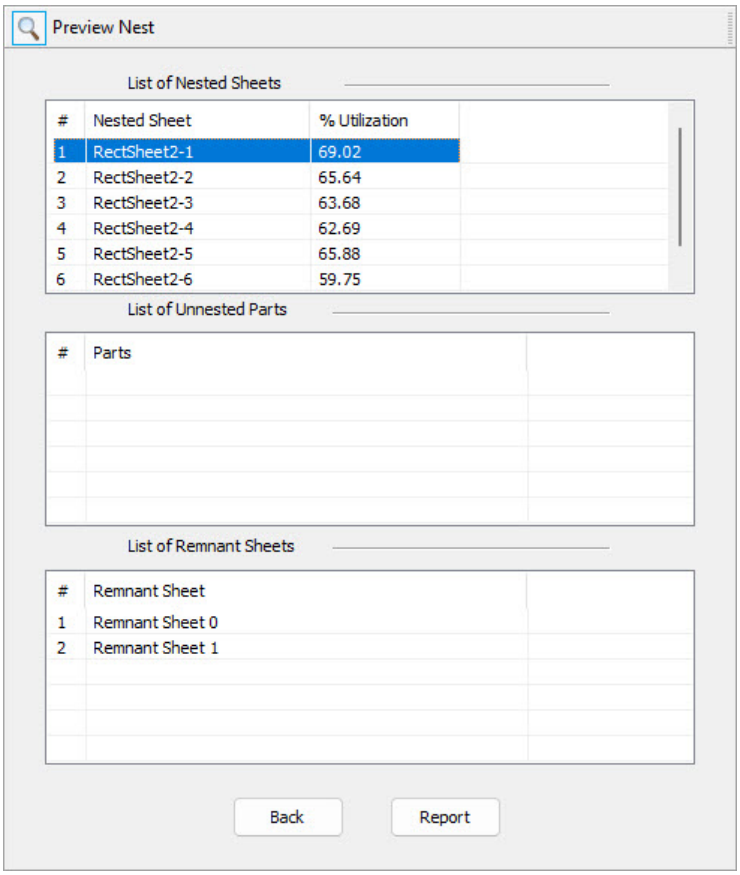
The system estimates the total number of sheets needed. You can also update the Count parameter in the Sheet(s) to Nest Parts in tab (see 5. above)

11. Execute the Nest

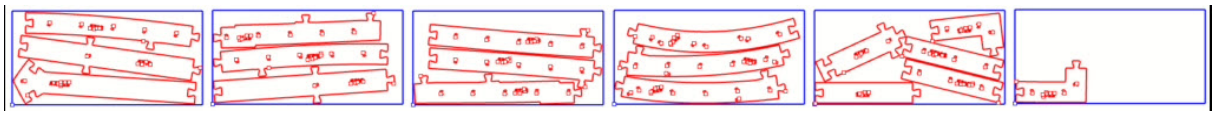
In this step, all of our parameters from the steps above are taken into account, the NEST is executed and then we are moved automatically to the Preview Nest tab of the NEST Browser.

12. Preview Nest

The Preview Nest tab shows three tables. They are, from top to bottom, List of Nested Sheets, List of Unnested Parts and List of Remnant Sheets. When you select a sheet from the List of Nested Sheets, a preview on that nested sheet is displayed. Because we selected “Along X” from the Nested Sheet Layout section of the Choose Nesting Parameters tab (see 7. above), the nested sheets are previewed one at a time beginning from 0,0,0 and extending along the X axis.



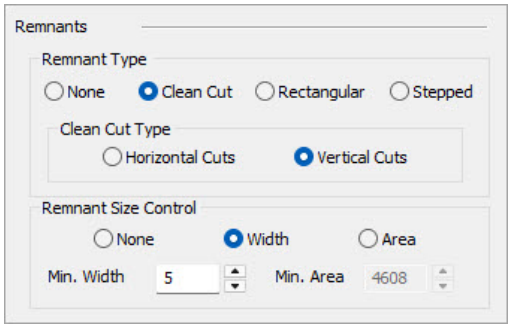
The Preview Nest tab lists the Nested Sheet, Unnested Sheets (if any) and the Remnant Sheets (if Remnants are enabled)



Here we see a preview of all 7 of the nested sheets.

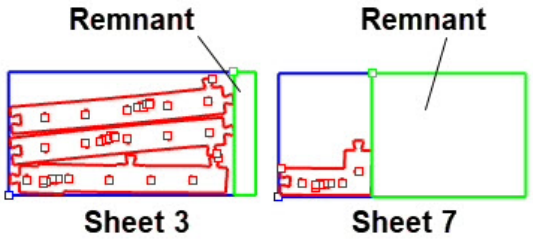
13. List of Remnant Sheets

If you recall, we enabled the Remnants section of the Nesting Parameters tab (see 8. above), and set the Remnant Type to Clean Cut, we set the Clean Cut type to Vertical Cuts, we set the remnant Size Control to Width and the Minimum Width to 5. The Remnants section of the Nesting Parameters tab is shown again below.



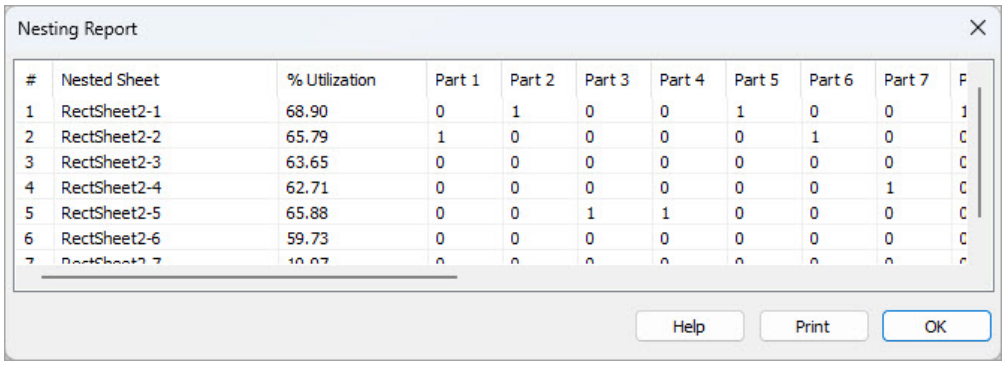
Here we see the Remnant parameters we set from the Nesting Parameters tab.

Now, from the Preview Nest tab we see that there are two Remnant Sheets listed in the List of Remnant Sheets. Selecting each Remnant Sheet from the list will poreview that sheet. For better understanding, the preview geometry is color-coded. The sheet perimeter is BLUE, the nested parts are RED and the Remnant is GREEN. This is shown in the illustration below.



14. Nesting Report

From the Preview Nest tab you can select the Report button to see a nesting report. The report lists each sheet in the nest and where in the nested sheets each part is located. The Nesting Report is shown below.

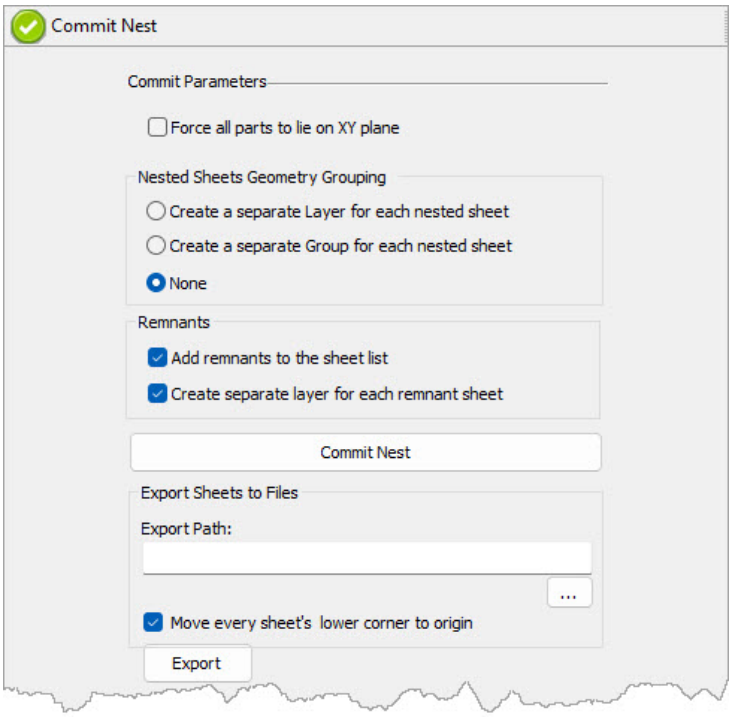


#	Nested Sheet	% Utilization	Part 1	Part 2	Part 3	Part 4	Part 5	Part 6	Part 7
1	RectSheet2-1	68.90	0	1	0	0	1	0	0
2	RectSheet2-2	65.79	1	0	0	0	0	1	0
3	RectSheet2-3	63.65	0	0	0	0	0	0	0
4	RectSheet2-4	62.71	0	0	0	0	0	0	1
5	RectSheet2-5	65.88	0	0	1	1	0	0	0
6	RectSheet2-6	59.73	0	0	0	0	0	0	0

The nesting report shows the sheets where each nested part is located.

15. Committing the Nest

After you have previewed the nest and are sure that it is to your liking, select the Commit Nest tab. The term “Commit Nest” means that once the nest is committed, you cannot undo or change any parameters. To change any parameters, you must start over with the nesting process.



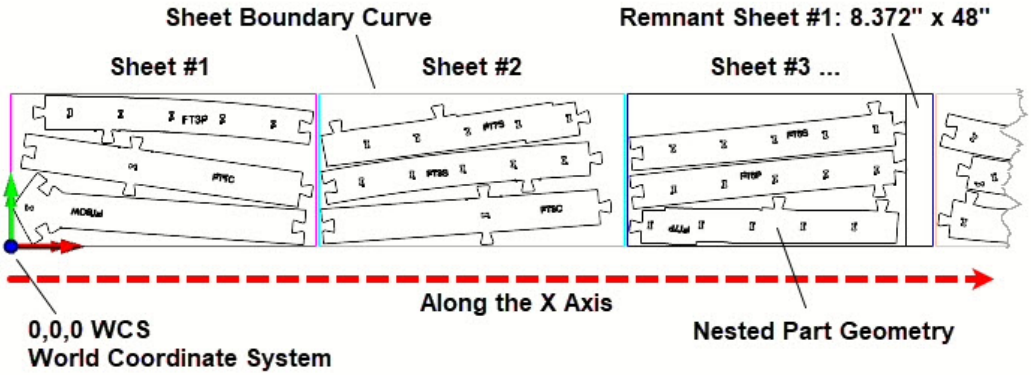
The Commit Nest tab includes parameters for committing the nested sheets to actual geometry that can then be used for other purposes inside Rhino such as generating toolpaths in RhinoCAM.

The parameters on this Commit Nest tab are pretty self-explanatory and we will not go into much detail. We do however, want to discuss the Remnants section of the dialog. Since we have identified 2 sheets with remnant material, we want to decide how those remnants are handled when the nest is committed. By default each remnant will be defined by creating a closed perimeter boundary. The closed perimeter curve geometry of these two remnants will then be added back into the sheets list on the Select Sheets to Nest Parts in tab (see 6. above). We also want each remnant geometry on a separate layer.

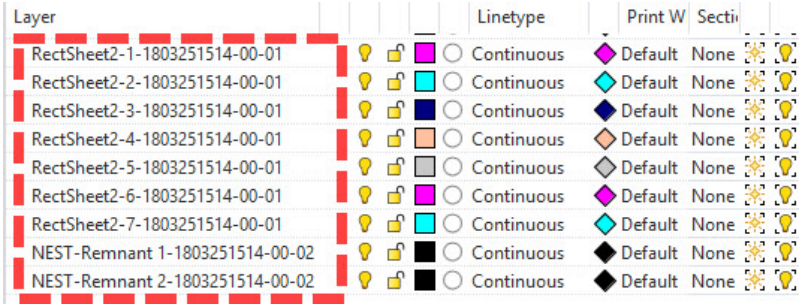
16. The Commit Nest Button

OK, all of our nesting parameters are set and now we pick the Commit Nest button. Each nested sheet is converted into actual Rhino geometry. Here is exactly what happens:

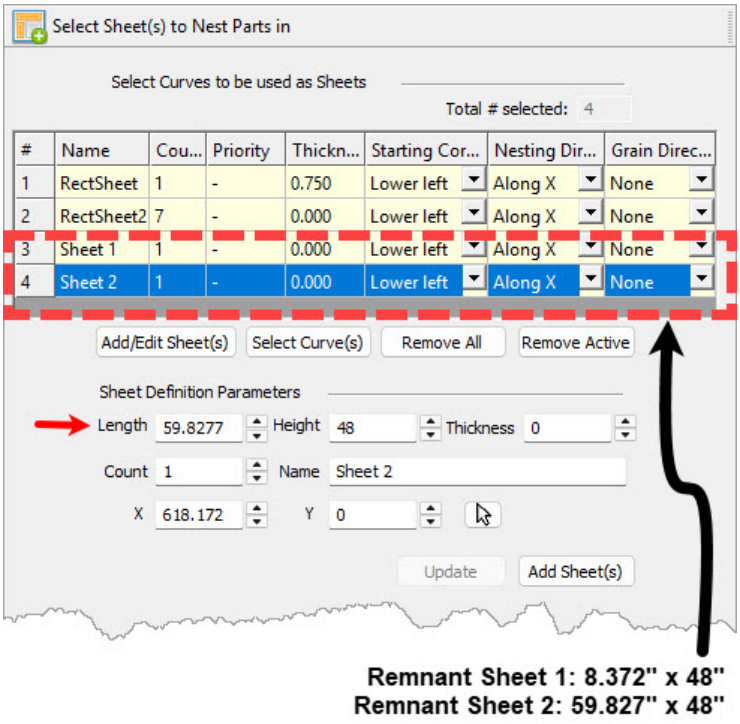
1. Each sheet is defined by a closed rectangle.
2. Each nested part geometry is added to its designated sheet.
3. Each sheet is placed side-by-side along the X axis.
4. Layers are created for each nested sheet and each remnant sheet.
5. The two remnant sheets are added to the Sheets list.



Above we see the first three sheets of the committed nest. The first of two remnant sheets are is shown on Sheet #3. The nested sheets traverse along the positive X axis.



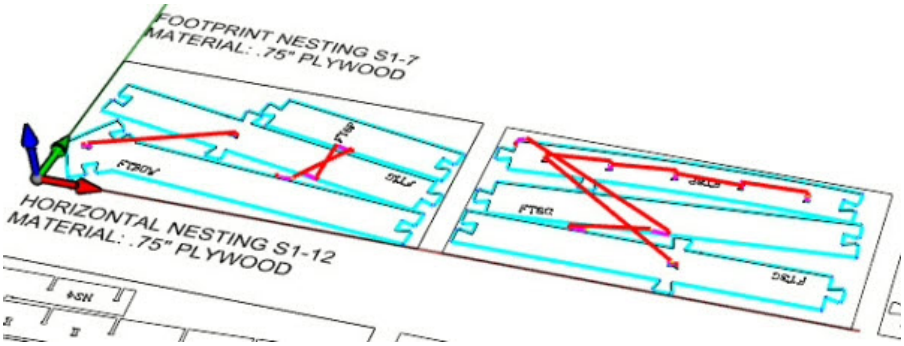
Above we see the Rhino layers that were created to contain the nested sheets. You will see that there is a layer for each of the seven nested sheets and that two sheets were created that each contain a remnant sheet.



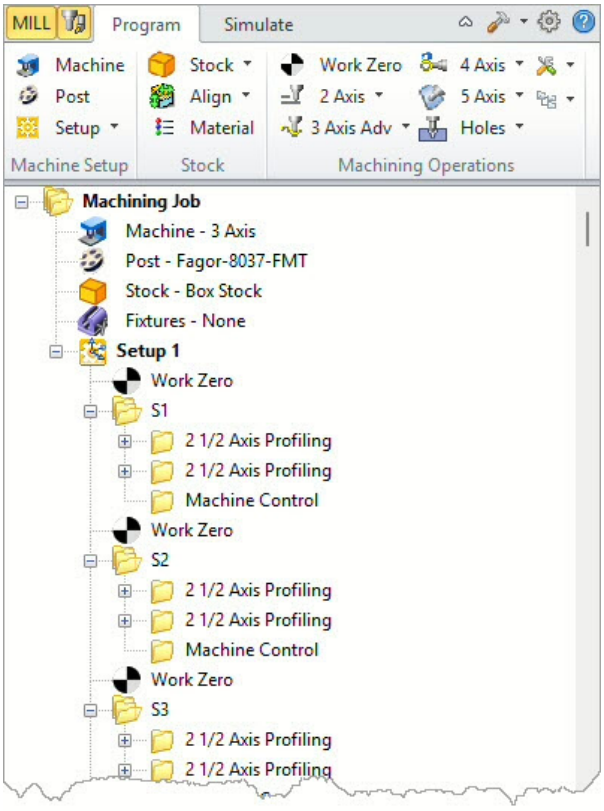
Above we see the Select Sheet(s) to Nest Parts in tab. Notice that the two remnant sheets were added into the Sheets list.

MILL Module

Once the nested sheets are generated, the RhinoCAM Milling module is used to generate and simulate the optimized 2½ Axis tool paths and custom G-Code needed to operate companie's 12'x5' C.R. Onsrud Panel Pro CNC machine. A sample Machining Job on nested sheets is illustrated in the images below.



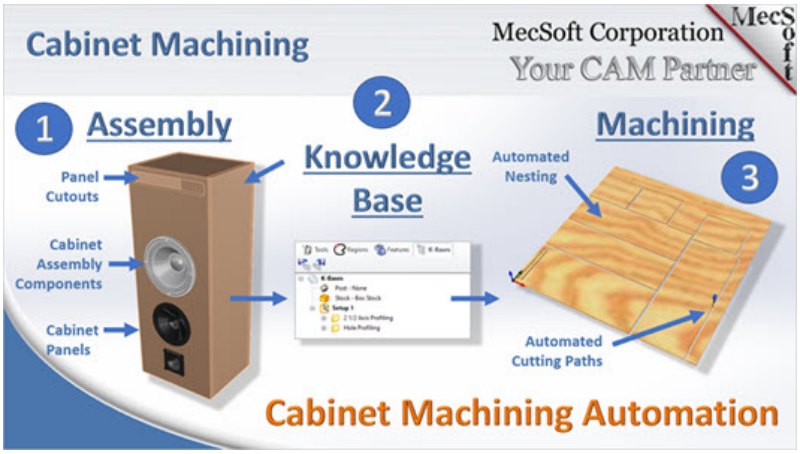
Above we see just a sample of nested sheets and CNC tool paths generated from the RhinoCAM NEST and MILL modules.



Above we see the RhinoCAM Machining Browser with the 2½ Axis tool paths generated for multiple nested sheets (S1, S2, S3, etc..).

Nesting & KBs in Practice

here we will review the 3-step process of automating the toolpath generation of a nested sheet of flat panel components. For this discussion will use the Speaker Cabinet assembly shown in the info-graphics below. We have created some videos of this process using the same speaker cabinet assembly. The videos are noted in each section below.

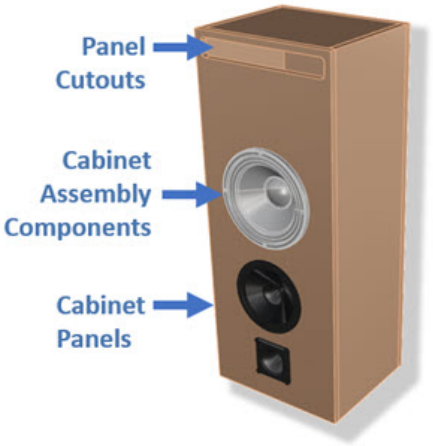


Phase 1:

Automated Cabinet Layout and Nesting

Cabinet Flat Panel Layout

In this phase the Flat Panel Layout utility included in RhinoCAM MILL Module (or VisualCADCAM) is used to decompose the speaker cabinet assembly and lay each panel out flat in preparation for nesting the components onto a sheet of stock material. In the illustrations below, the panels are laid out along the X axis. Notice that there are interior cutouts on some of the panels.



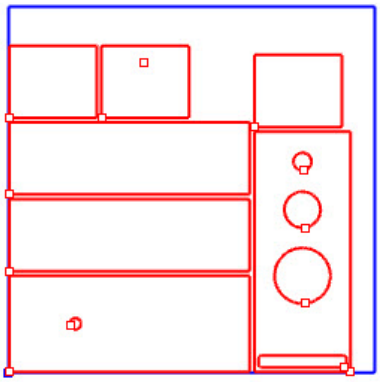
Above we see the speaker cabinet assembly.

The RhinoCAM Flat Panel Layout utility decomposes the speaker cabinet assembly and lays each panel flat onto the XY plane.

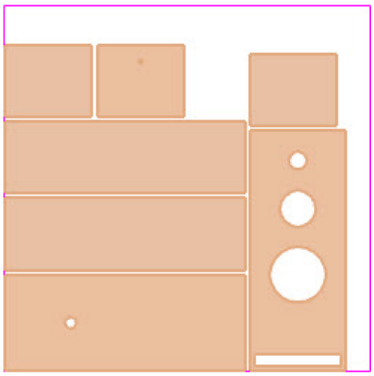
[Watch the Flat panel Layout Utility]

Cabinet Nesting onto a Sheet

Now that the cabinet panels are laid out flat on the XY plane we can load the RhinoCAM NEST Module. From here we can define the stock sheet as 60" x 60" x 0.75" and then add the component parts and quantities that we want included in the nest. There are many other parameters you can adjust if needed, such as the distance between component parts on the nested sheet. We can then Execute the Nest to see a preview of the nested sheet (bottom left image). Once we are satisfied we can Commit the Nest to generate the nested components (bottom right image)



A preview of the nested sheet



The nested sheet completed

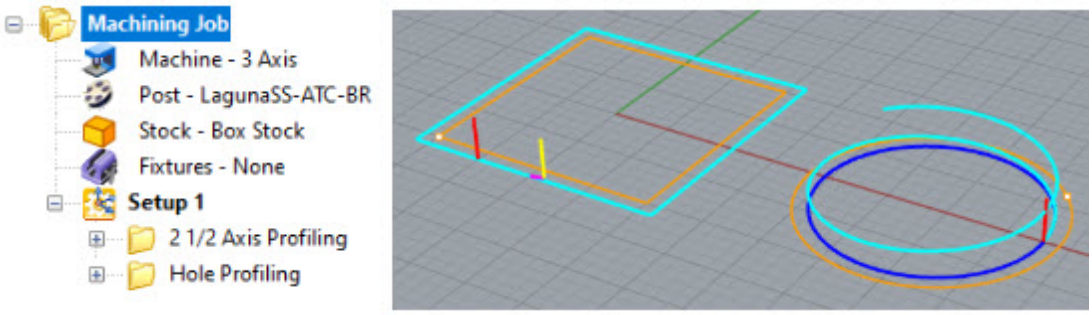
[Watch the Nesting of Components]

Phase 2:

Automated Knowledge Base of Machining Operations

Create the Toolpath Operations for the Knowledge Base

Now that we have a sheet containing all of the panels in our speaker cabinet, we can move on and create a knowledge base of the machining operations that we wish to use to machine the sheet. A review of the panels shows us that they contain both square and round interior cutouts. To cut these panels we will use one 2 axis Profiling operation for the square cutouts and one 2 axis Hole Profiling operation for the round cutouts.



These toolpath operations are used when cutting our sheet of nested components so we need to make sure they represent the exact way that we wish to machine, including cutting parameters, cut level parameters, entry/exit, etc. It's a good idea for us to simulate and review each toolpath operation before continuing further. While you can edit these toolpaths once they are added to the knowledge base, it is good to get it right the first time.

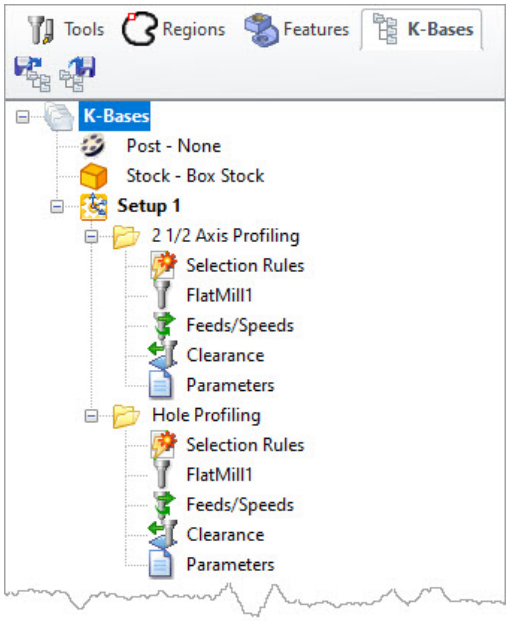
Watch the Creation of the Knowledge Base

Set the Selection Rules for the Knowledge Base

Once we're confident that each operation is correct to our liking, we can save the operations to a Knowledge Base KB file using the Save to KB command on the Program tab.

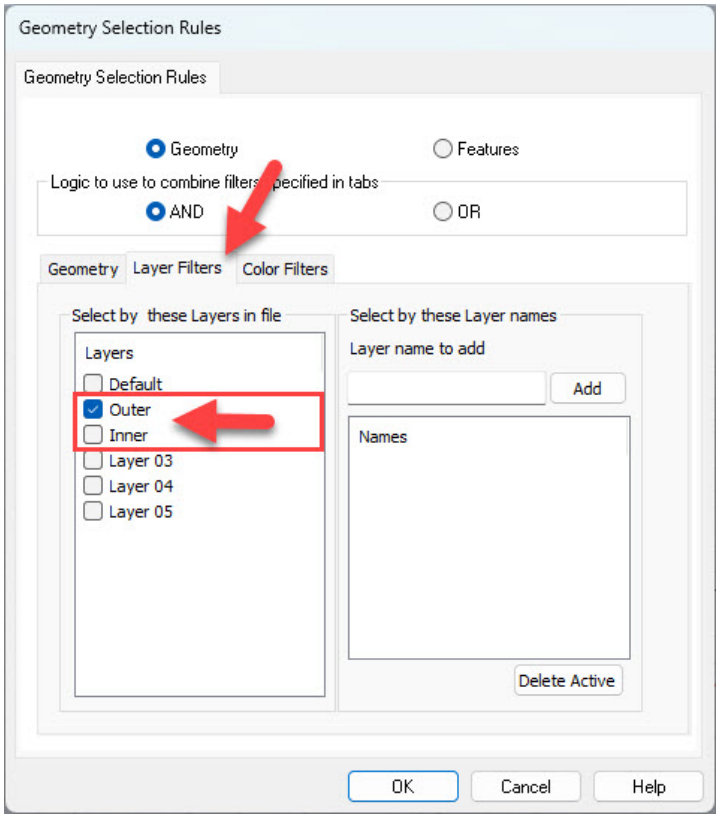
Now it is time to add the selection rules to each of our two operations in the Knowledge Base. This step will allow the operations to generate successfully once the knowledge base is loaded into the RhinoCAM Machining Job of our nested sheet file.

First we will load the Knowledge Base file into the K-Bases tab of the RhinoCAM Machining Objects Browser. This is shown here on the right. Notice that there is a flag next to the Selection Rules icon under each operation.



Each operation in the Knowledge Base can be set to automatically retrieve control geometry. This automated selection can be made by Geometry type, Layer name, Color or Feature type. In this case we will be selecting control geometry from two specific layers named “Inner” and “Outer”. First we create the two named layers in our Rhino file. Then we click on the flagged Selection Rules icon under each operation. This will display the Geometry Selection Rules dialog shown below. In the dialog, select the Layer Filters tab and you the two named layers that we created. For the 2 axis Profiling operation we will select the layer named Outer. Then for the 2 axis Hole profiling operation we will select the layer named Inner.

It is important to take a moment here and check to make sure your layer names are correct and that each layer is assigned to a specific operation in the Knowledge Base. Pick OK when you are done to close the dialog.



Phase 3:

Automated Sheet Machining

If you have stayed with us this far, congratulations! We’re now ready to automatically generate the toolpaths needed to machine our sheet of nested speaker component panels.

What We have Done So Far

Let’s take a moment to review what we have done so far:

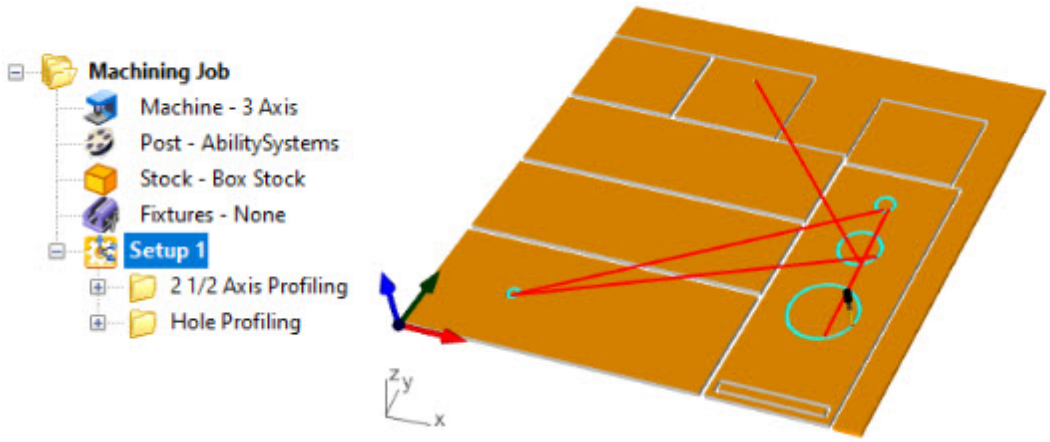
1. We used the Flat Panel Layout utility in RhinoCAM MILL (or VisualCADCAM) to decompose our speaker cabinet assembly onto the XY plane.

2. We used the RhinoCAM NEST module to nest the speaker panels onto a 60” x 60” x 0.75” sheet.
3. We created a Knowledge Base of the toolpath operations we wish to apply to our sheet of nested components.
4. We assigned selection rules to each operation in the Knowledge Base to automatically select from two existing layers

What Happens Automatically:

When loaded, the Knowledge Base file automatically populates the Machining Job with two operations. These are: 2 Axis Profiling and 2 Axis Hole Profiling.

When Setup 1 in the Machining Job is generated, the knowledge contained within our two operations automatically grabs the geometry from the correct layers and successfully regenerates the operations and displays the resulting toolpaths on the graphics screen.



Now consider this process being applied to a larger project with hundreds of nested components and you begin to see the benefits of Knowledge Base machining with RhinoCAM (or VisualCADCAM)!

Watch the Cabinet Machining Automation

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