TUTORIAL

[Solid Modeling] Hybrid Design
June 2012

RAPIDWorks

RapidWorks User Guide & Tutorial

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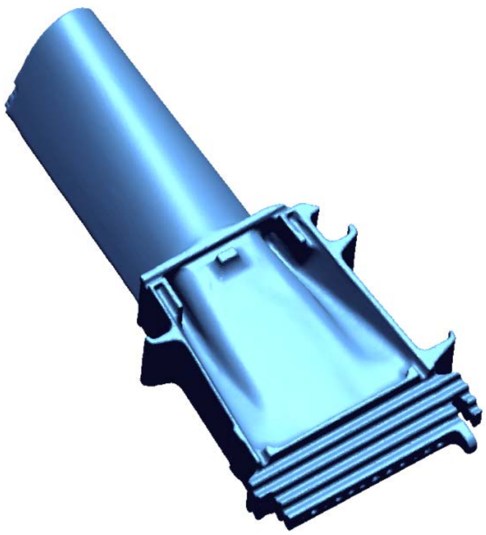
1. Introduction

This tutorial is intended for users who need to become quickly familiar with RapidWorks.
Before getting into the detailed instructions for using RapidWorks, this step-by-step tutorial aims at giving you a feel for what you can accomplish with the product.

This tutorial will guide you on how to completely design an industrial part by using powerful solid modeling methods. You will learn modeling methods that can use to create a solid body from 3D scan data. You will also learn how to transfer the fully designed model to a dedicated CAD program



Scan Data

Designed Model

Training time required : **approx. 90 min.**
Level of Difficulty : **Advanced**

2. Data Files

Hybrid_Design.rwl – preset modeling data of a turbine blade model.
[Download](http://nextwiki.s3.amazonaws.com/resources/application/octet-stream/536deaa0a2d8012f2eb800254b9c869c-pqf7co11v5sbc.rwl): http://nextwiki.s3.amazonaws.com/resources/application/octet-stream/536deaa0a2d8012f2eb800254b9c869c-pqf7co11v5sbc.rwl

The sample data for this tutorial is provided by **INUS Technology**. It is the property of INUS Technology and is used for informational purposes only. Other than to identify this software and publication, individuals or organizations purchasing the software are not entitled to use the sample data without INUS Technology's prior written consent.

3. Overview

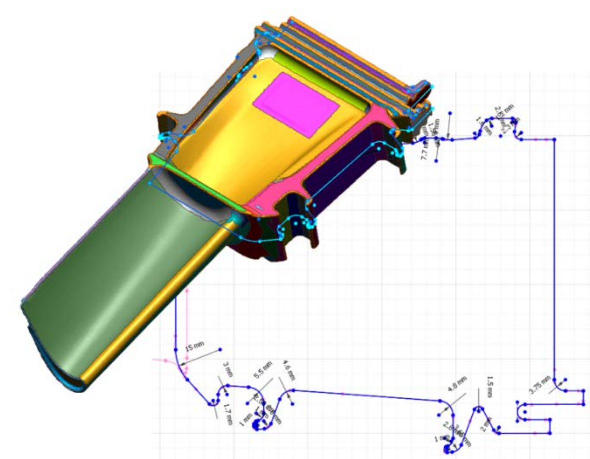
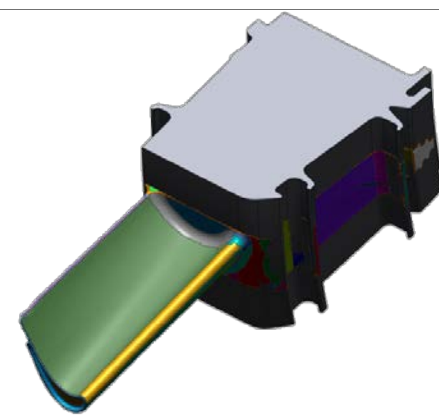
What will you learn in this course?

- Design 2D sketch profile for creating a root part of a turbine blade model
- Create a root part of a turbine blade model by using the created 2D sketch profile
- Create a vane part of a turbine blade model by using Loft surfacing method
- Add features
- Transfer a final model with modeling history to CAD system

What will you learn to do in this course?

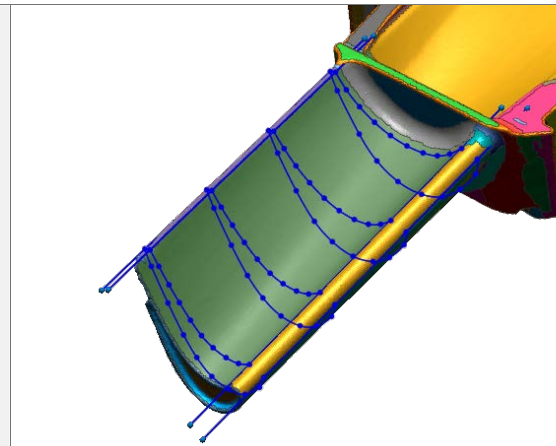
- You can extract 2D sketch profiles from a scan data.
- You can create a solid body by using the profiles for a root part of a turbine blade model.
- You can create a vane part of a turbine blade model from several sketch profiles.
- You can add more features onto the solid body.
- At the end of modeling process, you can easily check the deviation between the designed model and the scan data.
- Finally, you can transfer the final model with the whole modeling history at CAD program.

What does this exercise cover?

Step 1. Design 2D Sketch Profile You can design a 2D sketch profile for creating a root part of a blade model.	
Step 2. Create Root Part of Blade Model You can create a solid body by using the 2D sketch profile for a root part of a blade model.	

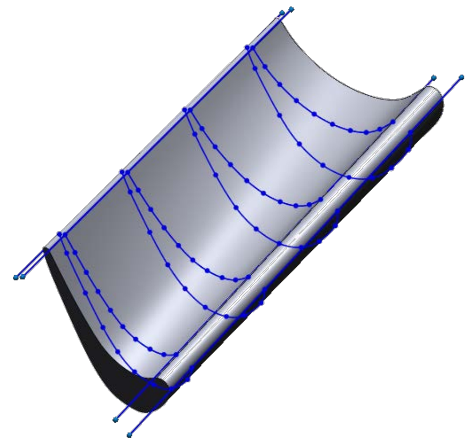
Step 3. Extract Feature Line and Design 2D Sketch Profiles

You can extract feature lines from the scan data and design 2D sketch profiles for creating a vane part of a blade model.



Step 4. Create Vane Part of Blade Model

You can create a vane part of a blade model by using Loft surfacing method and then combine with the root part of a blade model.

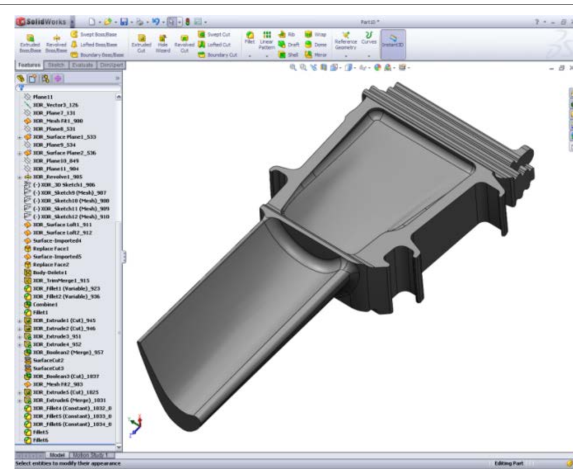


Step 5. Combine Parts and Add More Features

You can add more features onto the blade model.

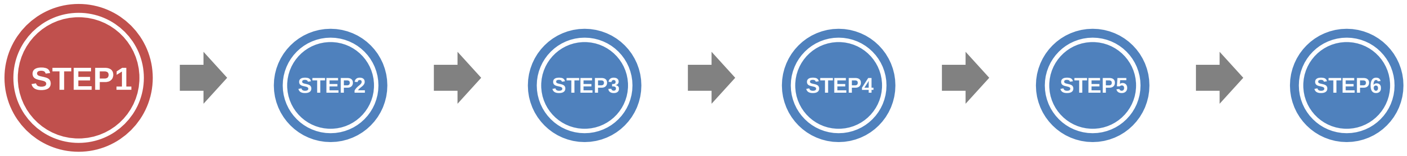
Step 6. Check Result and Transfer Model to CAD Program

Finally, you can check the result and then transfer it with the whole modeling history to the CAD program.

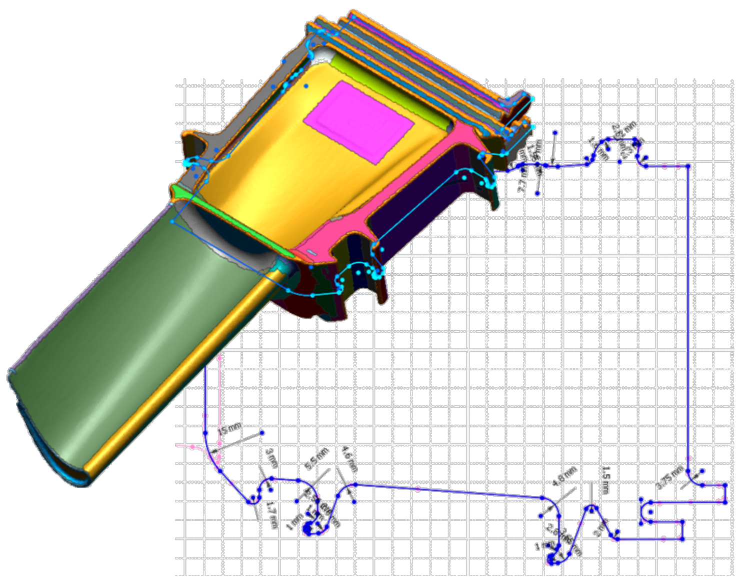


4. Modeling Process

Step1. Design 2D Sketch Profile



A turbine usually has a casing around the blades that contains and controls the working fluid.
The blade consists of a vane part which is thin and twisted and a root part which is fixed on a turbine rotor.
In this step, you will learn how you can extract a 2D sketch profile from the scan data for creating the root part of the blade model.

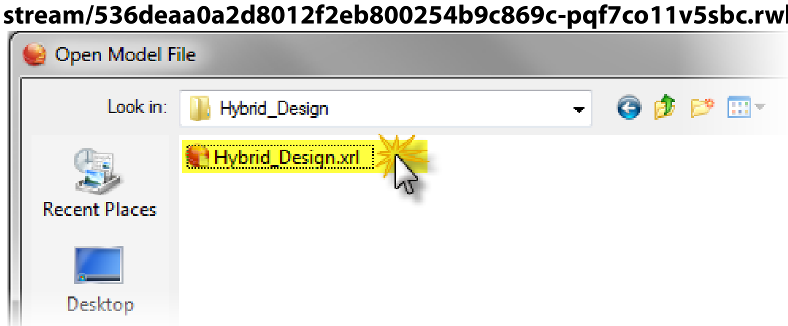


Created Sketch Profile

1. Open File

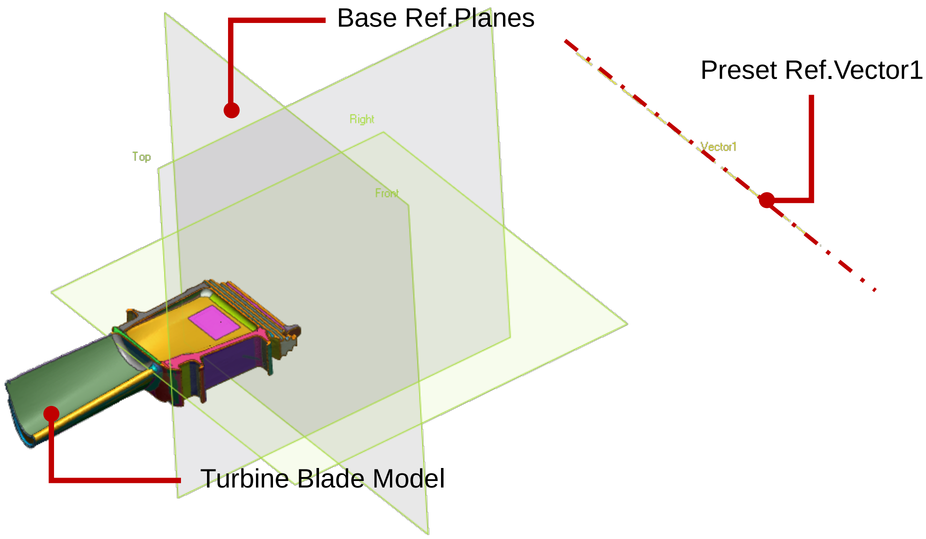
Follow this step by using the **Hybrid_Design.xrl** files.

① Click the **Open** button in the Toolbar or choose **File > Open** in the menu.
② Select the preset RWL data file (**Hybrid_Design.rwl**) and click the **Open** button.
Download:http://nextwiki.s3.amazonaws.com/resources/application/octet-stream/536deaa0a2d8012f2eb800254b9c869c-pqf7co11v5sbc.rwl



Note. Some sketch profiles and surfaces are already preset in this data file. These preset entities will be

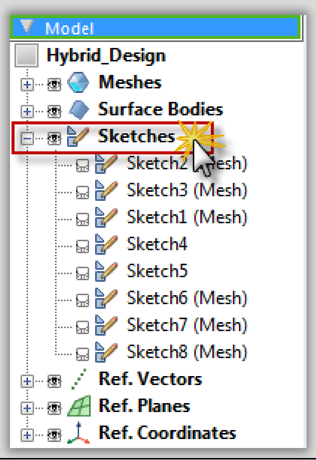
used for completely forming the blade model.
Hide all preset entities except the Ref.Vector 1 and the Base Ref.Planes (Front, Top, and Right) in the Model Tree to clearly see the model.



2. Set Mesh Sketch

You need to design a 2D sketch profile for creating a root part of the blade model. In the first step of the sketching process, you need to set a sketch base plane and define a Section Polyline

Note. *Check that the Sketches root entity is visible in the Model Tree.*
If the Sketches root entity is currently invisible, you may not be able to select the designed sketch entities in the Mesh Sketch or in the Sketch mode.

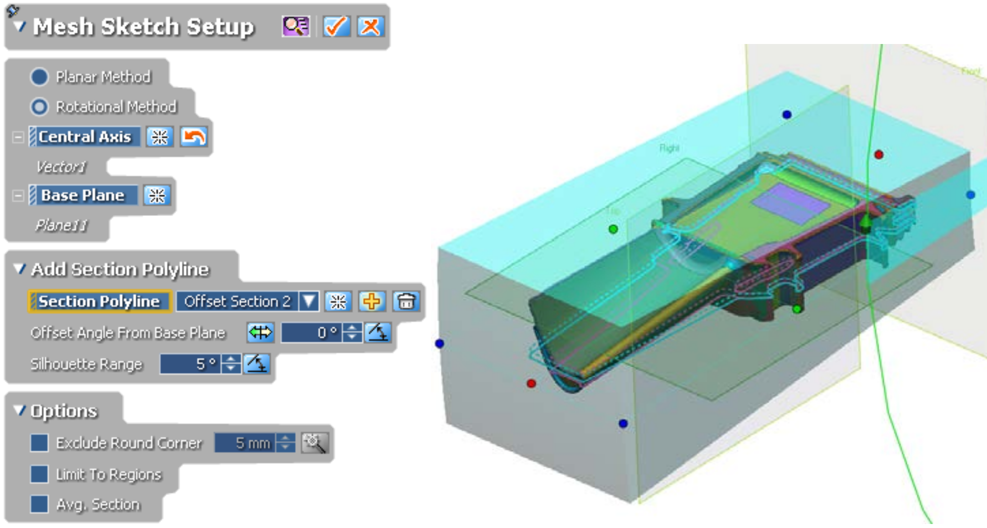


① Click the **Mesh Sketch** button in the Tool Palette to enter the Mesh Sketch mode.

② Select the **Rotation Method** and then select the vector (Vector1) as the target Central Axis and the preset Plane11 as the target Base Plane.
③ Click the **Add Section** button to register the current Offset Section as **Section Polyline**.
④ Check that the Section Polyline is currently set to **Offset Section2**.
⑤ Increase the value in the **Silhouette Range** to **5°**.

Note. *The silhouette range option helps you to easily extract silhouette of the target scan data.*
The silhouette of the target scan data will be projected onto the base plane and it will be used as Section Polyline.

⑥ Check that the Section Polylines are correctly projected onto the base plane.



⑦ Click the **OK** button.

Note. *You are sent automatically to the Mesh Sketch mode and now you are ready to design a sketch profile. The selected Ref.Plane will be automatically hidden for better sketch view. If you want to see the Ref.Planes, toggle the Eye icon of Ref.Planes entity on in the Model Tree*

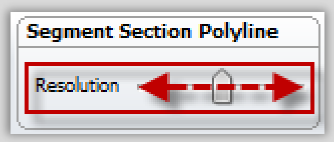
3. Design Sketch Profile

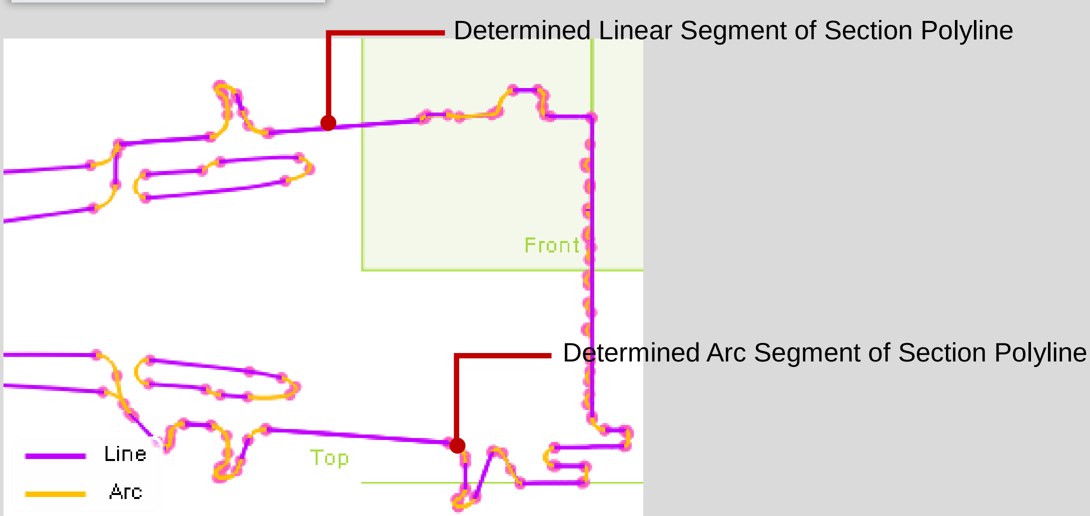
The Section Polyline consists of line and arc segments.

In the second stage of the sketching process, you can design line and arc sketches on the Section Polyline.

① Hide the scan data in the Model Tree.

Tip. *You can adjust resolution of the section polylines by moving the slide bar in the Segment Section Polyline tool palette.*
When you move the slide bar, the application automatically determines linear segment and arc segment in the Section Polyline and then you can easily fit sketches on the determined Section Polyline.

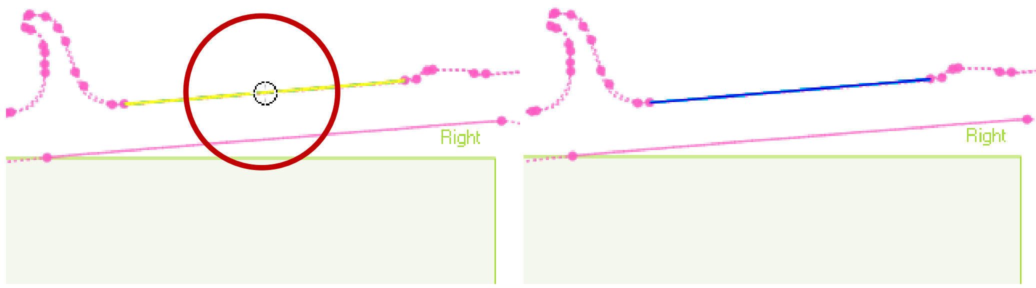




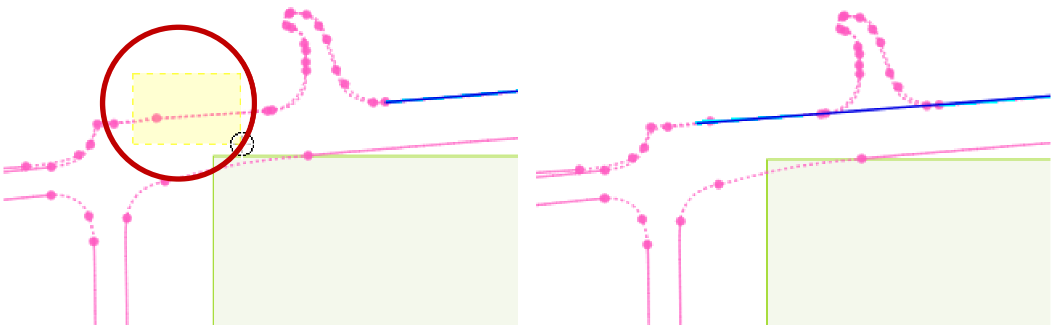
- **Design line sketch profiles**

① Click the **Line** in the Toolbar or choose **Tools > Sketch Entities > Line** in the menu.

② Check that the **Fit Polyline** option is enabled and then select the linear segment of the Section Polyline to create a line sketch on the Section Polyline, as shown in the image below.



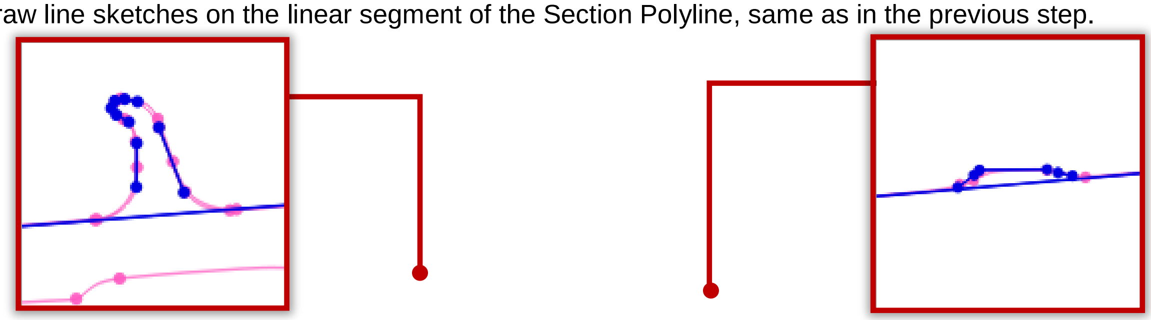
③ Additionally select adjacency the linear segment of the Section Polyline by dragging the mouse cursor, as shown in the image below.

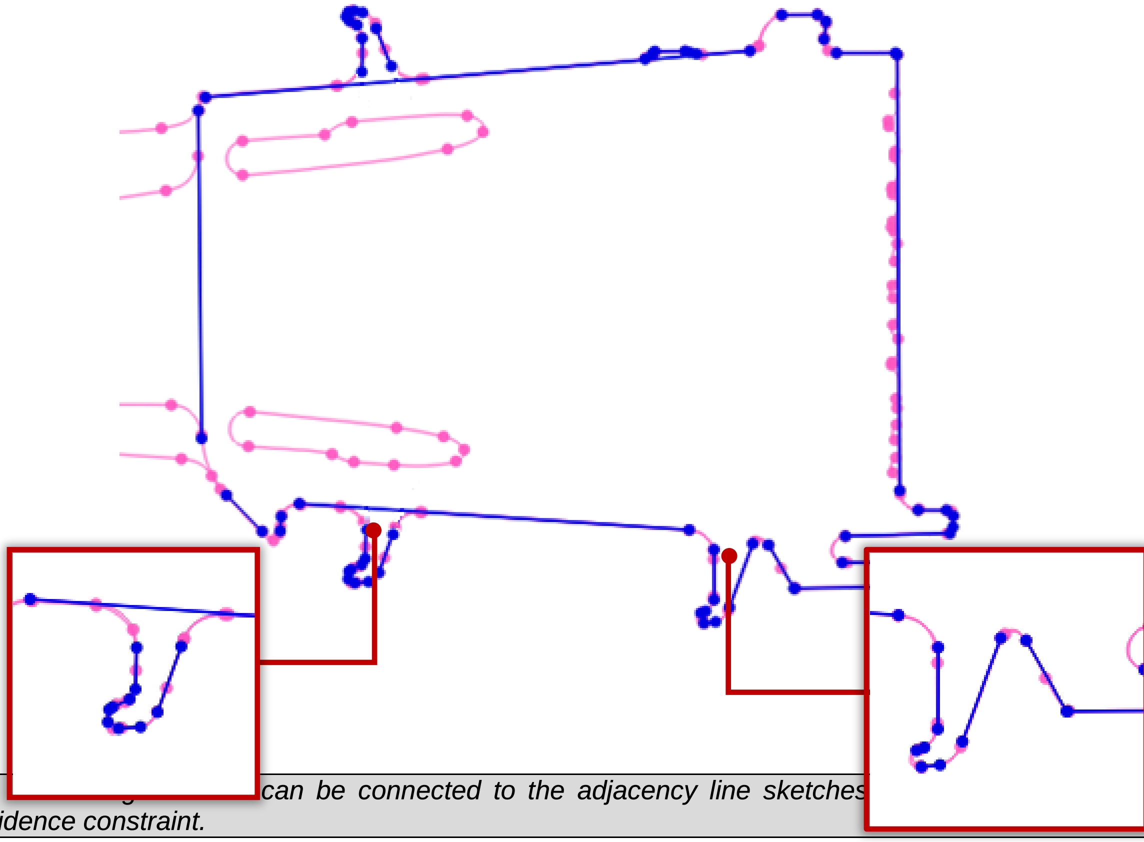


④ Click the **Accept Fitting** button to register the previewed sketch.

> ***Tip.*** *You can easily register the previewed sketch by clicking the Accept Fitting button or double-click the left mouse button on the Model View without leaving the command.*
> *If you click the OK button, you will leave the command.*

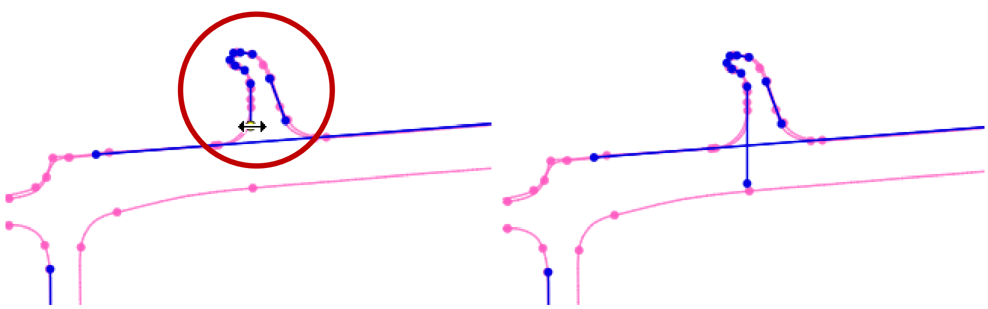
⑤ Draw line sketches on the linear segment of the Section Polyline, same as in the previous step.





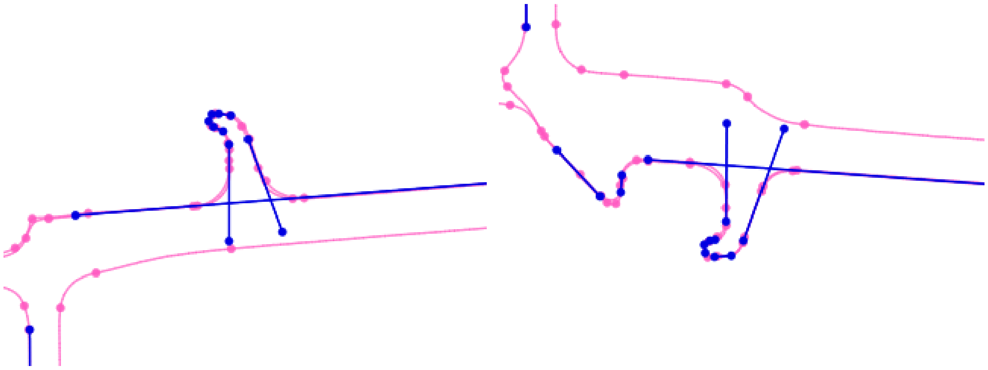
Note. ... can be connected to the adjacency line sketches ... nd coincidence constraint.

⑥ Click the **Resize** in the Toolbar or choose **Tools > Sketch Tools > Resize** in the menu.
⑦ Select the end point of the designed line and then increase the length of the line by dragging the point to the point of inner boundary of the other line sketch, as shown in the image below.

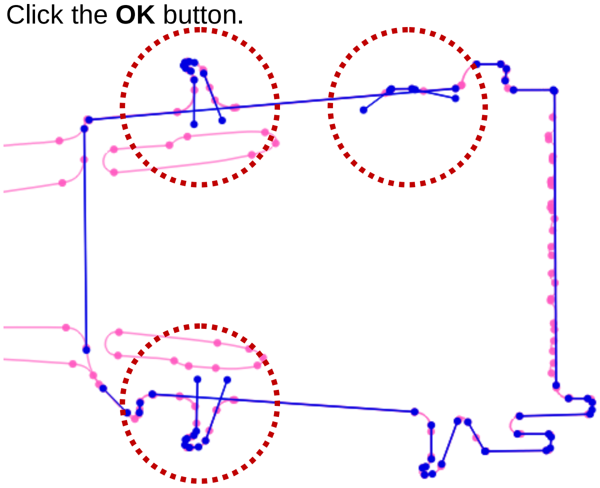


Note. *Check that the Sketches root entity is visible in the Model Tree.*
If the Sketches root entity is currently invisible, you may not be able to select the designed sketch entities in the Sketch mode or in the Mesh Sketch mode.

⑧ Increase the length of the other lines, same as in the previous step.

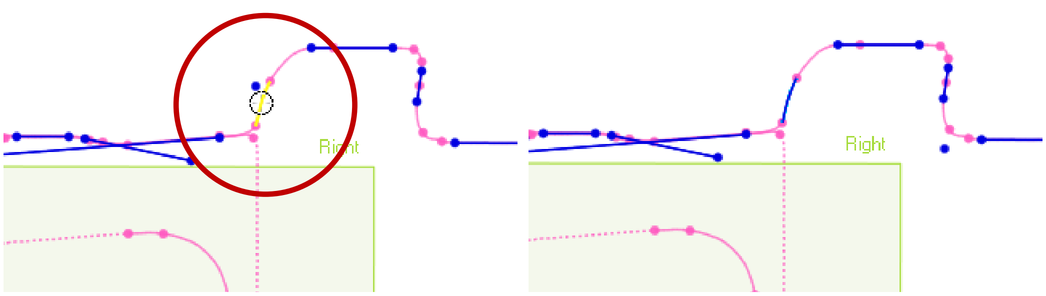


⑨ Click the **OK** button.



- **Design arc sketches**

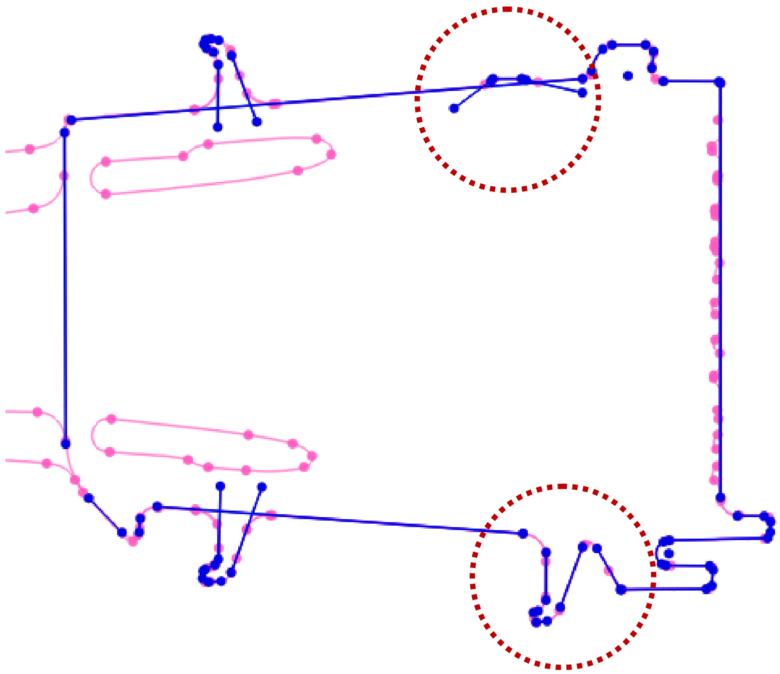
① Click the Centerpoint Arc in the Toolbar or choose Tools > Sketch Entities > Centerpoint Arc in the menu.
② Check that the **Fit Polyline** option is enabledZ and then select arc segment of the Section Polyline to create an arc sketch on the Section Polyline, as shown in the image below.



③ Click the **Accept Fitting** button to register the previewed sketch.

> ***Tip.*** *You can easily register the previewed sketch by clicking the Accept Fitting button or double-click the left mouse button on the Model View without leaving the command.*
> *If you click the OK button, you will leave the command.*

④ Draw arcs on the arc segment of the Section Polyline, same as in the previous step.
⑤ Click the **OK** button.



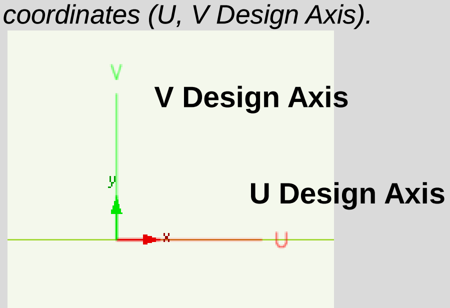
4. Join Sketches

So far you have designed sketches.

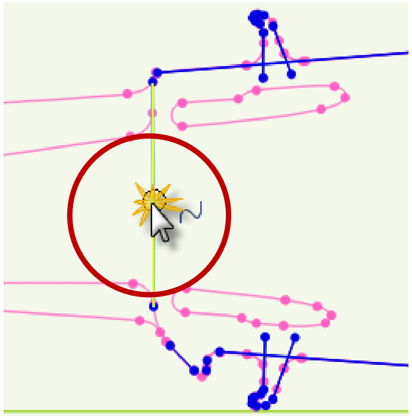
In the third stage of the sketching process, now, you can join the designed line and the arc sketches.

- **Set constraints**

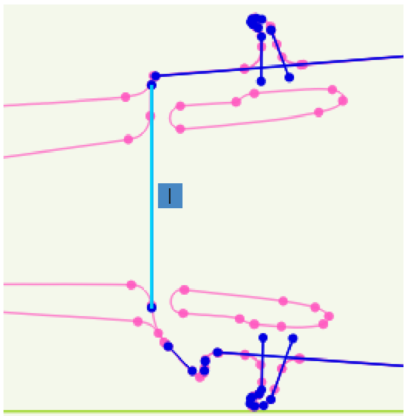
> ***Note.*** *The sketches have been generated by fitting operation so the sketches may not have the correct coordinate relationship with the design coordinate.*
> *If you set constraints (Vertical or Horizontal constraint) in the sketches, you can fit them to a design coordinates (U, V Design Axis).*



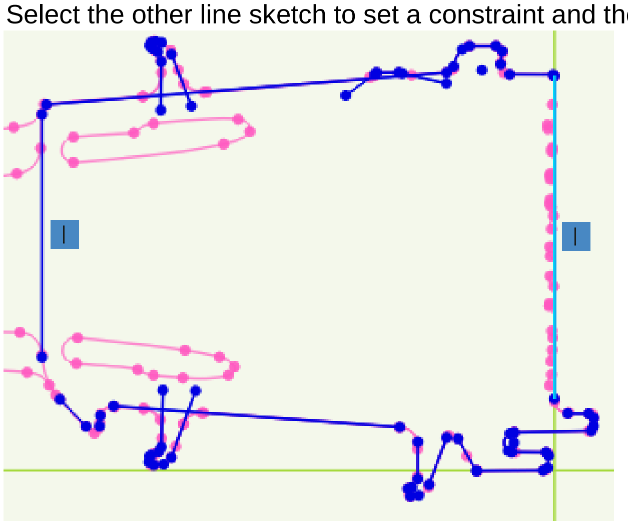
① Double-click the line sketch, as shown in the image below.



② Click the **Vertical** button in the Constraint section to fit the sketch to V Design Axis.

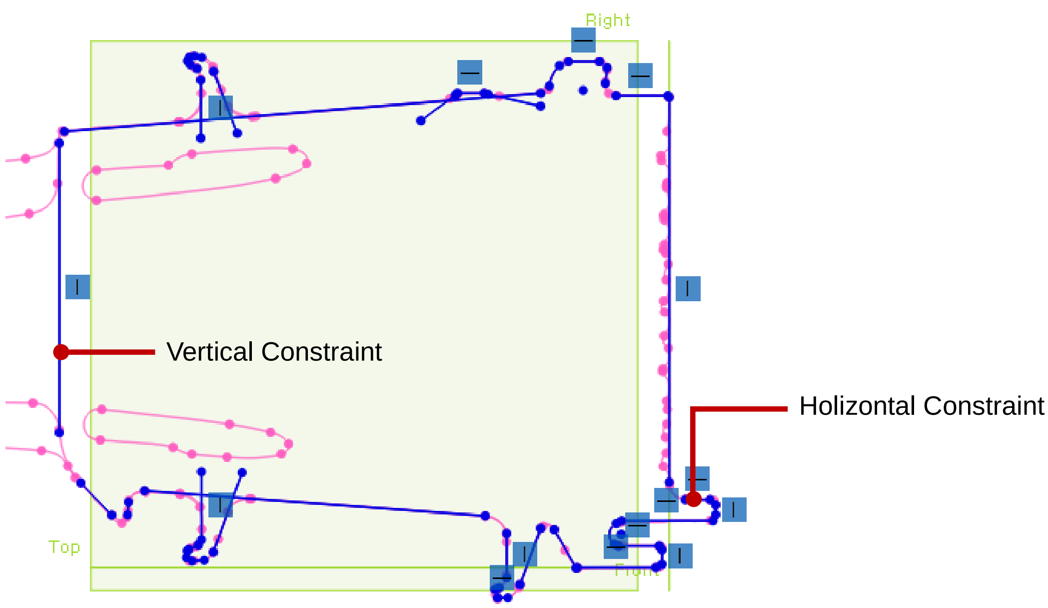


③ Select the other line sketch to set a constraint and then click the **Vertical** button, as shown in the image below.

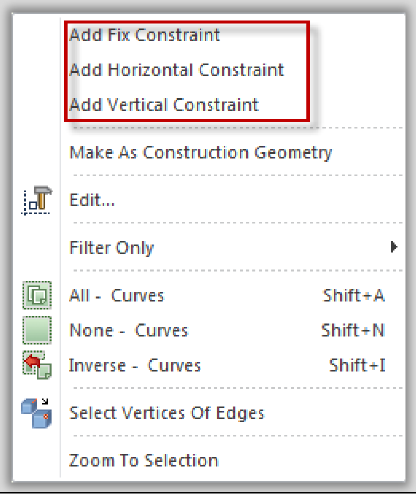


Tip. *If you also want to set a constraint to the other sketches after setting a constraint, just click on the target sketch. You can easily set a constraint to the sketch without exiting the menu.*

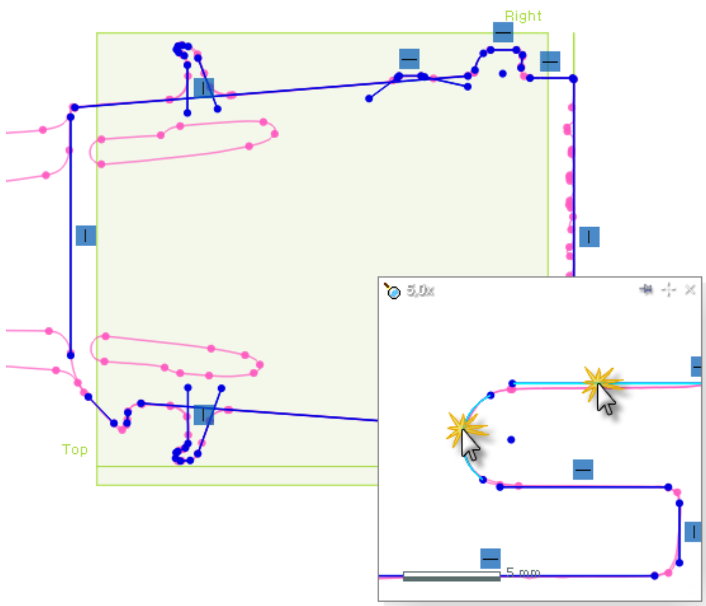
④ Set Vertical or Horizontal constraints to the other sketch, same as in the previous step.



Tip. If you click right mouse button on a sketch, you can also easily set a constraint to the sketch in the pop-up menu.



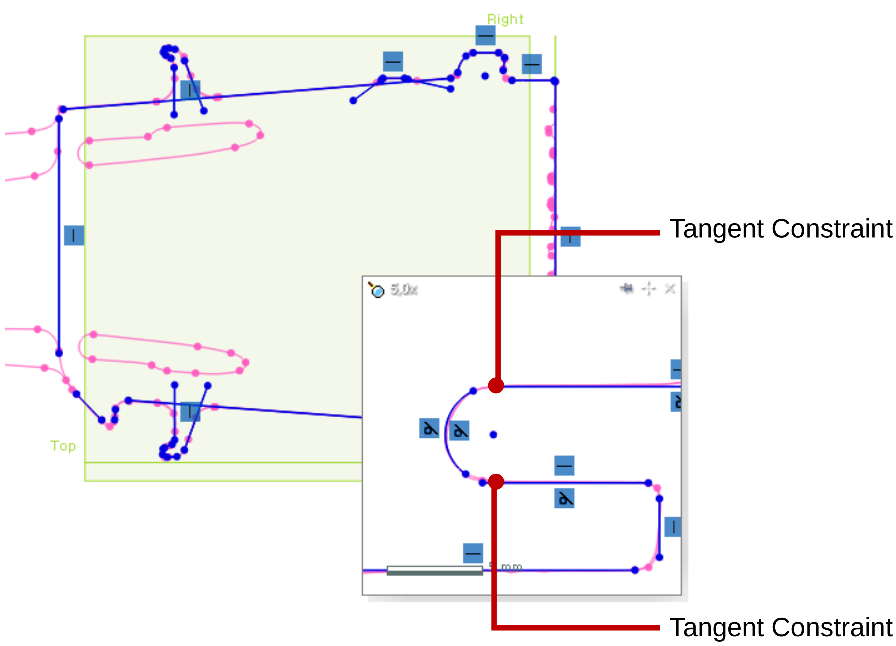
⑤ Select the arc sketch and then select the line sketch with Shift key, as shown in the image below.



⑥ Click the **Tangent** button to set a tangent constraint between the selected sketches.

Tip. *If you select sketches with Shift key, you can set a constraint between them.*

⑦ Set Tangent constraint to the other sketches, same as in the previous step.

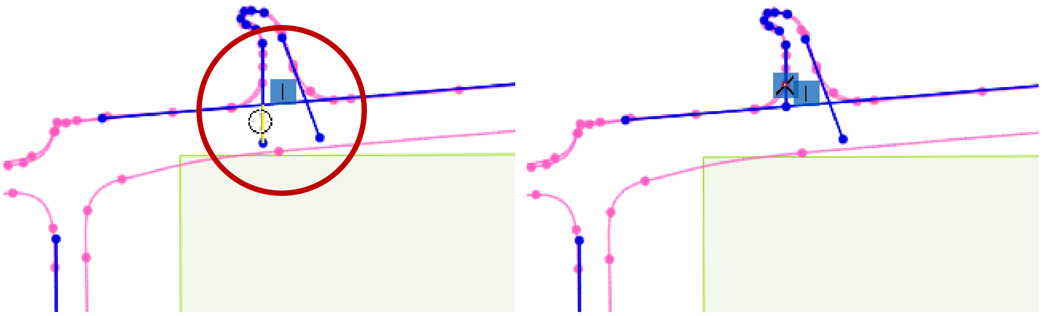


⑧ Click the **OK** button.

- **Trim sketches**

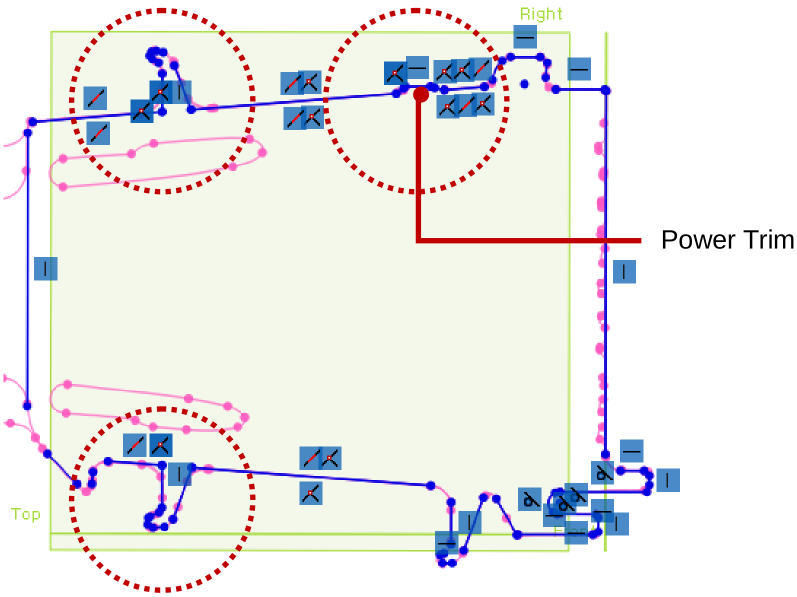
Note. *The application provides two trimming methods. One of the trimming methods is Power Trim. The Power Trim when applied to intersecting sketches and it easily trims out the unnecessary sketches.*
The other method is Corner Trim. The Corner Trim automatically extends sketches to be a coincident at the point of intersection and trims.

① Click the **Trim** in the Toolbar or choose **Tools > Sketch Tools > Trim** in the menu.
② Select the **Power Trim** option and then select the extended sketch, as shown in the image below.

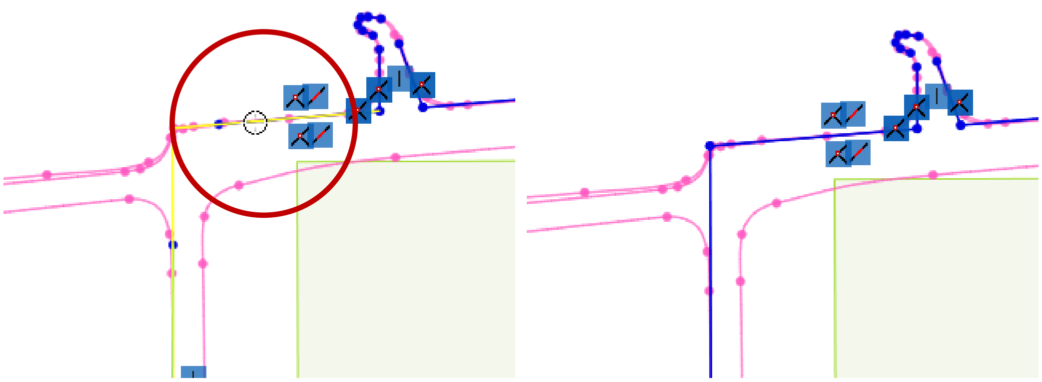


Tip. *You also can easily trim out the unnecessary area of sketches by dragging mouse cursor.*

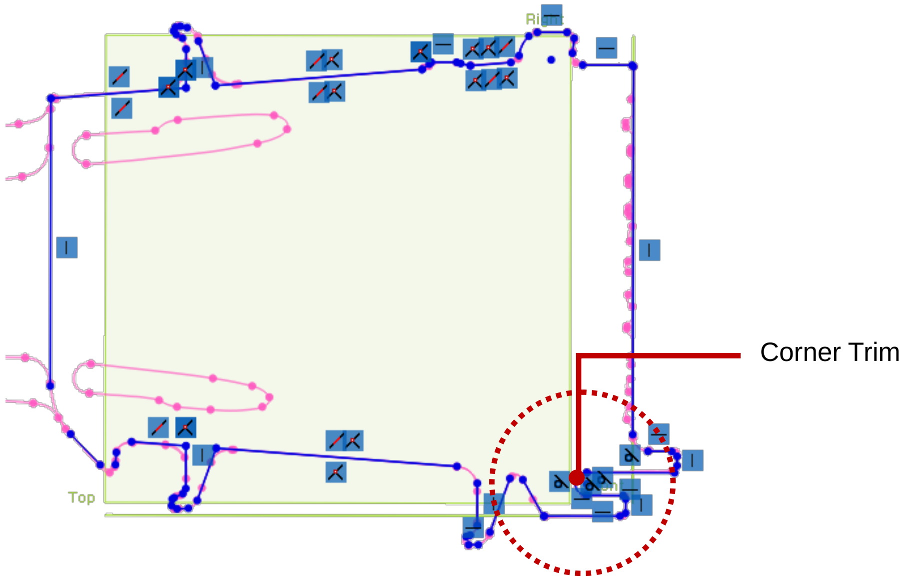
③ Trim the other sketches, as shown in the image below.



④ Select the **Corner Trim** option and then select the sketches, as shown in the image below.



⑤ Trim the other sketches, as shown in the image below.

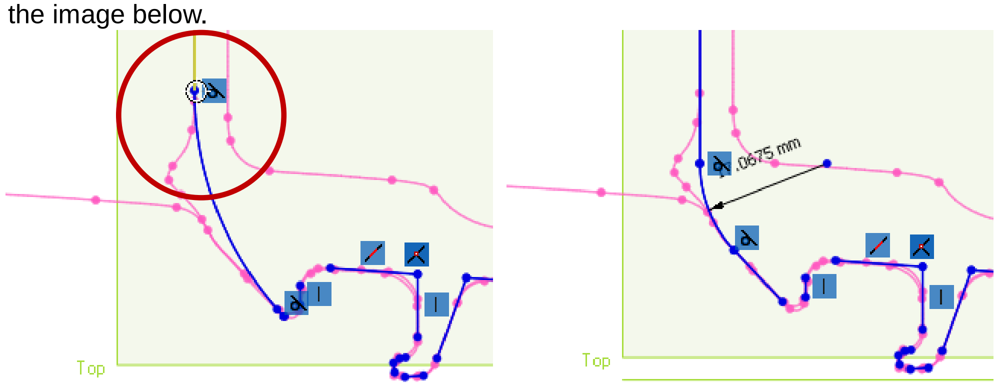


⑥ Click the **OK** button.

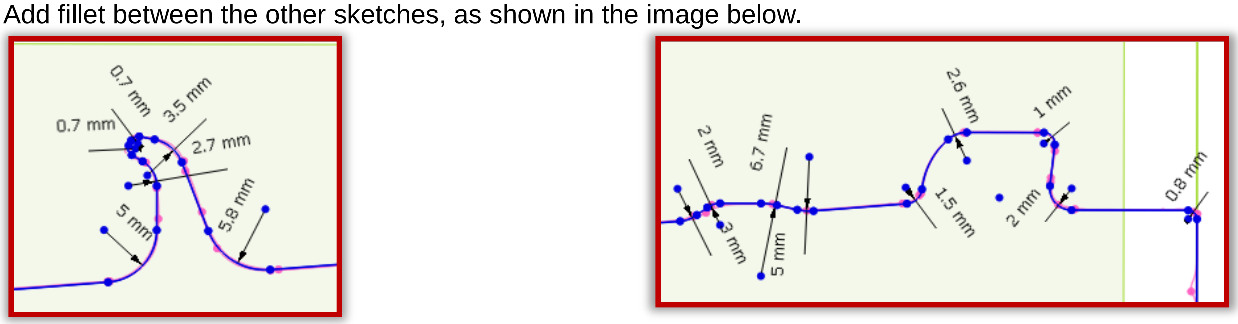
- **Add fillet**

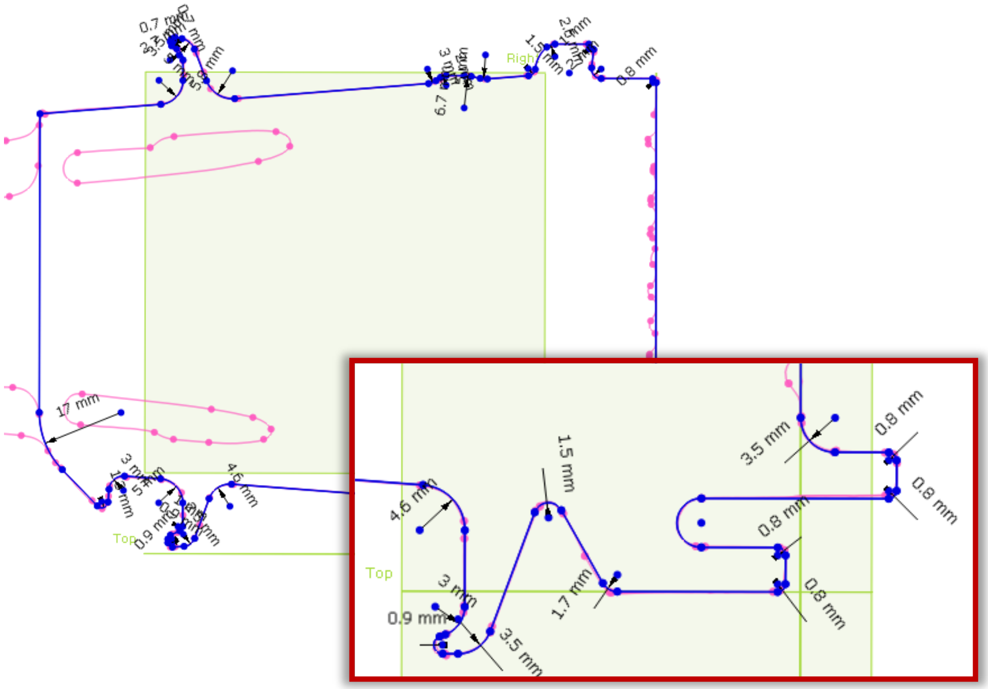
> ***Note.*** *The application provides you several methods to join the sketches. One of joining methods is to add round corner with a tangent arc at the intersection point of two sketches. This method is called Fillet.*

① Click the **Fillet** in the Toolbar or choose **Tools > Sketch Tools > Fillet** in the menu.
② Select a sketch and then select the other sketch and drag the tangent arc onto the Section Polyline, as shown in the image below.



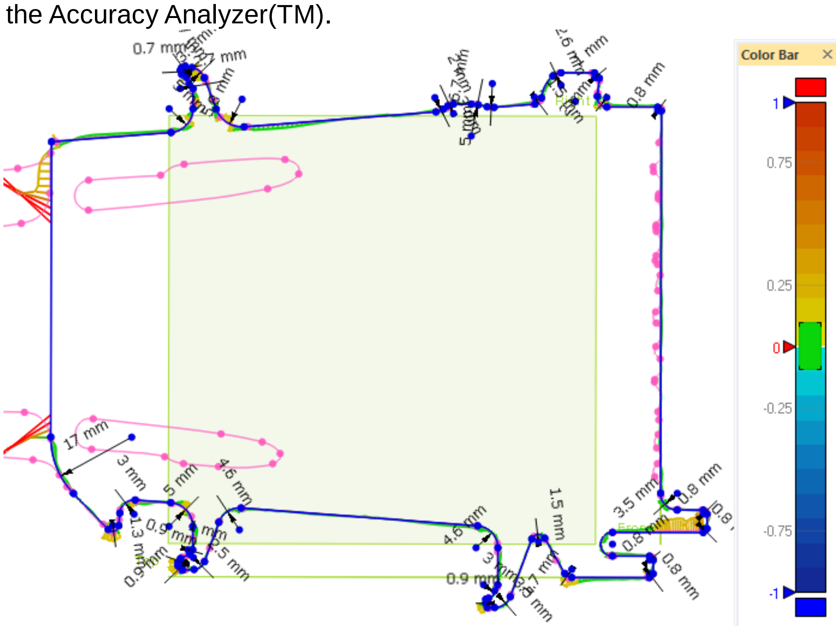
③ Add fillet between the other sketches, as shown in the image below.





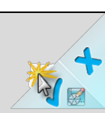
④ Click the **OK** button.

⑤ Check the deviation between the designed sketch and the Section Polyline by selecting the **Deviation** button in the Accuracy Analyzer(TM).

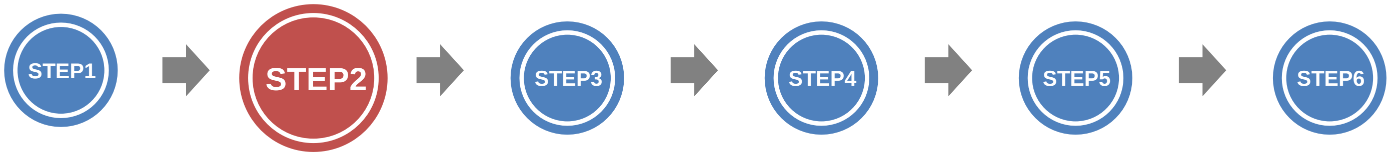


***Tip.** If the sketches have the deviation between the designed sketches and the Section Polyline, you can reduce the deviation value by picking-and-dragging sketches.*

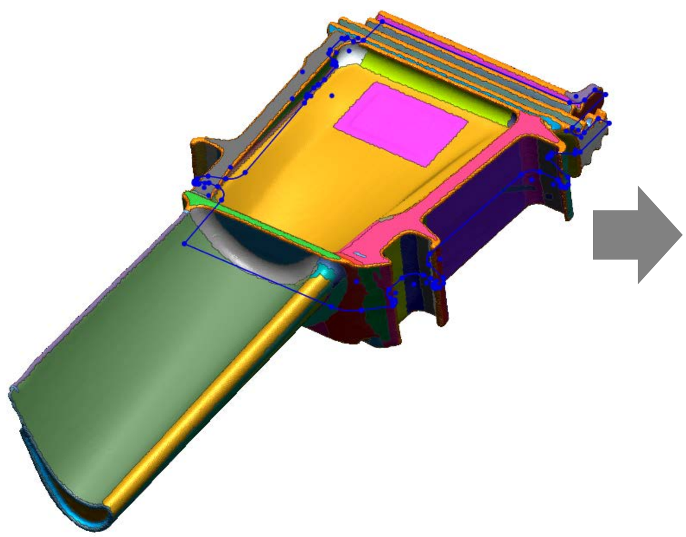
⑥ Click the **Mesh Sketch** Button in the Tool Palette to exit the mode.

Tip. *You can also exit the mode by clicking the*  *(Confirm) button on the right-bottom side of Model View.*
If you click the cancel button, all generated entities in the current mode will be discarded.

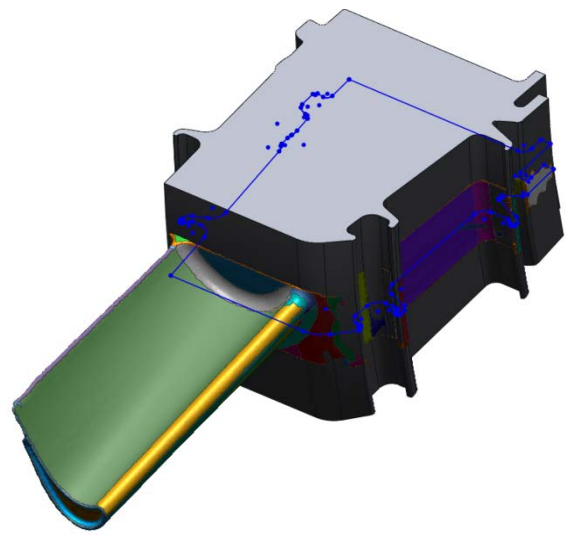
Step2. Create Root Part of Blade Model



You have designed a 2D sketch profile for creating the root part of the blade model so far.
In this step, you will learn how you can create a solid body for the root part of the blade model by using the 2D sketch profile and Revolve method.



Created Sketch Profile

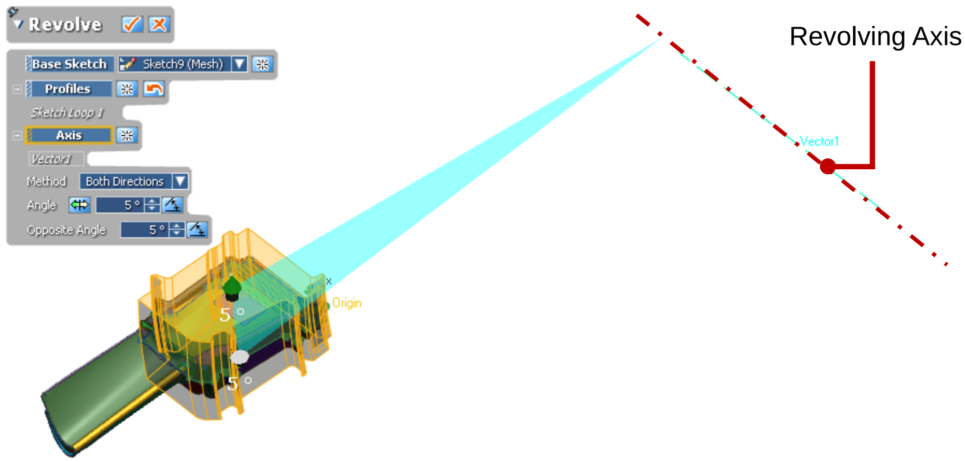


Created Revolved Solid Body

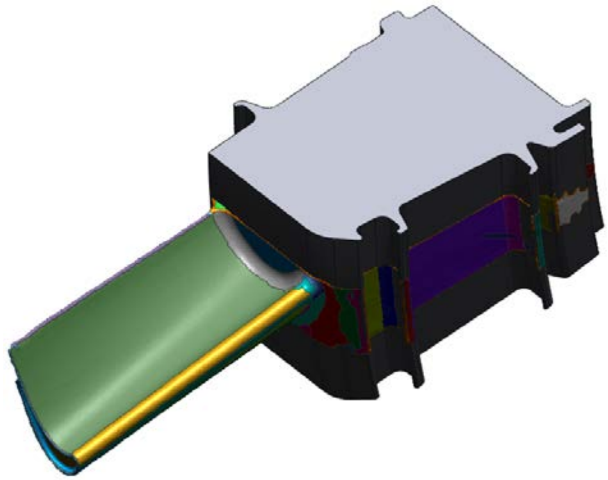
1. Create Root Part of Blade Model

You can create a solid body for the root part of blade model by using the designed 2D sketch profile and Revolve solid modeling method.

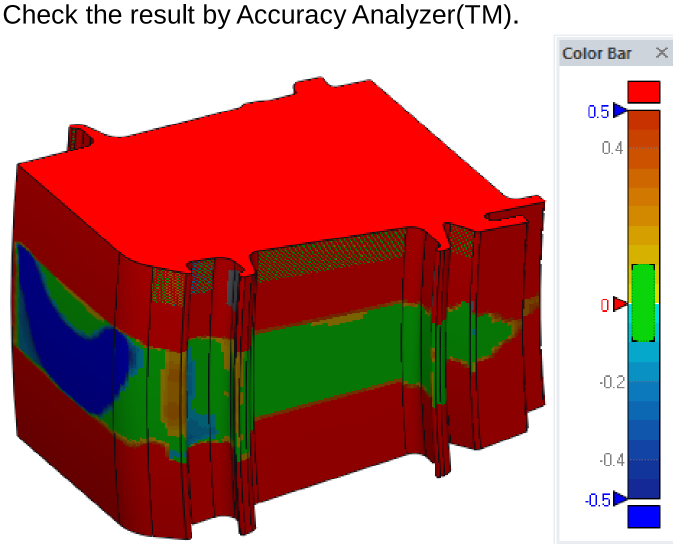
① Check that the Sketch9 and Vector 1 are visible in the Model View.
② Click the **Revolve** button in the Toolbar or choose **Insert > Solid > Revolve** in the menu.
③ Select the sketch profile (Sketch9) as the target Base Sketch and then select the **Axis** button to set a revolving axis.
④ Select the vector (Vector1) as the target Axis and then set the **Method** to **Both Directions**.
⑤ Set the **Angle** to **'5°'** and also set the **Opposite Angle** to **5°**, as shown in the image below.



⑥ Check the previewed result and then click the **OK** button.

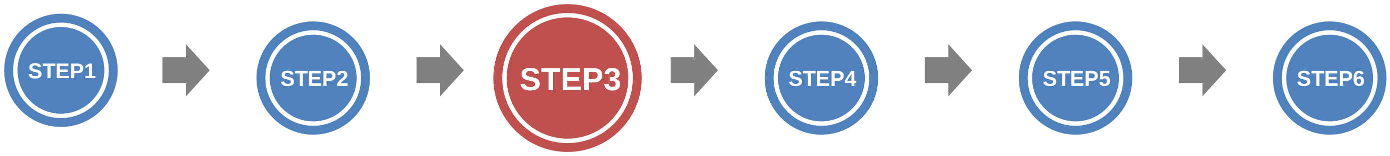


⑦ Check the result by Accuracy Analyzer(TM).

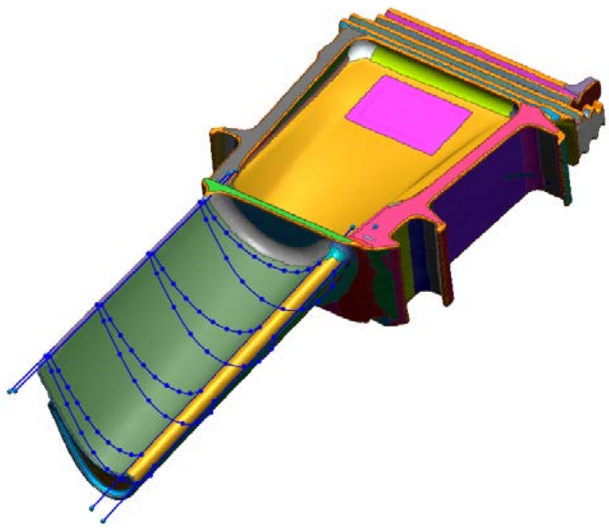


Note. *The Accuracy Analyzer(TM) enables you to analyze the result in real time during the modeling process.*

Step3. Extract Feature Line and Design 2D Sketch Profiles



You have generated a solid body for the root part of the blade model so far.
In this step, you will learn how you can extract feature lines and 2D sketch profiles from the scan data for creating the vane part of the blade model.



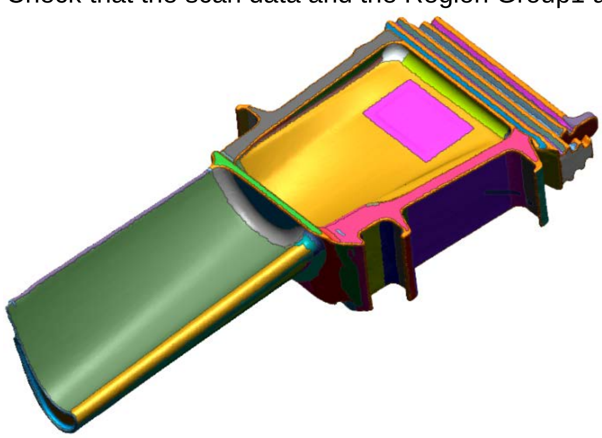
Created Sketch Profile for Creating Vane Part

1. Extract Feature Lines

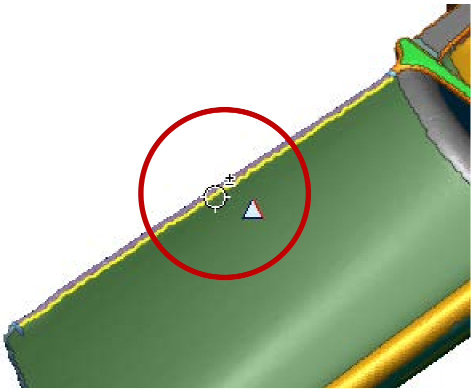
You need several freeform profiles for creating a vane part of the blade model which is thin and twisted.

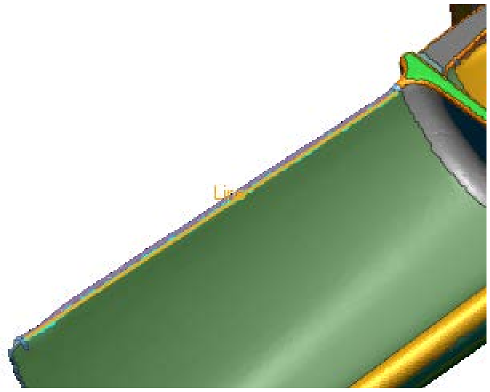
In the first of the sketching process, you need to extract feature lines. The features lines will be used for guiding the twisted shape of the vane part.

① Check that the scan data and the Region Group1 are visible in the Model View.

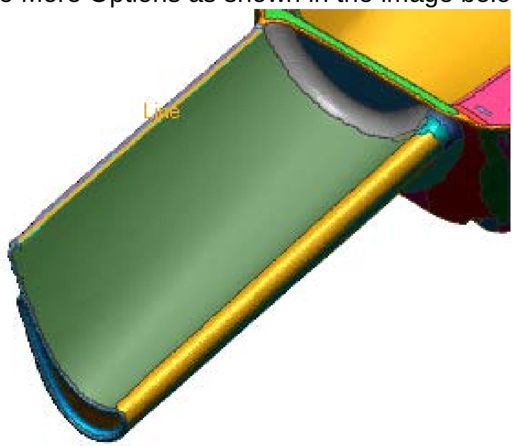


② Click the **3D Sketch** button in the Tool Palette to enter the 3D Sketch mode.
③ Choose **Tools > 3D Sketch Entities > Fit From Points** in the menu.
④ Select a boundary of the region as the Targets, as shown in the image below.

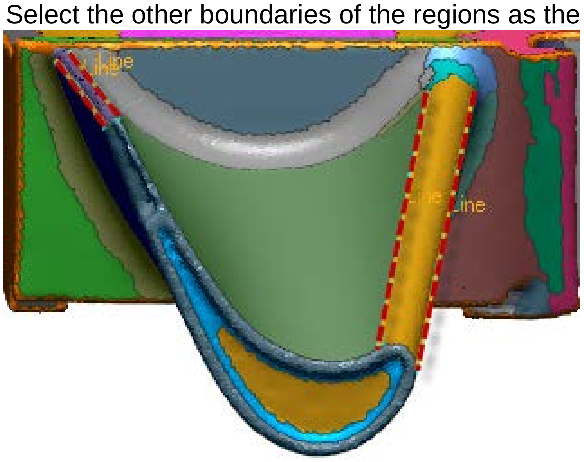




⑤ Set the **Fitting Shape** to **Line** and set the **Allowable Tolerance** to **1mm**.
⑥ Turn the **Remove Outlier** option off in the More Options as shown in the image below.



⑦ Select the other boundaries of the regions as the Targets.



⑧ Click the **OK** button.

Note. *The extracted lines will be used as guide lines for creating a twisted vane part of the blade model.*

⑨ Click the **3D Sketch** button in the Tool Palette to exit the mode.

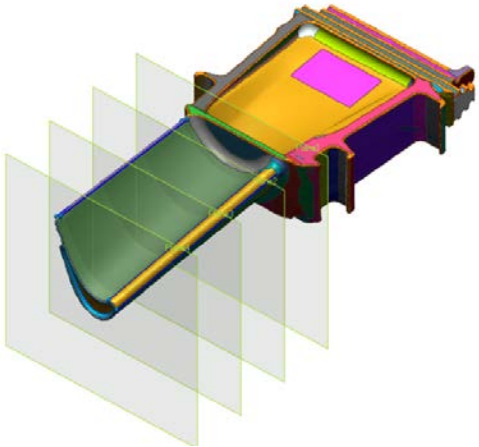
2. Design Sketch Profiles

You need several freeform profiles for creating a vane part of the blade model which is thin and twisted.

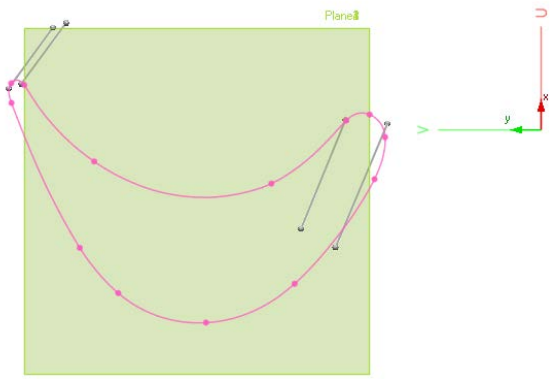
In the second of the sketching process, you need to design sketch profiles. The sketch profiles will be used for creating the twisted shape of the vane part.

- **Design spline sketch profiles**

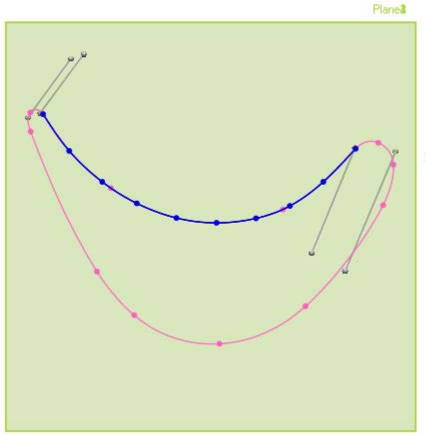
① Check that the 3D Sketch1 and the Ref.Plane1, 2, 3 and 4 are visible with the scan data in the Model View.



② Click the **Mesh Sketch** button in the Tool Palette to enter the Mesh Sketch mode.
③ Check that the **Planer Method** is selected and then select the Ref.Plane1 as the target Base Plane.
④ Check that the Section Polyline is created on the defined base plane and then click the **OK** button.
⑤ Hide the scan data in the Model View to clearly see the Section Polyline, as shown in the image below.

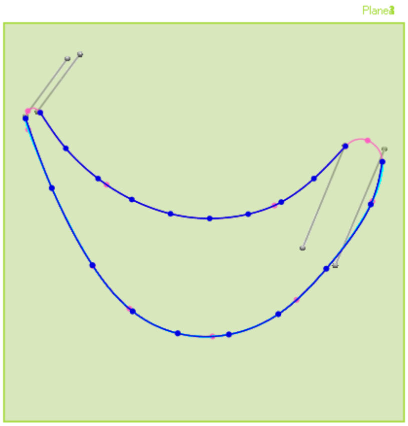


⑥ Click the **Spline** in the Toolbar or choose **Tools > Sketch Entities > Spline** in the menu.
⑦ Check that the **Fit Polyline** option is enabled and the value in the **No. Of Interpolation Points** option is set to **10**.
⑧ Select concave segments of the Section Polyline to fit a spline sketch, as shown in the image below.



Tip. *You can select the segmented Section Polyline to fit a spline sketch and you can also easily select that region by dragging your mouse cursor.*
If you select the regions that you don't want to, you can deselect the regions with Ctrl key.

⑨ Click the **Accept Fitting** button to register the previewed sketch.
⑩ Select convex segments of the Section Polyline to fit a spline sketch, as shown in the image below.

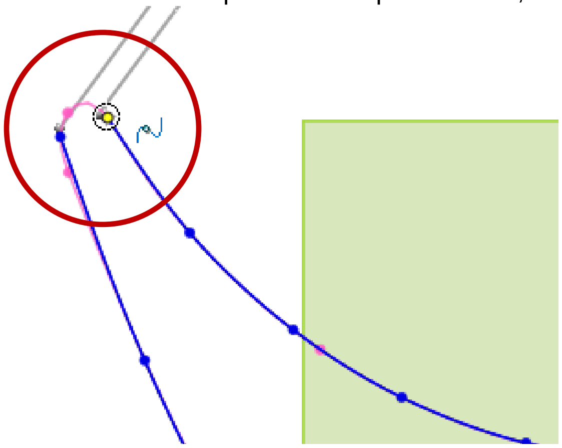


⑪ Click the **OK** button.

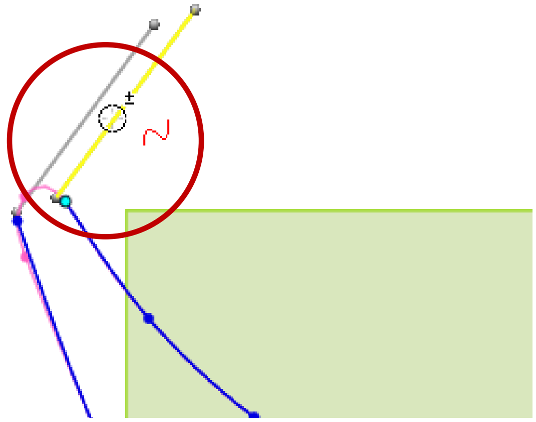
- **Set constraints**

> ***Note.*** *The spline sketches have been generated by fitting operation.*
> *And you will create the other spline sketches on the other base planes for creating a twisted vane part of the blade model.*
> *The spline sketches are free-formed on the Section Polyline and their end points may have different position with the other sketch profiles.*
> *If so, it could not be maintained a high quality freeform surface from a twisted vane part of the blade model.*
> *So, you need to set a pierce constraint to the end points of spline sketches with the generated guide line to array their position onto the guide line.*

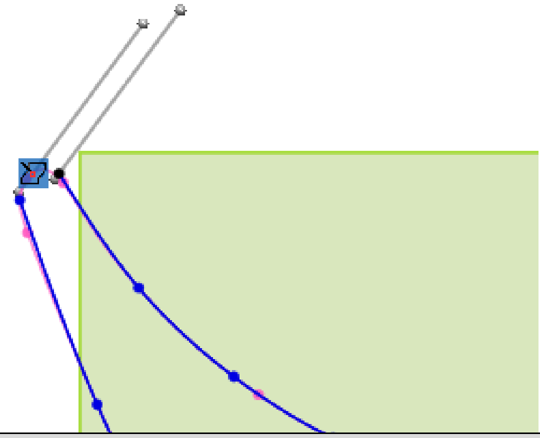
① Double-click the end point of the spline sketch, as shown in the image below.



② Select the one of the guide lines (3D Sketch1) with **Shift** key down.

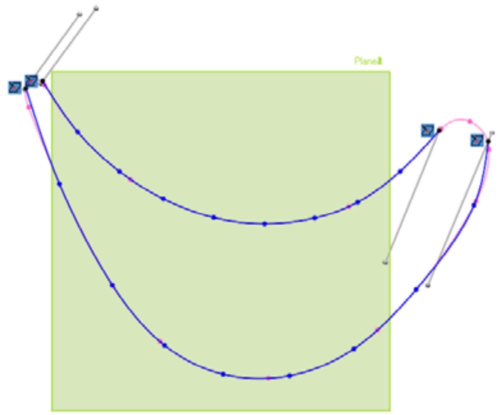


③ Click the **Pierce** button to set a constraint between the spline sketch and the guide line, as shown in the image below.



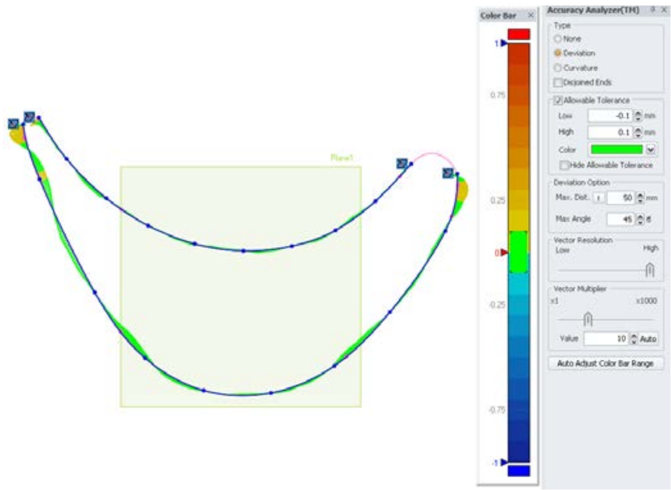
Note. *The end point of the spline sketch will be set with a coincident constraint with an intersection point which is created with the sketch base plane and the guide line.*

④ Set a pierce constraint to the other end points of the sketches, same as in the previous step.

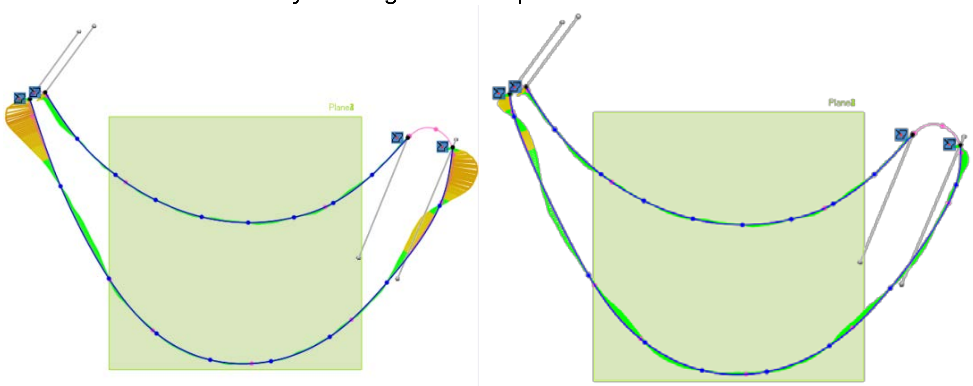


⑤ Click the **OK** button.

⑥ Check the deviation between the designed sketches and the Section Polyline by selecting the **Deviation** button in the Accuracy Analyzer(TM).



⑦ Minimize the deviation by moving the node points of the sketches on the Section Polyline as shown in the image.



⑧ Select the sketches and then check that the value in the **No. of Interp. Pts** of the Rebuild Splines tool pallet is set to **10**.

⑨ Check the **Evenly Spaced Points** option and then click the **OK** button.

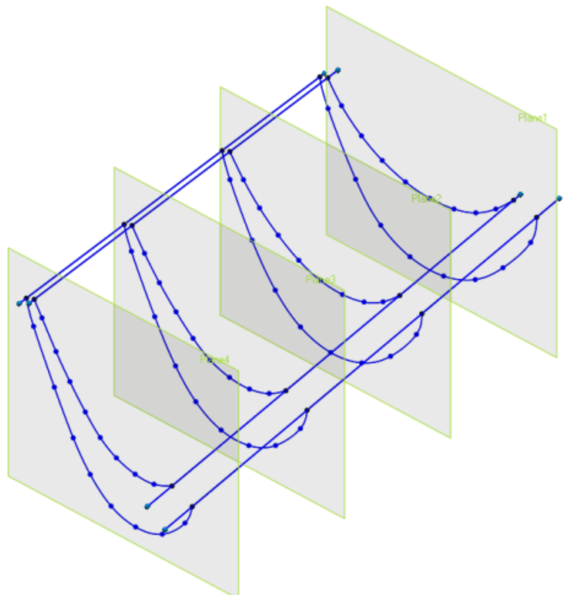
Note. *The Evenly Spaced Points option adjusts the space between the interpolation points on the sketches so that they are evenly distributed.*

⑩ Click the **Mesh Sketch** button in the Tool Palette to exit the mode.

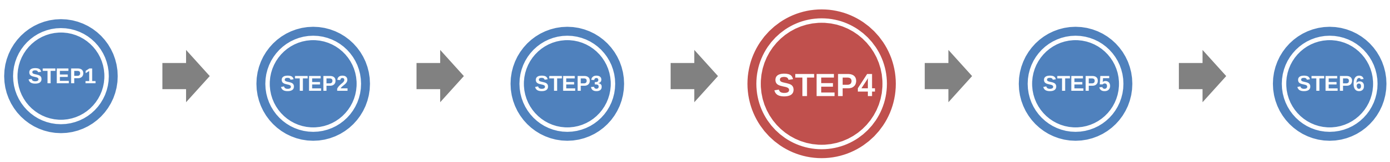
- **Design sketch profiles on the other sketch base planes**

① Design spline sketch profiles on the other sketch base planes (Ref.Plane2, 3, and 4) in the Mesh Sketch mode, same as in the previous step.

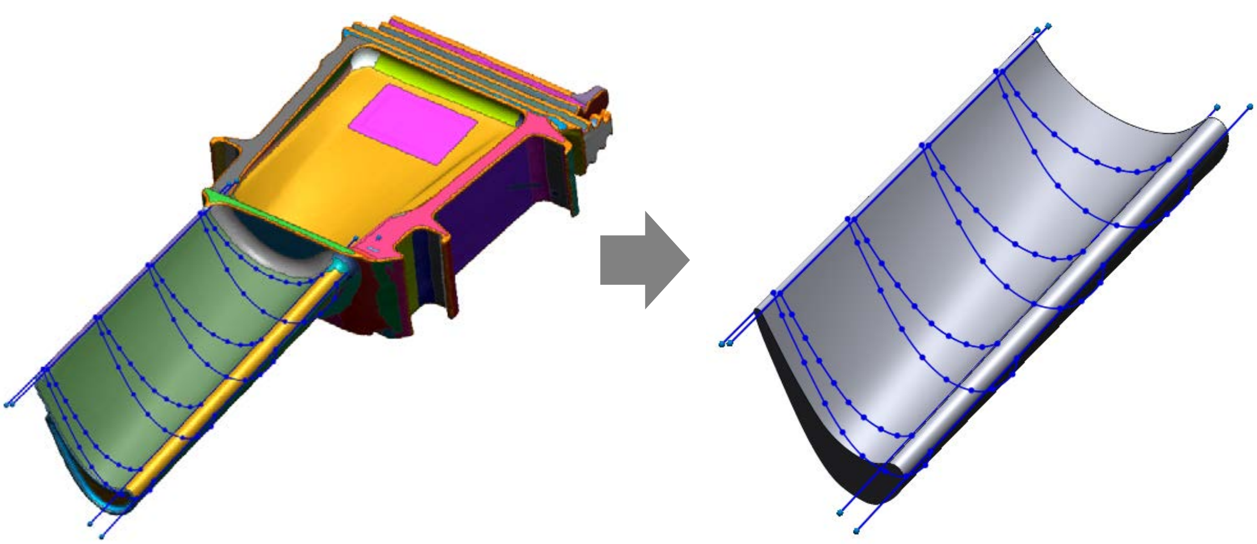
Note. *Whenever you design sketch profiles on a base plane, you need to enter the Mesh Sketch mode.*



Step4. Create Vane Part of Blade Model



You have extracted feature lines and designed 2D sketch profiles for creating the vane part of the blade model so far.
In this step, you will learn how you can create a solid body for the vane part of the blade model by using the 2D sketch profiles and Loft method.



Created Sketch Profiles **Created Vane Part of Blade Model**

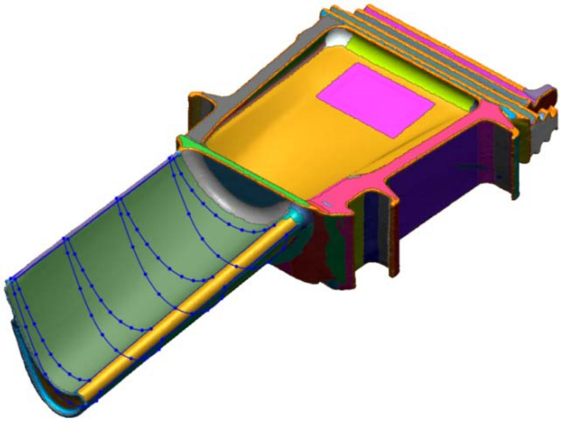
1. Create Freeform Surfaces

All sketch profiles have been prepared for creating a vane part of the blade model so far.

Now, you can create a freeform surface body by using the sketch profiles and Loft surface modeling method.

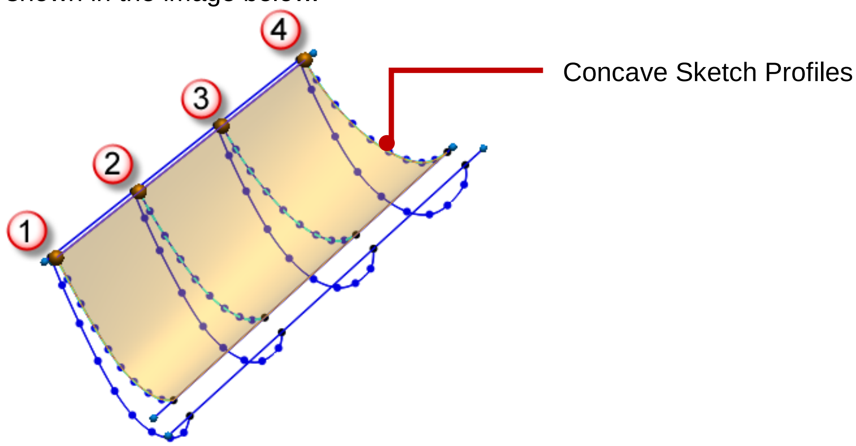
- **Create a concave freeform surface**

① Check that the designed sketch profiles (Sketch10, 11, 12, and 13) are visible in the Model View.

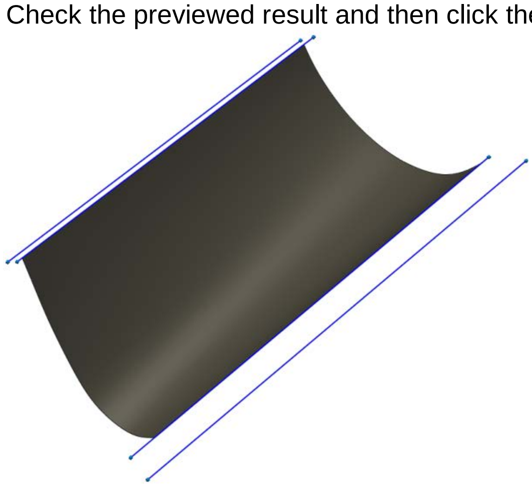


② Click the **Surface Loft** in the Toolbar or choose **Insert > Surface > Loft** in the menu.

③ Select the concave sketch profiles as the target Profiles in consecutive consecutive order order, as shown in the image below.

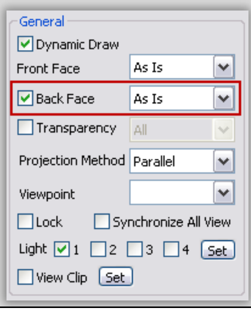


④ Check the previewed result and then click the **OK** button.



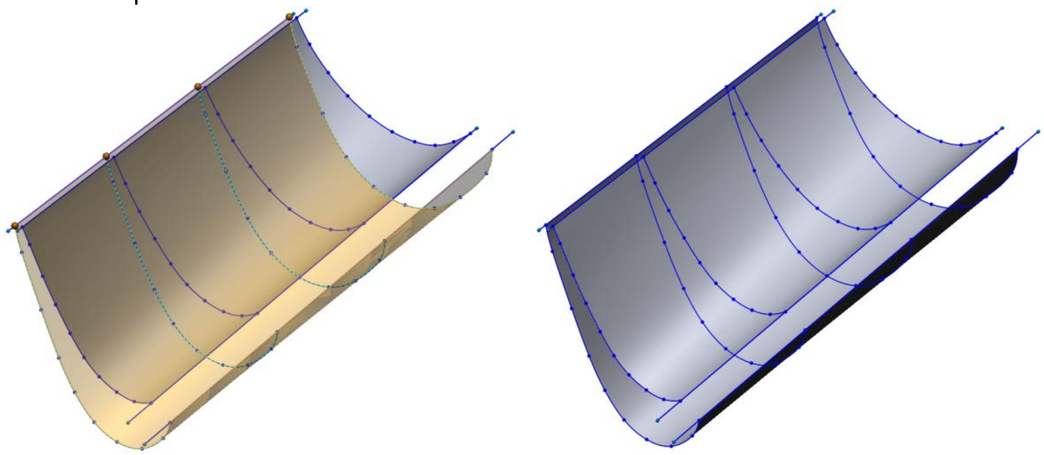
Tip. *The application basically provides 2 sided shading for the face of a mesh or a body. After the operation has been done, you may get a revered face of a surface body.*
If you are planning to create a solid body from the surface body, you don't need to change the normal direction of the face. The application automatically detects a correct direction of the face after the solid

body is finally completed.
But, if you want to see the face with better shading, you can easily change the shading method for the back face. Change the Back Face option to 'As Is' under the General section in the Display tab.



- **Create a convex freeform surface**

① Make the designed sketch profiles (Sketch 10, 11, 12, and 13) visible in the Model View.
② Click the **Surface Loft** in the Toolbar or choose **Insert > Surface > Loft** in the menu.
③ Select the convex sketch profiles as the target Profiles, same as in the previous step.
④ Check the previewed result and then click the **OK** button.



2. Join Freeform Surfaces

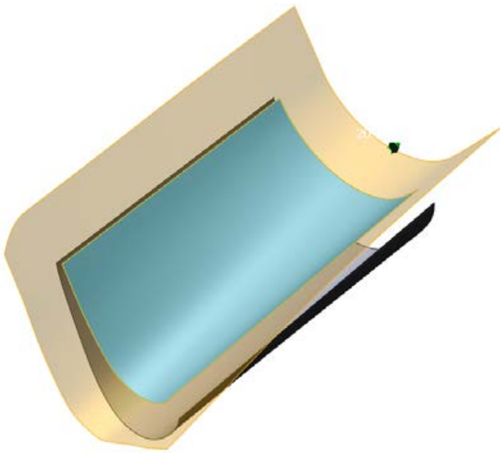
All freeform surfaces have been prepared for creating a vane part of the blade model so far. Now, you can join the surfaces to create a solid body.

- **Extend surface bodies**

① Check that the generated surface bodies (Surface Loft1 and 2) are visible in the Model View.
② Click the **Extend Surface** in the Toolbar or choose **Insert > Surface > Extend** in the menu.

Note. *Currently, the generated surface bodies are formed and positioned in the different coordinate. You need to intersect the surface bodies to join them with each other and the Extend command allows you to extend the surface bodies and intersect each other.*

③ Select the surface body (Surface Loft1) as the target Edges/Faces.
④ Select the **By Distance** option as the End Condition and then set the **Distance** to **20mm**.
⑤ Select the **Linear** option as the Extension Method and then check the previewed result.

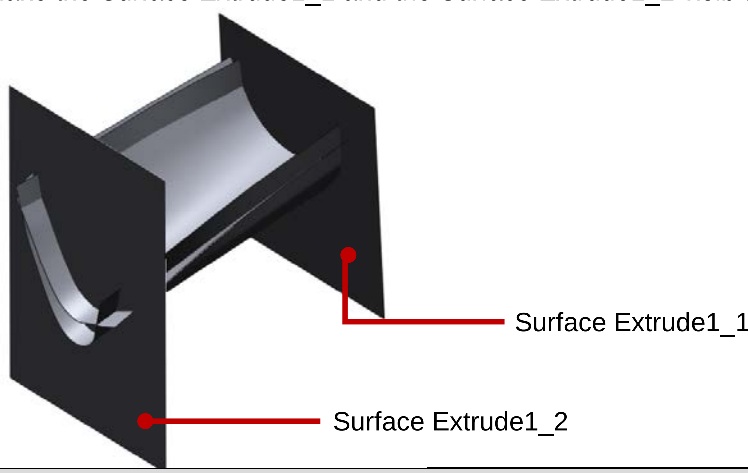


⑥ Click the **OK** button.

Note. *The extended and twisted face of the surface body will be removed after the surface bodies are joined with each other.*

⑦ Extend the convex surface body, same as in the previous step.

⑧ Make the Surface Extrude1_1 and the Surface Extrude1_2 visible in the Model Tree.

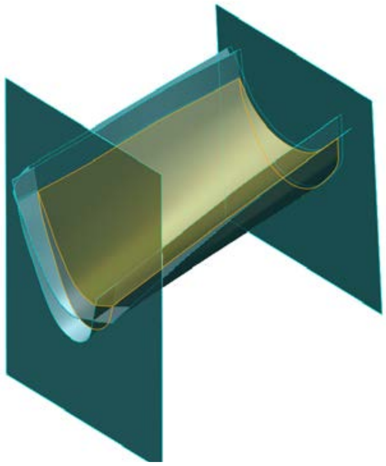


Note. *The Surface Extrude1_1 and the Surface Extrude1_2 which have been generated from the Sketch1 (Mesh) are preset surface bodies in the data file.*

- **Create a solid body by joining the surface bodies**

① Click the **Trim & Merge** in the Toolbar or choose **Insert > Surface > Trim & Merge** in the menu.

② Select the surface bodies (Surface Loft1, 2 and Surface Extrude1_1, 1_2) as the target Entities.
③ Click the **Preview** button and then check the previewed result.



④ Click the **OK** button.

3. Add Fillet

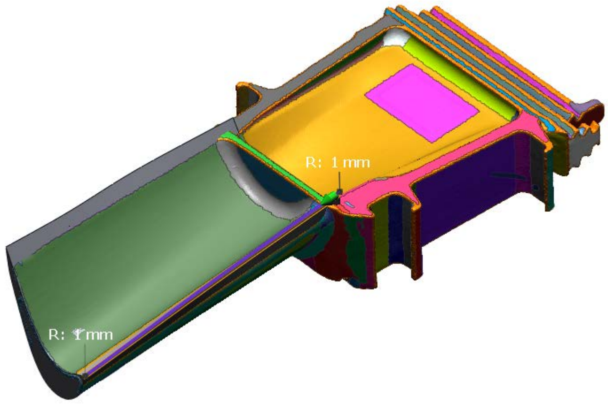
A solid body for a vane part of the blade model has been prepared.

Finally, you can complete the vane part of the blade model by adding fillet.

① Click the **Fillet** in the Toolbar or choose **Insert > Modeling Feature > Fillet** in the menu.

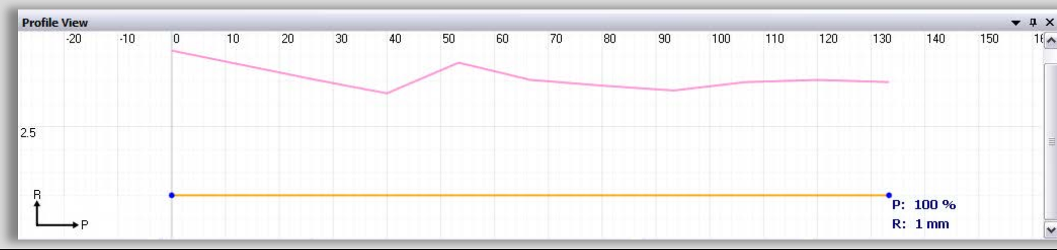
> ***Note.*** *The application provides you several methods to apply face filleting operation such as Constant Fillet, Variable Fillet, Face Fillet, and Full Face Fillet.*
> *The Variable Fillet operation is used for creating a round shape that has a change in fillet radius along the selected edge like this case.*
> *When you apply Variable Fillet to the model, you can control the radius in Profile View as well as in Input Widow.*

② Select the **Variable Fillet** option as the fillet method.
③ Select the edge of the solid body to apply filleting operation, as shown in the image below.



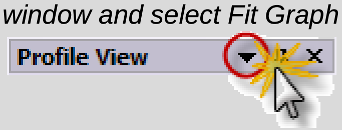
④ Click the **Estimate Radius From Mesh** button.

***Note.** If you click the Estimate Radii From Mesh button, then the application will calculate deviation between the scan data and the selected edge and recommend you a radius in a graph plot in the Profile View so that you can set the radius value.*



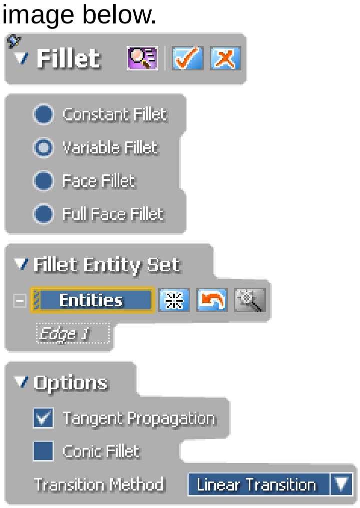
⑤ Change the **Transition Method** to **Linear Transition** and then control radius value in the Profile View or in the Input Window.

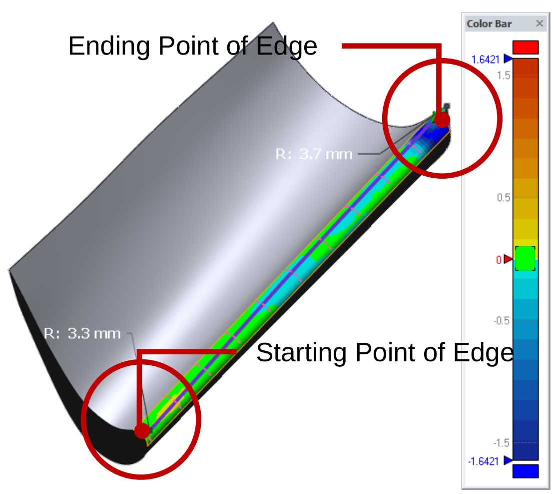
***Note.** If it is difficult to control the radius by graph, try to click Window Position button in Profile View window and select Fit Graph to mesh.*

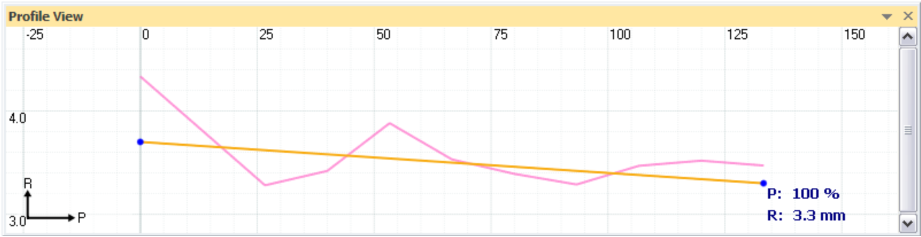


And if you use the Accuracy Analyzer(TM), you can also easily control the radius by checking the deviation between the scan data and the previewed result.

⑥ Set the radius at the start point of the edge to **3.7mm** and at the end point of the edge to **3.3mm**, as shown in the image below.

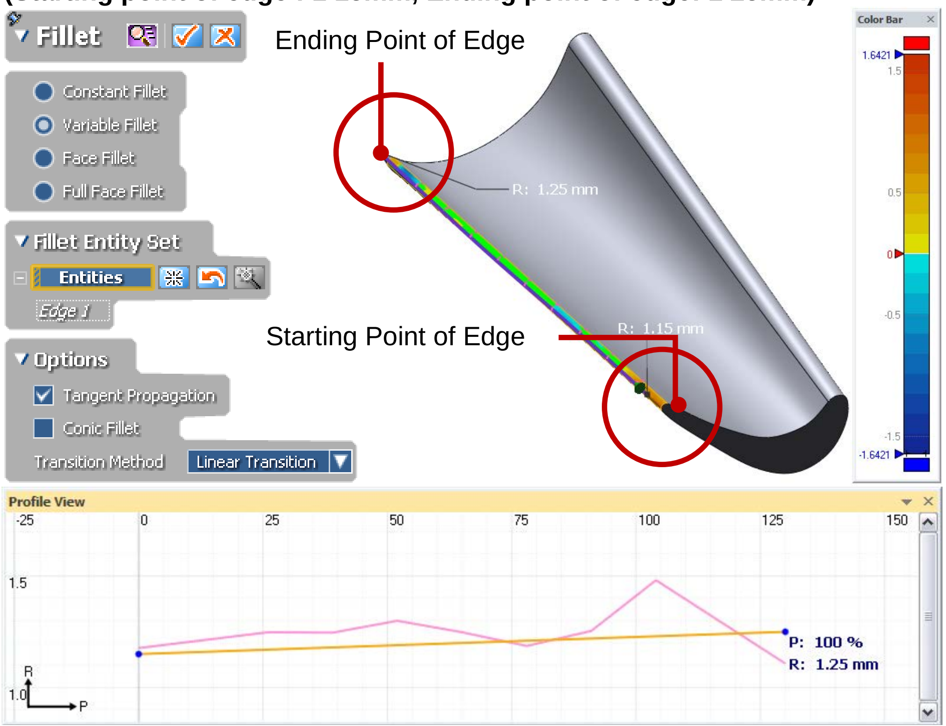




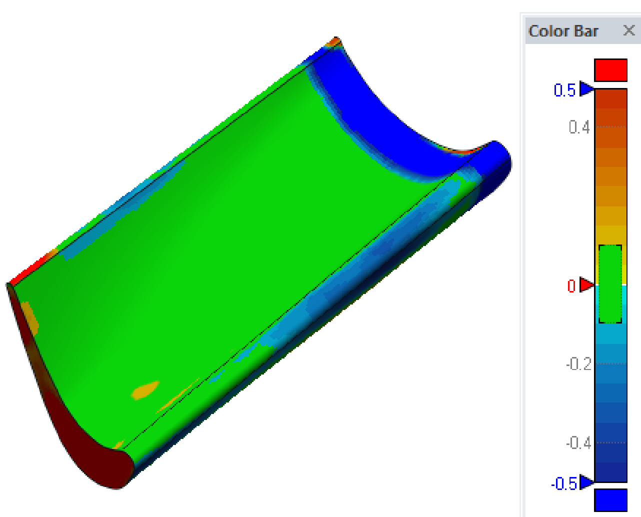


⑦ Click the **OK** button.

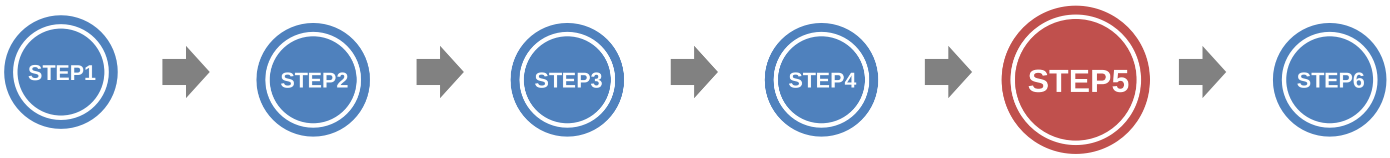
⑧ Add Fillet at the other edge, same as in the previous step.
(Starting point of edge : 1.15mm, Ending point of edge: 1.25mm)



⑨ Check the result with the Accuracy Analyzer(TM).



Step5: Combine Parts and Add More Features



You have generated solid bodies for the root part and the vane part of the blade model so far.
In this step, you will learn how you can combine the solid bodies and then you can add more features onto the combined body. Finally, the blade model will be completely designed.

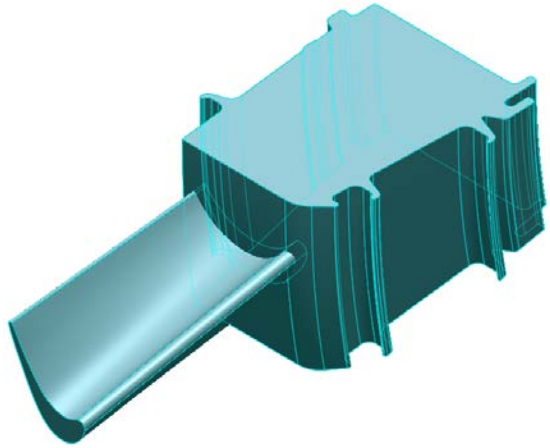
Completed Model

1. Combine Solid Bodies

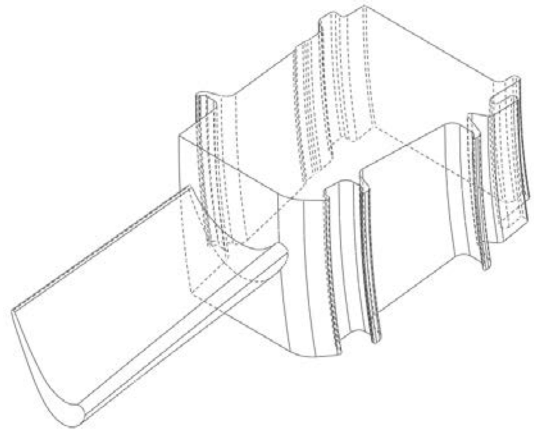
Now, you can combine the generated solid bodies as a single solid body.

- **Combine solid bodies**

① Make the generated solid bodies (Revolve1 and Fillet2 (Variable) visible in the Model View.
② Click the **Boolean** in the Toolbar or choose **Insert > Solid > Boolean** in the menu.
③ Select the **Merge** option as the Operation Method and then select the generated solid bodies as the target Tool Bodies.

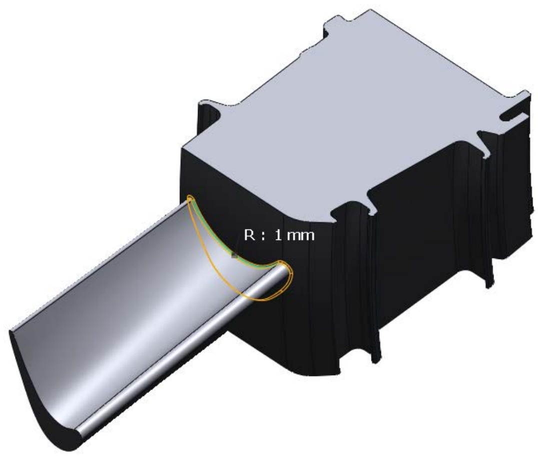


④ Click the **OK** button.
⑤ Check that the solid bodies are combined by clicking the **Hide Line** in the Toolbar or choosing **View > Body Display Mode > Hidden Line** in the menu.

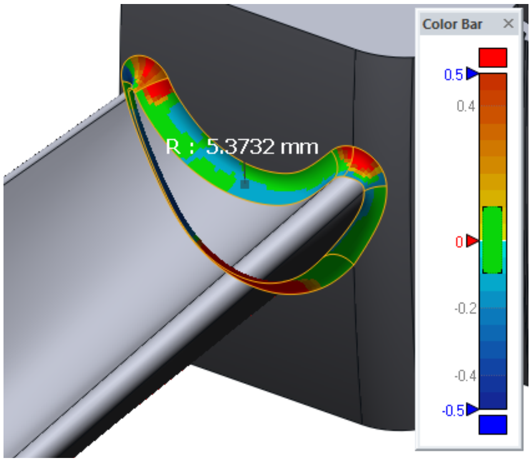


- **Add Fillet**

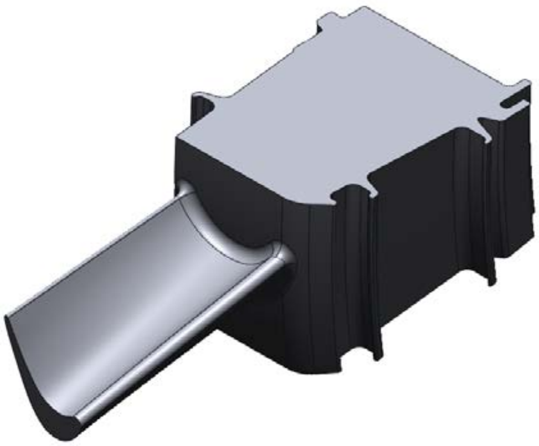
① Click the **Fillet** in the Toolbar or choose **Insert > Modeling Feature > Fillet** in the menu.
② Select the **Constant Fillet** option as the fillet method.
③ Select the edges between van part and root part to apply filleting operation, as shown in the image below.



④ Click the **Estimate Radius From Mesh** button and then check the deviation between the scan data and the solid body at the fillet area with the Accuracy Analyzer(TM).



⑤ Set the **Radius** to **5.5mm** and then click the **OK** button.

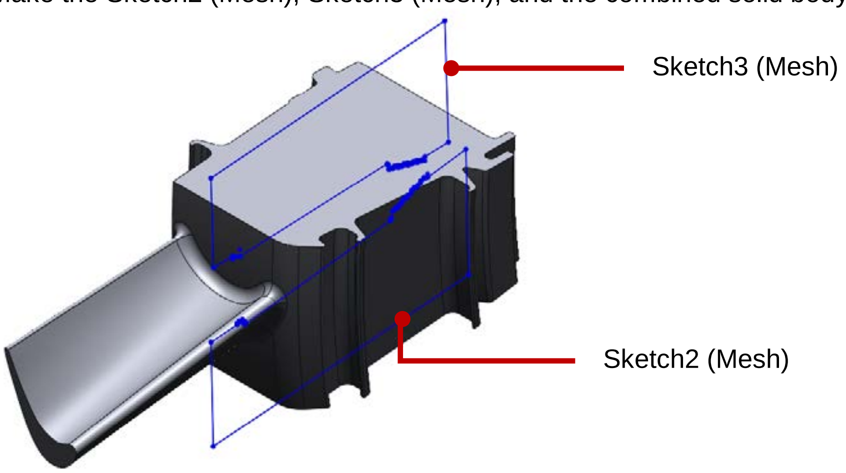


2. Add More Features

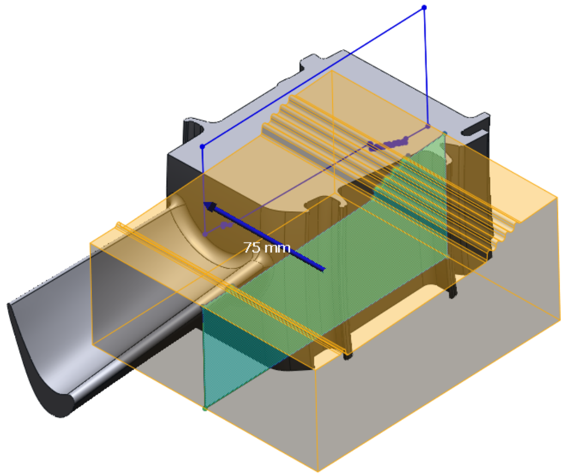
You can add more features to the combined solid body by using preset sketch profiles and surface bodies.

- **Cut upper and lower features off**

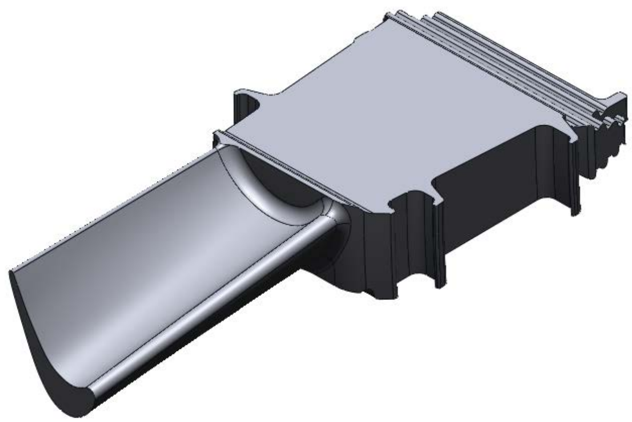
① Make the Sketch2 (Mesh), Sketch3 (Mesh), and the combined solid body visible in the Model View.



② Click the **Extrude** in the Toolbar or choose **Insert > Solid > Extrude** in the menu.
③ Select the sketch profile (Sketch2 (Mesh)) as the target Base Sketch and then set the **Method** to **Mid Plane**.
④ Increase the **Length** to **150mm** and then check the **Cut** option.

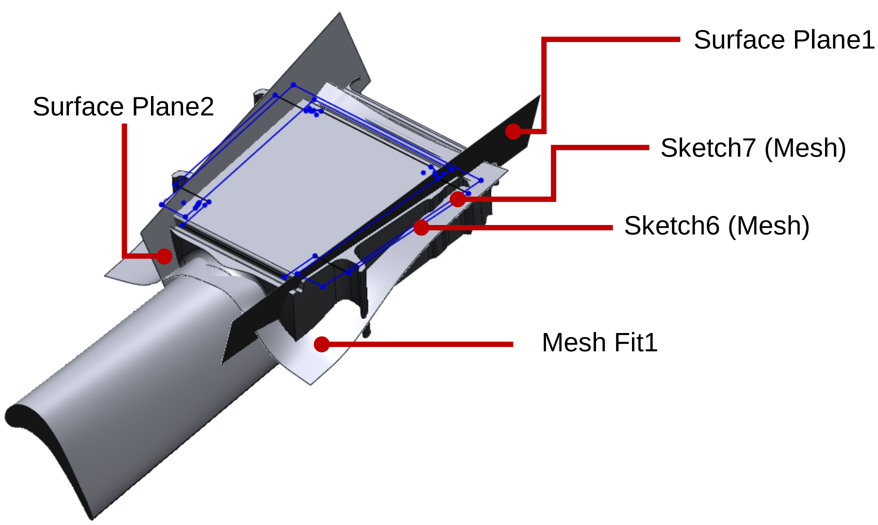


⑤ Click the **OK** button and check the result.
⑥ Cut the opposite feature off by using the other sketch profile (Sketch3 (Mesh)), same as in the previous step.



- **Create upper forms**

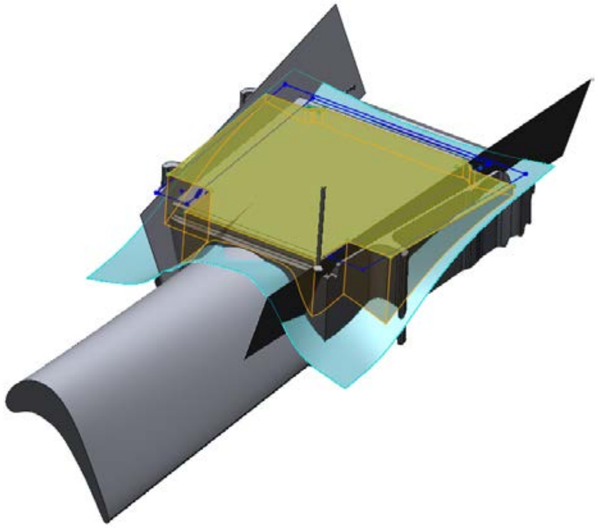
① Make the Sketch6 (Mesh), Sketch7 (Mesh), all preset Surface Bodies and the Solid Body visible in the Model View.



② Click the **Extrude** in the Toolbar or choose **Insert > Solid > Extrude** in the menu.
③ Select the sketch profile (Sketch6 (Mesh)) as the target Base Sketch and then set the **Method** to **Up To Surface**.

Note. *Even though you have a freeform face, you can create a solid body up to the freeform surface body.*

④ Select the surface body (Mesh Fit1) as the target Up To and then turn the other options off in the Result Operator.



Note. If you don't check any options in the Result Operator, An individual solid body is generated. If you want to combine the solid bodies afterward, you can combine those bodies by using Boolean command, same as in the previous step.

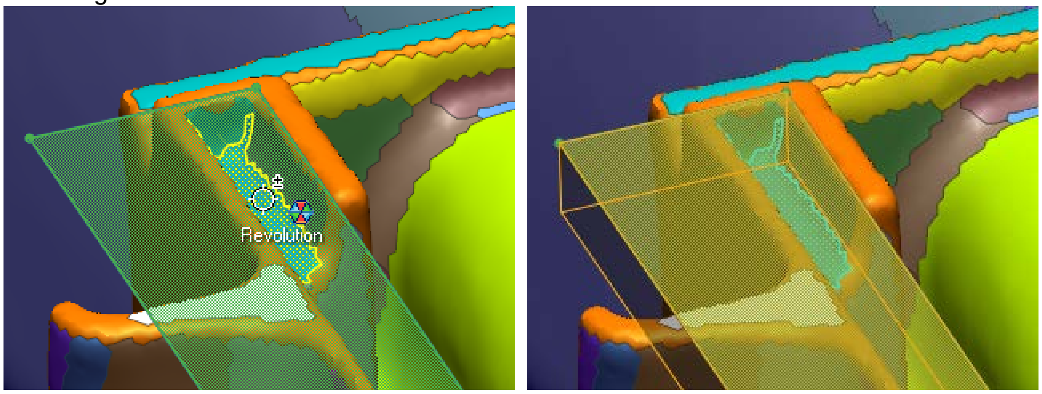
⑤ Check **Opposite Direction** option and then increase the **Length** to **5mm**.

Tip. If the Base Sketch plane is shared with the other face of the solid body, some computing errors might happen when the Cut or Boolean operation is performed.
If you increase the length of body in the opposite direction, you can avoid those computing errors.

⑥ Click the **OK** button.

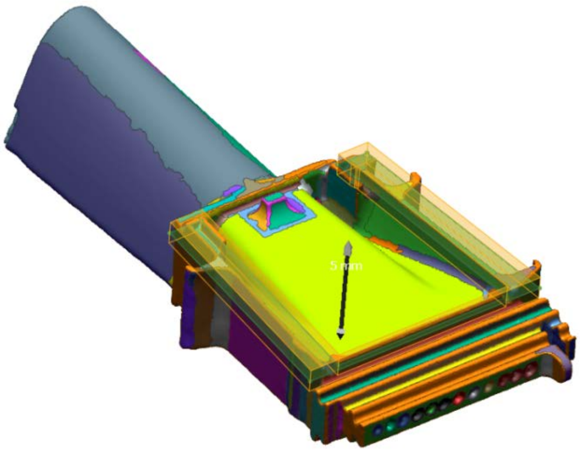
- **Combine created upper forms**

① Click the **Extrude** in the Toolbar or choose **Insert > Solid > Extrude** in the menu.
② Select the sketch profile (Sketch7 (Mesh)) as the target Base Sketch and then set the **Method** to **Up To Region**.
③ Set the **Sub Method** to **Max. Distance Position** and then select a feature region in the scan data, as shown in the image below.

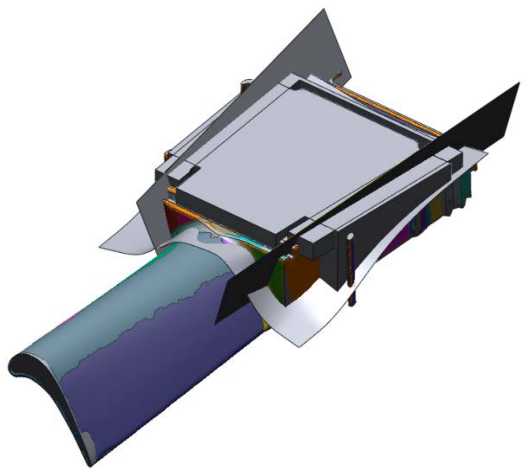


Tip. If you cannot see the region on the scan data, you can quickly hide the preset surface and solid bodies by hitting 'Ctrl + 4 and 5' key.

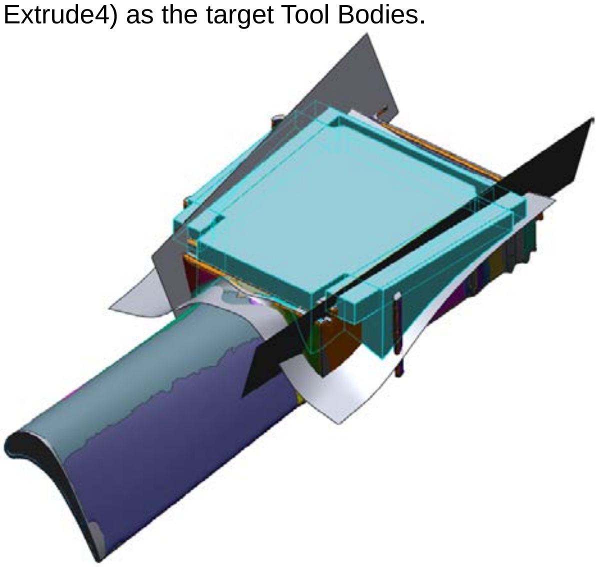
④ Check the **Opposite Direction** option and then increase the **Length** to **5mm**.



⑤ Click the **OK** button.
⑥ Click the **Boolean** in the Toolbar or choose **Insert > Solid > Boolean** in the menu.



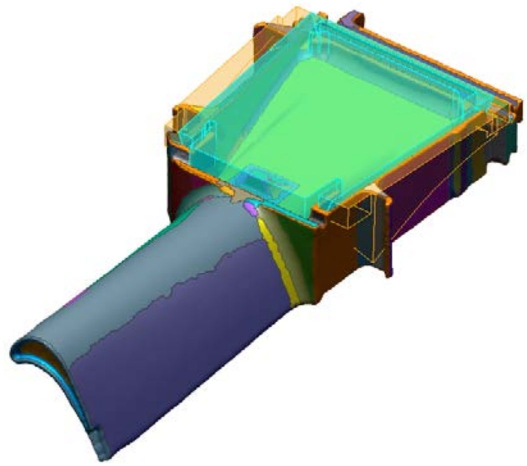
⑦ Check the **Merge** option in the Operation Method and then select the generated solid bodies (Extrude3 and Extrude4) as the target Tool Bodies.



⑧ Click the **OK** button.

- **Cut the combined upper form by using preset surface bodies**

① Click the **Cut** in the Toolbar or choose **Insert > Solid > Cut** in the menu.
② Select the surface bodies (**Surface Plane1** and **Surface Plane2**) as the target Tool Entities and then click the **Target Bodies** button to register the target body.
③ Select the solid body (**Extrude4**) as the Target Bodies and then click the **Next Stage** button.
④ Select the inner body as the Remaining Bodies, as shown in the image below.

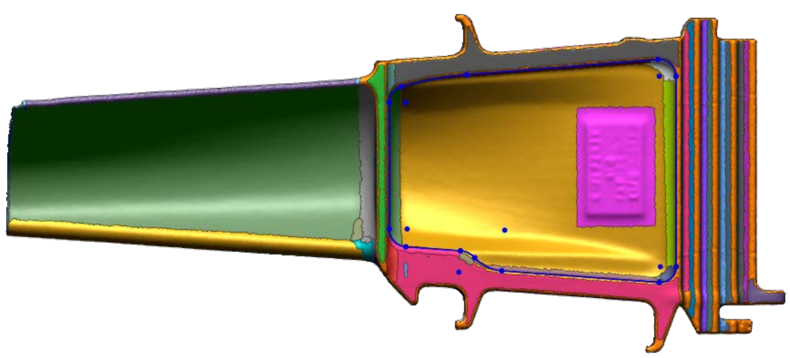


Note. *In the second stage of Cut operation, you can choose remaining bodies where you want to remain bodies.*

⑤ Click the **OK** button.

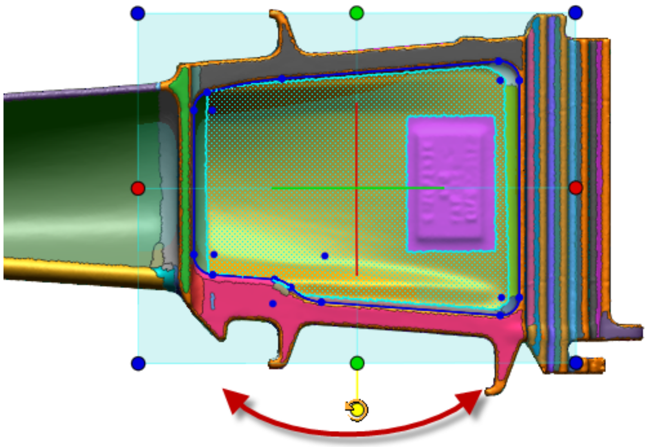
- **Create a lower form**

① Make the Sketch 8(Mesh) and the scan data visible only in the Model View.
② Click the **Right Viewpoint** in the Toolbar or choose **View > Viewpoint > Right** in the menu.



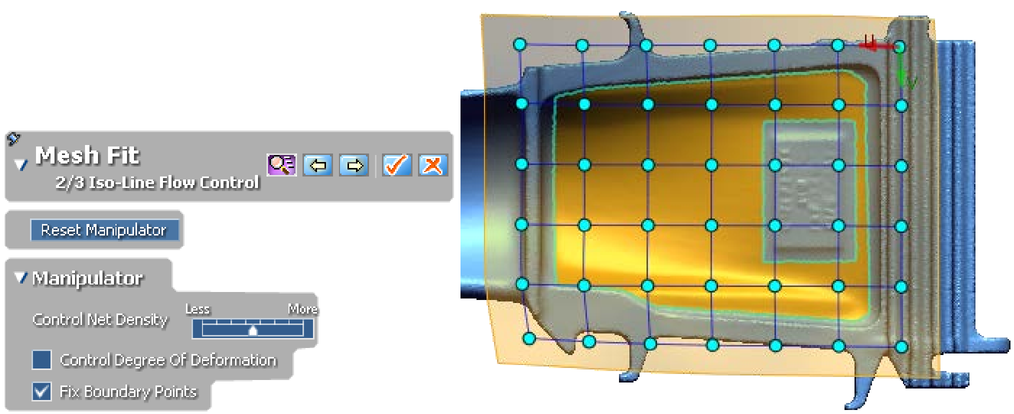
Note. *You can also use a hotkey ('Alt + 4') to set your viewpoint to Right.*

③ Click the **Mesh Fit** in the Toolbar or choose **Insert > Surface > Mesh Fit** in the menu.
④ Select the feature region as the target Regions/Poly-Faces and then rotate the U direction of the virtual plane so that it's parallel to the Z-Axis, as shown in the image below.

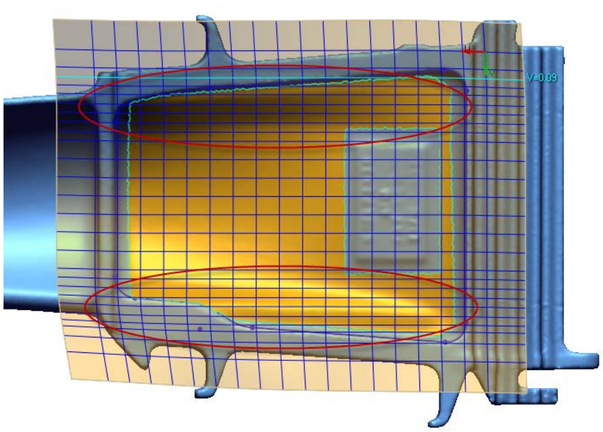


***Note.** The position and orientation of the virtual plane determines the flow of fitting surface body.*

⑤ Change **Resolution** type to **By No. Of Control Points** and then set the number of U and V control points to **20**.
⑥ Adjust options, as shown in the image below and then click the **Next Stage** button.

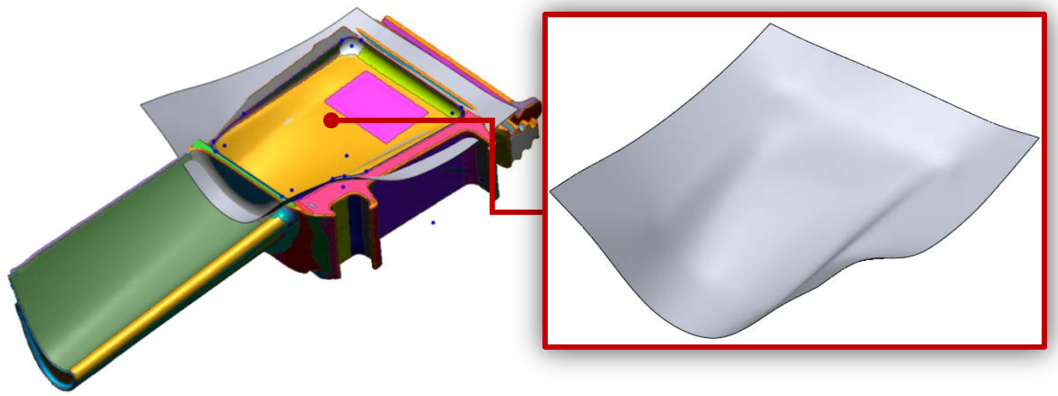


⑦ Check the flow of the fitting surface body and then click the **Next Stage** button.
⑧ Increase density of Iso-Line at the complex area of fitting surface, as shown in the image below.

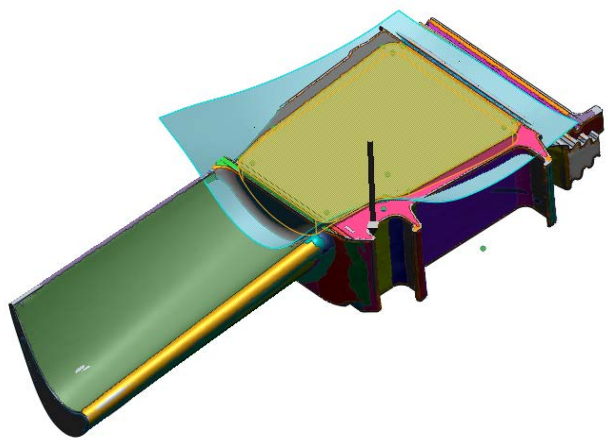


Tip. *You can easily move, add, or remove the Iso-Lines.*
Pick-and-Drag *: Move*
Pick-and-Drag with Ctrl key *: Add*
Delete key *: Remove*

⑨ Check the previewed result and then click the **OK** button.



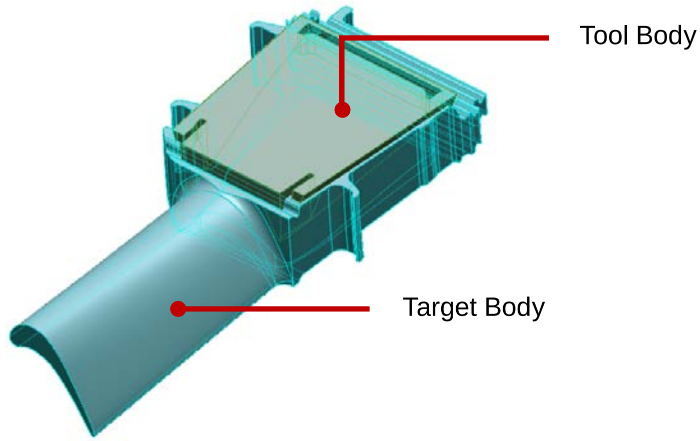
⑩ Make the Sketch8 (Mesh) visible in the Model View.
⑪ Click the **Extrude** in the Toolbar or choose **Insert > Solid > Extrude** in the menu.
⑫ Select the sketch profile (Sketch8 (Mesh)) as the target Base Sketch and then set the **Method** to **Up To Surface**.
⑬ Select the surface body (Mesh Fit2) as the target Up To and then click the **Flip Direction** button.
⑭ Check the **Cut** option in the Result Operator.



⑮ Click the **OK** button.

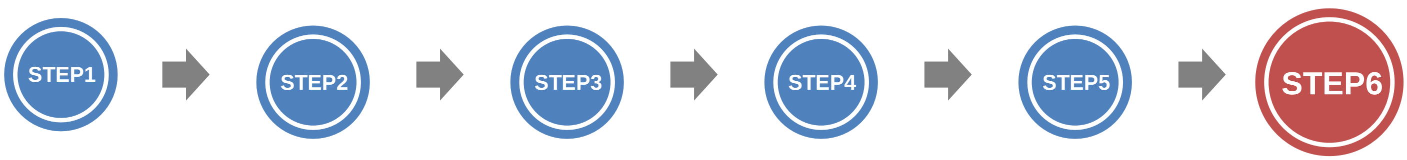
- **Complete the blade model**

① Make the generated solid bodies (**Extrude5 (Cut)** and **Cut1**) visible in the Model View.
② Click the **Boolean** in the Toolbar or choose **Insert > Solid > Boolean** in the menu.
③ Select the **Cut** option as the Operation Method and then select the solid body (**Cut1**) as the target Tool Bodies and the solid body (**Extrude5 (Cut)**) as the Target Bodies.

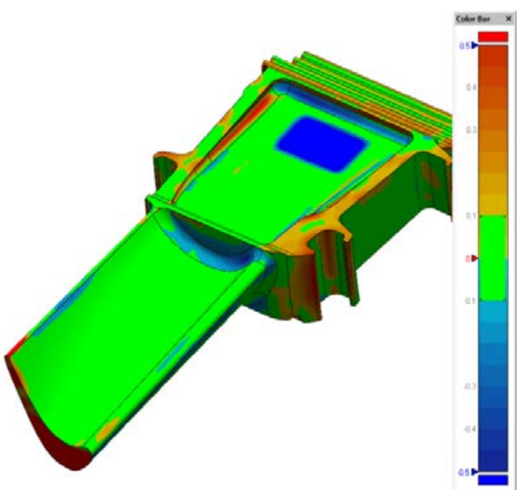


④ Click the **OK** button.

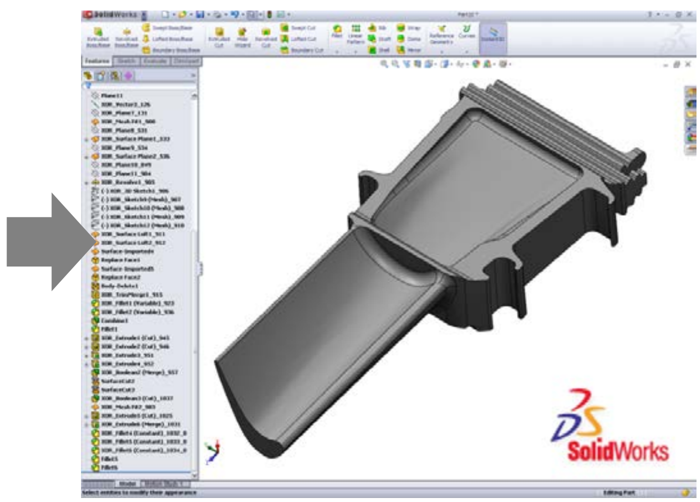
Step6: Check Result and Transfer Model to CAD Program



Finally, you will learn how you can check the modeling result and then transfer it with the whole modeling history to the CAD program.



Confirmed Accuracy

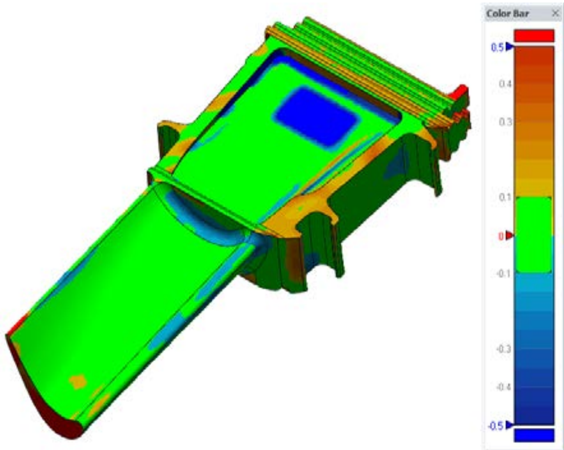


Transferred Model to SolidWorks

1. Check Result

Finally, you can check the result with Accuracy Analyzer(TM).

① Check the result with Accuracy Analyzer(TM).



Note. *You can check the deviation between the scan data and the designed solid body as well as the accuracy and quality of the result by using the Accuracy Analyzer(TM).*

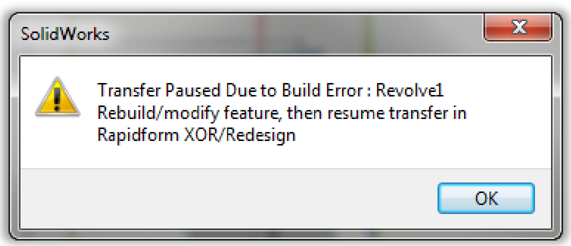
2. Transfer Model to CAD Program

If you want to have the designed model and the whole modeling history in your CAD program, you can transfer your model to the CAD program by using the LiveTransfer(TM) and you can also modify the model in your CAD program.

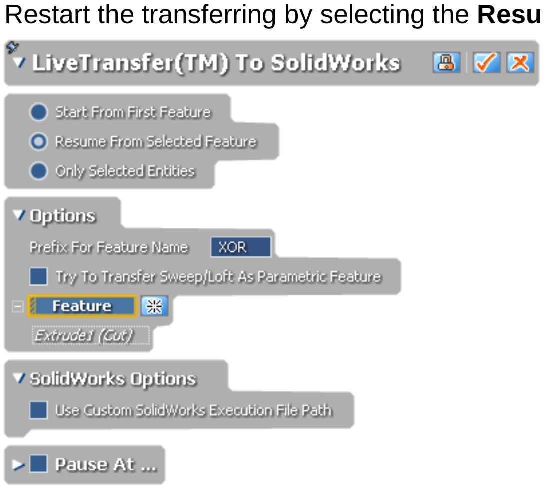
① Choose **File > LiveTransfer(TM) > To SolidWorks** in the menu.

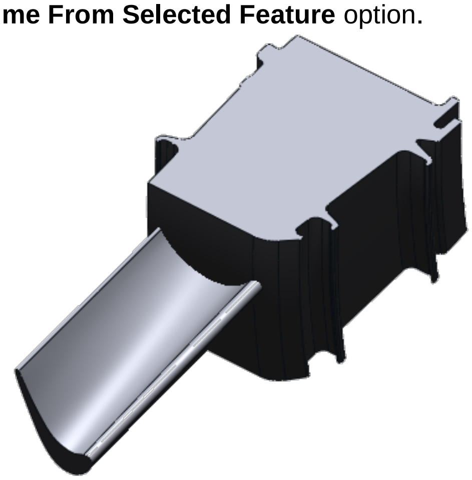
Note. *If you have your own CAD program in your PC, you can choose your CAD program in the menu. In this tutorial, SolidWorks software will be used.*

② Select the **Start From First Feature** option and then click the **OK** button.
③ If an error message occurs, click the **OK** button and then edit features in your CAD program.



④ Choose **File > LiveTransfer(TM) > To SolidWorks** in the menu.
⑤ Restart the transferring by selecting the **Resume From Selected Feature** option.





Note. *The application provides an interactive way to fix errors when you get them during the transferring.*

After you manually edit features in your CAD program, you can complete a model in your CAD program by resuming the transferring.