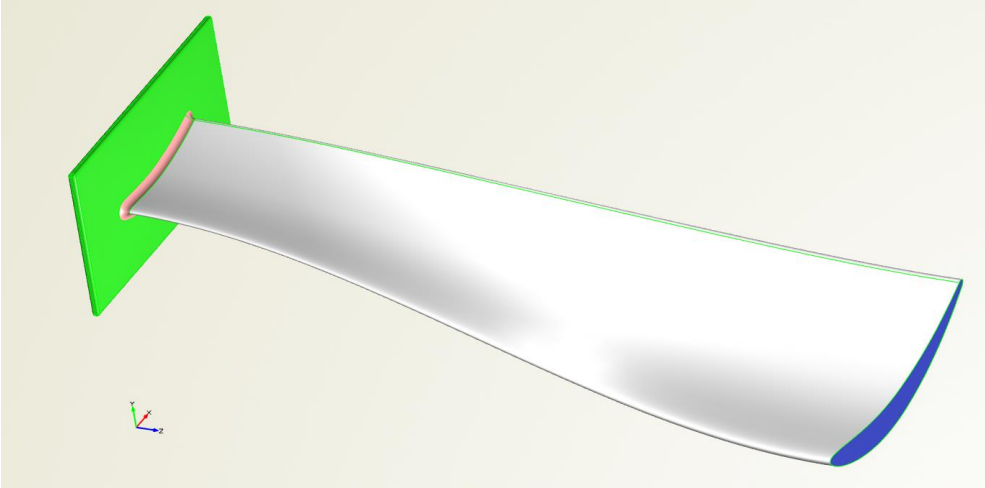
First Modeling Steps

This tutorial can be your starting point with CAESES. It shows you very basic modeling techniques and introduces the modeling concept of CAESES. As an example, a simple parametric rear wing model of a car gets created.

For more helpful information you can also check the tutorial videos on our website.

As a prerequisite you can also watch this video to get familiar with the graphical user interface.



After this tutorial, you will have a basic understanding of how to create parameterized geometries including curves, surfaces and solid bodies, which can later be used for automatic generation of design variants, CFD integration and design optimization.

CAESES Project

The resulting model can be found in the section *samples* > *tutorials* in the documentation browser.

Before you get started, make sure we use millimeters (at the bottom of the GUI):

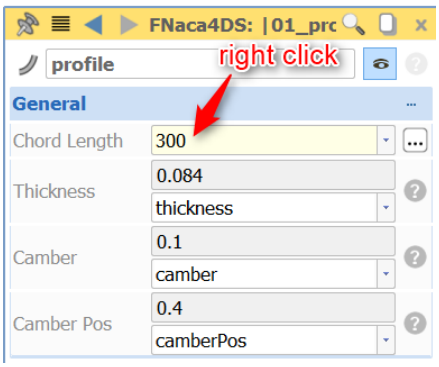
1 Base Profile

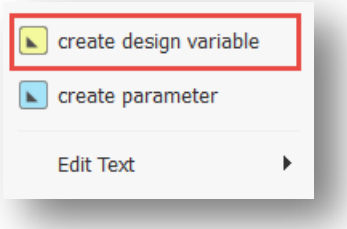
The starting point of this tutorial is a sketch of the 2D profile in the XY-plane. In this example, the wing shape will be based on the NACA 4-digit-series profile. NACA profiles are commonly used in the industry to form the basis of wings, airfoils and other aerodynamic shapes.

- Save your project first via *file > save project as* before you get started.
- Set the orientation to the XY-plane by clicking on the "Z"-icon at the bottom of the 3D view window.
- From the *CAD* pull-down menu (or from the *CAD* toolbar) select *curves > NACA-4DS curve*.
- Rename the created object to "profile".

The profile now appears in the 3D view and the leading edge is placed at the origin of the global coordinate system. The object editor shows the properties of the curve.

- Create a design variable by a right mouse button click on each marked value. Give them useful names and set the values according to the picture below:





2 Transform the 2D Profile: Image Curve and Transformation Chain

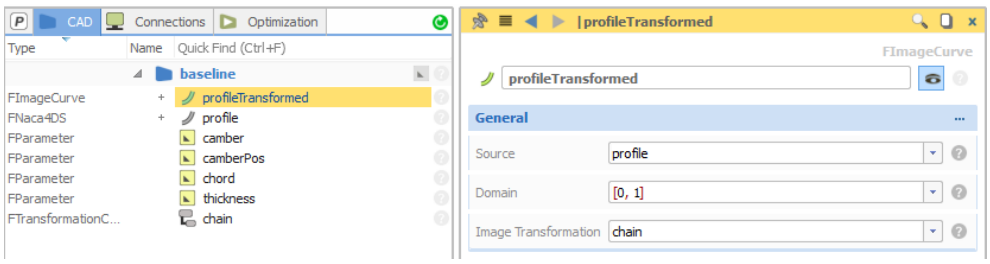
We want to transform the profile from the origin to a location in the 3D space. For this, an image curve is created which references the original curve. The image curve then also gets a transformation chain which will hold all the single transformations. You can still change the shape of the original profile, and the image will simple map everything into the new location.

- Create an image curve:
  CAD > curves > image curve.
- Rename the object to “profileTransformed”.
- Set the source of the image curve to “profile”.

✓ You can use the ALT-key, drag & drop or the pull down menu of the editor to enter the profile curve as input for the source.

Now we create a container for the upcoming transformations, and set it directly at the image:

- Create a transformation chain: *CAD > transformations > transformation chain.*
- Rename the new object to “chain”.
- Select the curve “profileTransformed” and set “chain” for the image transformation.



3 Transform the 2D Profile: Scaling and Rotation

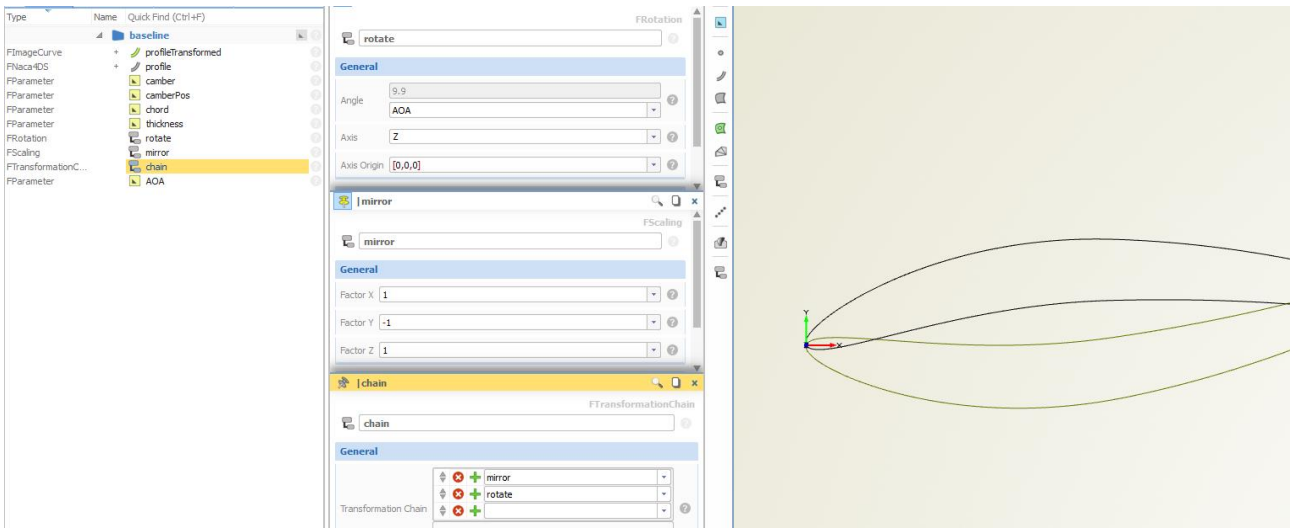
Now the transformations for the profile: We will create a scaling to flip the profile, and a rotation to apply an angle of attack.

- Create a scaling transformation: *CAD > transformations > scaling.*
- Rename the object to “mirror”. Set the Y-coordinate to -1.
- Put the transformation “mirror” into the transformation chain “chain”. Select “chain” and use drag & drop.

  See the screenshot below to check your chain and the mirror object.

Now the rotation:

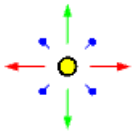
- Create a rotation transformation: *CAD > transformations > rotation.*
- Rename the object to “rotate”.
- Set the axis to “Z.
- Create a design variable for the angle and rename it to “AOA” (angle of attack).
- Put the transformation “rotate” into the transformation chain “chain”. Select “chain” and use drag & drop.
- Now you can try out to change the design variable “AOA”.



4

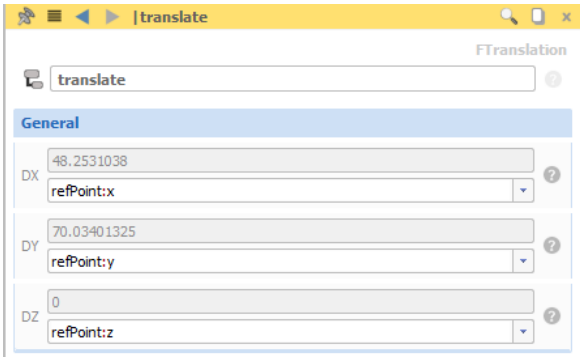
Transform the 2D Profile: Translation

Finally, we translate the image curve to the location in the 3D space.



- Create a point: *CAD > points > 3D point.*
- Rename the object to “refPoint”.
- Move the point in the 3D window by dragging the red and green arrows of the point somewhere above the origin.
- Keep the point selected and create a translation: *CAD > transformations > translation.*
- Rename the object to “translate”.

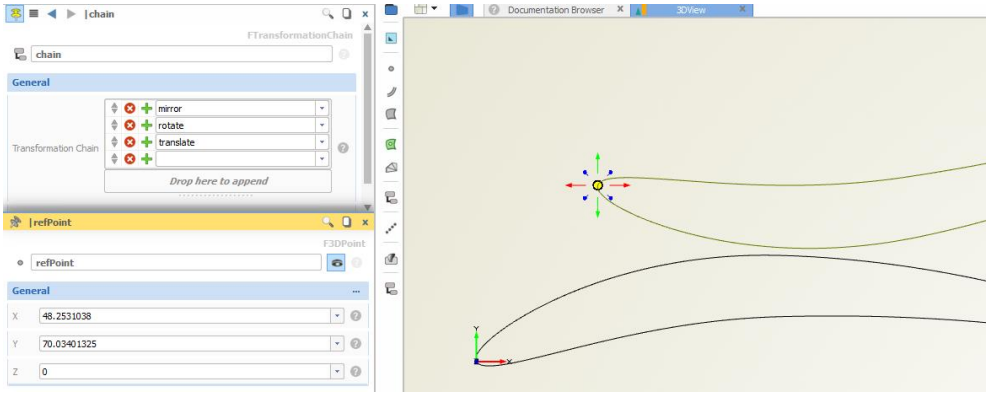
  You can see that the deltas are referencing the coordinates of the point.



- Put the transformation “translate” into the transformation chain “chain”:

  Select the object “translate” and use drag & drop.

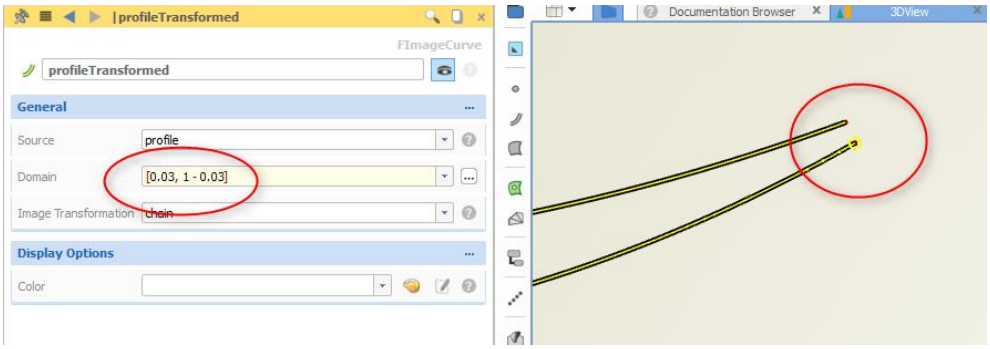
  Note the order of the transformations!



5 Fillet Preparation: “Cut Out” the Trailing Edge

In a later step, we want to create a nice rounded trailing edge. Hence, we need to create a little space at the trailing edge region.

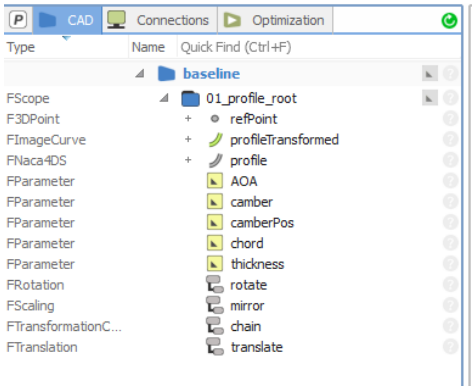
- Select the curve “profileTransformed” and set the curve domain to [0.03,1-0.03].



✓ Remember: All curves are parameterized definitions and the domain of the curve is in the range [0,1], referred as the parameter “t”. Thus in the above step, selecting a domain from 0.03 to 0.97 will create a subcurve. Also, the direction of the parameter domain is taken into account, and can be e.g. reversed if you make the selection from 0.97 to 0.03 within the image curve.

- Select all objects and create a scope via *CAD > scope*.
- Rename the scope to “01_profile_root”.

This puts all selected objects together into one scope.

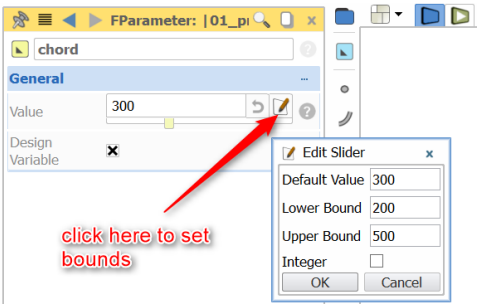


✓ Scopes are some folders that keep projects structured. By clicking on the scope icon in the tree, you can switch on and off the visibility of it.

6 Design Variables

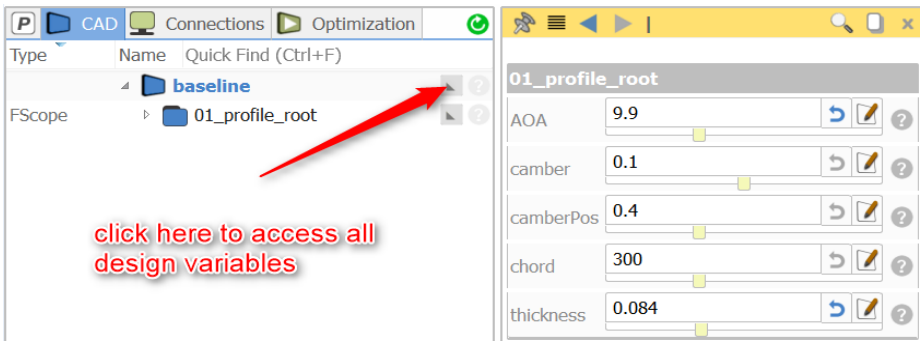
Now let's take a break and have some fun by creating sliders for all the design variables:

- Select a design variable and press the *edit* button next to the current value.
- Set the lower and the upper bounds.
- Press OK.
- Do this for all of your design variables. Use the following table to set the bounds:



Design Variable	Default Value	Lower Bound	Upper Bound
Chord	300	200	500
AOA	8	0	30
Camber	0.1	0	0.2
CamberPos	0.4	0.3	0.6
Thickness	0.1	0.05	0.15

- Click on the grey design variable icon next to the baseline, to access all the sliders for the design variables. Alternatively, select the baseline node or the scope 01_profile_root, which then also shows you these design variables. Modify the values – your model should now show these changes.

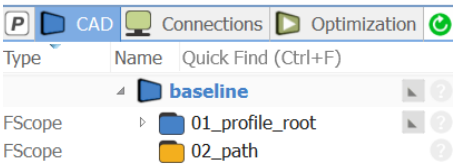


7 Sweep Path: Point Creation

The created profile will be swept along a 3D path. In this step we create the points of the path.

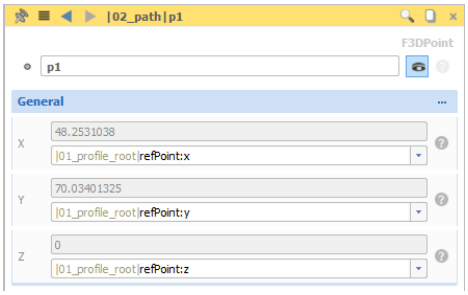
Since we want to work in a different "folder" (scope) here is a trick to enhance your workflow:

- Click on baseline and then create a new scope. Name it "02_path".
- Click with your middle mouse button on the scope. You can see that its color changed to orange.

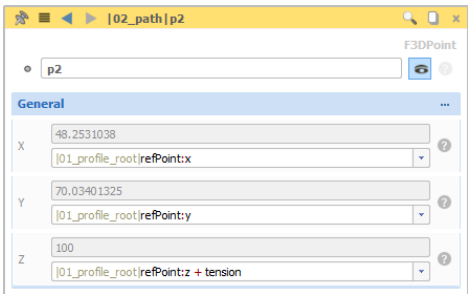


This scope is now your working scope, which means that each new object will be created in this scope. To undo this, click with the middle mouse button on the baseline. Test it!

- Select the point "refPoint" in the other scope and create a new point **"p1".**

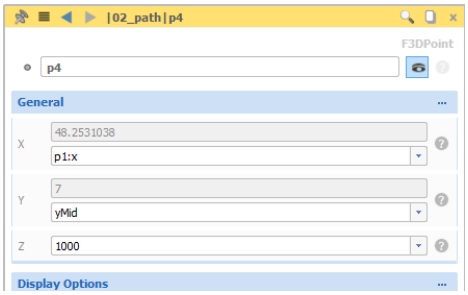


- Copy and paste this point within "02_path" to create a second point **"p2".**
- Select p2 and add the value 100 to the Z-coordinate.
- Mark the value 100 with the mouse and create a design variable "tension" for it (right click on the marked value).

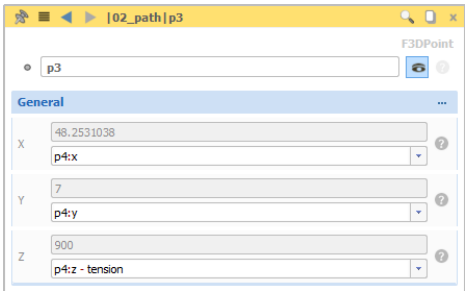


So far, we have two points, p1 and p2. Now create two more points, p4 and p3:

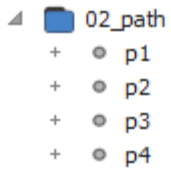
- Create a new point and rename it to **"p4"**.
- Set the coordinates of p4 to ➔ X = p1:x, Y = 7, Z = 1000.
- Create a design variable for the Y value and rename it to "yMid".



- Select p4 and create a new point **"p3"**.
- Modify the Z-coordinate and substract the tension design variable. Click behind p4:z, type the minus "-" and then press the alt-key to select the tension design variable in the tree.



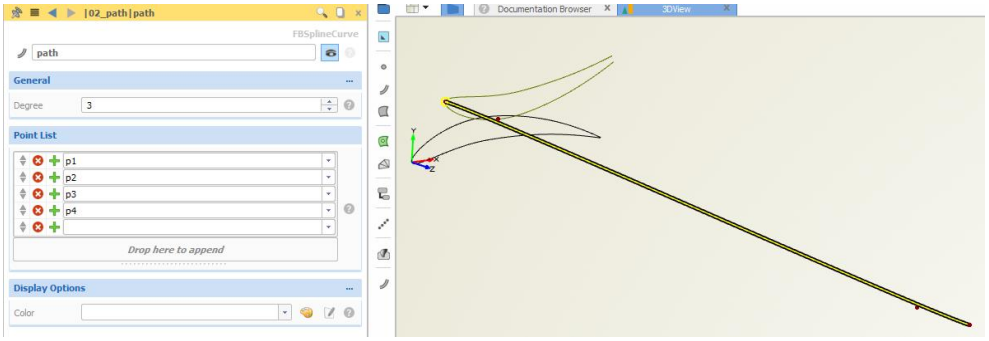
Your scope "02_path" now contains the four points:



8 Sweep Path: Curve Creation

Now, let's create a bspline curve using these 4 points:

- Select all points in the right order, starting with p1, and create a bspline curve: *CAD > Curves > B-Spline Curve.*
- Rename the curve to "path".
- Rotate the 3D view to check the curve.
  When being in a principal plane view (click on the XYZ-buttons at the bottom of the 3D window), you can also use the arrow buttons to change the view direction.
- Change value of "tension" – your curve should now changes, respectively.



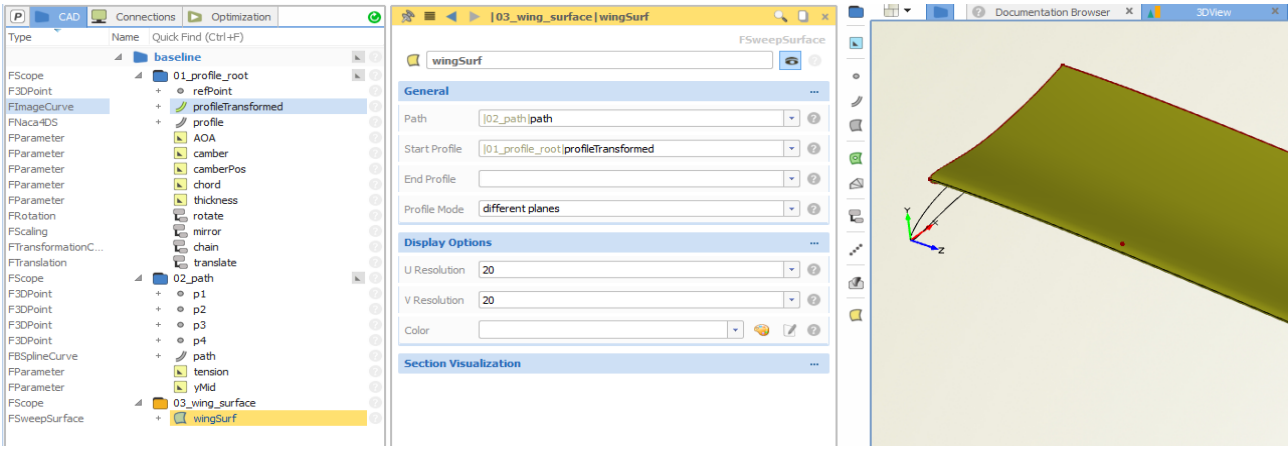
9 Wing Surface

In this step, we create the wing by sweeping the profile along the path.

- Click with the middle mouse button on baseline.
- Create a new scope and rename it to “03_wing_surface”.

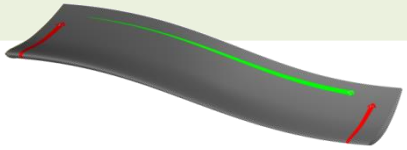
✓ If other objects are selected during a scope creation action, these will be automatically put into the new scope which is a convenience mechanism!

- Click with the middle mouse button on this scope to make it your working scope.
- Create a sweep surface: *CAD > Surfaces > Sweep Surface.*
- Rename the surface to “wingSurf”.
- Click into the field next to path. Press the alt-key to select the curve “path” from the previous scope and press enter.
- Set the start profile by choosing “profileTransformed” from the first scope.
- Now you should be able to see the wing surface in the 3D window:



✓ Surfaces have a mathematical domain, and hence they are similar to curves. They are parameterized surfaces, but instead of having a parameter “t”, the surface parameters are called “u” and “v”. Both run again in the interval [0,1]. This is of interest when working with commands.

You can visualize the uv-parametrization: do a right click on the surface and select “show UV Orientation”.

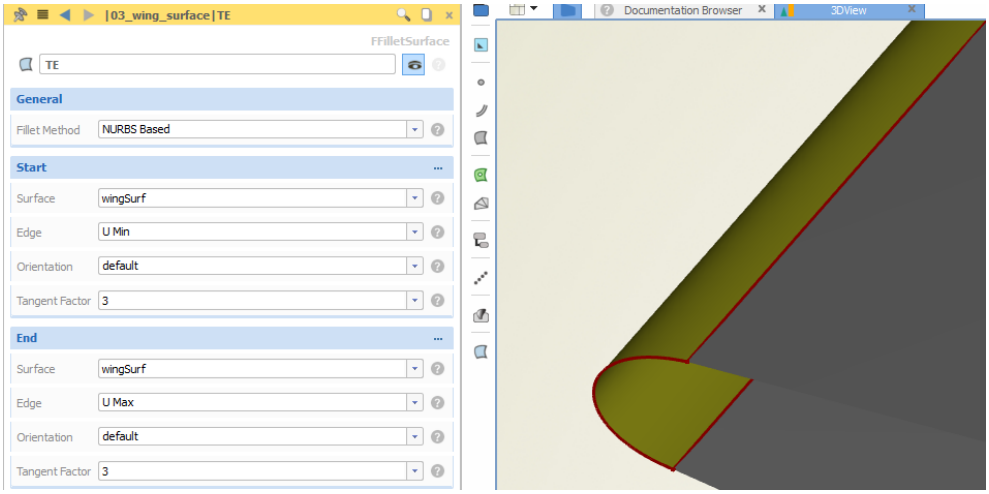


10 Trailing Edge Surface

To finalize the wing, we create a smooth fillet at the trailing edge of the wing:

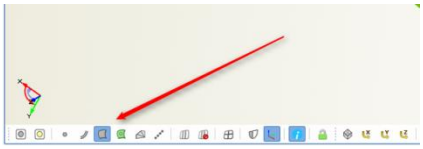
- Create a fillet surface: *CAD > Surfaces > Fillet Surface.*
- Rename the surface to “TE”.
- Set the attributes as shown in the image below.

We use tangent factors = 3 which defines how strong the effect of the tangential directions are taken into account.

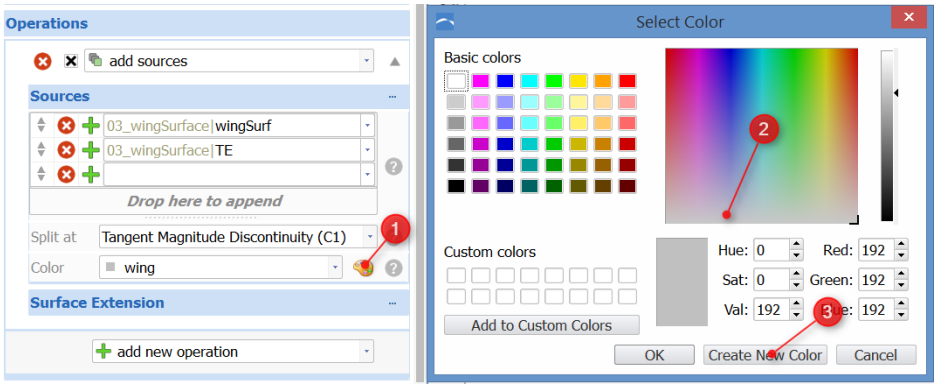


11 Solid Wing: BRep and Colors

Now we create a solid geometry for the wing, i.e. we want to close it. For this task we use so called BReps.



- Disable the surface filter at the bottom of the 3D view. *See also step 13 for more information.*
- Select the surfaces "wingSurf" and "TE" and create a BRep: *CAD > BReps > BRep.*
- Rename this BRep to "wingClosed".
- In order to give the surfaces a specific color, create a new color and rename it to "wing".



✓ Colors are not just used to make the model look nice; they play an important role to manage your boundaries! Depending on the export type, each face of the BRep which is marked by a different color will be exported as a separate boundary with the name of the color.

The BRep gives you the possibility to group different surfaces to one boundary; in the last step we combined "wingSurf" and "TE" to the boundary "wing".

But you could also create a new boundary for your "TE" surface. Just create a new "Add Sources Operation" and use the "TE" surface as source. Then assign a new color to this operation.

Remember: Once you set a color for a boundary, no other operation changes the color. For example you cannot change the color of the complete BRep when all sources already have a color.

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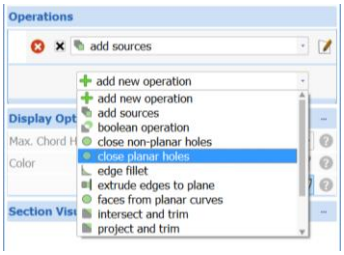
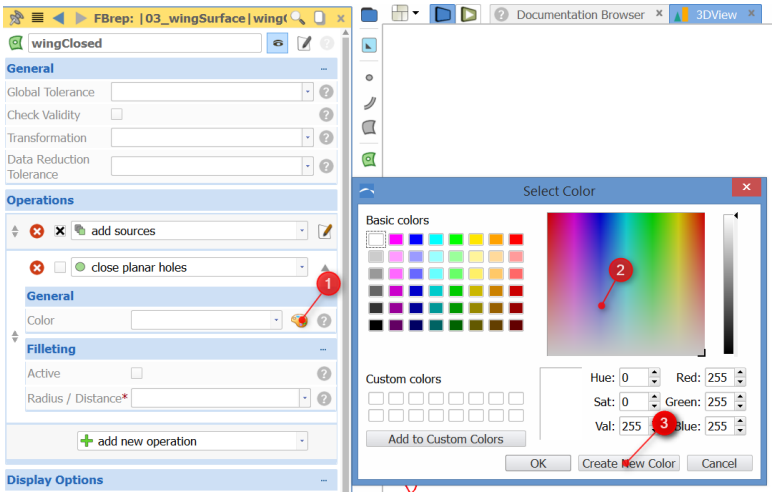
Solid Wing: Operation Close Planar Holes

For this new BRep object, we create an operation to close the planar holes:

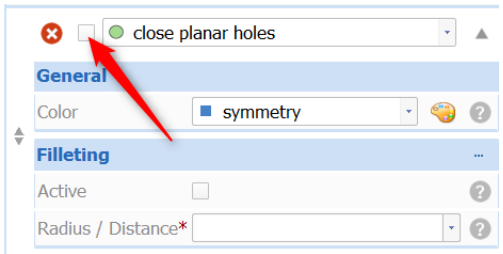
► Create the new operation “Close Planar Holes”:



► Create a new color and rename the color to “symmetry”.



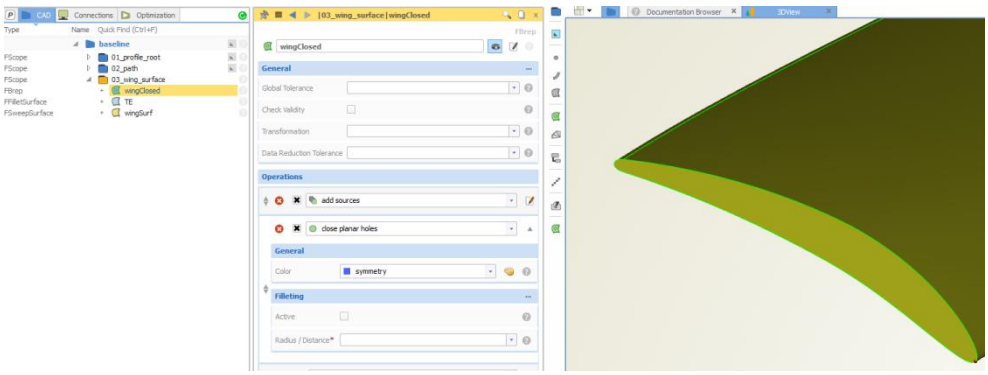
► Activate the operation:



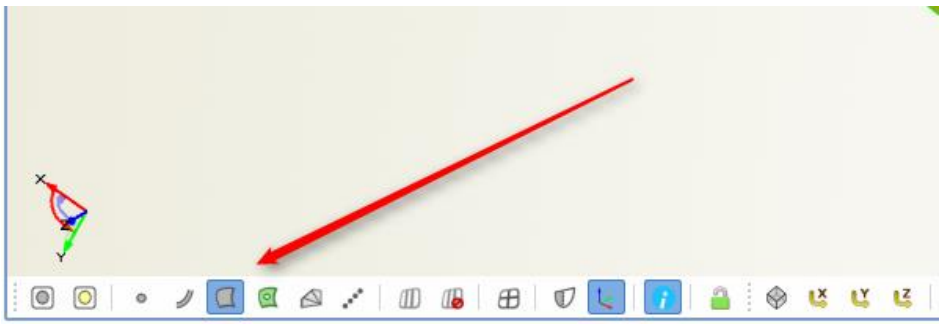
13

Displaying and Filters

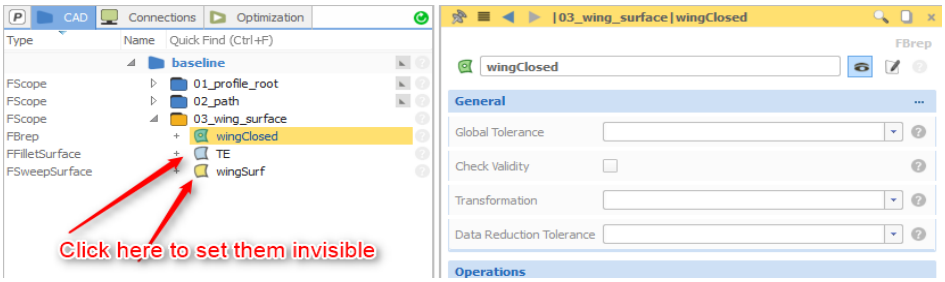
In CAESES, everything is 3D. All your objects are in a 3D view and you can use filters to show only objects of interest. In addition, you can change the visibility of objects to switch them off, for instance. So this is your situation right now:



► In order to look only at the BRep object, you can either use the type filter at the bottom of the 3D window (turns off the surfaces).



► Alternatively, set the surfaces invisible by clicking on the icons in the tree:



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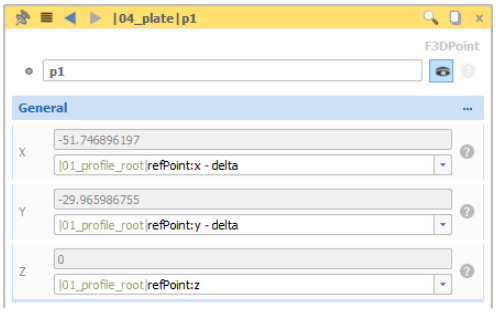
Side Plate Contour: Point 1

We will now start to create the side plate, where the wing is mounted. The points are referenced to the profile, so that the plate always moves according to changes of the wing. It's a bit tricky, but you learn a lot about dependencies ... ;-)

- Click with the middle mouse button on baseline.
- Create a new scope called "04_plate".
- Click with the middle mouse button on this scope to make it your working scope.

✓ If you don't do this scope activation with the middle mouse button, you can simply drag all your created objects into the new scope afterwards.

- Select the point "refPoint" from the first scope and create another point.
- For this point **p1** substract 100 from the X-coordinate.

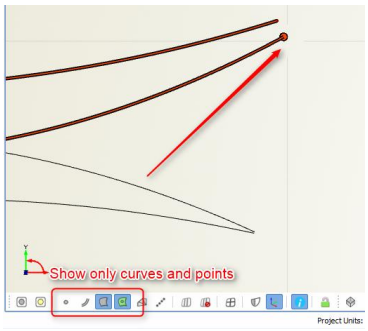


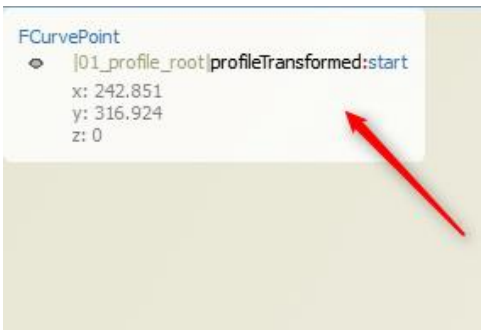
- Right click on the marked "100" to create a new parameter called "delta".
- Substract "delta" from the Y-coordinate.

15 Side Plate Contour: Point 2

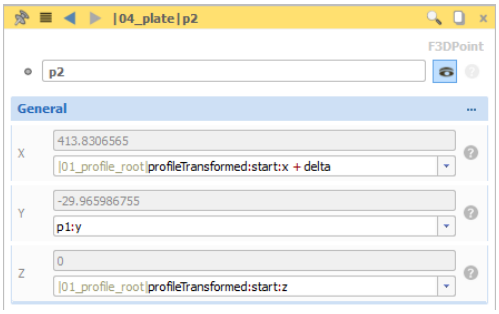
Create the second point of the side plate contour:

- Select the start point of the curve “profileTransformed” in the 3D view.
  Note that once you hover across the start point, it gets displayed in the upper left corner of the 3D view.





- While the start point is selected, create a new point **p2**.
- Add “delta” to the X-coordinate.
- Delete the entry for the Y-coordinate and set it as the Y-coordinate of p1, i.e. “p1:y”:



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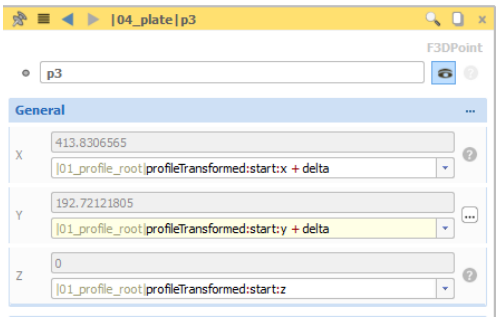
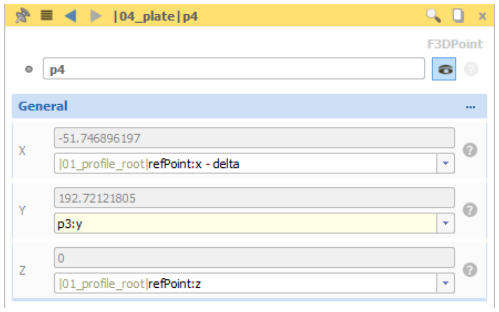
Side Plate Contour: Point 3 and Point 4

Two plate points are ready, now we need the other two corner points:

- Again, select the **start point** of the curve "profileTransformed" in the 3D view and create a new point **p3**.
- Add "delta" to the X- and Y-coordinate.



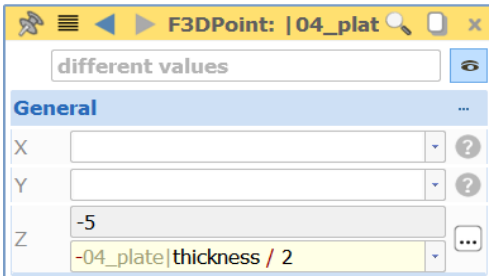
- Select the point "refPoint" from the first scope and create a last point **p4**.
- Substract "delta" from the X-coordinate.
- Set "p3:y" as the Y-coordinate.



17 Side Plate Contour: Thickness

For the Z-coordinates of the 4 points, we want to use a parameter to control the thickness:

- Create a new parameter called “thickness” via *CAD > parameters > parameter*.
- Set the value to 10.
- Select all 4 points and set the Z-coordinate to “ –thickness/2”.

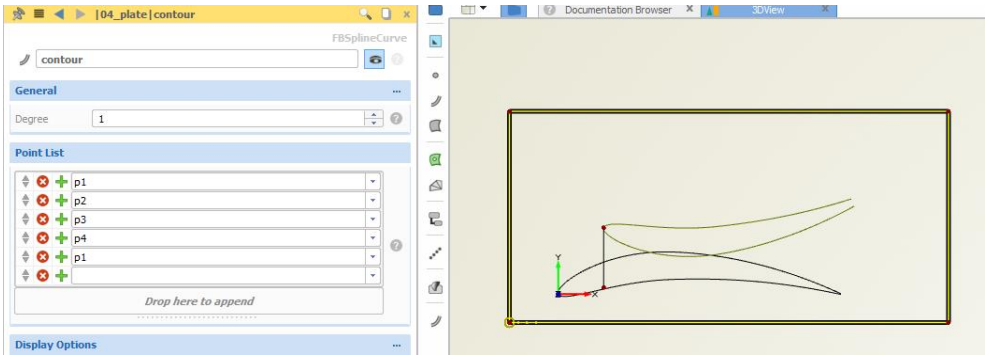


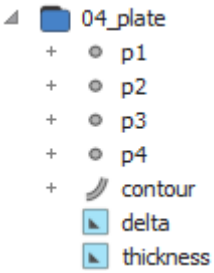
18

Side Plate Contour: Curve

Now we have manually set up the 4 points. Based on these, we create a linear bspline curve for the contour.

- Select all 4 points starting with p1 and create a bspline curve:
  CAD > Curves > B-Spline Curve.
- Rename the curve to “contour”.
- Set the degree to 1.
- Add p1 to the end of the list, so that the curve is closed (“closed loop”).





Great! Now we got this slightly tricky part. You can change the profile (e.g. by modifying “AOA”), and the side plate contour follows parametrically.

Let’s move on to create a solid from this contour …

19

Side Plate

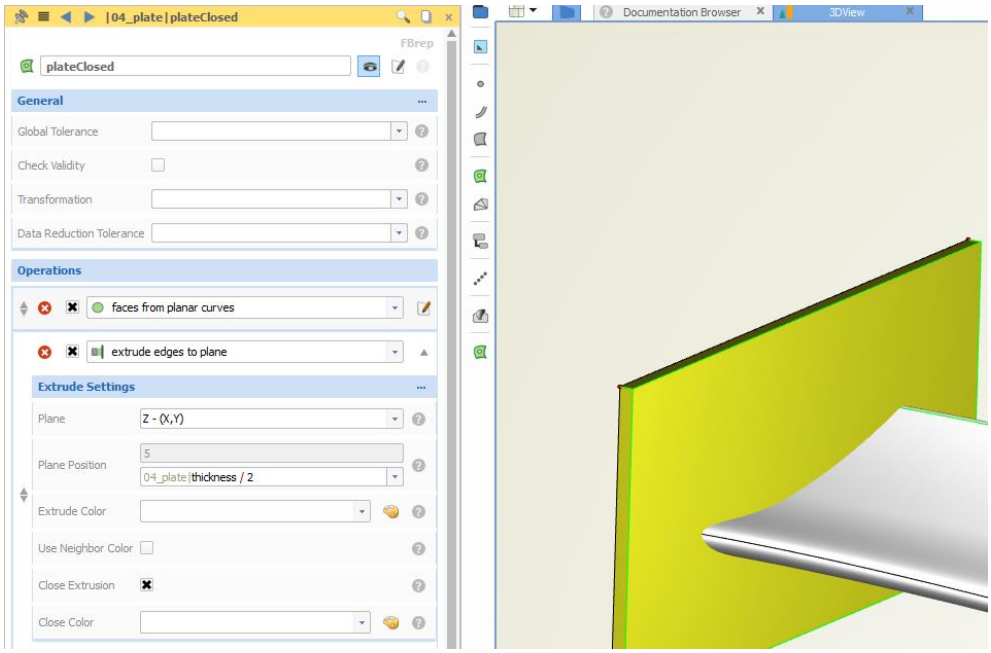
From this contour curve, we can create a solid body with a certain thickness.

- Select the curve “contour” and create a BRep: *CAD > BReps > BRep.*

The BRep object automatically takes the selection and detects the input type (FCurve), which is why the first operation is now “faces from planar *curves*”.

- Set a name, such as “plateClosed”.

- Add a new operation “extrude edges to plane”.

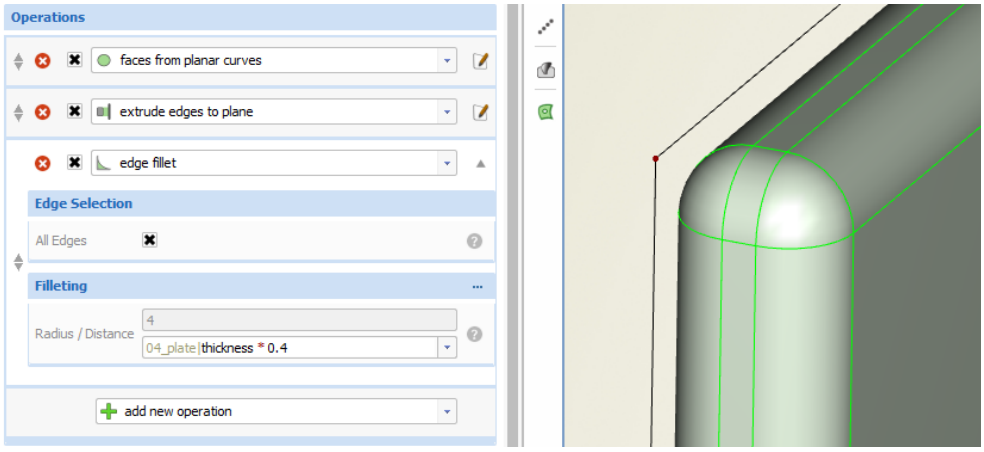


- Select the plane “Z”.
- Set the plane position to: thickness/2.
- Activate “close extrusion”.
- Activate the operation (i.e. activate the toogle next the the operation name)

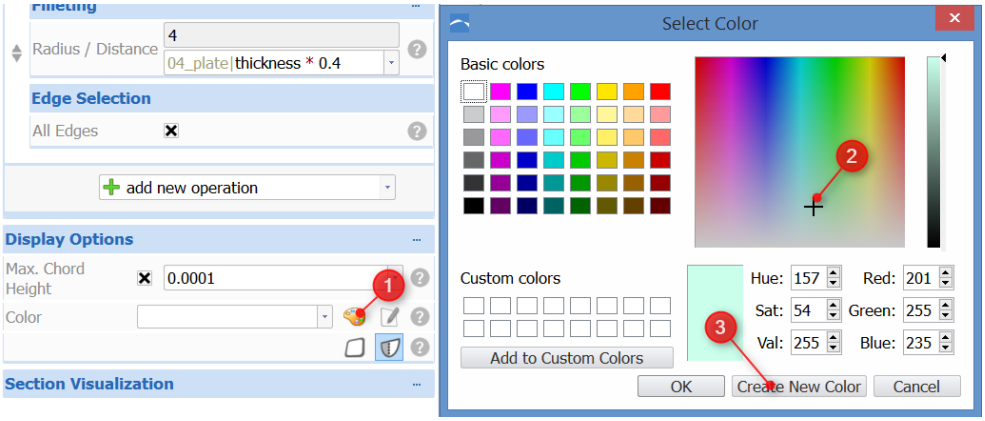
20 Side Plate Fillet

Finally, let's create a plate fillet:

- Add the new operation "edge fillet".
- Set the radius to : thickness*0.4.
- Activate the toggle "All edges".
- Activate the operation.



- Create a new color "plate" for the complete BRep (at the bottom of the object!):



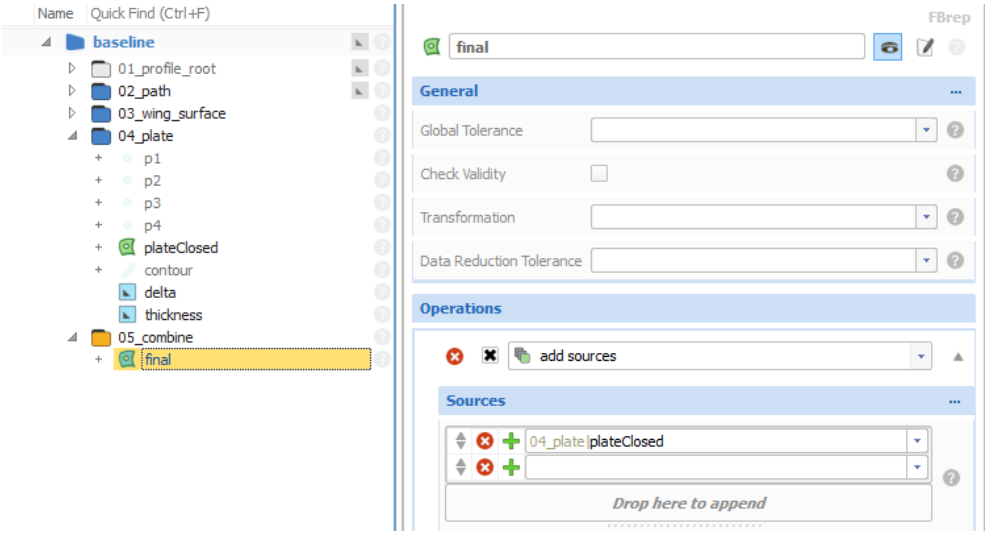
21

Combine Side Plate and Wing

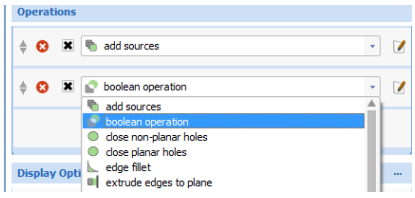
In this step we will create a solid by combining the plate and the wing.

- Click with the middle mouse button on baseline node.
- Create a new scope called “05_combine”.
- Click with the middle mouse button on this scope to make it your working scope.

- Create a new Brep: *CAD > BReps > BRep.*
- Rename it to “final”.
- Add the Brep “plateClosed” to the source of the first operation “add sources”.
  You can use drag & drop for this.



- Add the new operation “boolean operation”.
- Set the Brep “wingClosed” to the source of this operation.
- Set the operation of the “boolean operation” to “Union” (this is the default).
- Activate the operation.

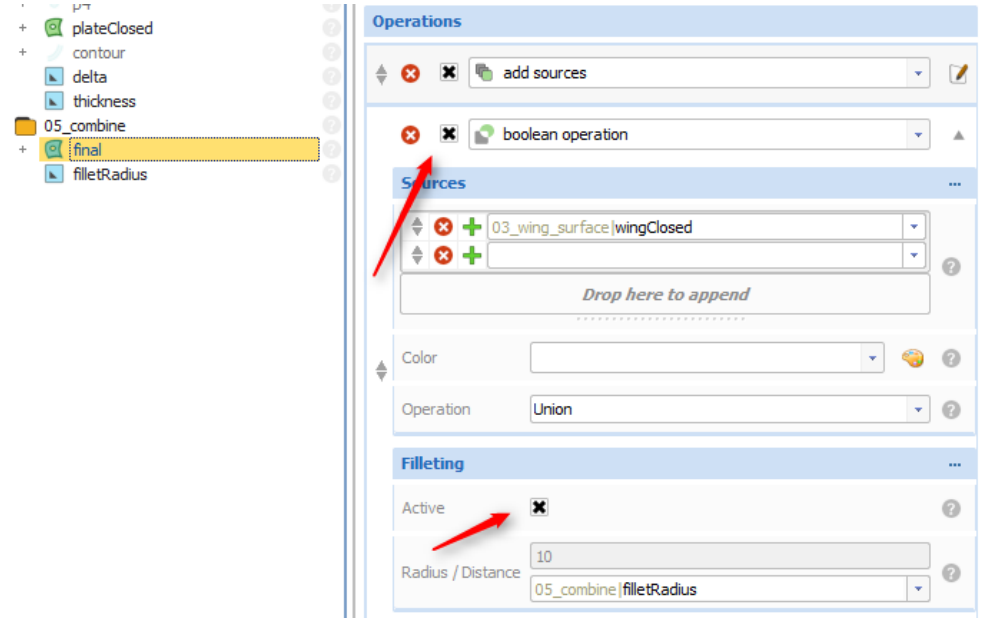


See the next page.

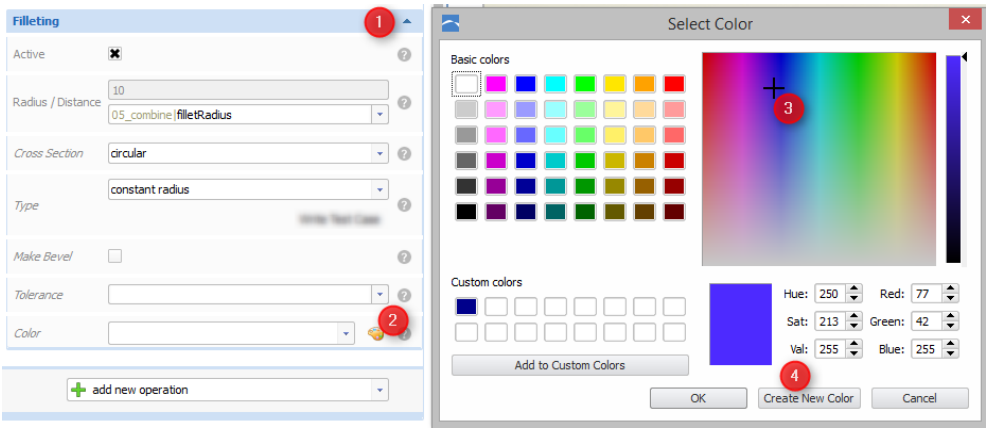
22 Fillet between Side Plate and Wing

We also want to have a nice fillet surface between the wing and the side plate:

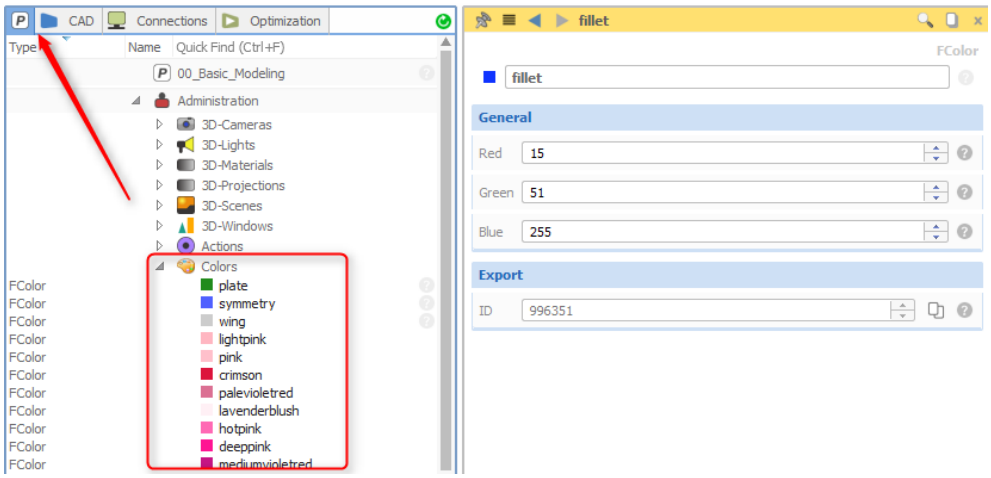
- Set the radius of the fillet to 10 and create a new parameter for this value called “filletRadius”.
- Activate the filleting process.



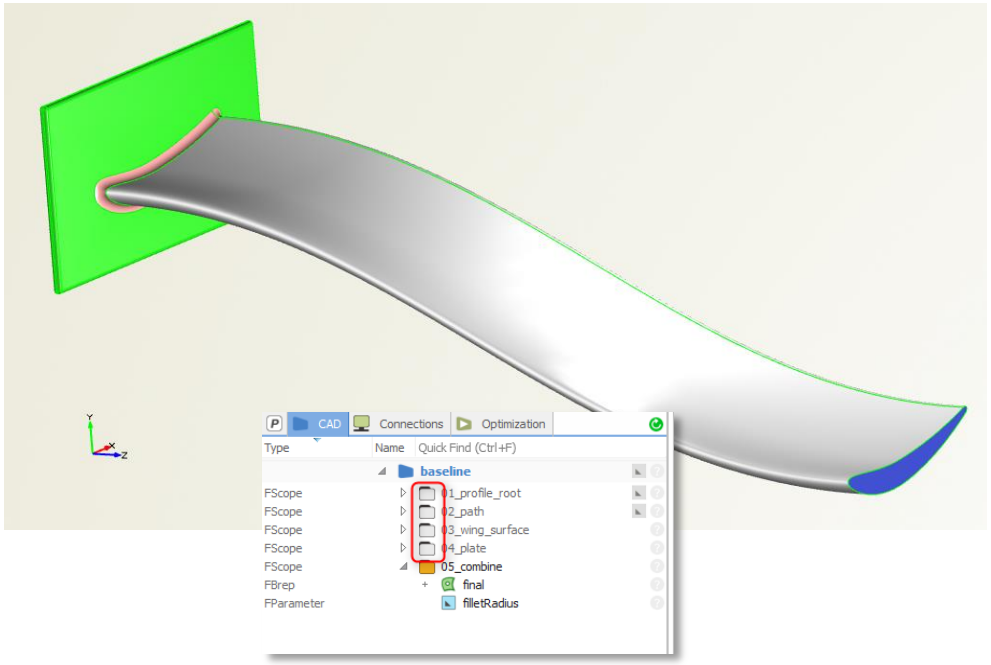
- Click on the dots of the category “Filleting” to get more fillet options.
- Create a new color called “fillet” for the fillet surface:



✓ If you want to change the values of your created colors, just click on the “P” (Project) tab in the tree and search your color in the “Colors” node. Select your color and modify the RGB values.



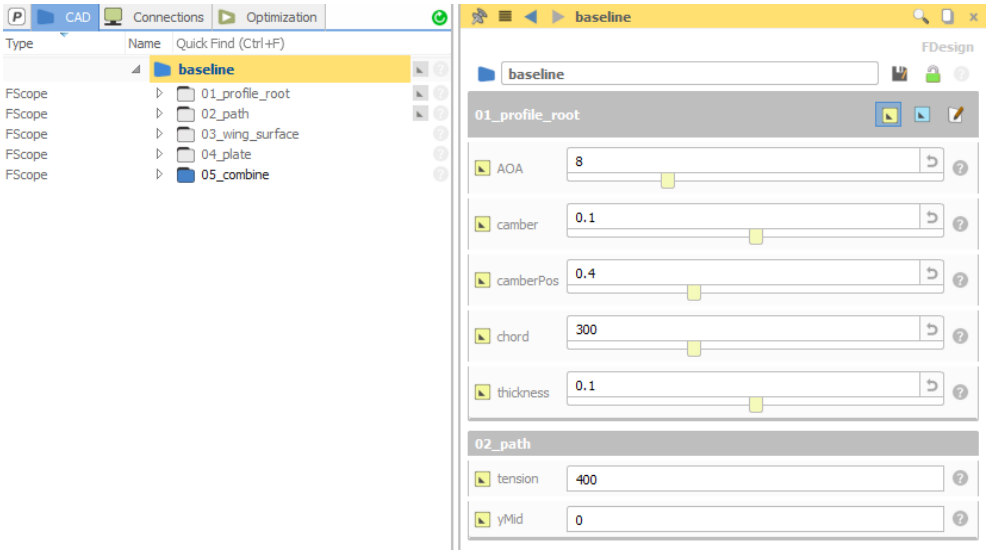
✓ Click on the 01-04 scopes to set them invisible. With this, you see only your resulting solid body in the 3D view.



23

Play with the Design Variables of Your Model!

Excellent! You've made it up to this point. Congratulations! This is your first parametric model in CAESES. Now, take your time again to try out your design variables. Simply select the baseline node and change the values. Does it work?



Are you ready for some additional steps? Then see the next pages!

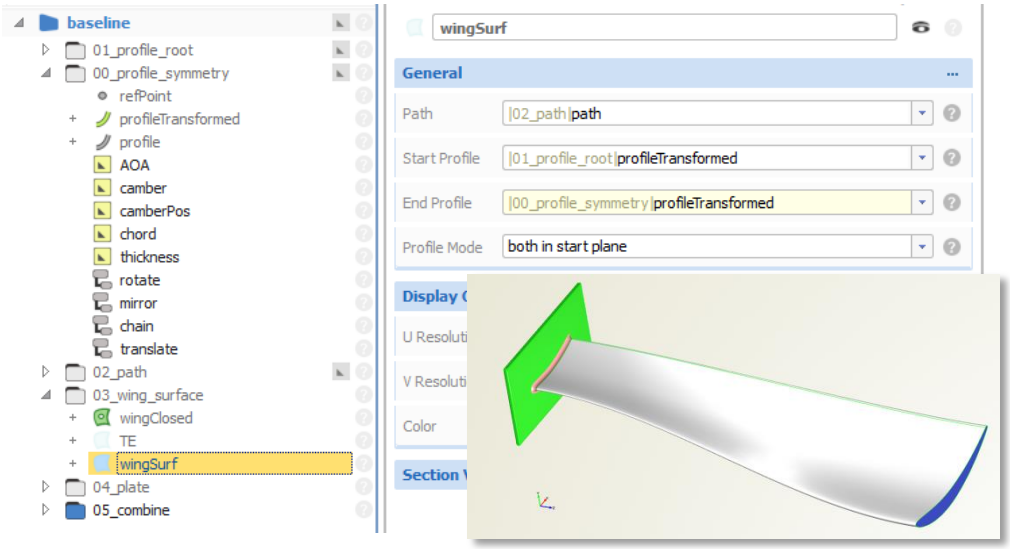
24

Separate Profile for the Symmetry Plane

Let's introduce a separate profile with separate control at the symmetry plane. For this, we use the ability of the sweep surface to have different start and end profiles:

- Click with the middle mouse button on baseline.
- Copy the scope "01_profile_root" and past it into the baseline directory.
- Rename the scope to "00_profile_symmetry".

- Select the sweep surface "wingSurf" from the third scope.
- Set the "Profile Mode" to "both in start plane".
- Set the end profile to "00_profile_symmetry|profileTransformed" from the new created folder.



- Select the scope "00_profile_symmetry" and change the values of the design variables in the object editor. The model should now change in the symmetry plane region.

✓ Try to export the solid model: Select the BRep "final" and choose *File > Export > STL*, for instance.