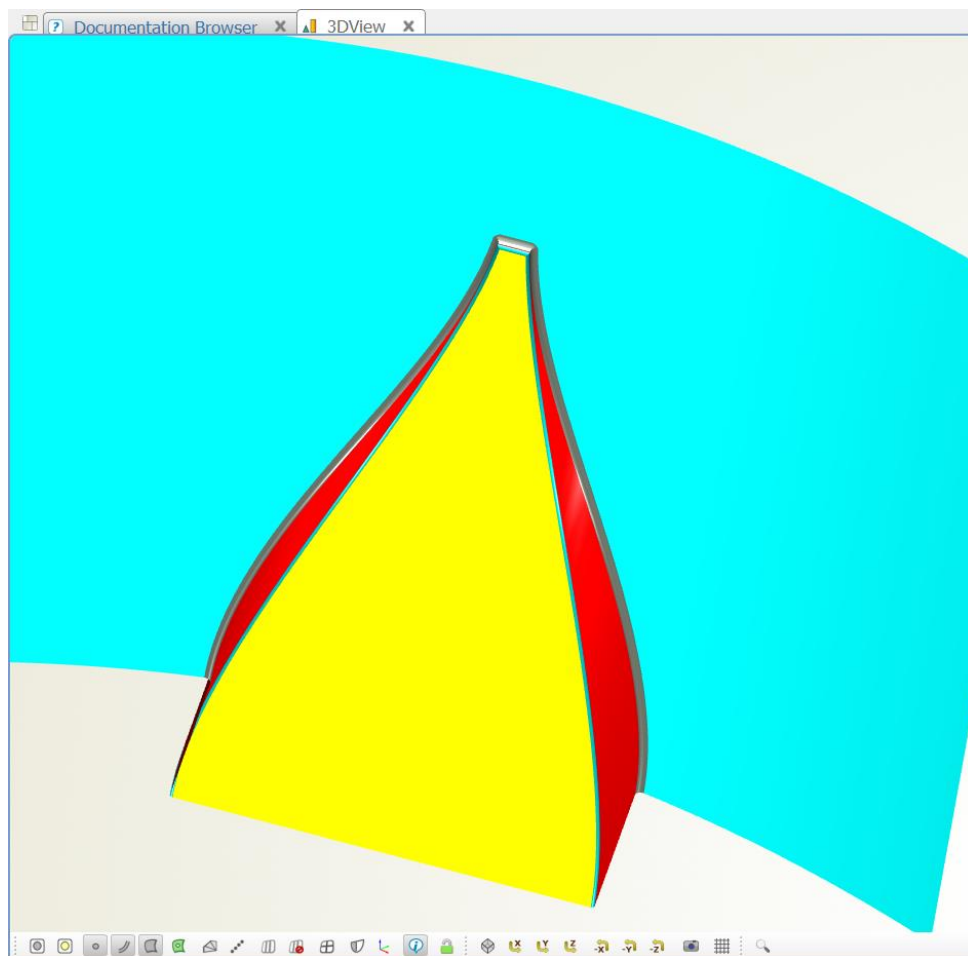


## BRep NACA Duct

This is another tutorial which gives you more practice with BReps. It is recommended to go through the tutorial “Introduction to BReps” beforehand.

NACA ducts – also known as NACA inlet or NACA scoop – are widely used in aerospace and motor sports applications. In this tutorial, such a duct gets created along with smooth fillets for the edges.



## CAESES Project

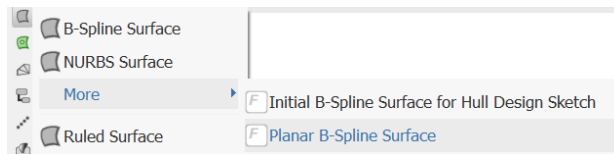
The resulting model can be found in the section *samples > tutorials* of the documentation browser.

# 1

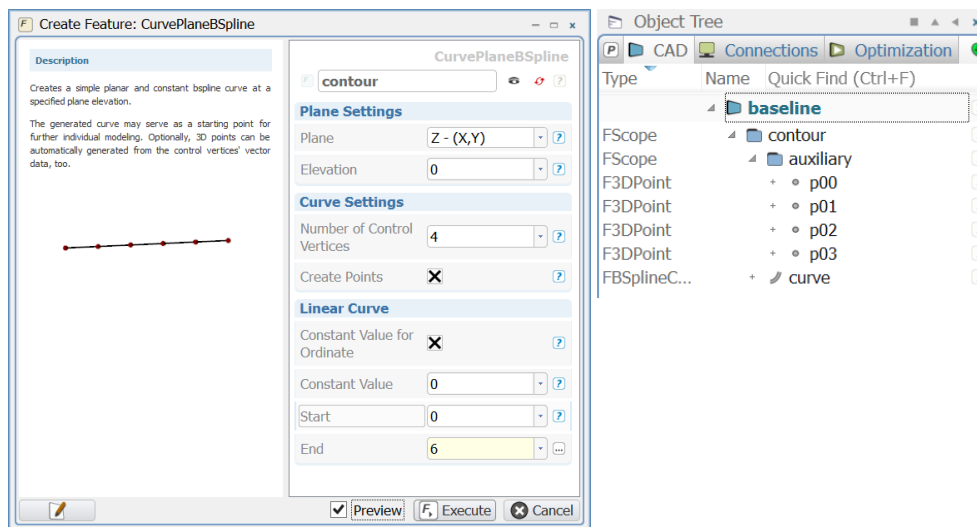
## Duct Contour

We start with a simple b-spline curve for the NACA duct contour.

- Save the project (CTRL + s).
- Create the feature “Planar B-Spline Curve” via *menu > CAD > curves > more > planar b-spline curve*.



- Change the name to “contour”.
- Set *Number of Control Vertices* to “4”.
- Set *End* to “6”.
- Select *Execute*.

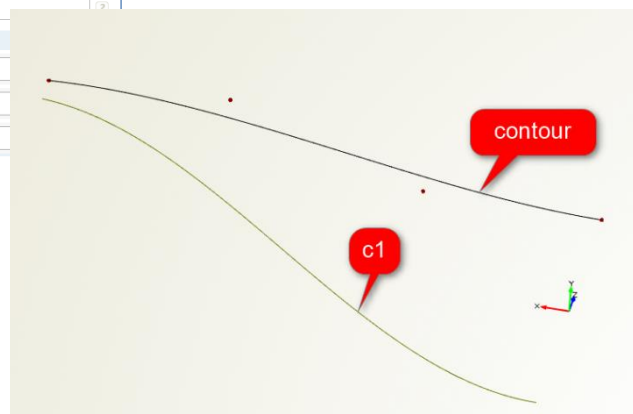
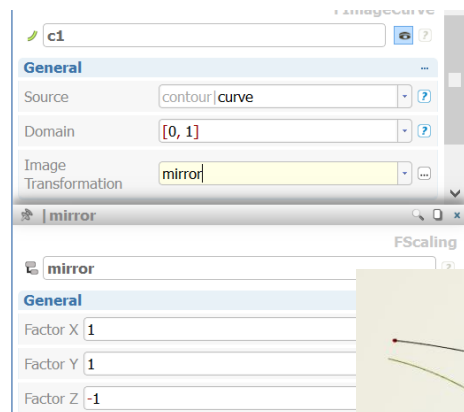
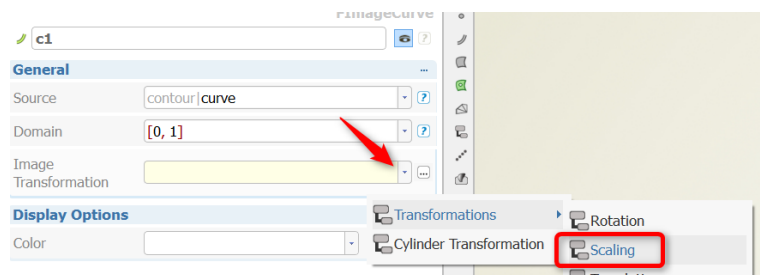


## 2

### Define Shape

We will set the coordinates of the four points and create an image curve.

- ▶ Press the CTRL key and select “p00” and “p01”.
- ▶ Set Z to “2”.
- ▶ Select “p02”, set Y to “1.75” and set Z to “0.4”.
- ▶ Select “p03”, set Y to “1.75” and set Z to “0.15”.
- ▶ Select “|contour|curve” and create an image curve via *menu > CAD > curves > imagecurve*.
- ▶ Click on the drop down menu of *Image Transformation* and create a scaling.
- ▶ Change the name to “mirror”.
- ▶ Set *Factor Z* to “-1”.

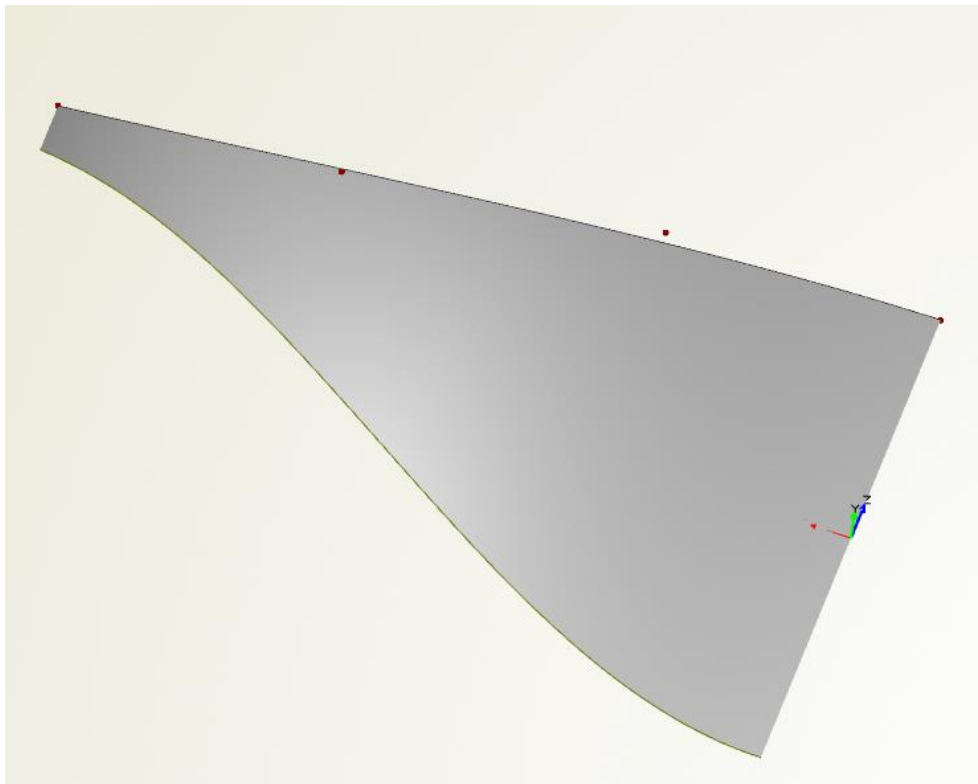


### 3

#### Main Surface

In this step you will create the duct surface between the contour curves.

- ▶ Press the CTRL key and select “curve” and “c1” and drag and drop them into the scope contour.
- ▶ Select the two curves “contour|curve” and “contour|c1”.
- ▶ Create a ruled surface via *menu > CAD > surfaces > ruled surface*.
- ▶ Change the name to “duct”.

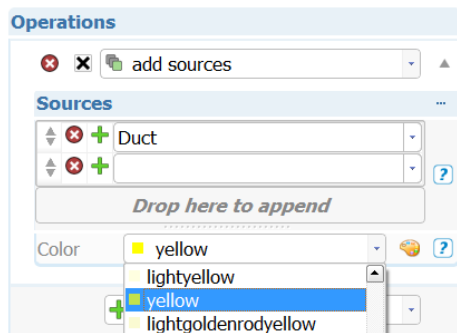
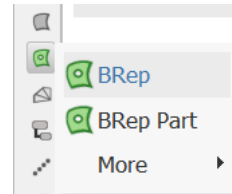


# 4

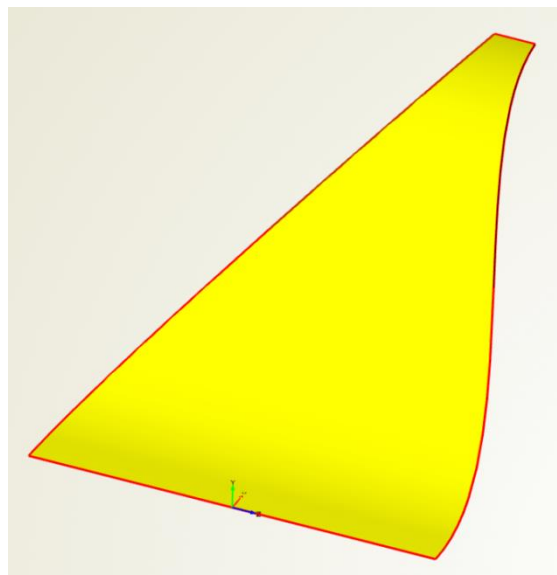
## Create a BRep

Based on the previous surface, we will now continue to work with BReps. The idea is, to extrude the edges to the z-plane later on.

- ▶ Select the surface "duct".
- ▶ Create a BRep via *menu > CAD > breps > BRep*.
- ▶ Change the name to "ClosedDuct".
- ▶ Set the color of the operation *add sources* to "yellow".



✓ Choosing colors for brep faces is very useful since, in operations, the faces can be addressed by colors. For instance, you can create a fillet between faces with two specified colors, or you can remove a face that has a specific color.

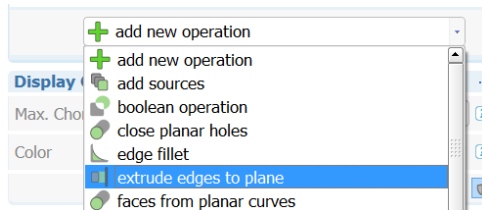


5

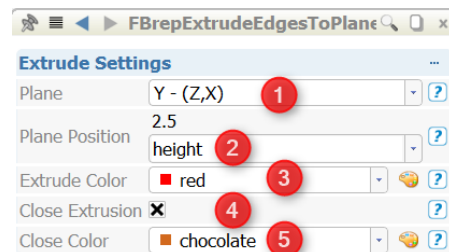
## BRep Operation Extrude Edges to Plane

In this step, we will extrude the edges of the Brep to a plane and close it at the top.

- Click on *add new operation* and create the operation *extrude edges to plane*.



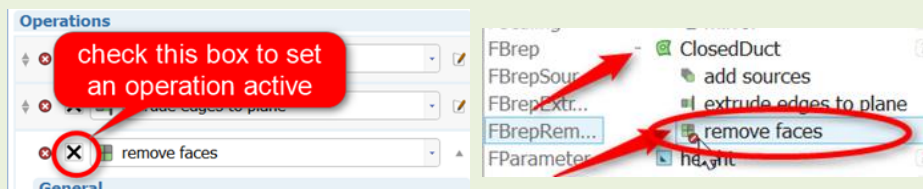
- Set *Plane* to "Y- (Z,X)".
- Create a parameter "height" for *Plane Position* and set the value to "2.5".
- Set *Extrude Color* to "red".
- Check *Close Extrusion*.



The Argument *Close Color* pops up after checking the close extrusion box.

- Set *Close Color* to "chocolate".
- First click on the "+" button in front of the icon of the brep "ClosedDuct", and click once on the icon of the brep operation *extrude edges to plane* in the object editor to set the brep operation active. See the comments below:

✓ BRep operations: New added BRep operations are created as inactive by default. After adding a new operation you can configure it with all inputs and set it active afterwards. Toggle the check box in front of the operation so as to set an operation active. Alternatively, you can click on the "X" button in front of the BRep object in the object tree and click on the operation. This also sets the operation active or inactive.

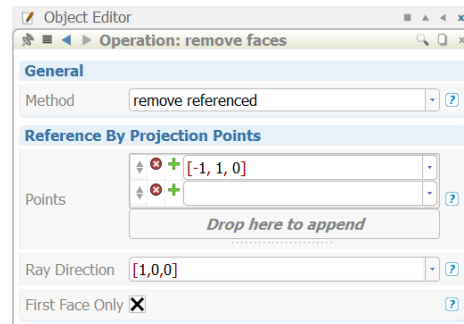


6

## BRep Operation Remove Face

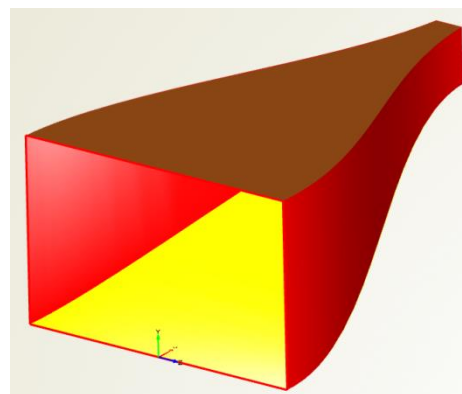
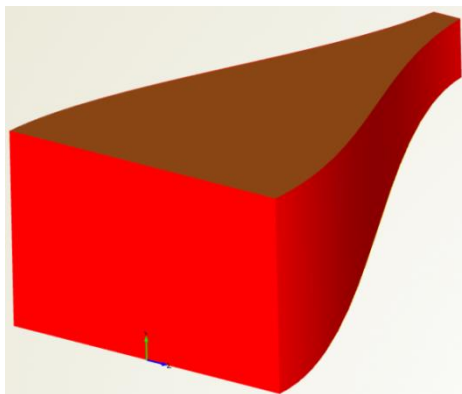
We will use the brep operation *remove face* in order to create an outlet surface.

- ▶ Select "ClosedDuct".
- ▶ Select *New Operation* and choose *remove face*.
- ▶ Change *Method* to "remove referenced".
- ▶ Set *Points* to "[1,1,0]".
- ▶ Set *Ray Direction* to "[1,0,0]".
- ▶ Activate the operation.



✓ *Ray Direction* defines the direction in which the point will be projected to address the face which will be kept or removed.

✓ BRep and colors: Faces with the same color will be treated as a group of faces. In this example the red faces are four in total and they are treated as one group. Because we want to create an outlet face we need to remove the front face of the group and close the face in the next step.



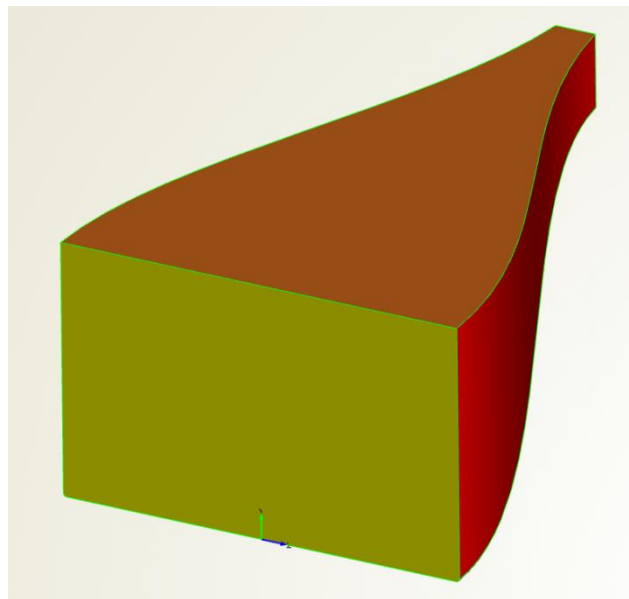
✓ The method "keep referenced" keeps the referenced face by the means of the point and the ray direction. "remove referenced" will remove the referenced face of the brep.

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### BRep Operation Close Planar Holes

Now we will close the duct again and assign a different color to have a separate face.

- ▶ Select "ClosedDuct".
- ▶ Select *New Operation* and choose *close planar holes*.
- ▶ Set *Color* to "olive".
- ▶ Activate the operation.



✓ The brep operation "close planar holes" closes all open edges which are planar. It can be used to close the inlet or outlet of your flow domain.

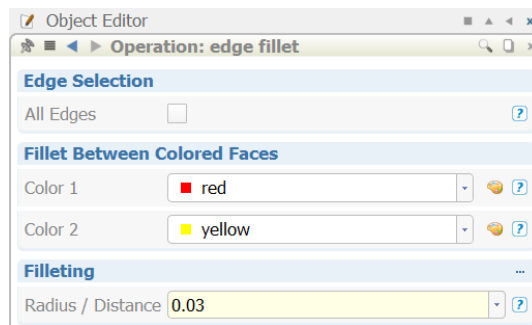


## 8

**BRep Operation Edge Fillet**

We will create a fillet and address the faces by colors. This step illustrates another reason, why we removed the face and created a new one. Now we are able to create the fillet just at the positions we want the fillet to be.

- ▶ Select "ClosedDuct".
- ▶ Select *New Operation* and choose *edge fillet*.
- ▶ Switch off *All Edges*.
- ▶ Set *Color 1* to "red".
- ▶ Set *Color 2* to "yellow".
- ▶ Set *Radius / Distance* to "0.03".
- ▶ Activate the operation.



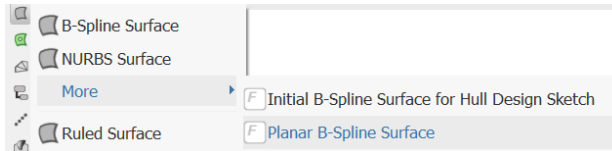
✓ This is a good example why you should make use of coloring brep faces. With the colors it is easy to create the fillet in a specified area. Some brep operations ("Boolean operations") readily offer a fillet option so for these there is no extra fillet operation required.

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## Create a Body Surface

We will create a b-spline surface which represents the body in which the NACA duct will be embedded.

- Create again a b-spline plane. Via *menu > CAD > surfaces > more > Planar B-Spline Surface*.



- Change the name to "Plane".
- Set *Plane* to "Y - (Z,X)".
- Set *Elevation* to "1.4".
- Set *CP in U-Direction* to "2".
- Set *CP in V-Direction* to "3".
- Set *Start First Axis* to "-5".
- Set *End First Axis* to "5".
- Set *End Second Axis* to "10".
- Click *Execute* to create the feature.

The 'SurfacePlaneBSpline' dialog box contains the following settings:

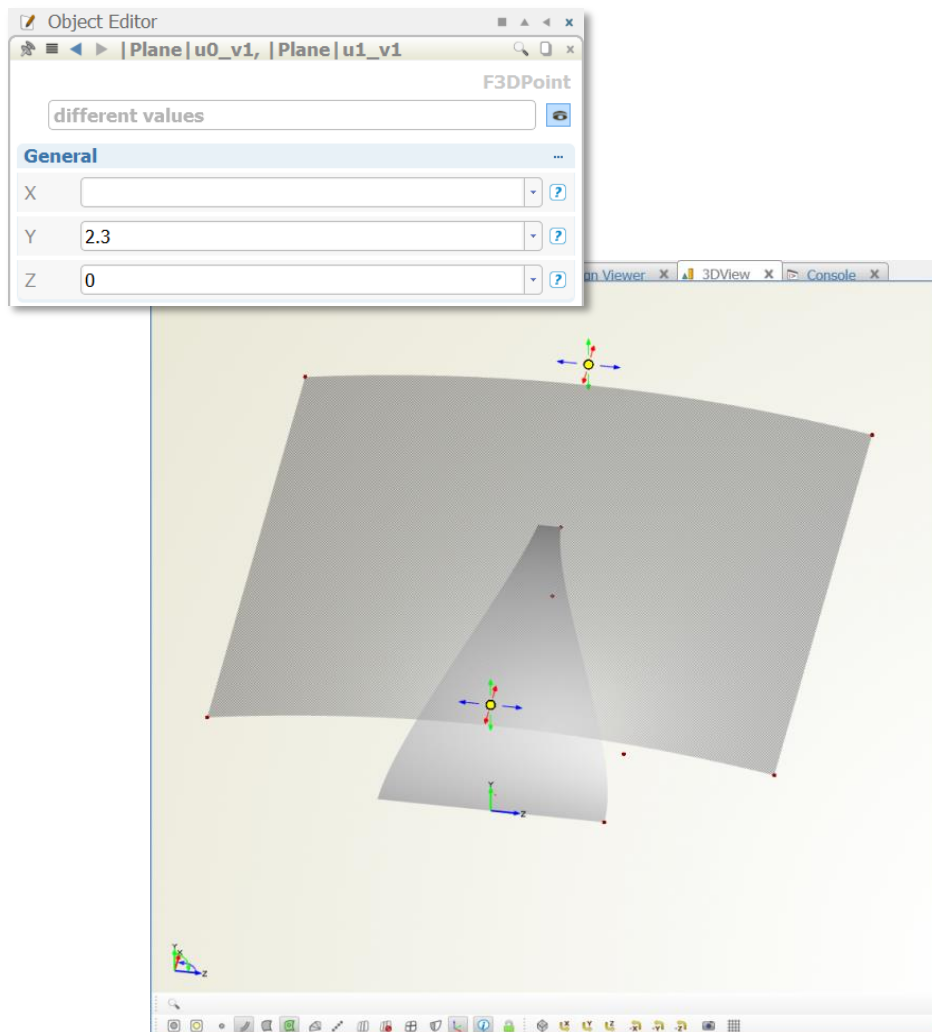
- Name:** Plane
- Plane Settings:**
  - Plane: Y - (Z,X)
  - Elevation: 1.4
- Control Points:**
  - CP in U-Direction: 2
  - CP in V-Direction: 3
  - Create Control Points: ☒
- Lower Corner Position:**
  - Start First Axis: -5
  - Start Second Axis: 0
- Upper Corner Position:**
  - End First Axis: 5
  - End Second Axis: 10
- Buttons:** ☒ Preview,  Execute,  Cancel

## 10

**Define the Body Surface**

We will change the coordinates of the surface points in order to change the shape from a planar surface to a curved surface.

- Select the two points "|Plane|u0\_v1" and "|Plane|u1\_v1".
- Set *Y* to "2.3".



# 11

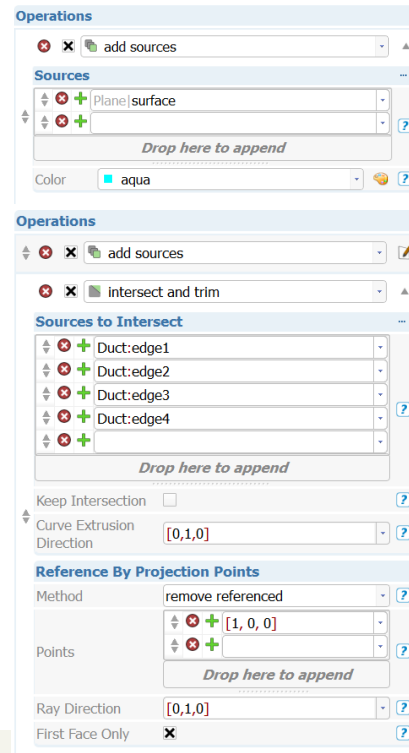
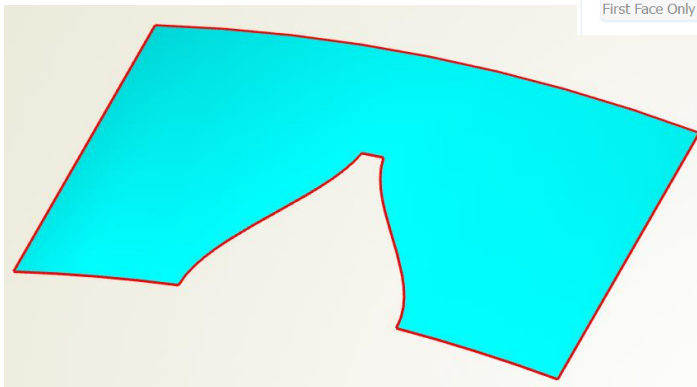
## Body Surface Brep

We will create a BRep of the curved surface and trim it with the duct.

- ▶ Select "Plane|Surface".
- ▶ Create a BRep via *menu > CAD > breps > BRep*.
- ▶ Change the name to "PlaneTrim".
- ▶ Change the color of the operation *add sources* to "aqua".

Now we want to create an intersection and remove it:

- ▶ Add a new operation *Intersect and Trim*.
- ▶ Add all four surface edges of "Duct" (select them either in the 3dViewer or in the object tree).
- ▶ Set *Curve Extrusion Direction* to "[0,1,0]".
- ▶ Change *Method* to "remove referenced".
- ▶ Set *Points* to "[1,0,0]".
- ▶ Set *Ray Direction* to "[0,1,0]".
- ▶ Activate the operation.



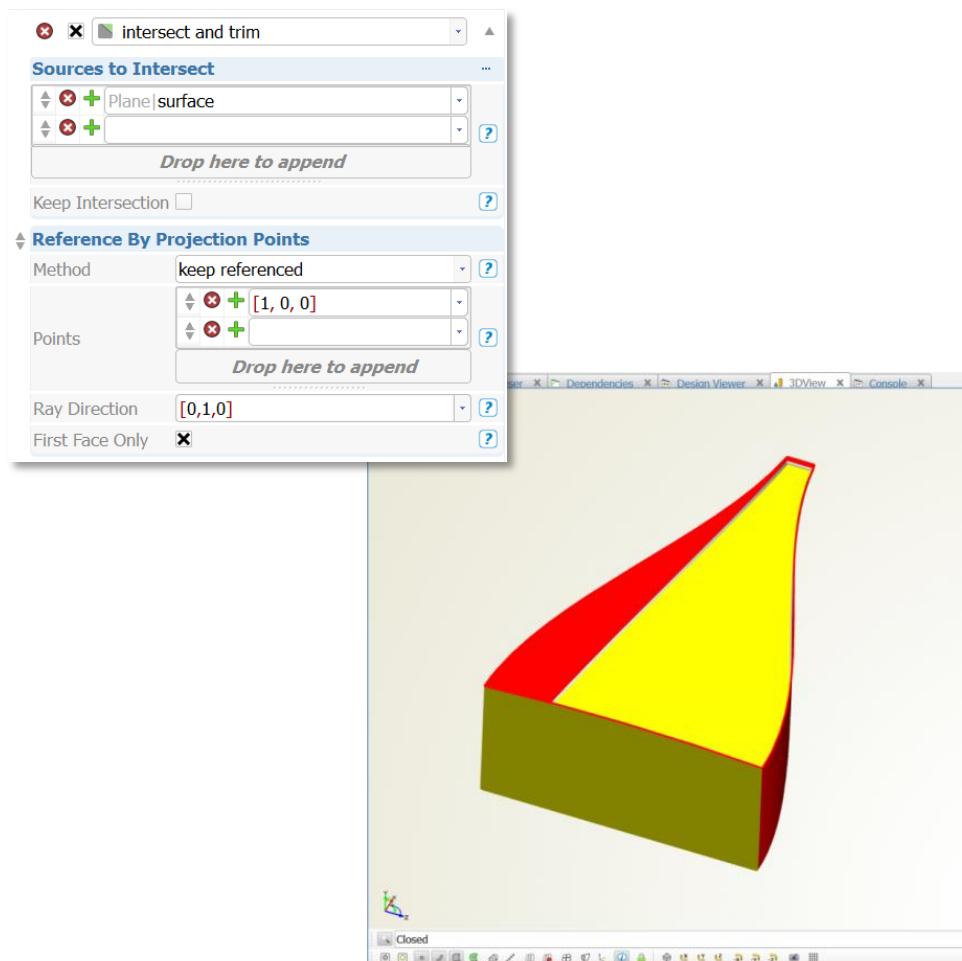
✓ *Intersect and Trim*: The group *sources to intersect* can hold breps, surfaces and curves. Curves will be internally extruded in a defined direction to find the intersection (*curve extrusion direction*).

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## BRep Operation Intersect and Trim

Now we will create an intersection between the closed duct and the body surface.

- ▶ Select "ClosedDuct".
- ▶ Select *New Operation* and choose *Intersect and Trim*.
- ▶ Choose "Plane|surface" in the drop down menu.
- ▶ Set *Points* to "[1,0,0]".
- ▶ Set *Ray Direction* to "[0,1,0]".
- ▶ Activate the operation.



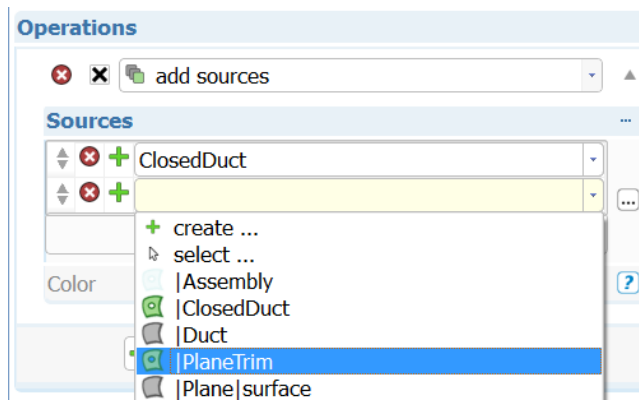
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### Assembling the BReps

Now we can assemble the two BReps “ClosedDuct” and “PlaneTrim” in order to apply more operations in the next two steps.

- ▶ Create a BRep. Via *menu > CAD > breps > BRep*.
- ▶ Change the name to “Assembly”.
- ▶ Choose “ClosedDuct” and “PlaneTrim” as sources.

You can use the drop down menu to add them.



14

## BRep Operation Remove Face

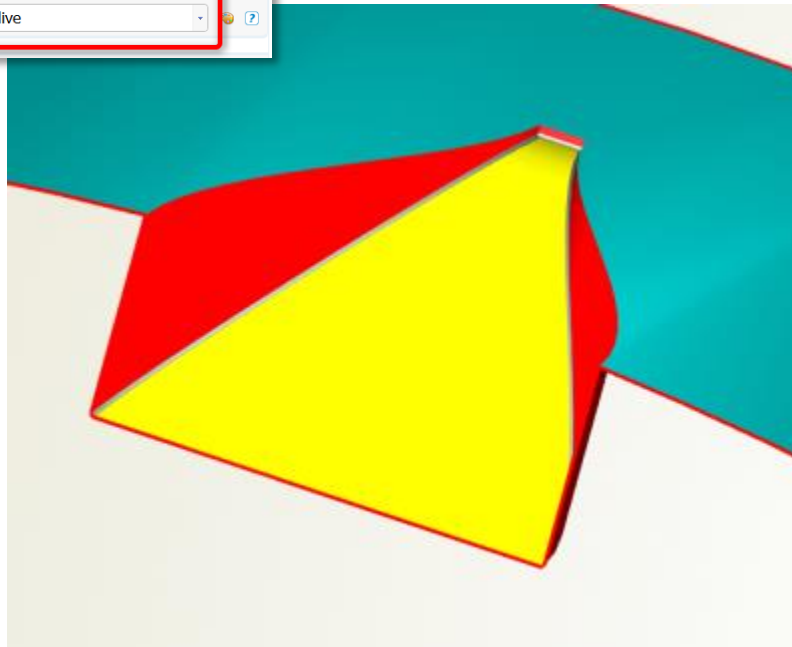
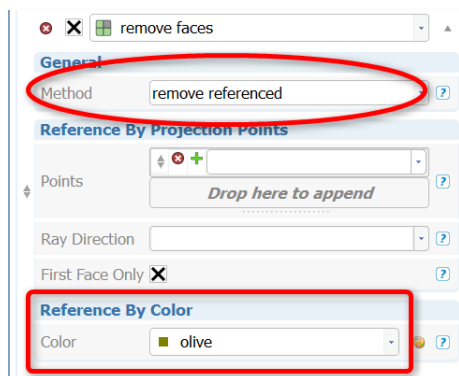
We will remove a face by its color.

- ▶ Add a new operation *remove faces*.
- ▶ Choose *referenced by color* for and set “olive” as color.

Now, the olive face will be removed from the BRep:

- ▶ Change *Method* to “remove referenced”.
- ▶ Activate the operation.

The olive face is removed now.



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### BRep Operation Edge Fillet

We will create a fillet between the curved body surface and the duct.

- ▶ Add a new operation *edge fillet*.
- ▶ Set *Color 1* to "aqua".
- ▶ Set *Color 2* to "red".
- ▶ Set *Radius / Distance* to "0.05".
- ▶ Activate the operation.

