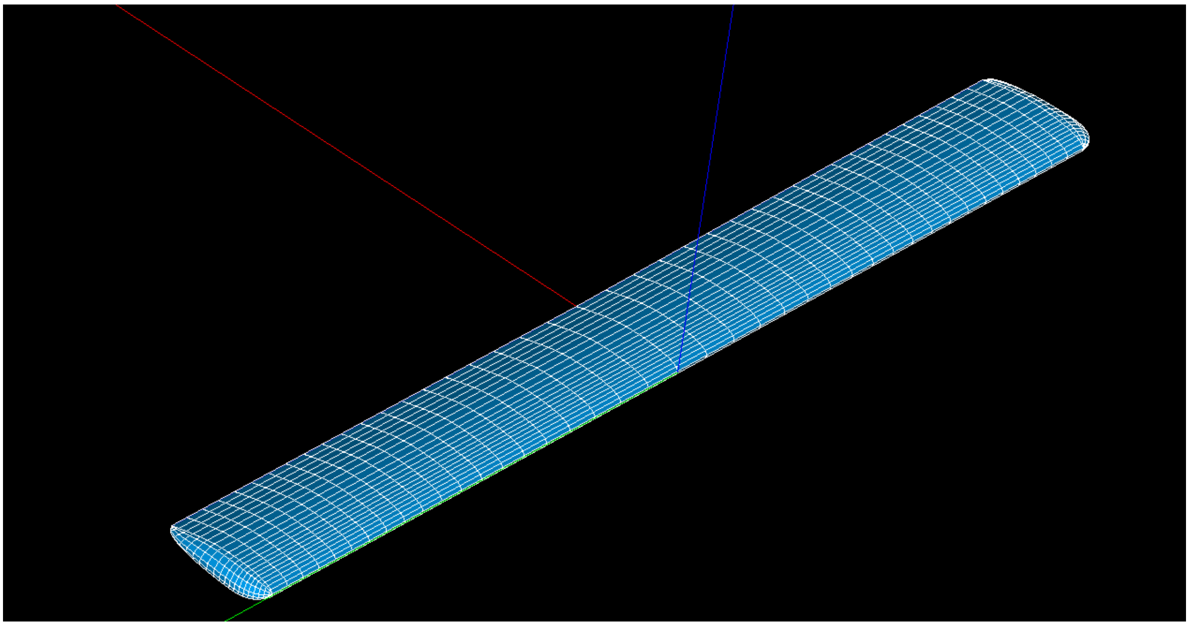
Wing meshing in SALOME



Report Number: *TUT1101R0*

Date of Issue: *20 July 2011*

Original Issue: *20 July 2011*

Author: *Etienne Vandame*

Signature: ....

Table of Contents

1 Log of revisions....1
2 Shortcuts & definitions....1
3 References....2
4 Introduction....2
5 Remarks about Salome....2
5.1 Installation....2
5.1.1 libgfortran.so.1 missing....2
5.1.2 Create a launcher on the Desktop....3
5.1.3 Set the Internet browser....3
5.2 SIGSEGV 'segment violation'....3
5.3 Working with Salome....4
5.3.1 Generalities....4
5.3.2 Show/hide....4
6 Prepare the study....4
6.1 Create a new Study....4
6.2 Import the file....4
7 Prepare the geometry for meshing....5
7.1 Create faces....5
7.2 Create the boundaries....7
8 Basic meshing....8
8.1 Wing_up....8
8.2 tip_up....13
8.3 tip_down....13
8.4 wing_down....14
9 Build the final mesh....14
9.1 Merge the meshes....14
9.2 Mirror the mesh....15
9.3 Merge & export....16
9.4 APAME....16

1 Log of revisions

Revision	Date	Author(s)	Notes
R0		E. Vandame	Initial issue

2 Shortcuts & definitions

CTRL	Control touch on the keyboard

3 References

[1] APAME 3D panel method, by Daniel Filković, Dipl. -Ing, www.3dpanelmethod.com

[2] www.salome-platform.org

[3] NASA-TN-D-8524 Aerodynamic characterisitcs of wing-body configuration with two advanced general aviation airfoil sections and simple flap systems. By Harry L. Morgan Jr and John W. Paulson Jr. August 1977

[4] Free your CFD http://code-saturne.blogspot.com/

[5] Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities http://geuz.org/gmsh/

4 Introduction

This tutorial explains how to do a surface meshing of an isolated aircraft wing using the SALOME meshing software [ref 2]. The mesh is made of both quadrangle and triangle patches, and is primarily intended for the APAME 3D panel method [ref 1]. The wing from NASA-TN-D-8542 [ref 3], using the GAW-1 airfoil was used. The geometry was previously prepared with a CAD software and exported to the neutral STEP format.

The tutorial explains how to import the geometry, prepare it, mesh individuals faces, modify and merge them, and export the whole wing mesh to a suitable format.

5 Remarks about Salome

5.1 *Installation*

On the blog *free your CFD* [ref 4] you can find good informations on how to install SALOME on your computer. I will just add small comments.

5.1.1 libgfortran.so.1 missing

Depending on your linux installation (even between several Ubuntu distribution), the path for the libraries can change, and may not be under **/usr/lib/.** So before trying to create the symbolic link as explained in [ref 4], do a search for your correct path of the libgfortran.so.3 library.

On Ubuntu with Gnome, it means on the main menubar:

PLACES → Search for files

Then, on the **file System** folder, search for **libfortran.so**, and adjust the path accordingly.

5.1.2 *Create a launcher on the Desktop*

On the Desktop, right-click and select **create launcher**.

Type → Application

Name → Salome_6.3.0

Command → Browse, and select the file **runAppli**, which is in the folder *salome_appli_6.3.0 under your personnal **home** folder.

You can add an icon by clicking on the default icon on the left. On path to the logo for the 6.3.0 version is:

.../salome_6.3.0/KERNEL_6.3.0/share/doc/salome/gui/KERNEL/icon_about.png

If for some reason, the icon is not working (as in my case), you will have to copy/paste it to the **Salome_appli_6.3.0** which is on you home folder.

5.1.3 *Set the Internet browser*

In order to access to the help, you have to set correctly the web browser. If you're using Firefox, open a terminal and type:

which firefox

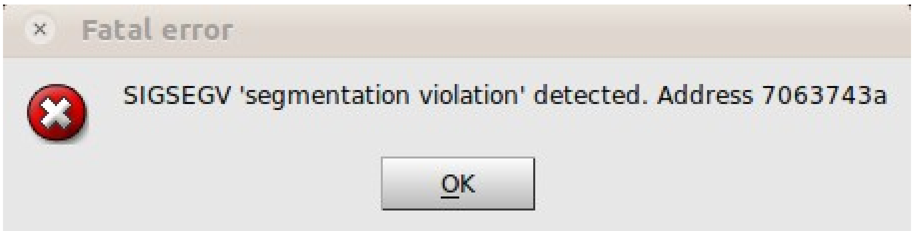
You'll get the path to the browser. Then go back to Salome, go to

File → Preferences

On the SALOME menu, go to the General tab, and under **external browser** line, type the path (for ubuntu 11.04, it is somethings like **/usr/bin/firefox**).

5.2 *SIGSEGV 'segment violation'*

There is a bug appearing from time to time when you are under the mesh module. The following windows is warning you:



I didn't find how to solve it, but if it appear to you, I recommend you to click OK, save your study, close Salome and reopen it. By this way you will not loose any data. This bug seems to appears on ALL versions of SALOME. Of course the address number will change each time...

5.3 *Working with Salome*

5.3.1 *Generalities*

Salome is not a *parametric* CAD. It means that all the entities, when created cannot be modified. In case something was wrong, you will have to delete the feature and create it again. It makes Salome not suitable for complex CAD geometry creation.

There is NO undo command ….

The pan and rotate commands are very cumbersome. In order to do one of this *operation*, you have first to click to the icon, and the desired movement will be available up to next mouse button release.

5.3.2 *Show/hide*

From version 6 of Salome, there is a shortcut in the object browser in order to hide or show elements (either meshes or geometry entities). This is marked by an eye at the left of each element. By clicking on the eye, you can switch on/off the visibility of the entity.

Right-clicking on the name opens a context menu with some other options (very useful one is the **show only**).

6 Prepare the study

6.1 *Create a new Study*

Open Salome

Create a new study

menu File → new

or

click on the icon 

6.2 *Import the file*

Switch to the geometry module

Press the arrow on the left of this icon 

and select **geometry**

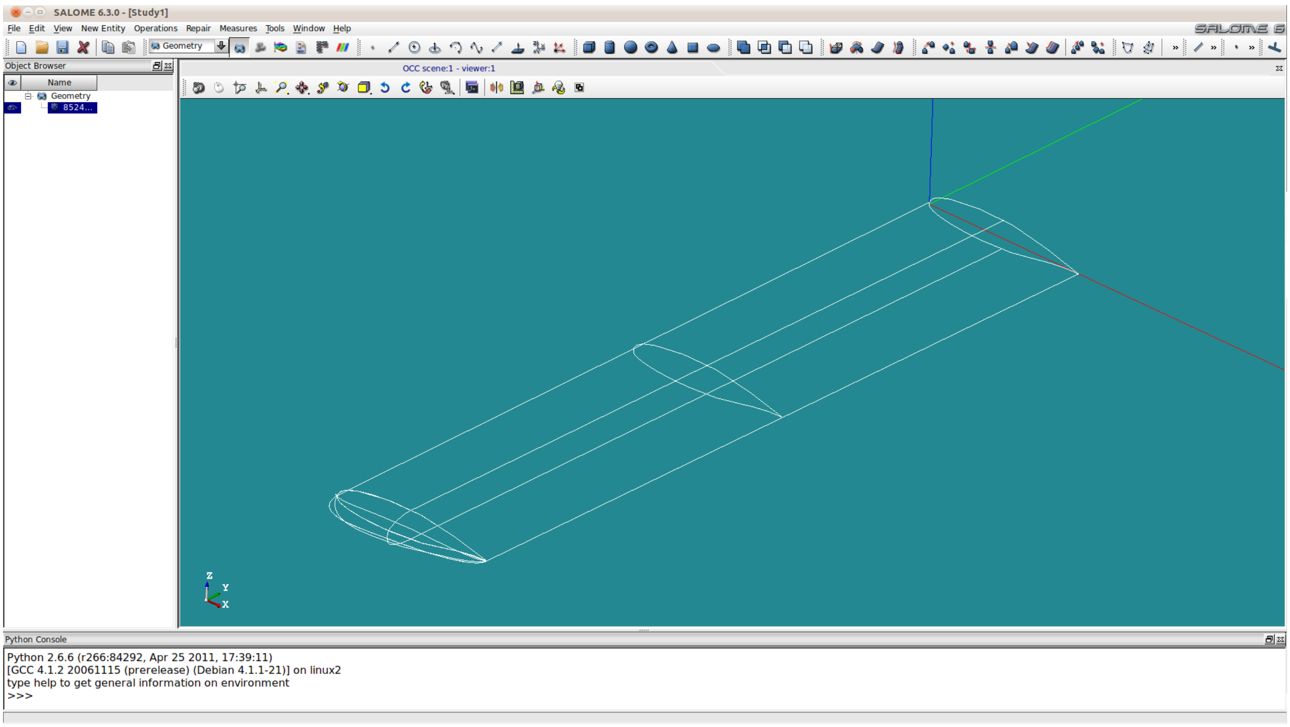
Import the file

menu File → Import

In the line “file of type” choose STEP

Select your file and click OK

The object appear in the **object browse** on the left of your screen



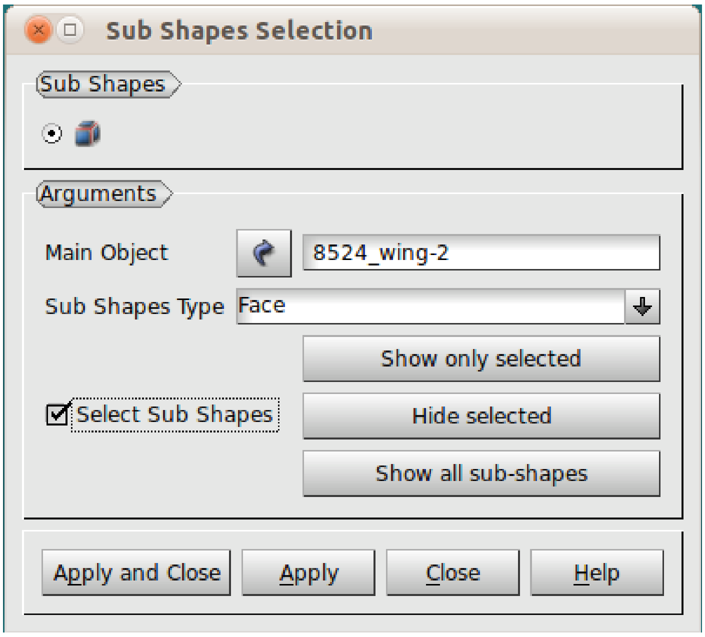
7 Prepare the geometry for meshing

In Salome, the geometry is imported as a “dead” object, so it is necessary to generate the basic geometries that will be meshed. In a first step it is necessary to generate faces, and then for each face, we will generate 2 edges for the faces boundaries meshing.

7.1 *Create faces*

Menu New entity → explode

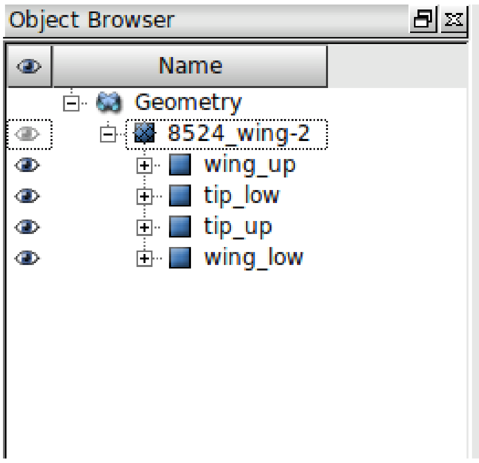
fill the fields as shown on the the following windows:



The **main object** is the object that will serve as the base for the shapes generation
In this step we need to generate **faces**.
Check **select sub shapes** box in order to be able to pick-up them with the mouse

Click on the geometry in the main window:
→ On the upper side of the wing, and then click **apply**.
→ On the lower side of the wing, and then click **apply**.
→ On the upper side of the wing-tip, and then click **apply**.
→ On the lower side of the wing, and then click **apply and close**.

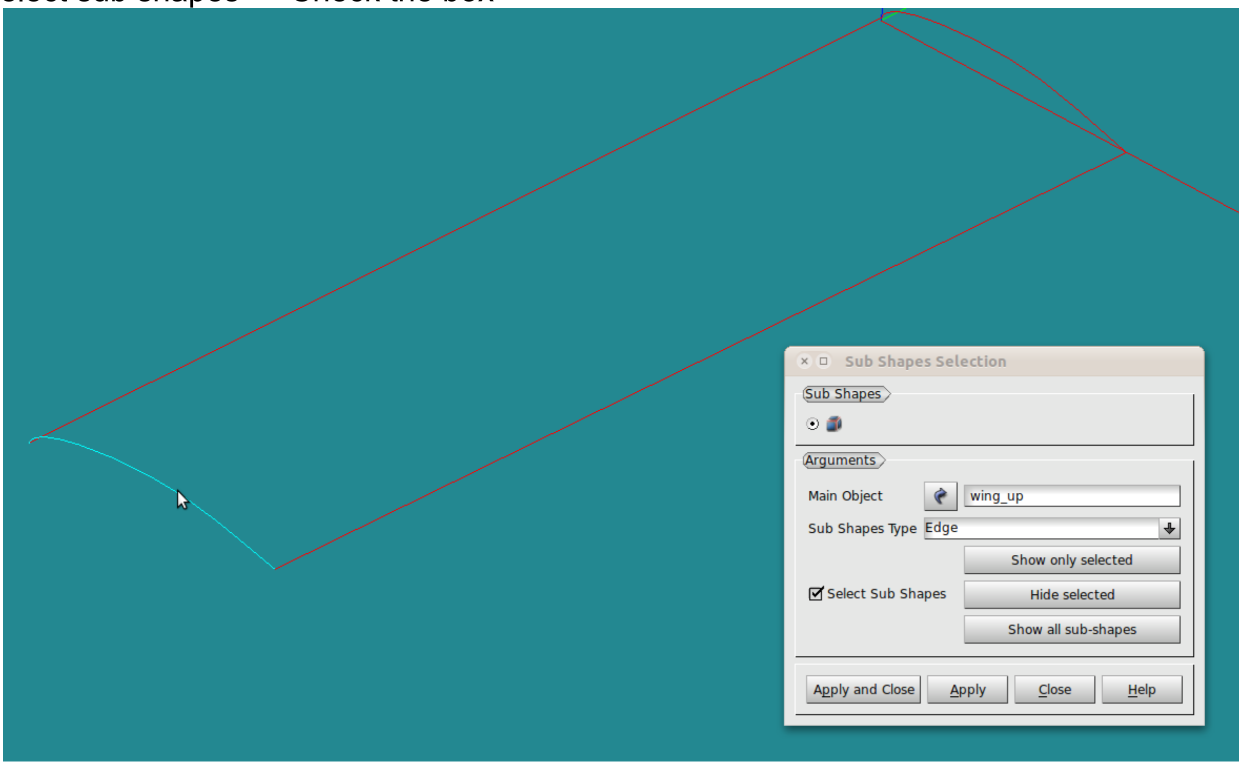
Create by this way 4 faces (upper side of wing, lower side of wing, upper wing tip, lower wing tip). You can rename the faces by double-clicking on the names of the faces on the Object browser.



7.2 *Create the boundaries*

Right-click on **wing_up**, then **show only**.

new entity → explode,
main object → wing_up
sub shape type → Edge
Select sub shapes → Check the box

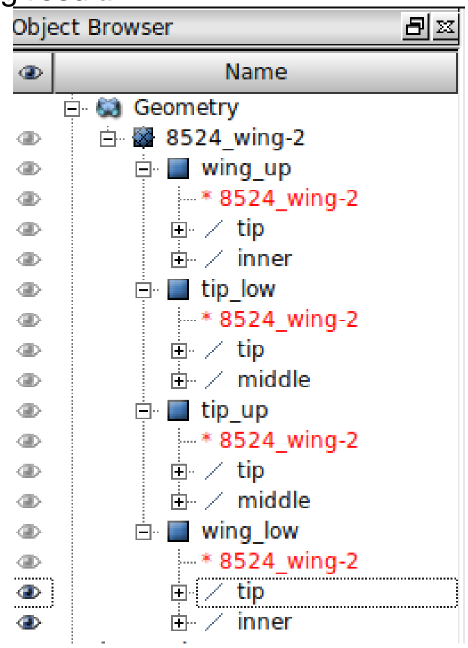


Select the edge of the wing section close to the wing tip, then apply.
Select the edge of the wing section at the center line, then apply and close.

Rename the edges **tip** and **inner**.

Do the same for the wing lower side (rename the edges also tip and inner), for the wing tips (rename the edges tip and middle).

You should have the following result:



Don't forget to save from time to time ...

8 Basic meshing

8.1 *Wing_up*

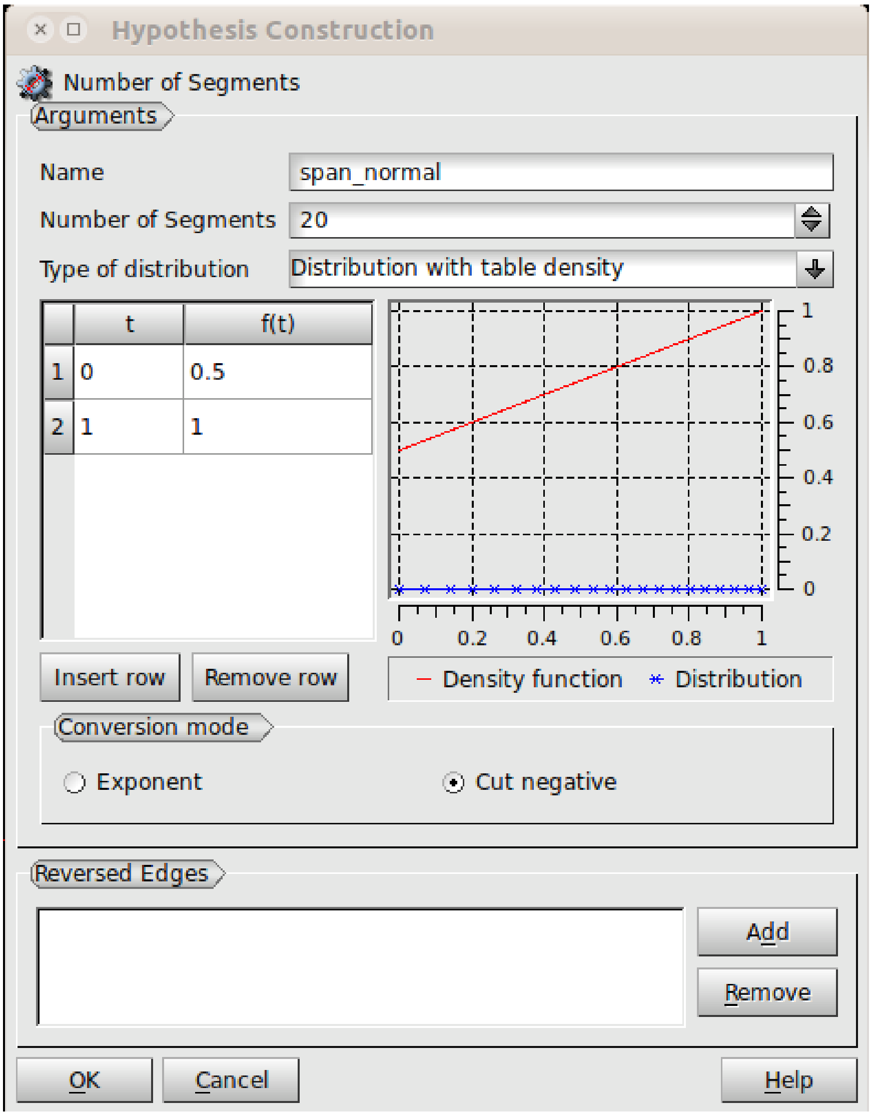
We will begin with the wing_up, so switch on its visibility on the the left panel.

Mesh → create mesh

geometry → select **wing_up**

tab **2D → assign a set of hypothseses → 2D automatic quadrangulation**

A sub-window **Hypothesis construction** opens**.** Fill it as shown on the next picture

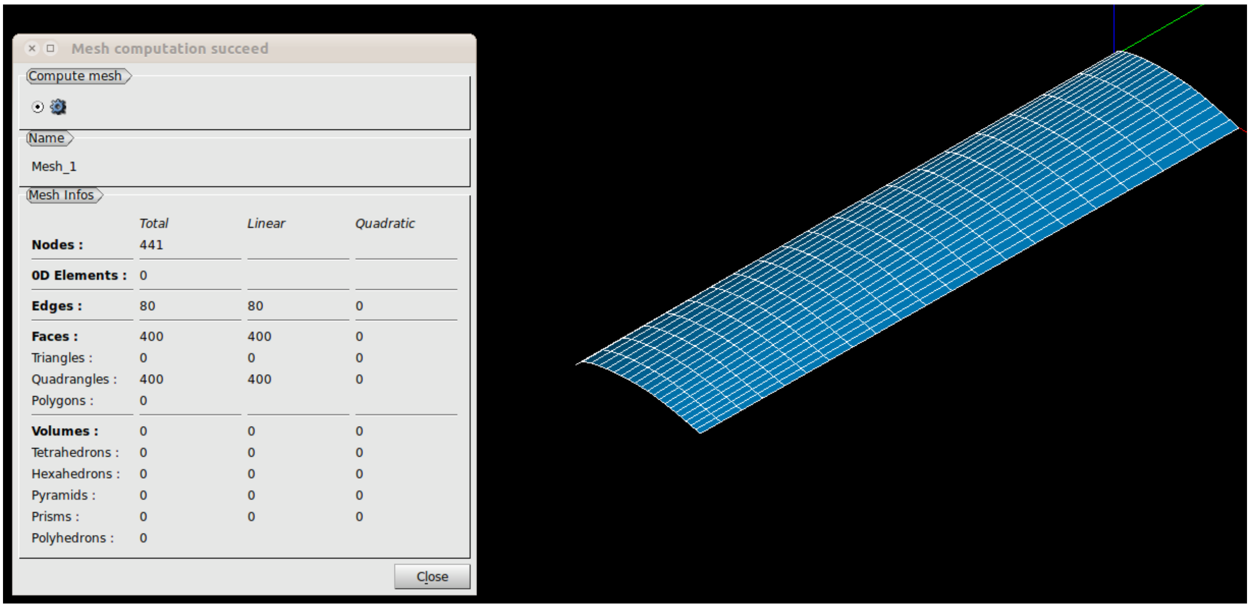


→ OK
→ **apply & close**

A **Mesh_1** is created. Rename it to **wing_up**,

You can notice a yellow triangle, signalizing that the mesh isn't computed. Right-click on it on the object browser, and **compute**

You will have the following resulting mesh:



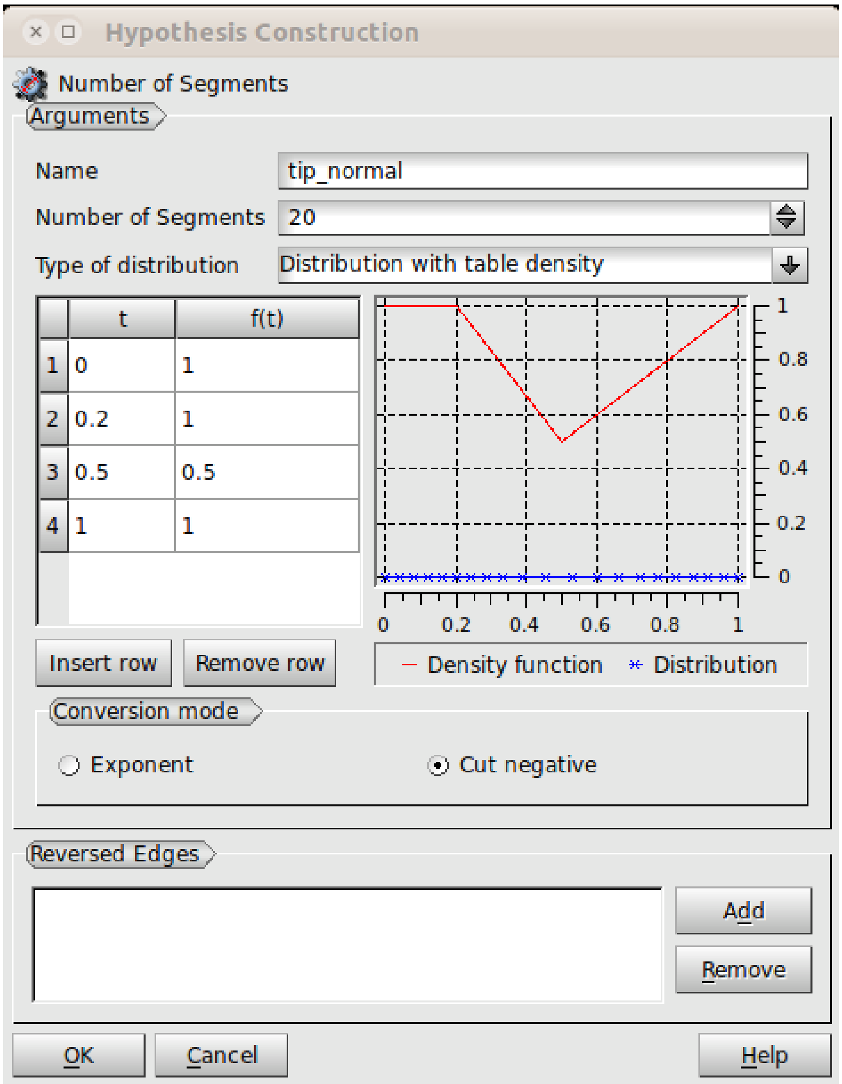
The four edges of the mesh boundary are meshed with the same hypothesis, **span_normal**, which is intended to satisfy our requirements along the span. It is now it is necessary to define precisely the parameters for the boundaries representing the wing profile.

Right-click on the name of the mesh, and, **create sub-mesh**.

Mesh → wing_up
Geometry, select the **tip** edge on the wing_up geometry.

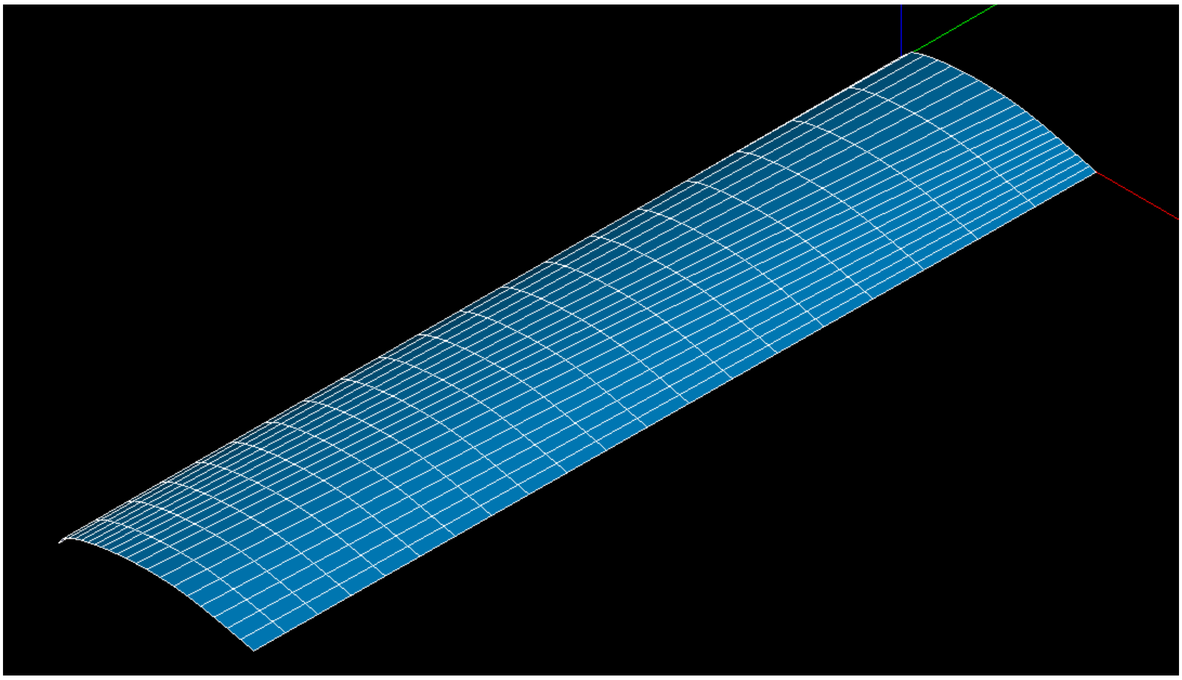
On the tab 1D, algorithm, select **wire discretisation**, then on the row Hypothesis, click on the icon at the right, and choose **Nb. Segments**.
A windows **hypothesis construction** opens.

In the arguments,
change the name to tip_normal
number of segments, set to 20
type of distribution, choose distribution with table density. Fill it as shown on the next picture



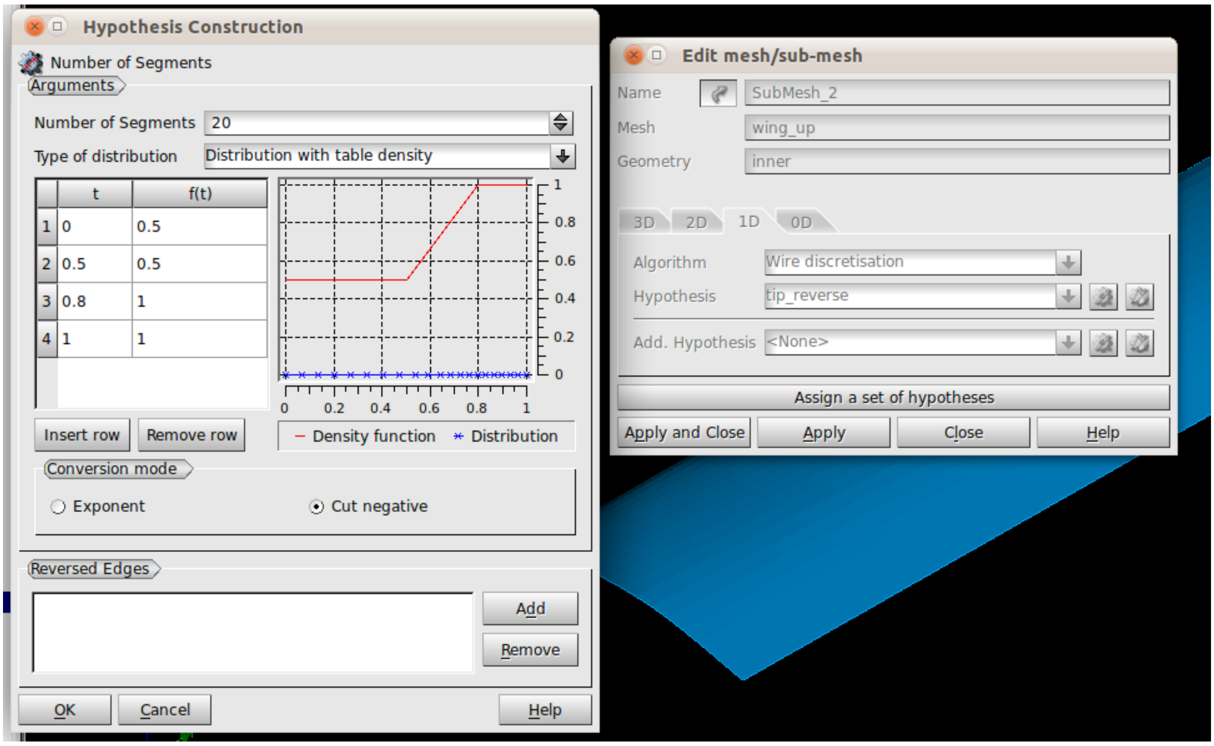
Press OK to close the windows, then apply

Then create another sub-mesh along the second half profile, using the **inner** geometry. Proceed the same way. Then, right-click on the **wing-up** mesh and **compute**.



You can see that the mesh is not good, it is given by the fact that the **inner** edge has another orientation than the tip edge. It is easy to correct it.

Open the **subMesh_2** under the **wing_up** mesh, and create another hypothesis with an inverted distribution:



→ OK
→ apply & close
→ Compute the mesh

The mesh is now looking as expected, we can begin to mesh another element. It will be easier because the basic hypothesis that we had to define will be used, and hence it will not be necessary to create them again. This is also insuring the continuity of the nodes and patches between two meshes.

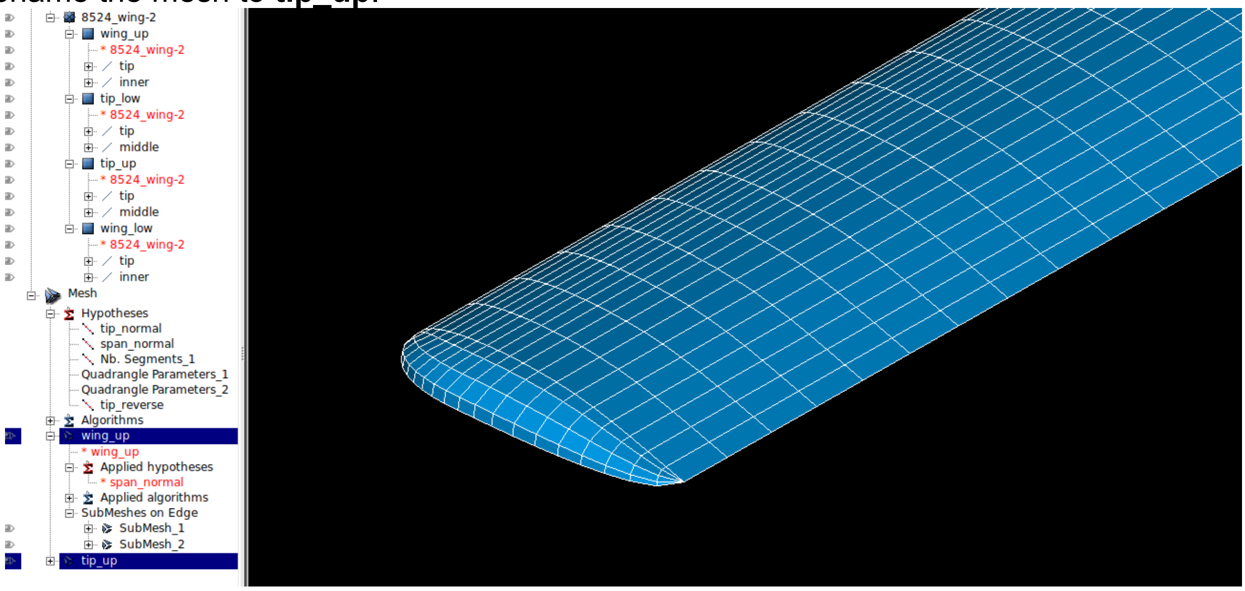
8.2 *tip_up*

We will now mesh the upper part of the wing tip:

Mesh → create mesh
Geometry → Select the **tip_up** geometry
On the 2D tan, select algorithm → Quadrangle (mapping)
Go to the tab 1D, choose **wire discretization** algorithm and select the hypothesis **tip_reverse**

Compute

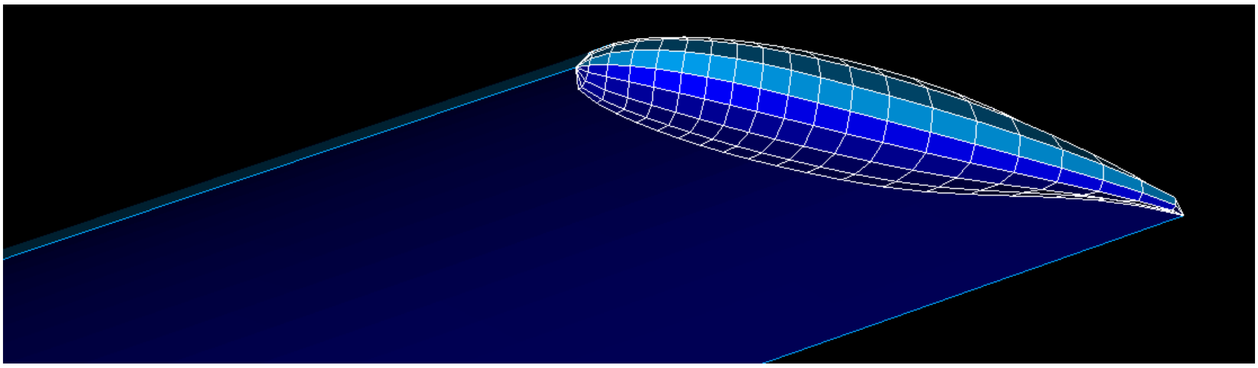
rename the mesh to **tip_up**.



The two meshes are fitting together. In order to easily see the discretization on the graphic display, select the meshes on the object browser (maintain CTRL pressed in order to select several meshes).

8.3 *tip_down*

Proceed exactly the same way, except that you will have to select the **tip_normal** as hypothesis. It is often a problem to guess the edge orientation, but you just have to try...



You can see that the **tip_down** mesh has another color. The **dark blue** means that you are seeing the inner part of the patches. We need to change the orientation of this mesh in order to have it correct:

Modification → orientation
check the **apply to all** box
select the **tip_down** mesh by clicking on one of its patches with the mouse on the graphic windows (you may have to hide the others meshes for ease), click **apply and close**.

8.4 *wing_down*

mesh → create mesh
select the wing_down geometry
on the 2D tab, choose the **quadrangle** algorithm
on the 1D tab, choose **wire discretization** algorithm, and **span_normal** hypothesis.

Now create the 2 sub-meshes for the wing profiles.
You will have to apply **tip_reverse** for the **tip geometry**, and **tip_normal** for the **inner** geometry.

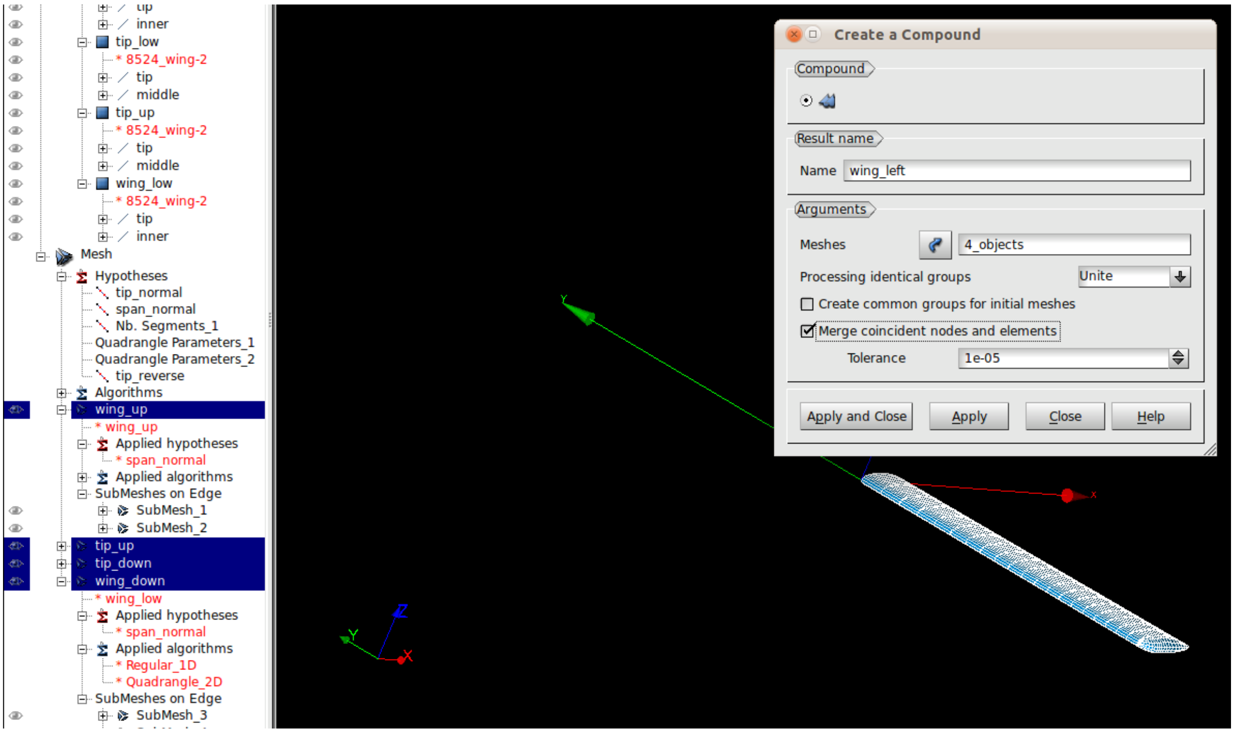
9 Build the final mesh

9.1 *Merge the meshes*

mesh ->build compound

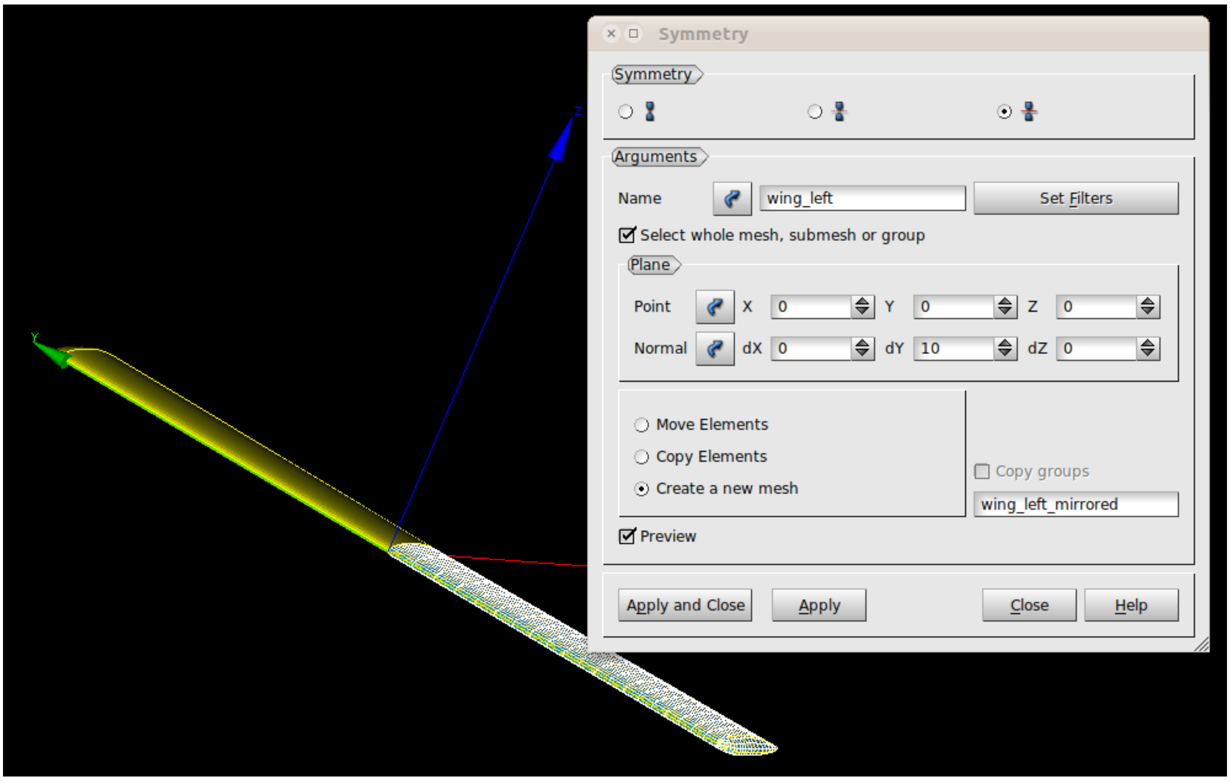
Select the four meshes (maintain the ctrl touch pressed).
Check the **merge coincident nodes and elements**.

Apply and close



9.2 *Mirror the mesh*

Modification → transformation → symmetry



Apply & close

You must then compute the newly created mesh:
Right-click on its name on the object browser, and compute

9.3 *Merge & export*

Select the compound mesh and its mirror, and create a final compound mesh.

select it and right-click on its name, you can choose to export it to UNV / MED / STL.

9.4 *APAME*

APAME is able to import directly NASTRAN meshes. As to date, you need another step (via the GMSH software, [ref 5]) to do the conversion:

→ Export to UNV format in Salome
→ Open GMSH, import the mesh.
→ save it as NASTRAN BULK DATA FILE
→ You can now open it in APAME