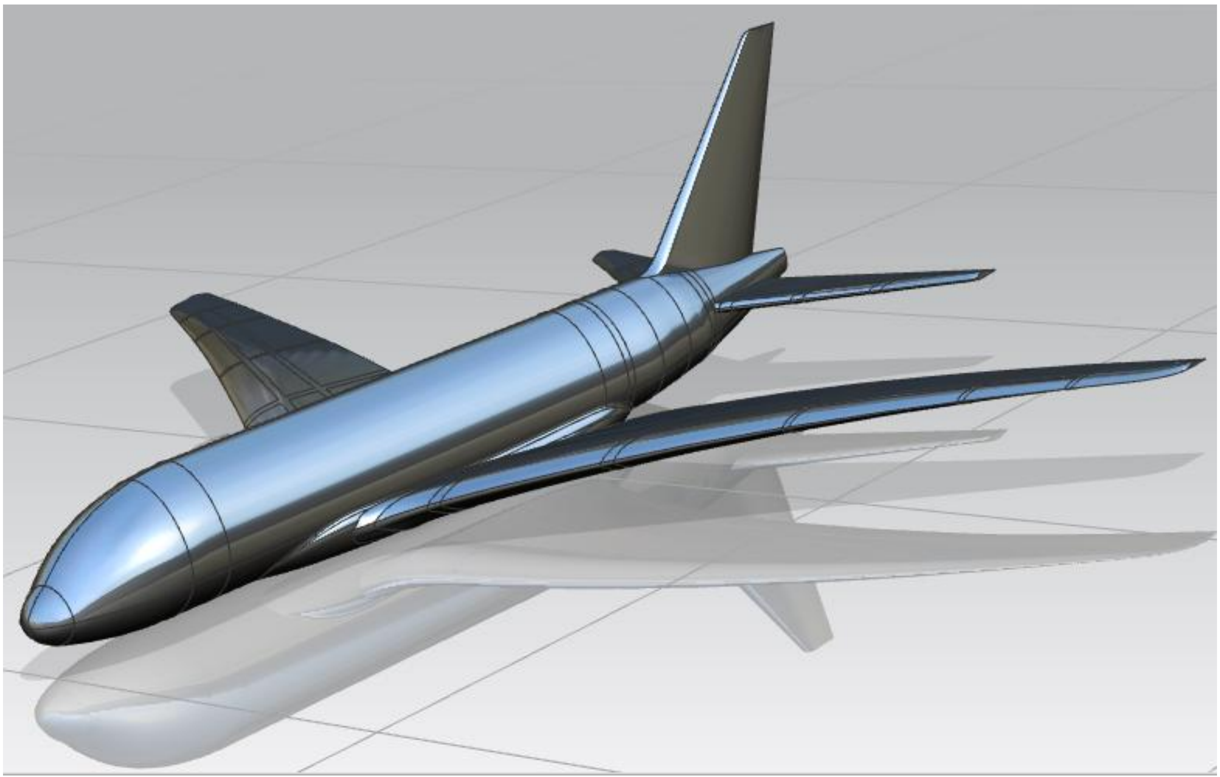
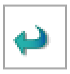
Siemens NX tutorial
Aircraft assembly and structure simulation

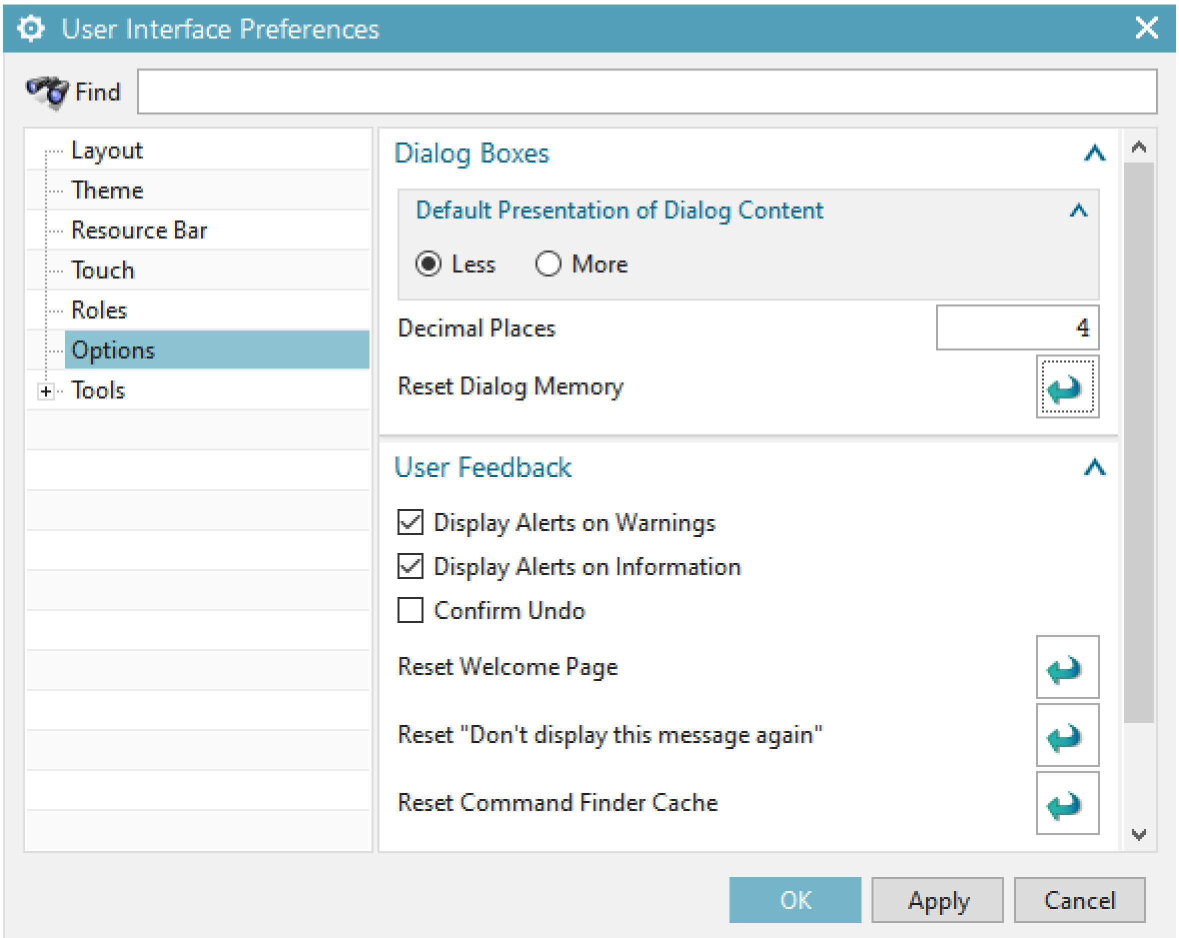
Aircraft structure designed from a solidworks tutorial[1]



[1] *https://www.youtube.com/watch?v=dRCKmc0ZxBk*

1. Reset dialog boxes

- Click on *Menu* →*Preferences* →*User Interface…*
- In the *User interface Preferences* dialog box, click on the *Options* item of the left tree, and then on the *Reset Dialog Memory*  button.
- Click *OK* to validate.



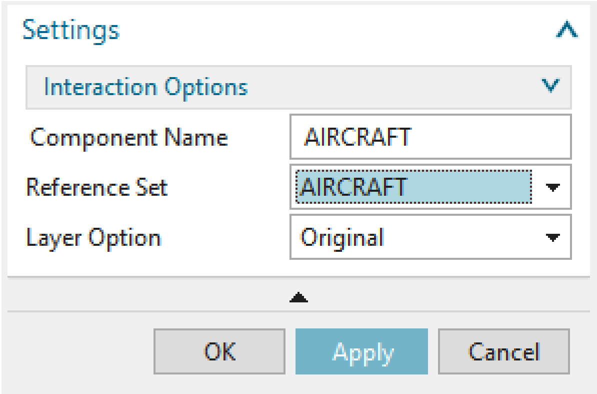
2. Structure assembly

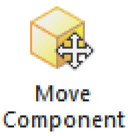
2.1. Adding the parts

- Download the zip file `aircraft.zip` from the course web page and unzip it into a folder. This file contains three part files:
    - `aircraft.prt` which contains the fuselage and tail of an airbus A380 on real scale.
    - `left wing.prt` which contains the left wing of the A380.
    - `right wing.prt` which contains the right wing of the A380.

- Create a new empty assembly file 

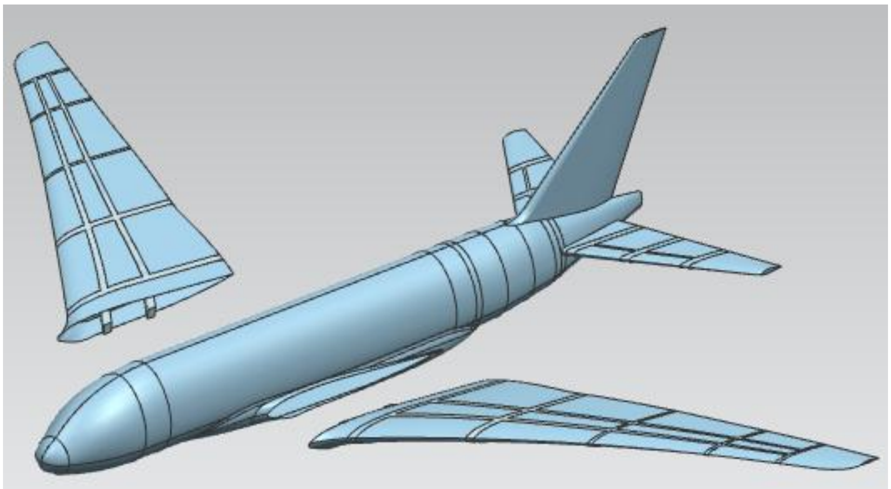
.
- Select as first part to insert the file `aircraft.prt`. Under *Settings* of the *Add Component* dialog box, select *AIRCRAFT* as *Reference Set.*



- Then, add the file `left wing.prt`, and select *LEFT WING* as *Reference Set*.
- Do the same with the file `right wing.prt`, with *RIGHT WING* as *Reference Set*
- Move 

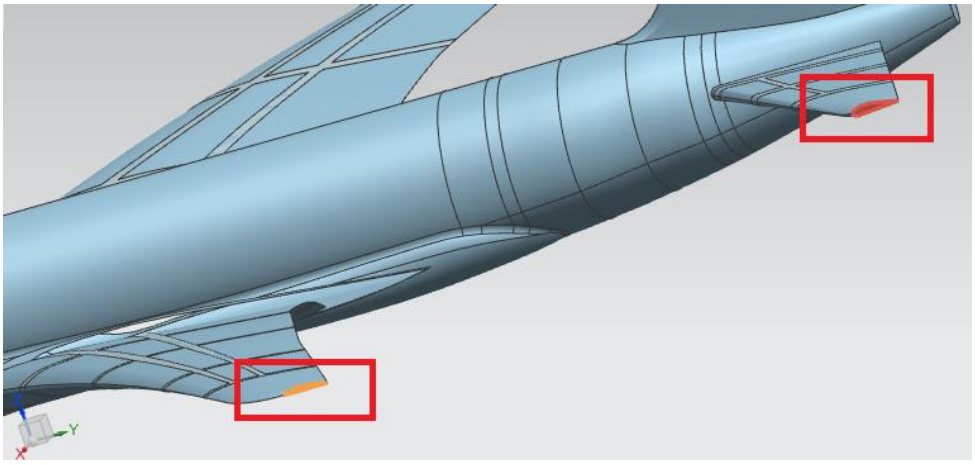
manually the wings so to position them approximately as shown below.

.

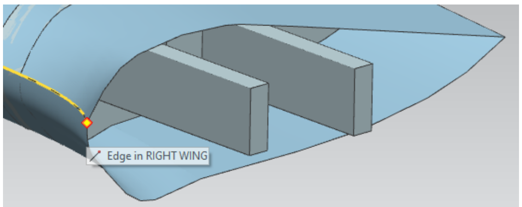


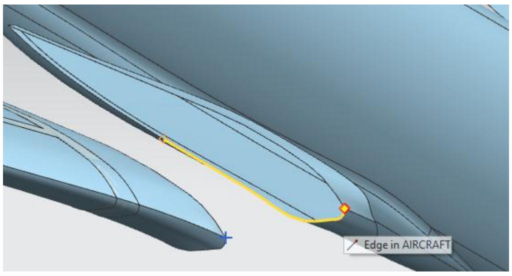
2.2. Setting the constraints

- If not already done, add a *Fix* constraint on the fuselage.
- Then, add two *Parallel* constraints between the tip of the left (resp. right) wing and the tip of the left (resp. right) tail.

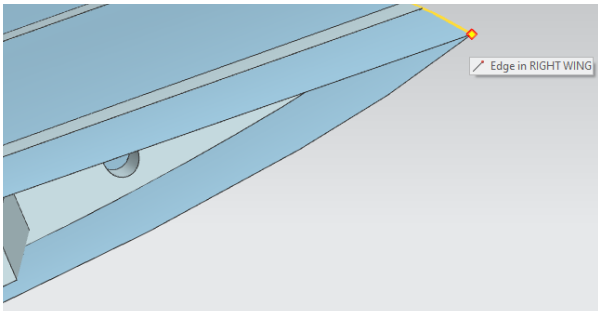


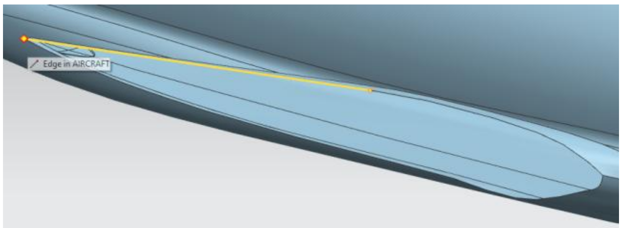
- Add a *Touch* Touch constraint between the front point of the base curve of each wing and the front point of the base curve of the corresponding hole in the fuselage (see figure). Note, in order to be able to select points, you will have to activate the *End Point* option in the toolbar.





- Redo the same operation for both wings, this time with the end points of the base curves of the wings and the corresponding holes in the fuselage.





- The assembly is done. Save it. There should be no more degree of freedom Show Degrees of Freedom existing in the assembly.

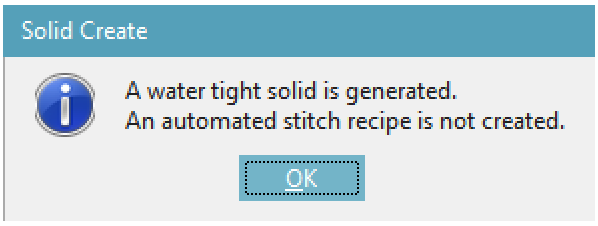
3. FEM meshes

3.1. Correcting geometric defects

- With the assembly file still open, create a new *NX Nastran FEM* file 

.
- We want to mesh the whole aircraft surface (fuselage + wing) using only one mesh. However, the wings and the fuselage do not share a common edge at their junctions. If we try to mesh it, we will end up with three different surface meshes. Moreover the fuselage contains some geometric defects (small holes) which will prevent any simulation from running.
- Use the *Stitch Edge* 

tool in order to correct these defects. Try to figure out how it is functioning and use it till you get the following message.

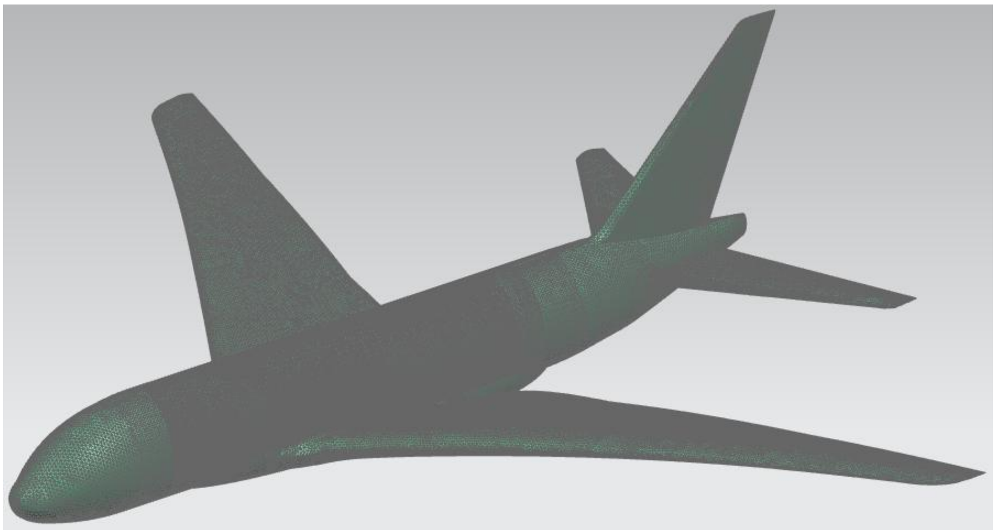


3.2. Creating the 2D (shell) mesh

- Create a 2D (shell) mesh 

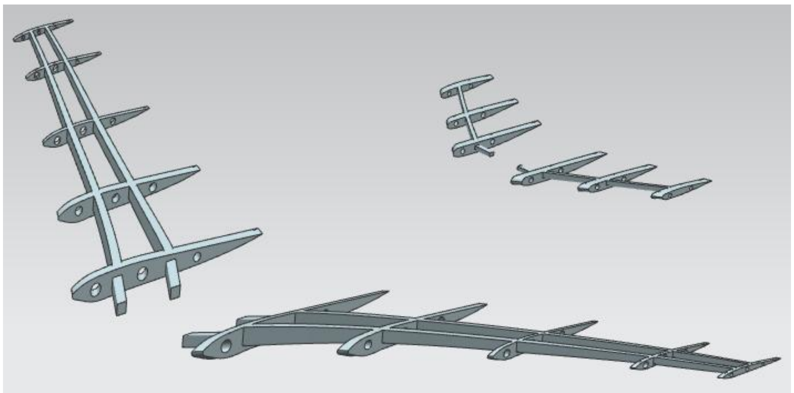
of the aircraft surface (NOT the solid frames inside the aircraft) with *CTRIA3* elements type 

and a mesh size of *300 mm*.
- Before creating the mesh, add to it a shell thickness of *10 mm*. See page 4 of the *Valve* tutorial if you do not know/remember how to do it.

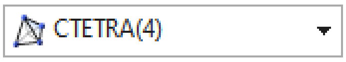


3.3. Creating the 3D meshes

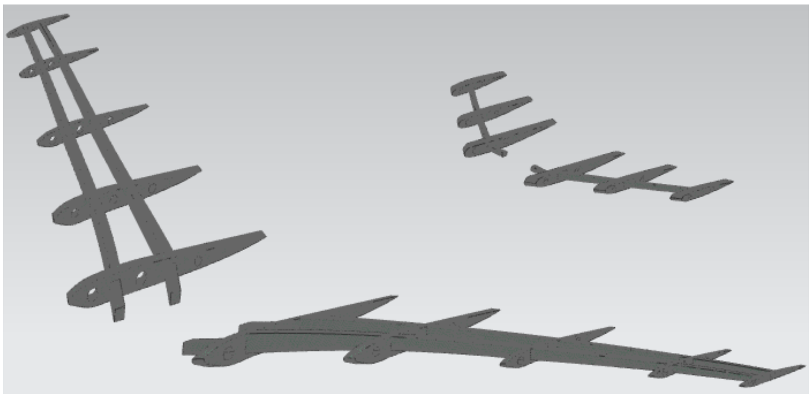
- Hide the newly created 2D-mesh as well as the aircraft surface, in order to keep only the wings and tails frames.



- Create 4 3D-meshes 

of elements type *CTETRA(4)* 

, with an element size of *200 mm*.



- Add the *Aluminum 2014* material 

to all 5 meshes.
- Save your FEM file.

4. Structure simulation

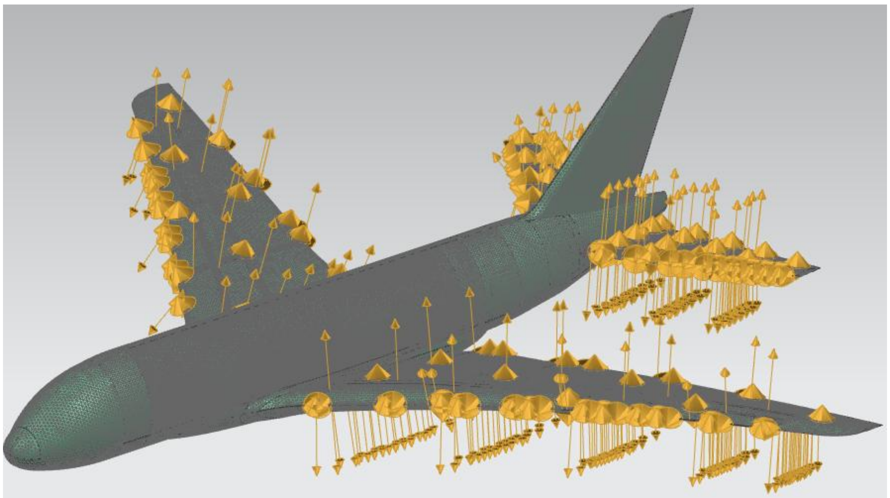
4.1. Coupling the 2D-mesh with the 3D-meshes

- With the FEM file still open, create a new *NX Nastran SIM* file 

.
- Use the *Surface-to-Surface Gluing* 

tool, under the *Simulation Object Type* 

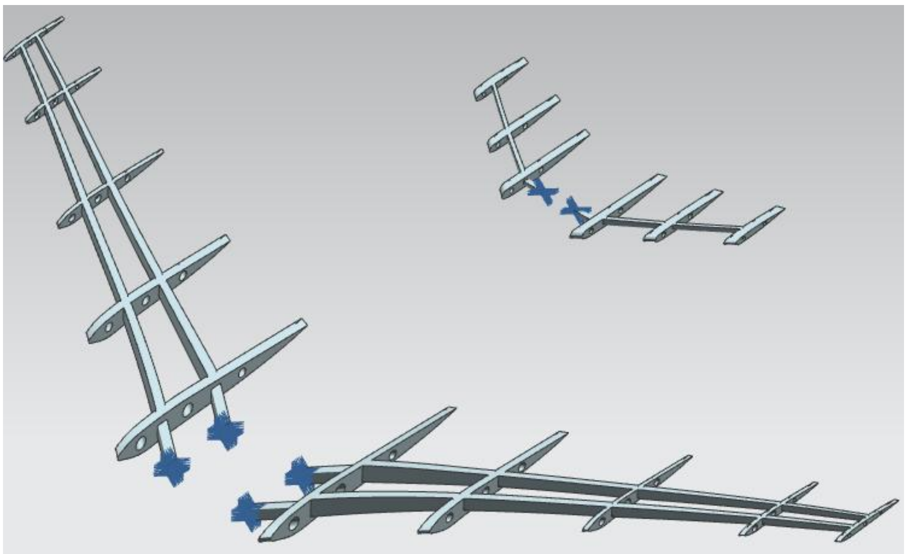
button. This will "glue" the 2D-mesh with the 3D-meshes at their contact points. We will later apply some loads on the wings and tails, the coupling with the 3D-meshes (frames) is here to guarantee that the wings and tails will not collapse under the loads.
- Try to figure out how this tool is functioning. You should obtain 4 glued regions (corresponding to the 4 3D-meshes).



4.2. Adding constraints

- Add 6 *Fixed Constraints* 

at the face extremities of the frame lying inside the fuselage.



4.3. Adding gravity and first simulation

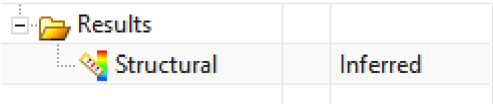
- Add a *Gravity* load of 9810 mm/s² 

along the –*Z* 

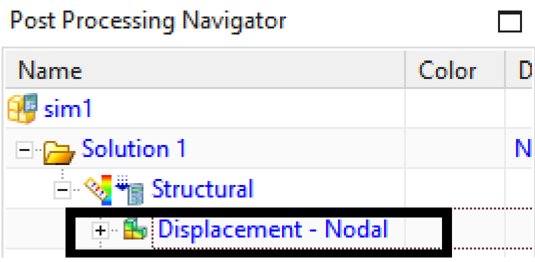
direction.

- Solve this simulation 

. This will take several minutes to complete.
WARNING: be sure to have at least 3 Gb RAM free.

- Once the simulation successfully completed, double-click on *Structural*, under *Result* at the bottom of the *Simulation Navigator* in order to visualize them


.

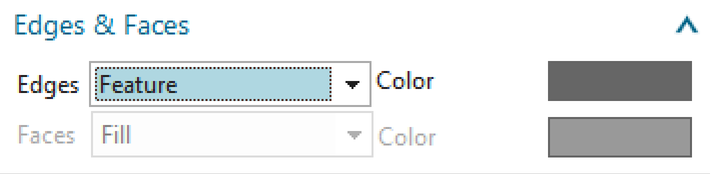
- In the *Post Processing Navigator*, double-click on *Displacement Nodal*, in order to visualize the displacements 

.

- As you can observe, the default visualization parameters are quite lame four our case. In order to improve it, use the *Edit Post View* tool 

.

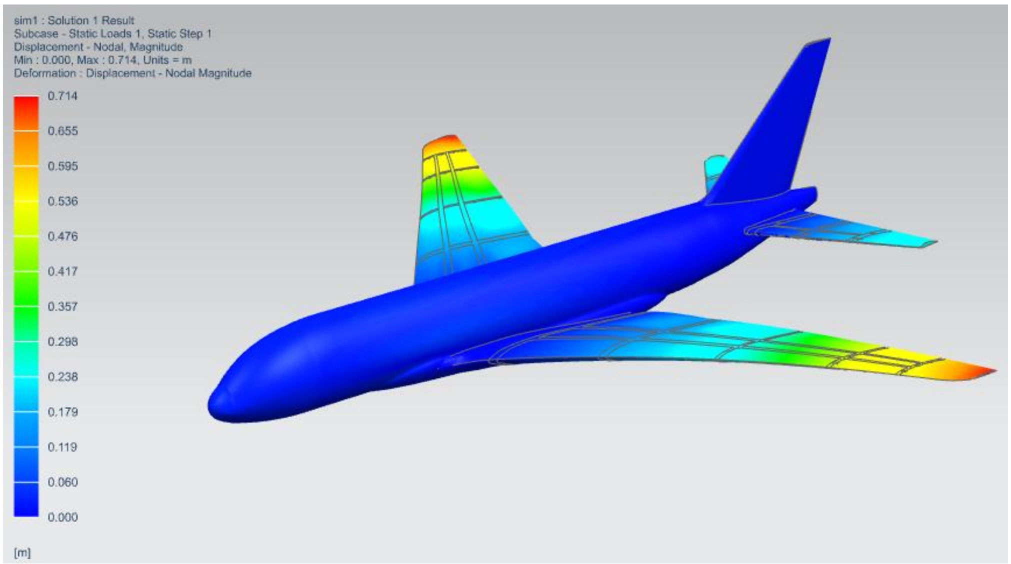
- In the *Post View* dialog box, do the following operations:
  - Under the tab *Result* set the *Units* option to meters (*m*)
  

  .
  The left colorbar will be more readable, showing deformation scale in meters instead of millimeters.
  - Under the *Display* tab, set the *Edges* option to *Feature*
  

  . This will only show the main geometric features of the airplane instead of all the triangles of the meshes.
  - Finally, under the *Deformation* tab, set the *Scale* options to *1.0* and *Absolute* 

  .
  This will show the true deformations ongoing on the plane.
  - Click *OK* to validate.

- Now the visualization of the deformations should look much better:



- You can also animate the deformation using the tool *Animate* 

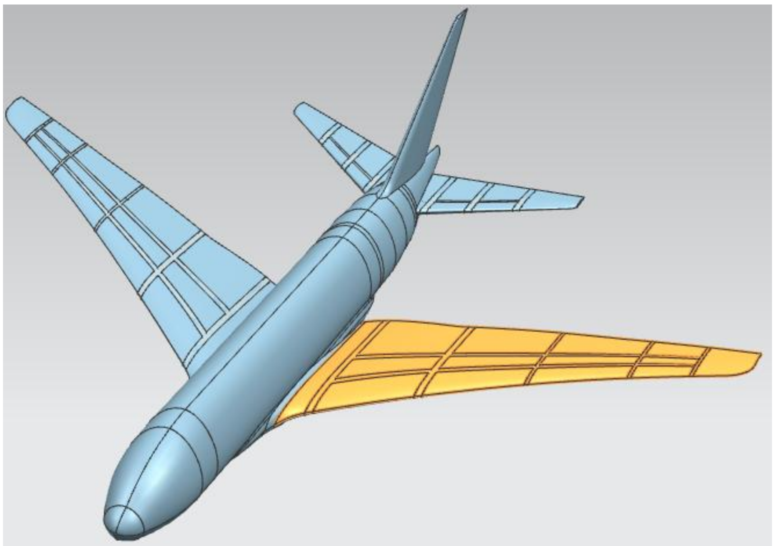
.

4.4. Adding local variable pressure field on the left wing

- **Before to continue**: be careful to follow the below instructions to the letter. These instructions involve opening many imbricate dialog boxes. Failure to set the right parameters into one, may lead to the failure of every other dialog box.

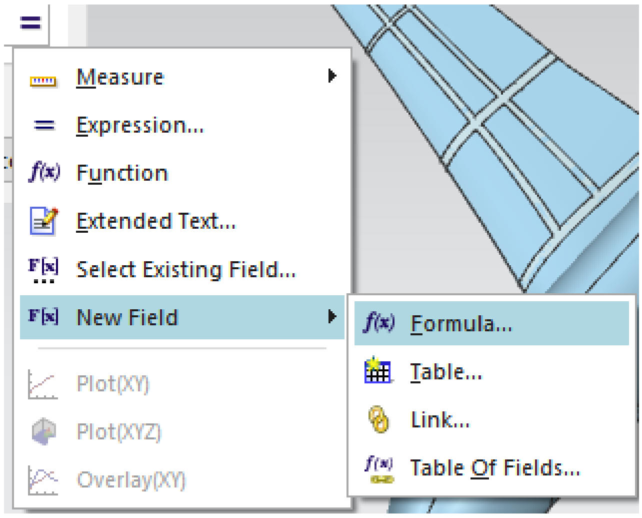
- **Tip 1**: if you selected the wrong object, you can deselect it with the *shift* key of your keyboard + left click of the mouse.
- **Tip 2**: if you think you messed-up the parameters of your current dialog box, you can reset them by clicking on the top right arrow of the dialog box ↻.

- Go back to the simulation by clicking on the *Return To Home* button [Return to Home].
- Click on the *Pressure* tool [Pressure]. In the *Pressure* tool box, select the surface of the left wing as well as its frame.

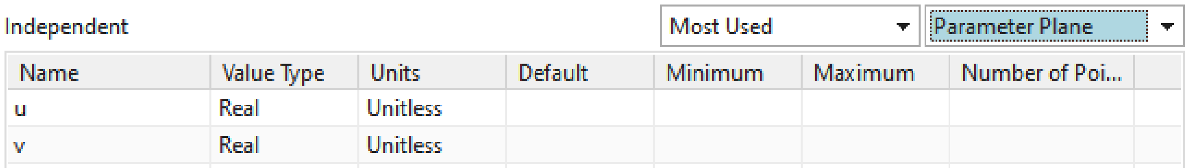


- In the *Pressure* option of the *Pressure* dialog box, click on the small "=" 

, and select *New Field*→*Formula*.

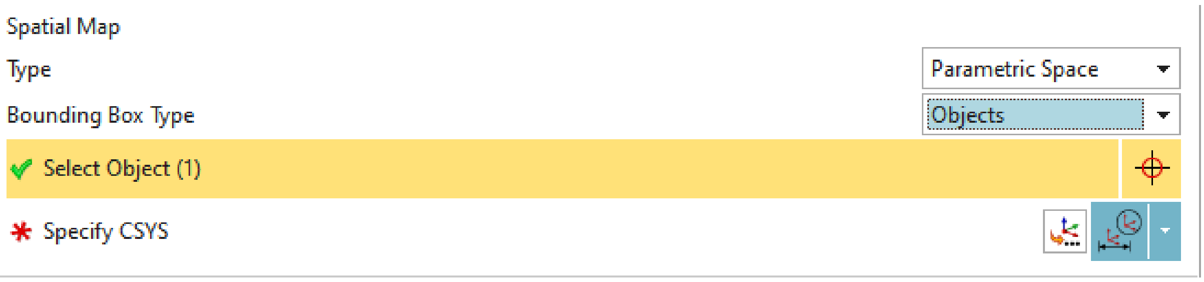


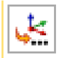
- A new dialog box *Formula Field* will open. We will write a simple formula describing the variation of an arbitrary pressure field on the left wing.
- In the *Domain* sub-box, select the option *Parameter Plane*.

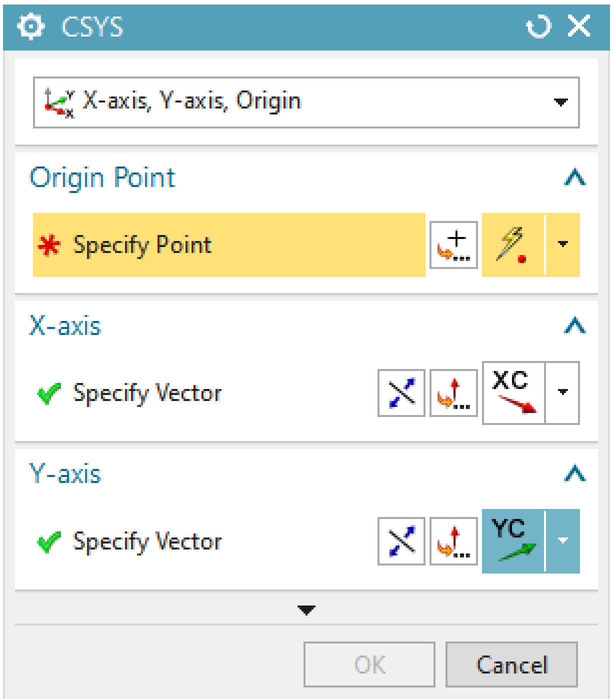


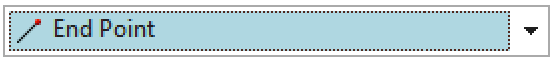
- You will see two new (dimensionless) variables appearing: *u* and *v*. These are variables that we will use in our pressure formula. These two variables correspond to the parametric coordinates of the B-Spline surface describing our left wing. See the course 5 of the CADCG course of E. Béchet for more details on the matter.
- What is needed to know here is that the variables *u* and *v* are real parameters both in the [0, 1] interval. Any point of the left wing surface can be accessed through these variables.

- The *Spatial Map* sub-box is here to set to which surface the parameters *u* and *v* will correspond. To set *u* and *v* to the left wing, select *Object* as *Bounding Box Type* and select the surface of the left wing.

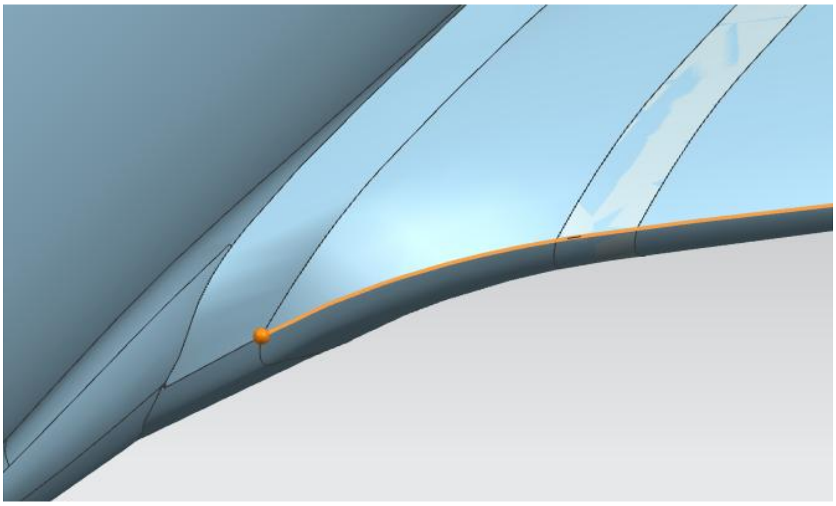


- For the local coordinates system (*CSYS*), we will use for the origin the front point of the base curve of the left wing, the same as the one chosen in paragraph 2.2. "Setting the constraints", and the X and Y axis of the global coordinates system.
- To this aim, click on the *CSYS Dialog* button , in order to open the *CSYS* dialog box. In this dialog box, select the *X-axis, Y-axis, Origin* option in the menu. Set the X axis as *XC* and the Y axis as *YC*.



- For specifying the origin of our local coordinates system, click on the *Point Dialog* button .
- In the *Point* dialog box, select the *End Point* option in the menu 

and select the front point of the base curve of the left wing, the same as the one chosen in paragraph 2.2. "Setting the constraints".



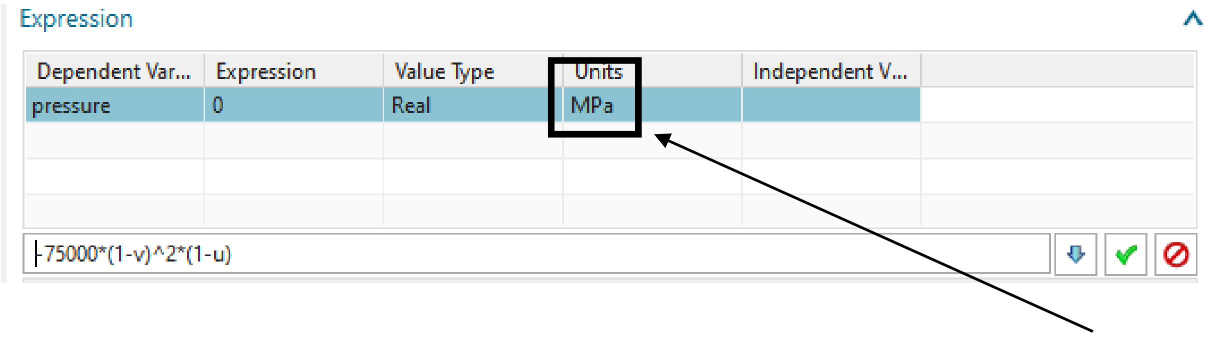
- Close the following dialog boxes by clicking on *OK*
    - *Point* dialog box.
    - *CSYS* dialog box.

- At this point you should be back in the *Formula Field* dialog box. Now that we have defined parametric coordinates, their corresponding B-Spline surface and a local coordinates system, we can finally set a pressure formula for the left wing.
- In the *Expression* sub-box of the *Formula Field* dialog box, enter the following pressure formula:

```
-75000*(1-v)^2*(1-u)
```

- This formula is expressed in terms of the parametric coordinates *u* and *v* and aims to mimic (very) qualitatively the pressure on an airbus wing[2]. The value *75000* is the mean pressure value on the wing in Pa. The negative value is there to tell that the pressure is in fact negative and the wing should be "pulled" towards the top (outward normal with respect to the surface).

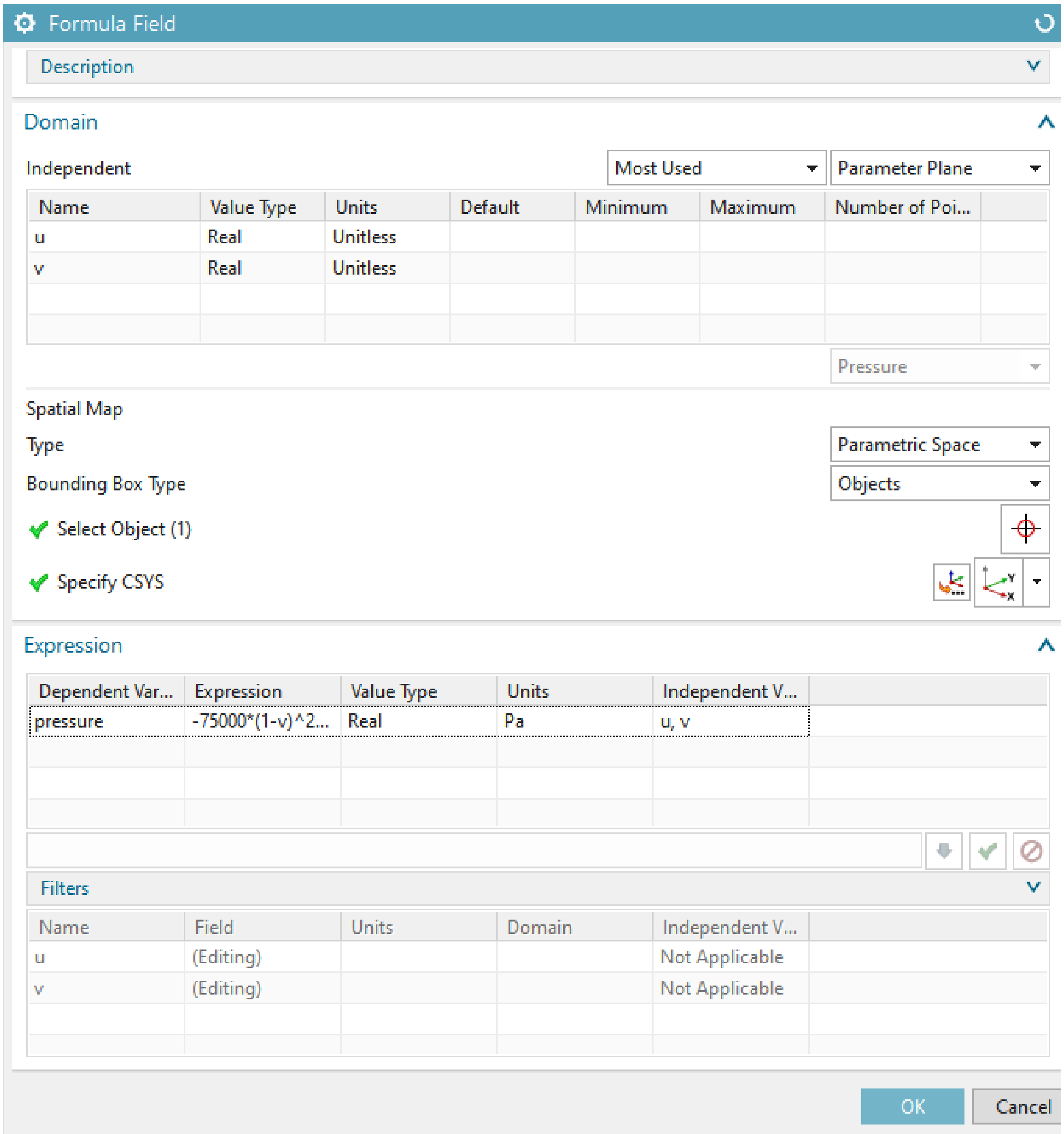
- Insert this formula into the appropriate field and click on the *Accept Edit* button .



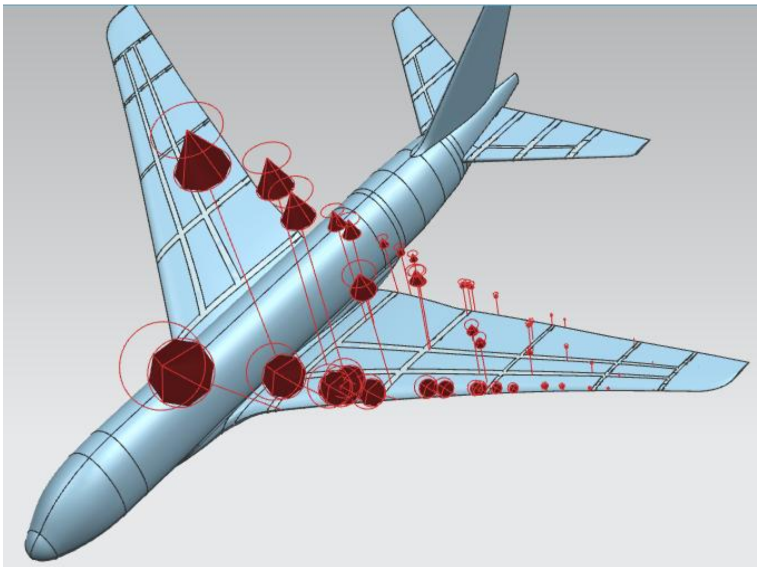
- Finally, change the pressure units in the *Expression* sub-box to ***Pa***, by right-clicking on *MPa*.

[2] Note : for real simulations, a better formula should be used.

- You should have the following *Formula Field* dialog box:



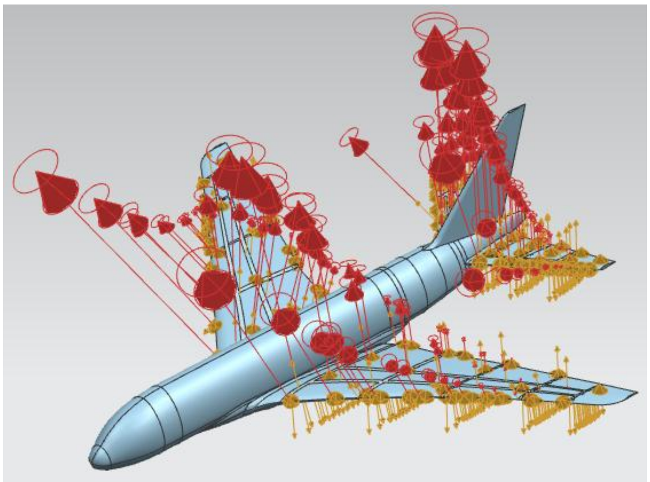
- Click *OK* to validate, and again *OK* in the *Pressure* dialog box.
- You should obtain read arrows figuring the local variable pressure field as follow:



4.5. Adding local variable pressure field on the other wings

- Redo the above operations for the right wing using this time for the local *CSYS* the front point of the base curve of the right wing, the same as the one chosen in paragraph 2.2. “Setting the constraints”. As pressure formula, use the following:

  `-75000*(1-v)^2*(u)`
- Redo the above operations for the left and right tails, using suitable local CSYS and the corresponding (left/right) pressure formula.
- You should obtain the following pressure distributions on all wings.



4.6. Final simulation

- Solve this new simulation 

, where variable pressure fields have been added to the, where variable pressure fields have been added to the wings. This will take several minutes to complete.
  WARNING: be sure to have at least 3 Gb RAM free.

- Again, visualize your result as explained in section 4.3 “Adding gravity and first simulation” of this tutorial. You should obtain a result similar to the one below.