2022

User manual

Version 0.90

Overview
ACE of Aircraft is a software made to help its users estimate performances of different aircraft designs.
In ACE of Aircraft, users can model the air flow, compute the aerodynamic coefficients, estimate the angle of stalling, etc., using different CFD methods.

Target audience
This user manual is directed to an audience with knowledge in aerodynamics and aeronautics. We assume that the user is accustomed with the concepts and terms presented or used in the software.

ACE of Aircraft

Table of contents

I. BEFORE USE **6**

1. Installation **6**
1.1. Requirements . . . 6
1.2. Windows 10 install . . . 6
1.3. Ubuntu 20.04 LTS install . . . 8
1.4. Setting-up a cluster running Ubuntu 20.04 LTS . . . 9

2. Updates **10**
2.1. upgrading ACE of Aircraft on Windows 10 . . . 10
2.2. upgrading ACE of Aircraft on Ubuntu . . . 11
2.3. upgrading ACE of Aircraft on a Ubuntu cluster . . . 11

3. License information **12**
3.1. License levels and corresponding features . . . 12
3.2. How to install a license . . . 13

II. USING THE SOFTWARE **14**

1. Navigation bar **14**

2. Projects **15**
2.1. Buttons . . . 15
2.2. Project types . . . 16
2.3. Analysis types . . . 19
2.4. Analysis contents . . . 20

3. Geometries **21**
3.1. Referential frame, forces and moments directions . . . 21
3.2. Parental bonds for geometries . . . 22
3.3. Parts VLM . . . 23
3.4. Parts NS2D / NS3D . . . 23

4. Models **26**
4.1. Importing a file . . . 27

5. Visualisations **28**

6. Charts **28**

7. Objectives **29**
7.1. Mission step types 29

8. Queue **30**

9. Settings **33**
9.1. General 33
9.2. User data folder 34
9.3. Third parties 34

10. Hardware **35**
10.1. Create a computing cluster 35
10.2. Buttons 36

11. Help **36**

III. LEXICON **37**

1. Airfoil lift coeff **37**

2. Cdmin / Cd0 **37**

3. Custom measurements **38**

4. Detail size **40**

5. Engine-propeller unit efficiency **44**

6. Engine power loss relative to altitude **45**

7. Flying altitude **45**

8. Custom refinements **45**

9. Parts movements and command angles **46**

10. Heat exchangers **48**

11. Hole detection points **48**

12. Mass and inertia **49**

13. Mean aero chord length **49**

14. Oswald efficiency factor **50**

15. Position of the center of gravity (X Y Z) **51**

16. Rolling / Pitching / Yawing moment with modified Center of Gravity positions **52**

17. Rotors **52**

18. Template NS **53**

19. Velocity for fixed pitch propeller **53**

20. Velocity for constant speed propeller **54**

21. Wing surface area **55**

IV. MODELING BEST PRACTICES 55

1. Avoiding bad comparisons **55**

2. Preparation of CAD for NS geometries **57**
2.1. Cutting the geometric modeling in several parts 57
2.2. Refinement of the adjacent and the detailed zones 58
2.3. Surface angles 61

3. STL Generation **62**

4. Exchanger heat exchange and pressure loss curves **63**

5. Self-evaluation of the calculation results **65**

V. CORRELATIONS 66

1. Detail size parameter **66**

2. Trailing edge thickness **69**

3. 2D turbulence model **71**

4. 3D turbulence model **73**

VI. HELP AND SUPPORT 77

1. Troubleshooting **77**

2. Contact us **78**

Part I.
BEFORE USE

1. Installation

1.1. Requirements

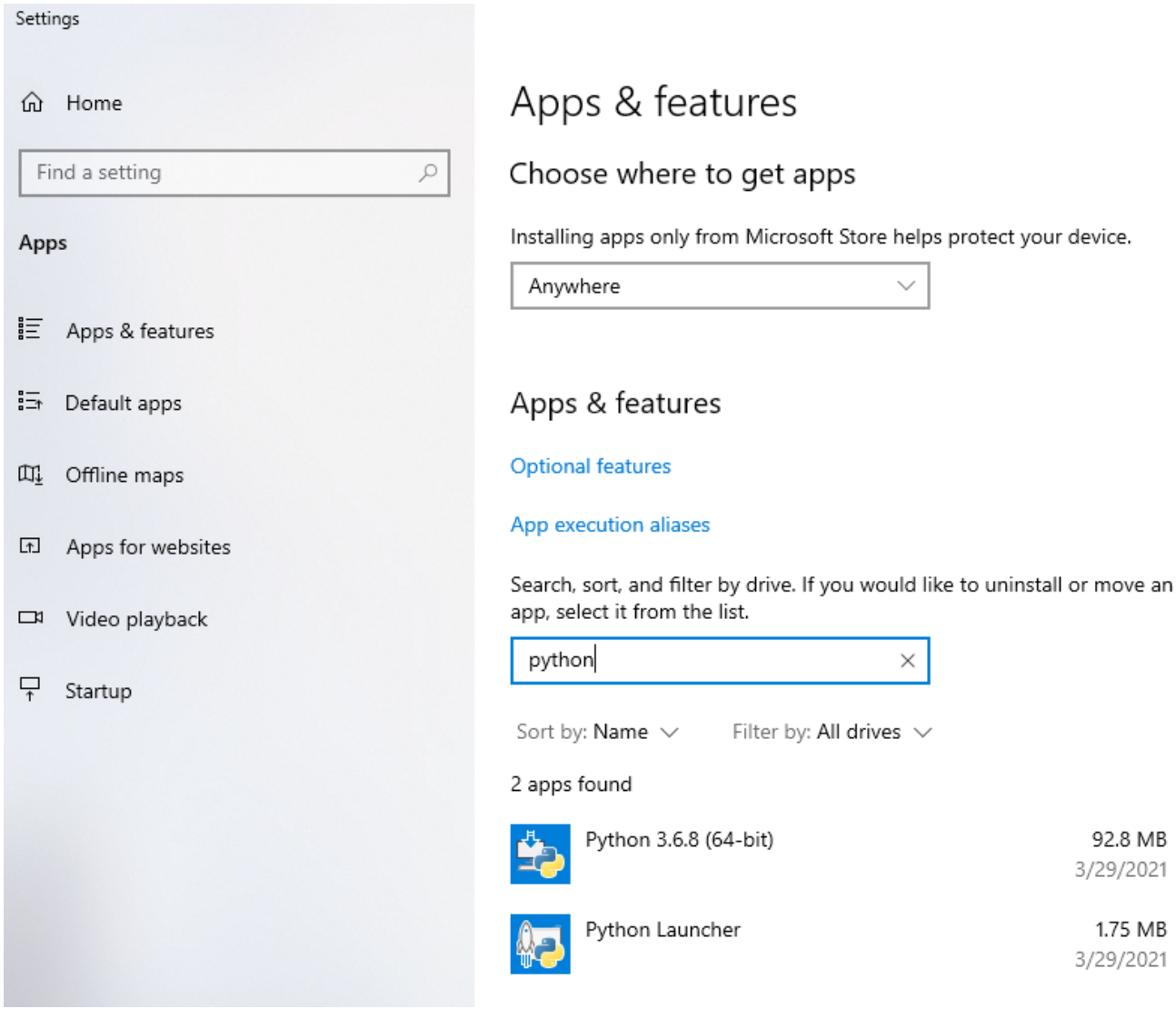
For basic features (Access and bronze licenses):
Supporting system : Windows 10 / Windows 11
Interpreter : Python 3.6+ 64 bits
CPU : no requirement
GPU : no requirement
RAM : 4 GB
HDD : 4 GB free

For advanced features (Gold and platinum licenses):
Supporting system : Ubuntu 20.04 LTS
Interpreter : Python 3.7+ 64 bits
CPU : 8 to 24 processing cores @ 2.2GHz or more, per computer. If possible a CPU designed for HPC, CFD-computing (AMD epyc, Intel Xeon Gold/Platinum, etc.). A better CPU will allow you to process Navier-Stokes computations faster.
GPU : GTX 1650 or higher
RAM : 21 to 128 Go of RAM per computer. More RAM will allow you to model geometriacal details of small scale.
HDD : 8GB free space for installation. Then we recommand to have 1 TB space or more to store the data that will be generated by Navier-Stokes computations.

1.2. Windows 10 install

Installation of Python 3

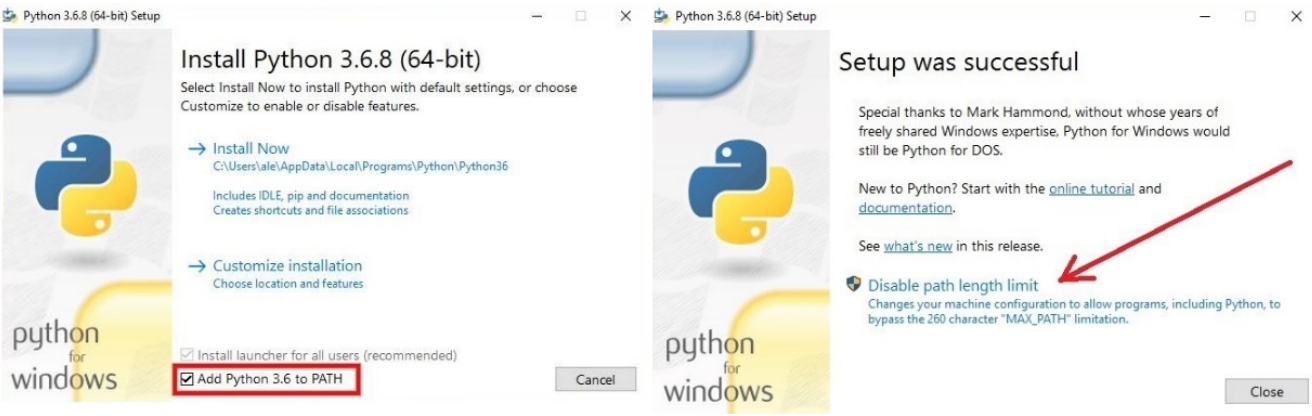
- Python 3.8+ 64 bits is required to use ACE of Aircraft. You may check if you already have this software by entering the following keywords in the windows search bar : “Add or remove programs”. This will lead you to a system settings window listing all software installed on the computer. In that window you wan use the search bar to search for “python”, as shown in the following image.



- If you already have an appropriate version of python, skip the following steps and proceed to install ACE of Aircraft. Otherwise, please download an appropriate version from python.org.

  Note: python versions 3.8.8 have been tested and is compatible. It is important to note that **only 64 bits versions are compatible.**

- Execute the installer for python 3.8.8 or higher.
- Make sure to validate "Add python 3.8 to PATH" during the installation, as shown in the following image.
- You can disable "path length limit", as shown in the following image.



Installation of ACE of Aircraft

- Find the installer named Ace_of_Aircraft_vX.XX_installer.exe.
- Execute this installer and follow the instructions of the installer.

1.3. Ubuntu 20.04 LTS install

The point of using the ubuntu version is to have access to the Navier-Stokes modeling which is powered by OpenFoam. This is available only at the gold and platinum license level. You will need an internet connection to install the software.

Before installing the software we advise to change the login screen of ubuntu from GDM3 to lightdm, because GDM3 often cause problems. In order to do that run the following commands:

sudo apt update
sudo apt upgrade
sudo apt install lightdm

When prompted, select lightdm then OK.

Go to the settings of ubuntu, in the power settings, and disable any kind of power saving setting such as the inactivity blank screen. This can cause computations to halt when the mouse and keyboard are not used.
Our software package has a couple of dependencies (OpenFoam and freeCAD). In order to allow your package manager to access them, you must run the following commands:

sudo apt install curl
curl -s https://dl.openfoam.com/add-debian-repo.sh — sudo bash
sudo add-apt-repository ppa:freecad-maintainers/freecad-daily
sudo apt-get update

Finally to install ACE of Aircraft, find the package file ace-of-aircraft-vX.XX.deb and install it with the command:

sudo apt install ./ace-of-aircraft-vX.XX.deb

1.4. Setting-up a cluster running Ubuntu 20.04 LTS

To reduce solving times of Navier-Stokes computations, you may want to build a cluster with several computer. One computer will run ACE of Aircraft graphical interface, we will call him "master" as he will be sending job orders to the other computers. The other computers will be called slave1, slave2, etc., as they merely execute Navier-Stokes simulations and send the data back to the master.

For the cluster to work properly it is important to set each of them exactly the same way. It is required that :

- The same user exists on all computers, with the same password
- This user account must have the same user ID on all computers. If possible, install users in the same order to facilitate the installation process
- All computers must have a fixed IP address
- All computers must have the same version of ACE of Aircraft installed as described in the preceding chapter. But only the master computer need to have a platinum license installed. The slave computers do not need any license.

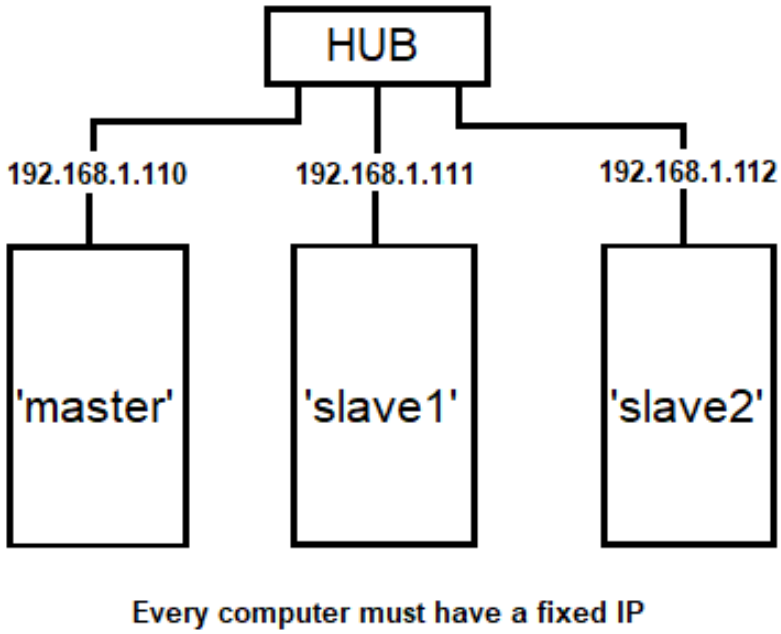


Figure 1: Representation of a computing cluster.

Slave computer set-up

On each slave computer, open a terminal and launch the command:

sudo apt install openssh-server nfs-kernel-server

Answer Yes when promped. You will require an internet connection for this step. The command will install software necessary for the computers to connect to each other and read data from network shares.
Then run:

sudo mkdir /Ace_of_Aircraft_datadir

This will create a folder were the data share will be mounted, so that the slave will be able to read and write data on the master computer. By default the data is situated on the master computer, at /usr/lib/ace-of-aircraft/ace-of-aircraft-vX/datadir . In the following example we will suppose this is the case, but the data can be placed somewhere else if needed. To mount this share on your slave computer, you must edit the fstab file with the following command:

sudo nano /etc/fstab

At the end of this file, add the line:

masterIPadress:/usr/lib/ace-of-aircraft/ace-of-aircraft-vX/datadir /Ace_of_Aircraft_datadir nfs -netdev,bg,user,hard,intr 0 0

Master computer set-up

On the master, open a terminal and launch the following command to create an authentification key:

ssh-keygen

Then for each slave computer execute the following command to associate the key onto the slave machine:

ssh-copy-id IP-adress-of-slave

And finally edit the exports file with the command:

sudo nano /etc/exports

At the end of this file, add the line (This is supposing the datadir is at the default adress):

/usr/lib/ace-of-aircraft/ace-of-aircraft-vX/datadir/ 192.168.1.(rw,sync,no_root_squash,no_subtree_check)*

Then restart all computers.

2. Updates

2.1. upgrading ACE of Aircraft on Windows 10

This is the recommanded way to upgrade from one Ace_ofAircraft version to another:

- Install the new version.
- Go to the installation directory, by default : %AppData%
- Copy your projects from the older version to the newest version. By default you projects are at the directory %AppData%/Ace_of_Aircraft_vX/datadir
- At the same location, copy the userdata folder, containing your settings and license
- Check the file %AppData%/Ace_of_Aircraft_vX/configuration.py, any special configuration made in this file should be copied to the file of the new version.
- Launch the new version and check that all your data and settings have been correctly transfered to the new version.
- Facultatively, you may uninstall the old version using windows system settings apps, "Add or remove programs section". Note that this will not delete your projects data, if you want to clear space you will have to delete files at %AppData%/Ace_of_Aircraft_vX/datadir manually.

2.2. upgrading ACE of Aircraft on Ubuntu

This is the recommanded way to upgrade from Ace_of_Aircraft, supposing you upgrade from version X to version Y:

- Install the new version Y with the .deb package. This will create the directory /usr/lib/ace-of-aircraft/ace-of-aircraft-vY
- Locate the application directory for the old version at /usr/lib/ace-of-aircraft/ace-of-aircraft-vX.
- Copy your project files and the userdata folder, from the old version data-directory to the new one (by default at /usr/lib/ace-of-aircraft/ace-of-aircraft-vX/datadir).
- Check the file /usr/lib/ace-of-aircraft/ace-of-aircraft-vX/configuration.py, any special configuration made manually in this file should be copied to the file of the new version Y.
- Launch the new version Y and check that all your data and settings have been correctly transferred to the new version.
- Facultatively, you may delete the old version's directory.

2.3. upgrading ACE of Aircraft on a Ubuntu cluster

Proceed as described on the preceding chapter on the master computer. Then check if the datadir path in the exports file must be updated, with the command:

sudo nano /etc/exports

For the slave computers, install the new version Y with the .deb package. If needed, edit the fstab line to mount the correct datadir share:

sudo nano /etc/fstab

3. License information

3.1. License levels and corresponding features

The following table lists the features unlocked by each license type.

License level	Access	Bronze	Gold	Platinum
Analytic tools	√	√	√	√
Models	√	√	√	√
Draft projects	√	√	√	√
VLM projects		√	√	√
NS projects			√	√
Additional options for NS projects *				√
OS compatibility	Windows or Ubuntu	Windows or Ubuntu	Ubuntu	Ubuntu

Figure 2: License levels.

To understand the differences between Draft, VLM and NS projects, see part 2.2.

*** Platinum additional options for NS projects:**

At gold level there is only one turbulence model available in NS projects, the SA model. The platinum license gives you an extended choice of turbulence models : SA, KW, SA_unsteady, KW_unsteady. You can find more details about these turbulence models at the end of part 2.2.

The platinum license also allow you to use a cluster of computers to solve your Navier-Stokes run queue, instead of a single computer.

Additionally the platinum license unlocks multiple boundary conditions. With the gold license,

any surface imported from CAD files will become an airtight wall surface in the model. With the platinum license more options are given, which enables the modeling of propeller flow, heat-exchangers, air-intake, air-exhaust and flow-report surfaces. See part 3.4 for more details.

3.2. How to install a license

You can execute ACE of Aircraft without installing a license, but most features will be disabled. To request a license, launch ACE of Aircraft and follow these steps:

- Click on the "Settings" tab on the top right corner of the application.
- In the bottom right corner of the window, you will find your machine ID. Copy it and use it in the form available at : *www.aero-ce.com/en/ace-of-aircraft-downloads*
- On the 'Settings' screen is also displayed the address of the **userdata folder**. Note down this address, you will need it later.
- ACE will return you a license file named "lic.dat" by email.
- Paste the license file into the **userdata folder**, or use the 'import a license file' button in the "Settings" tab.
- Restart ACE of Aircraft and check the license validity in the "Settings" tab of the application.

Part II.
USING THE SOFTWARE

1. Navigation bar

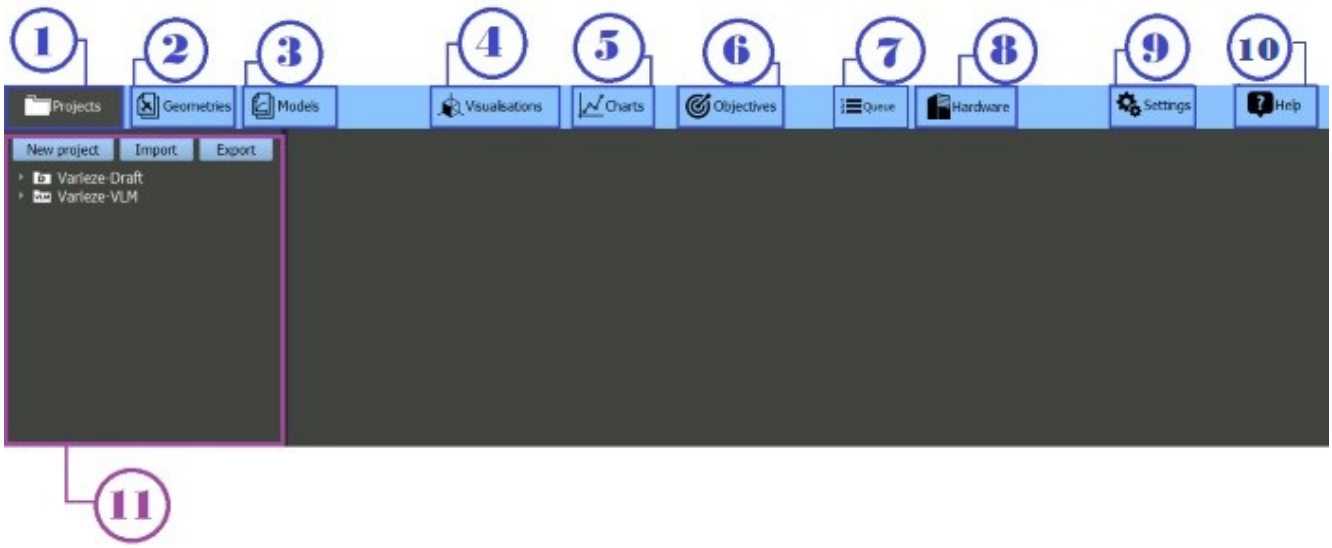


Figure 3: Screenshot of the software navigation bar.

1. **Projects tab**

   A project is a directory containing various data generated during a study.
   For more information, see 2. Projects

2. **Geometries tab**

   Geometry is the spatial modeling of the airplane shape cut into different parts.
   For more information, see 3. Geometries

3. **Models tab**

   Models are simple numerical laws binding a physical condition to another.
   For more information, see 4. Models

4. **Visualisations tab**

   3D visualisation and pictures to inspect and compare the Navier-Stokes results.
   For more information, see 5. Visualisations

5. **Charts tab**

   Charts is a tool to compare different configurations.
   For more information, see 6. Charts

6. **Objectives tab**

Define objectives to evaluate your designs and compare them.
For more information, see 7. Objectives

7. **Queue tab (or job manager)**

   Launch or stop runs of Navier-Stokes projects.
   For more information, see 8. Queue

8. **Hardware tab**

   See the CPU, RAM and disk usage. Add or remove computer(s) to the computing cluster.
   For more information, see 10. Hardware

9. **Settings tab**

   Add or remove a license. Get access to elements not allowed by your license. Define the default units.
   For more information, see 9. Settings

10. **Help tab**

    Manual, tutorials, support and contact.
    For more information, see 11. Help

11. **Tree view**

    Each project produced creates a folder containing all the data you enter via the Projects, Geometry and Models tab.
    You can expand each folder to see its content and open any data study you have created.

2. Projects

A project is a directory containing various data generated during a study.
Projects have an associated "type" indicating what kind of numerical modeling is used (Oswald efficiency factor, VLM, Navier-Stokes 2D or Navier-Stokes 3D).

2.1. Buttons

New project: create a new project.
Import: import a data folder from your computer.
Export: export a data folder. It is useful if you do not have a license and want to transfer your data folder to someone else.

The bin allows you to delete an analysis, a geometry, a data folder...
The pencil allows you to modify an analysis, a geometry or a model. If it is "locked" you cannot modify it anymore. You will probably need to delete children items in order to have this element come back to "unlocked" state.

2.2. Project types

Draft

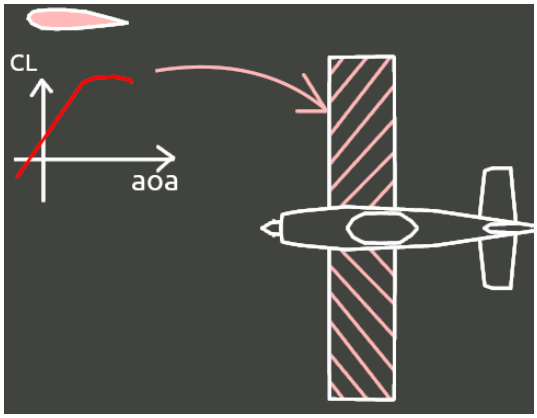


Figure 4: A draft analysis roughly models a lift law over a wing area.

A draft project requires the user to import a lift law from an airfoil polar, and will then apply this law according to the wing area given by the user. The Oswald efficiency factor (induced drag equation) is used to adjust the results from the 2D airfoil data to a more realistic 3D estimated lift and drag.

→ Estimate the performances of the lifting wing only.

- Pros :

  Quick. Only one equation to solve.

- Cons :

  Inaccurate, large approximation, no momentum analysis.

VLM

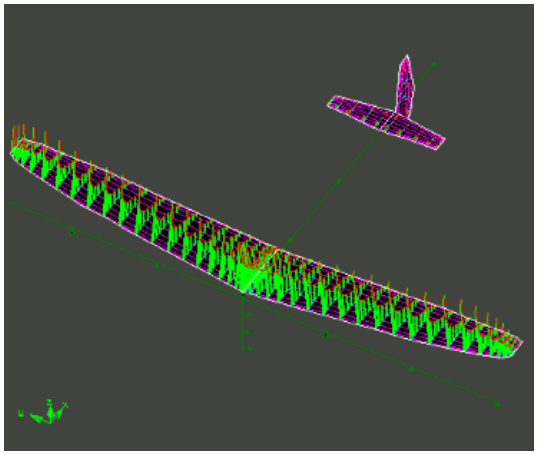


Figure 5: VLM computations are based on a rudimentary 3D geometry model.

Calculation using a third-party: AVL (Athena Vortex Lattice) by Mark Drela and Harold Youngren. This calculation is more precise than the Draft: a 3D geometry is used, modeling the airfoils in 3D, including the empennage.

→ Estimate the performances of an aircraft architecture.

- Pros :

  Momentum analysis (but outside of stall effects). The quickest model type for dynamic calculations. 3D interactions between aircraft parts.

- Cons :

  Basic geometry. Cannot model multi-element airfoils (slotted flaps) and under-cowl modeling. Inexact modeling of the drag and no modeling of the stall.

NS2D

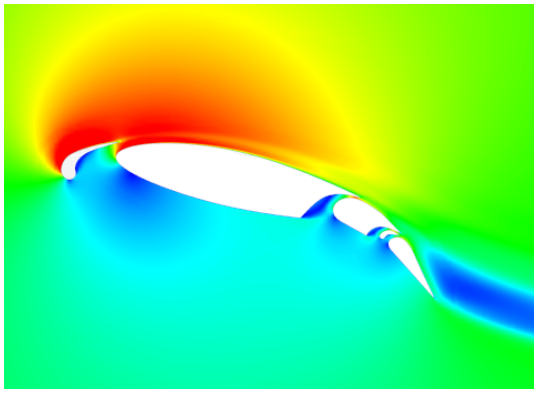


Figure 6: Example of a NS2D solution.

Calculation using a third-party: OpenFoam. Modeling of 2D airfoils. Accurate and detailed modeling of the air flow (speed, pressure and turbulence).

→ Estimate the performances of airfoils or 2D shapes. Can calculate slotted flaps and other multi-element airfoils. Generally used for the optimization of flaps and movement of control surfaces.

- Pros :

  Detailed air flow model, can evaluate multi-element airfoils.

- Cons :

  A precise CAD of the airfoil is required. The calculation is longer than the VLM.

NS3D

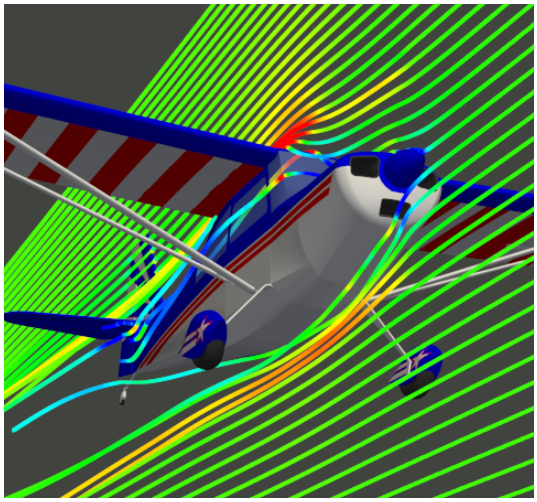


Figure 7: Example of a NS3D solution.

Calculation using a third-party: OpenFoam. Results in 3D for wings or detailed airplanes. Complete flow modeling with stalls effect, momentum, pressure gradients, ground effect. Can be used to study the impact of landing gear, optimize the shape of winglets, pods, fuselage, inflow ducts. Even good for modeling under-cowl environment and internal airflow. Possibility to add rotor models and porous zones (for the modeling of heat exchangers).

→ Estimate the performance of an airplane. Can calculate air flows precisely and optimise the airplane parts in detail.

- Pros :

  Closest to reality because of its accurate 3D geometry definition. Can find the ideal postion for Pitot tubes, static pressure tap or optimizes the external and undercowl part shapes.

- Cons :

  A precise and clean CAD is required. The calculation is the longest.

Turbulence models option

- SA

Turbulence model using the Spalart-Allmaras equation. Quick and simple model. Suitable for airfoils and tapered bodies.

- KW

Turbulence model using two equations : k and w (k-omega) with SST (Shear Stress Transport). This is a sligthly more complex model. It can be better for bluff bodies, modeling

the wake, modeling under-cowl elements. This model can give slight inaccuracy in stalling angle determination. However stalled results are generally more accurate.

- SA_unsteady

DDES (Detailed Detached Eddy Simulation) based on the SA model. Modeling several time steps, therefore modeling Vortex shedding and unsteady characteristics of the flow, which generally gives a better accuracy. Simulated duration is set automatically and the solution is averaged on the last few time-steps of the iteration. The computation duration will be long compared to the preceding models.

- KW_unsteady

IDDES (Improved Detailed Detached Eddy Simulation) based on the k-omega-SST model. This unsteady model with improved wall-modeling capability should be used when accuracy is paramount. The computation duration will be the longest.

For more information, see part V. Correlations.

2.3. Analysis types

Analysis types for VLM projects

- Aerodynamics and sideslip :

  Analysis of an airplane basic performances.
  Computation of several angles of attack and angles of sideslip, to check the stability of the airplane design.

- Forward longitudinal stability limit (VLM) :

  Investigation of the CoG forward limit for longitudinal stability.

- Stability derivatives (static,VLM) :

  Measurement of the rate of change in each aerodynamic coefficient as the angle of attack and angle of sideslip changes. (Calculation done for near-zero angles).

- Dynamic stability (VLM) :

  Estimation of dampening and frequency value when the aircraft faces various stability modes, including dutch-roll, phugoid and short period.

- Custom (VLM) :

  Define the parameters of your run(s) : add run(s), select the configuration you want to apply, add option(s).

- Manual :

  It opens an AVL terminal with the selected geometry pre-loaded.

Analysis types for NS projects (NS2D and NS3D)

- Aerodynamics :

  Computations at different angles of attack which will determine the main performances criteria (stalling angle, maximum lift, maximum lift to drag ratio...).

- Aerodynamics with Reynolds correction :

  Two sets of computation at different angles of attack, for two velocities. Interpolation of a polar at a constant mass. The main performances criteria are more precise.

- Custom :

  Define the parameters of your run(s) : add run(s), select the configuration you want to apply, add options.

Analysis types for NS3D projects

- Stability derivatives (static) :

  Creation of 3 runs depending on the angle of attack (AoA) and angle of sideslip (AoS) :

  1. A run with the AoA and the AoS at 0°.
  2. A run with the AoA at +1°.
  3. A run with the AoS at +1°.

  The results will give the stability derivatives of each aerodynamic coefficient.

- Forward longitudinal stability limit (VLM) :

  Investigation of the CoG forward limit for longitudinal stability.

2.4. Analysis contents

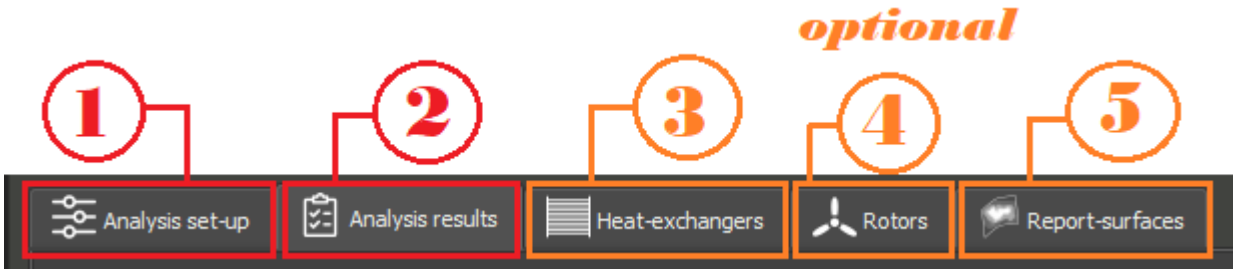


Figure 8: Software screenshot of the tabs presented in the analysis screen within the Projects tab.

1. **Analysis set-up**

   Summary of the information you entered when you created your analysis.
   See the advancement of your analysis.

2. **Analysis results**

   Results of your analysis.
   More information depending on the type of analysis.

3. **Heat-exchangers**

   Results of the heat-exchanger(s) calculation
   Including a results visualisation.

4. **Rotors**

   Results of the rotor(s) calculation
   Including a results visualisation.

5. **Report-surfaces**

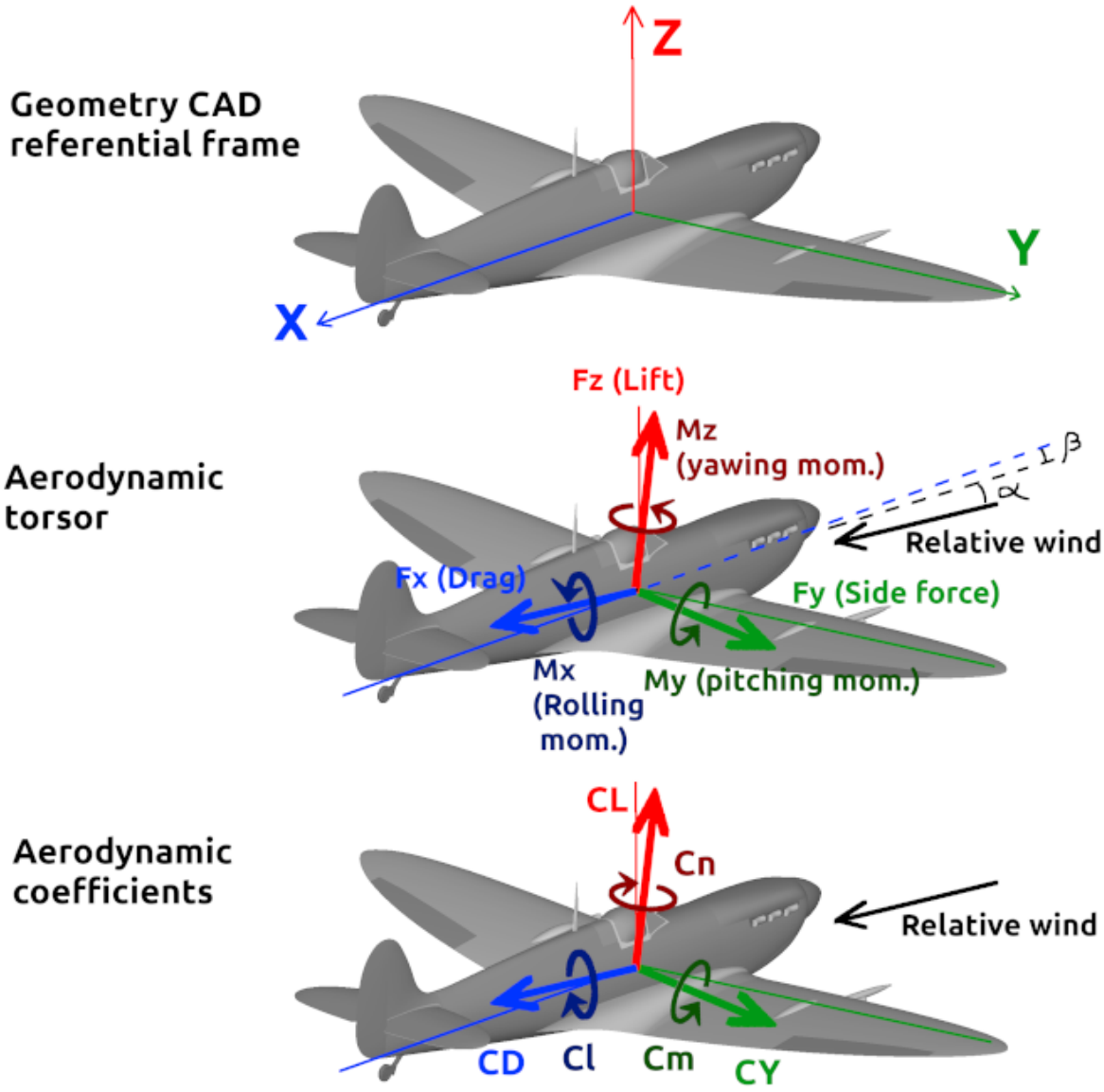
   Results of the report-surface(s) calculation
   Including a results visualisation.

3. Geometries

Spatial modeling of an airplane shape cut into different parts.
You need to define the placement and the delimitation of the parts of your aircraft.
Note: draft projects do not use geometries .

3.1. Referential frame, forces and moments directions

All parts created in the software or imported in this software should use the following referential frame:



In the results of the aerodynamic simulation you may find torsor values indicating the aerodynamic forces and moments, as well as the aerodynamic coefficients.

Please note the Cn is in the opposite direction of Mz, and Cl is in the opposite direction of Mx. This is because the standard aerodynamic referential frame as the Z axis down, and X axis forward, which is not the referential frame expected for the geometry in our case.

3.2. Parental bonds for geometries

When creating or editing a geometries, you can set if you wish another geometry as its 'main parent'. Then is is also possible to define one or more geometries as 'other parents'.
This functionality allows the user to set parental bonds between geometries. Their relative hierarchical positions can then be seen a tree (available from a button display on a geometry

page).
This parental bonds, with the use of comments in the geometry definition, can help the user to arrange and classify his data, to easily keep track of design branches.

3.3. Parts VLM

Define the different parts of your aircraft.
Each part is a geometrical element, it can be a wing, a fuselage, a winglet...
The aerodynamic forces applied will be calculated on each delimited part.

There are two types of parts : body and surface. The first type uses a slender body model, while the other should be prefered for wings and stability surfaces.

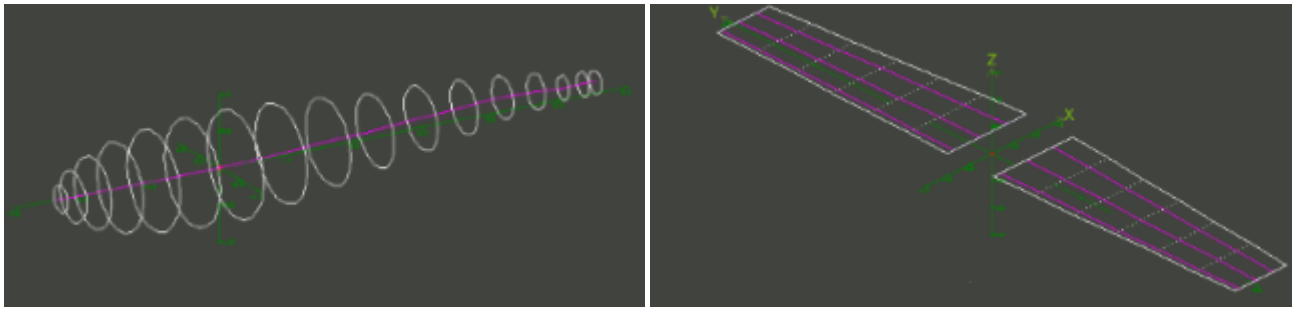


Figure 9: Software picture of the body and surface parts.

Add the center of gravity.
Make sure the unit used for the parts (mm, m, inch, ft) is correct.

For more information, see the AVL manual in the References compressed folder.

3.4. Parts NS2D / NS3D

Define the different parts of your aircraft.

Each part is a geometrical element, it can be a wing, a fuselage, a winglet, (...) and must be an .igs file or .stl file.
The aerodynamic forces applied will be calculated on each delimited part.

The software allows you to change the name of each part you have imported: it is easier for you to differenciate the parts in each geometry.
Make sure the unit used for the parts (mm, m, inch, ft) is correct.

NS3D only

Define a type for each imported file :

- Airtight surface :

  A hermetic surface where the air cannot get through.

- Flow report surface :

  A surface to visualize velocity and pressure (it will not impact the air flow).

- Air intake or exhaust :

  Model the air intake or exhaust. This is a surface boundary to which a flow rate is applied.

- Exchanger volume :

  Volume of the radiator core. It does not include the water box but only the volume where the heat exchange takes place. See 4. Exchanger models in 4. Models below.

- Exchanger inlet surface :

  Help to calculate the distance from the inlet to the outlet. The surface must be plane and parallel to the outlet.

- Exchanger outlet surface :

  The surface must be plane and parallel to the inlet.

- Rotor volume :

  It must be cylindrical and positioned where the propeler blades are. Cf Chapter 17.

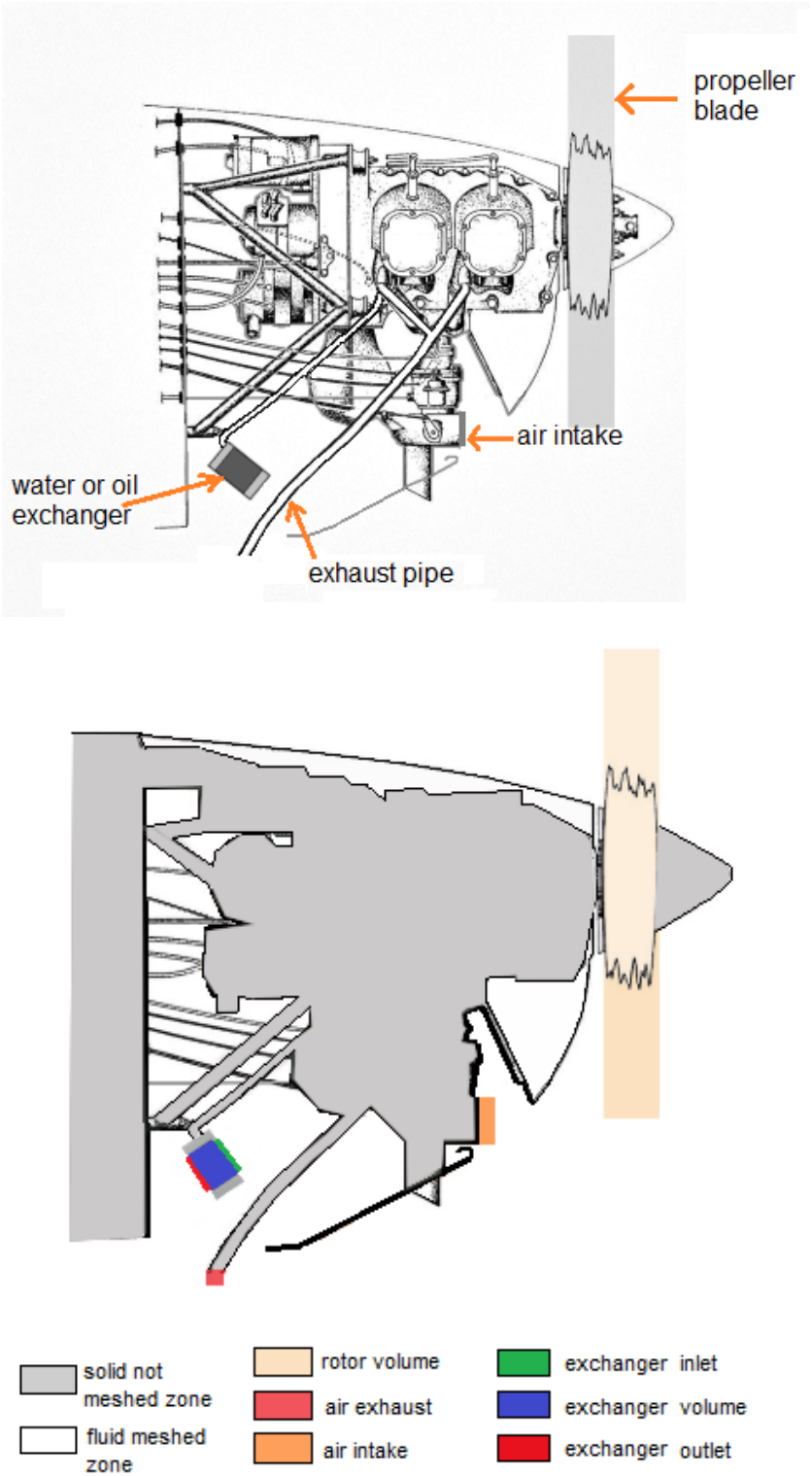


Figure 10: Aircraft under-cowl schemes.

4. Models

Models use user-provided values to create numerical laws binding a physical condition to another. These models are relatively simple and fast to solve.

Power models use experimental data (technical note N.640 from the NASA) to build a model of propeller thrust and torque.

For more information on the construction of the power models in ACE of Aircraft, see the ACE of Aircraft - power models file.
You can find the technical note and the ACE of Aircraft - power models PDF file in the References compressed folder provided.

Multiple types of models can be added to studies :

- **Power model (fixed pitch propeller)** :

  The addition of the engine and the fixed pitched propeller.
  Modeling of the power given to the airplane. This modeling will be determined by some parameters (speed of the propeller, density of air,...).

- **Power model (constant speed propeller)** :

  A variable speed propeller turning depending of the speed of the aircraft.
  Same modeling as the fixed pitch propeller with a constant speed propeller instead.

- **Exchanger models** :

  The software define two laws depending on the air velocity crossing the exchanger :
  - The pressure drop
  - The heat exchange

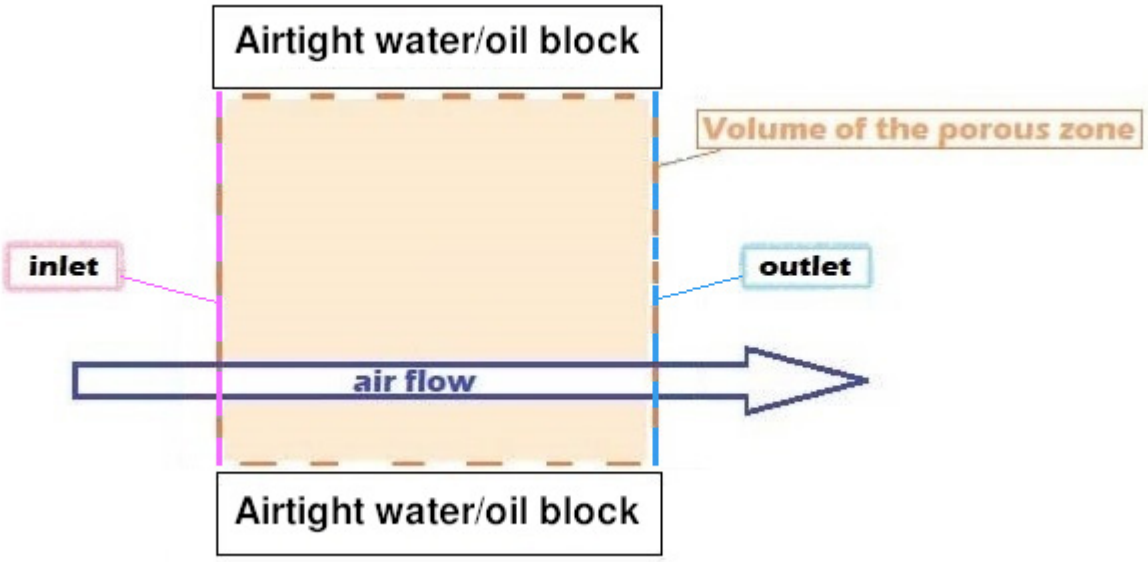


Figure 11: Air flow through an exchanger.

The heat exchanger is modeled by a porous zone.
The pressure drop and heat exchange will be calculated from the velocities passing through the zone.

Be cautious in your CAD modeling :
The inlet and outlet must be two parallel planar surfaces.

- **Airfoil model** :

  2D geometric definition of an airfoil, from a list of points with X and Y coordinates. A Selig format file should be used to define this list of points.

  The Selig format file must be in a .dat file extension to be exported in the software.
  → You can find .dat files for various airfoil at airfoiltools.com, or export .dat files from Xfoil or XFLR5 softwares.

  The Selig format file is a succession of precise points that must be defined in the order **(A)** → **(B)** → **(C)**.

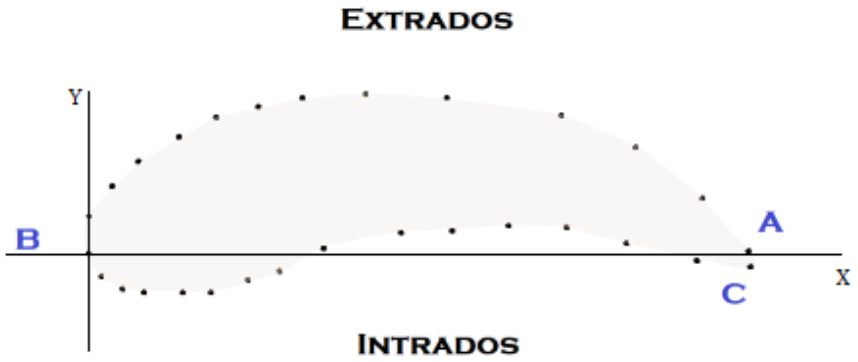


Figure 12: Content of a Selig file.

- **Aerodynamic Coefficient model** :

  Import external aerodynamic coefficient results to compare them in the Charts tab.
  Useful to import and compare experimental data.
  Complete the table of values or import a file containing the values.
  → To import the file, click on the "Load table from file" button.

4.1. Importing a file

Import a .csv or .dat file containing values of the angles of attack (AoA), sideslip (AoS) and the aerodynamic coefficients :

- It must contain 8 columns : AoA, AoS, CD, CY, CL, Cl, Cm, Cn.
- The first line has to start with "# " or "//" : the software will not read it because it is a commentary.
  Between each values, insert a comma (,) or a blank space.

– You need to provide data at least for the AoA and AoS columns.
If you do not have values for a column, do not leave it empty: type "NA" or any letters.

Here is an example to illustrate :

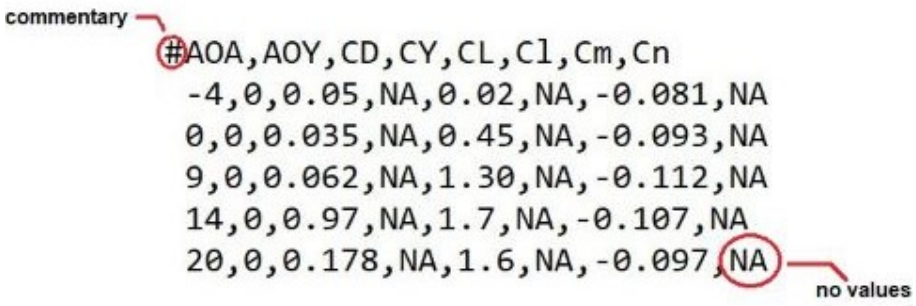


Figure 13: Example of a .csv file.

5. Visualisations

A tool to quickly get visualisations of Navier-Stokes runs (NS_2D or NS_3D projects) inside an interactive window. The tool can be set to visualise from 1 to 4 runs at a time in split frames. Selecting more than one run permits visual comparisons between the runs.

6. Charts

Fast chart building tool to compare the results of different analysis: select a type, select the different project and analysis, add option(s), filter the data you want to use.

→ The different types of comparisons :

- Global coef. :

  Comparison of aerodynamic coefficients. You can select multiple runs except for draft projects.

- Part forces :

  Comparison of force or moment coefficients. You can select multiple runs and parts.

- Rotor data :

  Comparison of thrust or torque. You can select multiple runs and rotors except for draft projects.

- Exchanger data :

  Comparison of flow rates, inlet pressures, outlet pressures and heat exchanges. You can select multiple runs and exchangers except for draft projects.

- Report surface flow rates :

  Comparison of flow rates and pressures. You can select multiple runs and surfaces except for draft projects.

→ Comparison with external analysis :
You can import external aerodynamic coefficients data to do a comparison with any analysis calculated in the software.

To add external data :

1. Go to the Model tab.
2. Select a folder project where you will save your data.
3. Select the "Aerodynamic Coefficient Model" model type.
4. Complete the Name imput.
5. Import the external results.

For more information about the importation, see 4. Aerodynamic Coefficient Model.

→ Save the results :
Comparisons will not be saved into the Charts tab.
To save your comparison, click on the save button at the top of comparison curve :

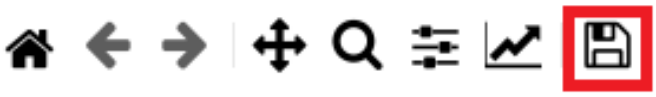


Figure 14: Software screenshot of the tool bar in the Charts tab.

7. Objectives

The objectives page contains two tools:

- **A mission tool :** A mission is defined by setting various masses, fuel and battery capacity for the aircraft, as well as a list of steps describing the mission profile. In the next chapter the different types of mission steps available are described in more details. Once the mission is created the user can select and aerodynamic analysis and power model to run the mission profile. This will give an estimation of fuel quantity used and duration.
- **A pugh matrix tool :** This let the user define scoring rules on several performance criteria or mission results. This rules can then be applied to an aerodynamic analysis to obtain scores and compare objectively concurrent designs.

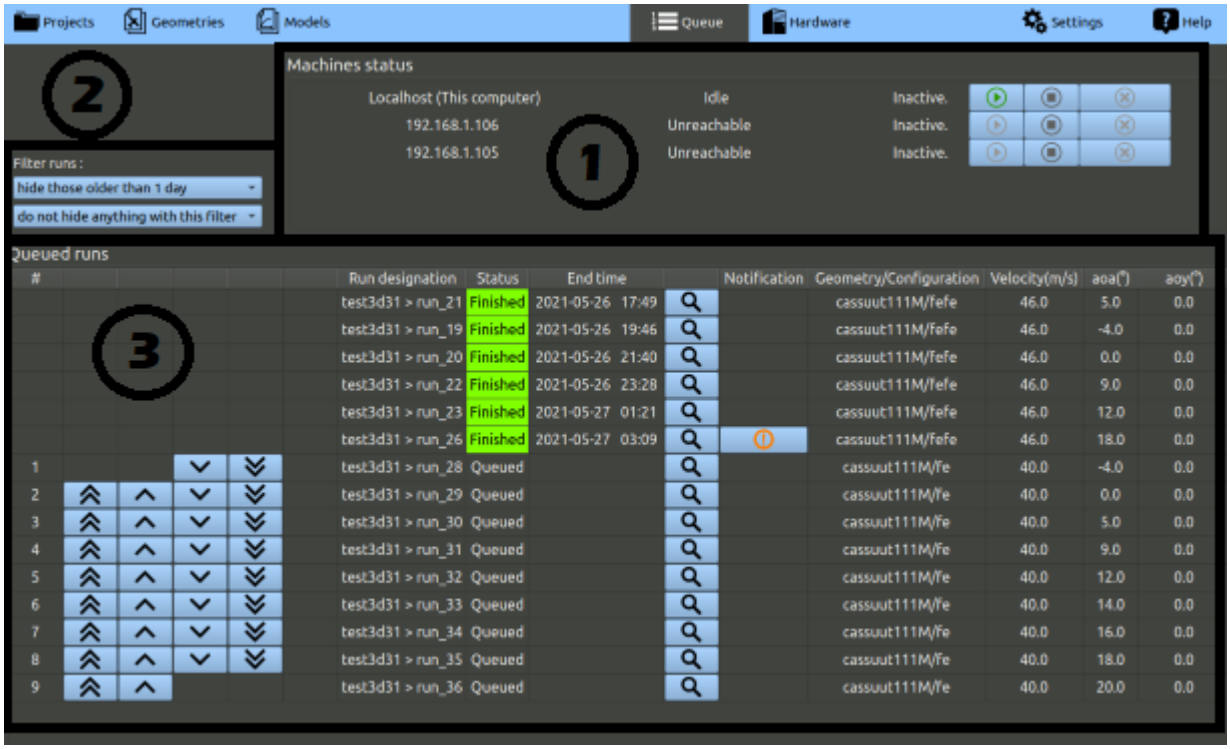
7.1. Mission step types

- **Altitude change :** to model a climb, or a descent if a negative value is given. The movement is calculated at the best climb rate speed found. The computation is actually made of 50 sub-steps (the weight and the best climb rate being re-evaluated at each sub-step).

- **Cruise over distance :** A distance and a speed value must be set. The speed can be automatically computed to be the speed giving the best range (longest distance flown per watt) or the best endurance (longest flight duration per watt). The step computation is actually made of 200 sub-steps.
- **Cruise over given time :** A duration and a speed value must be set. The speed can be automatically computed to be the speed giving the best range (longest distance flown per watt) or the best endurance (longest flight duration per watt). The step computation is actually made of 200 sub-steps.
- **Cruise until fuel threshold :** A threshold of fuel mass and a speed value must be set. The speed can be automatically computed to be the speed giving the best range (longest distance flown per watt) or the best endurance (longest flight duration per watt). The step computation is actually made of 200 sub-steps.
- **Cruise until battery threshold :** A threshold of battery capacity and a speed value must be set. The speed can be automatically computed to be the speed giving the best range (longest distance flown per watt) or the best endurance (longest flight duration per watt). The step computation is actually made of 200 sub-steps.
- **Fuel mass change :** A change of fuel mass, positive or negative, without movement calculation and instantaneous. It can be used to model fuel lost into taxiing or refueling for example. It can also be put at the last part of the mission and represent a given amount of reserve fuel that must not be used during the mission but kept for security.
- **Payload mass change :** A change of payload mass, positive or negative, without movement calculation and instantaneous. It can be used to model cargo-loading, cargo jettison, passengers loading, or skydivers jumping for example.

8. Queue

Queue or Job manager.
Launch or stop queue processing for different calculations.



1. **Machine status** :

   You can launch, pause or stop runs on the computer of your choice (including the computer(s) from your computing cluster).

2. **Filters runs** :

   You can hide or show runs depending on the day or hour.

3. **Queued runs** :

   See the status of your run (queued, finished or failed).
   Rank the runs according to the arrows.
   For more information, see Buttons below.

Buttons

If the status is 'inactive', click on this button to start or resume the runs.

The **black** button stops all runs after the end of the calculation.
The **red** button stops all runs and puts the calculation on failure.

Change the order of your runs with the arrows:
The double arrow will set the run at the top or at the bottom.
Simple arrow will set the run one position before or after.

This is a notification.
It is recommended to look at it.

There is a problem, you must look at it.
You can see more details in the logs of the run. Follow the instructions to spot the problem.

This button will open the results.

9. Settings

9.1. General

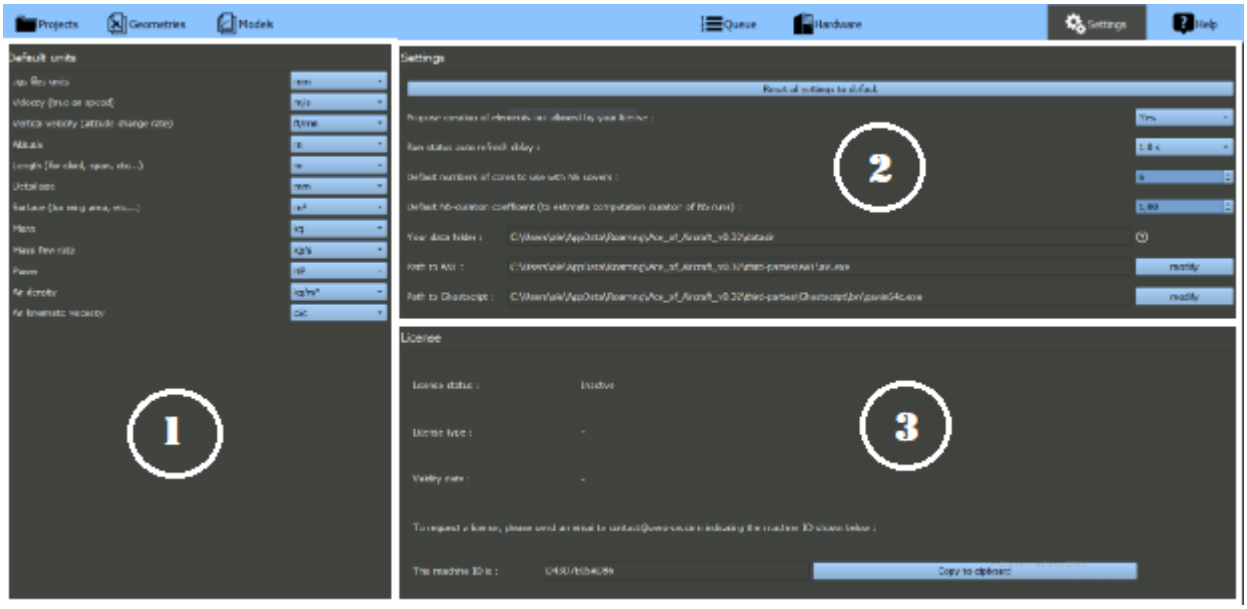


Figure 15: Software screenshot of the Settings tab.

1. **Default units** :

   Set the default units you will use in the software.

2. **Settings** :

   - Propose creation of elements not allowed by your license :

     Visualize elements not allowed in the free version. It allows you to prepare analyses without a license.
     To be able to run the calculation, you need to export the study data and send it to someone in possession of a license.

   - Run status auto-refresh delay :

     Set the number of second you want between updates on the pages having an auto-refresh feature.

   - Default numbers of cores to use with NS solvers :

     Set the default numbers of cores you want to use to solve Navier-Stokes calculations. This is the number set by default for every computer you add.
     You can change this number for each computer in the Hardware tab.

   - Default NS-duration coefficient :

     The time estimation has been calculated for 2.2GHz and 12 cores CPU.
     If your computer is faster, set the number below 1. Otherwise, set the number above 1.

3. **License** :

To install a license , please refer to chapter 3.2.

9.2. User data folder

The data folder indicated in the settings tab is composed of different sub-folders containing various data:

- userdata : the sub-folder where you copy / paste your license. This sub-folder also contains other user parameters.
- Project folders : each folder has the name of the project you created.
  In each of the project folders, you may find the Exports sub-folder containing the data you exported.

Manually modifying or deleting these sub-folders or their files could cause permanent loss of data or prevent the software from starting-up correctly.
The location of the data folder by default is the folder datadir within the application folder. However, you can change the path of the folder used as data folder in the file configuration.py within the application folder. This can also be used to have different users sharing the application. In order to do that:

1. Edit the file configuration.py
2. Find at the start of the file the dictionary called custom_user_data_paths.
3. Add a line within the dictionary brackets indicating your username in the operative system and the path of the folder you want to use as a data folder. Here is an example correctly formated for two users named Bob and Jennifer:

```
custom_user_data_paths = {
                          "Bob": r"/home/Bob/Documents/AoA-data",
                          "Jennifer": r"/NAS/Jennifer/ace-of-aircraft",
                         }
```

9.3. Third parties

Third parties are software independent from ACE of Aircraft that are needed to use some features. The path of each software must be properly set. There are four dependencies:

- AVL, needed to make vortex lattice computations.
- OpenFOAM, needed to make Navier Stokes computations.
- Ghostscript, needed to make image conversion.
- FreeCAD, needed to convert igs files to stl files

The paths of each software should be good by default. If needed you can force another path with the following way:

1. Edit the file configuration.py
2. Find near the start of the file the dictionary called default_settings.

3. Edit the line within the dictionary brackets starting by the software name. Here is an example correctly formatted to change the path of openFOAM:

```
default_settings = {
                    "gs_path": r"",
                    "avl_path": r"",
                    "of_path": r"/opt/OpenFOAM/of1912",
                    "FreeCAD_library_directory_path": r"",
                }
```

10. Hardware

Monitor the CPU, RAM and disk usages.
Add or remove computer(s) to the computing cluster.

10.1. Create a computing cluster

We do not recommend the use of computers with different processors, it could impact the results.

To add a computer to the computing cluster :

1. In ACE of Aircraft, go to the Hardware tab.
2. Click on the "Add a computer to the computing cluster" button.
3. Type manually the IP adress of the computer you want to add.

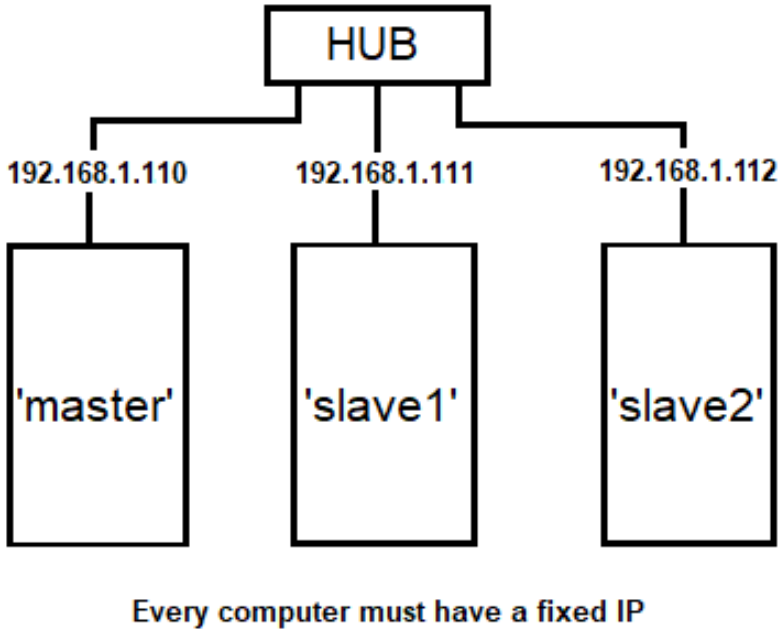
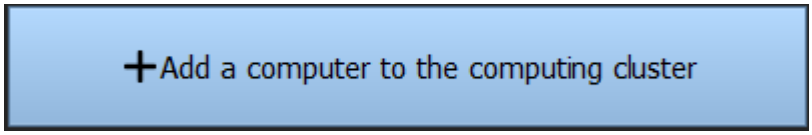


Figure 16: Representation of a computing cluster.

Your computer need to be connected to the same HUB and have a fixed IP adress. For more information about setting up the cluster, please refer to chapter 1.4.

10.2. Buttons



See above how to add a computer to the computing cluster.
You can add as many computers as you want.

You can change the number of cores you want your computer(s) to use to perform Navier-Stokes calculations.

To be efficient :

- do not use all your cores (keep at least 1 core for your GUI and other background processes).
- do not set a number higher than the cores you have.
- do not hyper-thread : it is preferable to use only 1 thread per core.

(The Draft and VLM project use 1 or 2 cores and do not need to be set to a certain number.)

11. Help

The help tab contains a user manual, tutorials, support and contact.
You can research word(s) in the research bar from the Help bar. It can be one word or an expression.
The result is composed of a colored square and the location(s) where your research was found:
A green square means the result is spotted in a title (in the manual).
A grey square means the result is spotted in a text (in the manual).

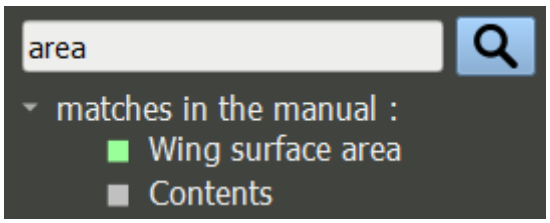


Figure 17: 'area' searched in the research bar.

Part III.
LEXICON

1. Airfoil lift coeff

The draft project needs values to establish the lift coefficient (CL) based on the angle of attack (AoA).
A .csv file is necessary to do this calculation as it defines lift values for a large range of angles of attack.

It contains the airfoil lift values for numerous angles of attack (when the airfoil is not stalled). You can find compatible .csv file at airfoiltools.com, or from Xfoil and XFLR5 softwares.

Here is an extract of a file from airfoiltools.com :

```
Xfoil polar. Reynolds number fixed. Mach  number fixed
Polar key,xf-naca23012-il-1000000
Airfoil,naca23012-il
Reynolds number,1000000
Ncrit,9
Mach,0
Max Cl/Cd,96.786
Max Cl/Cd alpha,8.75
Url,http://airfoiltools.com/polar/csv?polar=xf-naca23012-il-1000000

Alpha,Cl,Cd,Cdp,Cm,Top_Xtr,Bot_Xtr
-15.250,-1.0627,0.06220,0.05971,-0.0331,1.0000,0.0142
-15.000,-1.0952,0.05313,0.05040,-0.0395,1.0000,0.0142
-14.750,-1.1159,0.04706,0.04411,-0.0428,1.0000,0.0143
-14.500,-1.1291,0.04270,0.03955,-0.0442,1.0000,0.0144
-14.250,-1.1381,0.03932,0.03598,-0.0443,1.0000,0.0145
-14.000,-1.1426,0.03672,0.03321,-0.0434,1.0000,0.0146
-13.750,-1.1428,0.03474,0.03108,-0.0420,1.0000,0.0147
-13.500,-1.1396,0.03320,0.02940,-0.0402,1.0000,0.0148
-13.250,-1.1329,0.03211,0.02819,-0.0382,1.0000,0.0149
-13.000,-1.1469,0.02890,0.02470,-0.0341,1.0000,0.0153
```

Figure 18: Extract from a .csv file.

The software will use only the data highlighted in green and red.
The red numbers are interpreted as angles of attack and lift coefficients (in this order).

2. Cdmin / Cd0

The Cdmin is the minimum drag coefficient that you can obtain from an airfoil (or an aircraft).

The Cd0 is the drag coefficient that you obtain from an airfoil (or an aircraft) when it is set at an angle of attack producing zero lift.

The Cdmin and Cd0 can be equal if the airfoil is symmetrical or if a simple numerical model is used to compute the drag model from lift values.

In the draft project, the Oswald efficiency factor is used to model the induced drag coefficient. This model can only produce parabolic polars, which can be slightly inaccurate near the zero-lift point and stall point. In this case, Cdmin is equal to Cd0.

For actual wind tunnel measurements or accurate Navier-Stokes models of asymmetric airfoils, the polar obtained would be equivalent to the "actual drag polar" on the following figure. In this case Cd0 and Cdmin are close but not equal.

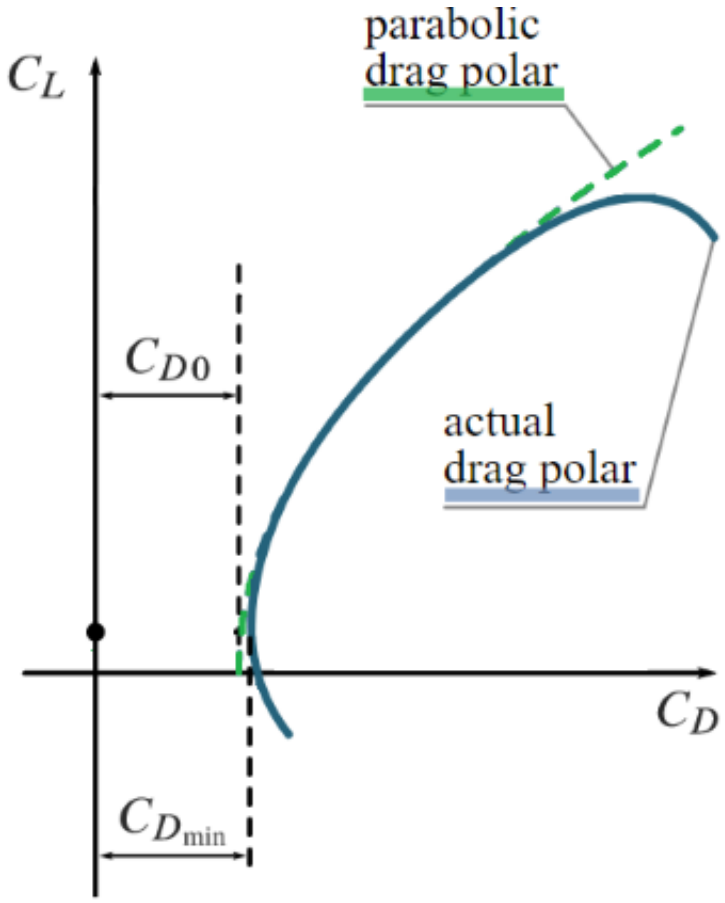


Figure 19: Parabolic drag and actual drag polar.

3. Custom measurements

Optional.

Custom measurements are defined in configurations. At the end of a run, additional values will be computed according to these custom measurments. To read these values, click the

export button on a run page or analysis page. This will generate a .csv file containing the custom measurments. There are 4 types of custom measurements : probes, moment readers, force slicers, torsor groups.

Probes are defined by a name and the XYZ coordinate of a point. This point must be placed outside of the plane to obtain results. The results will be the pressure and velocity of the airflow in that point. It can be used for example to check the pressure entering a inlet duct, or test potential position for pitot tubes or static pressure sensors.

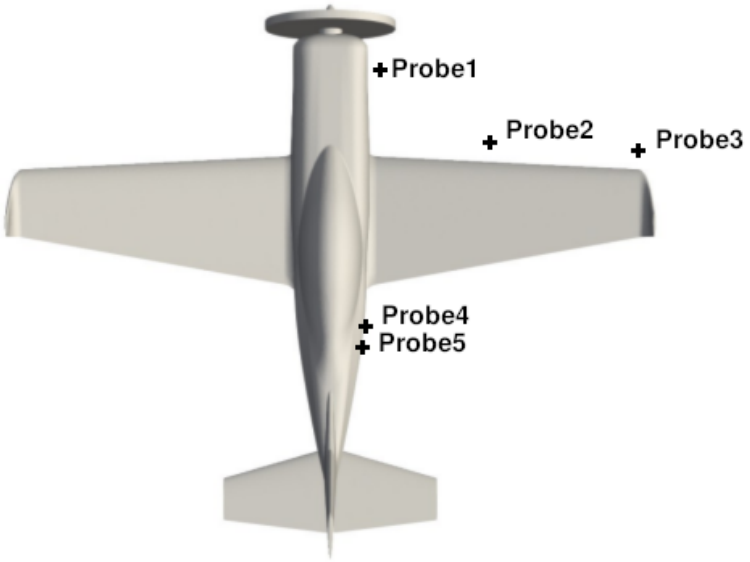


Figure 20: Example of probe points placed around the aircraft.

Moment readers are meant to compute the aerodynamic moment around a custom axis and for a custom part list. They can be used for example to evaluate the moment on an elevator or aileron axis. They are defined by :

- a name.
- a point and a direction (XYZ components), to define a rotation axis around which the moment is measured.
- the list of parts for which the aerodynamic moment computation is done.

Force slicers are meant to visualize the force distribution along an axis. It will virtually cut the parts selected into slices along the requested axis, and give torsor figures for each slices seperatly. It can be used to see the span-wise aerodynamic load of a wing for example. They are defined by :

- a name.
- a center point (XYZ coordinates).

- a direction (XYZ components).
- a number of slices to produce along the part length in the given direction.
- a list of parts. Only the aerodynamic forces on these parts are evaluated.

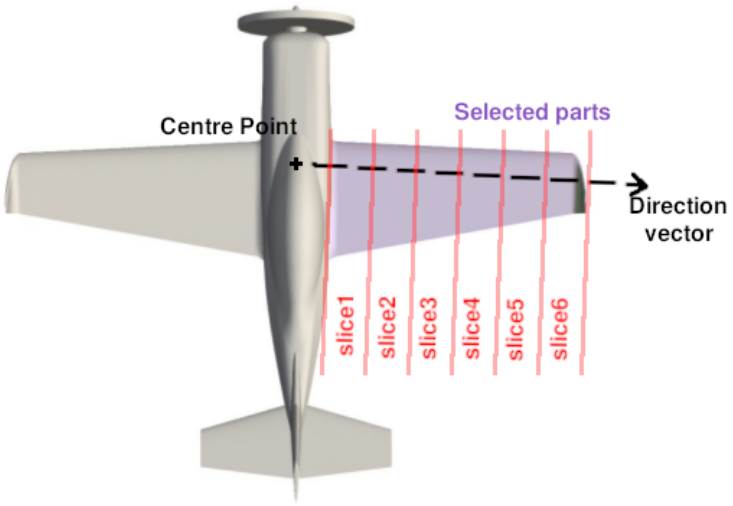


Figure 21: Example of slicer function used on the wing of an aircraft.

If you use this custom measurement options, several exports will be added to the results .csv file:

- Fx, Fy and Fz value of each slice. NB: in this file, x is the drag axis, z is the lift axis. These are not CAD axes.
- Mx, My, and Mz value of each slice. Measured around the center of gravity defined in the geometry page.
- Mc : moment measured around the axis defined with the point and direction given to define the slicer function.

Torsor groups are meant to group the torsor results of multiple parts. For example if a wing is split in several parts : primary, flap, aileron, etc. , this function will sum up the torsors of all these parts and produce a global torsor for the wing. They are defined by a name and a list of parts.

4. Detail size

It is recommended to take half of the trailing edge as detail size value.

Indications on the value of the detail size :
Be careful to choose a value neither too big nor too small.
- A value too big will give bad quality meshes, and unreliable solving (see examples below).
- A value too small will take a lot of RAM usage and a lot of computation time to get results (see the curves below).

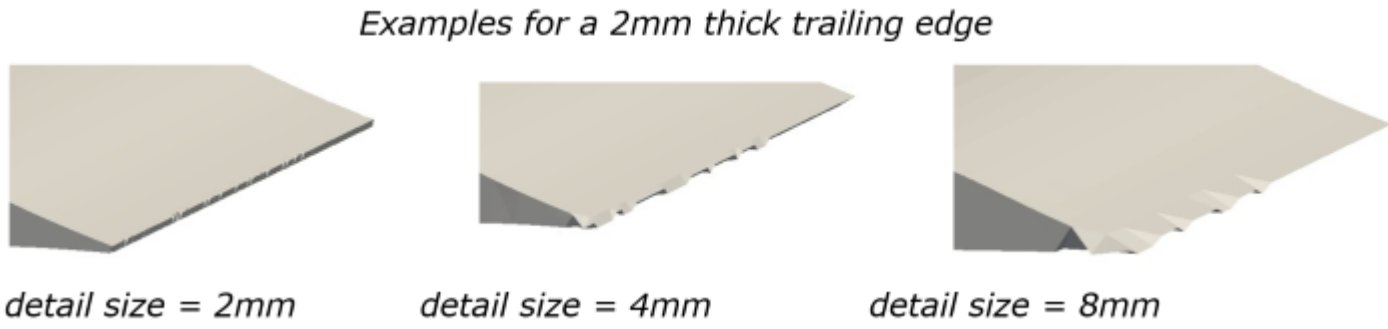


Figure 22: Different detail sizes resulting in various levels of precision.

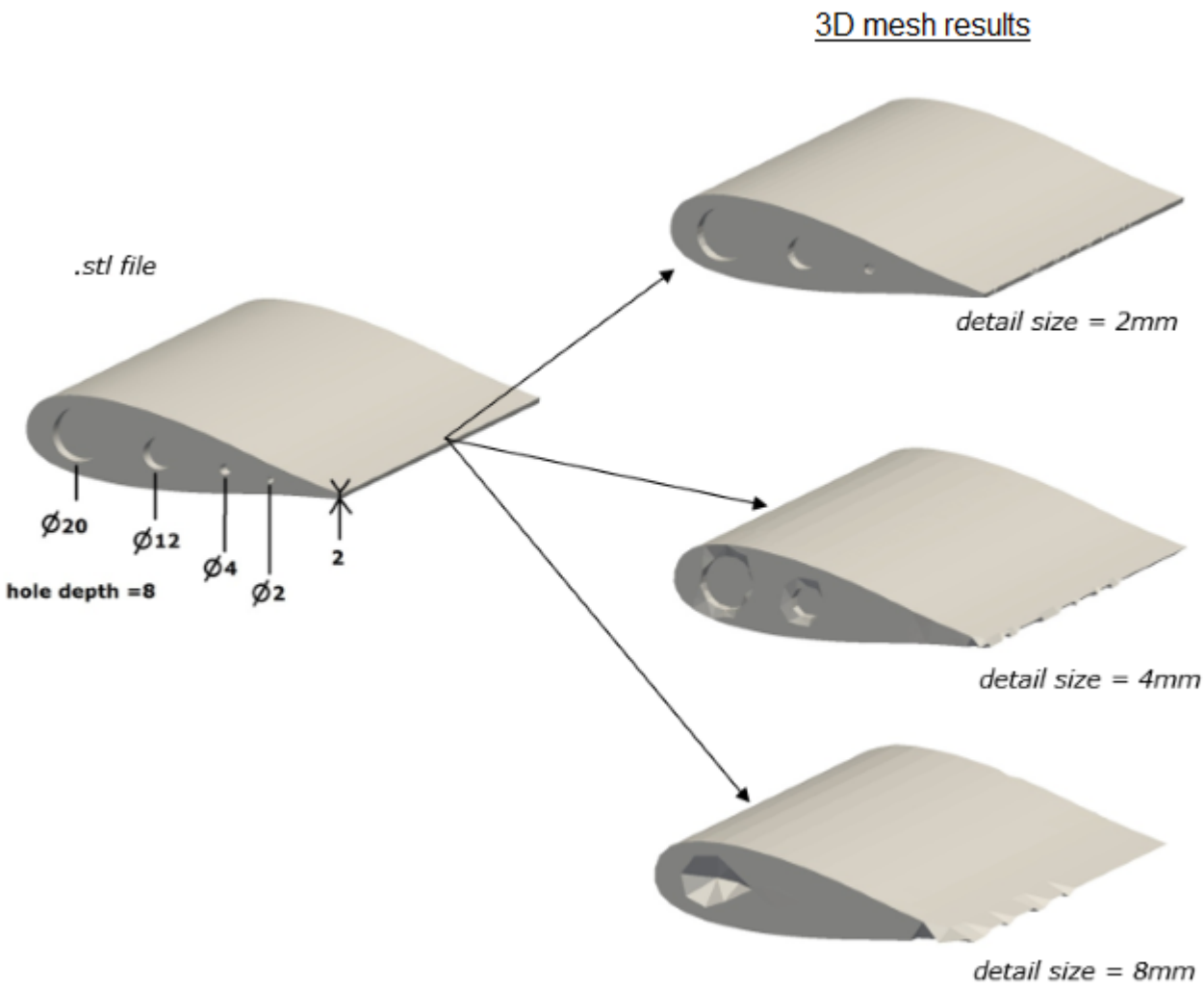


Figure 23: Representation of different detail sizes influencing the precision of the refinement.

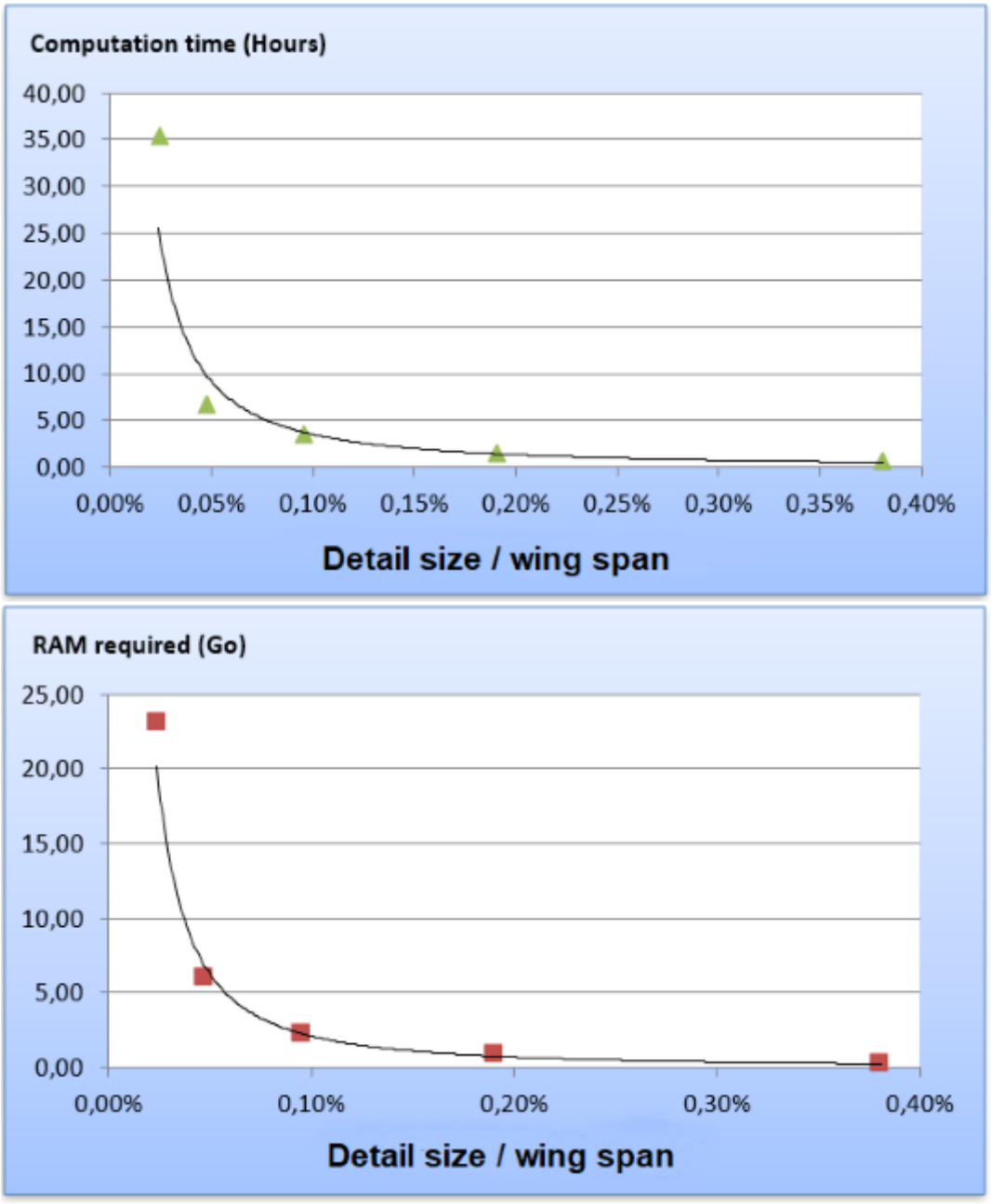


Figure 24: Computation time and RAM usage depending on the detail size over wing span ratio.

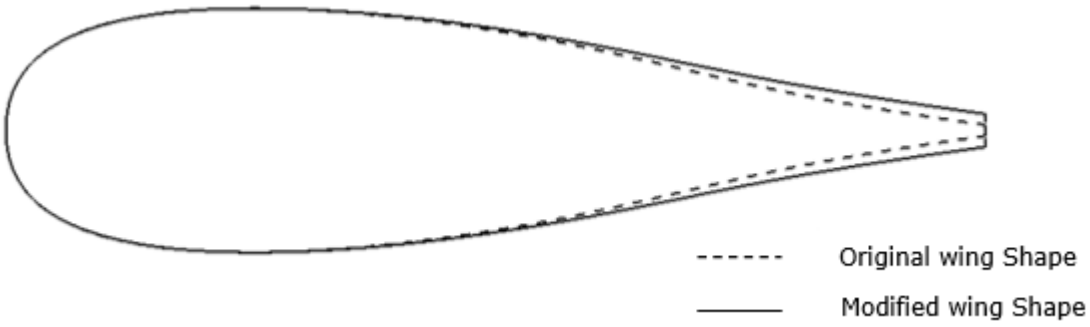


Figure 25: 1mm thick trailing edge modified to accept a 2mm detail size.

To reduce the computation time, it is possible to model the trailing edge a bit thicker than it really is, as shown in the previous image. This allows you to define a higher detail size value, and to reduce the solving time, while the effect on the drag coefficient is often neglectable.

To obtain additional details, see the part V. Correlations.

5. Engine-propeller unit efficiency

When setting a power model, the user must set a global efficiency for the engine-propeller unit, this includes power losses due to:

- engine accessories power draw
- gear reduction and transmission efficiency
- propeller efficiency

Generally the global efficicency factor is around 75-80% This means that with an engine of 100 HP rated power output, 75 to 80 HP will actually go to final effective thrust.

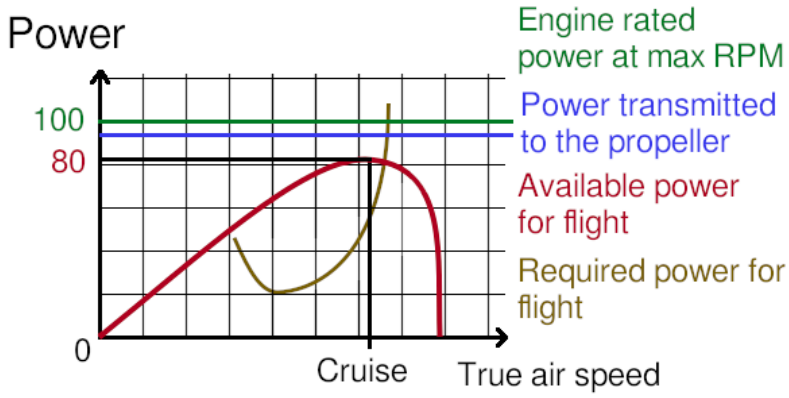


Figure 26: Engine-Propeller unit efficiency with respect to air velocity.

6. Engine power loss relative to altitude

The power models have a feature representing the engine power output being scaled down when the aircraft gains altitude.

This power loss is considered to be linear in the model. During the model creation, the user must specify how many HP is lost for a 1000 feet altitude gain. The power loss is then considered linear to this rule, for any altitude.

For a thermal engine, the power loss is generally 3% per thousand feet. (For a 100 HP engine, you would set the parameter to 3 HP)
For an electrical engine, you may set the parameter to 0 HP, as they don't breath air.

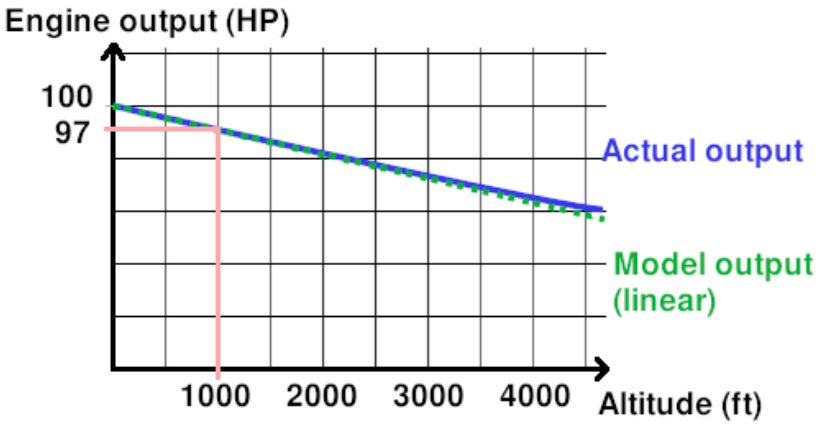


Figure 27: Modeling of the engine power output relative to the altitude for a 100 HP rated engine, with a power loss parameter of 3 HP per thousand feet.

7. Flying altitude

The software uses the altitude given by the user to find an equivalent of air density and/or air viscosity which will be used in calculations.

This is done by extrapolating values from the International Standard Atmosphere (ISA).
The ISA is a standard reference for Earth air pressure, temperature, viscosity and density values at various altitudes.

8. Custom refinements

Optional.

If you do not change this parameter, the detail size will be automatically adapted on the surface curvature (variation from detail size to detail size x 4).

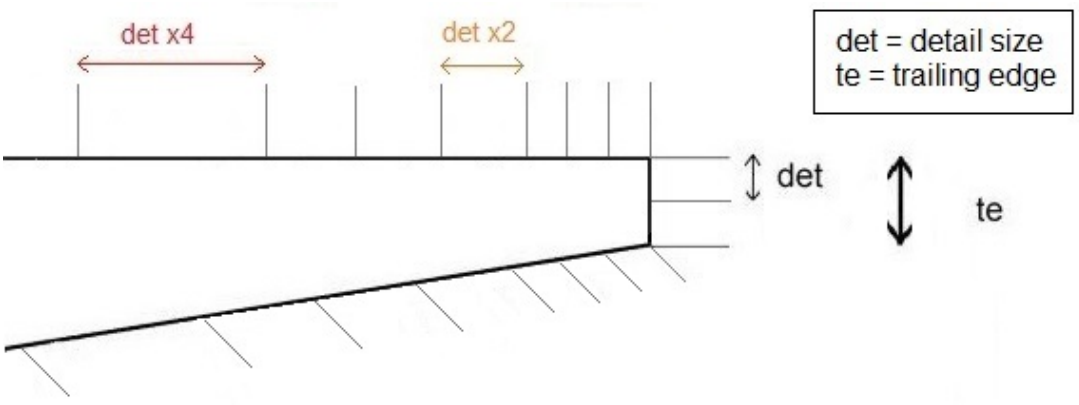


Figure 28: Detail size automatically adaptated on an airtight surface.

However you can use custom refinements options to mesh a part with a mesh finer than the detail size. If you want to establish a specific meshing size parameter :

1. Select the airtight surface you want to refine.
2. Select the refinement size parameter you want to apply.

Be careful, this refinements will likely increase the quantity of cells in the mesh, which means the computation will become slower and require more RAM.

9. Parts movements and command angles

This chapter describes how the motion of one or multiple parts can be modeled for a NS project/geometry/configuration. A geometry always has a list of fixed parts that are reference, in other word these parts are in a neutral position, have zero deflection or deformation.

During geometry creation or edition, part movements can be defined in two ways. The first way is to define a rotation of the part, around an axis, and this rotation will be bound to a certain command axis.
The second way is called "shifting". You need to select the fixed part you want to change in the drop down menu.
Import an .igs or .stl file with the part in the new position you want, or deformed. Then, specify the command angle parameters to indicate what this new position corresponds to.
This can also be very usefull to fill-up gaps when a part is retracted.

Example :
Assume you have various a couple of fixed parts defined in your geometry, including a 'flap' part, built inside a primary wing airfoil.

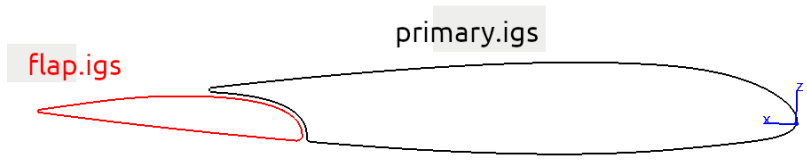


Figure 29: Fixed parts.

The flap part has a rotation definition around an axis, and is bound to the flaps command axis.

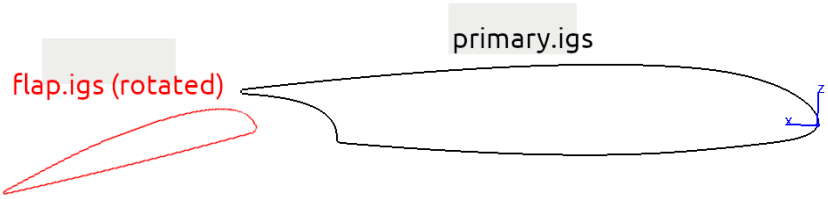


Figure 30: Possible rotation of the flap.

This means that if in the configuration a command angle is set to the flap, flap.igs will be rotated accordingly.

The problem is that when the flap is not rotated, the gap between it and the primary is very small, which can cause problem to the mesher. We can define an alternative version of the flap part that will fill-up this gap when the command angle is requesting a retracted flap. To do that a 'shifting' definition is made and bound to the following command angles :

- : pitch command: no angle specified
- : yaw command: no angle specified
- : roll command: no angle specified
- : flaps command: 0°
- : airbrake command: no angle specified

With this definition, everytime a flap command angle is requested, flap.igs will be rotated and used. When the command angle is zero, flap-bump.igs will replace flap.igs.

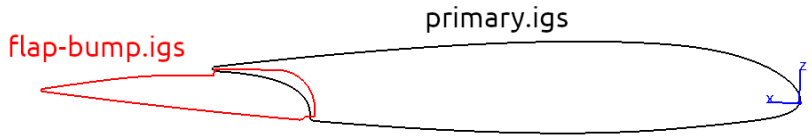


Figure 31: Shifting flap.igs to flap-bump.igs.

In cases where the airflow through the slot gap is neglectable, the flap can be modeled with a simple rotation and no shifting parts. But the primary should have modified shape to make sure no gaps appear, as shown in the following image.

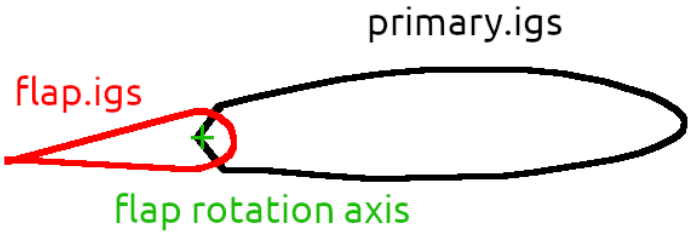


Figure 32: Simple flap rotation geometry.

10. Heat exchangers

When defining a configuration for a geometry of an NS project, with the correct license level you may have defined parts for heat exchange.

The heat exchanger models are porous volume

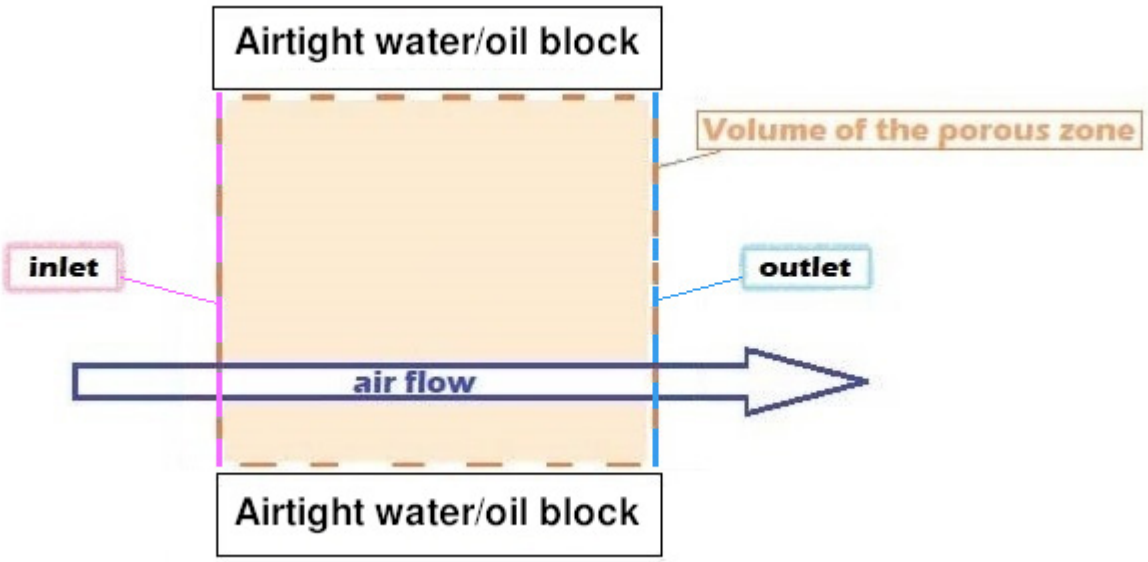


Figure 33: Air flow through an exchanger.

11. Hole detection points

The sum of your parts should create a closed volume, the software will verify if there is any hole left.

Write the coordinates of a point located inside the volume created by the parts you imported. You can add multiple coordinates if you have mutiple volumes. It is better to test the impermeability of every volume by adding a coordinate for each part in your geometry.

The software will search for a path between the point(s) you gave and a point in the farfield. If the part is impermeable, the software will not find a path.

Here is an example with a problematic situation, the flap is not closed :

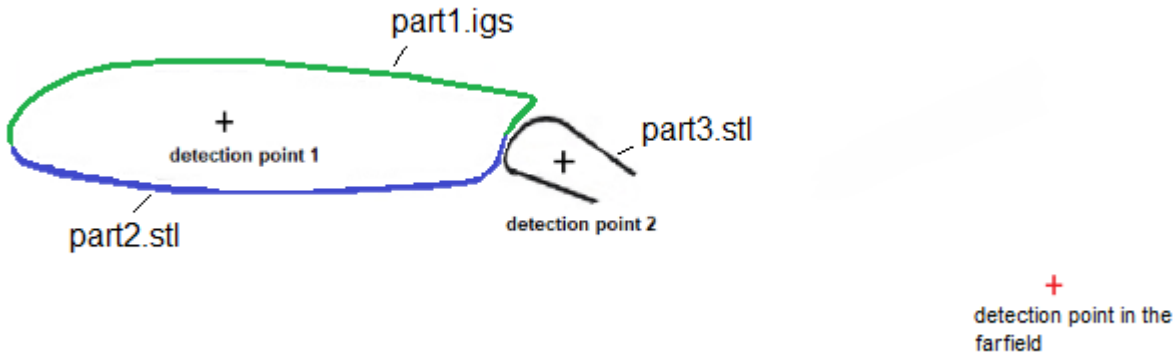


Figure 34: Example of an open volume.

The software will search for a path from detection point 1 and 2 to the detection point in the far field :
→ The primary wing is impermeable, the software will not find a path.
→ The elevator is open, the software will find a path and an error message will appear.

If you start the calculation with hole(s) in your parts, the software will try to compute an air flow inside the hollow part(s) which is unnecessary : it will take a lot of time and the results will be unreliable.

12. Mass and inertia

When defining a configuration for a geometry of a VLM project, you may define a list of mass and inertia values.

This definition will not be used in most analysis, but is required to perform dynamic stability analysis.

Provide a list of the elements inside your airplane. Define their mass and inertia characteristics.The list provided can be precise or global:

- You can separate each element and write one element per line to be more precise.
- You can write the global mass value of several elements or the whole aircraft in one line.

13. Mean aero chord length

- For 2D airfoil : the reference length used for the calculation of aerodynamic coefficients is generally **the chord of the airfoil.**

- For 3D wing (or entire aircraft) : the chord length can be different depending on where it is measured along the span. A representative chord value must be found from the geometric definition of the planform, this reference value is **the mean aerodynamic chord**.

The calculation you need to do to obtain the mean aerodynamic chord will depend on the shape of the wing.
You can see on the next figure that the mean aerodynamic chord is different for each shape of planform.

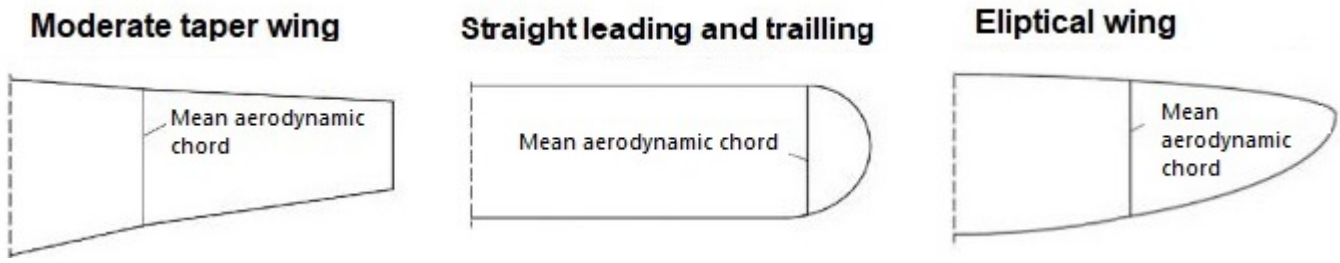


Figure 35: Different planforms and their mean aerodynamic chords.

14. Oswald efficiency factor

The Oswald efficiency factor depends on the lift distribution along the span.
It also depends on the twist of the wing, the fuselage and all appendixes added on the wing (winglet, landing gear or engine motor).

$$CD = Cdmin + \frac{CL^2}{Pi * e * S/c^2}$$

Figure 36: Oswald equation.
e is the Oswald efficiency factor.

This equation uses the Oswald efficiency factor to establish drag values from the lift values.
The lift values are obtained from the .csv file provided.
The number obtained will change based on the size and the shape of the wing.
→ Rectangular wings will have an e coefficient around 0.7.
→ A perfect elliptical wing will have an e coefficient of 1.

Here are a few examples for a few aircraft :

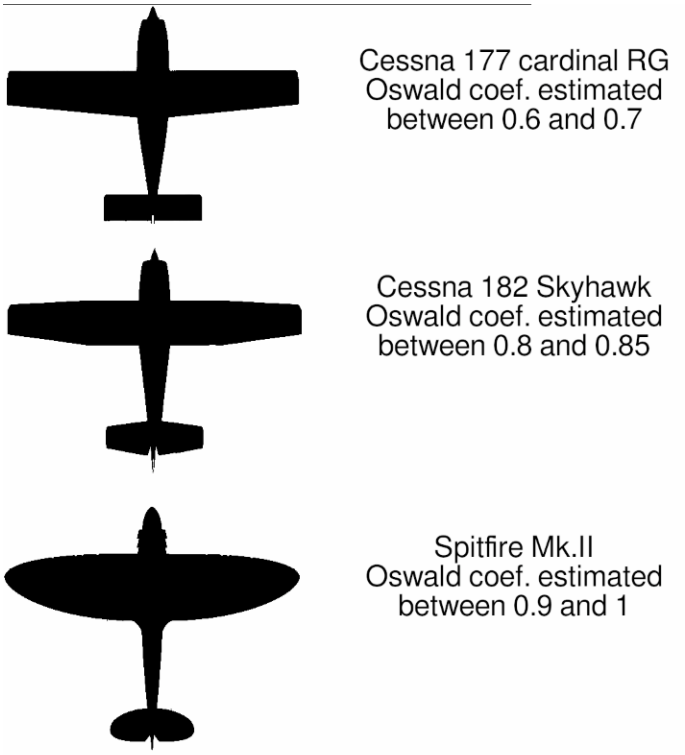


Figure 37: Oswald efficiency factor for different shapes.

The following paper is a good read on how to estimate the Oswald efficiency factor of an aircraft :
"Estimating the Oswald factor from basic aircraft geometrical parameters" by M.Nita and D.Scholz, Hamburg University of Applied Sciences, DocumentID 281424.
You can find the complete report in the References compressed folder provided.

15. Position of the center of gravity (X Y Z)

You must define the center of gravity (CoG) precisely to obtain correct moment values without re-calculations.

The center of gravity (also called center of mass) is used as a position reference point. It is used to calculate the aerodynamic moments (pitching, yawing and rolling moments).

This figure shows the CoG of an airplane :

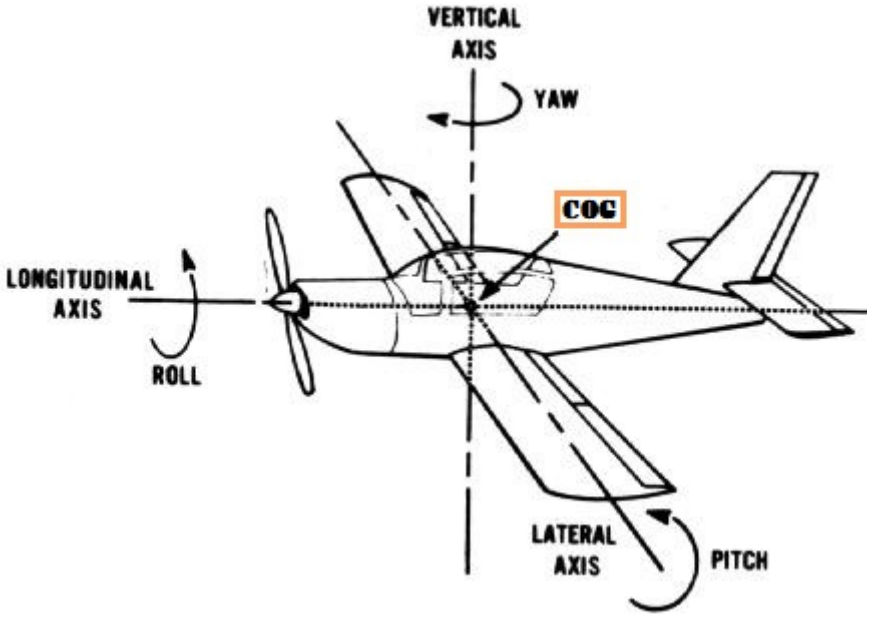


Figure 38: Center of gravity and axis of an airplane

16. Rolling / Pitching / Yawing moment with modified Center of Gravity positions

Visualisation of moment curves for various user-defined CoG.

→ CoG 1 is part of the geometry definition and cannot be modified.
→ You may input coordinates for other CoGs (2, 3 and 4) and visualize the impact of a CoG displacement on the moment curves.

In the aerodynamic type analysis, a suggestion of X coordinate for the aerodynamic center is displayed. This is generally the aft limit of the acceptable CoG range.
The software computes this value by searching an X coordinate nullifying the pitching moment for an AoA varying between –1° and +1°.

For the forward longitudinal stability limit analysis, a suggestion of X coordinate for the forward limit of the CoG range is given. It is the coordinates resulting in a null pitching moment at the reference AoA of the analysis (it should be the stall angle).

17. Rotors

When defining a configuration for a geometry of an NS project, with the correct license level you may have defined parts for modeling rotors.

A rotor volume represent the volume swept by the propeller blades, it should have the shape of a cylinder, or disc. For a good meshing, we recommend to make the thickness of this disc approximately 15 or 20 times the project's detail-size parameter.

The orientation of the geometry given is detected automatically. The software will always set the flow direction to the X positive direction when the orientation is X dominant. When the orientation is mainly Z dominant, it wil direct the airflow to the Z negative direction.

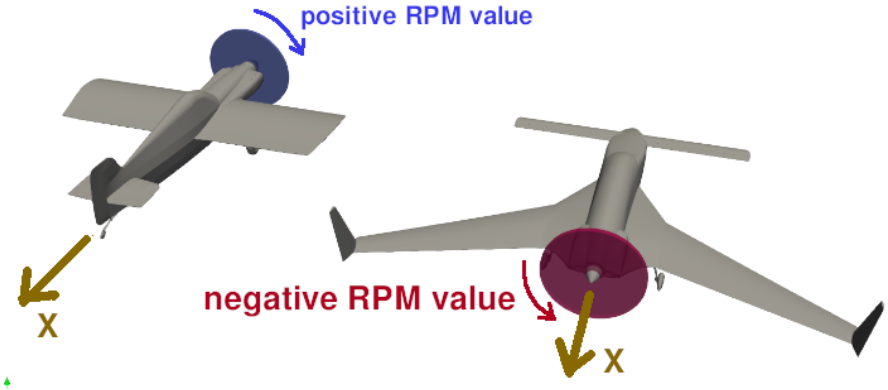


Figure 39: Rotor direction when main orientation is X.

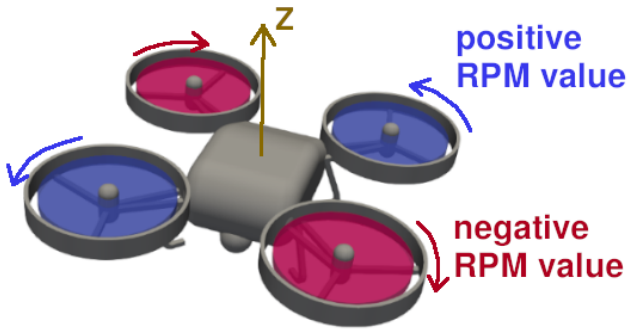


Figure 40: Rotor direction when main orientation is Z.

When setting up a configuration, you will have to set the RPM speed of the propeller. Note that if you set a positive value, the rotor will turn clockwise. If the value given is negative, it will rotate counter-clockwise (This is when observing the rotor from the downstream position).

18. Template NS

The Navier-Stokes template contains a list of instructions sent to OpenFoam (a third-party software). It is an automation script containing a list of commands.

Always use the latest available version of the template.
It could impact the results of the calculation.

19. Velocity for fixed pitch propeller

When setting up a power model with a fixed pitch propeller, the user must indicate the air velocity (TAS) for which the propeller is the most effective. In reality this is determined by

the blade pitch angle, which will correspond to an ideal advance ratio of the propeller.

For most aircraft, the ideal advance ratio of the propeller is chosen so that, when the engine runs at the reference RPM speed, the air speed is the cruise speed of the aircraft.

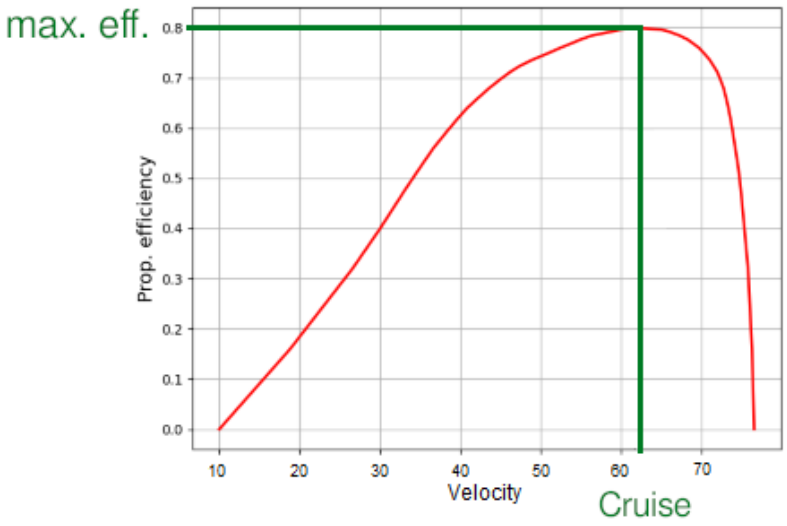


Figure 41: Propeller efficiency with respect to velocity.

20. Velocity for constant speed propeller

When setting up a power model with a constant-speed propeller, the user must indicate a couple of air velocity values (TAS) :

- **Min. velocity** : The true air speed resulting from the smallest blade pitch angle when at maximum propeller efficiency.
- **Max. velocity** : The true air speed resulting from the largest blade pitch angle when at maximum propeller efficiency.

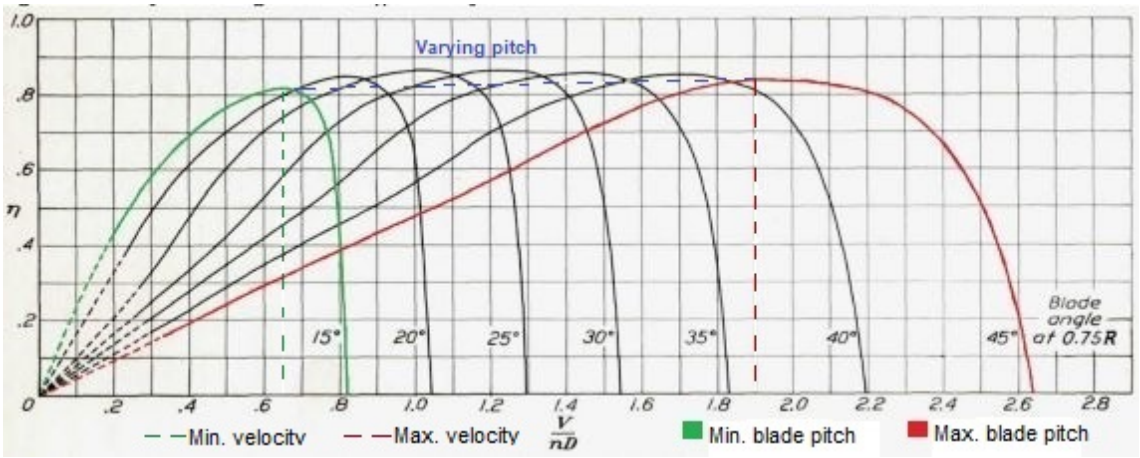


Figure 42: Propeller efficiency with respect to advance ratio for various blade pitch angles.

Generally, the max. velocity parameter can be set at the aircraft cruise speed. And the min. velocity can be set a bit above the stall speed.

21. Wing surface area

The wing surface area is the surface area of the wing including the fuselage surface between the wings.
Be careful to extend the wing shape in the fuselage as the figure shows:

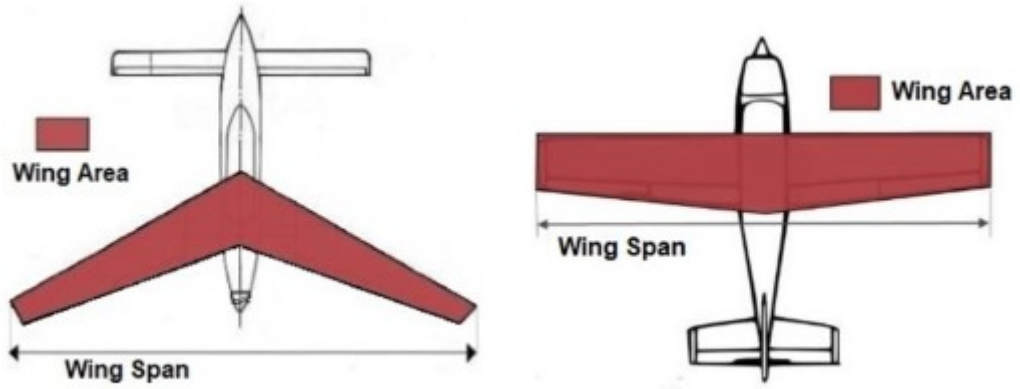


Figure 43: Different wing areas on various wing shapes.

The wing surface area is used as a reference in the aerodynamic coefficients calculations.

Part IV.
MODELING BEST PRACTICES

1. Avoiding bad comparisons

Avoid the comparison of different calculation results if the calculation process is not the same.

For example, we will try to compare the results of two wing profiles :
– The first wing profile was studied with a detail size parameter of 0.5mm for a 1m chord.
– The second wing profile was studied with a detail size parameter of 4mm for a 1m chord.

It is impossible to know if the differences between the results are due to the wing profiles or the *detail size / chord* ratio.
However, we will see below that this ratio has a huge impact on the results.

A difference in *detail size / chord* ratio creates large discrepancies between the results, making them incomparable. It is better to have the same ratio when making a comparison.

To illustrate, we are going to compare the detail sizes 0.5 and 4 for a 15° AoA on a single wing profile :

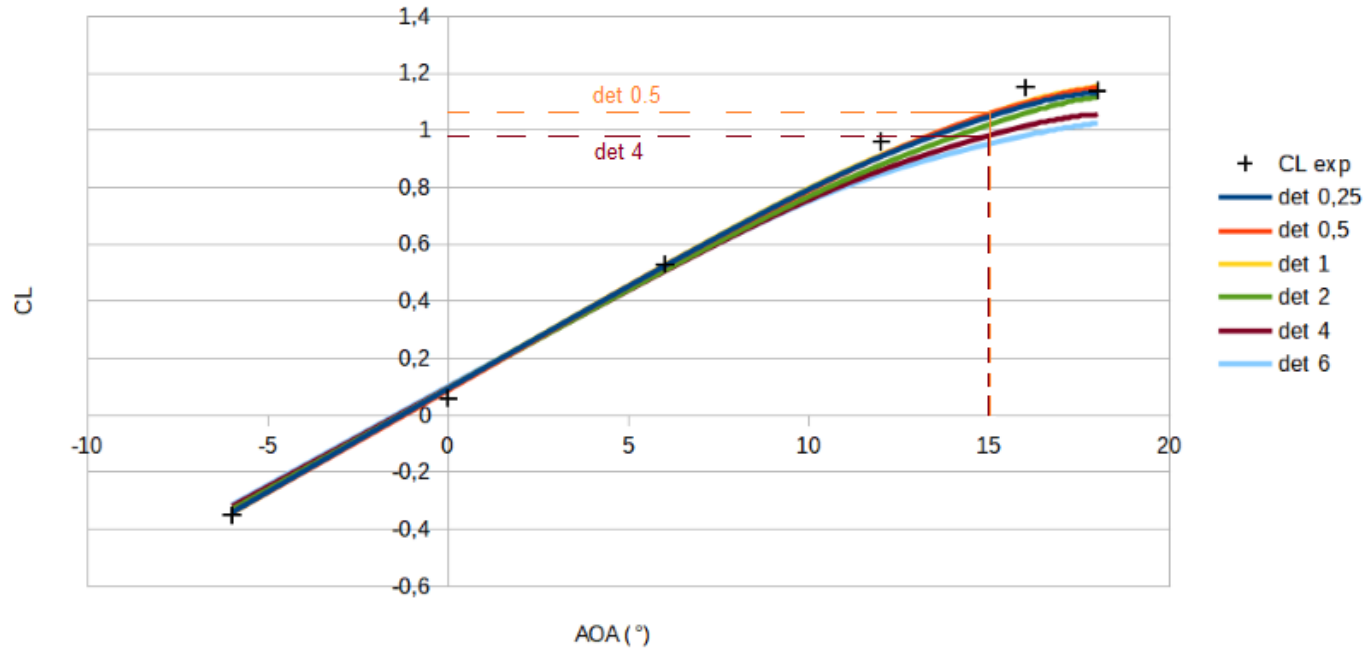


Figure 44: CL according to AoA (different detail sizes).

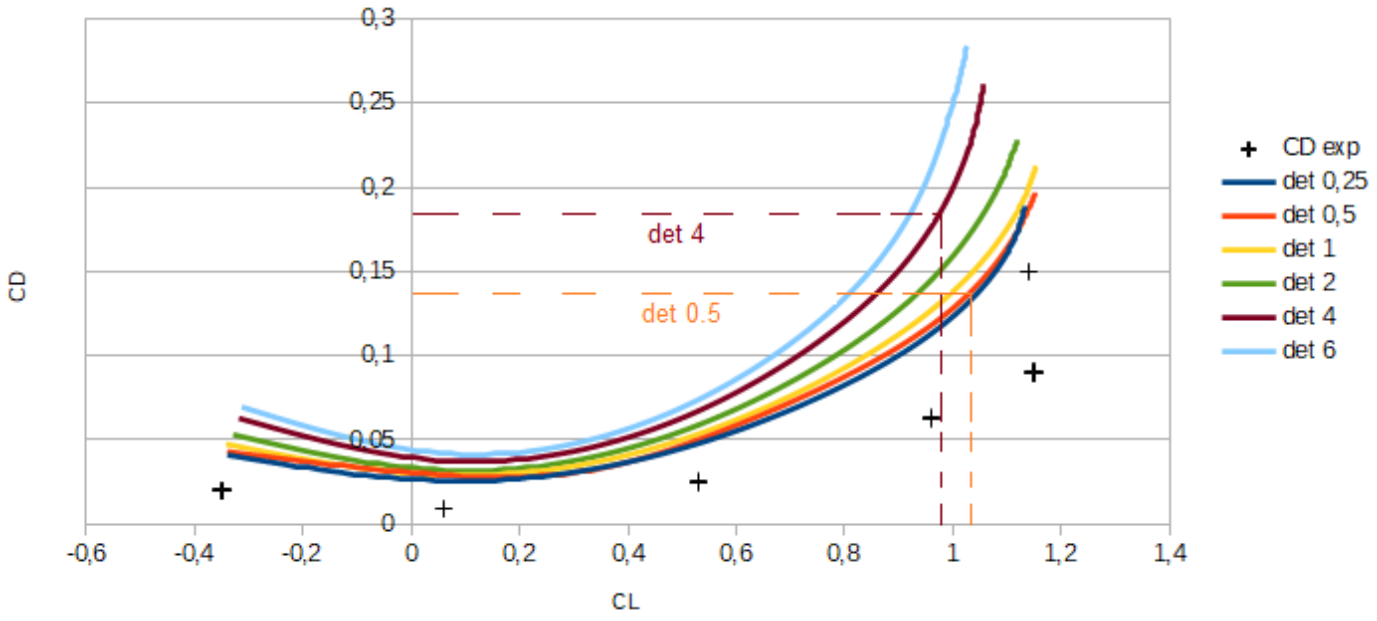


Figure 45: CD according to CL (different detail sizes).

The lift coefficient is higher for the detail size 0.5.
The drag coefficient is higher for the detail size 4.

As you can see, changing one parameter is enough to create a great disparity in results.

Therefore, during your analysis, it is recommended not to change more than one thing at a time in order to always identify clearly the cause of each consequence. Also keep in mind that each part of the geometry (may it be in the front or at the rear of the aircraft) can have a strong influence on the flow both upstream and downstream. To give you an example : a modification on the empennage can slightly change the airflow at the front of the plane.

2. Preparation of CAD for NS geometries

2.1. Cutting the geometric modeling in several parts

Cutting the surfaces of the plane in several parts allows you to evaluate the forces applied on each part.

Be careful: the sum of the parts must be a closed volume. Create a closed volume containing only .igs or .stl files :

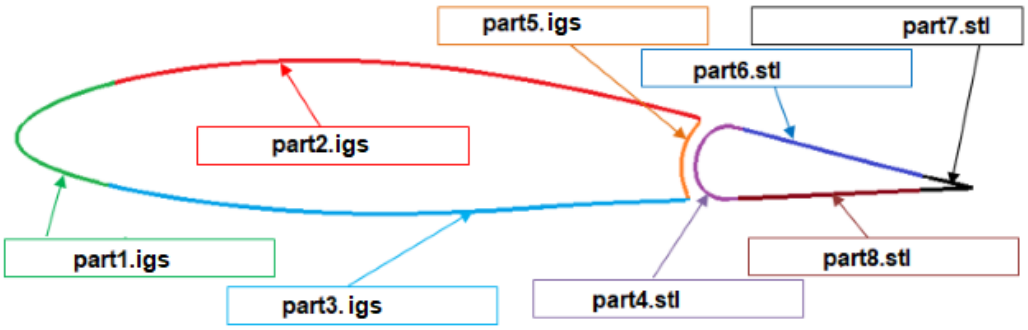


Figure 46: Closed volume composed of mutliple parts.

The hole detection points will verify if there are any holes left in the volume. Cf. chapter 11. To see the results, create a configuration : it will generate pictures in the Projects tab. The pictures named "STLholeDetection" indicate if there are holes. If there is a hole, a red highlight indicates its position.

NS2D exception :
It is expected for the NS2D to provide an extruded profile (the extrusion must be at least 2 chords long). The holes naturally present on both sides of the extrusion are acceptable.

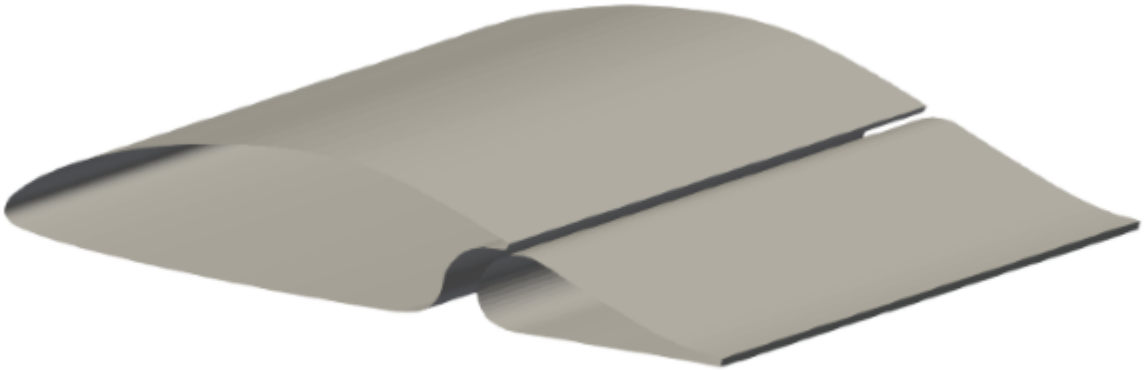


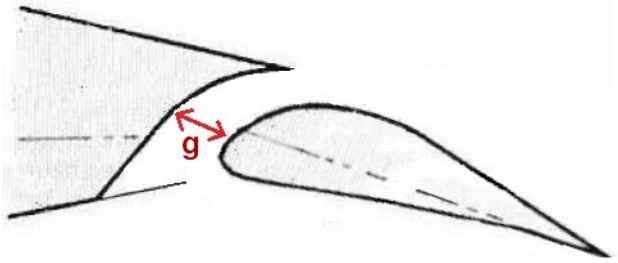
Figure 47: Extruded profile used for NS2D.

To obtain more details, see 11. Hole detection points

2.2. Refinement of the adjacent and the detailed zones

Refinement of the adjacent zones :
Refining adjacent zones increases computation time.
If no air flow is expected between your foils, you can fill the gap and avoid the calculation.

→ Refining the gap



In this example, g=1 and the detail size is 1.
We must refine the surfaces within the interstice with a custom refinement choosing *detail size / 4*, to insure that the gap is meshed properly.
Select the suitable ratio *detail size / n* for your geometry.

→ Filling the gap
If no airflow is expected between the foils, you can fill the gap between the parts. It will avoid long computation and inaccurate results due to a bad meshing.

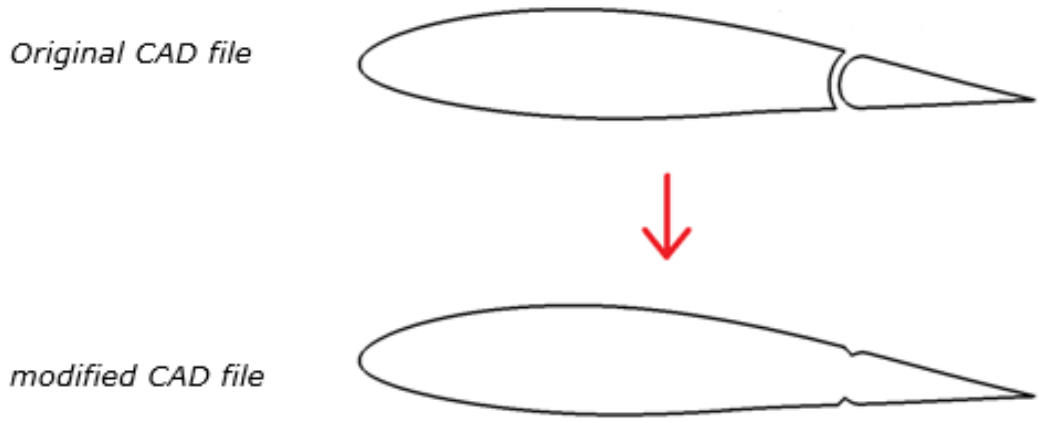


Figure 48: Modification of a CAD file to fill the distance between the parts.

Refinement of the detailed shapes :
Do not overdo it: the more refined, the longer the computation time.
Select the suitable ratio *detail size / n* for your geometry.

Smaller refinement sizes give more accurate results than bigger ones :

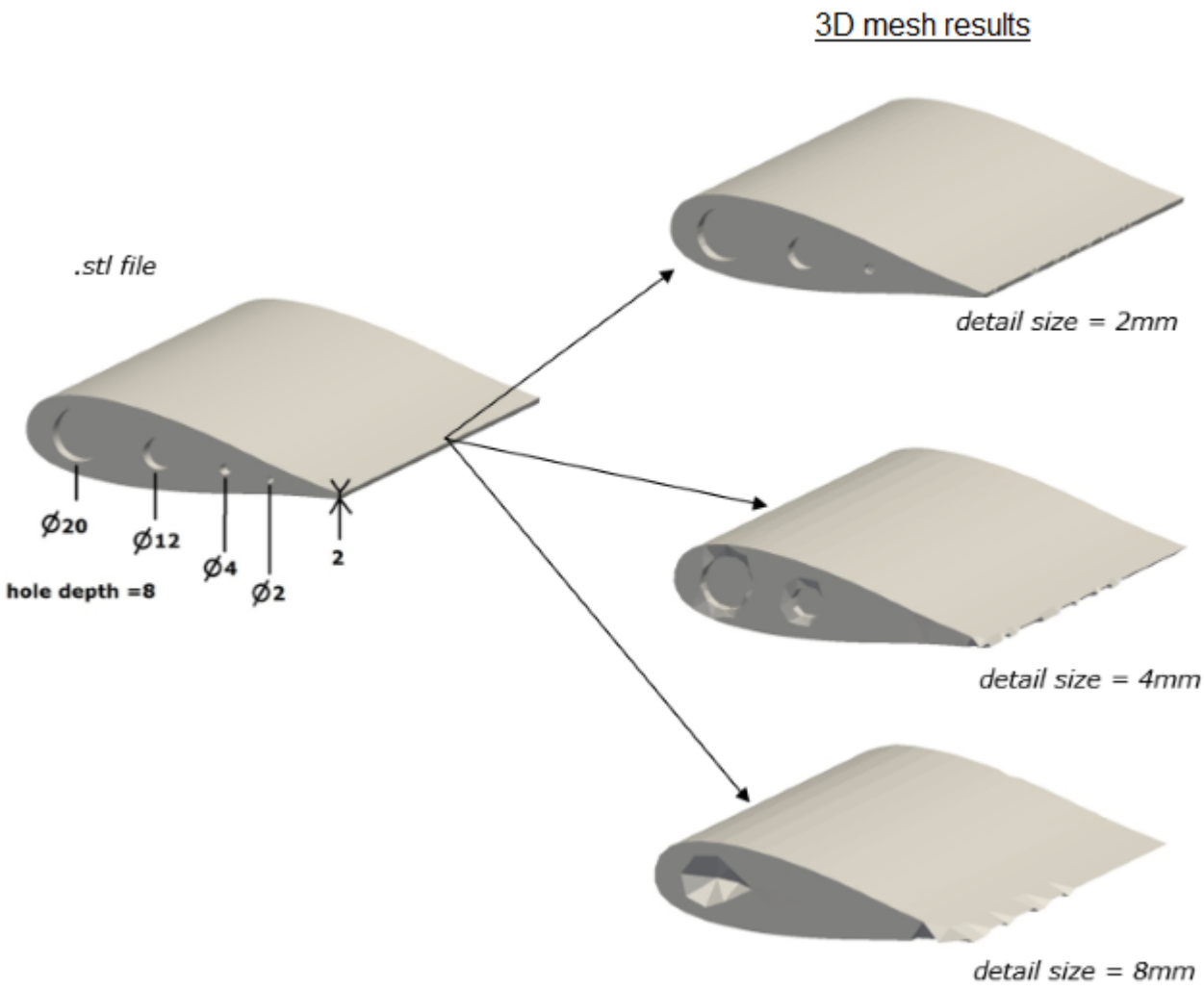


Figure 49: Representation of different detail sizes on a STL file.

When the detail size is too large, the meshed shapes can become inaccurate. The quality of the cells in the mesh can also become bad means runs become unreliable (randomly failing, diverging, giving inaccurate results).

Generally, it is best to use a detail size that is half the thickness of the smallest trailing edge in the CAD, and to modify the CAD to remove all details that would not be meshed well given this size. In the preceding image, the trailing edge thickness is 2mm, so the ideal detail size would be 1 mm. All the features that are less a couple of millimeters should be removed or edited in the CAD. If this is not possible, custom refinements can be used, see 8. Custom refinement

In case of cylindrical shapes, make sure the refinement provides enough sides (minimum 8 sides per circle) to constitute a almost cylindrical volume.

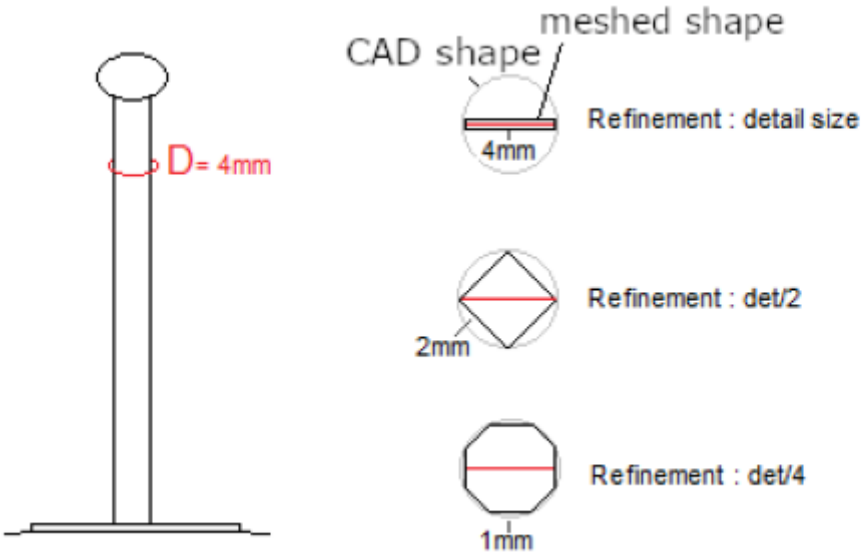


Figure 50: Aircraft antenna.

For more information, see 4. Detail size

2.3. Surface angles

Acute angles :
If you have any acute angles in your parts, the software could mesh the angle incorrectly. Edit the part to avoid it.

The following example shows a way to avoid an acute angle and get a mesh of better quality. The mesh must have at least 2 cells on the modified (red) surface.

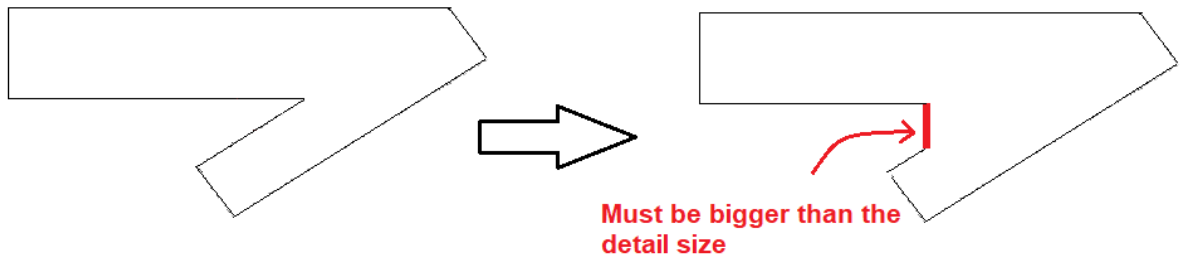
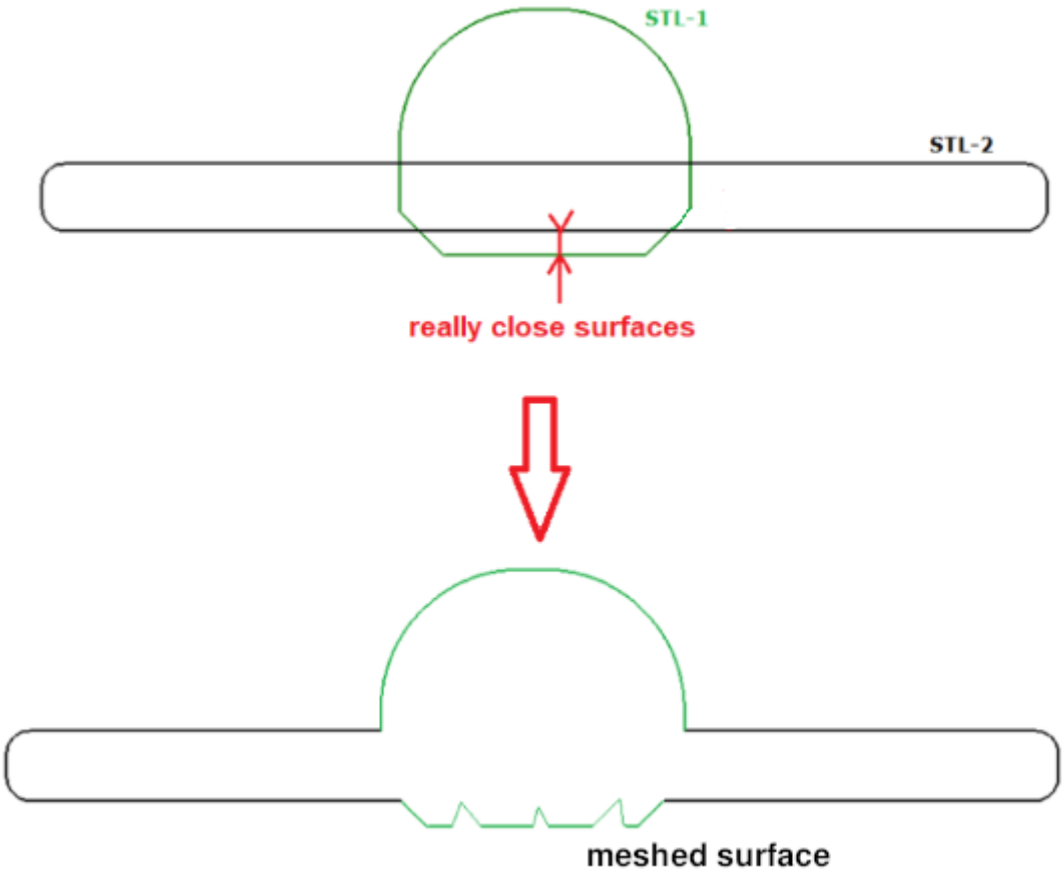


Figure 51: Modeling the part differently to avoid problems with meshing.

To avoid bad meshing, avoid angles below 90°.

Bad meshing near adjacent surfaces :
When two surfaces are close to each other, it is better to define only one surface for both in order to avoid bad meshing.

This example shows the mesh you can obtain with two closely spaced surfaces :



To prevent problems, avoid overlapping parts as much as possible.

3. STL Generation

Any .igs file imported into the software will be automatically converted into a .stl file by a script.
This automation is fallible and can create problems after the conversion of the part(s).

If your .isg file contains a few simple parts, you can proceed with it : it will be converted into a .stl file automatically.
If your .isg file contains numerous and complex parts, create and import a .stl file directly.

A high-quality .stl file :

- has its points coordinates close to the surface definition of the CAD.
- possesses enough coordinates to define precisely the surface.
- considers the placement of the sharp edges.

Exchanger surfaces and flow report surface :
Provide a well **refined** .stl file. *It is not recommended to use an* .igs *file.*

If your file is not refined enough, the mesh might be of bad quality and the computation will give inaccurate results.
The post-processing of the flow depends on the refinement of your .stl file because the interpolation is done on the meshing of the file.

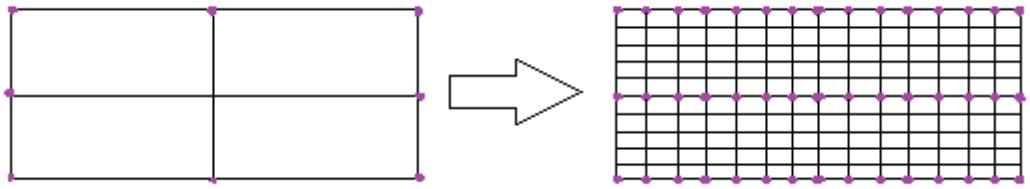


Figure 52: Refining part(s) avoids problems with the interpolation of results on the .stl mesh.

Be cautious when you create an exchanger :
The inlet and outlet must be parallel and flat.
The exchanger must be a closed volume.

4. Exchanger heat exchange and pressure loss curves

When you create an exchanger model, the software asks for a list of the exchanger operating points, that is, the difference of pressure between the inlet and the outlet at different velocities going through the exchanger.
This list of values determines a law of pressure loss and heat exchange depending of the airflow speed.

It is better to provide precise values for the largest velocity range possible. Add as many values as you want by clicking on the "+" button.

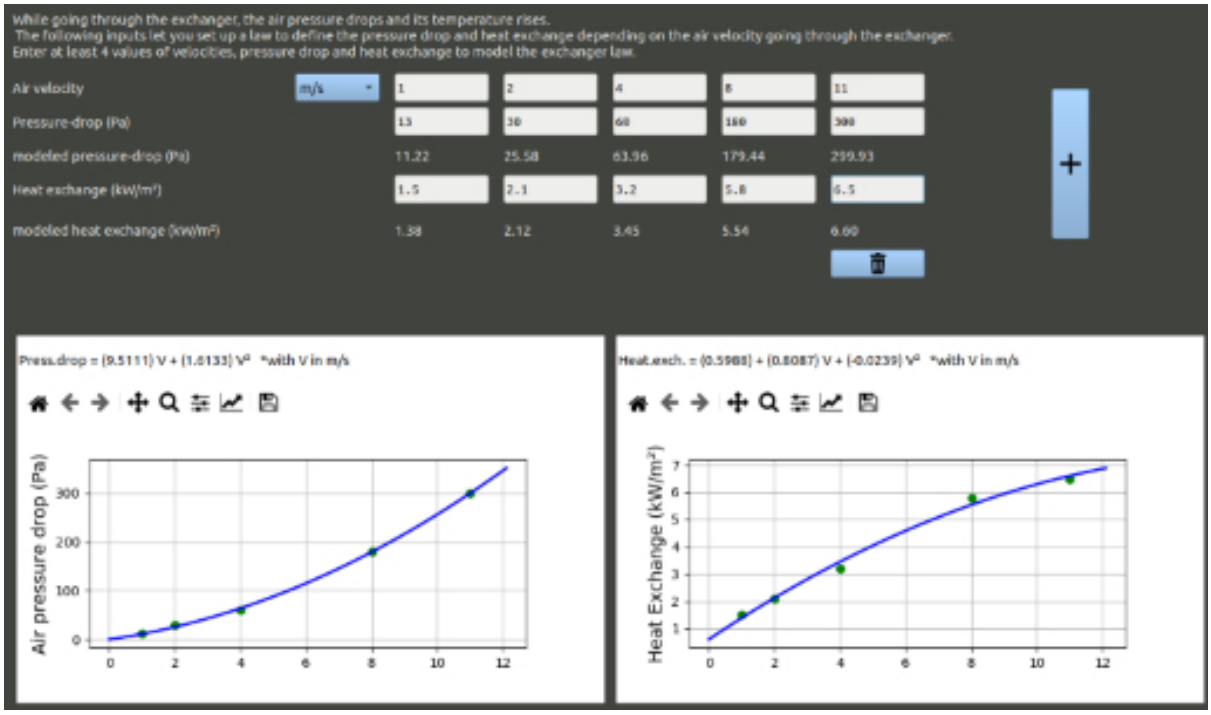


Figure 53: Software screenshot of the air pressure drop and heat exchanger curves.

→ To obtain the pressure loss data, contact the manufacturer or search the Internet for available data on your exchanger (make sure it is the same reference as yours). You may also measure the inlet and outlet pressures with a pressure gauge yourself :

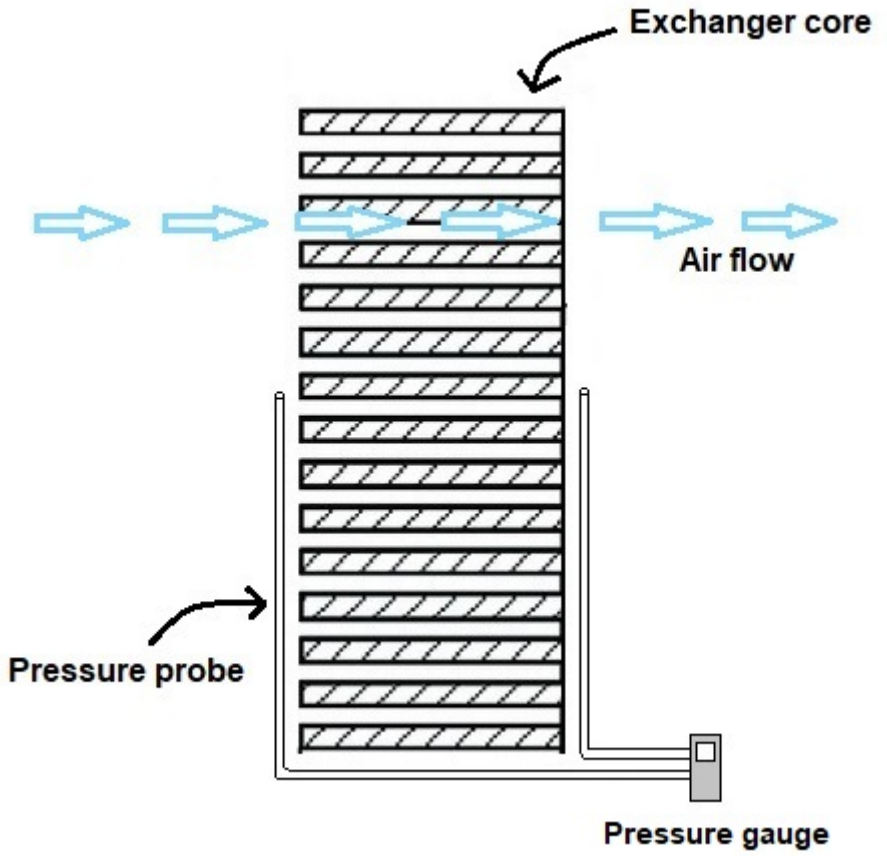


Figure 54: Representation of the pressure measurement for a heat exchanger.

→ If you have no heat exchange data, you may define a dummy curve where the heat exchange coefficient is equal to the air speed. However, keep in mind that this curve will be false and that you should not take the results into account.

Be careful: in your Navier-Stokes runs, the air speed going through the exchanger has to be in the range of values you provided or it will give you inaccurate results.

5. Self-evaluation of the calculation results

Use your own judgement to evaluate if the results seem logical or if there is a problem.

It is advised to verify the following checklist for the first calculation in a series :

In the configuration of your geometry tab, check :

- The hole detection points pictures :

  Red highlight in the pictures represent holes, it is advised to fill the hole or a problem might occur later.

- The view orientation pictures : The blue vector (x axis) is longitudinal and points to the rear.

The green vector (y axis) is spanwise and points to the right.
The red vector (z axis) is vertical and points upwards.
Verify if the parts are misplaced or if you did not forget any parts.

In the runs of your projects tab, check :

- MeshInternal pictures :

  The pictures must show holes representing the shapes inside your aircraft : it proves that these zones are not meshed.

- Prisms pictures :

  The quantity of surface layers on your aircraft must be at least 5. It is acceptable to have less layers in the zones where the viscous drag is neglectable.

- Convergence pictures :

  Within the "averaged for results" section the curve must be stable. Large jumps in values indicate a problem. If the curve is not stable, verify if the amplitudes are logical. The residuals are the estimations of the remaining error during computation. Their values should ideally decrease and remain as small as possible.

- RAM used :

  If your RAM used is at 90 or 100%, change your detail size parameters or add a computer to the computing cluster.

Error or warning message : (An orange message is a warning ; a red message indicates an error) :

- Errorview pictures :

  Errorview pictures appears only if your calculation has failed.
  Those pictures shows the bad quality meshes.

- Logs :

  Each log represents an OpenFoam operation, you can consult them in case of problem. If a warning icon appears, it is advised to look at it.

Part V.
CORRELATIONS

1. Detail size parameter

Investigation of the detail size parameter impact.
This investigation is based on a NASA experiment made on the geometry seen below (Wartime report L-373) and studied in the Langley Memorial Aeronautical Laboratory (Langley Field, Va.). *You can find the complete report in the References compressed folder provided.*

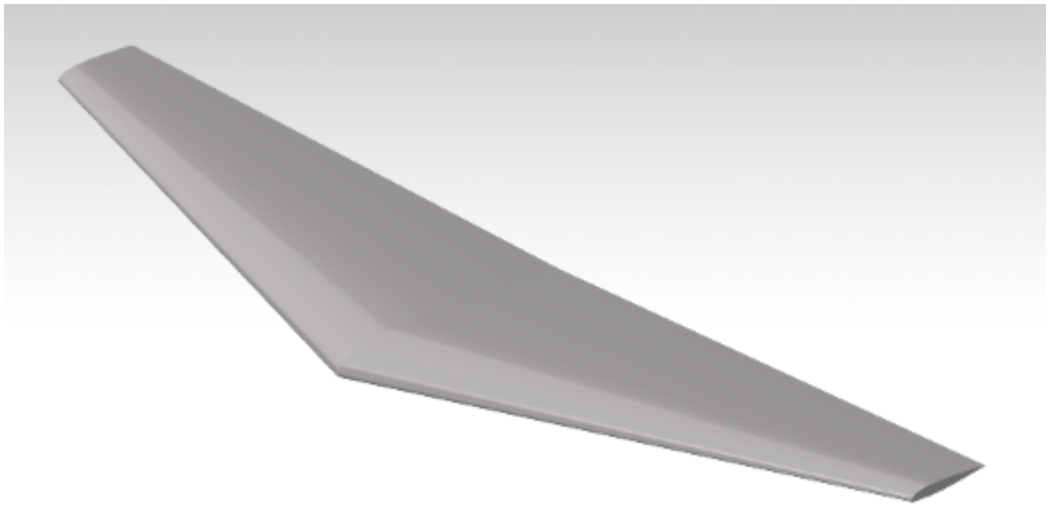


Figure 55: 3D Tappered NACA 23012.

Captions for figures 56 and 57 :
exp (black crosses) : NASA wind tunnel experiment.
det : detail size parameter.
te : trailing edge thickness.

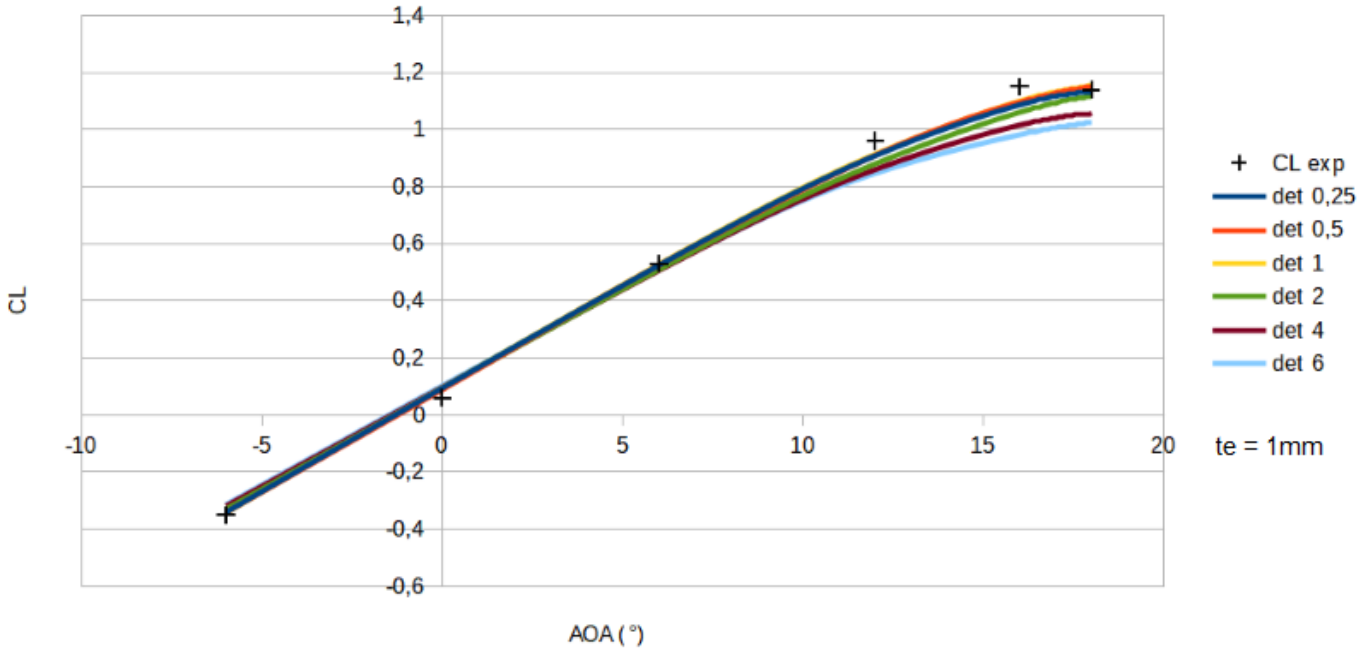


Figure 56: CL according to AoA (different detail sizes).

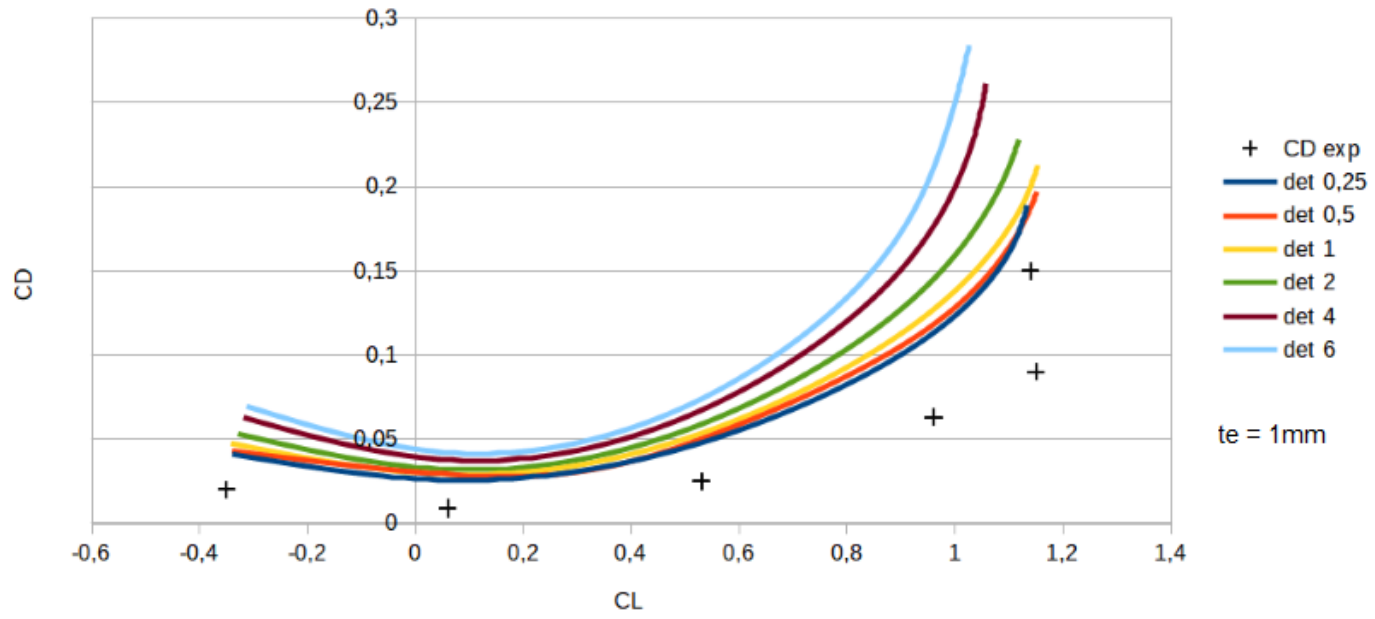
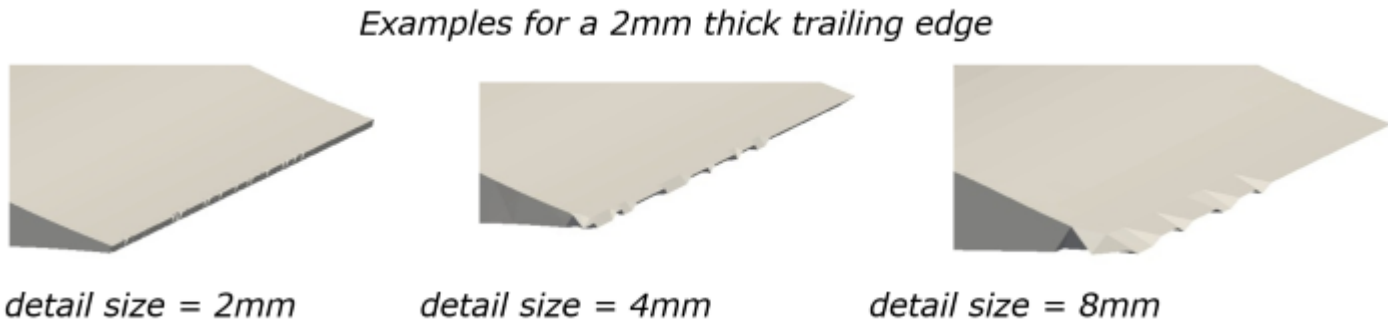


Figure 57: CD according to CL (different detail sizes).



If the detail size specified is larger than the trailing edge thickness, the mesh will not respect the trailing edge shape.

Conclusion :
The detail size defines the size of the meshing in the Navier-Stokes calculations.
The detail size number has a bigger impact on the drag coefficient (CD) than the lift coefficient (CL).
Set your detail size to a value neither too small nor too big :
If it is too small, the computation will take a lot of time and RAM.
If it is too big, it will be less accurate.
It is recommended to set the detail size value to half the trailing edge.

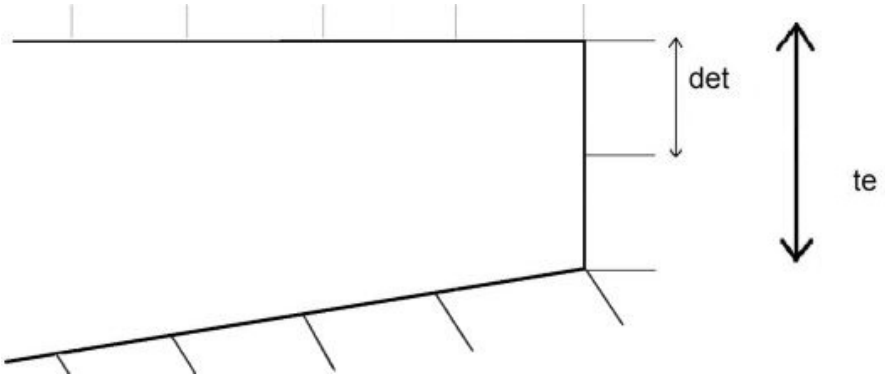


Figure 58: Detail size and resulting mesh.

For more information, see the 4. Detail size

2. Trailing edge thickness

Investigation of the trailing edge thickness impact.
This investigation is based on a NASA experiment made on the geometry seen below (Wartime report L-373) and studied in the Langley Memorial Aeronautical Laboratory (Langley Field, Va.).*You can find the complete report in the References compressed folder provided.*

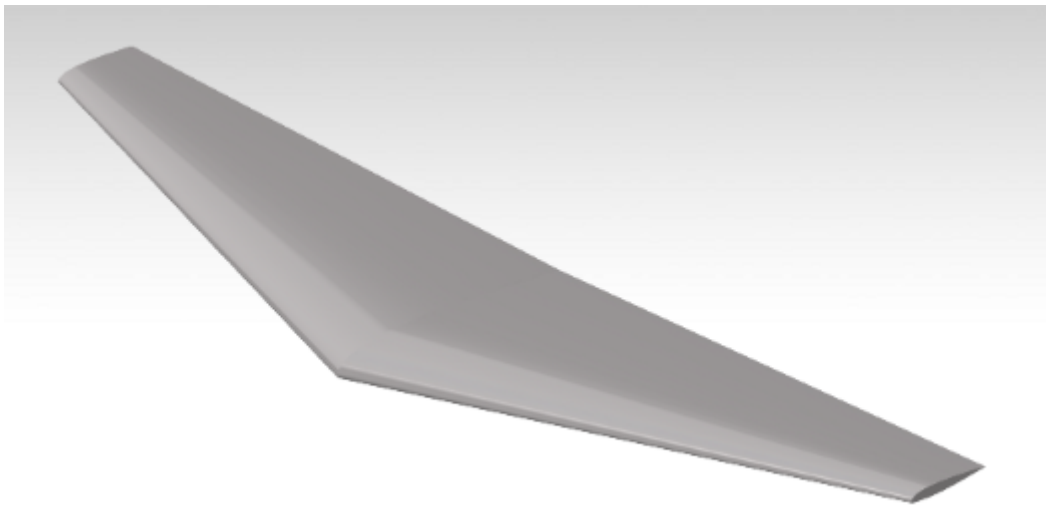


Figure 59: 3D Tappered NACA 23012.

Captions for figures 60 and 61 :
exp (black crosses) : NASA wind tunnel experiment.
te : trailing edge thickness.
The detail size is always half the trailing edge.

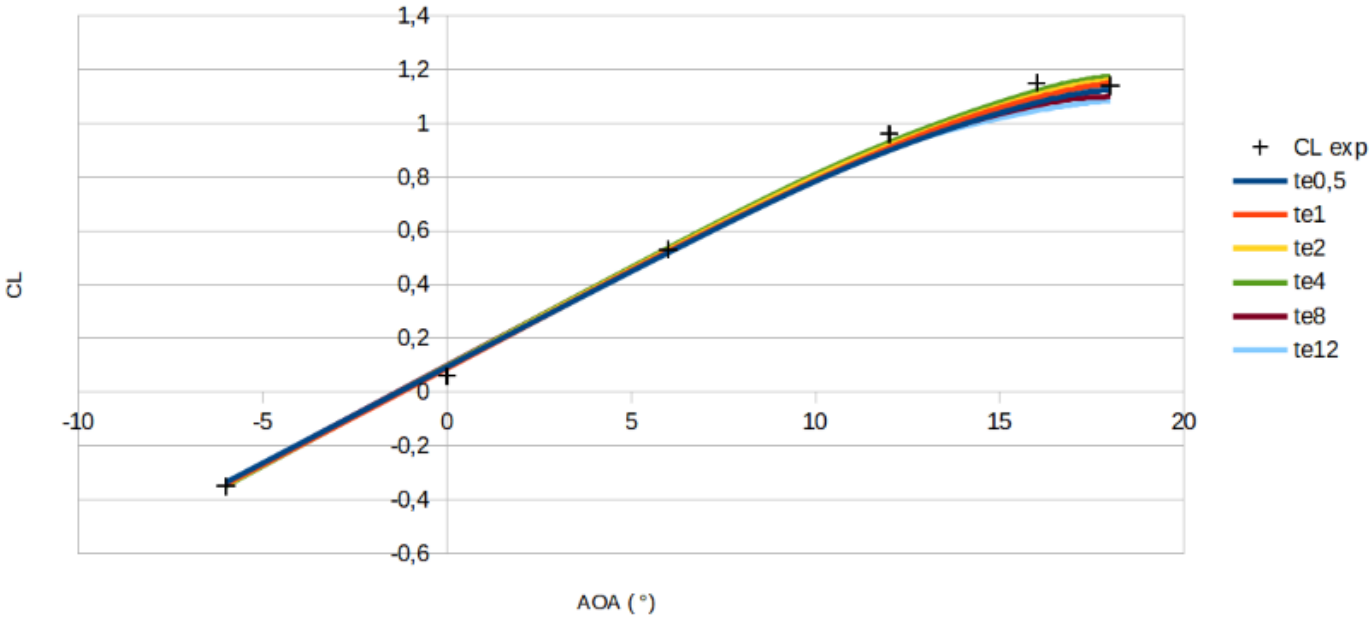


Figure 60: CL according to AoA.

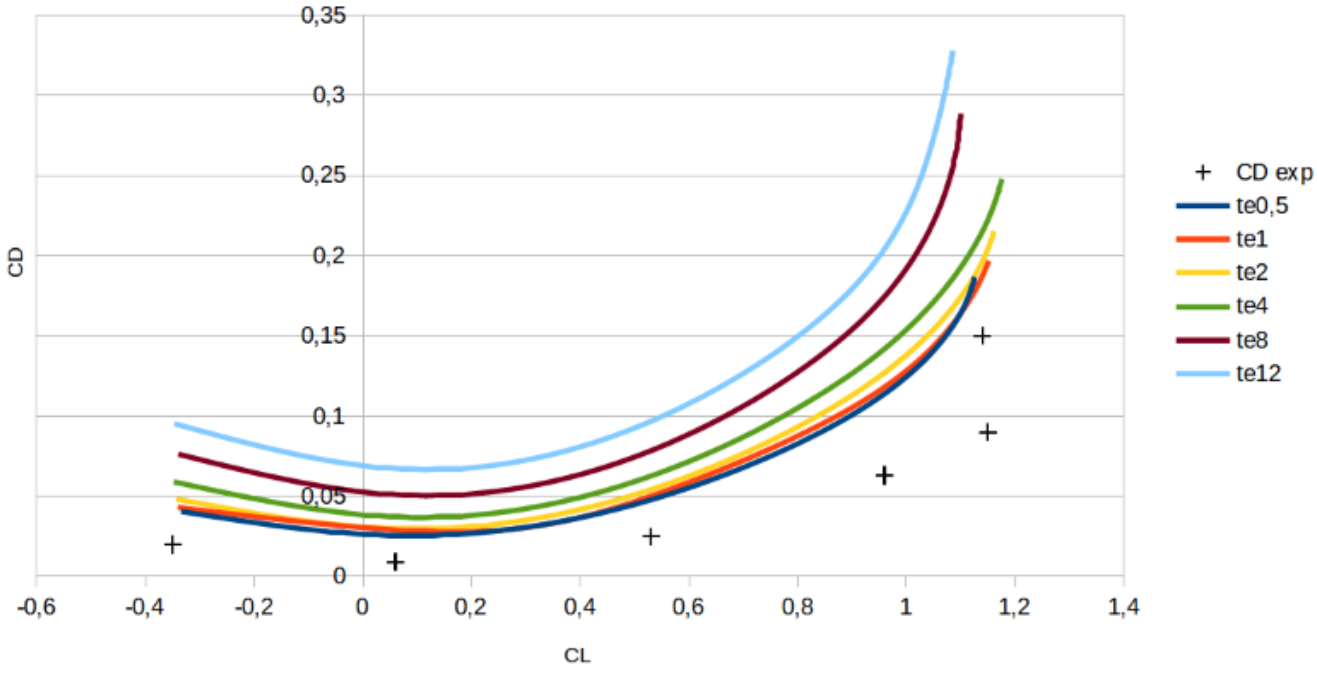


Figure 61: CD according to CL.

Conclusion:
The trailing edge thickness has a bigger impact on the drag coefficient (CD) than the lift coefficient (CL).
A trailing edge thickness closer to reality is better.
However, in case you need to limit the RAM used or the computation time, you may increase

the thickness in the CAD. This will allow you to define a bigger detail size and create a lighter mesh. Be aware that it may slightly overestimate the drag.

For more information, see the 4. Detail size

3. 2D turbulence model

Correlation between the NASA experiment and the 2D modeling turbulence models.
We will look at polars of steady and unsteady turbulence models to understand the pros and cons of each of them.

This investigation is based on a NASA experiment made on the geometry seen below (Wartime report L-697) and studied in the Langley Memorial Aeronautical Laboratory (Langley Field, Va.). *You can find the complete report in the References compressed folder provided.*

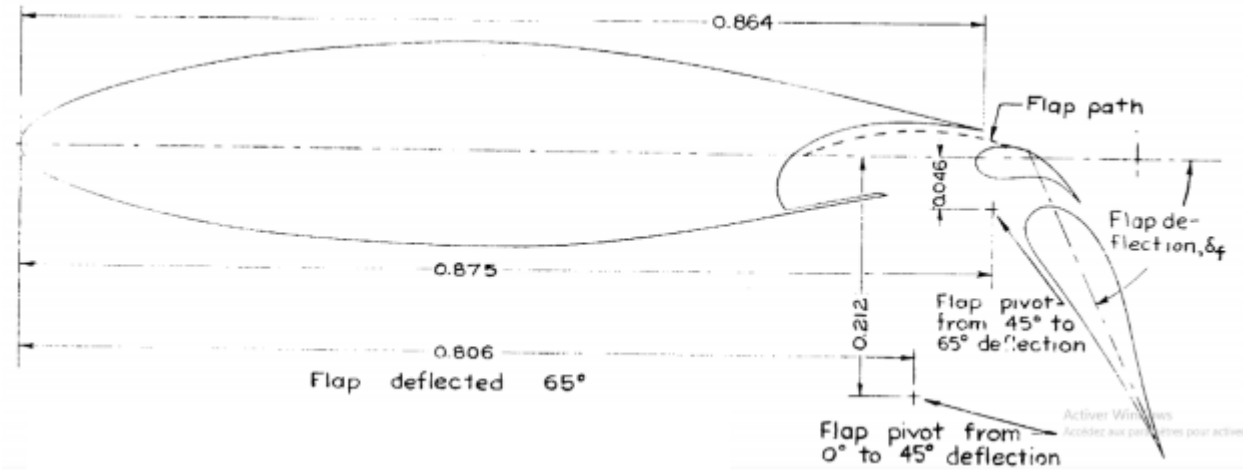


Figure 62: Flapped wings NACA 65,3-118, deflection 35°.

Captions for figures 63 to 66
exp (black crosses) : NASA wind tunnel experiment.
sa and kw : steady turbulence models.
sa-ddes and **kw-iddes** : unsteady turbulence models.

For more information about each type of turbulence model, see Steady turbulence model and Unsteady turbulence model in 2.2. Projects

In average, the meshes for each calculation are made of 12 million cells.

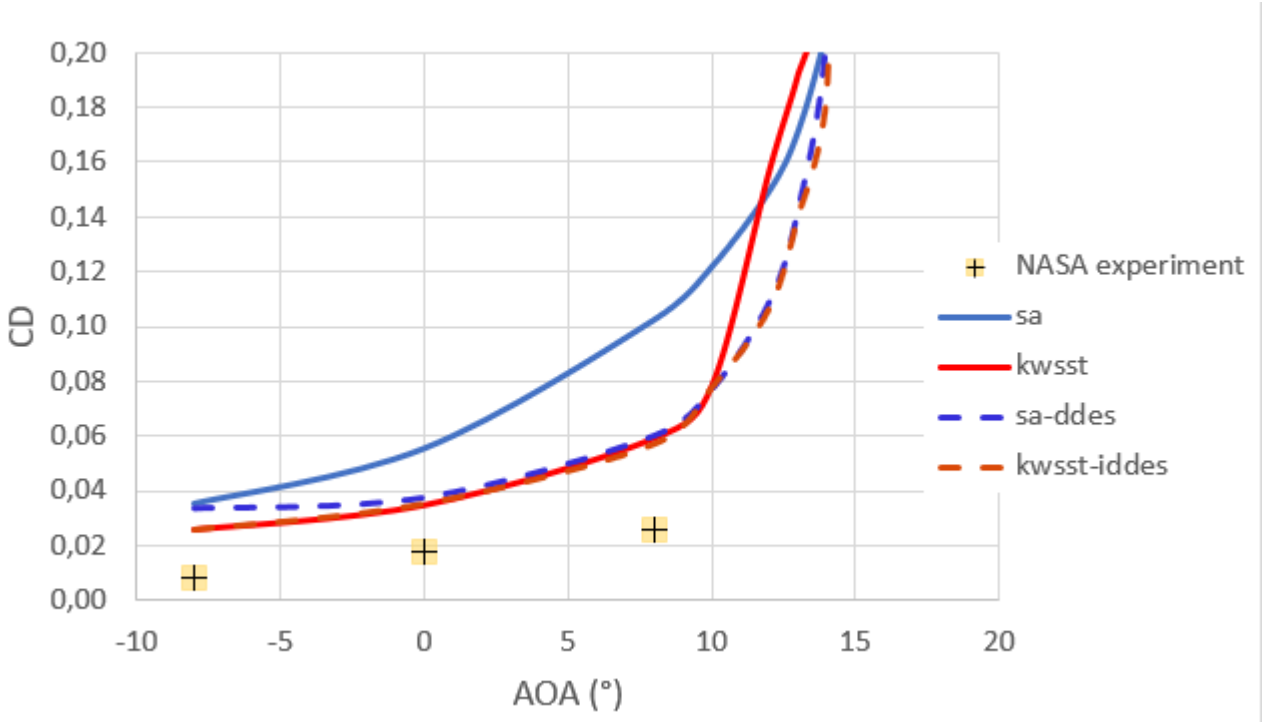


Figure 63: CD according to AoA(°).

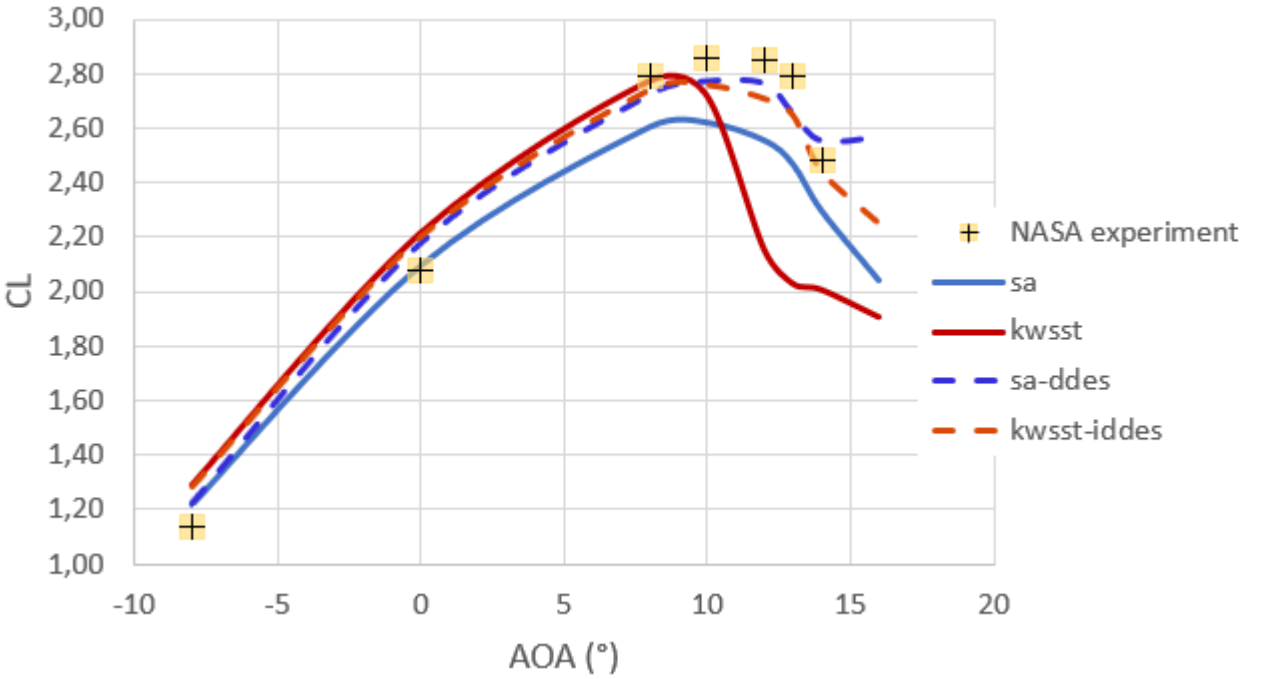


Figure 64: CL according to AoA(°).

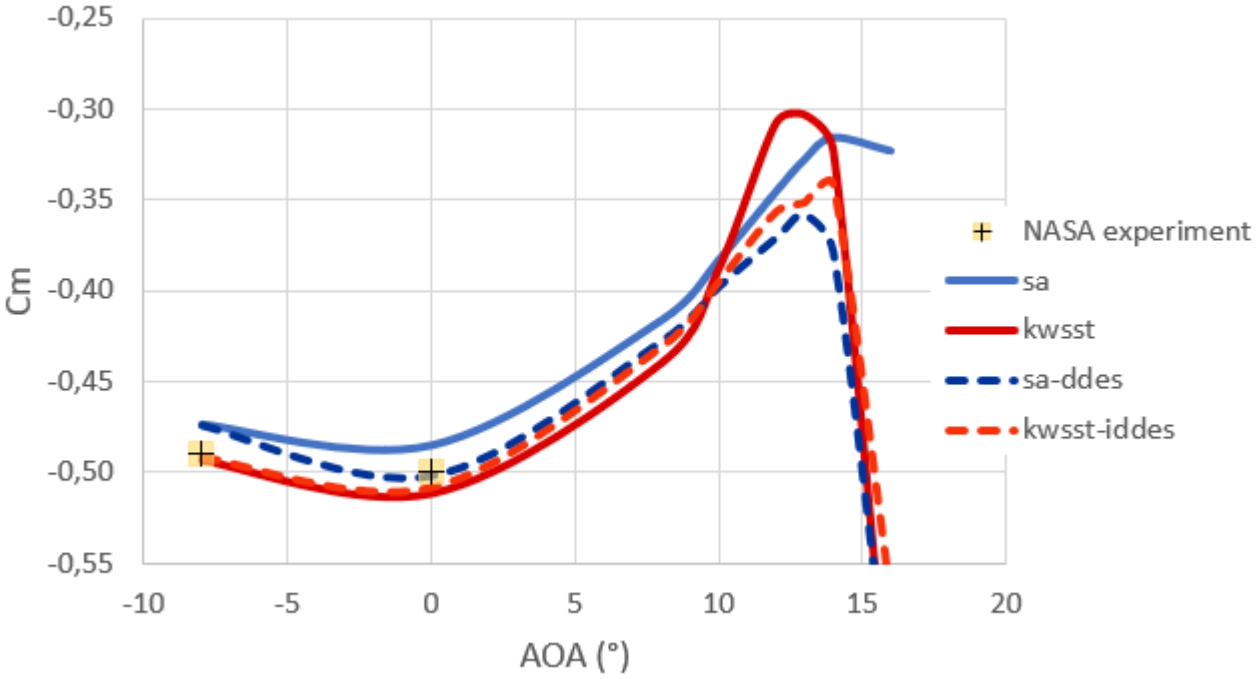


Figure 65: Cm according to AoA(°).

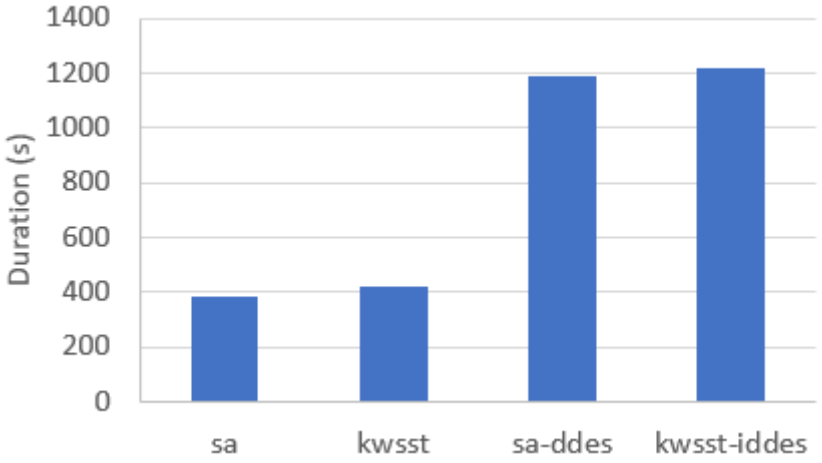


Figure 66: Histogram of steady and unsteady models according to 0° AoA.

4. 3D turbulence model

Correlation between the NASA experiment and the 3D modeling turbulence models. We will look at polars of steady and unsteady turbulence models to understand the pros and cons of each of them.

This investigation is based on a NASA experiment made on the geometry seen below (Technical Paper 2382) and studied in the Langley Research Center (Hampton, Virginia). *You can find the complete report in the References compressed folder.*

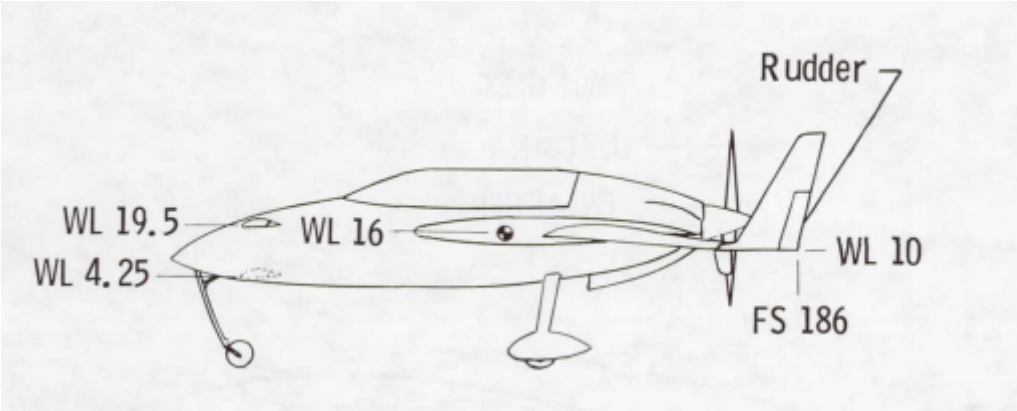


Figure 67: Varieze.

Caption for the figures 68 to 71 :
exp (black crosses) : experimental data from NASA (on the same aircraft).
sa and **kw** : steady turbulence models.
sa-ddes and **kw-iddes** : unsteady turbulence models.

For more information about each type of turbulence model, see Steady turbulence model and Unsteady turbulence model in 2.2. Projects.

In average, the meshes for each calculation are made of 19 million cells.

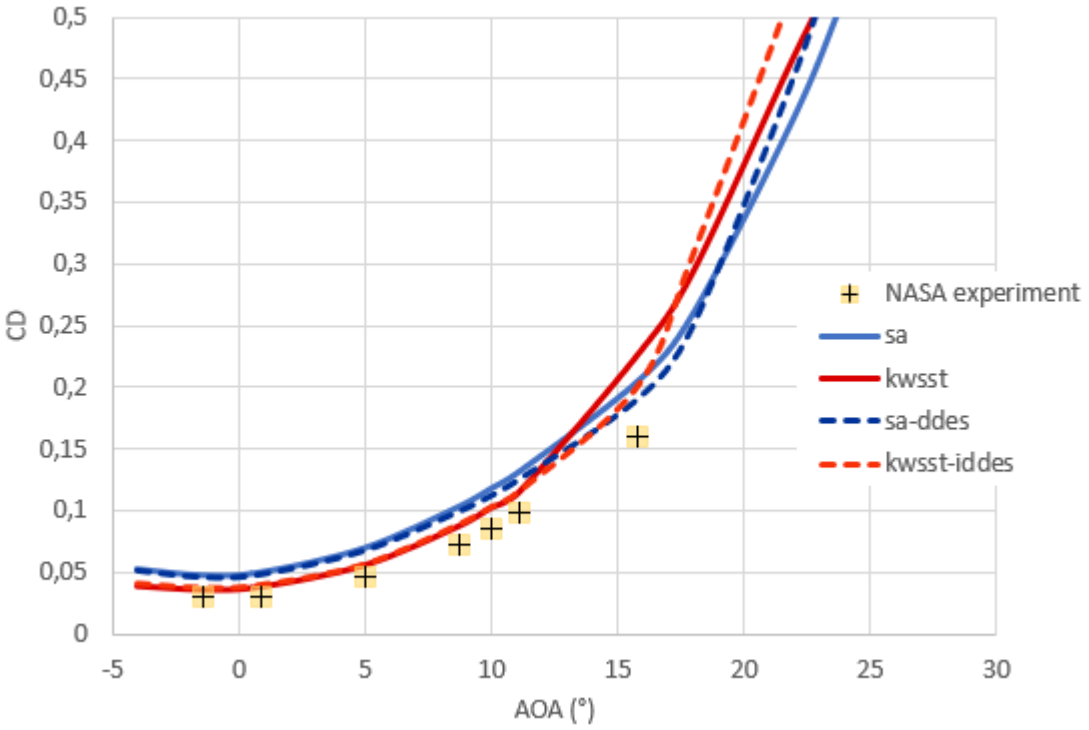


Figure 68: CD according to AoA(°).

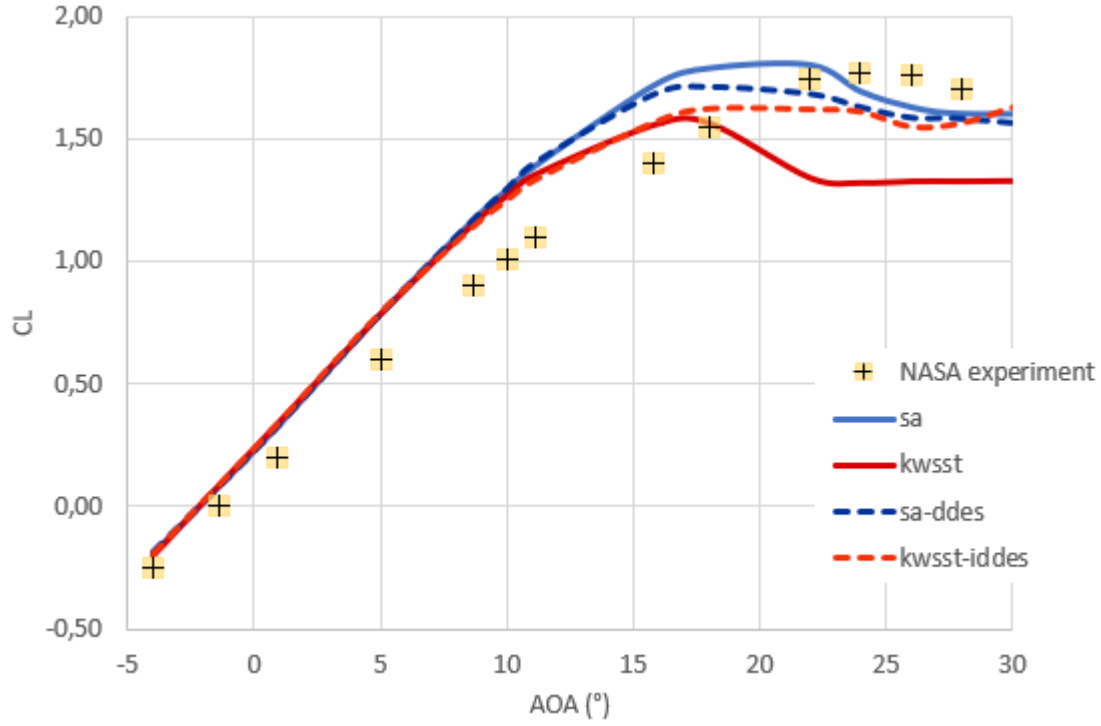


Figure 69: CL according to AoA(°).

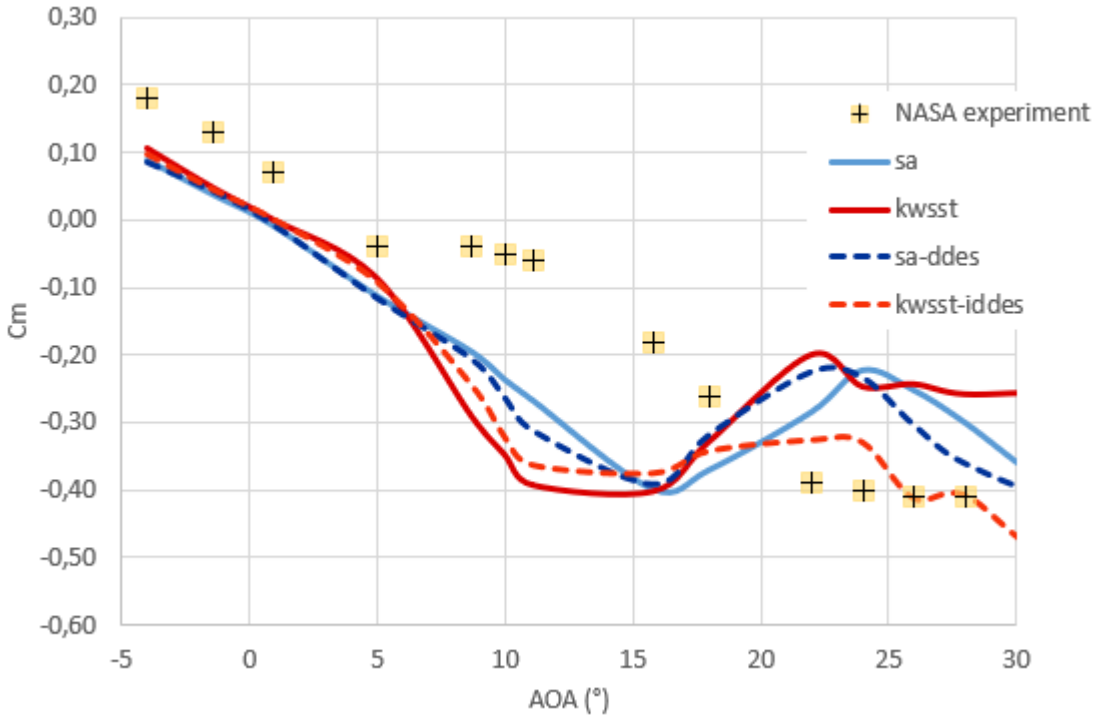


Figure 70: Cm according to AoA(°).

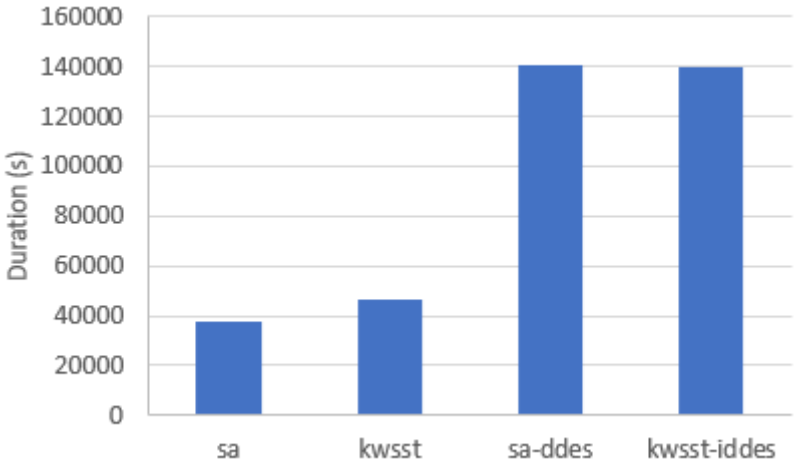


Figure 71: Histogram of steady and unsteady models according to 0° AoA

Part VI.
HELP AND SUPPORT

1. Troubleshooting

- **Error during ACE of Aircraft installation : pip not found**: Generally this happens when the user forgot to select the option 'Add python 3.8 to PATH' during the installation of python. Uninstalling an re-installing python, without forgetting to activate this option during the install, should fix the problem.

- **Error : python38.dll not found**: This indicates that the software cannot find a path to the python libraries. Generally this happens when the user forgot to select the option 'Add python 3.8 to PATH' during the installation of python. Uninstalling an re-installing python, without forgetting to activate this option during the install, should fix the problem.

- **Error 0xc000007b**: Indicates a 32bits vs 64 bits incompatibility. Ace_of_Aircraft only work in 64 bits. Make sure your python version is 64 bits as well.

- **Application crashing at launch without errors**: This generally indicates a python error, but the application window closes imediatly and there is no time to see the error. In order to visualise the error, launch the application with a terminal.

  On Windows, press simultaneously the windows key and the R key on the keyboard, and run cmd. This will open a terminal, in this terminal go to the application directory using the command cd. For example for the default installation directory type :
  cd %AppData%/Ace_of_Aircraft_vX
  Then execute the app by typing :
  Ace_of_Aircraft.exe

  On Ubuntu, proceed similarly, launch a terminal and use the commands:
  cd /usr/lib/ace-of-aircraft/ace-of-aircraft-vX
  ./Ace_of_Aircraft

- **Truncated text in the software**: This often happens with windows, depending on your screen resolution. Open the windows settings, select the 'system' category, then find the 'display' section. In there, find a percentage setting for size of texts and apps, set it to 100%.

- **Text and images appear very small in the software**: This can happen if you have a very high resolution on a relatively small display. In ACE of Aircraft, go to the settings tab, and make a small increase on the 'UI scale coefficient' parameter. Close and restart the application for the parameter to take effect. All texts and graphical elements should appear bigger on screen.

2. Contact us

Please read the User manual before contacting us. If you do not find what you need, or if you want to report a bug please send us an e-mail explaining your problem.

Website: aero-ce.com
e-mail : support-aoa@aero-ce.com