Disclaimer

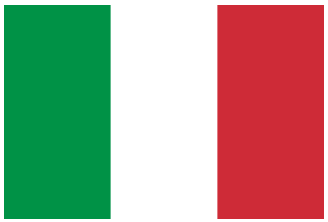
“This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks.”

Introductory OpenFOAM® Course

From 8th to 12th July, 2013

University of Genoa, DICCA

Dipartimento di **I**ngegneria **C**ivile, **C**himica e **A**mbientale



UNIVERSITÀ DEGLI STUDI
DI GENOVA

Your Lecturer

Joel GUERRERO

joel.guerrero@unige.it

guerrero@wolfdynamics.com

UNIVERSITÀ DEGLI STUDI DI GENOVA

wolf dynamics

Damiano NATALI

damiano.natali@unige.it

natali@wolfdynamics.com

UNIVERSITÀ DEGLI STUDI DI GENOVA

wolf dynamics

Acknowledgements

These slides and the tutorials presented are based upon personal experience, OpenFOAM® source code, OpenFOAM® user guide, OpenFOAM® programmer's guide, and presentations from previous OpenFOAM® training sessions and OpenFOAM® workshops.

We gratefully acknowledge the following OpenFOAM® users for their consent to use their material:

- Hrvoje Jasak. Wikki Ltd.
- Hakan Nilsson. Department of Applied Mechanics, Chalmers University of Technology.
- Eric Paterson. Applied Research Laboratory Professor of Mechanical Engineering, Pennsylvania State University.

Today's lecture

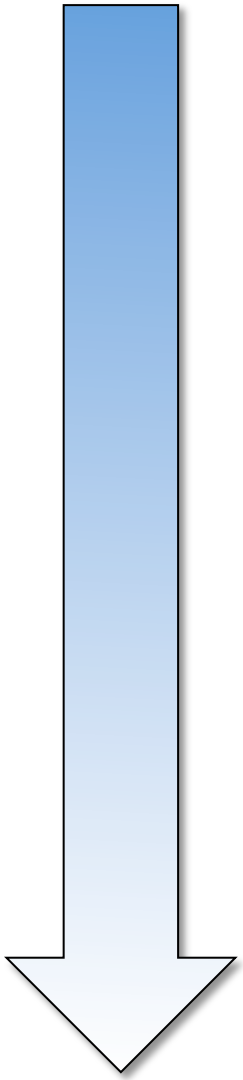
1. **CFD simulation workflow**
2. Geometry generation using open source tools
3. Hands-on session

CFD simulation workflow

Geometry	Meshing	Case setup and solver	Post processing
Salome	blockMesh	OpenFOAM® (FVM)	paraFoam
Blender	snappyHexMesh	Code Saturne (FVM)	paraView
Free-CAD	Salome	Overture (FDM)	VISIT
Google Sketch-Up	Engrid	ELMER (FEM-DG)	Gnuplot
	GMSH	OpenLB (LBM)	Octave
			Grace
			R computational statistics

This list does not enumerate all the open source applications available. It only shows those applications that I like to use or I feel confortable with.

CFD simulation workflow



Geometry

Meshing

Up to 70% of user time.

Case setup

Up to 10% of user time.

Post processing

Up to 20% of user time.

The percentages shown are based on my personal experience.

Today's lecture

1. ~~CFD simulation workflow~~
2. **Geometry generation using open source tools**
3. Hands-on session

Geometry generation using open source tools

- The best way to learn how to use the geometry generation tools is by doing.
- There are many video tutorials available on internet for each specific tool, so feel free to surf the web.
- Hereafter we are going to show you how to get started with the geometry generation tools. The rest is on you.

Geometry generation using open source tools

- There are always many ways to accomplish a task when creating a geometry, this give you the freedom to work in a way that is confortable to you. Hereafter I am going to show you my way.

- There is no wrong or right way to generate a geometry. The only rule you should keep in mind is that by the end of the day you should get a unique clean and watertight geometry.

Geometry generation using open source tools

- Remember, the quality of the mesh and hence of the solution, greatly depends on the geometry. So always do your best when creating the geometry.

Geometry generation using open source tools

Potential geometry issues

- Missing faces.
- Small faces.
- Misaligned faces.
- Overlapping faces.
- Sliver faces (high aspect-ratio).
- Repeated faces.
- Several surfaces connected to a single surface.
- Cracks.
- Gaps.
- Free faces, edges, nodes.
- Hard edges.
- Small edges.
- Sharp angles.
- Repeated edges.
- High curvature NURBS.

.. among others.

These issues must be fixed in order to create a smooth, clean, watertight body and to prevent meshing issues

Geometry generation using open source tools

Potential geometry issues

- Missing faces.
- Small faces.
- Misaligned faces.
- Overlapping faces.
- Sliver faces (high aspect-ratio).
- Repeated faces.
- Several surfaces connected to a single surface.
- Cracks.
- Gaps.
- Free faces, edges, nodes.
- Hard edges.
- Small edges.
- Sharp angles.
- Repeated edges.
- High curvature NURBS.

- In general, when generating the geometry and by using good geometry generation practices, we should not experience these geometry issues. At the end, we should get a smooth, clean, watertight body.

- Usually, we find these issues when importing or exporting the geometry from/to different formats.

Geometry generation using open source tools

Remember, before going to the meshing stage we must fix/cleanup the geometry.

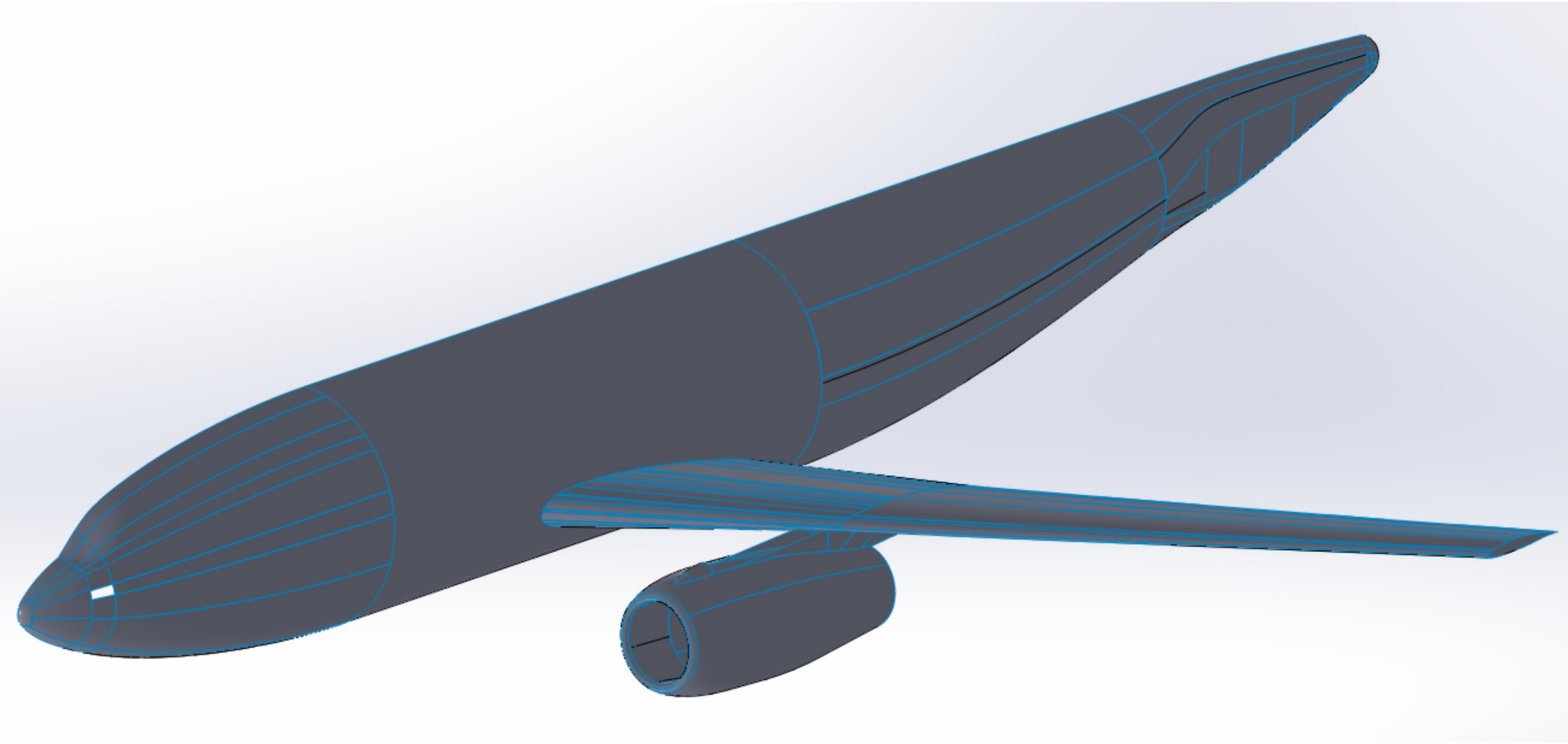
So, how do we prepare the geometry for mesh generation?

- Delete hard edges.
- Delete small edges/faces.
- Fill holes.
- Split surfaces with high curvature.
- Sew faces.
- Remove sliver faces.
- Connect/disconnected edges/faces.
- Delete sharp edges.
- Remove unnecessary details (defeaturing). This includes points, edges and faces.
- Decompose geometry into meshable sections.

Let us take a look at a corrupt or incomplete geometry

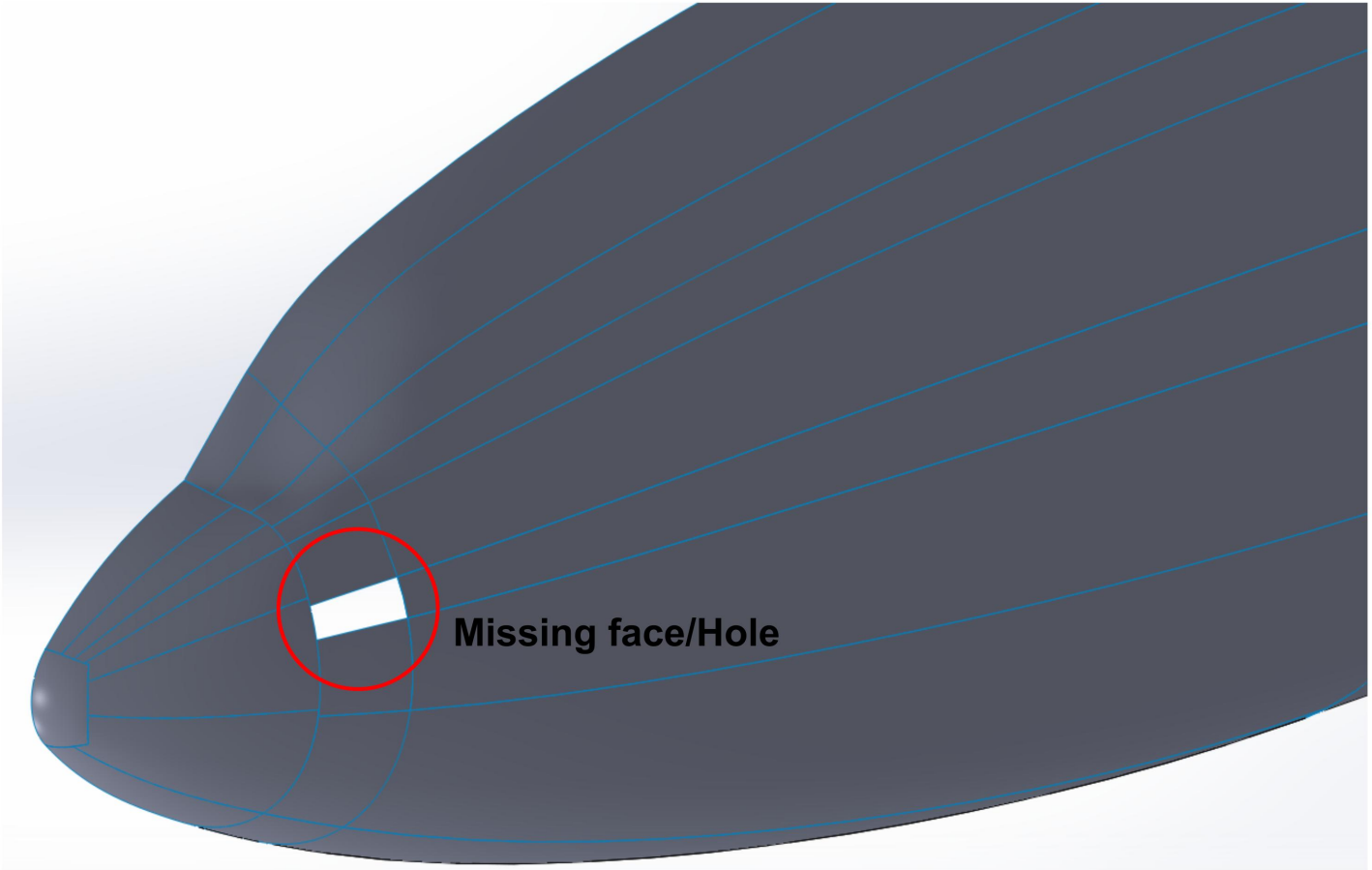
Geometry generation using open source tools

Geometry repair/cleanup



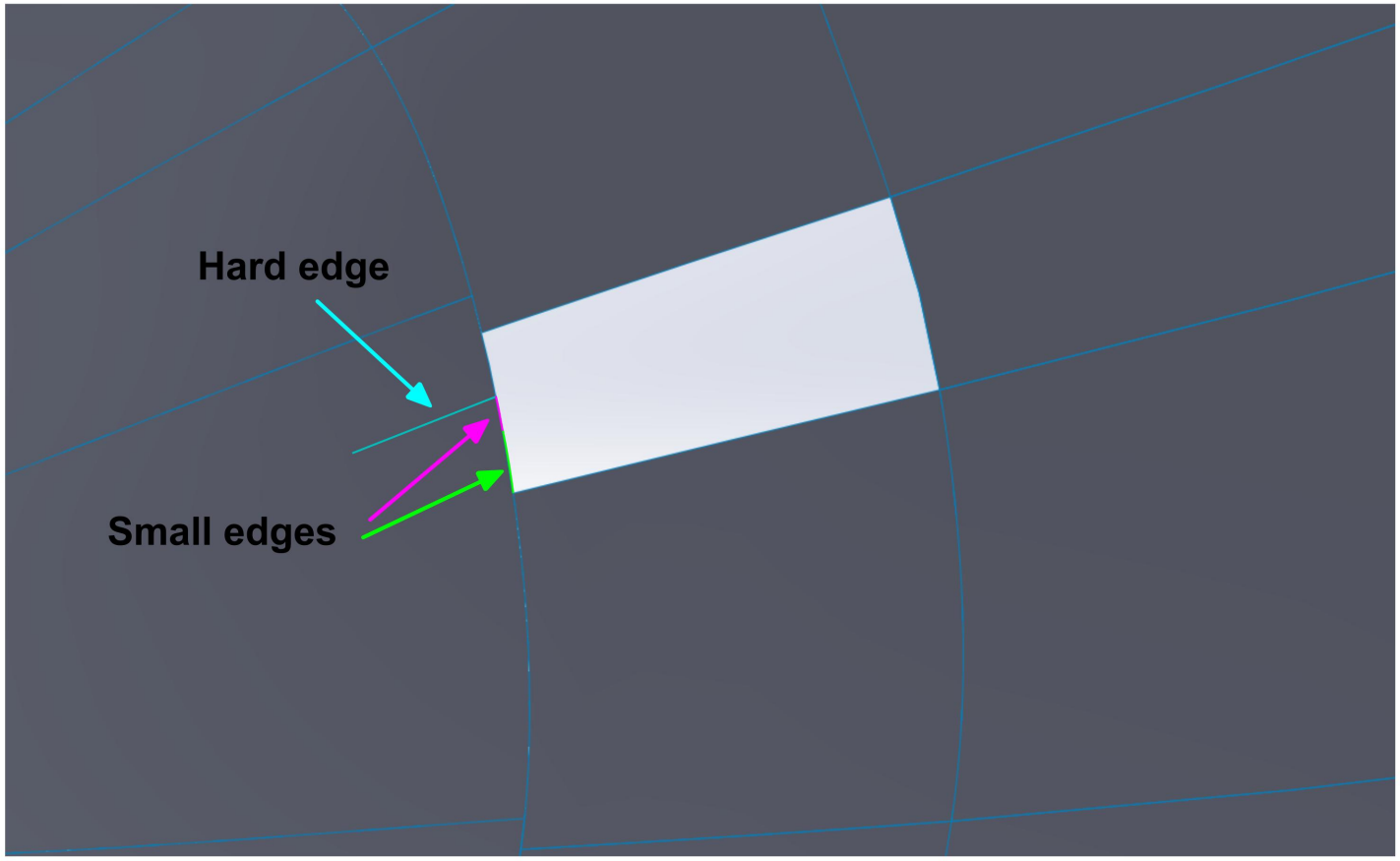
Geometry generation using open source tools

Geometry repair/cleanup



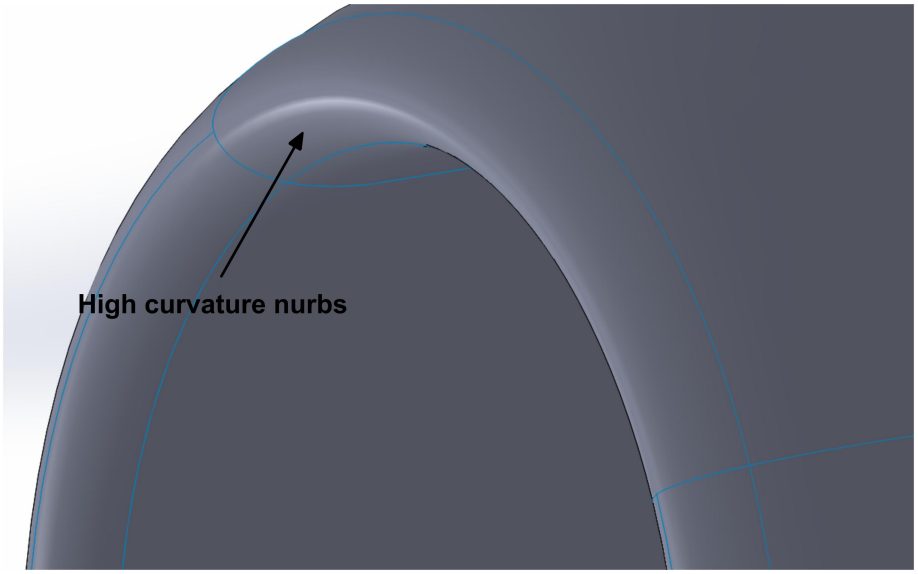
Geometry generation using open source tools

Geometry repair/cleanup

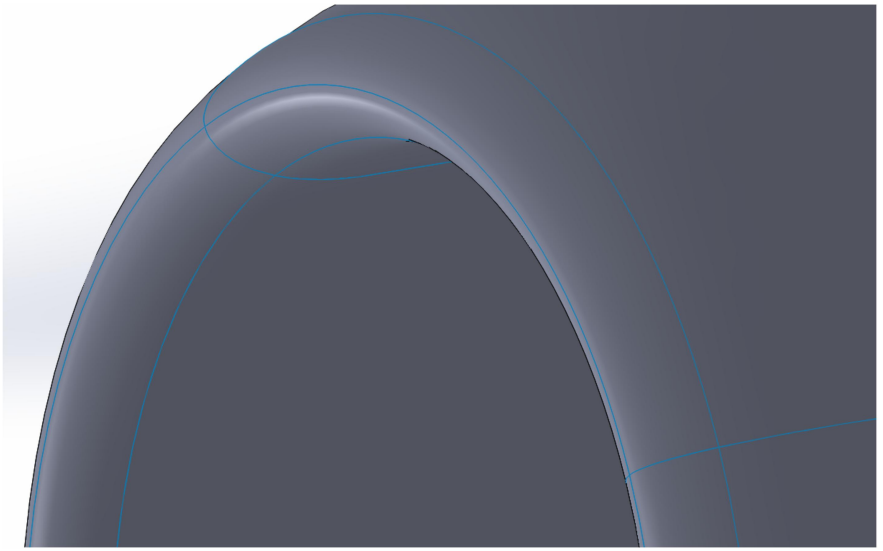


Geometry generation using open source tools

Geometry repair/cleanup

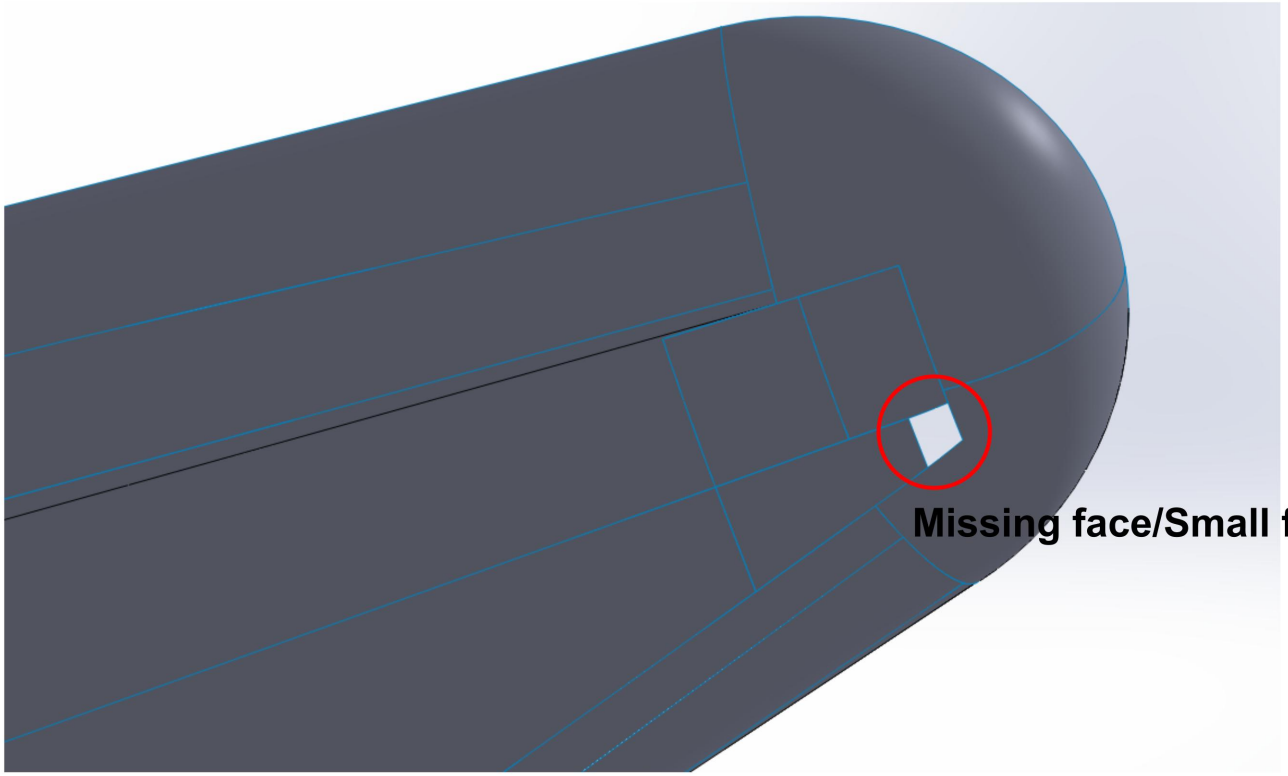


To improve quality, split the single surface into two surfaces



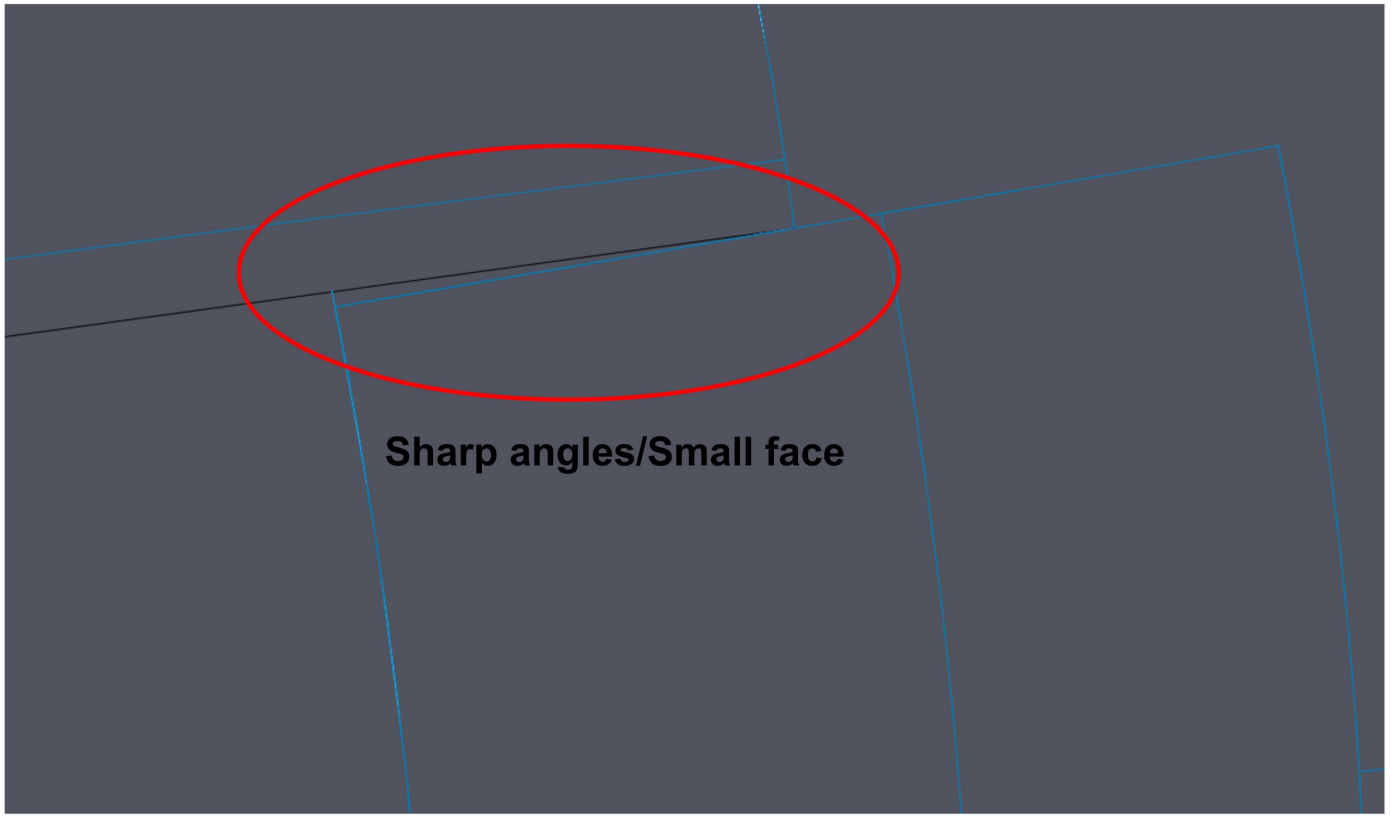
Geometry generation using open source tools

Geometry repair/cleanup



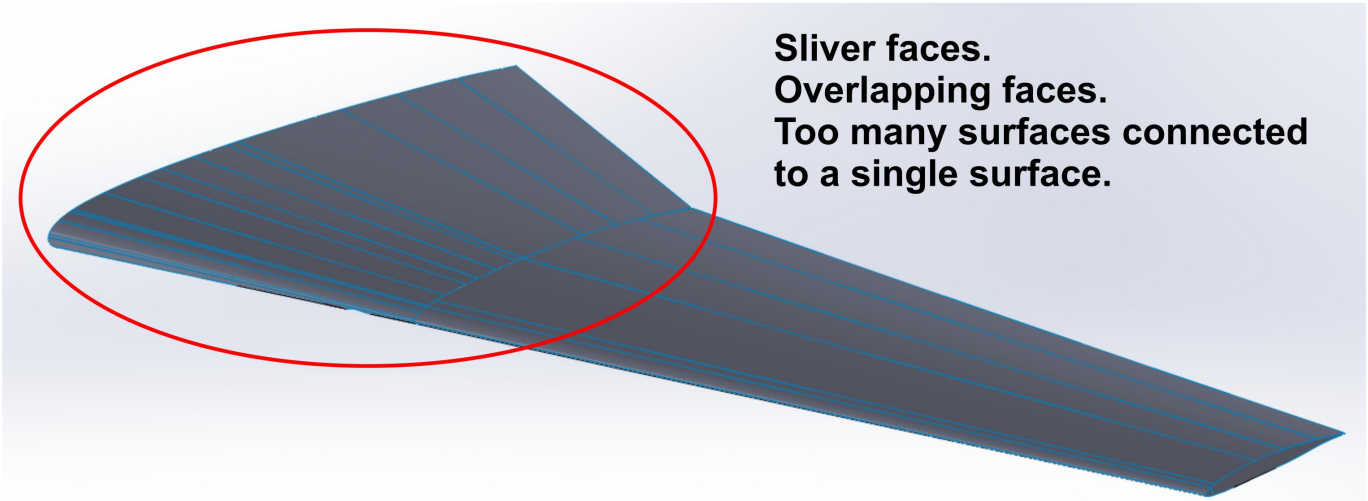
Geometry generation using open source tools

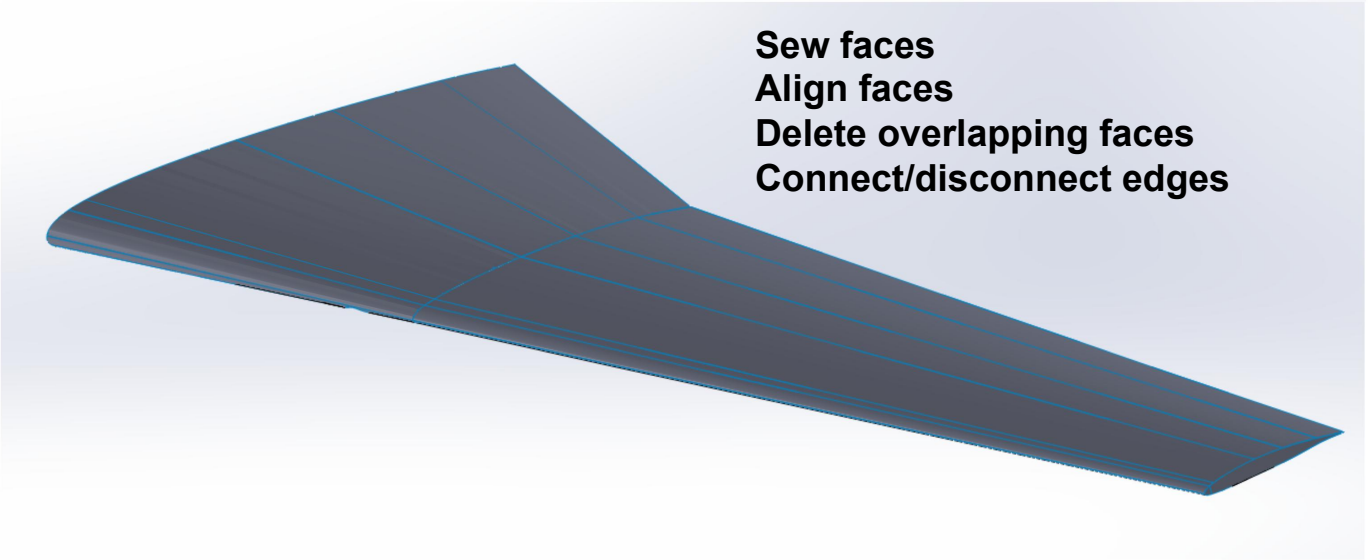
Geometry repair/cleanup



Geometry generation using open source tools

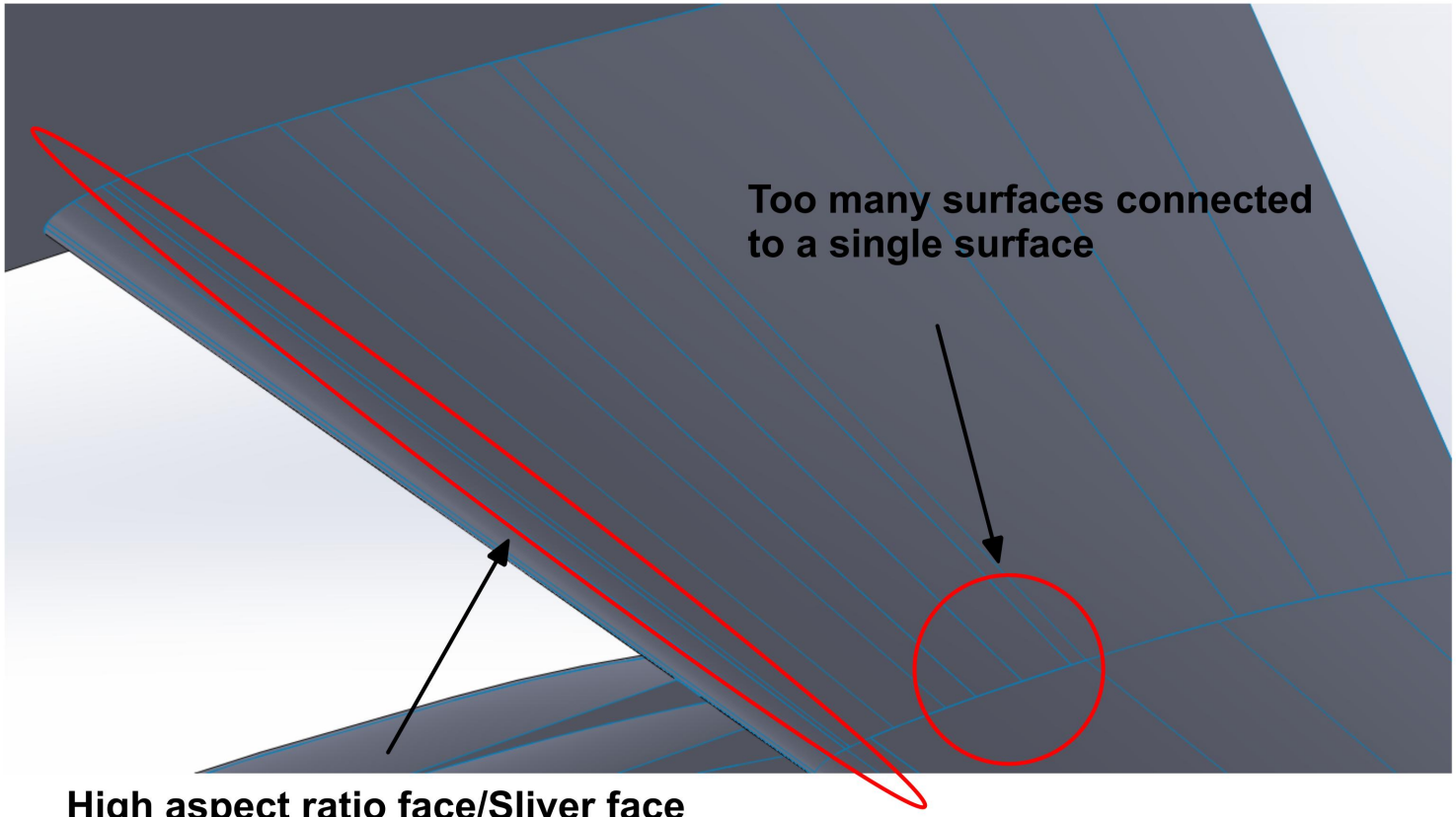
Geometry repair/cleanup





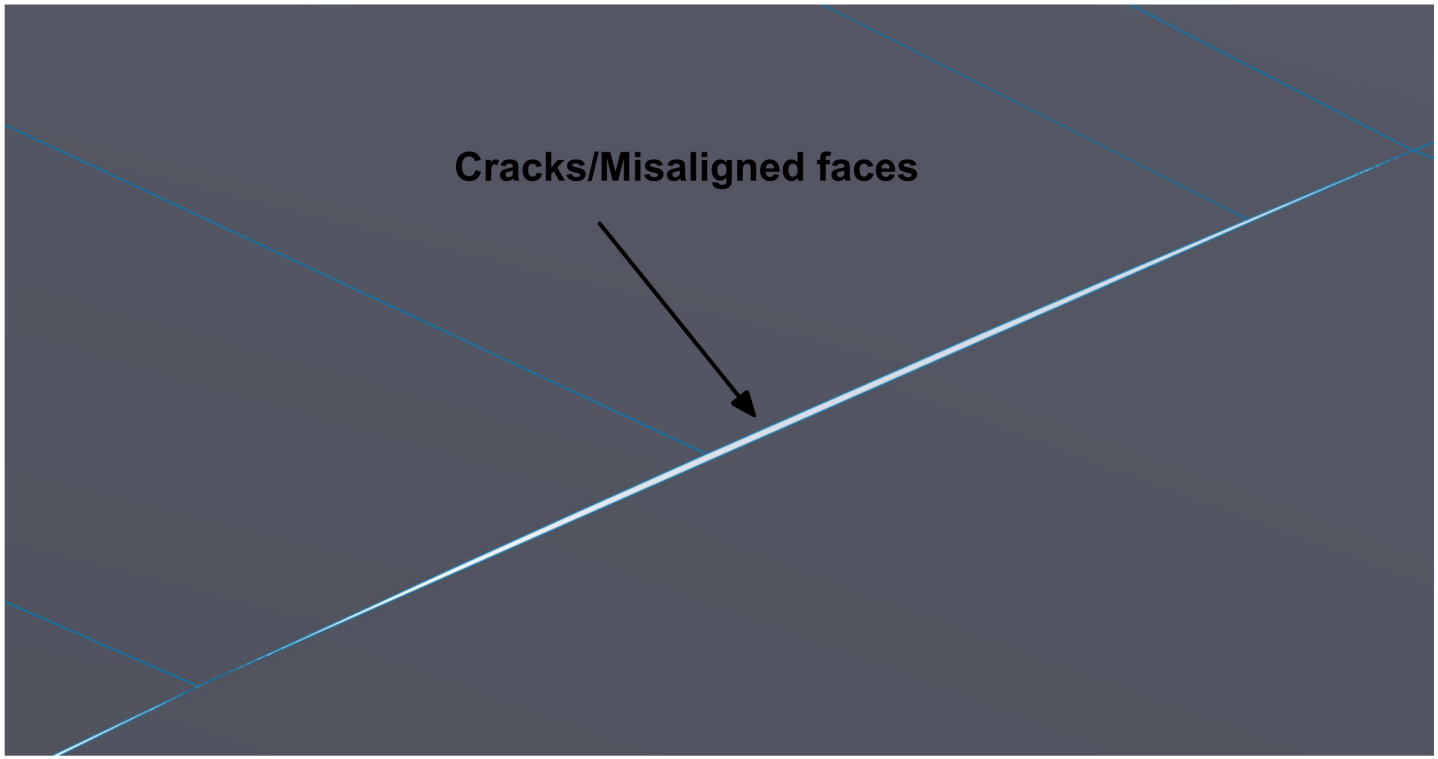
Geometry generation using open source tools

Geometry repair/cleanup



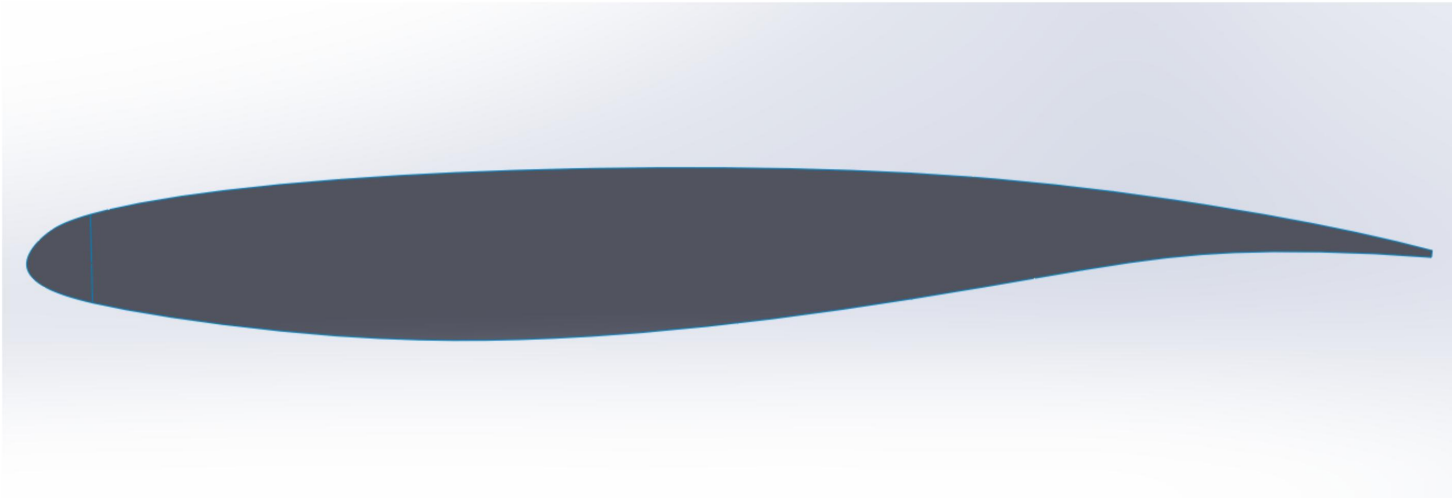
Geometry generation using open source tools

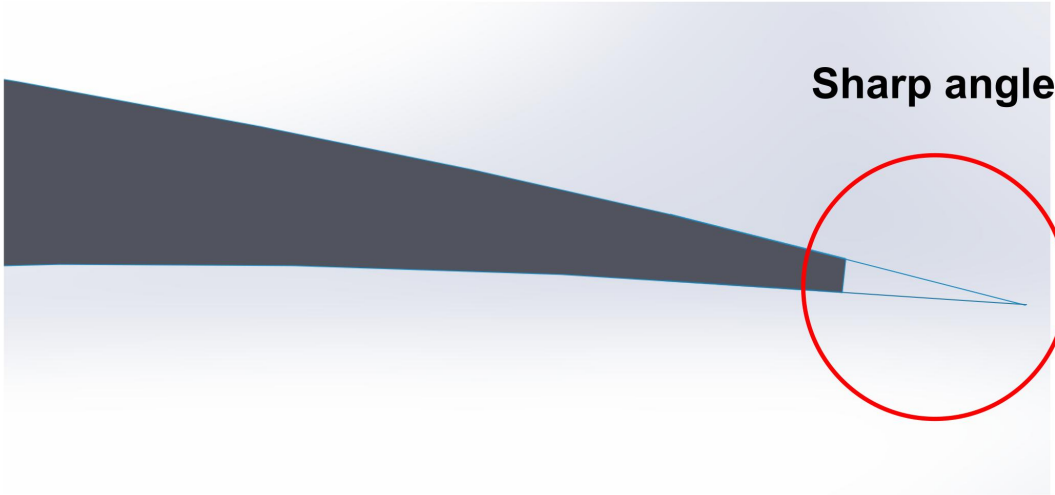
Geometry repair/cleanup



Geometry generation using open source tools

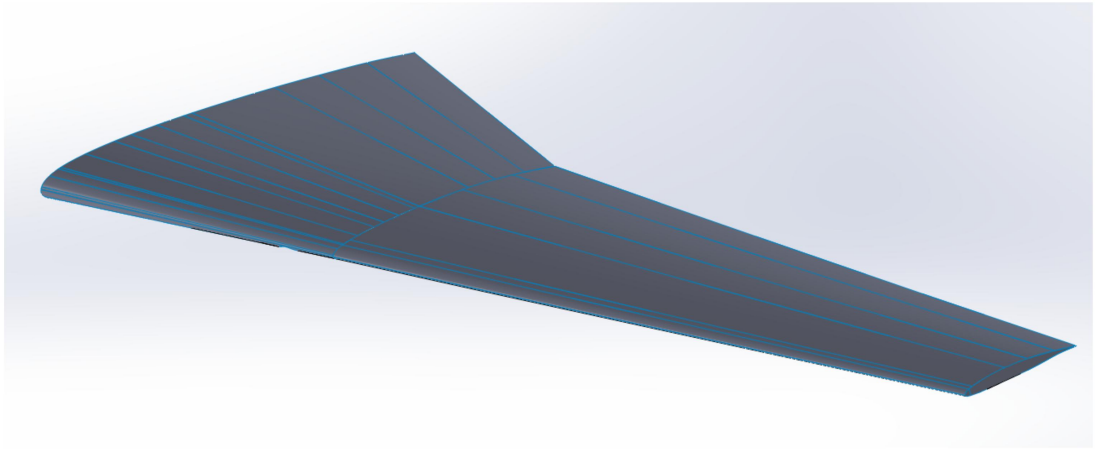
Geometry repair/cleanup

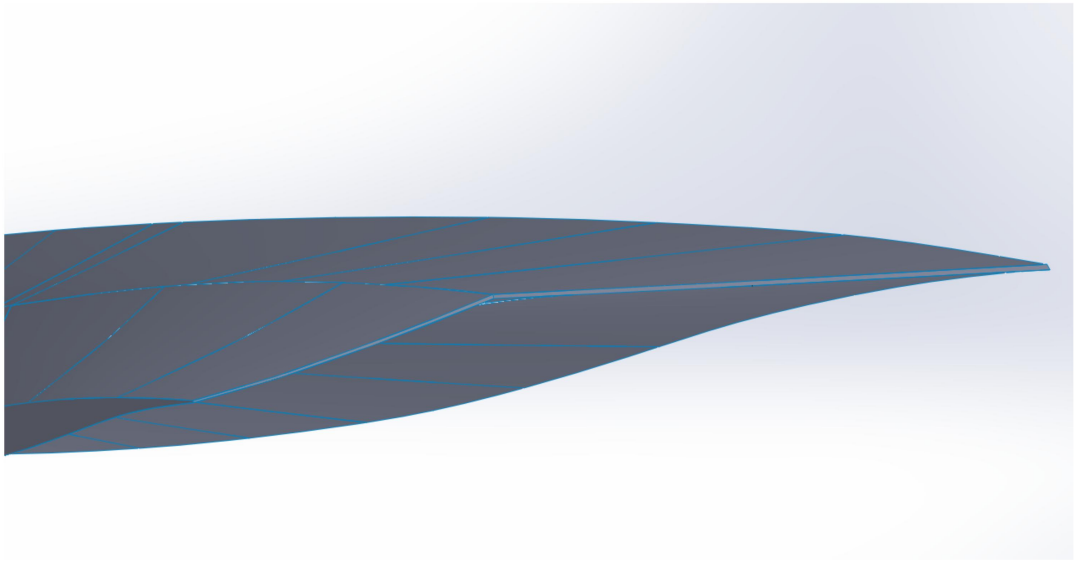




Geometry generation using open source tools

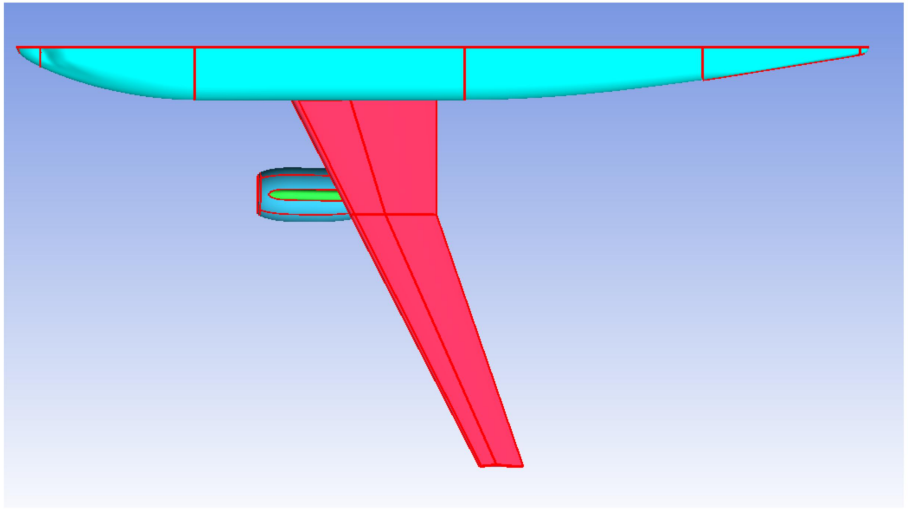
Geometry repair/cleanup

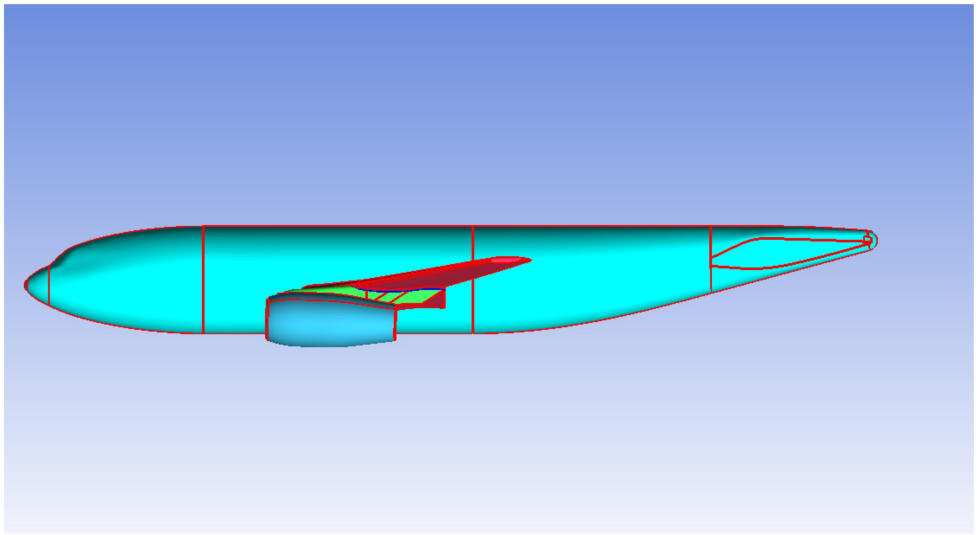


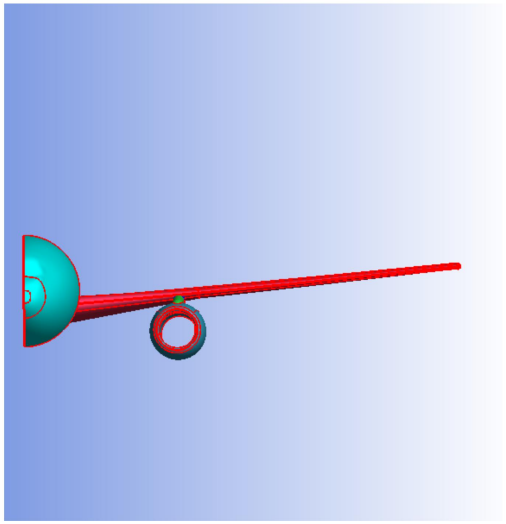


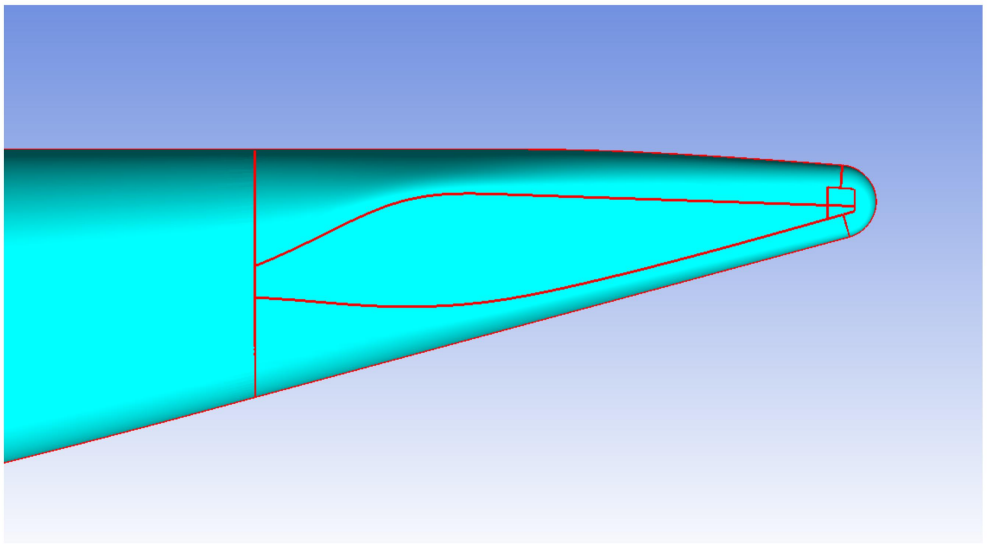
Geometry generation using open source tools

Geometry repair/cleanup





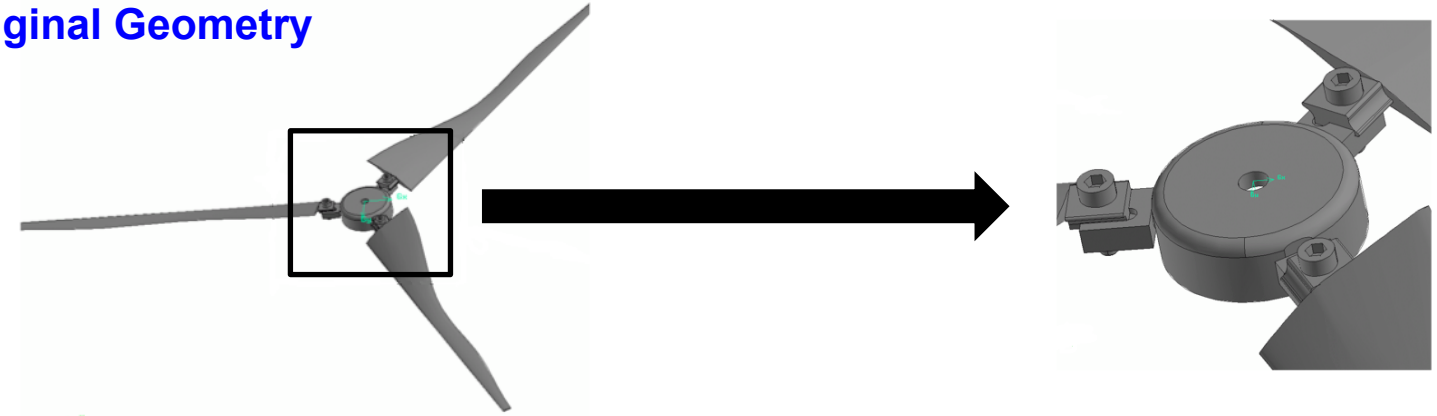




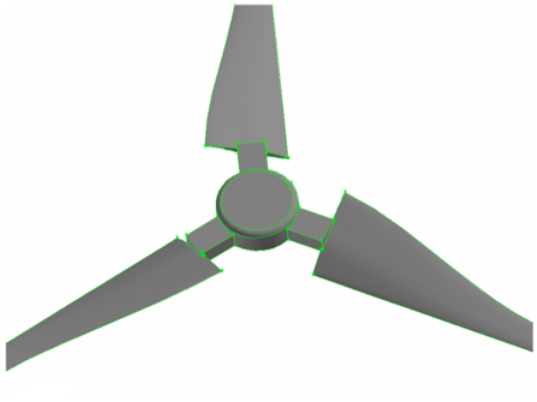
Geometry generation using open source tools

- Many times, it is not necessary to model all the details of the geometry. In these cases you should consider simplifying the geometry (geometry defeaturing).
- **Geometry defeaturing** can save you a lot of time when generating the mesh. So be smart, and use it whenever is possible.

Original Geometry



Defeatured Geometry



Geometry generation using open source tools

- **Salome:** history based (parametric design). It is a complete pre and post processing application. It has quite extensive capabilities for creation and manipulation of solid geometries.
  http://www.salome-platform.org/

- **Free-CAD:** history based (parametric design). Light CAD software, good for not very complicated mechanical designs.
  http://sourceforge.net/apps/mediawiki/free-cad/

- **Blender:** direct 3D modeling tool, it can be integrated with OpenFOAM. Extremely powerful for surface modeling and manipulation.
  http://www.blender.org/

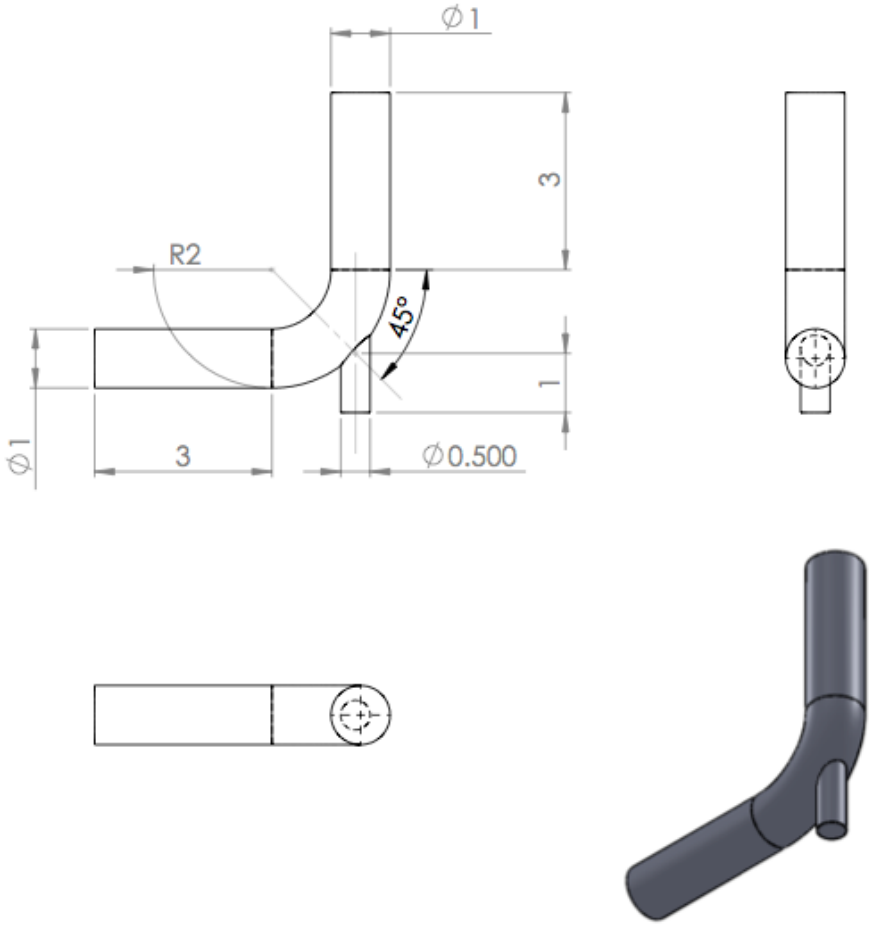
- **Google Sketch-Up:** direct 3D modeling tool. There are many plugins available that extend Google Sketch-Up capabilities.
  http://sketchup.google.com/

Friendliness/Easiness to use:  **Good**  **Fairly Good**  **Mild**  **Bad**  **Very Bad**

Geometry generation using open source tools

Geometry 1. Mixing elbow.

Let us create the following geometry by using Salome or Google Sketch-Up.

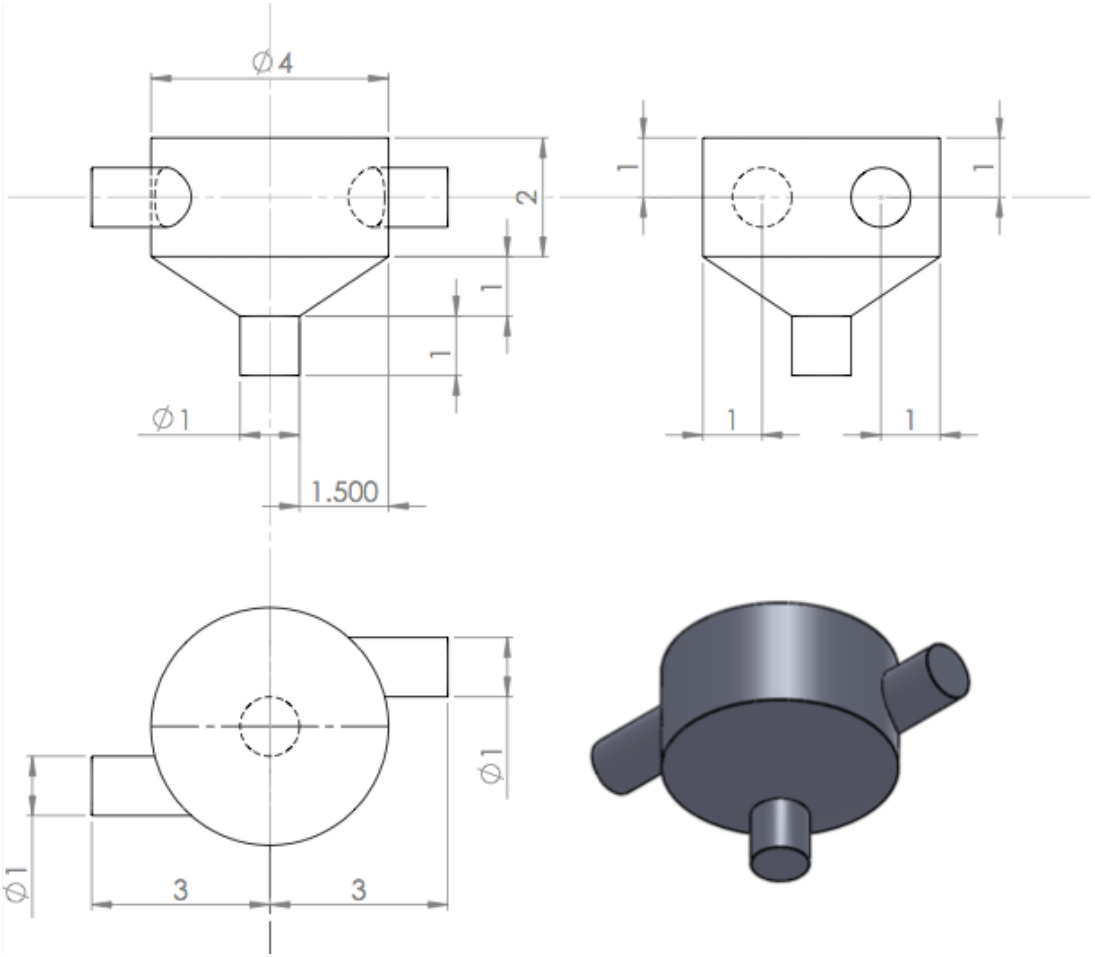


All the dimensions are in centimeters

Geometry generation using open source tools

Geometry 2. Static mixer.

Do you take the challenge?. Create this geometry using Salome or Google Sketch-Up and in no more than 10 minutes.



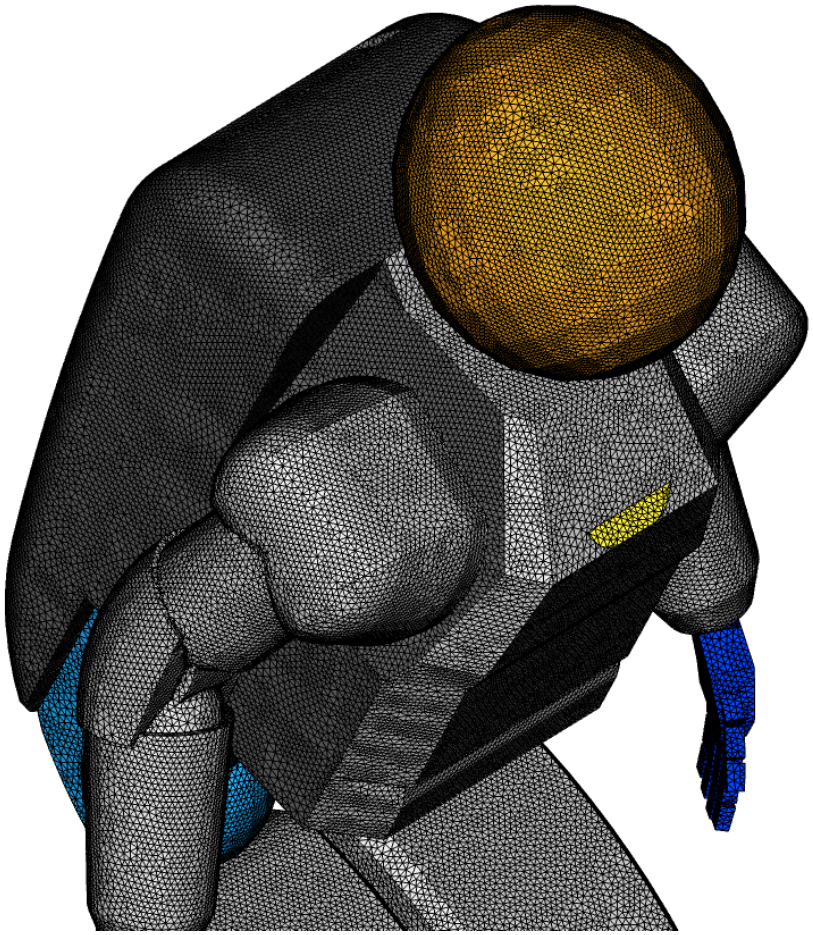
All the dimensions are in centimeters

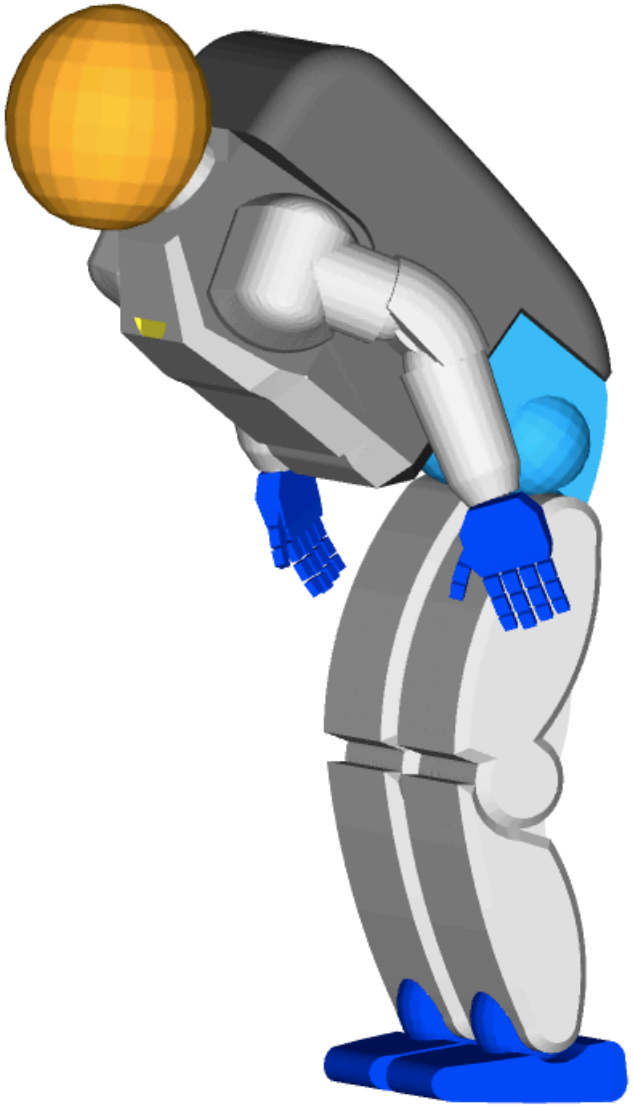
Geometry generation using open source tools

Additional tutorials

In the folder **$path_to_openfoamcourse/geometries_meshers_tutorials,** you will find many tutorials, try to go through each one to understand and get functional using the geometry modeling.

Thank you for your attention

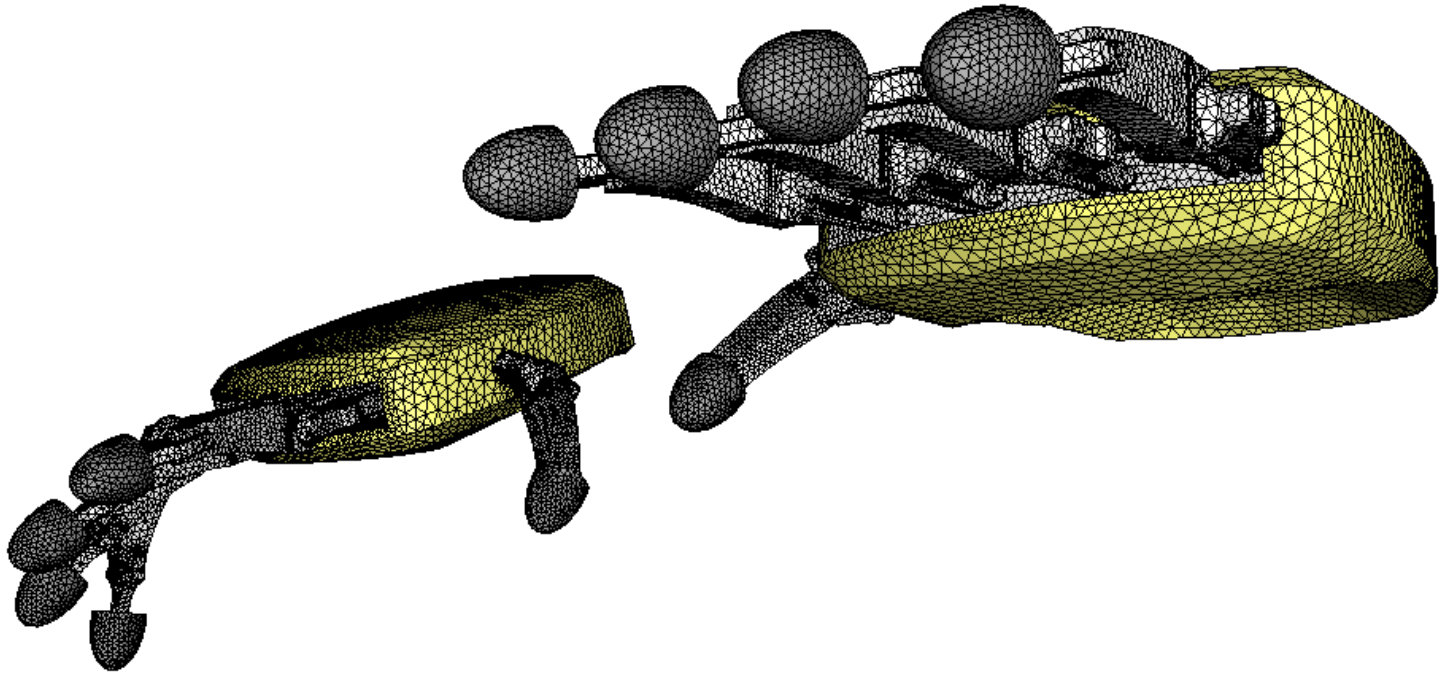




Today's lecture

1. ~~CFD simulation workflow~~
2. ~~Geometry generation using open source tools~~
3. **Hands-on session**

Hands-on session



In the course's directory **($path_to_openfoamcourse)** you will find many tutorials (which are different from those that come with the OpenFOAM® installation), let us try to go through each one to understand and get functional using OpenFOAM®.

If you have a case of your own, let me know and I will try to do my best to help you to setup your case. But remember, the physics is yours.