THE CHALLENGE OF MANY DIFFERENT APPLICATIONS SOLVED BY ANSA

Tom Fahner, Roland Broers
Actiflow B.V., The Netherlands

KEYWORDS –
Variety, workflow, challenge, CFD, consultancy

ABSTRACT –
Actiflow is a Dutch CFD Consultancy company involved in many markets. Although the application is almost always CFD, our clients stem from various markets and each market and company has their own specific requirements. This means a large variety in CAD models, a large variety in level of detail and a large variety in solver physics, which results in a large variety of mesh requirements.

The main challenge is to have a standard workflow that is flexible enough to handle the variety in projects that we see, while still resulting in an efficient time management. ANSA is the software packages that works, since the workflow is always similar, from CAD import to clean-up followed by surface meshing and volume meshing finally to output of the mesh and solver settings, with flexibility in all steps.

Important tools for clean-up and a clear interface with a lot of meshing options: ANSA provides us with the right set of tools as shown in (1) to work with large differences in applications without having to reinvent the wheel every time. This bottom-up approach works well and helps in communication between the engineers whenever they see a new model outside of their own specialty.

This presentation will show the differences in our projects and how the repetition in the workflow in ANSA helps our engineers to get a good quality mesh in a short amount of time.

1. INTRODUCTION

Actiflow is an independent Dutch CFD consultancy company and as such Actiflow is working in many different markets. In order to be efficient our engineers need to have a workflow that is efficient and as similar as possible besides the differences in the particular application. As an example, the typical dimensions of some of the projects that we work on are shown in table 1.

Case	Minimal length (mm)	Maximal length (mm)	Domain size (lxbxh)	Typical flow length scale
Blood heater	0.018	0.862	0.041m x 0.0093m x 0.16m	2 mm
City center	10	50 000.0	2000 m x 2000 m x 600 m	5 m
Nuon Solar Car	0.245	1000.0	46 m x 28 m x 12.8 m	5 cm
Chocolate production facility	4.490	496.8	109 m x 54 m x 5.2 m	5 cm
Lock near a port	1.560	2995.0	271 m x 165 m x 21 m	1 cm – 1 m
Aircraft fuselage	0.007	100 000.0	1350 m radius sphere	10 m

Table 1: Typical problem dimensions for Actiflow projects

Table 1 shows already the differences in flow problems that are carried out by Actiflow. The variation is also shown in the markets that Actiflow is active in:

1. Building industry
    - Wind comfort (pedestrians/balconies)
    - Wind loads
    - Exhaust fume studies
    - Indoor climate
    - thermal comfort
    - Fire and smoke simulations
2. Aerospace industry
    - Gliders
    - Rocket igniters
    - Load calculations
3. Automotive industry
    - Drag reduction
    - Downforce optimalisation
    - Thermal load studies (underhood/brake cooling)
    - Soiling
4. Industrial processing
    - Mixing simulations
    - Fluidized beds
    - Combustion
    - Pipe flows
5. Maritime and coastal technology
    - Lock filling
    - Smoke dispersion around ships/offshore structures
    - Sloshing studies
6. Medical industry
    - Operating room clean area design
    - Incubators
    - Blood heaters
    - Steralisation chambers

<u>The problem of the mesh</u>
This also means that the type of flow problem varies a lot. From laminar incompressible single phase single species steady state simulation problems up to time dependent transonic reacting flows, but also multiphase flows, both segregated and dispersed. Furthermore there are buoyancy driven flow problems including radiation and also some chemically reacting flows.

Not only that, the types and cleanliness of the CAD from our customers varies from optimised for CFD to sketches useful for visualisation. This means that most of the time significant clean-up is necessary and in some cases large parts of the geometry have to be (re)made by our engineers.

Each project therefore requires a different solver, mostly one from OpenFOAM®[1](2). For each solver and each project there are different demands on the mesh. This means that in order to be efficient the engineers must use a robust mesher that works for all these different cases and provides the right mesh with good quality without having to find new workflows for each project.

2. THE USE OF ANSA TO COPE WITH THE VARYING DEMANDS

OpenFOAM's own mesher, snappyHexMesh is a good mesher especially when the work you do is repetitive and when the subject of you study always has similar geometry. This means

1 *OPENFOAM® is a registered trade mark of OpenCFD Limited, producer and distributor of the OpenFOAM software via wwww.openfoam.com. This offering is not approved or endorsed by OpenCFD Limited, producer and distributor of the OpenFOAM software via www.openfoam.com, and owner of the OPENFOAM® andOpenCFD® trade marks.*

you can generate a working set-up once and use this over and over for each new project. With the vast differences in each project, the time needed to get a good quality mesh for each project using snappyHexMesh takes too much time. Instead in ANSA the workflow is always the same.

Furthermore viscous layer meshing in snappyHexMesh is troublesome. With ANSA over 95 % of the time the prismatic layer can be generated without much trouble and with high quality volume elements in the first go.

Whatever the scale of the problem, the required fineness or coarseness of the mesh follows from local curvature, distances and proximities. This means that after some geometry clean-up, using the auto CFD spacing combined with some refinement boxes gives a nice starting mesh point distribution on the perimeters. After some small tweaking the Mesh>CFD see (1) algorithm gives a first mesh which can be improved locally if necessary using standardized quality settings in the hidden mode view for meshing. From this high quality surface mesh the generation of the volume mesh is straightforward. The use of prismatic layers to capture the boundary layer and subsequently automatic detection of the remaining volume is robust to let ANSA do the work, while working on a different project during the mesh generation period. Than just 1 or 2 volume mesh quality improvement steps and the mesh is ready to be exported.

<u>The various mesh types</u>
The most commonly used mesh type is the hybrid prism-tet mesh, but in some cases quad surface cells are combined with the layers algorithm, followd by hexainterior and a conv2poly to generate a combination of hexahedral and polyhedral volume elements that typically have better orthogonality and higher accuracy for the same amount of cells. In some rare cases a pure hexahedral mesh using the hexablock mesher is used. The conv2poly also helps to change the hybrid prism-tet mesh to a polyhedral mesh, but this does not always provide a high quality mesh in the boundary layer for complex geometries.

3. SOME EXAMPLES FROM THE WIDE RANGE OF PROJECTS
The following examples show some of the meshes that have been created, with a short explanation on the particular project.

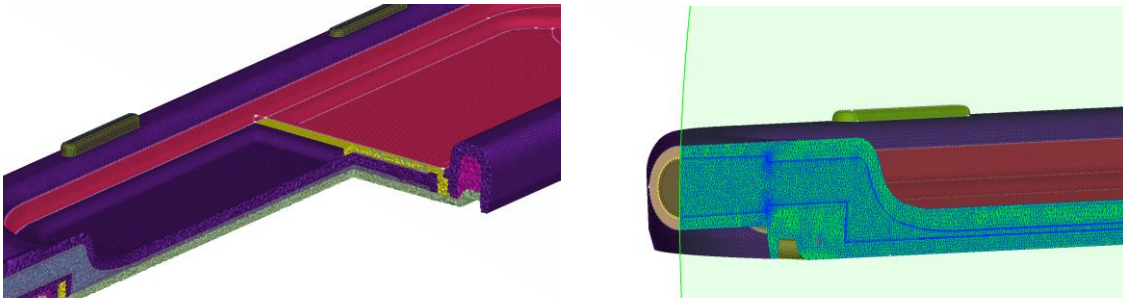

Figure 1 – A mesh for a blood heater

Figure 1 shows two visualisations of the mesh that was used for the simulation of a blood heater. This device is used in hospitals to warm blood up to body temperature during operations. The conjugate heat transfer simulation involved several different regions to represent the various materials and flow regimes. During the project several designs were analyzed and special care was taken not to introduce any hotspots.

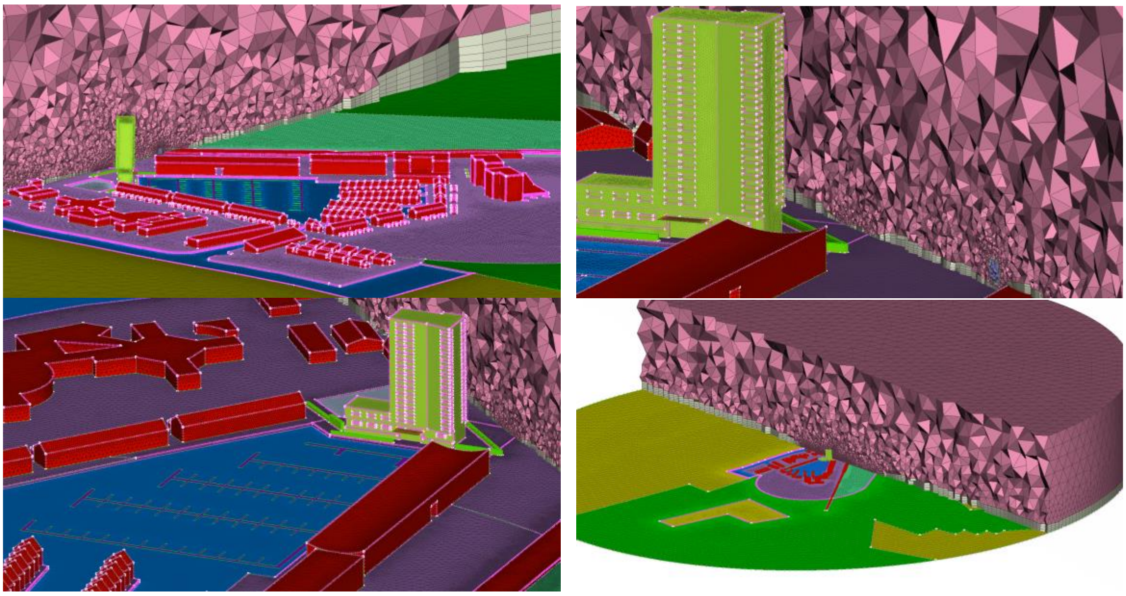

Figure 2 – A mesh for a city

Figure 2 shows part of a city in the Netherlands where a new high-rise building was planned close to a small harbour. The project was aimed at the investigation of the windclimate in the current situation and with the new building present. There is an influence of different roughness elements in the surrounding and from trees and small hedges. Some design modifications to improve the wind climate were investigated for wind coming from 12 different directions. The statistical distribution of wind over these 12 directions was used to give an expected level of comfort throughout the year.

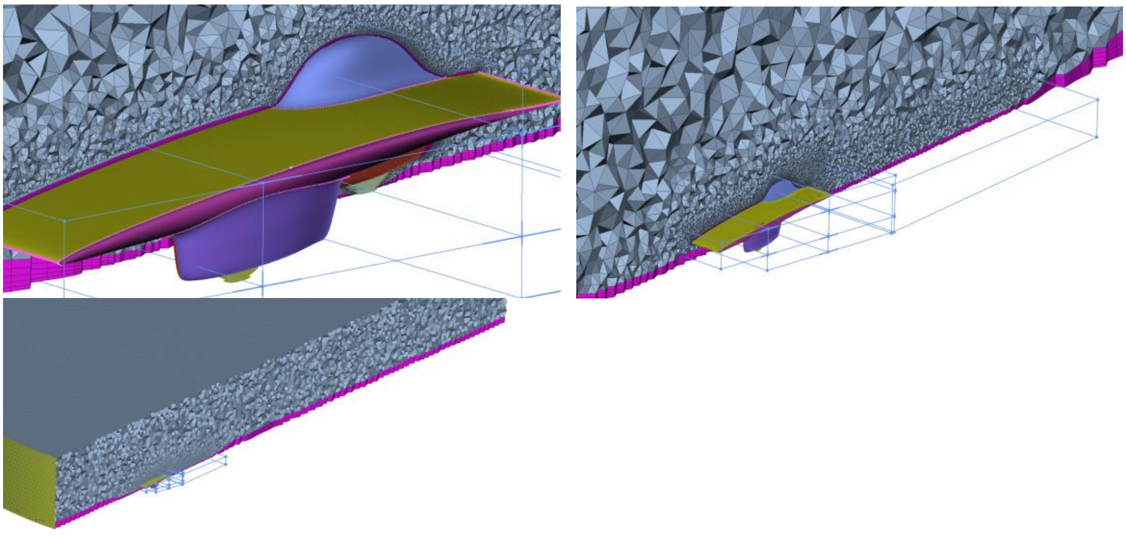

Figure 3 – A mesh for a solar racing car

Figure 3 shows the mesh around one of the cars from the Nuon solar team, the succesful solar racing team from the Delft University of Technology. During the design phase the drag of the new car was reduced significantly compared to the car of the year before. Several design iterations have been studied.

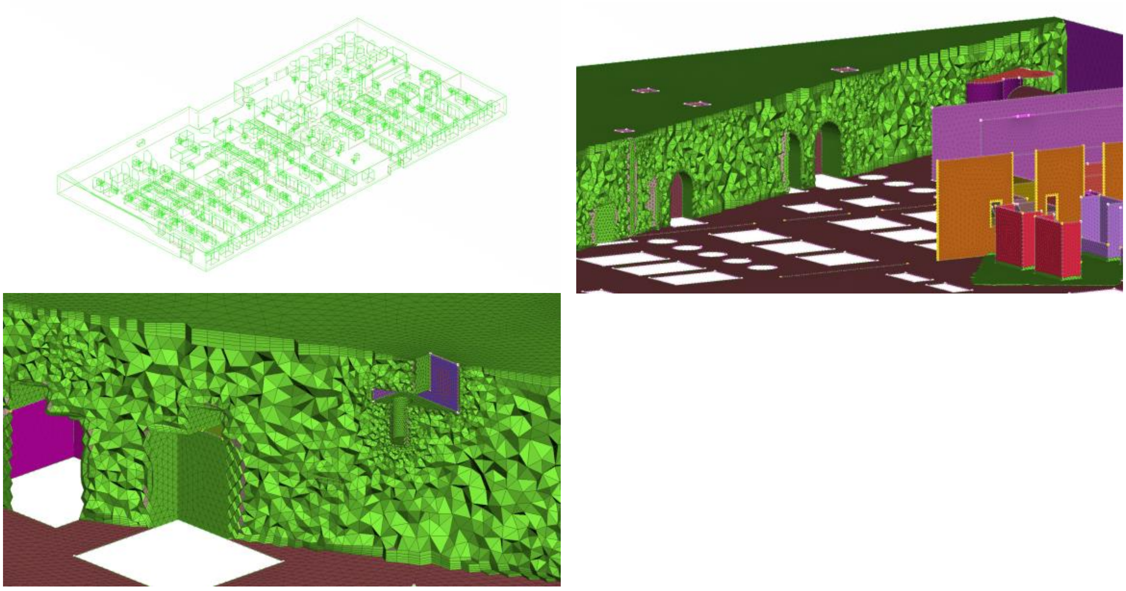

Figure 4 – A mesh for a chocolate factory

Figure 4 shows the lay-out of a chocolate factory and two visualisations of the mesh. The factory hall contains many of hot production locations, while the workers are working next to them. For their comfort a ventilation system was designed that cools the working areas. While expanding the factory, the ventilation system was no longer working correctly to cool in all areas. The thermal climate and the complex ventilation system where studied in separate simulations. First the ventilation system was studied in order to find the correct boundary conditions for the thermal comfort simulation. Some ideas for improving the thermal climate were simulated and later tested in the factory.

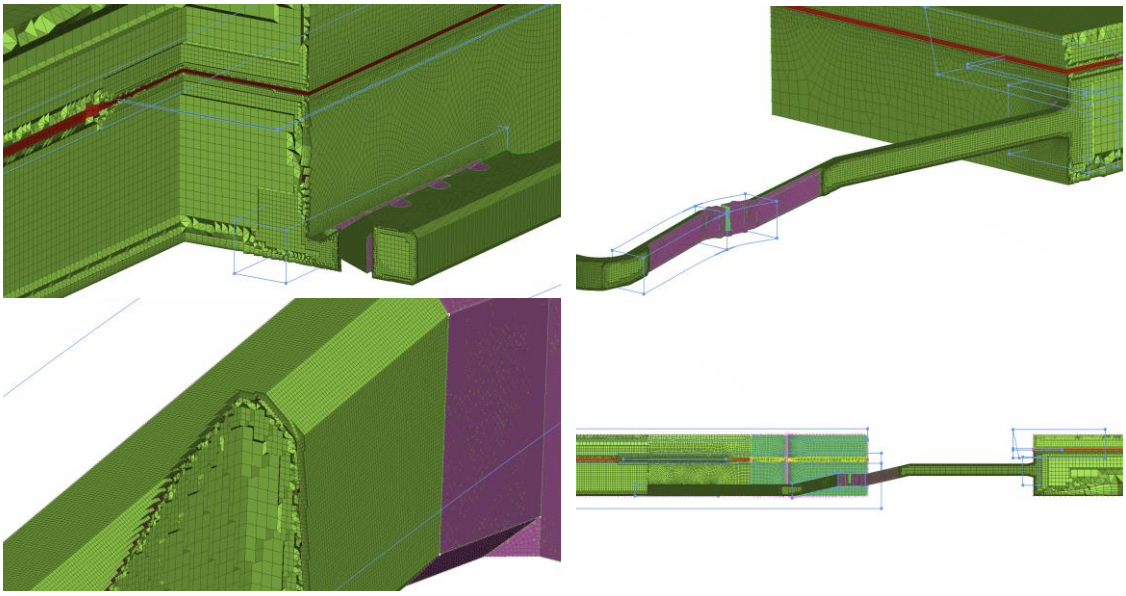

Figure 5 – A mesh for a simulation of a lock

Figure 5 shows some visualisation of the mesh used for the flow in a lock. In order to use the VOF model of OpenFOAM a nice structured mesh was created near the interface between air and water, which differed between cases. The goal of the simulations was to find typical flow patterns in key moments during the filling and emptying of the lock, making sure any ships present in the lock would not be subjected to large forces. A particular challenge was the requirement of high resolution near the interface and a good capturing of the boundary layer in the connecting channels between both sides of the lock doors. The combination os several meshing algorithms and steps helped in creating this complex mesh.

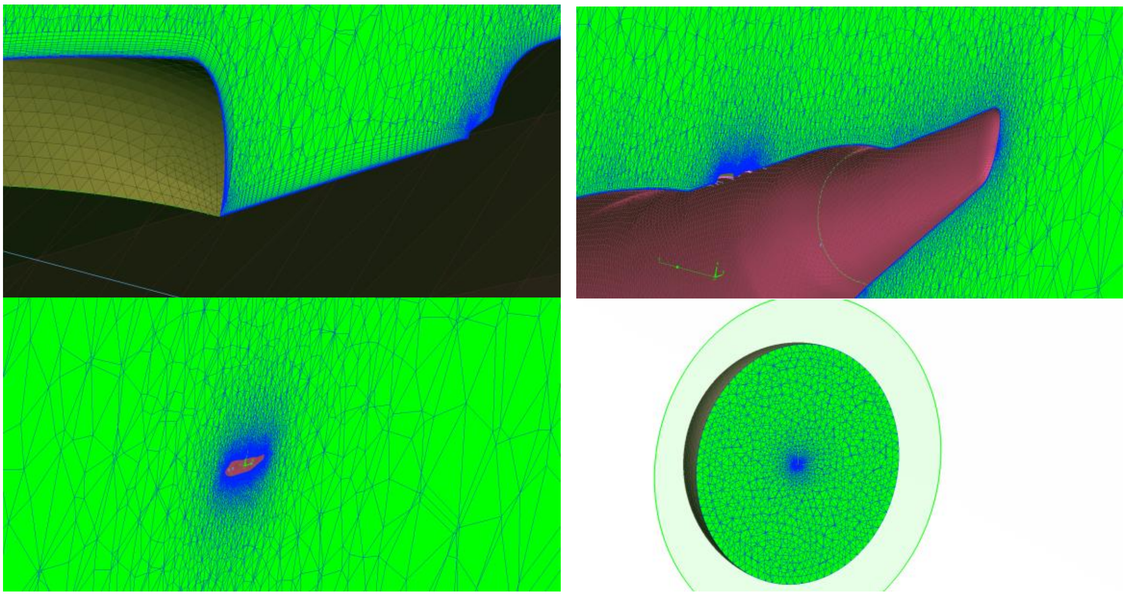

Figure 6 – A mesh for the simulation of transonic flow around an aircraft

Figure 6 shows the mesh around an aircraft, where few antennae where added to the fuselage. CFD simualtions where required to investigate local shocks and pressure distribution around the antennae in order to satisfy maximal loads for the construction along several points of the flight envelope. The simulations where performed using SU2 (3), which requires a boundary layer resolution with y+ of 1, meaning that the first cells of the wall have a height only a few micrometers high, while a spherical domain with a radious of about 1.35 km was used.

4. CONCLUSIONS

Actiflow uses ANSA in order to keep the workflow for the engineers constant even if the subject of study and type of simulation varies a lot. The large flexibility in meshing algorithms and the many CAD tools available make it our choice to have an efficient and similar workflow for each new project. This robust workflow makes it possible to deliver correct results in a timely manner.

REFERENCES

(1) ANSA version 17.0.0 User’s Guide, BETA CAE Systems, July 2015

(2) https://openfoam.org/

(3) F. Palacios, M. R. Colonno, A. C. Aranake, A. Campos, S. R. Copeland, T. D. Economon, A. K. Lonkar, T. W. Lukaczyk, T. W. R. Taylor, and J. J. Alonso, “Stanford University Unstructured (SU2): An open-source integrated computational environment for multi-physics simulation and design”, AIAA Paper 2013-0287, 51st AIAA Aerospace Sciences Meeting and Exhibit. January 7th - 10th, 2013. Grapevine, Texas, USA