

Use of Open-Source Solver Technology for Design-related CFD

Ulrich Heck, Martin Becker

DHCAE Tools GmbH, Krefeld, Germany

Summary:

The Open-Source CFD Toolbox OpenFOAM has very good prerequisites for a usage in CAD oriented CFD processes. This includes e.g. the strong encapsulation of the respective CFD solution capabilities into individual solvers. The solver approach simplifies using CFD as an application-specific tool: In contrast to the multi-purpose CFD systems, the designer can focus more easily on problem-specific settings for his specific area of application. The open-source structure of OpenFOAM also facilitates the adaptation of CFD methods and solution procedures, e.g. by a CFD expert to the designer's requirements in terms of model capabilities or individual workflow.

On the other hand, OpenFOAM's hurdles for direct deployment into the design-related environment are the relatively complex application of the system, the comparatively poor integration into a CAD- and GUI-based workflow, and the intended work on a text file base. Thus, OpenFOAM appears to be generally unsuitable for widespread use in the design environment. However, in combination with additional third-party tools, used as wrappers around the actual CFD solver OpenFOAM, these deficits may be overcome: They retain the advantages of the CFD solver in the design-related environment, on the other hand reduce or even eliminate the disadvantages that arise from the direct use of OpenFOAM. These tools can then be adapted to a high degree for the individual needs of the designer.

The use of OpenFOAM's solution capabilities for the designer becomes interesting especially in the case of recurring problems with similar flow conditions and objectives. However, even in the case of complex CFD problems, such as multiphase or multi-physics applications, which are to be implemented as simply and automatically as possible, OpenFOAM has particularly strong capabilities. These solution capabilities are easily expandable by CFD experts and can be made available to the designer.

Different levels for automation are presented for a usage of OpenFOAM in combination with the third-party tool CastNet: This covers a pure template approach for solution settings without geometric association to CAD model entities as well as abstract modelling techniques. With the help of abstract modelling it is possible for certain CAD systems to fully define the CFD model in the design program and also for design changes to continue to use the existing settings and geometric references.

Keywords:

CAD, CFD, OpenFOAM, Open-Source, CastNet, Workflow, Abstract Modelling

1 Background

The integration of CFD into the design engineering process faces a number of specific challenges. These challenges result from pre-conditions given by the design engineer knowledge, CAD-model related issues and workflow requirements:

Design engineer:

- The design engineer has typically only restricted knowledge of numerical methods and their usage.
- In most cases, the design engineer focuses on very specific applications.
- The design engineer is used to work rather with graphical user interfaces than text file edited systems.

CAD-Modell:

- The input models are presented by manifold CAD models and assemblies.
- The models represent the solid parts and not the fluid domain.
- The model includes a large number of geometric details (dirty CAD), which are typically not relevant for the flow.

Workflow:

- Often a larger number of designs must be investigated for optimization.
- A Windows environment is in general state of the art.

These conditions result in a number of requirements for the CFD tool. To satisfy the needs of the design engineer and considering his typical knowledge and working style, a CFD application with predefined solution methods, automated meshing and automated case setup for single purpose CFD in a graphical user interface is desired. To consider the CAD model or assembly input, an automated generation of non-manifold models is required. Often techniques for cleaning and automated generation of fluid domains based on solid models are needed. Finally, to fulfil the workflow requirement some degree of reusability of CAD models and automated workflows on Windows are indispensable.

2 OpenFOAM

The Open-Source CFD Toolbox OpenFOAM became more and more popular in last years. OpenFOAM (Open Field Operation and Manipulation) is a CFD Toolbox to simulate complex fluid flows involving chemical reactions, turbulence, multiphase flows and heat transfer. The base of OpenFOAM has been developed at the Imperial College London. OpenFOAM is freely available and open source, licensed under the GNU General Public Licence [1]. The major advantages of OpenFOAM are:

- Very strong solution capabilities covering a wide range of flow phenomena
- Fully transparent (source code is part of the system)
- No license costs
- Designed for customization and for implementation of extensions

However, it is regarded to be too complex to integrate the system into a typical CAD oriented workflow directly: The original OpenFOAM requires a text-file editing for the case setup with difficult to understand entries or keywords while the learning curve is steep even for advanced CFD users. The meshing is based on discrete data input (e.g. stl) and the original versions of OpenFOAM run only on Linux OS.

On the other side, OpenFOAM offers some advantages, which represent a promising base for a CAD orientated usage:

- The so-called "solvers" in OpenFOAM encapsulate a specific range of physical models and therefore they can be used for specific applications. E.g. if a designer has to focus on pressure losses in steady state flow with constant physical properties, he only needs a single specific solver in this toolbox (in particular simpleFoam). Due to specific applications the design-engineer typically works on, there is no need to be familiar with the complete solution

capabilities in wide range of solvers: He can focus on his single solver for his specific application.

- The OpenFOAM meshing tool snappyHexMesh allows building meshes for the flow domain even if no fluid region representation exists in the CAD model.
- The text-file oriented case setup and scripting capabilities in OpenFOAM represent a perfect base for the generation of automated workflows by a CFD expert and hand them over to the design engineer.

3 OpenFOAM in combination with CastNet

Beside the base functionality OpenFOAM already provides, additional third-party tools exist, which present wrappers around the CFD solver and offer additional functionality for a usage in the design-related CFD workflow. One of these third-party tools that provides additional functionality in this context is CastNet; a GUI with CAD-kernel based case setup and meshing functionality for OpenFOAM [2].

The drawbacks of OpenFOAM for design engineers of complex structure and text-file editing can be overcome with the use of CastNet: CastNet allows setting up OpenFOAM cases directly, including the definition of solver settings, job control parameters, materials and boundary conditions. CastNet exports all files OpenFOAM needs to perform the analysis and is able to export special mesh zones directly. Exported cases uses only functionality provided by OpenFOAM, so the complete compatibility is established. The definitions of OpenFOAM settings are also associated with the CAD-geometry (e.g. boundary condition related to geometry faces). This allows exchanging these data easily to perform analysis for a number of CAD-models.

Beside the pure pre-processing and case setup system tool an additional monitoring is provided. This tool named "RunGui" enables a detailed convergence control e.g. by residual plots and monitor value graphs. Furthermore, it allows to change solution setting such as schemes or relaxation factors during runtime. This can be done either manually or automatically, e.g. to have solution settings for stable start and accurate end results.

3.1 CAD-Input and meshing

CastNet is based on CAD preparation, meshing and abstract modelling of Simmetrix Inc [3]. The base system "SimModeler" has been extended with adapted GUIs and added solver functionality, in particular to support open-source solvers such as OpenFOAM (CFD) and CalculiX (CSM).

CastNet reads CAD kernel data (Parasolid, ACIS, Granite) or discrete mesh data. The CAD kernel import results on the one hand in good quality of the imported models. On the other hand, it allows to conduct manipulations during the import: For a manifold CAD assembly the non-manifold can be generated automatically resulting in single interfaces between the parts of the assembly. This simplifies the generation of conform meshes while still preserving the multi-region capability (e.g. for porous, MRF or CHT zones).

Basically, two meshing approaches are supported:

- Internal hybrid meshing approach:

The hybrid meshing uses the Simmetrix meshing technology. This meshing approach requires a fluid domain as CAD input.

The user defines meshing parameters (e.g. mesh sizes, boundary layers) with respect to geometrical objects. The complete meshing process is optimized considering all meshing parameters for surface meshing, boundary layer meshing and volume meshing. The hybrid meshing approach includes powerful boundary-layer meshing capabilities and a hex core meshing.

Using relative element size and curvature control attributes CastNet generates meshes complied with the numerical models without the need of defining many mesh parameters. CAD clean up functionality (e.g. to suppress sliver faces or partial sliver sections on faces) are provided.

- External hexahedral dominant polyhedral meshing based on snappyHexMesh

Beside the hybrid meshing, a hexahedral dominant meshing based on the OpenFOAM utility snappyHexMesh is integrated into the graphical user-interface CastNet. The definition of the mesh parameters is associated with CAD entities: The user e.g. can pick CAD faces for local refinement

or boundary layer definitions. Based on the user settings the snappyHexMesh-dictionary is generated. The external meshing process is integrated into an automated workflow. For the meshing approach with snappyHexMesh in CastNet, no fluid domain is required, and the mesh input can be build based on the solid assembly. Only few mesh settings are typically required for standard cases, see figure 1.

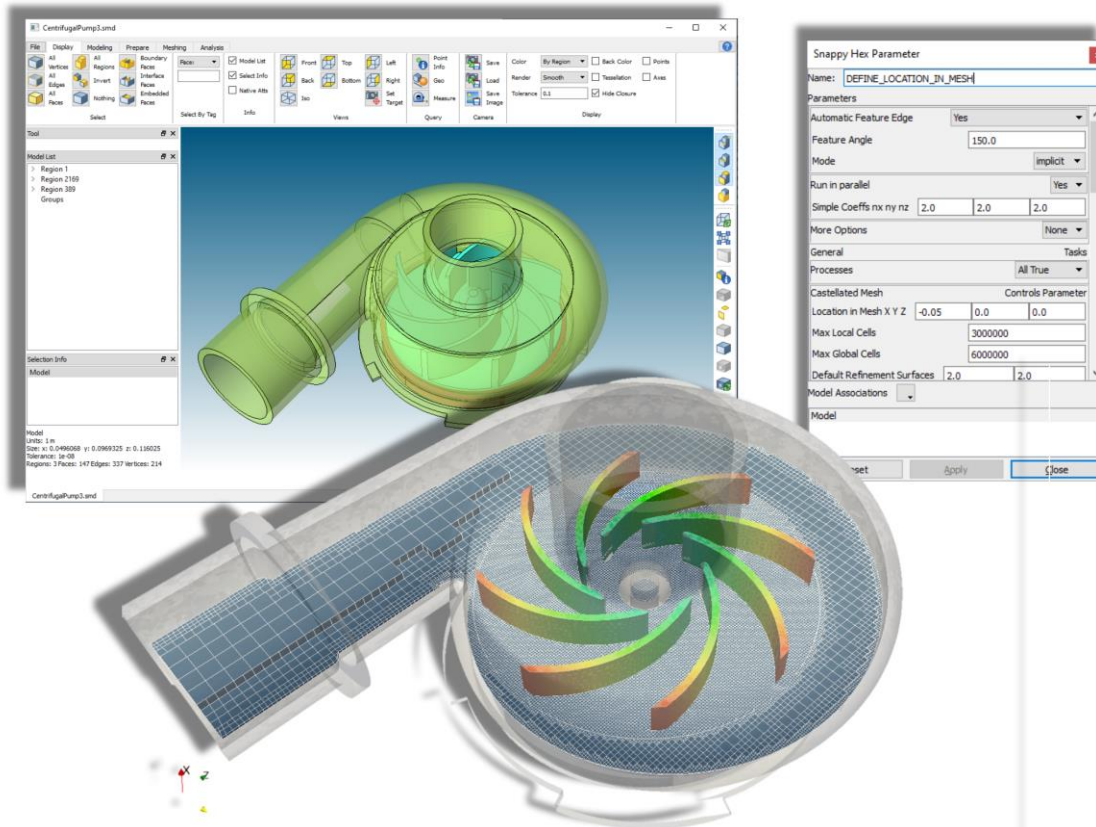


Figure 1: GUI based setup for a pump - meshing with snappyHexMesh (hex-dominant mesh) based on CAD solid domain input.

3.2 Template approach

CastNet uses a template approach to simplify the case-setup for OpenFOAM and to apply the mesh settings. The template approach can be adapted to the specific needs, knowledge and grade of automation for the design engineer:

- In the first stage, the design engineer can select a standard template for the CFD specific solver such as simpleFoam for steady state, incompressible analysis. In this case he must add case specific settings, e.g. associate the geometry faces with inflow conditions and enter the application specific boundary conditions (e.g. inflow velocity).
- The next higher level would be a customized template where all flow model specific settings such as boundary conditions, flow properties etc. are predefined for a specific analysis case. In this case, the design engineer must only associate the settings with geometric entities such as geometry faces for inflow etc.
- The highest level of automation is given by a complete abstract model for a specific application. In this case, all geometric entities (such as inflow faces or specific regions) are found by a predefined logic in the abstract model. Here the design engineer only needs to import his CAD model, apply the abstract model and export the case. All settings to run the case are associated with the current CAD model and result in a run-ready case even in very different CAD variants.

The template approach covers several needs of a CFD-tool in the design related usage. First the template approach simplifies the case setup: The design engineer can apply application specific predefined settings to his various CAD geometries and does not need to worry about neither detailed solution settings nor numerical methods. Furthermore, the template approach allows efficiency design

studies: The design engineer can simply import his different design cases, apply the template and run the case. This procedure results in a fail safe and reliable design study. The method of the abstract modelling with an example will be discussed in the following section.

3.3 Abstract Modelling for CAD variants

If a larger number of strongly differing CAD-designs must be considered, abstract modelling is the most powerful and most general modelling technique.

Abstract modelling allows defining model attributes detached from exact CAD geometry and then applies it on different CAD-Models.

Advantages and areas of application of abstract modelling are:

- Implementing calculation variants even on very different designs
- The abstract model can already be designed even though no CAD geometry is provided and only a vague concept exists, e.g. position of in- and outlet with complete freedom in between
- Abstract modelling can very early be integrated in the process of developing, optimising, and changing
- The abstract model saves all model- and solver-specific settings, means that the abstract model can be built by CFD experts (In-house or consultant), while the application of the attributes on the CAD model can be done by a trained design-engineer
- Abstract modelling supports building personnel templates e.g. for specific solver settings

There are different options to identify geometric entities. In some CAD systems geometric entities such as faces, regions etc. can be already tagged, and the models can be built based on the tagged elements. Changes can be applied to the tagged geometry in the CAD system and simulation models can be generated directly for the new CAD geometry as long as the tagged elements are available.

Another option to identify the different geometry components is based on logic operations (e.g. Boolean operations between different CAD elements) and search rules (e.g. bounding boxes) for certain designs. An example for this approach and the complete resulting work is discussed in the following section.

4 Example

4.1 Problem specification

A number of designs of a heat exchanger must be investigated. The base designs and boundary condition are shown in figure 2 and figure 3. In this case, the fluid domain can be easily generated in the CAD system and is used as input. Otherwise, a modelling based on the solid domain using snappyHexMesh as mesher would be possible as well.

The inflow and outflow section are identical in all designs (s. figure 2). The design engineer must investigate different arrangements of the heating walls for the middle region. The mesh on heating walls is an important parameter for the result quality. For a good compatibility of the various designs, all heating surfaces should provide the same boundary layer mesh. A free mesh is desired in the middle region to have most flexibility in the design of the heating walls, while extrude meshes should be used in the inflow and outflow section (see figure 3).

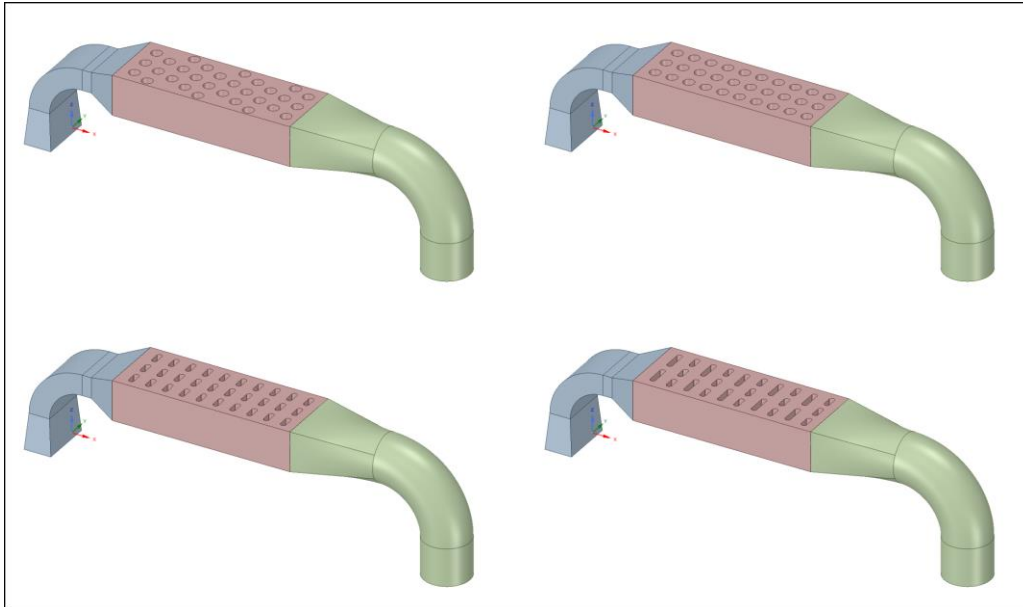


Figure 2: CAD variants in the abstract modelling.

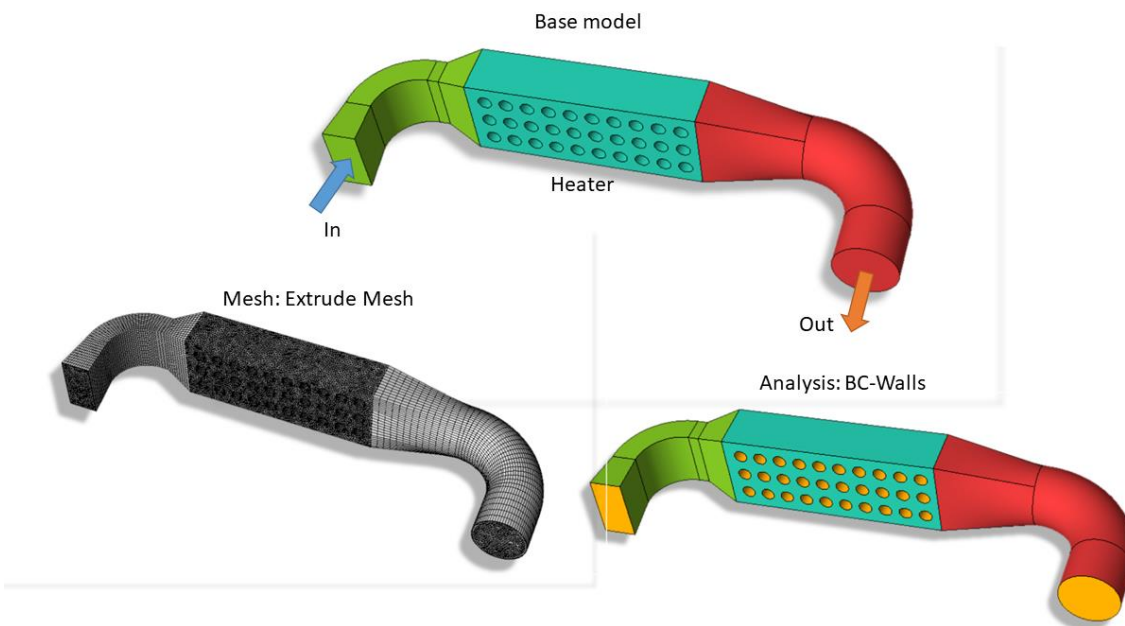


Figure 3: Abstract model requirements.

4.2 Definition of the abstract components

The user defines the identification of abstract components in a special GUI (figure 4).

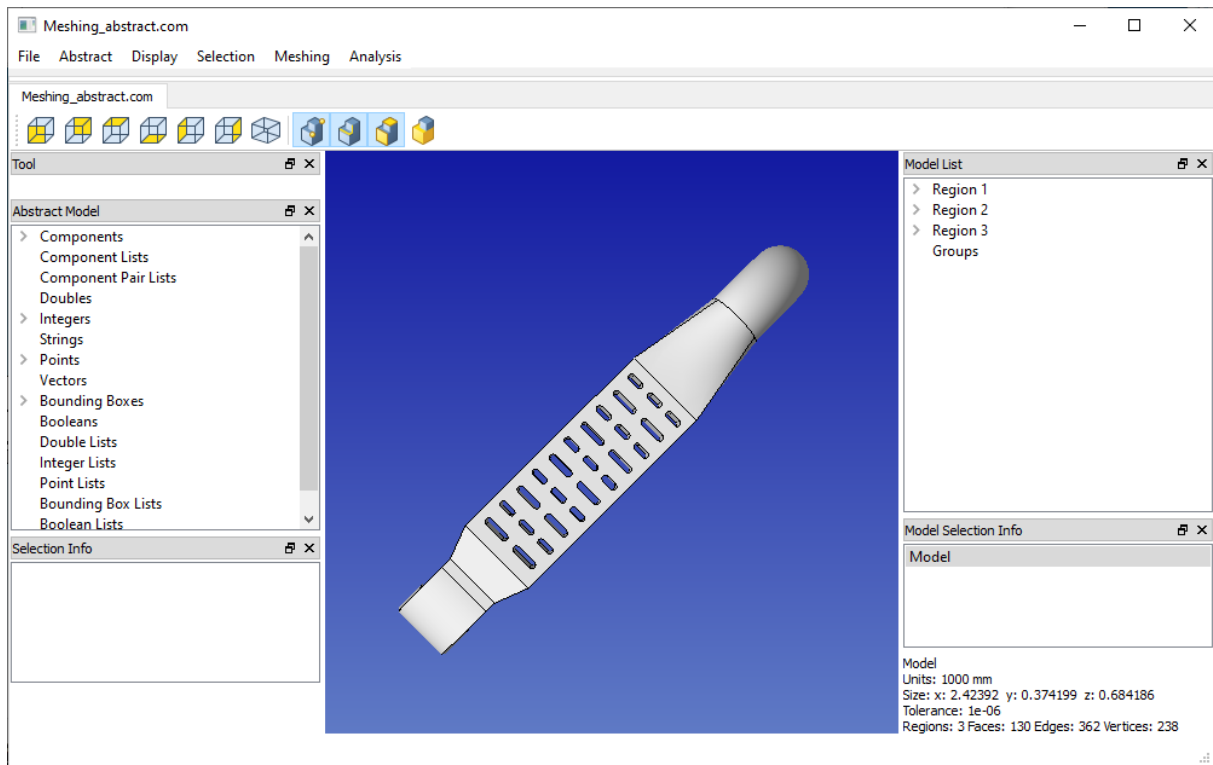


Figure 4: Definition of abstract components.

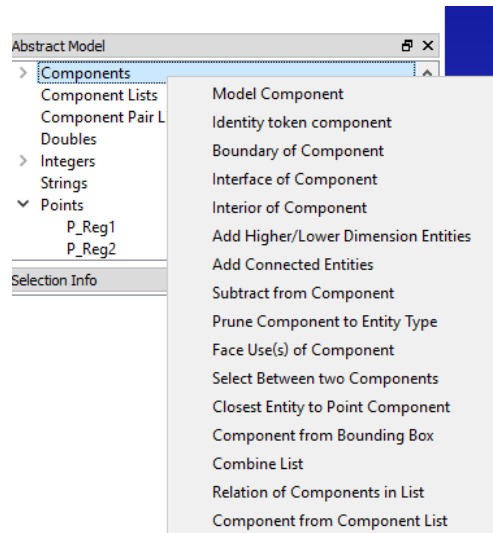
First it is recommended to define a general strategy for the abstract components:

Here the user should consider which entities remain fixed in space for the CAD variants and which are changing. Based on this the user must decide which operation for the abstract component is the best to fulfil the requirements of the topology changes.

Various options and operations to identify the abstract components are available.

Typical operations are:

- Bounding boxes: E.g. to pick all faces within a specified tolerance of a bounding box.
- Closest CAD entities to a defined point
- Loops over geometrical objects, e.g. adding of connected entities
- Boolean operation (e.g. subtract all faces found in a certain bounding box or to identify sharing faces of two regions)
- Geometrical orientation (e.g. all top faces of a certain region)



In our case, we consider

- The inflow and outflow faces to be fixed in space due to some external connectors. So, these components can be identified in the different designs by an operation: “Closest Entity to Point Component”.
- The heating faces differ in number and shape. But they are all located in a rectangular region. Here obviously a bounding box might be good choice to detect the faces.

The user defines the attributes accordingly.

4.3 Connecting the abstract components to CAD entities

When the user loads the CAD models finally and applies the attributes, a connection is established between the abstract components and the real CAD geometry entities (Figure 5). So, the placeholder of the abstract component is replaced by the real geometry faces. This method is applied to all CAD models. Hereby all CAD models get the same meshing and solver specific modelling attributes. The user generates the mesh, exports the case with analysis settings and starts the job.

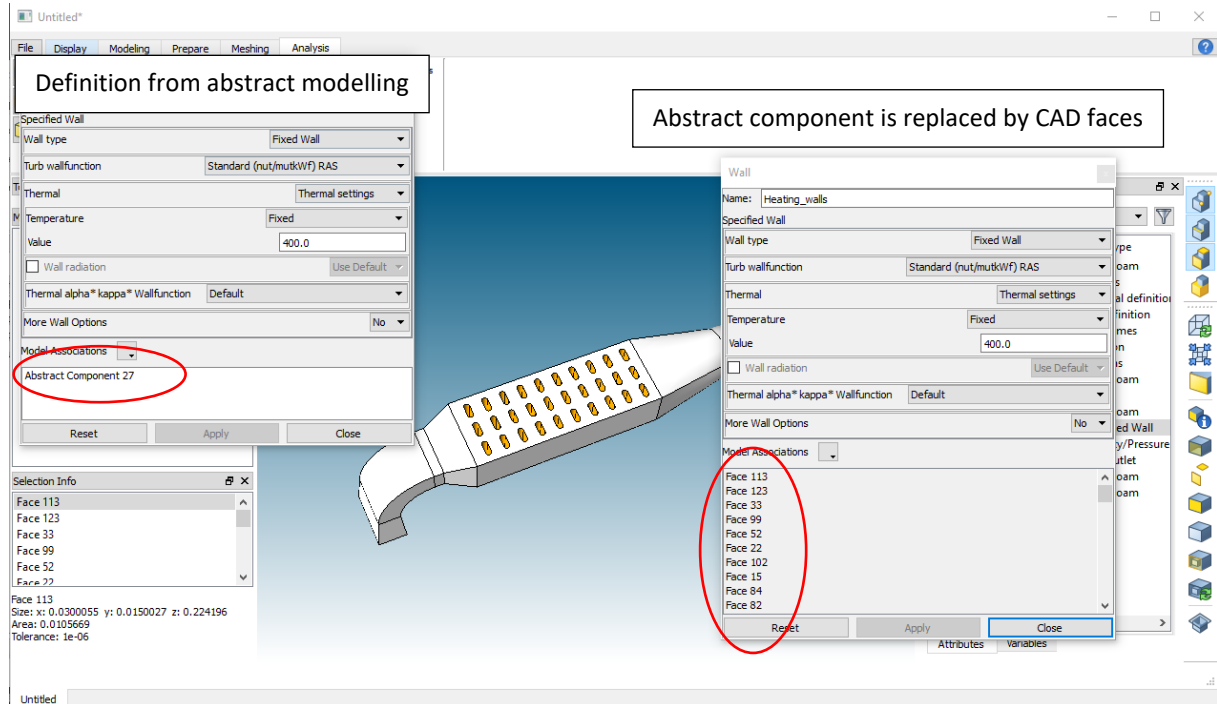


Figure 5: Applying the abstract components to CAD models.

4.4 Solving

As OpenFOAM is originally only available on Linux, a Windows port is used (BlueCFD [4]) to solve the case on Windows OS. The solution process based on CastNet output is fully automated with stability and accuracy improvements: Typically, in a heat transfer analysis the solving of energy can lead to divergence if the flow field is not established. Therefore, the solving of the energy equation is retarded until a stable velocity field is established. Here the base function of OpenFOAM to modify solution procedures during runtime is used. Furthermore, an accuracy improvement is automatically implemented: To establish a stable start, the solution begins with first order upwind-schemes. After some iterations the solution is stable, the settings are updated and the solver switches to higher order upwind schemes for higher accuracy. These run optimizers are integrated into the monitoring tool RunGui, by parsing the OpenFOAM log-files and changing the solution procedure (relaxations factors, schemes) during run-time. So, the design engineering doesn't have to know the CFD-solution procedure with the particular settings in detail. The settings are already prepared in the CastNet template e.g. by the CFD expert.

Furthermore the monitoring tool RunGui allows a detailed monitoring of the simulation: In this case the design engineer can expect a stable solution if the monitored values for temperature at the outlet and pressure at the inlet do not change anymore.

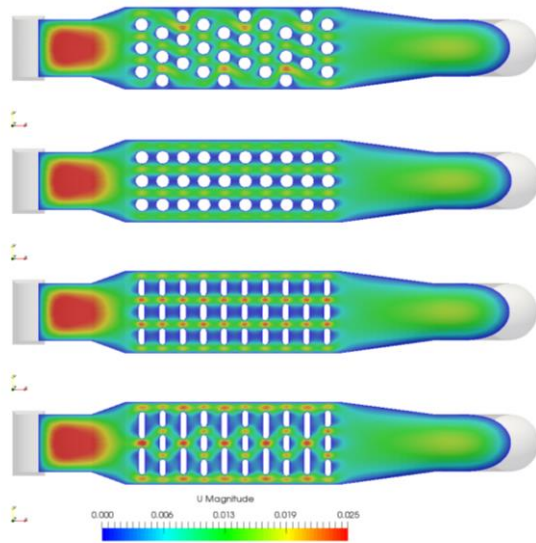


Figure 6: Velocity distribution.

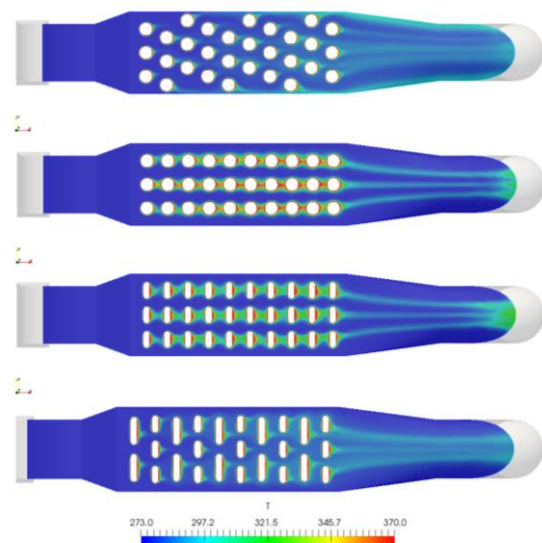


Figure 7: Temperature distribution.

5 Summary

Integration of CFD into the design process has specific requirements, which appear to be hardly met with open-source CFD, in particular OpenFOAM. But in combination with the wrapper CastNet the specific requirements can be met. Following figure demonstrate how the particular workflow can be realized with OpenFOAM and CastNet.

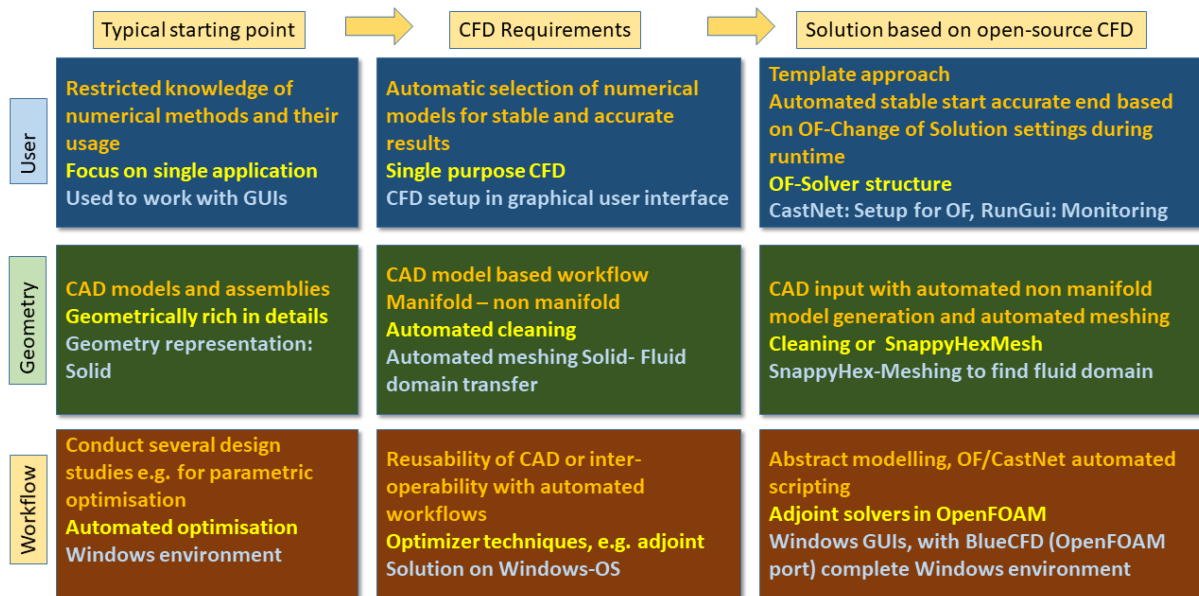


Figure 8: Usage of open-source solvers in a design related process.

- Models can be built and analysed based on the solid domain. No fluid region must be built if the GUI based setup of the OpenFOAM-mesher snappyHexMesh is used.
- The complexity of solution settings can be hidden in a template approach which allows a case related setup for OpenFOAM.
- Design studies can be easily carried out by using the abstract modelling technology. The particular CFD specific requirements for an application are condensed in the abstract model. Applying the abstract model to CAD-geometries, running the cases and visualizing the results for target parameters can be performed by the design engineer without advanced CFD knowledge.

- The simulation runs are optimised by using the run-time modifiability of the solution procedure in OpenFOAM resulting in stable simulations and accurate results at the end.

This finally allows taking advantage of the usage of open-source CFD solution. Particular advantages are: no license costs, transparency, expandability, comprehensive physical modelling capabilities and reliability for the CAD design related CFD area.

6 References

- [1] OpenFOAM: www.openfoam.com (ESI/OpenCFD-release) www.openfoam.org (CFDDirect-release)
- [2] Simmetrix: www.simmetrix.com
- [3] CastNet provided by DHCAE Tools www.dhcae-tools.com
- [4] BlueCFD project: <http://bluecfd.github.io/Core/>

7 Contact/Further Information

Please Contact
Dr. Ulrich Heck
DHCAE-Tools GmbH
Friedrich-Ebert-Str. 368
47800 Krefeld
Phone + 49 2151 821493
Fax + 49 2151 821494

International Website:
www.dhcae-tools.com

German Website:
www.dhcae-tools.de