LAMINAR AIR FLOW SIMULATION OF A COIL WITH REAL GEOMETRY, POROUS MEDIUM, AND GAP MODEL IN FLOWVISION

Tolga Güler
ÜNTES Heating Air Conditioning Company Inc.
Ankara, Turkey

Sinan Soğancı
Capvidia NV
Leuven, Belgium

Mehmet Oğuz Tutkun
Akana Engineering
Ankara, Turkey

SUMMARY

In the air conditioning sector, batteries are essential equipment. For this reason they are included in almost every analysis. In many cases we need accurate solutions to make changes on our HVAC unit design. CFD analysis can show us internal pressure loses and other airflow characteristics of an HVAC unit. During the design process we make many changes on the unit design. At this point CFD analysis provides us economical solution for testing our preliminary designs before the final design. For this reason we need to create correct model for coils in our CFD analysis.

In CFD analysis meshing a coil's real geometry is a very expensive job. You have to create a very dense mesh between coil's lamellas. Creating a dense mesh extends the solution time and consumes too much system resources but it gives more accurate solutions. If you don't have enough system resources then you should try porous medium definition for the coil. For porous medium definition neither a dense mesh nor a high system resource is necessary. In many cases porous medium definition works well if you can define the coil's volume resistance correctly in three dimensions. Porous medium solutions are less accurate then real geometry solutions.
Here we must make a decision between accuracy and resources.

Third option is FlowVision's unique feature named "Gap Model" which provides the real geometry solutions without need for dense meshing.

Current study covers comparison of these three different coil definitions according to their positive and negative aspects.

NOMENCLATURE

HVAC	Heating Ventilation Air Conditioning
CFD	Computational Fluid Dynamics
SGGR	Sub-Grid Geometry Resolution

INTRODUCTION

In the current study, three different CFD simulation approaches for a cooling coil are evaluated in comparison to each other and further assessed with respect to outputs gathered from a coil selection software.

The main intention of realization of this work is to contribute, by sharing the experience, to accumulated knowledge among CFD simulation engineers working in the HVAC industry.

Design and performance of heating/cooling coils play a vital role in determination of thermal performance and energy efficiency of HVAC systems. Although there are design and/or selection software available on the market, any kind of deviation from the standard geometries inevitably leads to obscurities in design cycle. In such cases, CFD simulations, comprised of momentum and heat transfer and in some cases with conjugate heat transfer calculations including the solids, are the leading solutions for design evaluation and performance optimization. However, the complexity of coil geometries with relatively small (generally 1.6 - 5.4 mm) gaps between lamellas turn out to be a significant problem for generation of computational grids.

The main target followed in this study is elaboration of possible grid generation and corresponding simulation methodologies for calculating pressure drop and heat transfer through a cooling coil. Resulting pressure and temperature distributions accompanied with user and computation resource requirements are the main parameters of interest for the assessment of different approaches.

FlowVision (1) (Capvidia, Belgium), a general purpose commercial CFD software package, is utilized for the CFD calculations. Having a C++ implemented solver based on finite-volume method, FlowVision covers 2D/3D inviscid and Navier-Stokes formulation for laminar and turbulent flow regimes accompanied with various physical modules such as heat and mass transfer, phase interactions, chemical reactions and ablation. Grid generation starts with a Cartesian initial grid followed local and dynamic adaptations. At the same time, CAD boundaries are resolved by SGGR (2) technology, allowing to deal with complex geometries (such as the coil used in the scope of this study) without sacrificing the accuracy. Gap Model (3) enclosed in the software avoids the necessity to resolve the small (down to sub-microns) clearances with grid elements, resulting in a significant decrease in total number of cells.

Friterm Standard Product Selection Software (4) is used in this study to make bulk analytical calculations of air and water inlet/outlet conditions for the same coil design which is also used in CFD calculations. The software, being certified by Eurovent, is widely accepted and used among the HVAC and energy industries.

CASE DESCRIPTION

The case studied in this work consists of a cooling coil with outer border cross-section size of 0.46x0.2 m and width in flow direction of 0.065 m (Figure1). The coil is located in a rectangular duct with the same cross section and 2.065 m length.

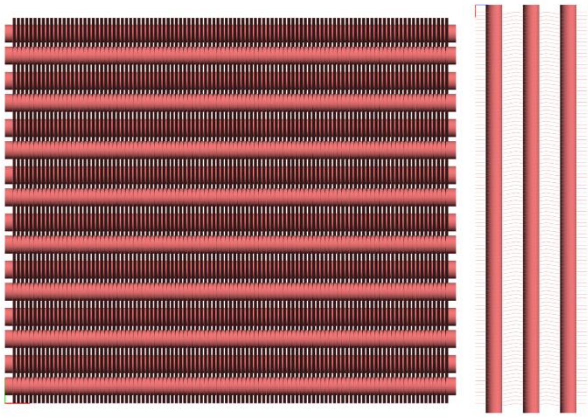

Figure1 Coil geometry (Left; flow direction, right: top view)

Air is modeled as ideal gas with inlet boundary condition of 0.1044 kg/s mass flow rate and initial velocity of 0.95 m/s, being correspondent with each other. Pressure in duct outlet boundary condition and the whole region initial condition is defined to be atmospheric. Temperature is assigned to be 27°C at inlet and upstream of coil whereas it is 15°C at downstream and zero gradient at outlet. Considering the cooling water inlet and outlet temperatures; temperature of coil surfaces are defined as a function of height, being 7°C at the top and 12°C at the bottom (Figure2).

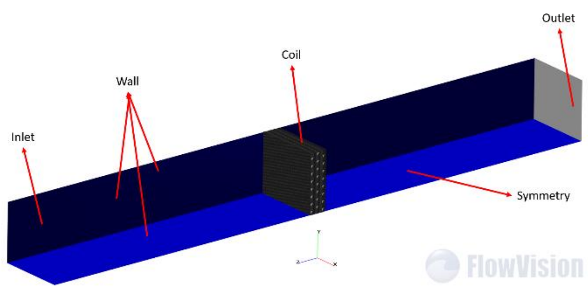

Figure2 Boundary conditions

APPROACHES

In this study, three different CFD approaches are followed and additionally an analytical calculation is carried on using a coil product selection software.

In all CFD simulations, half sections of duct and coil are used with the help of symmetry boundary condition, consequently decreasing the computational requirements.

Coil resolution by grid

In this approach; initial grid consists of cells with sizes equal to 6.72, 26.88 and 26.88 – 53.76 mm in x, y and z directions respectively. Following the initial grid generation, 4^{th} level of adaptation is applied in the vicinity of coil, resulting in computational cells with sizes of 0.42 ($6.72 / 2^4$) mm which ensures 5 cells between each lamella pairs which are located with distances of 2.1 mm. In addition to that, multiple (6-8) layers of 1^{st}, 2^{nd} and 3^{rd} level adapted cells are used in the upstream and downstream of coil, for the purpose of capturing flow gradients. Resultantly, the computational grid has 5.78 M cells.

Advantages: Using actual geometries of coils in CFD analysis, being very close representations of real life applications, provides more realistic and more accurate outputs which are also in high acceptance with experimental results.

In this way we can see the pressure, velocity and temperature distributions on coil's surface and vicinity very close to physical applications. So we can prescience the undesired conditions or results before occurrence. For example if there is not a homogeneous velocity distribution on coil's surface, it can be inferred that that there will be some heating or cooling capacity loses. On the other hand, the airflow characteristics such as turbulent or laminar airflow regions can be observed via CFD calculations where real coil geometries are resolved. By using these characteristics, acoustic calculations can also be performed as a post process.

Enabling the heat transfer equations, heat flux and temperature distribution on coil can be examined. Depending on the thermal boundary conditions, temperature distribution of air stream and the downstream temperature changes are to be observed in detail and with relatively high spatial and numerical accuracy.

Disadvantages: Different objects and geometries are used daily in CFD analysis. In some cases, geometries are created specifically for the simulation purposes and sometimes they are received from manufacturers. In both cases the most important thing is making the geometries the same with all characteristics for both CFD analysis and the experimental tests. But generally it is not possible to get a coil's 3D model from its manufacturer, which usually turns out to be a serious problem to generate a well-defined 3D CAD data to be used in engineering simulations. Trying to extract geometric details of a physical sample requires a lot of dimensional measurements, which is difficult to ensure tolerances, especially for fins. The measurement mistakes, encountered during this process, are likely to deviate the 3D model significantly from the original geometry. This, consequently, causes completely mislead CFD solutions.

Another problem is lamellas distance and coil's exterior dimensions ratio being too small like 1/100 or smaller. Thus it is required to generate very dense meshes such that for convergent solutions there must be a minimum of 3-5 elements between lamellas. Creating a very dense mesh requires too much memory and system resources. Also high mesh count extends the solution time on the same system. Creating a very dense mesh for coil is not enough for reaching to an accurate solution. In many cases due to system resource limitations we can't create soft transition between coarse meshes and fine meshes. This kind of sharp transition is not good for algebraic solvers. Generally it creates convergence problems. Sometimes it results in losing the solution and project completely, meaning requirement to start the run from the beginning.

Trying to solve a whole HVAC system by using actual geometry of components, very large cell counts are required, making such CFD analysis impossible to be performed with desktop workstations.

Another technique is dividing the HVAC system into parts. By that way the system requirement can be decreased and solutions for each part can be reached separately. Following that separate solutions can be combined together but generally by sacrificing some amount of precision and creating deviation from actual physics, due to data loses between separate solutions.

Coil resolution by Gap Model

Once the Gap Model is activated, FlowVision automatically recognizes two surfaces within a distance interval specified by user as gap-forming boundaries and Gap Cells are generated within these clearances. In this task, minimum and maximum Gap intervals are specified respectively as 1 μ and 2.5 mm, ensuring the 2.1 mm distances to be identified as Gap Cells.

In this approach; initial grid consists of cells with sizes equal to 3, 3 and 10 – 20 mm in x, y and z directions respectively. Following the initial grid generation, adaptation to solution (velocity gradient) up to 1st level is applied in the downstream of coil, for the purpose of capturing flow gradients. Resultantly, the computational grid has 870k cells plus 40k Gap Cells (`Figure3`).

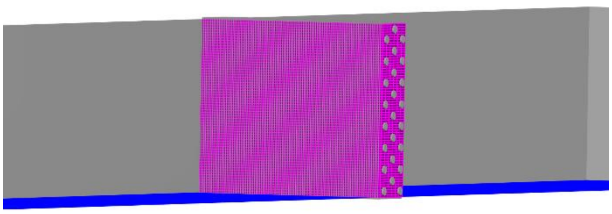

Figure3 Gap Cells between lamellas

Advantages: Gap model, embedded in FlowVision software, is intended to address dimensionality problems where clearances (down to sub-microns) are resolved by only one row of elements between the wall surfaces forming the clearance. The clearances, either static or dynamic, is to be automatically recognized by the software based on the interval entered by user as to indicate to be behaved as gap cells. Resultantly, using this model, a significant decrease in computational power requirement can be achieved by, in return, losing only industry-accepted levels of results accuracy.

Disadvantages: Although gap model is evaluated as a productive tool for calculating accurate flow rate, pressure drop and bulk heat transfer values among clearances, this modeling approach is not sufficient to capture complex spatial flow gradients, such as capturing supersonic shocks, in those regions.

Coil approximation by porous modeling

In this approach; the coil geometry is not imported into computational region and instead of that a rectangular box is positioned just at the same location of coil. Initial grid consists of cells with sizes equal to 5 mm in all directions, resulting in a computational grid with 758k cells.

Advantages: The most important advantage of porous media definition for coils is that it does not require a high number of meshes. Therefore, using this approach gives the capability to complete CFD analysis with low system resources and even more get solutions in shorter times. As a simulation strategy, system resources to be used for the coil modeled with actual geometry can be saved and instead used for whole HVAC unit analysis. In this way the data loss due to discrete solutions, long modeling times and possible modeling errors can be avoided.

Disadvantages: The definition of porous media is often useful only if we are concerned with pressure losses or outlet temperatures. However, this definition does not cover the actual resistance vector created by the battery in physical application. Deriving a mathematical expression to represent actual coil resistance, requires a number of assumptions and physical interpretations, trying to correlate parameters like permeability or porosity with a resistant coefficient or function. Even though it is considered to be sufficient to use non-unique products in a specific design phase, the coils used in HVAC industry have different geometric patterns in 3D and generally pressure drops are known only at some points. Therefore, it is, in most cases, very likely to fail in successful representation of coil geometry considering the momentum, heat transfer and turbulence effects.

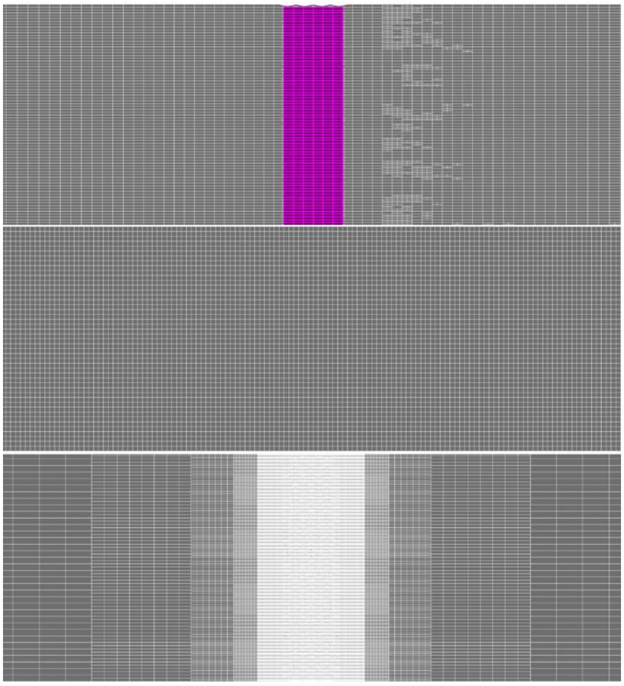

Figure4 Computational grid (top view) (from top to bottom; Gap Model, porous, grid resolved)

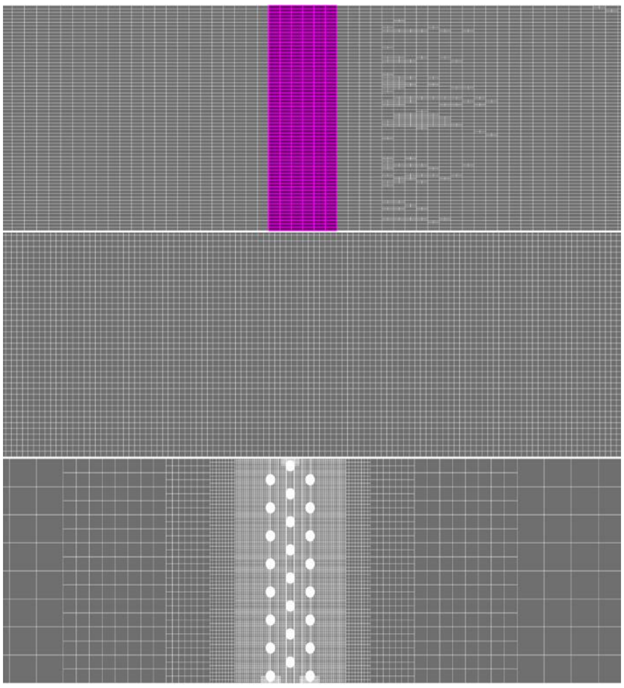

Figure5 Computational grid (side view) (from top to bottom; Gap Model, porous, grid resolved

RESULTS

Depending on the purposes of the current study, the main simulation outputs of interest are pressure drop across the coil, heat flux from air to coil surfaces and the air outlet temperature (Table1).

Table1 Overall simulation results

	Grid Resolved	**Gap Model**	**Porous Modeling**	**Coil Selection Software**
Geometry	Reverse engineered CAD (rough measurement)	Reverse engineered CAD (rough measurement)	No CAD (rectangular box)	Actual product (ideal manufacturing assumption)
Grid Size [# of cells]	5.78 M	870k + 40k Gap Cells	758k	N/A
Pressure Drop [Pa]	8.0	8.1	7.93	12.3
Outlet Temperature [°C]	15.5	15.2	15.35	11.5
Heat Flux [W]	1380	1356	1356	1600 (capacity)

Grid Resolved Modeling has the finest mesh (5.78 M) among all configurations where the distances between lamellas are ensured to be resolved by five elements. Regarding the computational power requirement, the case where coil is resolved by Gap Model, is following the first approach but still with a total cell number less than a million. On the other hand, porous modeling is accomplished by slightly less number of cells with respect to Gap Model and obviously, the analytical calculations with a product selection software is incomparably cheap in regards of computational cost.

The output values of pressure drop, outlet temperature and heat flux obtained by three different CFD approaches, well agree with each other where, considering the grid resolved case as reference, Gap Model approach and porous modeling approaches deviate less than 1% and 2% respectively.

On the other hand, the results obtained via product selection software Friterm seem not to be in the same amount of acceptance with CFD results as they are in between each other. Although the discrepancies in absolute values (of pressure drop, outlet temperature and heat flux) still seem to be acceptable, the percentage deviations are significantly larger. However, since the reversed engineered 3D CAD data is just a rough representation of the actual coil, the end results are still valuable in the sense to meet orders of magnitude and moreover constitute a base for future works.

On the color contours, results are from top to bottom; Gap Model, Porous Modeling and Grid Resolved. On side and top views flow direction is from left to right.

The inlet pressure is higher than the outlet pressure because the outlet boundary condition is free outlet. As can be seen from Table 1, pressure drop is around 8 Pa. As seen from the distributions, there is no significant change in pressure before the coil and the pressure decreases through the coil. As the stream strikes the coil pipes, the velocity rises and falls at some points, followed by the same phenomenon in the pressure. Thus, at some point the pressure increases to around 9 Pa.

In the temperature distribution, since the inlet temperature is higher than the outlet temperature, the temperature in the flow direction gradually decreases. In addition, since temperature on coil surfaces is defined to be dependent on height, 7°C at the top and 12°C at the bottom, temperature in the downstream of coil also demonstrates a gradual distribution (Figure 8-9).

Inlet mass velocity is 0.1044 kg/s, corresponding to 0.95 m/s. As expected, due to conservation of momentum, increased velocities (up to 2.5 m/s) are observed in the gaps between lamellas. Additionally, especially in the velocity contours from side view, the wakes of the water pipes and their effects on air stream can easily be observed.

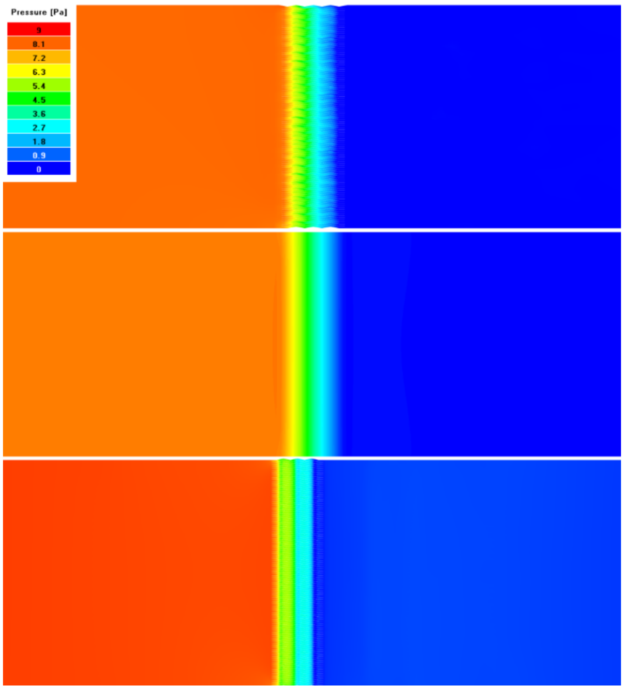

Figure6 Pressure Distribution (top view)

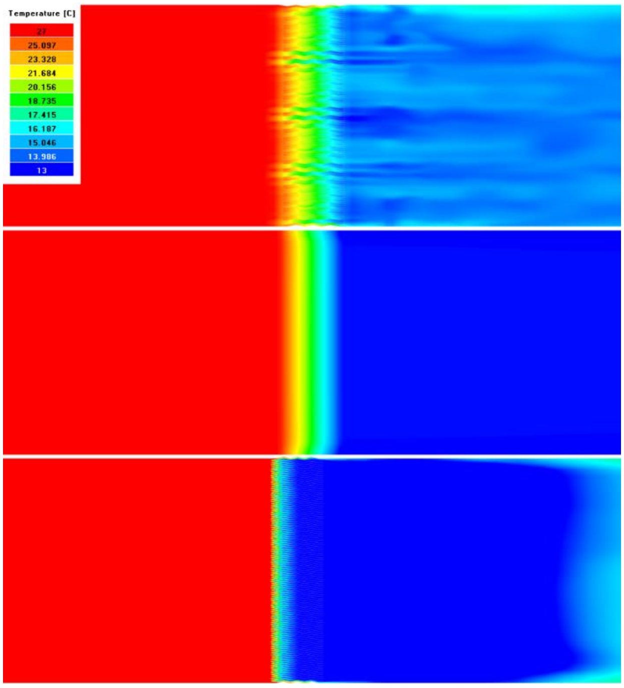

Figure8 Temperature Distribution (top view)

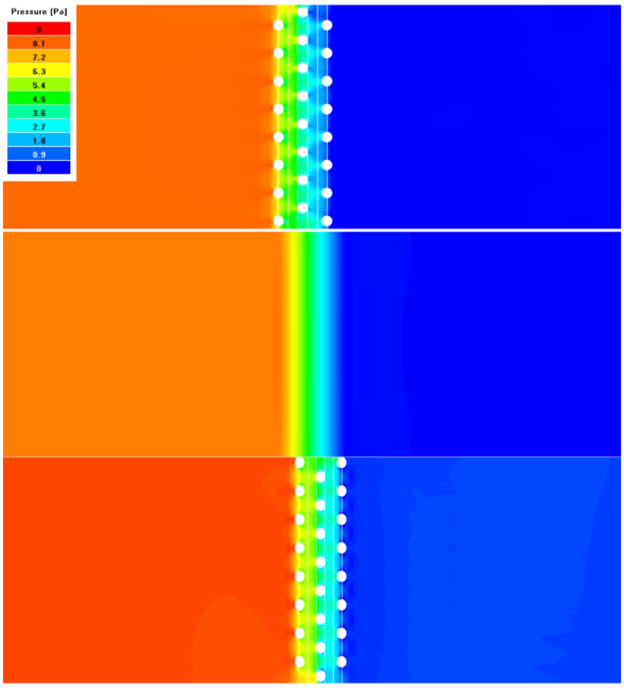

Figure7 Pressure Distribution (side view)

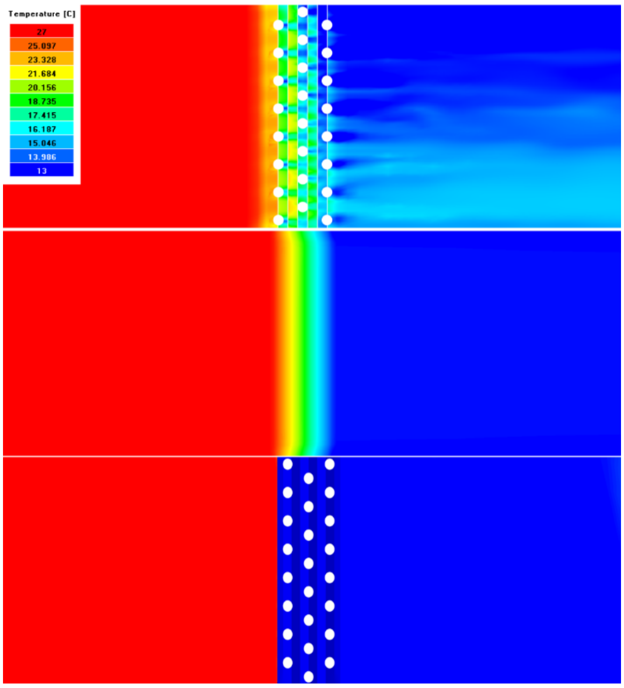

Figure9 Temperature Distribution (side view)

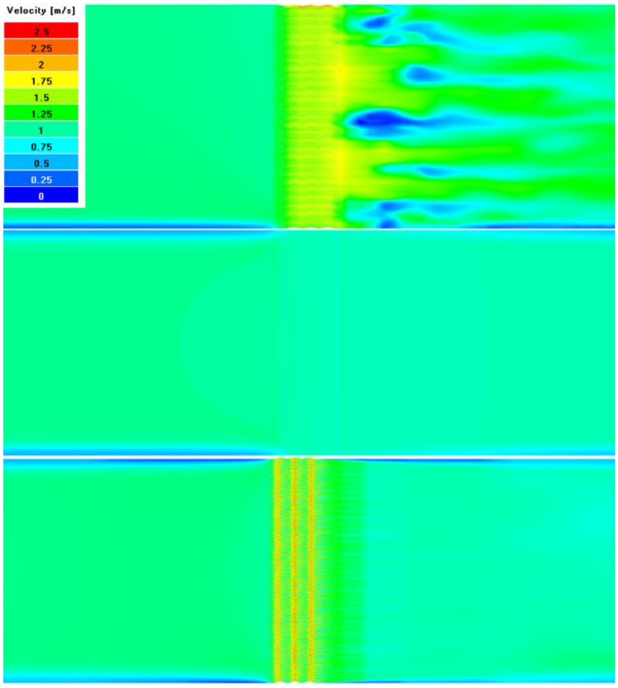

Figure10 Velocity Distribution (top view)

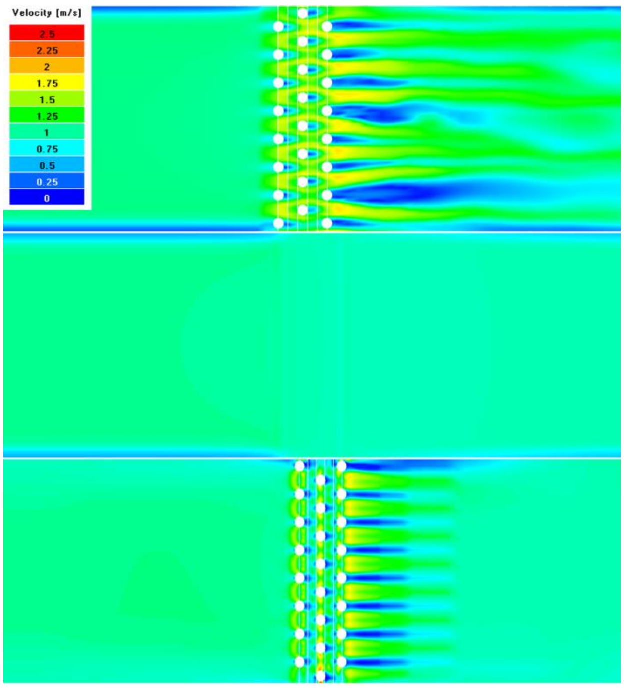

Figure11 Velocity Distribution (side view)

DISCUSSION

The main purpose of this study is to investigate possible ways to perform CFD simulations of cooling/heating coils used in HVAC systems. In this respect, three different CFD approaches are applied and evaluated with respect to each other and physical expectations.

The differences (relative errors) are obtained to be less than 2% for all CFD cases which is thought to be acceptable to be benefited from in the scope of heating/cooling coil design activities in HVAC industry.

The most remarkable outcome of this study is considered to be the utilization of Gap Model which seems to saving enormous amount of computational power, thus resulting in a very limited loss of accuracy from the traditional CFD calculations where the clearances between lamellas are resolved by a large number of cell elements.

Finally, it is strictly considered by the authors that related future work is ought to be performed; first of all, on a precisely reverse engineered (or already existing) CAD representation of a coil, thus being able to use certified product selection software as a more reliable basis. Once this is available, further assessments are to be made by performing grid resolved and Gap Model CFD simulations on a coil, if possible, on different operating conditions.

REFERENCES

1. **Capvidia NV.***FlowVision User's Guide Version 3.10.02.* Moscow : Capvidia NV, 2017.

2. *Overcoming of Barrier between CAD and CFD by Modified Finite Volume Method.* **Aksenov, Andrey, Dyadkin, Alexander and Pokhilko, Victor.** San Diego : ASME Pressure Vessels and Piping Division Conference, 1998.

3. **Capvidia NV.** FlowVision Gap Model. *FlowVision Website.* [Online] [Cited: February 23, 2018.] http://flowvisioncfd.com/gapmodel.

4. **Friterm A.Ş.** Friterm Standard Product Selection Software. *Friterm Website.* [Online] [Cited: February 23, 2018.] http://www.friterm.com/en-US/friterm-standard-product-selection-software/26097.