Automatic Hex-Dominant Mesh Generation with cfMeshPRO for CFD Analysis of HVAC Flows

Alen Cukrov and Franjo Juretić
Creative Fields Ltd.
Ulica Dragutina Golika 63, Tehnološki park Zagreb, 10000 Zagreb, Croatia

1 Introduction

This report presents automatic hex-dominant mesh generation of a detailed room model, carried out by using *cfMeshPRO*. *cfMeshPRO* is the core product of *Creative Fields Ltd.* and the general information regarding the underlying algorithm and other software features are given in [1]. In addition, the quality of the generated mesh was tested by performing a numerical simulation of a draft flow through the room. The study is conducted on a 16 GB RAM machine with 3.20GHz x 4 Intel Core i5 processor.

2 Test Case

2.1 Geometry Preparation

Within this study a detailed model of a living room (Figure 1) is used. This model is created by Frederico Costa and published on GrabCAD, a digital manufacturing hub. The model is originally stored in IGS format and is converted to STL using *Salome*. In order to make the geometry suitable for CFD analysis, the geometry is closed. This is done as follows:

- A planar face is created in *Salome* and exported in STL format.
- The created planar face is imported several times in *Meshmixer* to obtain enough faces for closing the domain.
- Each of the faces is adopted and associated to the domain as shown in Figure 2.
- The associated parts are merged to the geometry using *Paraview* and exported in binary STL format.

The computational domain does not require to be watertight and therefore it is ready for volume mesh generation. The geometry in binary STL format is imported in *cfMeshPRO*. From this binary STL, *cfMeshPRO* made its native FMS format and use it for mesh generation.

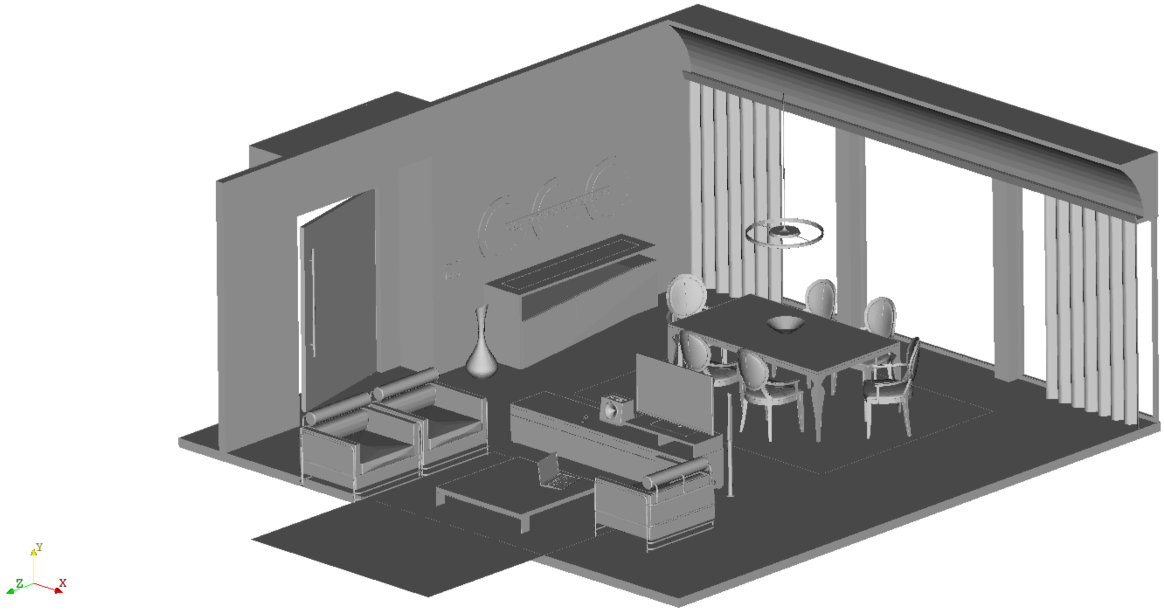

Figure 1: The detailed room model.

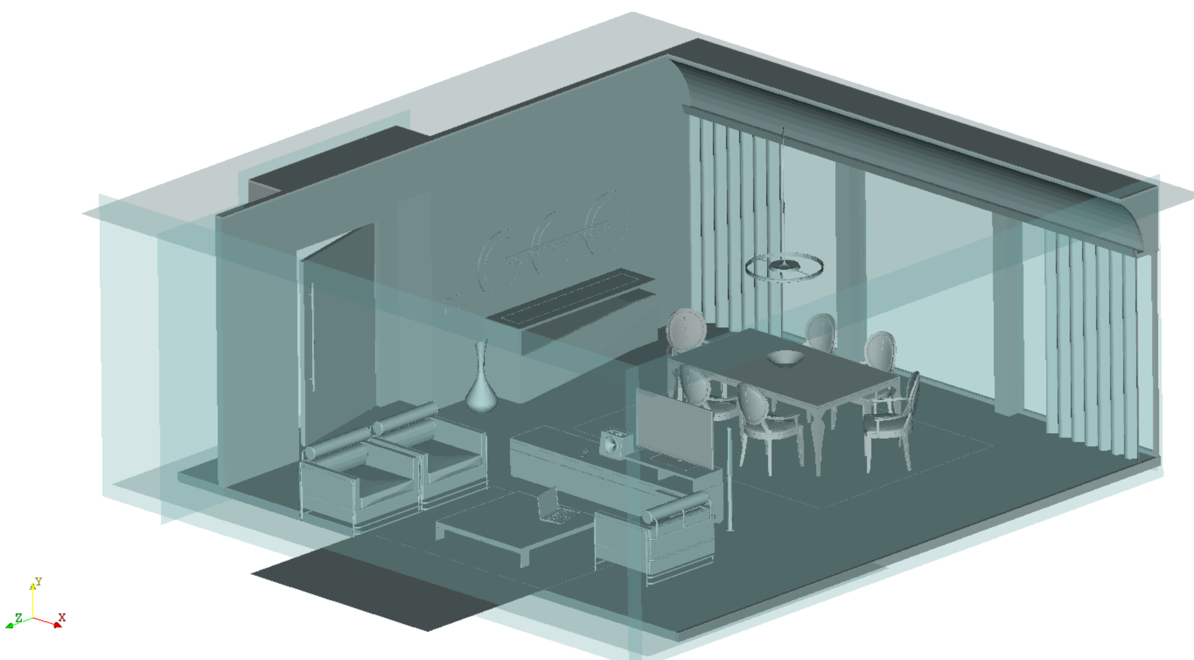

Figure 2: The planar faces are created and merged with the original geometry in order to make the geometry closed and suitable for CFD analysis.

2.2 Mesh Generation

As was previously shown, the geometry is composed of many complex parts which need to be meshed. The complex room parts can be selected into separate subsets, so called *facetSubsets*, that are later used to specify mesh properties.

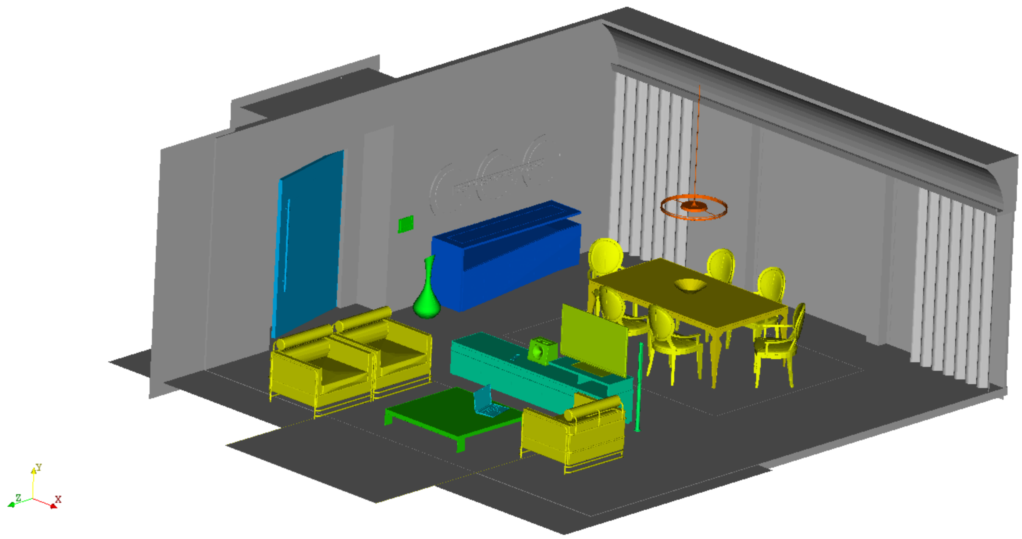

Figure 3: The *facetSubset* entities are assigned to the coloured parts of the geometry in order to control the mesh refinement level on them, i.e. to better capture them with the mesh.

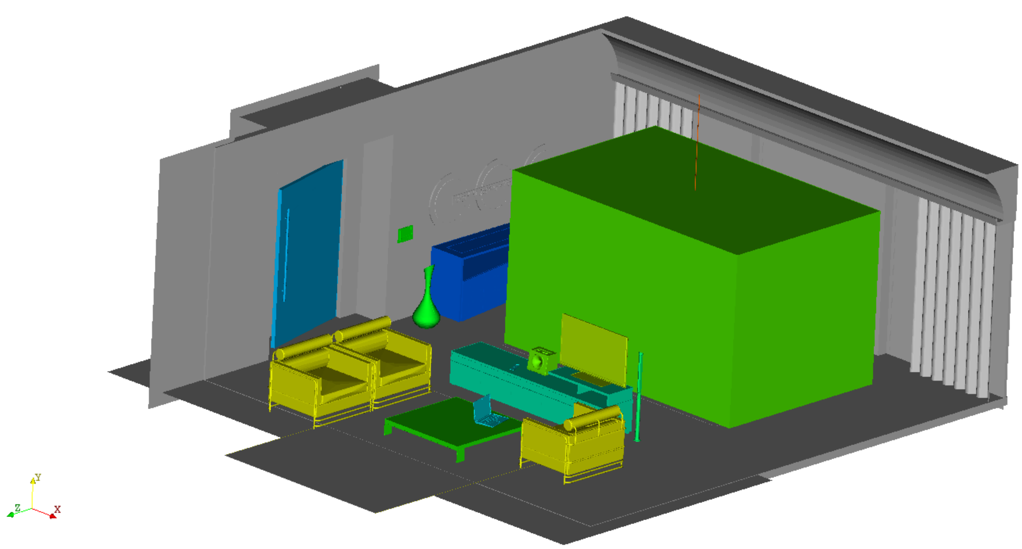

Figure 4: The created primitive (green box) denotes the zone where the user can specify the higher mesh resolution.

The *facetSubset* entities enable the user to control the refinement level at particular parts of the surface mesh. Here *facetSubsets* are applied on the parts which are coloured in Figure 3.

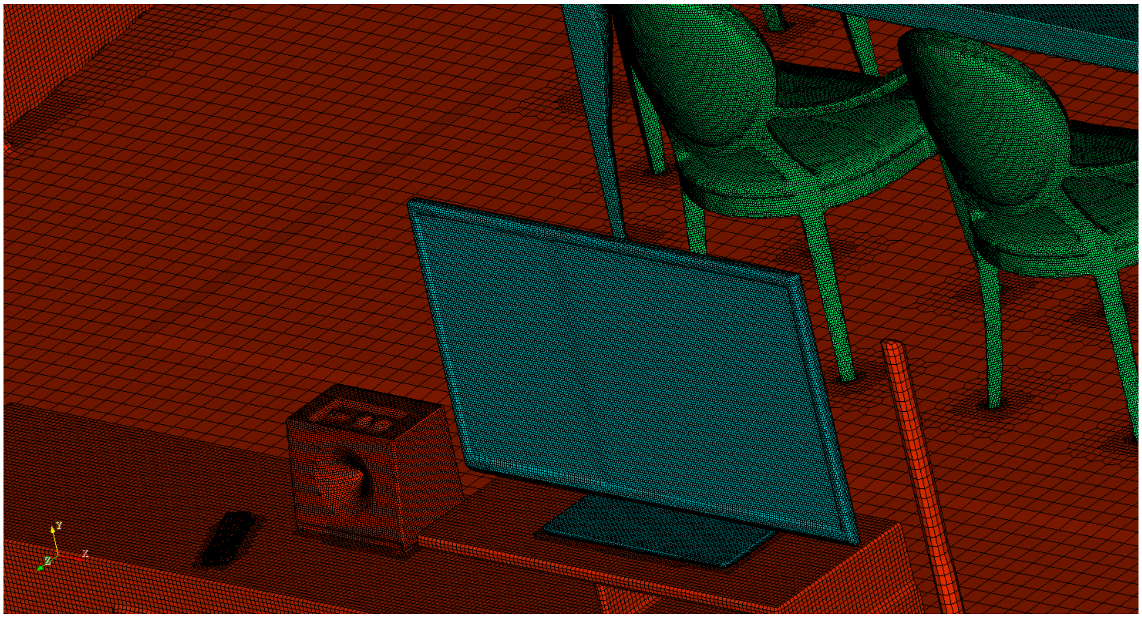

Figure 5: A detailed view on the surface of the volume mesh of the television, loudspeaker, remote control, part of the television desk and part of the dining table.

Figure 5 – 7 gives detailed view on the surface of the generated volume mesh. Please note how very complex parts are nicely resolved with a high quality mesh. Useful information regarding *facetSubset* entities can also be found in [2] while the ease of their creation on the complex geometries is shown in [3].

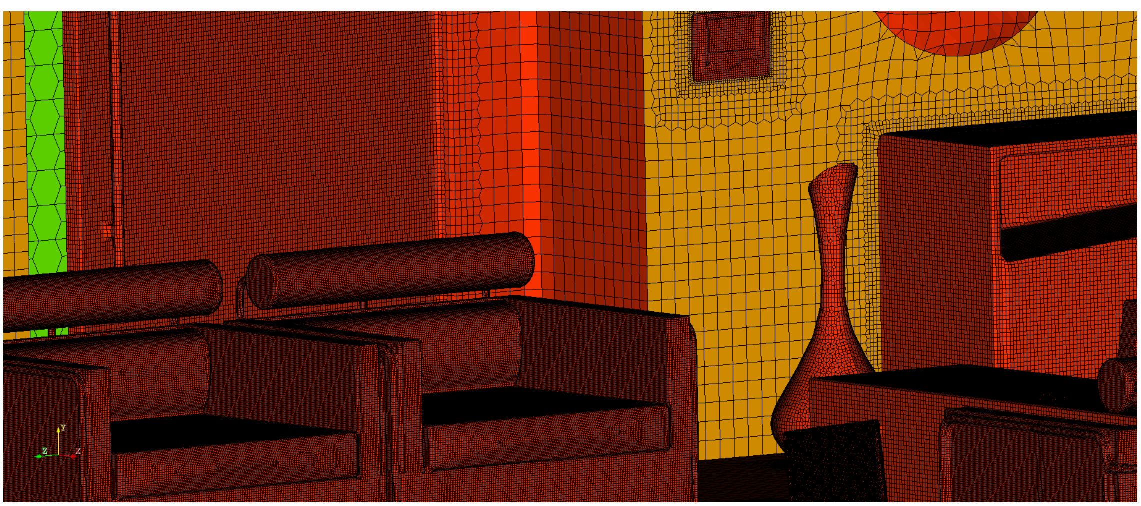

Figure 6: The discretized vase, armchairs, inlet door, thermostat, part of the television desk and part of the another desk.

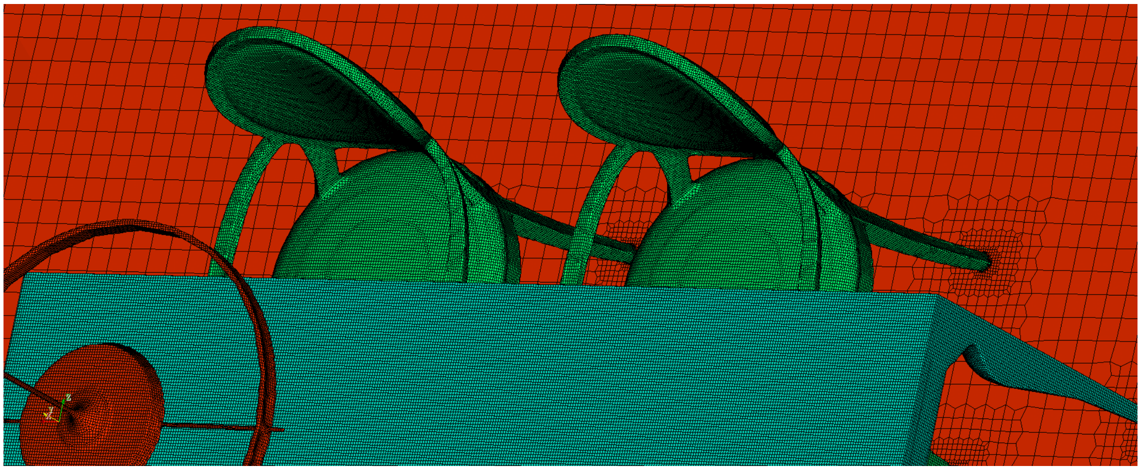

Figure 7: A view from above on the discretized ceiling lamp, dining table and chairs.

The mesh resolution in the zone of the special interest can be easily handled via *Object-based refinement* option as shown in Figure 4. The procedure starts by defining a primitive object (e.g. box, sphere or cylinder), and positioning it where needed followed by specification of the desired mesh resolution inside this region. Please note that the zone embedded in a box from Figure 4 has increased mesh resolution (see Figure 8). More details about available refinement options one can found in [2] and examples are provided in [4].

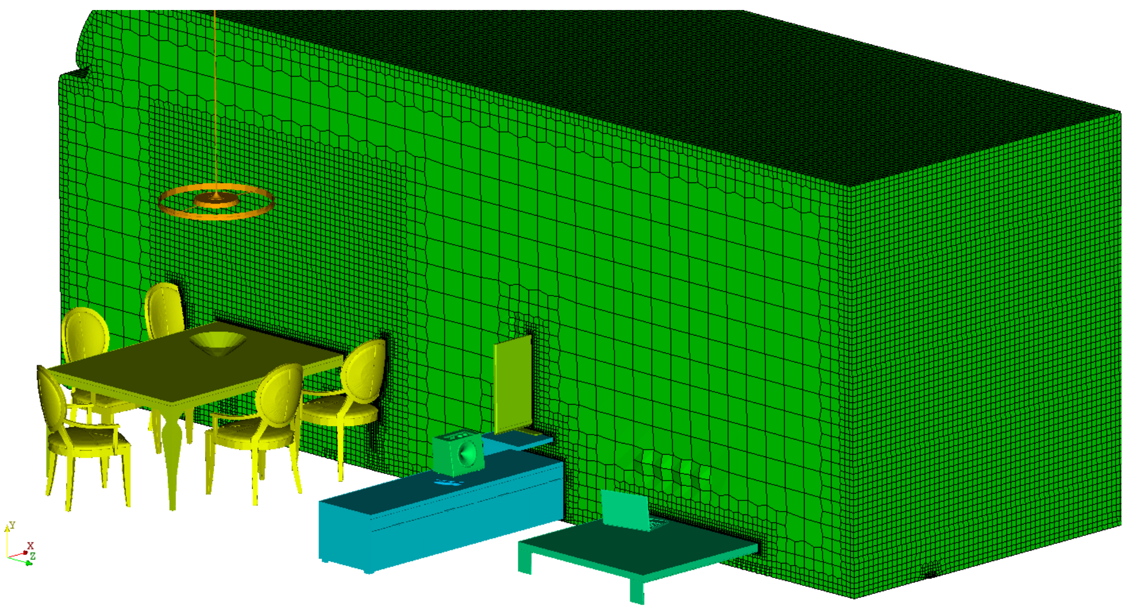

Figure 8: The cross-sectional view on the volume mesh. The rectangular zone with the fine mesh resolution inside the refinement box is shown in Figure 4.

The boundary layers can be defined on every patch in the domain. When defined, they are extruded over the complete part they are applied to. This can be seen in Figure 9 and 10 where the boundary layer is extruded over the television and the chair respectively.

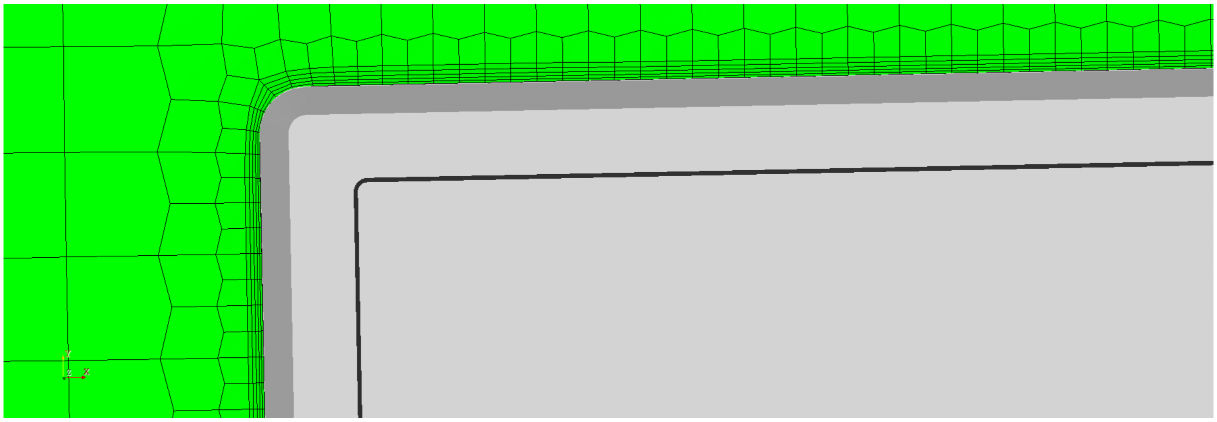

Figure 9: The boundary layer on the television.

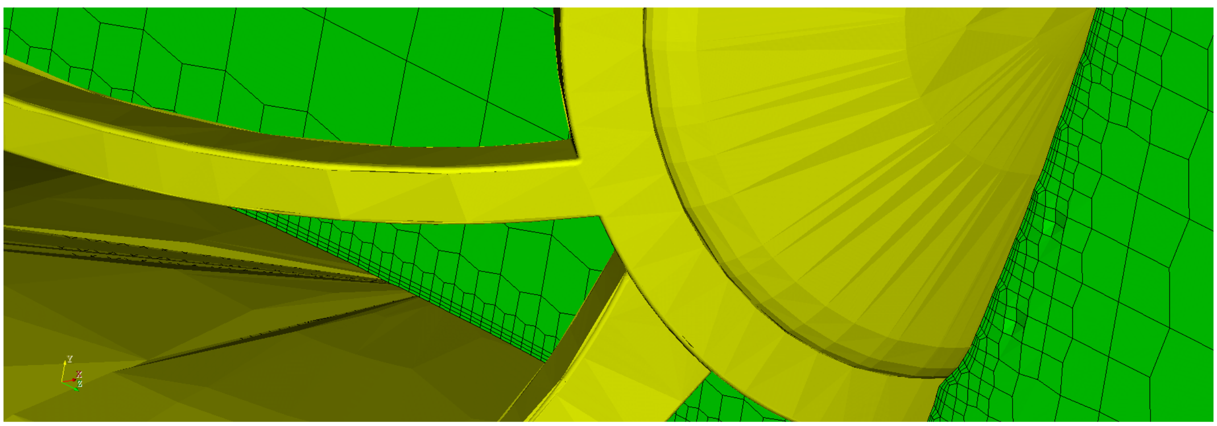

Figure 10: When extruded over a certain patch, the boundary layer topology is present over the complete patch. Therefore, one can notice that the chair seat as well as the chair backrest have three boundary layers.

The complete mesh consists of 6828911 cells where 6231339 cells (91.25%) are hexahedral cells for which is proven that result in the highest solution accuracy [5]. In addition, there are 596824 (8.74 %) polyhedral cells which follow the hexahedral cells in solution accuracy [5]. Finally, there are 748 (0.01%) prisms.

3 Numerical Simulation

An unsteady, turbulent, incompressible, isothermal flow is computed on the generated mesh. The mathematical model consists of the continuity and momentum equations while the turbulence is modelled using the k-ω SST turbulence model with the standard coefficients. The volumetric flow rate of air through the inlet door is 500 m^3/h. The rightmost window from Figure 3 is considered to be open and is defined as outlet. The remainder of the domain are walls. The computation is carried out using *OpenFOAM*'s `pimpleFoam` solver. The computational results are shown in Figure 11.

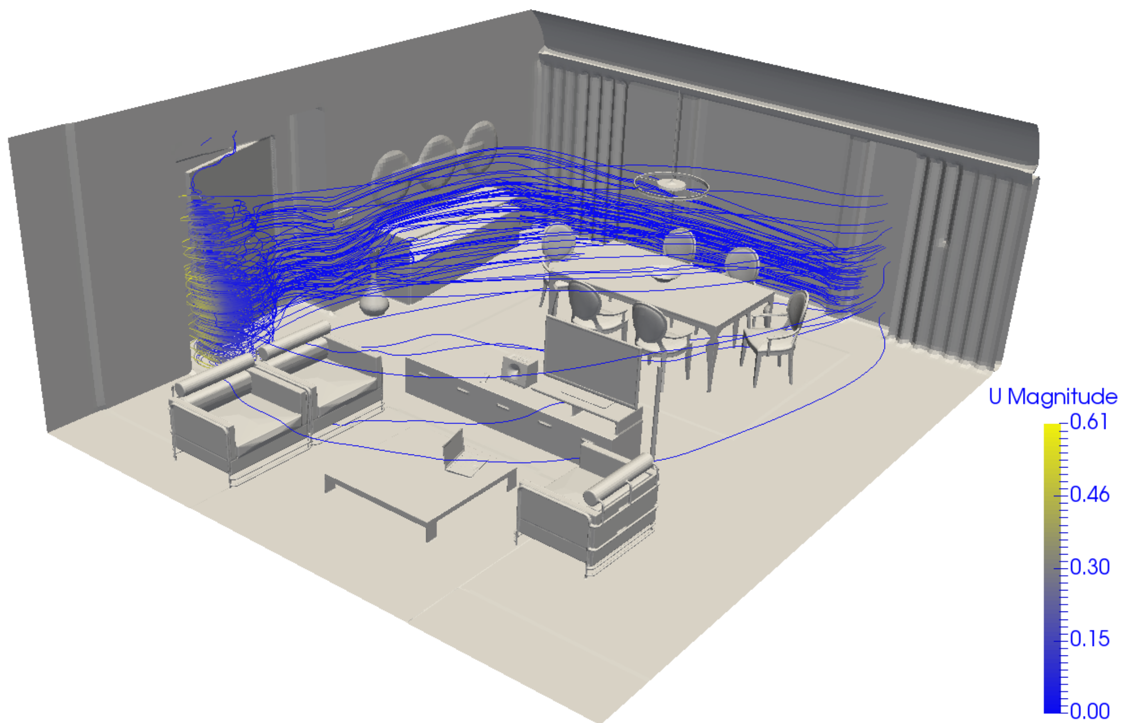

Figure 11: The flow streamlines coloured in the colors of the velocity magnitude. The flow enters the room through the partially open door and leaves the room through the rightmost window.

4 Conclusion

This report presents an application of *cfMeshPRO* in CFD analysis of HVAC flows on a detailed geometry. It is also shown that the available mesh refinement options enable fast and simple mesh generation. Finally, the quality of the generated mesh is validated by the performed numerical simulation.

Acknowledgement

The authors thank to Frederico Costa for providing us the room geometry.

References

[1] Cukrov, A., Lugarić, T. and Juretić, F.: *cfMeshPRO - An Automatic Finite Volume Mesh Generator*, Proceedings of 7th Meeting of Croatian Society of Mechanics, Split, 2016. (in Croatian)

[2] Juretić, F.: *cfMesh User Guide*, Creative Fields, Zagreb, 2015.

[3] Cukrov, A. and Juretić, F.: *Automatic Hex-Dominant Mesh Generation for CFD Analysis of Formula One Car with cfMeshPRO*, White paper, Creative Fields, Zagreb, 2016.

[4] Cukrov, A.: *Quick Start With cfMeshPRO*, Creative Fields, Zagreb, 2016.

[5] Juretić, F.: *Error Analysis in Finite Volume CFD*, PhD Thesis, Imperial College London, United Kingdom, 2004.