DEVELOPMENT OF WEB PLATFORM OF CFD ANALYSIS USING OPENFOAM AND PARAVIEW

JINWON SEO[1], BONGKEUN CHOI[2], HYUNKANG PARK[3], HYUNWOONG LEE[4]
[1]Hyundai Mobis CAE Team, seojw@mobis.co.kr
[2]Hyundai Mobis CAE Team, bong82@mobis.co.kr
[3]Nextfoam Research Center, hkpark@nextfoam.co.kr
[4]Nextfoam Research Center, hwlee@nextfoam.co.kr

Keywords: *OpenFOAM, Paraview Automation, Self-Verification, Web Platform, CFD*

Global automobile engineers are utilizing various numerical analysis methods for product development. Traditionally, simulation engineers predict and improve the performance of product through virtual test just after the product design. However, every commercial software used in this process (such as FLUENT, STAR-CCM+, ANSYS CFX, etc.) is expensive and requires specialized knowledge of simulation for engineers to conduct virtual tests. With the Fourth Industrial Revolution underway, the traditional difference in role between designers and simulation experts is breaking down, which is increasing demand for designers to directly verify their own design in the product development process. This trend plays an important role in improving product quality, as it enables designers to identify and resolve problems in early stage of development.

The purpose of this study was to create an open web platform that would enable designers to use simulations directly in product development and would result in better design and improved performance. To build this platform, we employed open-source code for the solver (OpenFOAM) and post-processing program (Paraview). For pre-processing software, we used the commercial program ANSA [1]. While open-source codes such as SnappyHexMesh or cfMesh were available, we found it necessary to choose ANSA due to its ability to quickly and reliably produce analysis models (grid) during the mass production development process. When using the original OpenFOAM code, the complexity of shapes and physical phenomena in the analysis model may cause convergence problems. Additionally, turbulence models such as wall functions are difficult to simulate in detail compared to commercial software. To overcome these limitations, this study relied on the research published by Seo et al. [2].
In this study we developed a web-based analysis platform for computational fluid dynamics (CFD) named MOCA (MOBIS CFD ANALYSIS). In MOCA designers easily upload CAD models and input essential boundary conditions through web page. The modeling team is responsible for pre-processing, which involves downloading the CAD model, generating the mesh model, and uploading output to the website before initiating the analysis execution. MOCA utilizes the OpenFOAM solver to perform the analysis, and upon completion of the calculation, automatically uses Paraview to generate predefined analysis results such as streamlines, vector plots, and contours in PPT format. The web platform used in this study is built on IBM's EnginFrame, with the scheduler for the analysis cluster utilizing LSF. The overall process is summarized in Figure 1.

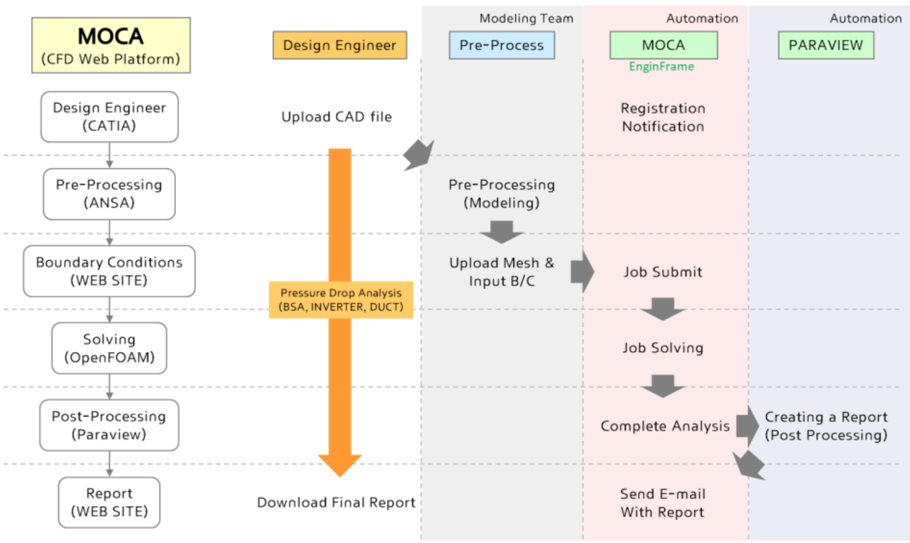
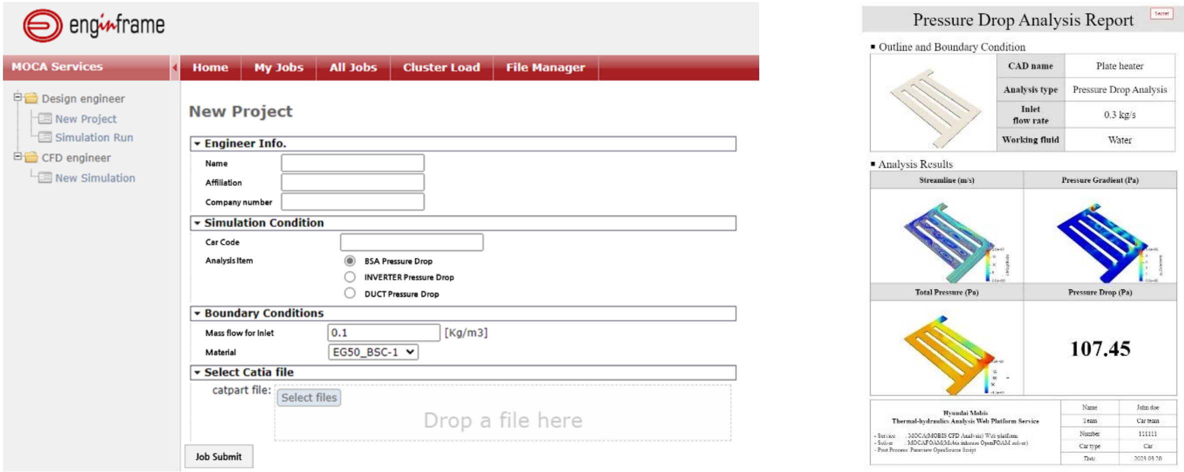


Figure 1: The Workflow on the MOCA Web Platform

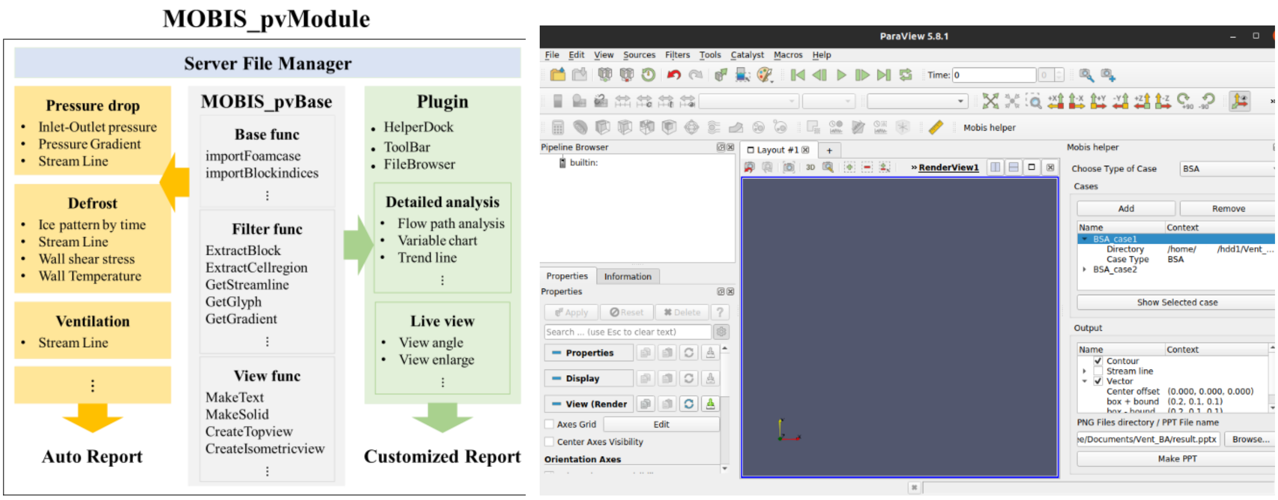
Using the MOCA service, design engineers can perform direct analysis for pressure drop of the automotive battery system, inverter, and cockpit air duct. Main page of MOCA is presented in Figure 2(a). The analysis results are automatically organized and submitted in report form, as shown in Figure 2(b). After the analysis is complete, a Python script of Paraview automatically produces an analysis report that includes essential information, such as the design engineer's name and analysis model, as well as a 3D geometry information, pressure gradient contour, streamlines, pressure drop values of the inlet and outlet, and other details.



(a) Main Page **(b) Automation Report Form**

Figure 2: MOCA Web Platform

For design engineers who are not familiar with CFD, it is necessary to easily post-process the simulation results. Using the pvpython library of Paraview, post-processing can be fully automated. We have modularized the analysis process required for performance verification for each part of the automobile. In addition, to treat massive data from the local server, we have inserted the concept of server file manager, which allows users to easily manipulate data stored on the Paraview server. Detailed information about the Python module can be found in Figure 3(a). Moreover, we have developed a separate plug-in for CFD engineers who require more information, as illustrated in Figure 3(b). This plug-in enables CFD engineers to access flow path distribution, characteristic charts based on independent variables, and trend lines that can generate customized reports. Furthermore, various post-processing techniques can be attempted in real-time through live view.



(a) a Schematic of Python Module **(b) a plug-in of Paraview**

Figure 3: The Paraview Module and Plug-in developed for the MOCA Service

References

[1] ANSA Pre Processor User Guide ver. 20.0, BETA CAE System, Sep. 2019.

[2] J.W. Seo, H.S. SEO, B.K. Choi, "Development of CFD Program for Automotive Ventilation and Defrost Simulation Using OpenFOAM", SAE 2020-01-0154, Apr. 2020.