Title:
Intelligent CFD Analysis Regime Validation and Selection in Feature-based Cyclic CAD/CFD Interaction Process

Authors:
Lei Li, lli7@ualberta.ca, University of Alberta
Carlos F. Lange, carlos.lange@ualberta.ca, University of Alberta
Yongsheng Ma, yongsheng.ma@ualberta.ca, University of Alberta

Keywords:
CFD, Analysis View, Expert System, Multi-view Feature Modelling

DOI: 10.14733/cadconfP.2017.263-267

Introduction:

Multi-view feature modelling provides a specific view for each phase in product development [1]. The analysis view [7] should be fully integrated with CAD models in a multi-view product development environment for simulation-based design. In the development of fluid flow products, CFD (Computational Fluid Dynamics) is increasingly used as an advanced support. However, the successful application of CFD requires special knowledge and rich experience, which is a barrier for the conversion from the design view to the analysis view, and the maintenance of information consistency.

Several approaches to multiple feature views have been proposed, such as design by features, feature recognition [3] and feature conversion [2],[8]. In one-way feature conversion, features in a specific view are usually derived from the original design view [4]. Bronsvoort and Noort [1] put forward a multiple-way approach which enables a designer to modify the product model from an arbitrary view. In this paper, the CAE interface protocol [5] is used to convert the features in the design view into the *CAE boundary features* [5] in the analysis view. Based on the physical knowledge, an expert system is established to further process those features and generate a robust simulation model with the help of *fluid physics features* and *dynamic physics features* [6] in the analysis view.

Main idea:

Based on the engineering knowledge and using design by features, the geometry of the product can be created parametrically. Based on the functional requirement derived from design intent, the *CAD fluid functional feature* is defined as a class of design intent attributes which are composed of design parameters and functional descriptions, as well as functional fluid geometry [6]. The CAD fluid functional features not only convey the design intent to the downstream applications but also entail the design view which covers both geometrical and non-geometrical features.

The features in the design view are transformed into features in the CFD analysis view through feature conversion. To achieve this, the fluid domain needs to be abstracted with necessary defeaturing. Based on the function of the product, the non-geometrical features like fluid properties and boundary conditions can be derived from the design intent attributes. In order to keep the consistency, the faces of the fluid domain are assigned specific tags to identify their boundary type. The tag is an identifier which can be recognized by both CAD and CFD systems. It works as part of the CAE interface protocol. Tab. 1 shows the mechanism of the geometric feature conversion between the design view and the CFD analysis view, in which m, n, p, and q are the numbers of the corresponding faces in the abstracted fluid domain [5]. Then, the mesh can be generated and the boundary conditions can be assigned, accordingly. Thus, the *CAE boundary features* are generated, which include the fluid attributes inherited from the design intent attributes and the meshed fluid domain with boundary conditions attached. This feature conversion process is depicted in Fig. 1.

Tag	*Attribute*	*Boundary condition*
$I_1, I_2, I_3, \ldots\ldots, I_m$	Inlet	Velocity or pressure inlet
$O_1, O_2, O_3, \ldots\ldots, O_n$	Outlet	Velocity or pressure outlet
$W_1, W_2, W_3, \ldots\ldots, W_p$	Wall	No-slip wall
$S_1, S_2, S_3, \ldots\ldots, S_q$	Symmetrical plane	Symmetry

Tab. 1: Geometric information transmission in the feature conversion. [5]

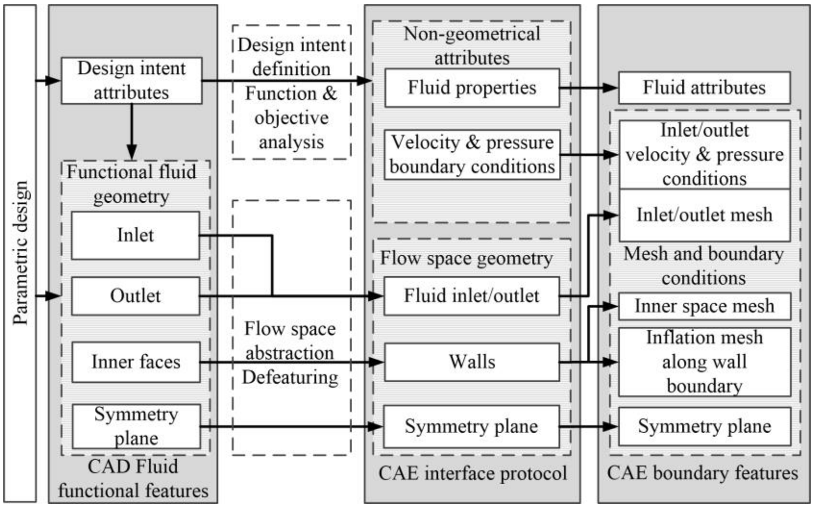

Fig. 1: Feature conversion between the design view and the analysis view.

Ideally, the analysis view should be a feature model to propagate the changes in the design view [7]. However, the current solver structure does not support this generally. To assist the solver setup, CFD expert systems can be used by applying artificial intelligence [9]. Here, we propose to develop intelligent CFD solver functions for dry steam simulation, which are shown in Fig. 2.

The initial values are obtained from the fluid attributes which are a part of the CAE boundary features. The parameters in the following steps can be derived using equations [6]. Here, Q is the flow rate of gas, d is the inner diameter of duct, μ is the dynamic viscosity of gas, ρ is the density of gas, R is the gas constant, T is the temperature of gas, k is the specific heat ratio of gas, A is the cross-sectional area of duct, p is the pressure of gas, a is the speed of sound of gas, v is the velocity of gas, Re is the Reynolds number, and Ma is the Mach number. If the Reynolds number exceeds the critical value in the duct, a turbulence model will be selected. Meanwhile, the Mach number judges whether the flow is compressible. If the compressibility effects cannot be ignored, the total energy model should be selected and the reference pressure, as well as proper boundary conditions should be setup to trigger the compressible flow simulation. In the beginning of the simulation or at the time the simulation has convergence problems, lower order discretization schemes like UDS and Euler implicit, as well as k-ε turbulence model if applicable, are preferred to assist convergence.

The index i (iteration), C (Convergence) and D (Divergence) will be updated after each simulation run. If a simulation converged, post processing will be conducted to check whether the solution matches the initial assumptions and expected accuracy. If not, grid adaption will be activated based on the existing simulation result. According to the peak value of Reynolds number and Mach number, the flow regime is double-checked to see whether the simulation model needs to be changed. If a simulation diverged, the solver setup should be modified to achieve convergence. It should be noted that each time when a new iteration starts, only one change is allowed in the solver configuration to obtain the sensitivity towards different simulation schemes. If the simulation still has convergence problems after several successive runs, human intervention is needed to diagnose the problem.

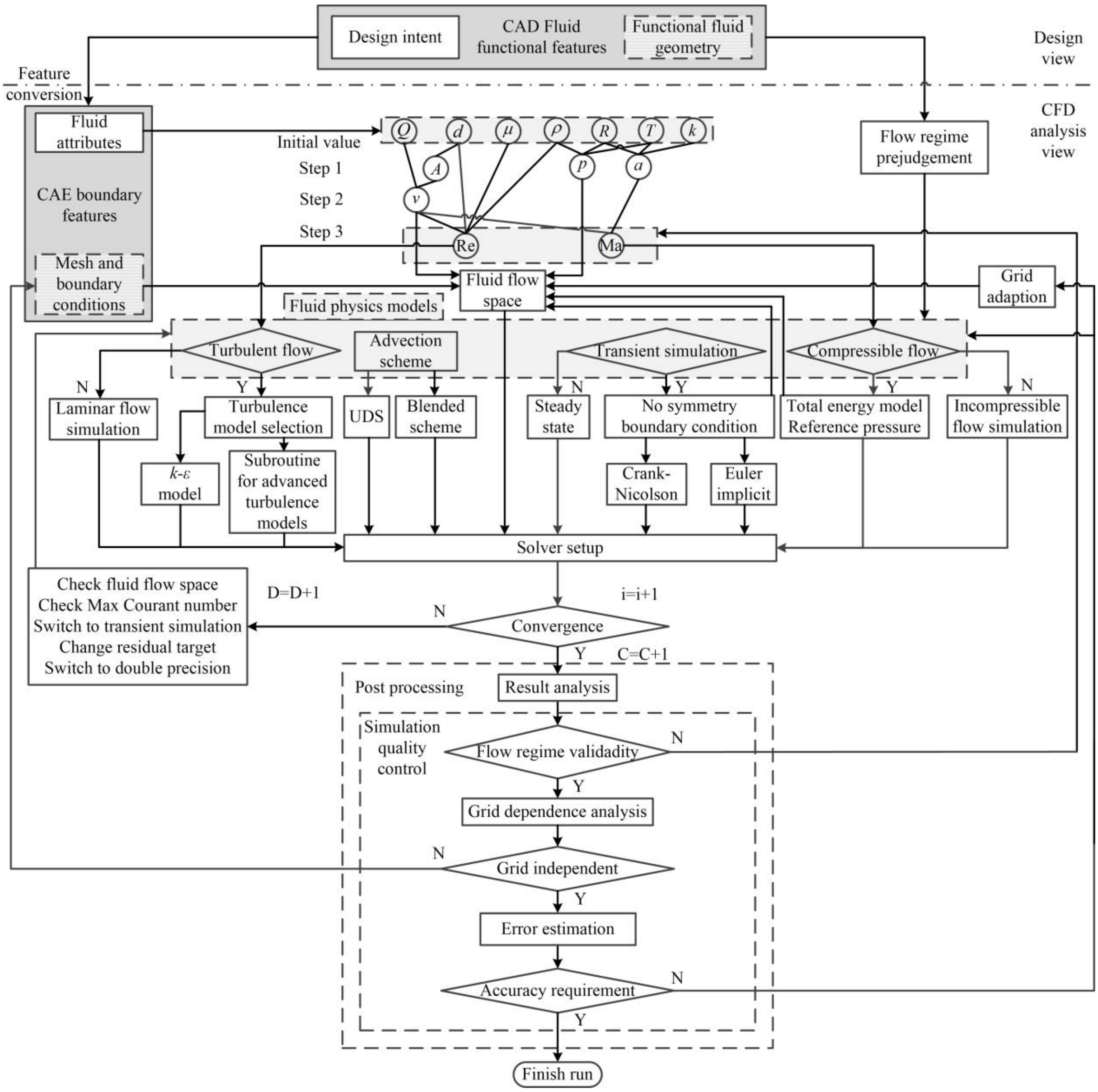

Fig. 2: Intelligent solver functions for the dry steam simulation.

Higher order schemes can be applied after rounds of successful simulations because the mesh will be further refined. In such kind of situation, a subroutine, shown in Fig. 3, will be entered to select one advanced turbulence model if the flow is turbulent. This program stops when the precision of the turbulence model is satisfactory. If the flow regime used to judge the fluid physics models is valid, grid independence analysis will be conducted to see whether the simulation is still affected by the grid refinement. By this analysis, the error of the discretization can be estimated if the grid is independent. Consequently, the accuracy of the final simulation can be guaranteed. During this process, the *dynamic physics feature* is developed to facilitate the generation of the robust simulation model [6].

Fig. 4(a) shows a section of a pipe with a contraction. The design and analysis of the piping system are selected as the case study because the pressure drop under a certain flow rate in the piping system can be determined by head loss calculation [10], and it can be used as a benchmark for the simulation results. The fluid domain is created by feature conversion and is shown in Fig. 4(b). Under the control of CAE boundary features, the mesh is generated as shown in Fig. 4(c).

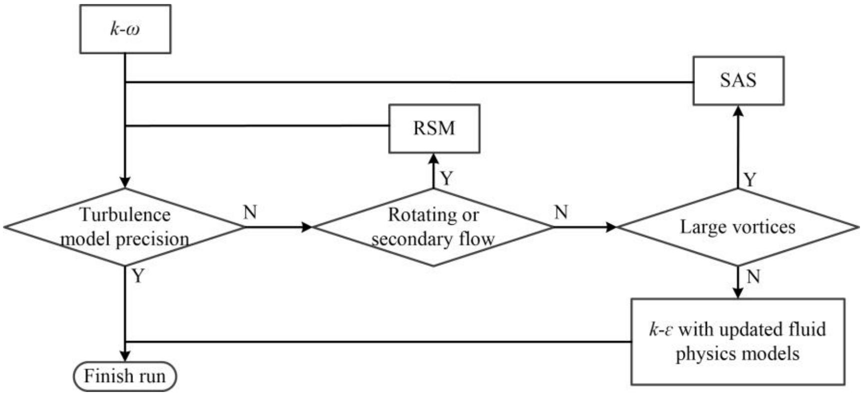

Fig. 3: Subroutine for advanced turbulence models.

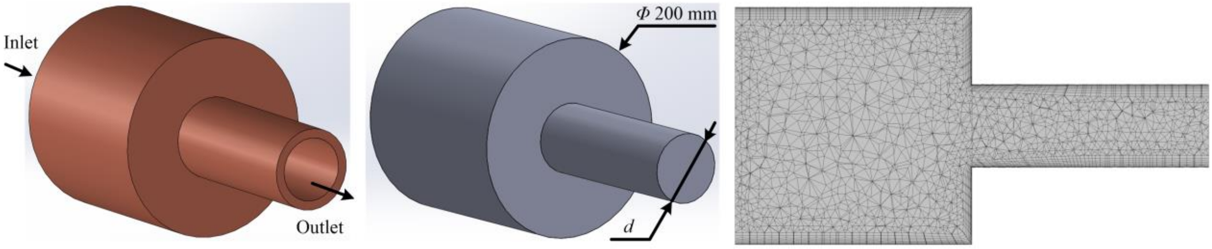

Fig. 4: Feature conversion in the analysis of the piping system: (a) Contracted pipe, (b) Fluid domain, (c) Mesh generation.

Design point	d (mm)	Δp (Pa)	Δp_1 (Pa)	δ_1 (%)	Δp_2 (Pa)	δ_2 (%)
DP1	70	29.05	32.18	10.77	30.62	5.40
DP2	60	54.32	60.81	11.94	57.39	5.65
DP3	50	113.45	129.64	14.27	120.04	5.81
DP4	40	278.35	325.01	16.76	296.57	6.55
DP5	30	882.52	1065.67	20.75	959.09	8.68
DP6	20	4478.13	6156.05	37.46	5057.10	12.93

Tab. 2: Pressure drop calculation based on different methods.

At the inlet of the fluid domain, dry steam flows at 1 m/s. The pressure of 101325 Pa is assigned to the outlet. The pressure drop between the inlet and outlet is tabulated in Tab. 2. Here, d is the small inner diameter of the pipe, Δp is calculated from the published head loss coefficient plot for flow through contracted pipe [10], Δp_1 is calculated by ANSYS CFX under the batch mode, which is a kind of routine analysis using the default setup for each design point. δ_1 is the relative error between Δp_1 and Δp. Correspondingly, Δp_2 is calculated using the intelligent solver functions we put forward, and δ_2 is the relative error between Δp_2 and Δp. Seen from Tab. 2, the pressure drop increases with the decreased small inner diameter. The δ_2 error of the intelligent solver scheme is in the order of the uncertainty of Δp obtained from empirical results which cannot be improved further. The δ_1 error is significantly bigger especially when much higher velocity occurs with very small d, which means the compressibility effect is already not negligible. The reason for the difference in the error control is that the intelligent solver functions support the validation of the CFD results and the reselection of correct solver regimes if there is any validity issue. Therefore, the CFD analysis view proposed in the paper achieves the automatic feature conversion from the design view and provides the convincible input for another view.

The proposed intelligent CFD solver functions can be implemented based on ANSYS Workbench. Specifically, CCL (CFX Command Language) can be applied as a session to manipulate CFX-Pre and CFD-Post. Besides, Workbench scripting can be used to create the whole project and invoke various applications to complete the created project [11].

Conclusions:

On top of the design view supported by the *CAD fluid functional features*, in this paper, the CFD analysis view is developed as a feature model which consists of associative *CAE boundary features*, *fluid physics features*, and *dynamic physics features*. The feature conversion between the design view and the CFD analysis view is achieved by the CAE interface protocol. Especially, the application of fluid physics feature and dynamic physics feature enables computer-assisted solver regime selection and validation. It should be highlighted that the quality of the simulation is guaranteed by the grid independence analysis and error estimation. The subroutine for advanced turbulence models is developed to enhance its ability to model complex turbulent flow. The effectiveness of the proposed method is shown by the investigation of pressure drop in a benchmark case of contracted pipe under different designs. This approach can be applied in other contexts by adapting the relevant knowledge bases. The intelligent CFD solver functions will be developed by using CCL and Workbench scripting.

Acknowledgement:

The authors would like to acknowledge Natural Sciences and Engineering Research Council of Canada (NSERC), RGL Reservoir Management, China Scholarship Council (CSC), University of Alberta and Alberta Innovates Technology Futures (AITF) for the financial support.

References:

[1] Bronsvoort, W. F.; Noort, A.: Multiple-view feature modelling for integral product development, Computer-Aided Design, 36(10), 2004, 929-946. https://doi.org/10.1016/j.cad.2003.09.008

[2] Bronsvoort, W. F.; Jansen, F. W.: Feature modelling and conversion—key concepts to concurrent engineering, Computers in Industry, 21(1), 1993, 61-86. https://doi.org/10.1016/0166-3615(93)90045-3

[3] De Martino, T.; Giannini, F.: Feature-based Product Modelling in Concurrent Engineering, In Globalization of Manufacturing in the Digital Communications Era of the 21st Century, Springer US, 1998, 351-362. http://dx.doi.org/10.1007/978-0-387-35351-7_28

[4] Hoffmann, C. M.; Joan-Arinyo, R.: Distributed maintenance of multiple product views, Computer-Aided Design, 32(7), 2000, 421-431. https://doi.org/10.1016/S0010-4485(00)00023-3

[5] Li, L.; Ma, Y.: CAD/CAE associative features for cyclic fluid control effect modeling, Computer-Aided Design and Applications, 13(2), 2016, 208-220. https://doi.org/10.1080/16864360.2015.1084190

[6] Li, L.; Ma, Y.; Lange, C. F.: Association of Design and Simulation Intent in CAD/CFD Integration, Procedia CIRP, 56, 2016, 1-6. https://doi.org/10.1016/j.procir.2016.10.006

[7] Smit, M. S.; Bronsvoort, W. F.: Integration of design and analysis models, Computer-Aided Design and Applications, 6(6), 2009, 795-808. https://doi.org/10.3722/cadaps.2009.795-808

[8] Suh, Y. S.; Wozny, M. J.: Interactive feature extraction for a form feature conversion system, In Proceedings of the fourth ACM symposium on Solid modeling and applications, 1997, 111-122. https://doi.org/10.1145/267734.267762

[9] Tong, S. S.: Design of aerodynamic bodies using artificial intelligence/expert system technique, In 23rd Aerospace Sciences Meeting, American Institute of Aeronautics and Astronautics, 14(17), 1985. https://doi.org/10.2514/6.1985-112

[10] White, F. M.: Fluid Mechanics Seventh Edition, McGraw-Hill, 2003.

[11] Workbench scripting Guide, ANSYS, Inc., Release 15.0, 2013, http://148.204.81.206/Ansys/150/Workbench%20Scripting%20Guide.pdf.