Association of Design and Simulation Intent in CAD/CFD Integration

Conference Paper · March 2016

Yongsheng Ma
University of Alberta

Carlos F Lange
University of Alberta

Available online at www.sciencedirect.com

ScienceDirect

Procedia CIRP 00 (2014) 000–000

www.elsevier.com/locate/procedia

9th International Conference on Digital Enterprise Technology - DET 2016 – "Intelligent Manufacturing in the Knowledge Economy Era

Association of Design and Simulation Intent in CAD/CFD Integration

Lei Li[a], Yongsheng Ma[a,]*, Carlos F. Lange[a]

[a]*Department of Mechanical Engineering, University of Alberta, Edmonton, T6G 1H9, Canada*

* Corresponding author. Tel.: +1-780-492-4443; *E-mail address:* yongsheng.ma@ualberta.ca

Abstract

CFD (Computational Fluid Dynamics) requires strong expertise and extensive training to obtain accurate results. To improve the usability in the complex product development process, two new types of engineering features, fluid physics feature and dynamic physics feature, which convey the simulation intent, are proposed in this paper to achieve CFD solver setup automation and robust simulation model generation in an ideal CAD/CAE integration system. Further, the association between simulation intent and design intent is integrated with another newly defined fluid functional feature in order to achieve the consistency. Consequently, an optimal design could be achieved by considering production operation, manufacturability and cost analysis concurrently. A case study of steam assisted gravity drainage (SAGD) outflow control device (OCD) is presented to show the prospective benefits of the method.

Peer-review under responsibility of the Scientific Committee of the "9th International Conference on Digital Enterprise Technology - DET 2016.

Keywords: Design Intent; Simulation Intent; Fluid Functional Feature; Fluid Physics Feature; Dynamic Physics Feature; CAD/CFD Integration

1. Introduction

In order to facilitate SBD (Simulation Based Design), CAD and CAE are expected to be integrated seamlessly. However, design and simulation are usually conducted by different engineers because of their own expertise [1]. Current systems suffer from inconsistency of engineering intent, intensive human intervention and reduced accuracy. CAE solver setup, especially for CFD (Computational Fluid Dynamics), requires rich experience and strong background knowledge. CFD setup involves flow regime judgement and physical model selection.

In order to improve the CAD/CFD integration, this paper proposes a framework to automate the CFD solver setup and adaptive control. Three novel feature concepts are highlighted in this paper, where *fluid physics feature* and *dynamic physics feature*, are used to convey simulation intent; while *fluid functional feature* is used for defining design intent and propelling the formation of inter-feature associations. This work is a continued effort of CAE *boundary feature* and *effect feature* [2] concepts. This paper also presents the mechanism of how design and simulation intents should interact.

Integrating AI (Artificial Intelligence) approach with CFD has been reported in [3][4] with an expert cooling fan design system and a hybrid system for conducting local compressible flow analysis respectively. One useful direction of expert system application in CFD environment was to diagnose problems, aid decision making and provide best practices. For example, Wesley et al. [5] integrated AI and CFD to monitor the user input and inspect undesirable situations. Stremel et al. [6] implemented BPX, a best practice expert system, to guide the CFD projects. Though helpful to novice users, they served as just CFD auxiliary tools. So far, automated CFD solver selection and setup function has not been achieved.

Historically, CFD expert systems had been used when CFD solvers were still in the open-code form, and their integration requires "merge-coding" with special knowledge and training. Nowadays, more and more CFD commercial packages have offered user-friendly interfaces to improve the usability. However, the knowledge embedded in the CFD solvers is still unfamiliar to many users other than experts. Moreover, the benefits of introducing automated cycles of CAD/CFD integration have not been fully explored. In fact, the reliance

2212-8271 © 2016 The Authors. Published by Elsevier B.V.
Peer-review under responsibility of the Scientific Committee of the "9th International Conference on Digital Enterprise Technology - DET 2016.

on experts also makes it difficult to consistently support cyclic interactions from different engineering optimization domains and their invoked analysis simulators [7]. In our framework, an automated CFD solver package should inherit the design information from the design stage and transform it into the optimal simulation model with more detailed inputs and then, in turn, generate credible results. Thus, the modular integration of CAD/CFD with a coherent data flow structure will be useful to improve robustness and efficiency.

Nomenclature

Q	flow rate of gas
d	inner diameter of duct
μ	dynamic viscosity of gas
ρ	density of gas
R	gas constant
T	temperature of gas
k	specific heat ratio of gas
A	cross sectional area of duct
p	pressure of gas
a	speed of sound of gas
v	velocity of gas
Re	Reynolds number
Ma	Mach number

2. CAD/CFD integration framework

Conventionally, *design intent* mainly focuses on geometric modelling aspects [8]. Mun et al. [9] defined design intent as the functional requirement provided by customers which are a set of geometric and functional rules satisfied by the final product. From this definition, it is obvious that the formation of design intent starts from the customer's requirement for functions. Fig. 1 presents this whole product development process. Note that iterations are necessary before the final design is achieved. Designers fulfill the functional requirement based on engineering knowledge and develop the initial conceptual design with CAD model. In the transformation process, not only the CAD geometric model is transformed into the CFD mesh model, but also the simulation conditions as well as the setup parameters are transmitted into a CFD meta model.

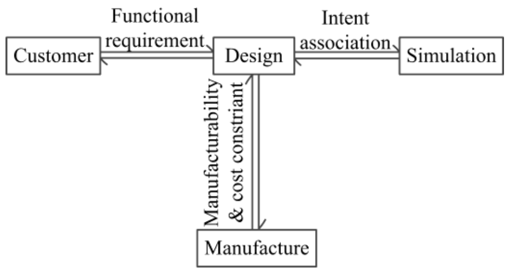

Fig. 1. Product development routine.

Nolan et al. [10] defined *simulation intent* as a collection of all the analysis, modelling and idealization decisions, and all the parameters required to create an adequate analysis model from an input CAD geometry. In the similar view, the authors suggest that the generation of simulation intent should occur in the transition process where the association with design intent should be available. Commonly, simulation result is used to check the initial design assumption validity and guide design optimization. Subsequently, the design is further modified to meet the manufacturability and cost constraint. Eventually, an acceptable design is returned for customer endorsement.

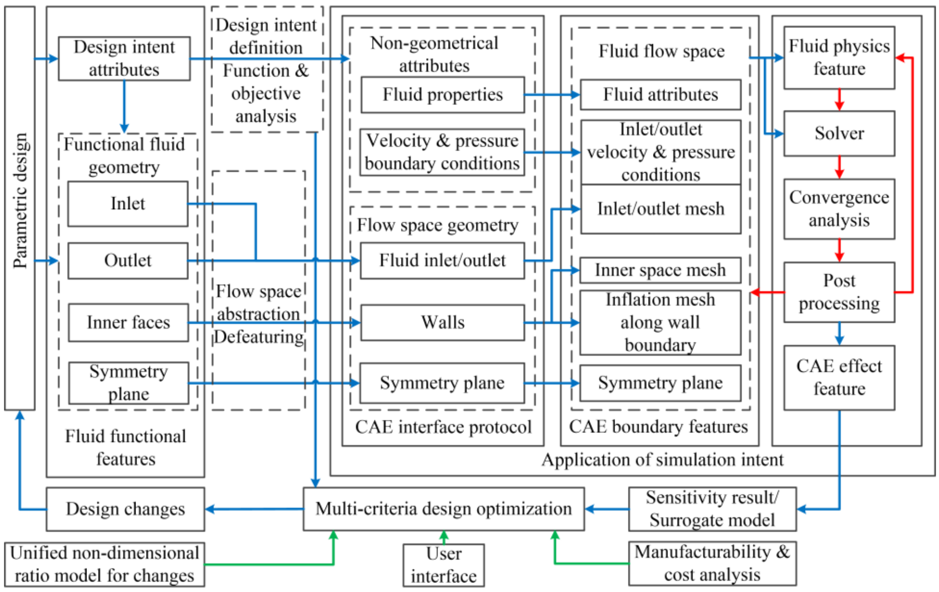

Fig. 2. CAD/CAE integration framework (Refined based on [2]).

In order to keep the consistency of design intent in different product development stages and to facilitate the correspondence of the design and simulation models, a CAD/CFD integration framework is proposed as shown in Fig. 2. The product design can be parameterized according to engineering knowledge. Based on the functional requirements, *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 geometry which is controlled by those attributes. The functional fluid geometry can be itemized as inlet, outlet, and inner faces enclosing fluid space and symmetry plane if there is any. In this way, the design intent can be fully conveyed to the downstream analysis stage with the fluid functional features. The CAD model of flow spaces can be extracted by Boolean operations. The face IDs of flow space geometry are assigned specific tags with attributions and boundary conditions attached, which will be recognized by CAE boundary feature [2]. Under the control of CAE boundary features, the fluid flow space can be meshed with designated mesh type according to different boundary properties. For example, inflation layer is applied along wall boundary to capture the boundary layer accurately. Meanwhile, the boundary conditions are assigned accordingly. The fluid flow space including discrete geometry, boundary conditions and non-geometrical fluid attributes, which are inherited from the design intent, is treated as the input of solving stage.

At the upper-right corner of Fig. 2, *fluid physics feature* is defined as object class with a characteristic set of fluid simulation setup parameters that entails a generic data structure and the related methods. Then, this fluid physics feature module also models a set of rules to select appropriate CFD solver regime applicable to each round of simulation. This module is also designed to implement knowledge and best practices which enable the conversion of input data, automated CFD solver setup and robust CFD model generation. Therefore, the simulation intent is embedded in the fluid physics feature instances. The detailed description of fluid physics feature will be illustrated in next section. Post processing could be conducted based on any converged run. The CFD model including mesh and solver setup parameters could be updated iteratively leading to a robust model setup. This is a unique requirement of CFD analysis which distinguishes it from linear engineering problems, such as stress analysis.

After achieving the result from the simulation of robust CFD model, the initial design can be modified heuristically to approach design objectives. Then, based on the new design, the updated CFD analysis will be obtained accordingly under the aforementioned scheme. The CAE effect features are extracted from the difference between the incremental analysis results [2]. Following that, sensitivity result or surrogate model could be obtained to provide optimization input. Coupling with optimization objectives derived from design intent, operational performance, manufacturability and cost analysis, the optimization process takes different constraints into consideration. Here, a unified non-dimensional ratio model is proposed to calculate the weights of different design criteria and to enable the measurement of performance increments between different designs. Finally, a closed CAD/CFD loop forms, which links the CAD domain and CFD domain consistently. Evidently, the design intent is adhered throughout the whole process, which is denoted by blue lines. At the same time, design intent and simulation intent are associated through the control of fluid functional feature, CAE boundary feature and fluid physics feature.

3. Implementation of the proposed feature concepts

In this paper, the fluid physics feature is established based on compressible flow scenario which is a complicated case in CFD application. The process of physical parameter analysis, CFD solver setup, convergence analysis and robust model generation is shown in Fig. 3.

The initial values are the fluid attributes which are carried over by design intent to achieve specific product functional performance. The physical relationships between these parameters are applied as rules which have the form of equations listed below.

$$A = \frac{1}{4}\pi d^2 \quad (1); \qquad p = \rho RT \quad (2); \qquad a = \sqrt{kRT} \quad (3)$$

$$v = \frac{Q}{A} \quad (4); \qquad \mathrm{Re} = \frac{\rho v d}{\mu} \quad (5); \qquad \mathrm{Ma} = \frac{v}{a} \quad (6)$$

Consecutively, the parameters in step 1, 2 and 3 can be acquired through forward-chaining [4]. It should be noted that the medium used in this case is ideal gas. If not, the knowledge base should be extended correspondingly. Parameters like velocity and pressure are usually assigned to the boundaries as boundary conditions. As a consequence, they can be used to check the validity of fluid flow space which will be the input of the CFD solver. The Reynolds number and Mach number are dimensionless quantities which determine the flow regime. Based on the Reynolds number, a turbulence model will be selected if the flow is turbulent. The Mach number judges the compressibility of the flow. Special model and setup are needed if the compressibility effects cannot be ignored. When the simulation is in the startup stage or faced with convergence problems, lower order discretization schemes like UDS and Euler implicit, as well as k-epsilon turbulence model if applicable, have the priority to be selected in order to assist convergence.

The index i will be updated after each simulation run. Index C and D denoting the status of convergence will also be updated accordingly. All the simulation solver setup is recorded no matter if the simulation is converged or not. If a simulation is converged, post processing will be conducted to check whether simulation results match the initial assumptions and expected accuracy. If not, grid adaption will be triggered based on the existing simulation result to examine where local mesh refinement is required. Simultaneously, the peak value of Reynolds number and Mach number will be checked to see whether the flow regime needs to be changed. If a simulation diverged, the system would try to alter the solver setup to achieve convergence. It is highlighted here that each time when a new iteration starts, only one change is made in the solver configuration to obtain the sensitivity towards different simulation schemes. If the simulation still has convergence problem after several successive runs, human intervention is needed to diagnose the problem which requests more expert knowledge.

After rounds of successful simulations, the mesh is further refined, which is favorable for the application of higher order schemes and advanced turbulence model if the flow is turbulent. Higher order schemes always have priority, when the iteration index is high. Thus, the final simulation quality can be guaranteed. The cycles stop when all the simulation requirements are satisfied. As mentioned earlier, all the simulation setup is recorded dynamically for each iteration during this process. Li et al. [11] define the interim features between various manufacturing operations as dynamic features. Applying the similar dynamic feature concept, in this work with the situation of CFD model generation, we define *dynamic physics feature* as the intermediate states of the fluid simulation model including flow properties, grid distribution and discretization scheme. The dynamic physics feature is developed to facilitate the generation of *robust simulation model* which is defined as the applicable CFD regime and simulation setup template with validated physics conditions, and converges into reasonable and accurate results.

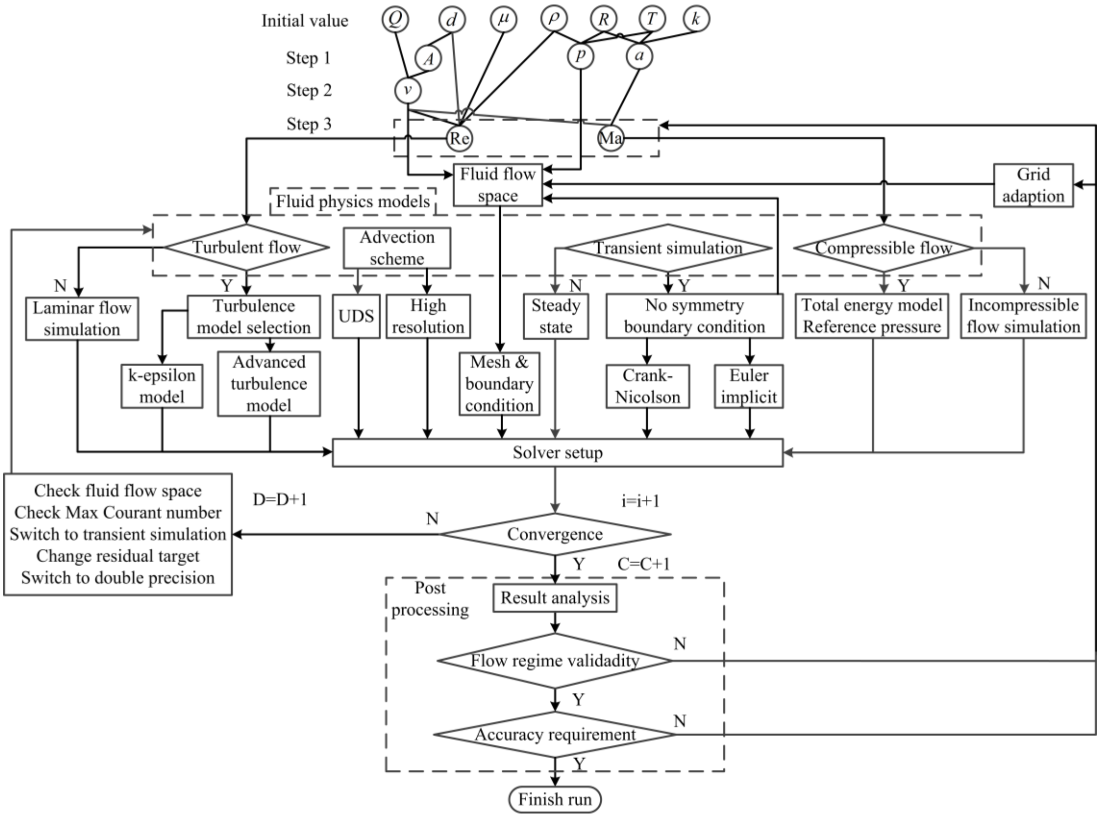

Fig. 3. CFD simulation robust model generation cycles.

4. Case study

The case study used an OCD (Outflow Control Device) applied in SAGD (Steam Assisted Gravity Drainage) process to show how the system functions. As shown in Fig. 4 (a), OCDs regulate the flow rate of steam flowing to the formation under a given pressure drop. The slotted liner covering the OCD is used to protect the device against the sand surrounding the well. When steam flows into the device, a portion of it flows to the outer space through the nozzles radially located on the device. The majority of steam flows to the downstream. Dry steam at the temperature of 500 K is assumed to be pumped into the injection well at a flow rate of 0.25 m^3/s. Fig. 4 (b) shows that the inner diameter *d* of the control parameter for the functional fluid geometry which is itemized as inlet, outlets, inner faces and symmetry planes. In this way, the fluid functional feature is fully defined, which conveys the design intent to the next stage in the integration loop. Because of the symmetrical geometry, a quarter of the CAD model is extracted in Fig. 5 (a). Tags with attributions are assigned to the real and virtual faces to transmit boundary information in CAD/CFD conversion. The fluid domain (see Fig. 5 (b)) is abstracted by Boolean operation. The faces which are associated with the part geometry take over the corresponding tags which are used to guide the mesh generation shown in Fig. 5 (c). Consequently, CAE boundary features are established resulting in the generation of fluid flow space which is the input of fluid physics feature module.

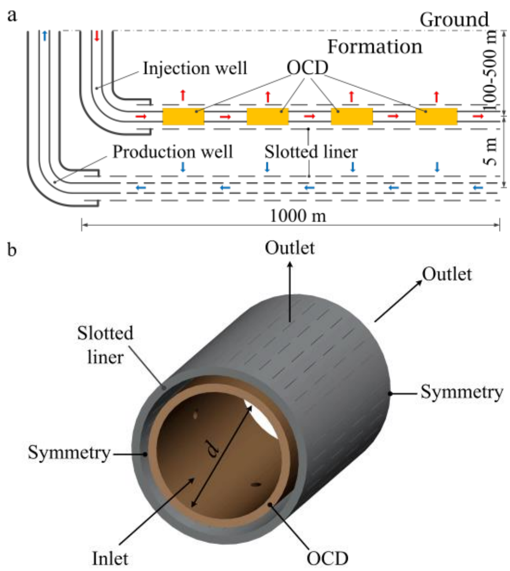

Fig. 4. (a) SAGD well completion [12]; (b) OCD and slotted liner.

Table 1. Initial value of physical parameters.

Physical parameter	Value	Unit
Q	0.25	m^3/s
d	0.15	m
μ	1.66×10^{-5}	kg/(m·s)
ρ	13	kg/m^3
R	461.5	J/(kg·K)
T	500	K
k	1.327	N/A

Table 2. Value of physical parameters in step 2.

Physical parameter	Value	Unit
A	0.0177	m^2
p	3000	kPa
a	553	m/s

The initial values of the physical parameters are given in Table 1. Table 2 shows the parameters calculated in step 2. The inlet velocity is found to be 14.1 m/s in step 3. In step 4, the Reynolds number is calculated to be 1.66×10^6, which is much bigger than the turbulence transition Reynolds number in a pipe. The Mach number is 0.02, which is much less than 0.3. So, the flow is assumed to be uncompressible turbulent flow. Using ANSYS CFX as the solver, the simulation converges and the Mach number contour obtained from this initial run is shown in Fig. 6. The fluid physics models selected in this simulation is shown in Table 3.

Though the initial run is converged, it is found that the maximum Mach number is bigger than 0.3 which means the compressibility effects cannot be ignored. Based on the result of the simulation, grid adaption is conducted, which is shown in Fig. 7. Total energy model is selected to activate the compressible flow simulation. The other solver setup parameters remain unchanged in the next run.

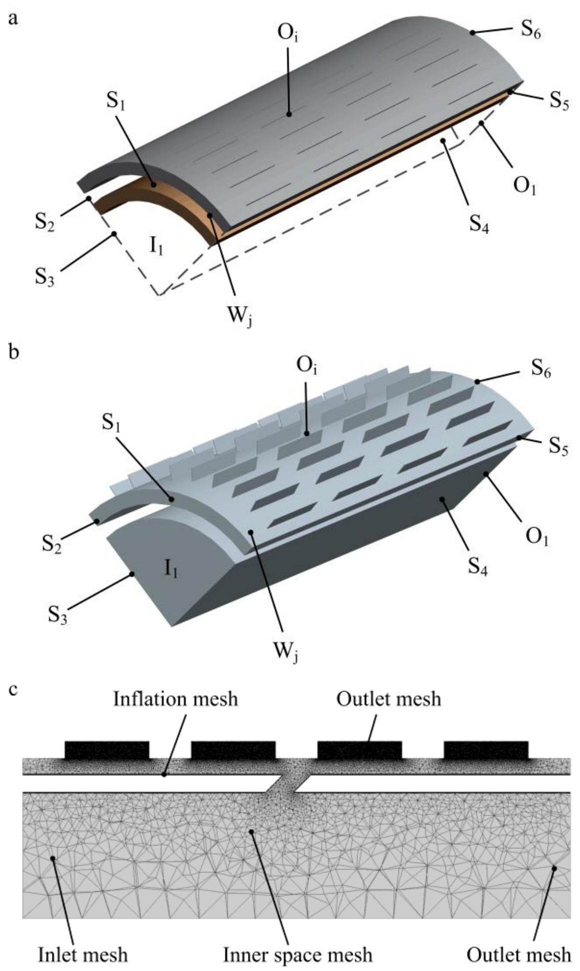

Fig. 5. (a) extracted quarter part geometry; (b) abstracted quarter fluid domain; (c) initial mesh generation.

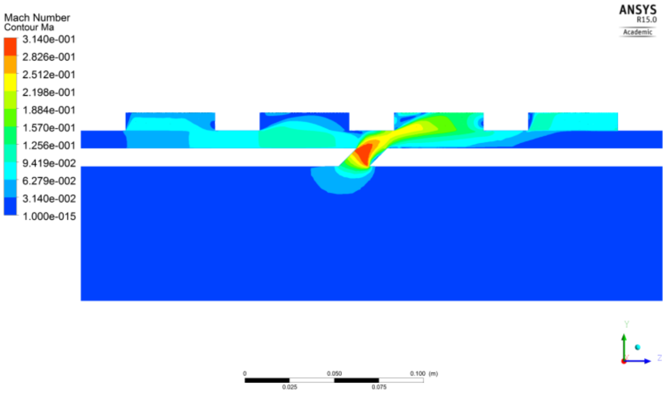

Fig. 6. Mach number contour of initial run.

Table 3. Fluid physics models of initial run.

Fluid physics model (i=1; C=1; D=0)	Selection
k-epsilon	Y
Advanced turbulence model	N
UDS	Y
High resolution	N
Steady state	Y
Transient	N
Incompressible flow model	Y
Compressible flow model	N

After rounds of this process, the dynamic physics feature is developed to enable the acquisition of sensitivity towards different physical models. Consequently, a robust simulation model is obtained. The physics models selected for this robust simulation model is shown in Table 4. The velocity vectors and streamlines derived from this final run are shown in Fig. 8 and Fig. 9 respectively. Fig. 10 illustrates the mechanism of how simulation intent is associated with design intent seamlessly in this OCD case.

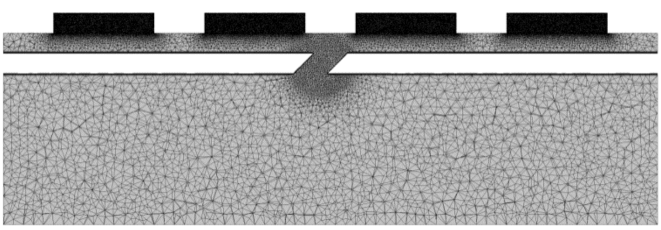

Fig. 7. Grid adaption.

Table 4. Fluid physics models of final run.

Fluid physics model (i=4; C=4; D=0)	Selection
k-epsilon	N
Advanced turbulence model	Y
UDS	N
High resolution	Y
Steady state	Y
Transient	N
Incompressible flow model	N
Compressible flow model	Y

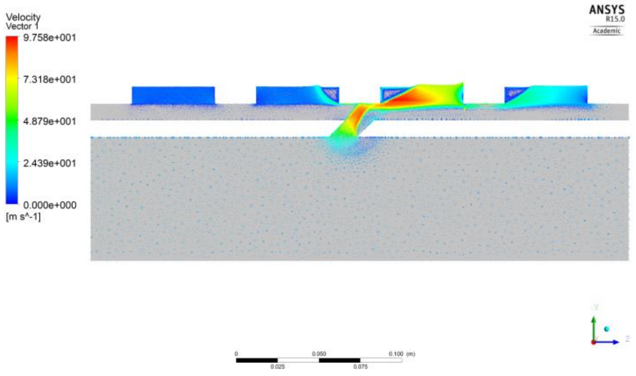

Fig. 8. Velocity vectors of final run.

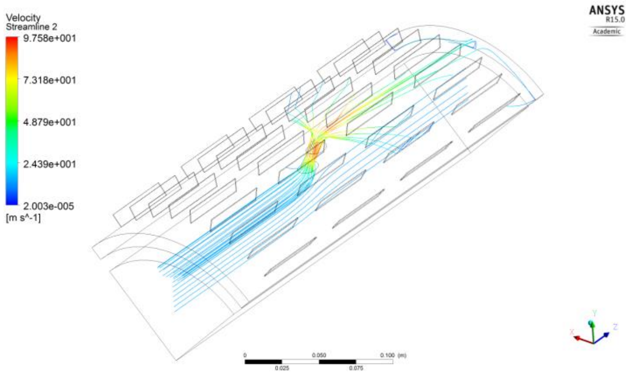

Fig. 9. Streamlines of final run.

5. Conclusion

This paper contributes to the smooth CAD/CFD integration, and a framework has been proposed to address the solver usability problem faced by many CFD users in product development process. Fluid physics feature and dynamic physics feature, which convey the simulation intent, have been proposed to assist input-data processing, solver setup, convergence analysis, and robust simulation model generation. In this way, semi-automation could be achieved and CAD/CFD integration could be boosted to a higher coherence and consistency level. Through the collaboration of *fluid functional features, CAE boundary features, fluid physics features, and dynamic physics features*, design intent and simulation intent are associated seamlessly, leading to the reuse of design intent throughout the whole integrated CAD/CFD system. The engineering effort involved in product development and the reliance on CFD experts are expected to be reduced. To show the potential benefits of the proposed forward CAD/CFD integration method, a case study of SAGD outflow control device is presented. The prototype of the CAD/CFD integration system will be developed in the future. Currently, the authors also intend to explore topology and material optimization for such flow control devices by incorporating previously developed gradient-based [13] and surrogate model-based [14] optimization techniques into CFD solvers.

Acknowledgements

The authors would like to acknowledge the financial support from China Scholarship Council, University of Alberta Doctoral Recruitment Scholarship, NSERC Discovery Grant, RGL Collaboration Fund and Alberta Innovates Graduate Student Scholarship.

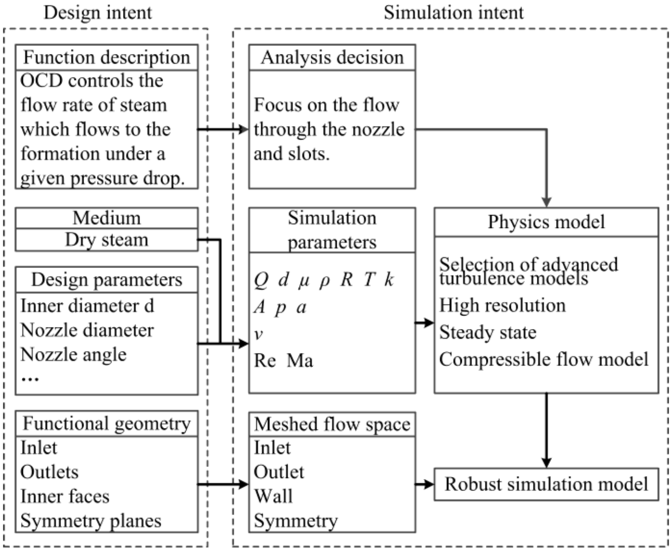

Fig. 10. Design and simulation intent association.

References

[1] Shephard MS, Beall MW, O'bara RM, Webster BE. Toward simulation-based design. Finite Elements in Analysis and Design, 40.12 (2004): 1575-1598.

[2] Li L, Ma Y. CAD/CAE associative features for cyclic fluid control effect modeling. Computer-Aided Design and Applications, 13.2 (2016): 208-220.

[3] Andrews AE. Progress and challenges in the application of artificial intelligence to computational fluid dynamics. AIAA journal, 26.1 (1988): 40-46.

[4] Dannenhoffer JF, Baron JR. A hybrid expert system for complex CFD problems. Proceedings of AIAA 8th Comput. Fluid Dyn. Conf., AIAA Paper, (1987): 1111.

[5] Wesley LP, Lee JD, Rodman LC, Childs RE. Toward an integrated CFD expert system environment. AIAA Paper, (1998): 1005.

[6] Stremel PM, Mendenhall MR, Hegedus MC. BPX–A Best Practices Expert System for CFD. 45th AIAA Aerospace Sciences Meeting and Exhibit, (2007): 974

[7] Gelsey A. Intelligent automated quality control for computational simulation. Artificial Intelligence for Engineering, Design, Analysis and Manufacturing, 9.05 (1995): 387-400.

[8] Price MA, Robinson TT, Soban D, Murphy A, Armstrong CG, McConnell R, Roy R. Maintaining design intent for aircraft manufacture. CIRP Annals-Manufacturing Technology, 62.1 (2013): 99-102.

[9] Mun D, Han S, Kim J, Oh Y. A set of standard modeling commands for the history-based parametric approach. Computer-aided design, 35.13 (2003): 1171-1179.

[10] Nolan DC, Tierney CM, Armstrong CG, Robinson TT. Defining Simulation Intent. Computer-Aided Design, 59 (2015): 50-63.

[11] Li Y, Liu X, Gao JX, Maropoulos PG. A dynamic feature information model for integrated manufacturing planning and optimization. CIRP Annals-Manufacturing Technology, 61.1 (2012): 167-170.

[12] Li L, Lange CF, Ma Y. Modelling of outflow control in steam assisted gravity drainage process. 23rd Annual Conference of the Computational Fluid Dynamics Society of Canada, (2015).

[13] Liu J, Ma Y. 3D level-set topology optimization: a machining feature-based approach. Structural and Multidisciplinary Optimization, 52 (2015): 563-582.

[14] Liu J, Ma Y, Fu J, Duke K. A novel CACD/CAD/CAE integrated design framework for fiber-reinforced plastic parts. Advances in Engineering Software, 87 (2015): 13-29.