SSRG International Journal of Mechanical Engineering
ISSN: 2348 – 8360 /doi:10.14445/23488360/IJME-V8I8P102

Volume 8 Issue 8, 7-10, Aug, 2021

Discontinuous Finite Element Analysis of Counter Flow Heat Exchanger Unit Cell

Suliman Alfarawi[1], Azeldin El-sawi[2], Hossin Omar[3]

[1,2,3]*Staff member, Department of Mechanical Engineering, University of Benghazi, Benghazi, Libya*

Received Date: 12 July 2021
Revised Date: 15 August 2021
Accepted Date: 29 August 2021

Abstract *- Computational fluid dynamics (CFD) analysis was conducted on parallel-plated counter flow heat exchanger using continuous and discontinuous meshing schemes. A unit cell of the counter flow heat exchanger was initially selected as a computational domain for testing the CFD metrics. The results of Nusselt number and friction factor using continuous meshing were compared to available methods in literature. Good agreement was found with 6 % and 1 % maximum deviations in Nusselt number and friction factor results, respectively. The CFD simulations were performed at different Reynolds numbers ranging from 100 to 2000 using the two approaches. The results of the two approaches were compared in terms of accuracy and computational time. It was found that the results of Nusselt number of discontinuous meshing approach are 8% overestimated only at higher Reynolds numbers, while the results of pressure drop of discontinuous meshing approach are 8.5% underestimated at higher Reynolds numbers. The discontinuous meshing approach is recommended for the preliminary design of a heat exchanger regardless of the complexity of the geometry as less memory and time are required.*

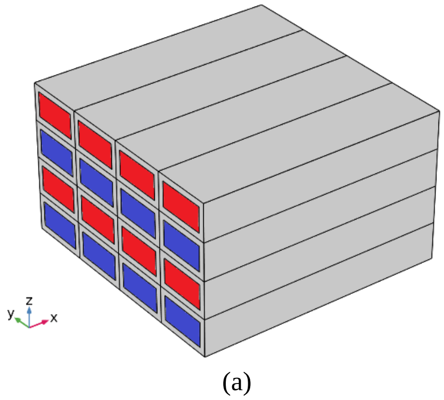
Keywords - *CFD, Counter-flow, Discontinuous, Heat Exchanger, Meshing.*

I. INTRODUCTION

Different types of heat exchangers can be found in various applications in industry. Thus, optimizing the heat exchanger's performance can lead to an efficient conversion of heat transfer. In literature, there are numerous studies to analyse and optimize heat exchangers based on analytical, numerical and experimental approaches depending on the problem complexity and the feasibility of the approach. Numerical methods approximate the solution of the problem (normally partial differential equations) that cannot be solved analytically based on the type of discretization. Those commonly encountered include finite difference method (FDM), finite volume method (FVM) and finite element method (FEM). FVM and FEM are suited for unstructured meshing and FEM is highly recognised in solving complex boundary value problems. However, the experimental and analytical approaches are needed when it comes to validation stage of the numerical methods. Nowadays, with the development of computational power, researchers and engineers use computational fluid dynamics (CFD) in the analysis of complex heat exchangers [1]. Comsol multiphysics software is one of the commercially available CFD packages that are based on the FEM. The FEM in the early stage, also known as Galerkin method, was applied to structural mechanics and took decades to be used in fluid dynamic problems. Different formulations of the FEM have been developed for thermal and fluids problems including continuous and discontinuous FEM [2-4]. In discontinuous FEM, the shape functions can be chosen so that the dependent variable or its derivative (or both) becomes discontinuous across the element boundaries, while maintaining continuity of the computational domain. This feature results in a reduction of core memory. Numerous studies can be found in literature using continuous FEM for simulations of different types of heat exchangers proving robust and accurate methodology [5-7]. However, few studies have been reported in literature on using discontinuous FEM for conjugate heat transfer applications. One attempt was made [8] to apply discontinuous Galerkin (DG) method in conjugate heat transfer simulations of gas turbines. The developed code was verified with number of heat transfer applications suggesting the use of efficient time integration methods to ease the higher computational time. Within the scope of the present paper, it is motivated to analyse a counter flow heat exchanger using continuous and discontinuous FEM within Comsol multiphysics environment. Comparisons of the two methods in terms of accuracy and computational time to be presented.

II. MODEL DESCRIPTION

The heat exchanger under study is a parallel-plated counter flow heat exchanger (Fig.1.a). The computational domain is a unit cell having two channels with hot and cold water (Fig.1.b). The mini-channels have a rectangular cross section with 1-mm height and 2-mm width. The length of the channels is 1 cm. The solid material of the channels is made of copper with wall thickness of 0.25 mm. The geometrical and operational parameters are summarized in Table 1.



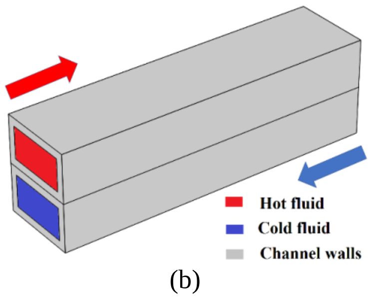


Fig.1: Counter flow heat exchanger, a) multi-layered, b) unit cell.

Table 1: Boundary conditions and geometrical parameters.

Designation	Value/Unit
Inlet hot water temperature	315 K
Inlet cold water temperature	300 K
Length× Height× Width	1 cm×1 mm×2 mm
Thickness (t)	0.25 mm
Working fluid	Water
Solid material	Copper

A. Boundary Conditions

All boundary conditions are set based on the model assumptions. The inlet temperature of the hot water is 315 K, and the inlet temperature of the cold water is 300 K. The flow is considered to be incompressible fully developed laminar flow. The inlet velocity of the fluid is determined according to Reynolds numbers range used which varied from 100 to 500 with a step of 100, and from 1000 to 2000 with a step of 500. Moreover, the outlet is exposed to zero gage pressure with normal flow condition. Thermal insulation boundary condition is applied on the external surfaces of the unit cell. "***Pair Continuity***" boundary condition is only applied when discontinuous meshing is used. This enforces continuity of fields and balances fluxes between two adjacent objects (the solid and all of the fluid domains).

B. Mesh Selection

a) Continuous Mesh Settings

The geometry of the investigated counter flow heat exchanger, was created and meshed by using the model builder inside COMSOL Multiphysics 5.5 (Fig.2). The mesh is plotted below, and it can be seen that it is continuous between the fluid domain and the solid domain. The mesh is composed primarily of tetrahedral elements, with a boundary layer mesh applied on the fluid side of the channel walls to resolve the velocity and thermal gradients near the wall. Since the channels are uniform in shape, the left faces of the hot and cold fluids can be meshed with free triangular elements and swept over the entire channel length.

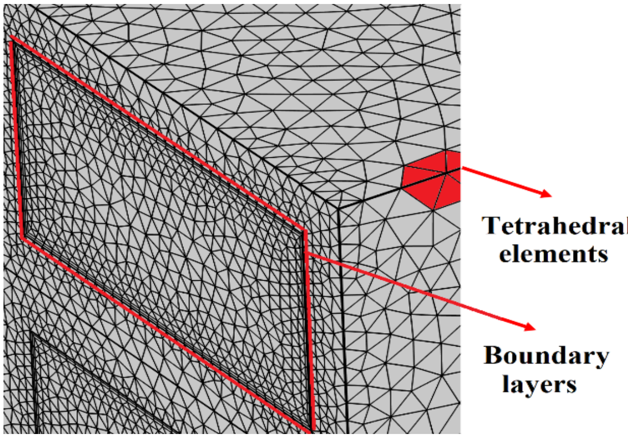


Fig.2: Continuous meshed geometry.

b) Discontinuous Mesh Settings

Two different objects are defined within the geometry sequence before meshing. The first object is the metal part, the solid through which the fluid flows. The second object is the combination of all of the fluid flow domains; that is, a single object that is composed of several different domains, which are created in the geometry sequence. These domains are nonoverlapping to form an assembly in which all mating faces between these objects are recognized automatically by the software as ***identity pairs***.

C. Mathematical Model

Conjugate heat transfer model is adopted in this study using stationary incompressible non-isothermal flow with Boussinesq approximation. Based on the finite element method, the discretised governing equations of fluid flow and heat transfer (continuity, momentum and energy equations) are solved by segregated solvers using iterative methods which require less memory compared to direct solver in a fully coupled mode. The algebraic multi-grid (AMG) solver with Parallel Sparse Direct Linear Solver (PARDISO) as a pre-conditioner provide robust solutions for large CFD simulations [9]. All simulations were performed on Intel® core™ CPU i7-3632QM PC runs at speed of 2.2 GHz with 16 GB RAM memory.

After solving continuity, momentum and energy equations, The heat transfer coefficient, h is calculated from the knowledge of net heat flux transferred from hot fluid to cold fluid and the fluid bulk temperature.

The friction factor, f from CFD results can be calculated based on the pressure drop obtained from simulations. Since the outlet of the channel is exposed to atmospheric pressure (zero-gauge pressure), the pressure drop is directly calculated at the inlet. The friction factor formula for fully developed laminar flow through rectangular cross section [10] is to be used for validating the friction factor results of CFD simulations.

III. Results and Discussion

A. Results Validation

In order to judge the CFD results (the Nusselt number and friction factor) using continuous mesh, sensitivity analysis of the results was performed using four element sizes as shown in Table 2 at minimum and maximum Reynolds numbers of 100 and 2000, respectively. Grid-4 with total number of elements counts for 263,904, was chosen for CFD simulations. The results of Nusselt number were first compared to the LMTD method results at different Reynolds numbers. Good agreement was found with an average deviation of 6%. Meanwhile, the results of friction factor showed a deviation of less than 1% when compared to the friction factor formula for fully developed laminar flow (equation 8). This gives confidence in CFD metrics and the numerical analysis.

Table 2: Continuous mesh sequence and its sensitivity.

Grid	Number of elements	Re = 100		Re = 2000	
		Nu	f	Nu	f
1	25795	9.3214	0.612	22.865	0.02248
2	49530	9.2824	0.614	22.624	0.02485
3	114544	9.2230	0.615	22.420	0.02780
4	263904	9.1215	0.617	22.377	0.03026

In order to compare discontinuous meshing results with continuous meshing approach, a grid size must be chosen. It should be emphasized here that if the same meshing sequence and number of elements were selected, both results of the two approaches would converge to the same solution [9]. However, discontinuous meshing approach offers a unique feature that must be utilized in the computations. As discussed in section 2.2.2, the identity pair feature disconnects fluid domains from solid domains giving an advantage to mesh each domain separately. Therefore, if fluid domain is meshed with finer size, the solid domain can be meshed with coarser size. Also, (equation 9) is applied to enforce continuity between the identity pairs. From that point of view, the potential is to examine the discontinuous mesh with less number of elements for saving time and memory. Therefore, a minimum grid size (Grid-1) was selected to carry out the simulations for discontinuous meshing approach.

B. Comparison of the Two Approaches

a) Heat Transfer

A comparison was made between Nusselt number results for continuous and discontinuous mesh approaches as shown in Fig.3. As depicted, the results of discontinuous mesh approach are overestimated since the mesh density is coarse. However, at higher Reynolds number of 2000, the maximum deviation reaches 8% with an average deviation of 5% over the whole range of Reynolds number.

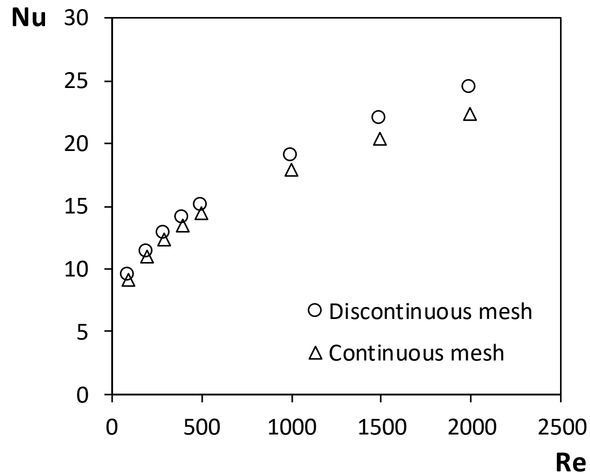


Fig. 3: Comparison of Nusselt number results for the two approaches.

b) Pressure Drop

In order to clearly pinpoint the deviation between the two approaches, the pressure drop results from the CFD simulations were plotted versus the mass flow rates for the two approaches as shown in Fig.4. The discontinuous mesh approach results are underestimated with a maximum deviation of 8.5% at Re = 2000 and with an average deviation of 4.5% over all range of mass flow rates.

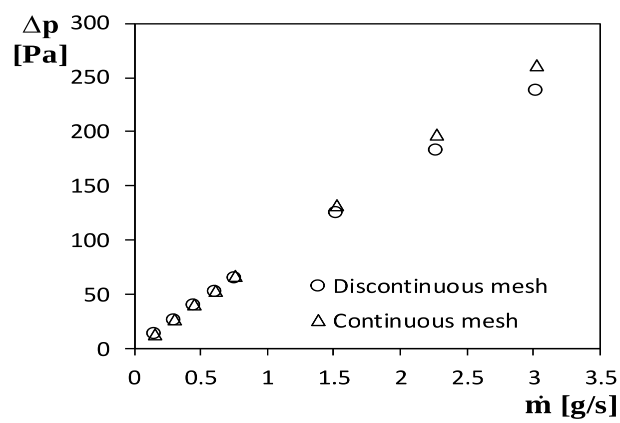


Fig. 4: Comparison of pressure drop results for the two approaches.

c) Computational Time

A comparison is made between the two approaches in terms of computational time for each simulation case. With less sacrifice with element quality and small counts of the elements and thus less memory, it can be seen from Fig.5 that running a simulation case with discontinuous mesh consumes just a few minutes regardless of the value of Reynolds number. Meanwhile, it took up to 39 minutes to run the simulation as Reynolds number increases to 2000.

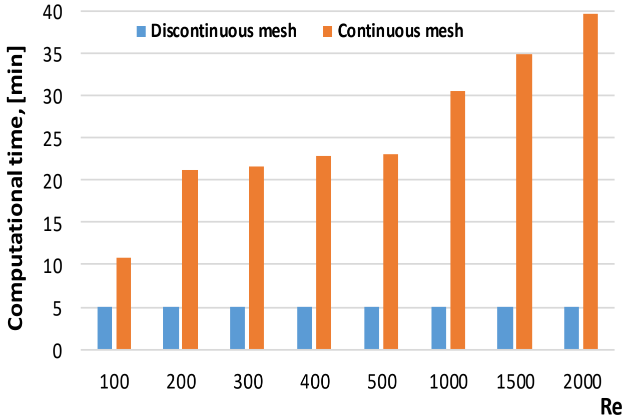


Fig. 5: Comparison of computational time for the two approaches.

IV. CONCLUSIONS

Using discontinuous meshing for a 3D CFD finite element based analysis of counter flow heat exchanger was presented in this study. A unit cell of a multi-layered parallel-plated counter flow heat exchanger was examined. CFD metrics initially was performed on the results of Nusselt number and friction factor using continuous meshing with good agreement found with available methods. Then, CFD simulations were performed at different Reynolds numbers using the continuous and discontinuous meshing schemes. The results of the two approaches were compared in terms of accuracy and computational time. The following points can be withdrawn from the present study:

• The discontinuous meshing approach is very useful in the stage of preliminary design of a heat exchanger for saving time and memory with little sacrifice of accuracy.

• The higher deviation between the results of the two approaches is more pronounced at higher Reynolds number range [1000-2000], Therefore, it is recommended to explore the use of discontinuous meshing approach for turbulent flow encountered in many heat exchangers applications.

REFERENCES

[1] M. M. A. Bhutta, , N. Hayat, M. H. Bashir, A. R. Khan, K. N. Ahmad, and S. Khan, CFD applications in various heat exchangers design: A review, Applied Thermal Engineering. 32 (2012) 1-12.

[2] J. N. Reddy, Introduction to the finite element method, 4th ed., McGraw-Hill Education. (2019).

[3] B. Q. Li, Discontinuous finite elements in fluid dynamics and heat transfer, Springer Science & Business Media. (2006).

[4] C. Ranganayakulu and K. N. Seetharamu, Compact heat exchangers: Analysis, design and optimization using FEM and CFD approach, John Wiley & Sons. (2018).

[5] N. H. Saeid and K. N. Seetharamu, Finite element analysis for co-current and counter-current parallel flow three-fluid heat exchanger, International Journal of Numerical Methods for Heat & Fluid Flow. 16(3) (2006) 324-337.

[6] T. Y. Ozudogru, O. Ghasemi-Fare, C. G. Olgun and P. Basu, Numerical modelling of vertical geothermal heat exchangers using finite difference and finite element techniques, Geotechnical and Geological Engineering. 33(2) (2015) 291-306.

[7] S. Alfarawi, Evaluation of hydro-thermal shell-side performance in a shell-and-tube heat exchanger: CFD approach, Journal of Advanced Research in Fluid Mechanics and Thermal Sciences. 66(1) (2020) 104-119.

[8] Z. R. Hao, C. W. Gu, and X. D. Ren, The application of discontinuous Galerkin methods in conjugate heat transfer simulations of gas turbines, Energies. 7(12) (2014) 7857-7877.

[9] COMSOL Multiphysics® v. 5.5. www.comsol.com. COMSOL AB, Stockholm, Sweden.

[10] J. P. Holman, Heat transfer. McGraw-Hill. (2010).