



HAL
open science

Semi mesh-less model for combined conduction-convection-radiative heat transfer : application to porous heat exchanger

Raj Narayan Konduru, Olivier Farges, Gilles Parent, Vincent Schick

► To cite this version:

Raj Narayan Konduru, Olivier Farges, Gilles Parent, Vincent Schick. Semi mesh-less model for combined conduction-convection-radiative heat transfer : application to porous heat exchanger. International Heat Transfer Conference 17, Aug 2023, Cape Town, South Africa. pp.10, 10.1615/IHTC17.50-20 . hal-04362764v2

HAL Id: hal-04362764

<https://hal.science/hal-04362764v2>

Submitted on 16 Jan 2024

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



HAL
open science

SEMI MESH-LESS MODEL FOR COMBINED CONDUCTION-CONVECTION-RADIATIVE HEAT TRANSFER: APPLICATION TO POROUS HEAT EXCHANGER

Raj Narayan Konduru, Olivier Farges, Gilles Parent, Vincent Schick

► **To cite this version:**

Raj Narayan Konduru, Olivier Farges, Gilles Parent, Vincent Schick. SEMI MESH-LESS MODEL FOR COMBINED CONDUCTION-CONVECTION-RADIATIVE HEAT TRANSFER: APPLICATION TO POROUS HEAT EXCHANGER. International Heat Transfer Conference 17, Aug 2023, Cape Town, France. pp.10, 10.1615/IHTC17.50-20 . hal-04362764

HAL Id: hal-04362764

<https://hal.science/hal-04362764>

Submitted on 1 Jan 2024

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

SEMI MESH-LESS MODEL FOR COMBINED CONDUCTION-CONVECTION-RADIATIVE HEAT TRANSFER : APPLICATION TO POROUS HEAT EXCHANGER

Raj Narayan KONDURU ^{1*}, Olivier FARGES ¹, Gilles PARENT ¹, Vincent SCHICK ¹

¹Université de Lorraine, CNRS, LEMTA, F-54000 Nancy, France

ABSTRACT

Existing deterministic methods can solve coupled heat transfer problems in complex 3D geometries, but the increasing complexity results in high computational time and memory usage due to grid-based approximations. This paper presents a semi-meshless method for solving coupled heat transfer problems. This method is based on a stochastic approximation, i.e., the Monte Carlo method. The resulting algorithm is less sensitive to the system's complexity. Thanks to advancements in computer acceleration technology, it is now easier to implement this method. This coupled algorithm was applied to high-temperature porous heat exchanger technology, which could be a potential solution for improved heat recovery. A Kelvin cell structure was used as the porous geometry and placed inside the heat exchanger. The heat recovered in terms of temperature rise was evaluated at different solid thermal conductivity and fluid inlet velocity parameters using the Monte Carlo method and compared to the deterministic method. The salient features of the Monte Carlo and deterministic methods were also discussed.

KEY WORDS: Radiation, Convection, Heat exchanger, Porous media, High temperature, Monte Carlo Method

1. INTRODUCTION

Burning fossil fuels to meet energy demands releases vast amounts of greenhouse gases (GHGs) such as CO₂, CH₄, and NO₂, causing egregious effects on the climate [1]. GHG emissions are primarily generated by energy, industry, building, and transport sectors [2]. As the global population continues to increase, energy demands also rise, increasing the likelihood of elevating GHG emissions. From 1990 to 2021, the warming effect of our planet's climate (Radiative forcing) due to long-lived GHGs rose by nearly 50 %, with carbon dioxide accounting for approximately 80 % of this increase. To reduce GHG emissions, with a primary focus on the energy sector, improving energy efficiency in high-temperature industries such as metallurgy, steel, cement, and glass is essential. The development of high-temperature heat recovery/storage/transport solutions is required to improve energy efficiency. In this work, we present a potential solution for heat recovery.

Shell-and-tube heat exchangers are widely used in process industries, as well as in many types of HVAC equipment, for high-temperature heat recovery at an industrial scale [3]. Although stochastic foams are also used, they are less in demand. Deployment of new heat exchanger technologies could revolutionize the way industries use energy. Research on foam heat exchangers is steadily gaining momentum. To improve heat exchanger efficiency, the use of porous materials could be an attractive alternative to conventional heat exchanger technologies. To achieve better heat recovery, foams should be designed to have less pressure drop and a high temperature difference between the entry and exit of an element filled with such foams. However, optimizing these foams in terms of geometrical properties such as strut diameter, pore diameter, and foam design types like Kelvin-cell, structural, and stochastic foams, to obtain better heat recovery, is a challenging task.

Before performing optimization, it is essential to understand the physics and control the computational time. Deterministic methods, which are commonly used to solve coupled heat transfer and fluid flow

*Corresponding Author: raj-narayan.konduru@univ-lorraine.fr

through porous heat exchangers in 3D, use numerical approximations that consume significant computational time and memory. This computational effort is directly proportional to the complexity of the geometry. To address this problem, a semi-meshless method can be used for simulating coupled conduction-convection-radiation heat transfer. We use the finite volume method to solve the velocity fields and the Monte Carlo method [4, 5] to solve coupled heat transfer. The algorithm accounts for the coupling of conduction in the solid phase, convection, and radiation in the fluid/void phase. This approach is shown to use less memory and computation time for complex geometries.

2. PROBLEM DESCRIPTION

In this paper, we propose using porous structures, such as Kelvin-cell, structural, or stochastic foams, inside a heat exchanger to improve heat recovery. We consider these porous structures as complex geometries for numerical analysis, rather than solving the equations using porous properties. Specifically, we use the Kelvin-cell type of porous structure. The geometrical model of the Kelvin-cell foam was provided by CTIF (Centre Technique des Industries de la Fonderie), and samples made of this foam were manufactured in cast iron by CTIF. We conduct experiments in our lab using air as the fluid medium for convective analysis. For radiative analysis, we treat the fluid as a transparent medium and the solid as opaque. To solve for conduction, we use a solid medium with the properties listed in table 1. All properties mentioned in table 1 are constant, and temperature dependence has not been taken into account. The solid medium corresponds to metallic materials such as cast iron, and its typical thermal properties are mentioned in table 1 except for λ_s . The solid thermal conductivity λ_s of cast iron is $40 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, but in our analysis, we also consider λ to be $1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$ to decrease conduction and thus increase the role of radiation in heat transfer. In order to validate our proposed method, we present a case study as shown in fig. 1 only for better visual understanding.

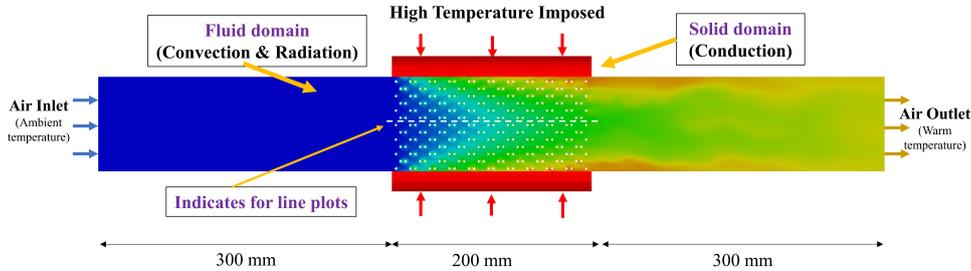


Fig. 1: Porous Heat exchanger - Sample simulation

Temperature contour : 300 K(Blue color) ; 1073 K (Red color) for $V_{in} = 2 \text{ m} \cdot \text{s}^{-1}$

Medium	ρ (in $\text{kg} \cdot \text{m}^{-3}$)	C_p (in $\text{J} \cdot \text{kg}^{-1} \cdot \text{K}^{-1}$)	λ (in $\text{W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$)	ϵ	μ (in $\text{Pa} \cdot \text{s}$)
Solid	8000	450	1 & 40	0.9	-
Fluid	1.225	1005	0.025	-	$1.81 \cdot 10^{-5}$

Table 1: Solid and Fluid media properties

The foam is a structured lattice with a Kelvin-cell pattern, as shown in fig. 2. The cell diameter (d_c) is equal to 14 mm and the strut diameter (d_{strut}) is equal to 2.5 mm. The foam part has a length (l) of 200 mm, an inner diameter (d_i) of 100 mm and its porosity (ϕ) is 85 %. For the given operating conditions of maximum temperature (1073 K) and V_{in} to be $2 \text{ m} \cdot \text{s}^{-1}$, Knudsen number was calculated to be less than

0.01. This indicates that the fluid can be treated by the continuum model, which assumes that the fluid properties can be defined at every point in the fluid. Mach number was calculated to be 0.003, which indicates that the flow can be considered incompressible and the incompressible flow equations can be used.

3. NUMERICAL MODELS

Numerical modelling is widely applied technique to tackle complex problems by computational simulation of such scenarios. It uses mathematical models to describe the physical conditions of the scenario using equations. Some partial differential equations are difficult to solve directly, these numerical models can approximate the solution. In this work, two types of numerical methods were used and are discussed in following sections.

3.1. Deterministic Method: These methods involve numerical approximations such as Finite Difference method (FDM), Finite Volume method (FVM), and Finite Element method (FEM). In this work, the Finite Volume method (FVM) was used to solve the coupled heat transfer problem using a Conjugate heat transfer model. ANSYS Fluent® software was employed for the simulation. It is important to note that only steady-state simulations were carried out in this study. This numerical simulation involves three steps.

Geometry. The Kelvin-cell structure, which serves as the solid medium is shown in fig. 2. An enclosure was developed around this structure to serve as the fluid medium, and both media were perfectly interconnected to form a perfect interface. The geometry was checked for errors and, once confirmed to be error-free, moved on to the next operation.

Meshing. Meshing is a process of dividing the volume of interest into smaller sub-volumes called elements. In this work, polyhedral elements were used as a mesh type for better stability when dealing with complex geometries [6], [7]. Mesh errors were checked and a grid independence test was performed. An element size of 0.5 mm and 1.0 mm were found to provide a stable solution, and the latter was chosen for further simulations due to its lower computational memory and time compared to the former.

Solver and Post-processing. In this study, the Fluent® solver was employed to simulate the heat transfer process. The Conjugate heat transfer model was utilized for both constant density and perfect gas density cases. To account for radiation heat transfer, the surface-to-surface radiation model was employed. The results obtained from the simulation were then post-processed and discussed in further sections.

It should be noted that the simulation was performed on the full 3D geometry. The geometry is not axially symmetric, but has two planes of symmetry. The entire structure was modeled in our study to test its capability in handling complex geometries. Indeed, our aim is to develop a model that can work for any complex and even stochastic structures. For testing purposes, we considered a Kelvin cell structure. However, for non-structured foams, symmetry boundary conditions cannot be applied, and the entire domain must be computed. Hence, it is better to compare the computational resources required for whole domains and complex geometries to evaluate the model's capabilities.

3.2. Semi Mesh-less Method: This method is a combination of deterministic methods using FVM (Mesh-based approximation) to obtain velocity fields and a stochastic method, i.e., Monte Carlo method (Mesh-less), to solve the coupled heat transfer problem, as proposed by Loris Ibarrart et al. [8], to obtain the temperature at any prescribed point in the domain. As per our application, we could calculate the temperature at the outlet directly using this method instead of calculating the whole field as performed

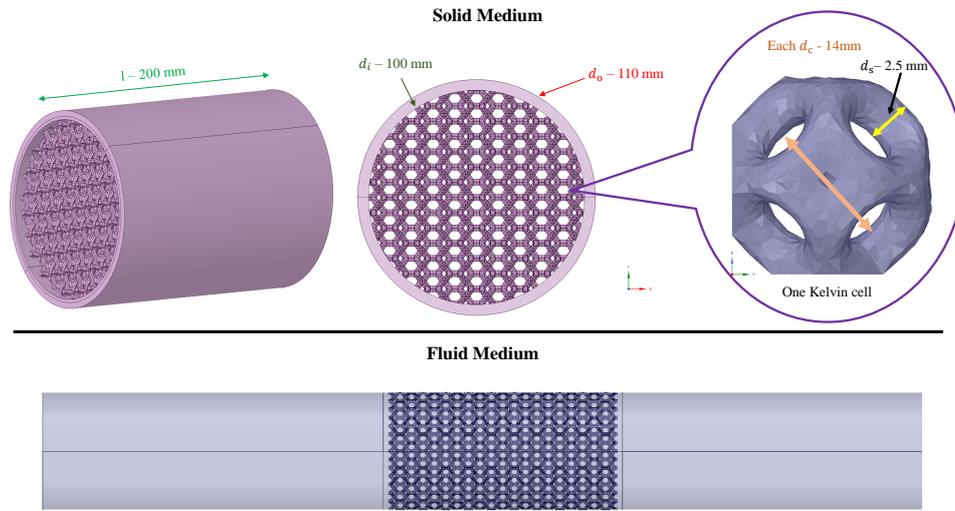


Fig. 2: Geometry Model

in DM. This helps to compute the global heat flux by knowing the inlet and outlet temperatures and flow rates of the fluid which allows to quantify the capabilities of the heat exchanger.

The Monte Carlo method only requires the surface mesh of the interface separating the solid medium from the fluid medium, whereas deterministic methods need a volumetric mesh of both media. The random walk technique is the main principle behind this method. To compute the temperature at location \mathbf{x} , a large number of paths crossing the complex geometry are calculated. The temperature is the expectation of a random variable $T(\mathbf{x})$. This random variable is determined from a path inside the calculation domain. A path starts at the location and at the time where and when the temperature needs to be computed. Each path can follow a conductive path in the solid (considered here opaque) or a radiative path in the void phase (considered here perfectly transparent) or a convective path. The walk/path step size and direction are based on random number generators and probabilities derived from partial differential equations. Some famous random walk techniques are shown in fig. 3 for one realization only. Multiple realizations are required to initialize, and then the expectation is computed. It is suggested that the reader refers to [8] for a detailed mathematical formulation and analytical validation of the coupled algorithm. For the sake of brevity, we will not discuss the derivation in this paper. For better understanding, the algorithm is described in fig. 4.

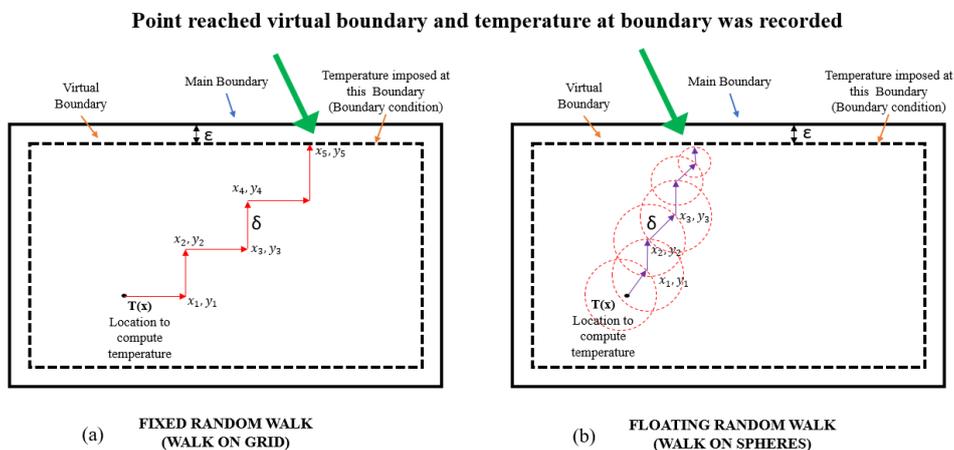


Fig. 3: Random walk techniques

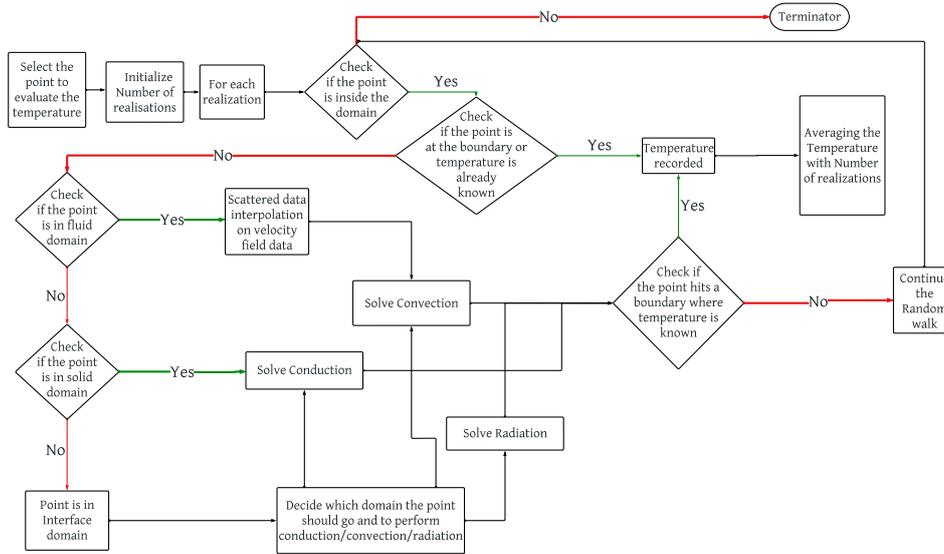


Fig. 4: Semi Mesh-less Algorithm

Conduction is solved by using the floating random walk method in the solid medium as shown in fig. 3(a) [9]. As air is considered a transparent medium, the surface-to-surface interaction model is used to calculate radiation in the fluid medium [10]. Convection is solved by transforming the energy equation to a random walk on the grid in the fluid domain as shown in fig. 3(b). To solve the energy equation in eq. (1), prior knowledge of the velocity field is required, as the path moves randomly through the domain, as shown in fig. 3. Thus, an interpolation technique is needed to estimate velocities between grid points. To handle complex data, we employed scattered data interpolation [11]. We performed a validation test on a 3D complex geometry, specifically the flow through a Kelvin-cell foam placed at the center of a long tube with a diameter of 100 mm and a length of 800 mm. The inflow velocity V_{in} was set to $0.1 \text{ m} \cdot \text{s}^{-1}$, and the fluid medium was air with a Reynolds number of approximately 660. We validated our interpolation technique by comparing the velocity profile along the horizontal axis with the expected results, as shown in fig. 5b. For reference, the velocity contour for this test case is shown in fig. 5a.

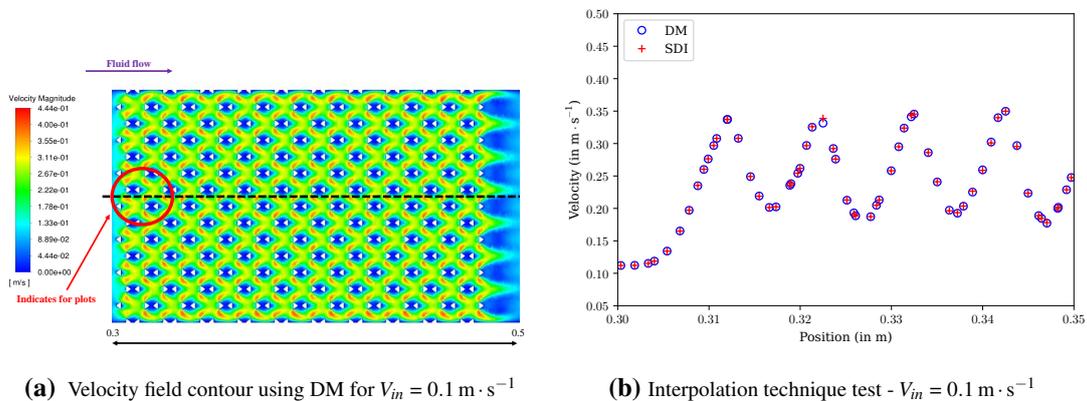


Fig. 5: Scattered data interpolation test.

The results of the interpolation technique were found to be in good agreement with the results obtained from the deterministic method (Fluent®). However, it should be noted that an increase in the amount of data results in longer computational times. In comparison to the radial basis interpolation technique [11], this method is preferred as it requires less computational memory. The radial basis technique needs to

calculate the Euclidean distance between each point for interpolation, which results in the development of a large matrix. The second step involves discretizing eq. (1) using FDM and determining direction probabilities to enable the random walk. This is a type of fixed random walk.

$$\nabla \cdot (-\lambda_f \nabla T(\mathbf{x}) + \rho_f C_{p_f} \mathbf{V}T(\mathbf{x})) = 0, \quad x \in \Omega_f \quad (1)$$

The rearranged equation is written as,

$$T(x, y) = P_{x+}T_{x+} + P_{x-}T_{x-} + P_{y+}T_{y+} + P_{y-}T_{y-} + P_{z+}T_{z+} + P_{z-}T_{z-} \quad (2)$$

where,

$$P_{k+} = \frac{\frac{\alpha_f}{\delta}}{\frac{6\alpha_f}{\delta} + u + v + w} \quad P_{k-} = \frac{\frac{\alpha_f}{\delta} + j}{\frac{6\alpha_f}{\delta} + u + v + w}$$

$$k \in [x, y, z] \quad j \in [u, v, w] \quad \text{resp.} \quad \alpha_f = \frac{\lambda_f}{\rho_f C_{p_f}}$$

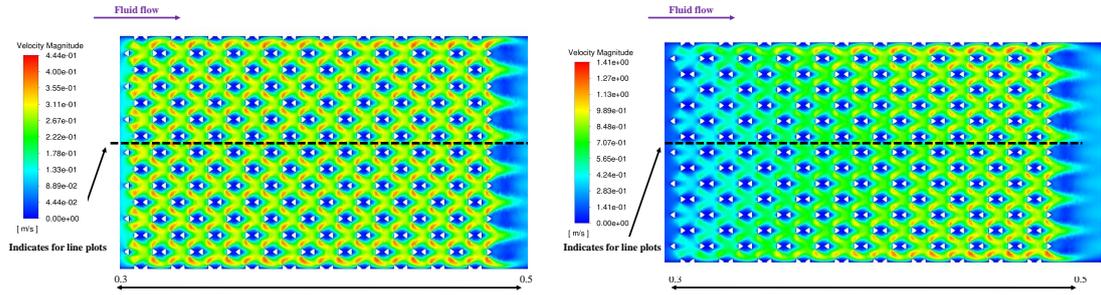
where k is location coordinate vector and j is the velocity vector. The finite difference method formulation (Method-1) and Pecllet formulation (Method-2) were both applied to the energy equation in eq. (1), as suggested by [8]. Both methods were found to be in good agreement. Method-2, which uses Pecllet formulation, was found to be more accurate than Method-1, which uses FDM, by a margin of 0.1 % when compared to the finite volume method (FVM). Nonetheless, Method-2 requires more computational time than Method-1. Therefore, we used Method-1 in this study based on a trade-off between computational time and accuracy.

4. VALIDATION

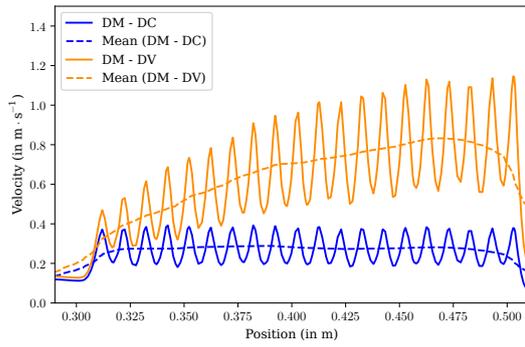
Using both deterministic and semi mesh-less methods, simulations were performed by considering inlet ambient temperature of air at 300 K, T_{imp} on the solid outer surface at 1073 K, air V_{in} at $0.1 \text{ m} \cdot \text{s}^{-1}$, $0.2 \text{ m} \cdot \text{s}^{-1}$ and $0.5 \text{ m} \cdot \text{s}^{-1}$ and λ_s at $1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$ and $40 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$ to simulate the case. No gravitational effects were considered. Near central horizontal profile as shown in fig. 1 is used to draw the plots. This profile is only located in fluid medium. Monte Carlo algorithm results are compared to Fluent® results, the latter being considered the reference solution. As per the fundamentals of this algorithm, velocity fields is used to compute the temperature but in return, no effect of temperature on velocity fields was considered (one-way coupling). So, this algorithm is equivalent to the constant density deterministic approach. Temperature in the domain was computed in deterministic approach using both variable density (perfect gas law) and constant density.

The objective to is study the performance of the algorithm and compare results generated by deterministic approach (Fluent®). Primarily results obtained by Fluent® will be discussed followed by comparison with Monte carlo algorithm. As per Mach number, we have assumed the flow is incompressible. Using Fluent®, we have performed the simulations for velocity magnitude and temperature for λ_s at $1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$ and V_{in} at $0.1 \text{ m} \cdot \text{s}^{-1}$ and $0.5 \text{ m} \cdot \text{s}^{-1}$ as shown in figs. 6c to 6f. The referenced velocity contours for fig. 6c are shown in fig. 6a and fig. 6b.

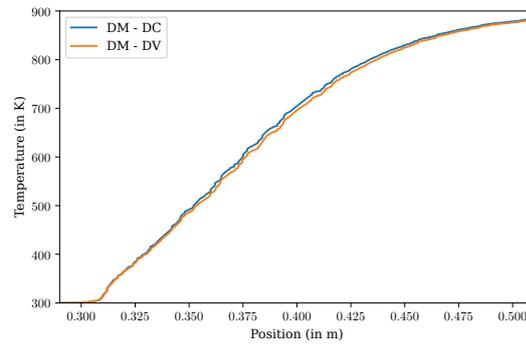
It was observed that a significant difference in velocity magnitude at lower V_{in} but no difference in temperatures for both chosen inlet velocities by keeping density as constant (DC) and variable (DV). For the case considering variable density, increase in velocity is due to the thermal expansion of the gas. It is lower at high V_{in} because the temperature rise is lower and it is due to the energy that is transferred by conduction from the outer part of the tube to the inner part and the amount of energy transported is limited by the thermal conductivity. If the gas flow rate increases, the temperature rise should decrease.



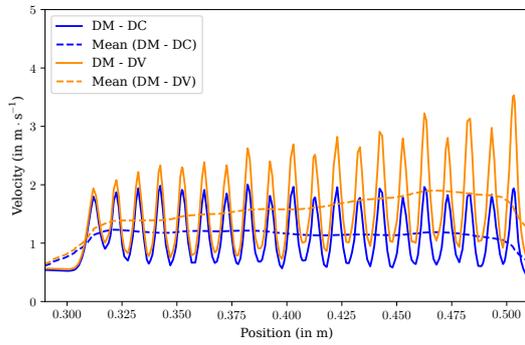
(a) Velocity contour (DC) : $\lambda_s = 1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, $T_{imp} = 1073 \text{ K}$, $V_{in} = 0.1 \text{ m} \cdot \text{s}^{-1}$, (b) Velocity contour (DV) : $\lambda_s = 1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, $T_{imp} = 1073 \text{ K}$, $V_{in} = 0.1 \text{ m} \cdot \text{s}^{-1}$



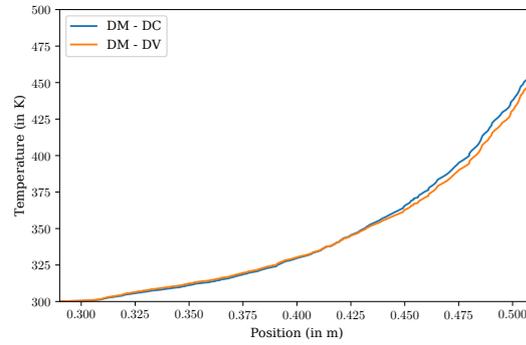
(c) $\lambda_s = 1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, $T_{imp} = 1073 \text{ K}$, $V_{in} = 0.1 \text{ m} \cdot \text{s}^{-1}$



(d) $\lambda_s = 1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, $T_{imp} = 1073 \text{ K}$, $V_{in} = 0.1 \text{ m} \cdot \text{s}^{-1}$



(e) $\lambda_s = 1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, $T_{imp} = 1073 \text{ K}$, $V_{in} = 0.5 \text{ m} \cdot \text{s}^{-1}$



(f) $\lambda_s = 1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, $T_{imp} = 1073 \text{ K}$, $V_{in} = 0.5 \text{ m} \cdot \text{s}^{-1}$

Fig. 6: Variation in temperature and velocity magnitude keeping density as constant and variable using deterministic method.

Because constant density and variable density are equivalent from the thermal point of view, for further analysis, we will consider the simulations of constant density as reference solution for comparing Monte Carlo algorithm.

This algorithm was validated against Fluent® results. The algorithm used 3000 realizations to compute the temperature at each chosen probe point which belongs to fluid medium only as shown in fig. 1. All results were at 1% standard deviation. But in fig. 7, we consider $\pm 2\sigma$ interval in plot representations to have a better confidence (i.e. 95% instead of 67%). As per our application, the main interest is all about the temperature rise between inlet and outlet of the tube. Based on this, L1 error was computed between DM results and MC results using eq. (3).

$$L_1 = \frac{|T_{DM}| - |T_{MC}|}{|T_{DM}|} \quad (3)$$

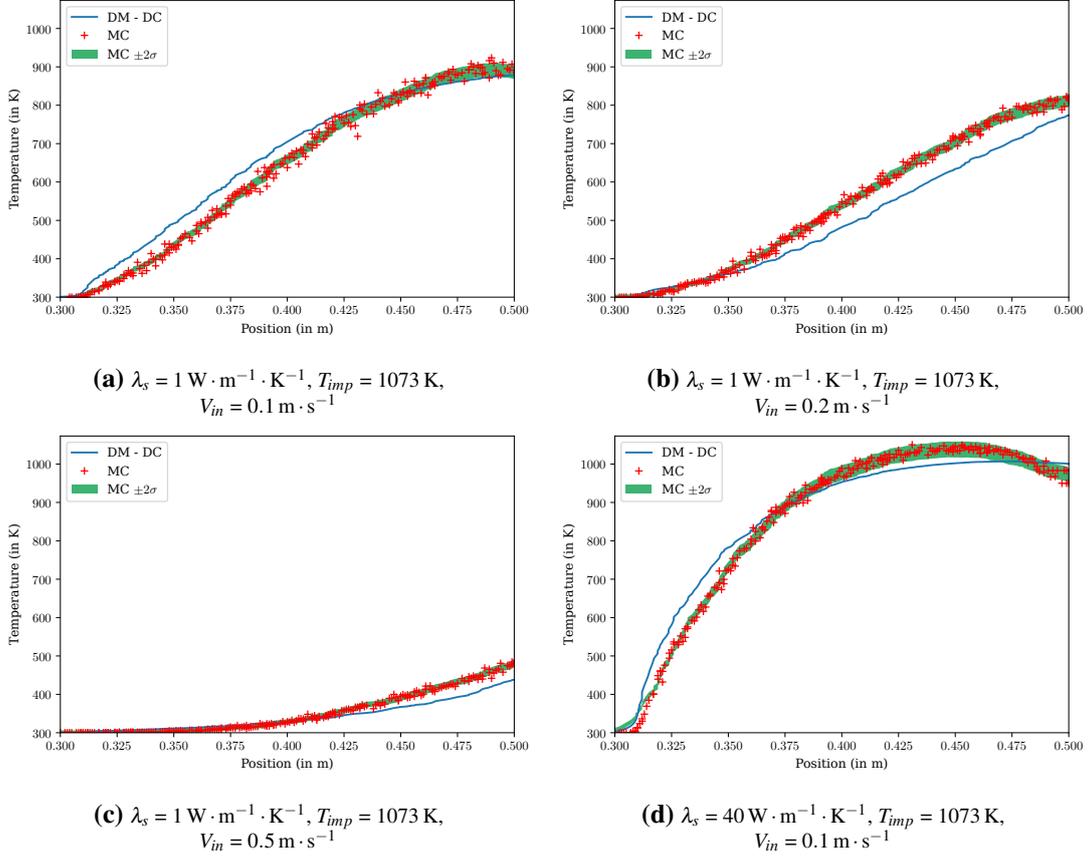


Fig. 7: Temperature profile computed using Monte Carlo algorithm compared with deterministic method.

As from figs. 7a to 7d, the trend is maintained throughout foam length. Since our goal is to compute the outlet temperature of the foam, eq. (3) was used to calculate the error at the outlet section only. For λ_s at $1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, L_1 error was found to be 2%, 3% and 5% for V_{in} at $0.1 \text{ m} \cdot \text{s}^{-1}$, $0.2 \text{ m} \cdot \text{s}^{-1}$ and $0.5 \text{ m} \cdot \text{s}^{-1}$ respectively. Similarly at λ_s at $40 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, L_1 error was found to be 2% for V_{in} at $0.1 \text{ m} \cdot \text{s}^{-1}$. It is also observed that changing λ_s does not lead to a significant change in the error %, but changing V_{in} increases the error. It was also observed that increasing λ_s also increases the temperature at the outlet. In general, this algorithm has good agreement with low input speeds and the percentage error increases with increasing input speed. The reason could be that increasing the input velocity in complex geometries leads to large fluctuations in temperature and velocity, which makes the algorithm less efficient to predict accurately. Secondly, it was ensured that the Deterministic Method (DM) solution has converged and is well approximated but it is not the exact solution.

Table 2 summarizes the features of Monte Carlo algorithm compared with deterministic method. In this application, the objective is to quantify the foam efficiency which depends both on the temperature rise along the foam and the flow rate. In the DM method, the entire temperature field needs to be calculated for every change in parameter, which can be computationally expensive and time-consuming. However, with the proposed method, we only need to obtain the outlet temperature at a few selected points, which significantly reduces the computational resources required. In contrast, the DM method calculates the whole field, which is not necessary for our objective of optimizing the porous structures. Furthermore,

Type	Monte Carlo Algorithm	Deterministic method
Geometry input file	Surface geometry (.obj)	Solid geometry (.stp)
Mesh type	Surface mesh	Volume mesh
Geometry complexity effect on memory	Consumes less memory	Consumes more memory
Results obtained	Probe locations	Total field
Computational time	45 min for each plot in fig. 7 150 probe points for each plot	6 hours (Steady State)
Parallelization	Yes	Yes
No. of processors used	32	32
Input data	Velocity fields required	Not required ; V & T are calculated

Table 2: Features of Monte Carlo Method over Deterministic method

for every change in parameter, the DM method requires calculating the velocity field from the beginning. For instance, if we need to obtain the temperature for two different solid thermal conductivity's, $1 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$ and $40 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, we can use the same $0.1 \text{ m} \cdot \text{s}^{-1}$ velocity field data from the DM method as the input for Monte Carlo simulation, and then we can obtain the temperature for both solid thermal conductivity's without recalculating the velocity field, which saves a considerable amount of time and resources.

5. CONCLUSIONS

Semi mesh-less method was applied to porous heat exchanger at high temperatures and compared with deterministic method at stationary state. This algorithm was then compared with results obtained by FVM method. Results were discussed along with the reasons of agreements and disagreements. Linearization of radiative term is one of the main assumption of this algorithm. In occurrence of high non-linearity situations, this assumption would not be valid. The mesh-less method could be very interesting to use in terms of reducing computational time and memory while maintaining the accuracy given by FVM methods. Future is development of full mesh-less method because for now the calculation is still limited by the CFD calculation for velocity fields. Porous heat exchanger experiments will be performed as an upcoming work. The experimental results will be used to validate the model discussed in this work. The transient behaviour of these system will be recorded experimentally to analyse the storage capacity.

ACKNOWLEDGEMENT

The Monte Carlo algorithm was implemented using Star-Engine libraries developed by Meso-star Company <https://gitlab.com/meso-star/star-engine.git>. Région Grand Est is acknowledged for funding Raj Konduru Phd Thesis. This work is part of the THERMOUSSE project funded by ICEEL. This work was performed using HPC resources from the EXPLOR mesocentre of University of Lorraine. <https://explor.univ-lorraine.fr/>

NOMENCLATURE

α	Thermal diffusivity [$\text{m}^2 \cdot \text{s}^{-1}$]	ε	Emissivity[]
δ	Step size[m]	λ	Thermal Conductivity[$\text{W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$]

μ	Dynamic viscosity [Pa · s]	l	Length [mm]
Ω	Domain []	MC	Monte Carlo Method []
ϕ	Porosity [%]	P	Probability []
ρ	Density [kg · m ⁻³]	T	Temperature [K]
\mathbf{V}	Velocity [m · s ⁻¹]	u	Velocity component in x-axis direction [m · s ⁻¹]
C_p	Heat Capacity [J · kg ⁻¹ · K ⁻¹]	v	Velocity component in y-axis direction [m · s ⁻¹]
d	Diameter [mm]	w	Velocity component in z-axis direction [m · s ⁻¹]
DC	Constant density []	x	Coordinate x-axis [m]
DM	Deterministic Method []	y	Coordinate y-axis [m]
DV	Variable density []	z	Coordinate z-axis [m]
j	Velocity vector []		
k	Coordinate vector []		

Subscripts

imp	Imposed	in	Inlet	i	Inner	f	Fluid
s	Solid	o	Outer	$strut$	Porous struts	c	Porous cell

REFERENCES

- [1] Abhinandan Kumar, Pardeep Singh, Pankaj Raizada, and Chaudhery Mustansar Hussain. Impact of covid-19 on greenhouse gases emissions: A critical review. *Science of The Total Environment*, 806:150349, 2022.
- [2] WF Lamb, T Wiedmann, J Pongratz, R Andrew, M Crippa, JGJ Olivier, D Wiedenhofer, G Mattioli, A Al Khourdajie, J House, et al. A review of trends and drivers of greenhouse gas emissions by sector from 1990 to 2018. *Environmental research letters*, 2021.
- [3] Sahan Trushad Wickramasooriya Kuruneru, Kambiz Vafai, Emilie Sauret, and YuanTong Gu. Application of porous metal foam heat exchangers and the implications of particulate fouling for energy-intensive industries. *Chemical Engineering Science*, 228:115968, 2020.
- [4] John R Howell. Application of monte carlo to heat transfer problems. In *Advances in heat transfer*, volume 5, pages 1–54. Elsevier, 1969.
- [5] Oleg Mikhailovich Belotserkovskii and Yury Iv Khlopkov. *Monte Carlo Methods in Mechanics of Fluid and Gas*. World Scientific, 2010.
- [6] Marcin Sosnowski, Jaroslaw Krzywanski, Karolina Grabowska, and Renata Gnatowska. Polyhedral meshing in numerical analysis of conjugate heat transfer. In *EPJ Web of Conferences*, volume 180, page 02096. EDP Sciences, 2018.
- [7] Wei Wang, Yong Cao, and Tsubasa Okaze. Comparison of hexahedral, tetrahedral and polyhedral cells for reproducing the wind field around an isolated building by les. *Building and Environment*, 195:107717, 2021.
- [8] Loris Ibarrart, Stéphane Blanco, Cyril Caliot, Jérémie Dauchet, Simon Eibner, Mouna El-Hafi, Olivier Farges, Vincent Forest, Richard Fournier, Jacques Gautrais, Raj Konduru, Léa Penazzi, Jean-Marc Tregan, Thomas Vourc’h, and Daniel Yaacoub. Advection, diffusion and linear transport in a single path-sampling Monte-Carlo algorithm : getting insensitive to geometrical refinement. working paper or preprint, October 2022.
- [9] Abdolhossein Haji-Sheikh and Ephraim M Sparrow. The floating random walk and its application to monte carlo solutions of heat equations. *SIAM Journal on Applied Mathematics*, 14(2):370–389, 1966.
- [10] M. F. (Michael F.) Modest. *Radiative heat transfer Michael F. Modest*. Academic Press, New York, 3rd ed. edition, 2013.
- [11] Isaac Amidror. Scattered data interpolation methods for electronic imaging systems: a survey. *Journal of electronic imaging*, 11(2):157–176, 2002.