IOP Conference Series: Materials Science and Engineering

PAPER • **OPEN ACCESS**

Performance Simulation for Gas Flow through a Porous Media in Packed bed Columns using CFD

To cite this article: Asaad Salim Bded 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **579** 012001

IOP Conf. Series: Materials Science and Engineering **579** (2019) 012001 doi:10.1088/1757-899X/579/1/012001

Performance Simulation for Gas Flow through a Porous Media in Packed bed Columns using CFD

Asaad Salim Bded

Petroleum Engineering Department, Kerbala University, Karbala, Iraq

Corresponding email: asaadsalim78@gmail.com

Abstract. Two dimensions of Computational Fluid Dynamics simulation for gas flow in the column with filling material as bed at low particle Reynolds numbers and high particle Reynolds numbers have been presented. The simulation test has been done with the column length 0.9 m. Porous media model was used to simulate the gas flow inside porous granular media. Model of standard k-e for the flow turbulent range was used; the difference on the mesh size used to study effect it on the performance of gas flow through porous media. The result of flow rate of the gas was an important variable influencing bed and there are good results between the CFD simulation of pressure drop and the predicted values from Ergun model, and Computational Fluid Dynamics gave us simulation with high details for porous flow in packed bed equipment.

1. Introduction

The flow through column filled with granular particles is of frequent occurrence in petroleum industry and chemical process, therefore an expression are needed to predict the pressure drop within material of beds due to the resistance caused by the filling material inside of column [1]. The flow at low velocity and high velocity through the spaces between the particles have been retained by porous medium to allow us obtaining the influence of some of parameters on the performance of flow inside bed columns. Therefore this study carried out a computational fluid dynamics (CFD) based on the standard k-e model investigation of flow through porous media ranging from laminar to turbulent flow. CFD is relatively lower in cost than the physical testing, ability to simulate real conditions, many flows and transfer processes cannot be easily tested [2]. By using a Computational Fluid Dynamics (CFD) the studies were extended to treat various geometries with different boundary conditions, more applications for flow studies in porous media have produced extensive basis information in published literature, of which R.S.Maier & R.S. Bernard.[3]used the lattice Boltzmann method to the velocity distribution under the spatial resolution in packing without gives the details of the packing to simulate viscous fluid flow through a column of glass beads.

J. Gunnar I. Hellstr, & P. Jonsson [4] analysis of a quadratic array of cylinders shown that a number of different Reynolds number values can be defined for porous media, CFD simulations for flow through a quadratic array of cylinders at three solid fractions with a turbulent, found that turbulence flow needs be considered when Rep is above 300, since the laminar setup of equations failed to predict the experimental results over this value F.Augier&J.Y.Delenne[5] verified the transport and transfer properties inside packed beds of particles by CFD simulations at low to moderate Reynolds numbers. They found a problem of contacting points between particles, which is inherent to finite-volume numerical methods and solved by applying a shrinking of 2% along of material of the bed when transfer properties been estimated of increasing in arranging packing complexity. Kamyar M. & Ali A.&

Eckehard S.[6] used the porous media model as a method for simulation to ability the prediction of CFD using as a tools design for such packed columns to flow in the reaction media, The results of computational fluid dynamics simulation was excepted with results of the experimental in all cases studies.
The present research uses the software ANSYS-Fluent to develop the porous media in column with filling materials as bed to simulate of performance the flow of gas from the laminar to turbulent gas flow through the filling materials (porous media) within the range of Rep (7.87 – 1115.87) respectively, throughout change in axial temperature distributions along materials column.

2. Material Properties

The details of system are a gas phase without change in its density and uniform of Glass balls size. The properties, both of the (air) and material (Glass balls), are tabulated in Tables (1) to (3). Based on an experimental setup [5] and the thermal properties of glass balls are taken from Peter, S.T. [8], and the gas properties based on the ANSYS database.

Table 1. Properties of gas phase at the temp. 293 K

Parameter / unit	value
(air) density (kg/ m^3)	1.225
(air) viscosity (kg/ m. s)	1.7894
Heat capacity (j/kg. k)	1006.43
Coefficient of thermal conductivity (w/m. k)	0.0242

Table 2. the Physical properties of column (Steel) from ANSYS database

Parameter	value
density (kg/ m^3)	8030
specific heat (j/ kg. k)	502.48
heat conductivity coefficient (w/ m. k)	16.27

Table 3. Material properties of simulated sphere [7]

Parameter	value
Particle diameter (m)	0.005
Density (kg/ m^3)	2590
Heat capacity (j/ k_g.k)	780
Heat conductivity coefficient (W/m.k)	0.9656

3. Models of Porous Media

Forchheimer suggested the flow with in porous medium and described his suggestion when the flow and the pressure gradient have a linear relationship (linear Darcy's law) as shown in below form [9]:

$$\frac{\Delta p}{L} = \frac{\mu u}{k} + \rho\, k_F u^2 = \alpha u + \beta u^2 \qquad (1)$$

Where k_F is the inertial resistance (Forchheimer coefficient) at low Reynolds number, the equation (1) considered (Darcy law) that the viscous forces be controlled in porous media and Δp is proportional to the Darcy velocity (inertial effects can be neglected), while as the Reynolds number increases, the inertial forces are considered and equation (1) gives non-linear relationship between Δp and Darcy velocity, and at high Reynolds number, the inertial forces became strong, Δp is proportional to the squared velocity [10]

In the ANSYS-FLUENT the superficial phase velocities can be calculated depending on volumetric flow rate in a porous media of the regime [11, 16]. Porous Media Model for homogeneous medium consists of two portions when the momentum term is added; inertial loss and viscous resistances portions. Will be mixed with other factors of Darcy's equation to predict the pressure drop through the porous zone inside the gas cell zone (fluid volume) in the axial direction, this is called a Porous Media Model by cell zone conditions [11].

The concept of permeability was developed by Kozeny and Carman the following equation (2). It depends on the porosity and glass sphere diameter and medium's permeability also depends on physical and geometric model characteristics.

$$K = \frac{\rho g}{5\mu}\frac{\varepsilon^3}{(1-\varepsilon)^2}\left(\frac{d}{6}\right)^2 \qquad (2)$$

Where ε is the porosity of medium's, It is calculated from equation (3) Furnas [12] is proposed an equation for the porosity in bed column as function of particle and bed diameter, as in the following equation:

$$\varepsilon = 0.375 + 0.34\,\frac{d}{D} \qquad (3)$$

The coefficient β is the inertial resistance in the porous media along the flow direction per unit length at high velocities [10]. Values of equation (1) are obtained from physical regime by solution of Navier-Stokes equation relating to velocity and pressure subject to corresponding boundary condition in CFD simulation. In this paper, air was passed through the bed to simulate the flow and to estimate the pressure drop depending on properties of the fluid, and porosity of medium. Ergun equation [13], is defined as follows:

$$\frac{\Delta p}{L} = 150\,\mu\,\frac{(1-\varepsilon)^2}{\varepsilon^3\ d^2}\,u + 1.75\rho\,\frac{(1-\varepsilon)}{\varepsilon^3\ d\emptyset}u^2 \qquad (4)$$

The pressure drop predicted from CFD simulation of bed columns has been compared with values from equation (4).

4. Modelling Fluid Flow

The basis for all CFD fluid flow modeling is the Navier-Stokes equations. The term Navier-Stokes equations are used to describe three equations; the momentum equation, the continuity equation, and the energy equation.

5. Navier- Stokes Equation

The Navier Stokes equation is the equation of motion for a Newtonian fluid with constant viscosity and density. The equation is greatly simplified when applied to 2D flow with the assumption that velocity is only in the axial direction [11 14 17].

6. Standard k-e Turbulence Model

At high Reynolds numbers, the viscous dissipation of kinetic energy (ε) will rise because of high inertial forces, multiplied by the fluctuating vorticity and from Navier-Stokes equation it can be derived the viscous dissipation at the exact transport fluctuating vorticity equation [10 14 17].

7. Preparation of media and design of column

The first step in CFD simulation of bed columns is Pre-processing, which has been done by ANSYS Workbench 12.0 tool to form the geometric model with (hexahedral grids) of numerical density: dividing 20 interval counts and 40000 quadrilateral cells (40k), 1600 2D wall faces, 40851 nodes, (40k) computational cells as shown in Fig.(2).The geometric model was designed as two dimension (2D geometry) on the (x, y) coordinates, with 0.041m in a diameter of column, 0.9 m length with four surface boundaries: inlet (for inflow, t, u), outlet (for outflow, t, P,.etc), wall and porous medium. The bed materials in this model of glass balls are packed into the column randomly as porous media. The model assumption is shown in Fig. (1).

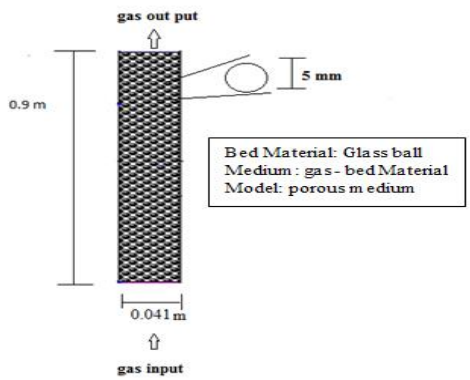

Figure 1. 2D schematic of the Bed column

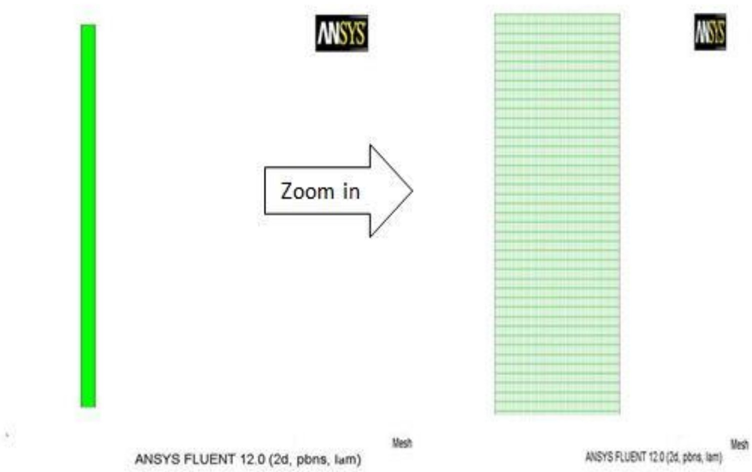

Figure 2. 2D hexahedral meshing bed column.

8. Computational Fluid Dynamics (CFD) model using (ANSYS- FLUENT)

The method of (CFD) used to simulate gas flow in the assumed bed column by using the ANSYS-FLUENT, the pre-processor, solver and post-processor are the basic parts of the ANSYS-FLUENT software to analyse the process. The pre-processor step is usually carried out through a graphical interface and describe in section 3. The velocity of gas, and the data of the porous cell condition, density of the gas and all coefficient values are entered to the solver software (user input values), to solve the numerical equations based on the entered values and the results are obtained to provide the post-processor step software [15]. In this work the single phase flow is initially in static condition inside the bed column and the gas passes through bed from bottom of the column by velocity inlet at the central of porous cell zone that is specified for the porous medium conditions, there are options related to inertial loss and viscous resistances and axial direction vectors to be solved , void fraction has been calculated from Equation (3), and the permeability has been calculated from Equation (2) for each of the space coordinates. Flow Analysis by FLUENT through bed column using 2D model is solved in steady state laminar and turbulent flow simulation. all models that described previously are available in FLUENT software [15]. Select physical velocity under porous formulation, will take the volume blockage into consideration and provides a more accurate representation of flow within bed column. The boundary condition for laminar model and turbulent model flow will range from 0.023 to 3.23 as inlet velocity and is used for inlet temperature equals 293^{o}k (isothermal boundary condition), column wall temperature be constant and equals to 373^{o}k (heating wall), boundary condition for porous zone are estimated in section 2. After the simulation is iterated, the final steps are post-processing and displaying the output in contours or in vectors, the data that is obtained by the solver can be visualized and displayed such as static pressure, velocity magnitude, static temperature as shown in next section. The procedure analysis by CFD using model porous flow is shown in the Flowchart below:

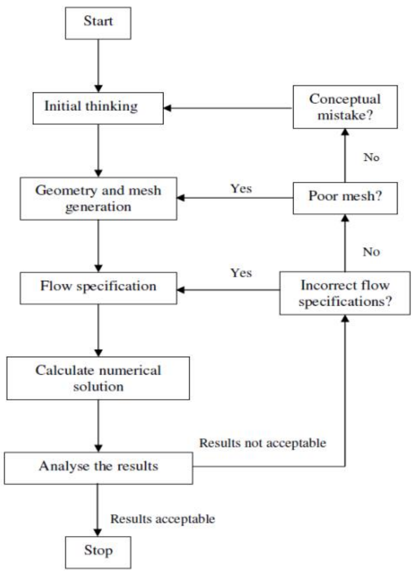

Figure 3. Flowchart of (CFD) analysis process [15]

9. Results and Discussion

9.1. Effect of porous flow properties on pressure drop

The different velocity flow (porous flow) ranges have effected on pressure drop through the packed column, the pressure difference between the bottom and top of the bed is (ΔP), the profile of pressure has been simulated over the Reynolds numbers range of (7.8 – 1115.876) as shown in Fig (4) CFD contours simulation of static pressure, and as function to superficial gas velocity and length of bed column (0.9) m, the pressure drop CFD results are compared with Ergun correlation as shown in Table (4). The Fig (5) shows the results from both models and gives reasonable data with error (0.6% – 0.63%) between data results from (CFD) analysis and values from Ergun equation, this error comes from the equation (3) for the porosity of medium. for bed height with (0.9 m), maximum pressure drop is (33381.57) (Pascal) under Reynolds numbers (turbulent flow), the reason is the effect of changes in porous flow regime to turbulent flow (strong inertial forces) when the bed length increases and (ΔP) along column increases

Table.4. CFD results compared with Ergun equation

Re_p	(ΔP) (Pascal) CFD model (0.9) m length column	(ΔP) (Pascal) Ergun model (0.9) m length column
7.872750643	2.7494	10.52869951
188.2614284	972.0385	946.9017
290.94948	2299.466	2261.608
328.60177	2927.186	2884.842
376.522857	3835.525	3787.607
492.90265	6551.711	6490.893
554.51548	8282.316	8215.037
835.196155	18729.81	18636.28
1115.8768	33381.57	33267.08

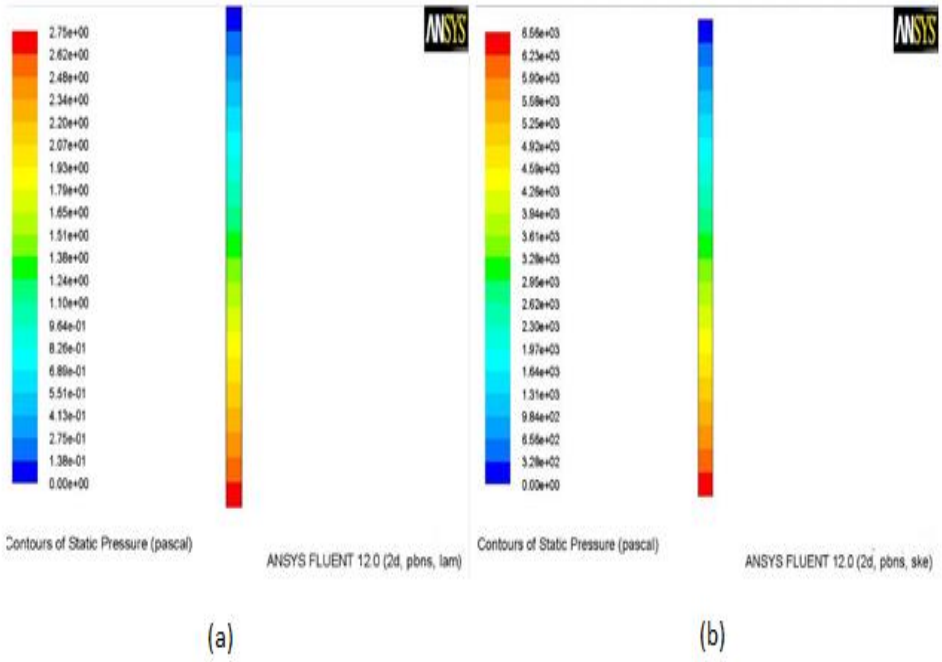

Figure 4. CFD contours simulation of static pressure (Pascal) (a) laminar flow (b) Turbulent flow.

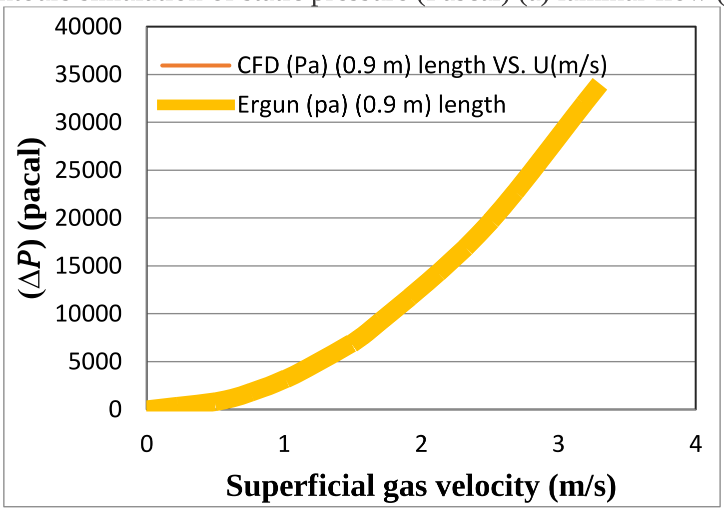

Figure 5. (ΔP) (Pascal), CFD results compared with Ergun equation variations with superficial gas velocity (m/s) along length of column 0.9 m.

9.2. Effect of Mesh size:

the concept of mesh density can be examined at different mesh sizes by temperature distribution in the model as shown in Fig.(6).The comparison between steady-state temperature distributions is the plot of temperature (K) vs. radial position (m) for a velocity of 0.023 (m/s) for four mesh sizes (12k, 30k, 40k, 64k) at high that equals to 0.594 m from bottom of packed bed model, the profiles of change in temperature inside column model comes from changes in mesh sizes on the flow gas for 0.9 m model with the appearance of very little difference in temperature between the mesh (40k and 30k) in the middle of the column and a great convergence in temperature at the wall of the column temperature. The mesh 64k has a clear difference in temperature, indicated to mesh size and refinement created a major little difference in temperature. So the variation in the mesh density should be smooth to reduce numerical diffusion.

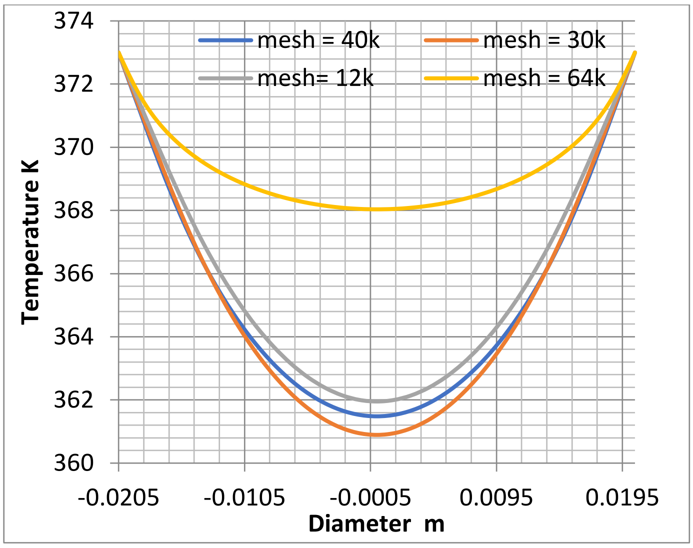

Figure 6. Temperature distributions at four different mesh sizes from bed model.

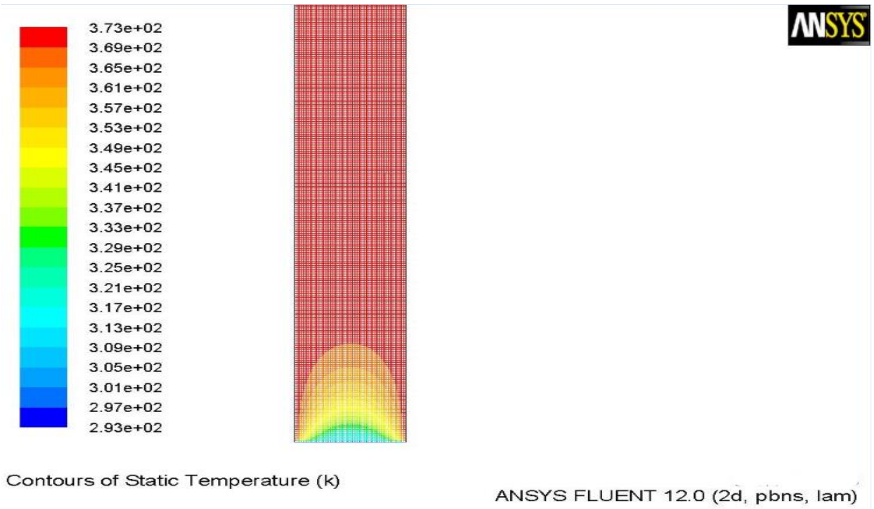

Figure 7. Static temperature in (K) for axial position (m) at a velocity = 0.023 (m/ s) with 40K mesh size.

9.3. Effect of gas flow rate:

The gas flow rates have an important variable in bed column system and have direct influence on the process model. Fig.8 shows simulation Ansys-Fluent for Contour of static temperature values along bed model and affects it by gas flow inside system model at laminar and turbulent flow. from Fig.8 the total difference in temperature for laminar flow (Rep=7.8) is (79K- 80K) to reach steady – state condition, but for turbulent flow Rep = 1115.87 the total difference in temperature is (59K) along bed column model, and this difference is ranging by an increase in the gas velocity with (x/L) for model column, The increase in temperature with increasing Reynolds number has the effect on amount of heat into the bed at wall temperature is constant.(CFD) simulations give suitable prediction in amount of heat inside geometry model column at change in gas flow regime. Table.(5) illustrates CFD results temperature (K) profiles inside bed column model at Reynolds number ranged from laminar to turbulent flow for different positions. Fig.9 illustrates porous medium model distribution of radial temperature by CFD simulation at different positions with high Reynolds number that equals to 1115.87 variations with radial

IOP Conf. Series: Materials Science and Engineering **579** (2019) 012001 doi:10.1088/1757-899X/579/1/012001

(r) m. so the prediction from CFD simulation gives high details for porous flow in packed bed equipment.

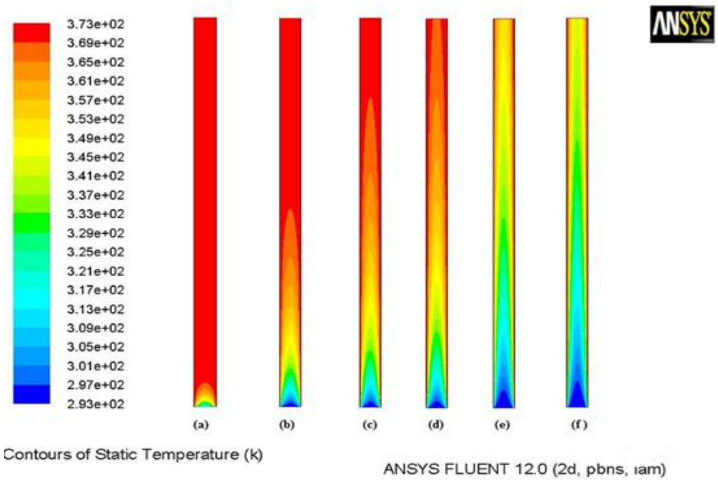

Fig.8. Static temperature in (K) along geometry model (a) Re_p=7.872, (b, c, d, e, f) Re_p=turbulent flow

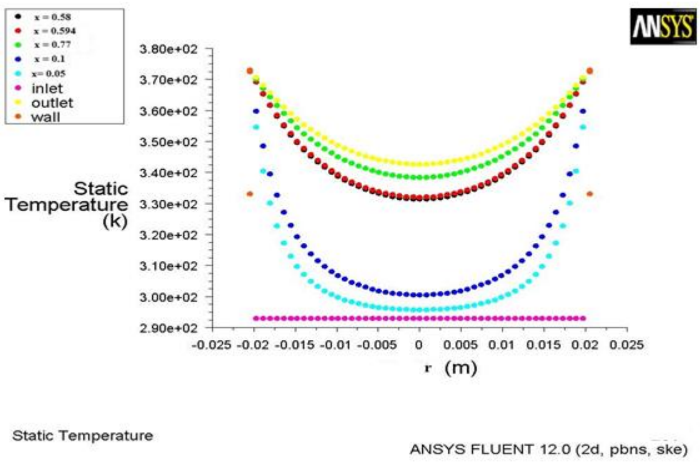

Figure 9. Porous media model illustration radial temperature distributions by CFD simulation at different positions.

Table.5.CFD results temperature (K) profiles inside bed column model at laminar and turbulent flow for different position

Reynolds number	**Different positions along bed material column (m)**					
	0.05	**0.1**	**0.4**	**0.6**	**0.8**	**0.9, outlet**
7.872	370.00	372.82	372.89	373.00	373.00	373.00
188.26	335.55	352.34	368.61	371.04	372	372.6
290.94	326.98	341.20	363.28	371.15	371.06	371.21
328.60	324.81	338.14	361.37	369.28	370.49	370.78
376.52	322.51	334.87	359.05	367.32	368.94	369.49

492.90	318.28	328.98	353.94	362.40	365.58	366.97
554.51	316.56	326.65	351.54	360.03	362.92	364.76
835.19	311.2	319.38	342.76	350.24	355.89	358.15
1115.8	308.02	314.96	336.48	344.55	348.64	352

10. Conclusion

The pressure drop simulation has been compared with the values calculated from Ergun equation, the relative error is (0.6% – 0.63%) for our model and It was found that there is a good agreement between CFD and Ergun model, and maximum (ΔP) for model bed with (0.9 m) height is (33381.57) (Pascal) under different in flow regimes and It illustrated the effect of changes in porous flow regime to turbulent flow, It becomes (strong inertial forces), and gives increases in the (ΔP) along column.

The profiles of change in temperature inside column model comes from changes in mesh size on the flow gas, there is a very little difference in temperature between the meshes (40k and 30k) in the middle of the column it is and very close in the wall.

influence on the process model. The total difference in temperature for laminar flow (Rep=7.8) is (79K- 80K) to reach steady – state condition, but for turbulent flow Rep = 1115.87 the total difference in temperature is (59K) along bed column model, and this difference is ranging by an increase in the gas velocity with (x/L) for model column. The prediction from Computational Fluid Dynamics gives us simulation with high details for porous flow in packed bed equipment, design its and performance flow of gas through porous media in packed bed column.

References

[1] Burke S P and Plummer W P 1928 Gas flow through packed column *Ind. Eng. Chem.* **20** 1196
[2] Bark H 2009 *CFD Advantages and Practical Applications* Glumac Inc.
[3] Maier R S, Kroll D M, Kutsovsky Y E, Davis H T and Bernard R S 1998 *Phy. Fluid*, **10** 60
[4] Gunnar J, Hellstr I, Jonsson P and Staffan Lundstr 2010 *Journal of Porous Media* **13** 1073
[5] Kamyar M, Alkhalaf A and Specht E 2018 *Computational Particle Mechanics* **6** 157
[6] Augier F, Idoux F and Delenne J Y 2010 *Chemical Eng. Science* **65** 1055
[7] Wen D and Ding Y 2006 *Chem. Eng. Sci*. **61** 3532
[8] Peters S T 1998 *Handbook of composites* Springer-Verlag (Houston: Springer-Verlag) ed 2 p 1118
[9] G K Batchelor 1967 *An Introduction to Fluid Dynamics* (Cambridge: University Press Cambridge) ed 1 p 615
[10] Hlushkou D and Tallarek U 2006 *Journal of Chromatography* **1126** 70
[11] ANSYS Fluent Documentation ANSYS Inc. 2010 ANSYS Fluent 12.0/12.1 Documentation
[12] Furnas C C 1931 Ind. *Eng. Chem* **23** 1052
[13] Stewart B R B 1960 *Transport Phenomena* (New York: John Wiley and Sons Inc.) ed 1 p 780
[14] Launder B E and Spalding D 1972 *Lectures in Mathematical Models of Turbulence* (London: Academic Press) ed 1 p 24
[15] ANSYS Fluent Documentation ANSYS Inc. 2009 Release 12.0
[16] Gunnar J, Hellstrom I, Jonas P and Jonsson P 2010 *Journal of Porous Media* **13** 1073
[17] Festuccia A and Bella G 2018 Journal *of Porous Media* **21** 5