International Journal of Energy Applications and Technologies
Available online at
http://dergipark.gov.tr/ijeat/issue/28599
Research Article
Vol. 4
Issue 1
pp. 12-17
Year
2017

METHANOL COMBUSTION SIMULATION VIA CFD

Ali Hussein Abdulkarim[1]*

[1]Kirkuk University, Mechanical Engineering Department, Kirkuk, Iraq

Abstract

Methanol combustion can take place in various mediums ranging from internal combustion engines to burners and such. Consequently combustion efficiency and the dimensional system characteristics vary from system to system. Recent researches are going on to identify these aforementioned characteristics. Present paper is a part of such effort. A combustion domain representing the geometrical parameters of a burner was modelled and governing equations for combustion process were selected in a commercial CFD solver. Results constitute base for future work focusing on a similar burner performance. Static pressure distribution, mesh structure, temperature distribution, turbulence intensity, density distribution and velocity vectors are presented in both 2D planes and 3D domain. Results indicate the importance of combustion volume entrance design. There are dead regions adjacent to the combustion volume entrance. It is proposed that a new entrance region should be designed.

Key words: *CFD, Combustion, Fluid Dynamics, Methanol, Simulation*

Note: This paper has been presented at the International Conference on Advanced Technology & Sciences (ICAT'16) held in Konya (Turkey).

Paper submitted: February 23, 2017

Paper accepted: March 24, 2017

Corresponding author, dr.ali@uokirkuk.edu.iq – alihuseyin81@yahoo.com

1. Introduction

Porous medium as an alternative to conventional burners attracts industrial and scientific communities. Their compact and complex inner structure leads to efficient burning of the fuels while its general geometry from outside can be adapted literally to any volume. Figure 1 is provided in order to give an idea for a porous medium burner.

Methanol combustion can take place in various mediums ranging from internal combustion engines to burners and such. Consequently combustion efficiency and the dimensional system characteristics vary from system to system. Recent researches are going on to identify these aforementioned characteristics. There are remarkable studies in the literature about modelling porous combustion mediums and fuels such as methanol [1-3]. These studies focus on geometrical modelling of the medium, proper models for the flow and combustion and comparison of the obtained results with literature and experimental results.

As a preliminary evaluation tool, CFD was utilized in case of a cylindrical body representing a porous metal matrix burner. Methanol was simulated with air as an ideal gas. Boundary conditions were selected according to the real application operational parameters of the so called porous metal matrix burner. By this way, static pressure distribution, velocity profiles, density of the fluid, enthalpy, total pressure, turbulent kinetic energy and dynamic viscosity changes were presented versus radius of the examined body. Fluent was employed for the modelling and calculation software.

Fig. 1. Combustion in a porous medium burner.

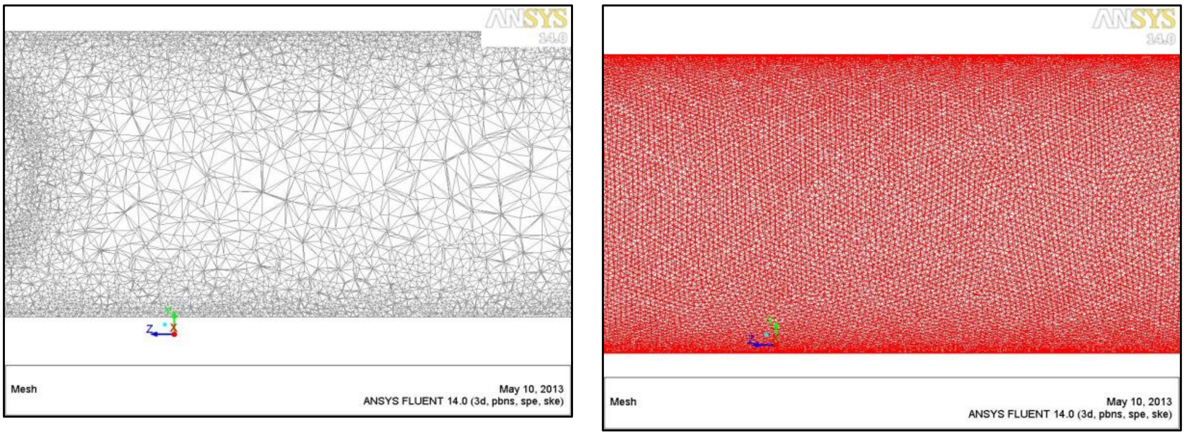
Computational analyses are attracting more attention recently. Al-Turaihi and Oleiwi [4] used computational numerical heat transfer analysis for investigating a channel flow. Their study includes liquid and gas phases. Also ribs in the channel were considered in order to detect their effects in respect of heat transfer improvement. Their results were validated with correlations from the literature. Although the intensity of the mesh seems to be relatively low, flow and heat transfer information can be caught.

Another interesting computational heat transfer study is belonged to Kent [5]. He investigated natural laminar convection in triangular channels. He changed boundary conditions and the Rayleigh number. Fluent was utilized with SIMPLE algorithm. A fine mesh was structured and isotherms was used for the results.

2. Analysis

Ansys Fluent was used for the computational analysis. Cartesian coordinates were used for the computational domain but curvilinear adapted mesh was utilized. This doesn't lead to any complexities in the major fraction of the computational domain and the CFD code can handle this situation. Air was selected as the fluid and thermo physical properties were selected accordingly.

A fine mesh setting was applied to the cylindrical porous medium model geometry and the general view of the mesh structure is provided in Figure 2. k-ε turbulence model was utilized as a general and justified turbulence model for the examined geometry. Since internal boundaries are avoided for the present preliminary investigation, the utilization of the k-ε turbulence model was appropriate. The governing equations for this model are not provided here for the convenience of the space however as a well-established and validated model, governing equations can be found easily in the literature.



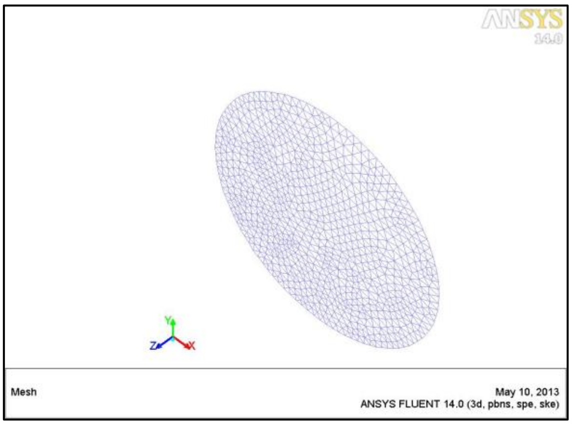


Fig. 2. Mesh structure view of the model: a- General Mesh View b-Outlet Mesh View c- Inlet Mesh View.

3. Results

Since the symmetry exists in the analysis, mainly axial change is presented and evaluated here. The graphics are presented on a 2D plane and in a 3D volume. The quantitative values can be extracted from the color scale which also includes and indicates numbers.

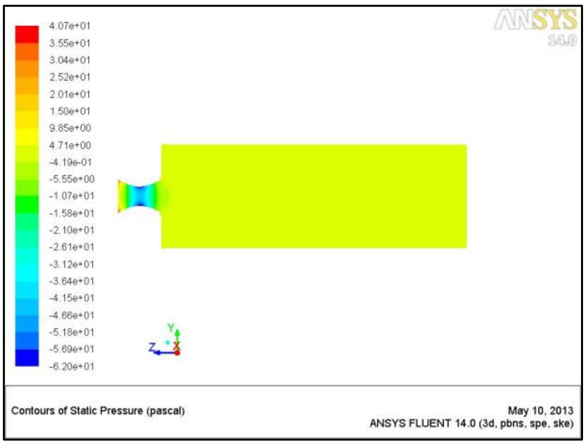


Fig. 3. Static pressure contours changing on a 2D centre plane according to axial distance.

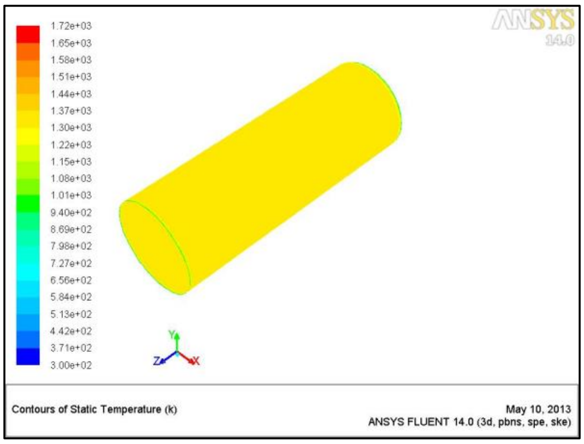


Fig. 4. Static temperature contours changing on a 3D domain according to axial distance.

International Journal of Energy Applications and Technologies, Year 2017, Vol. 4, No. 1, pp. 12-17

Due to the energy dissipation by the friction, static and total pressure values decrease in the flow direction. Pressure gradient in the axial direction occurs as the maximum pressure value exists at the entrance or in other words inlet of the medium. Quantitative results are presented below for static pressure in Figure 3, Static Temperature in Figure 4, Contours of static temperature in Figure 5, Velocity vectors in Figure 6, Contours of velocity magnitude in Figure 7, turbulent kinetic energy distribution in Figure 8 and Density distribution in Figure 9 respectively.

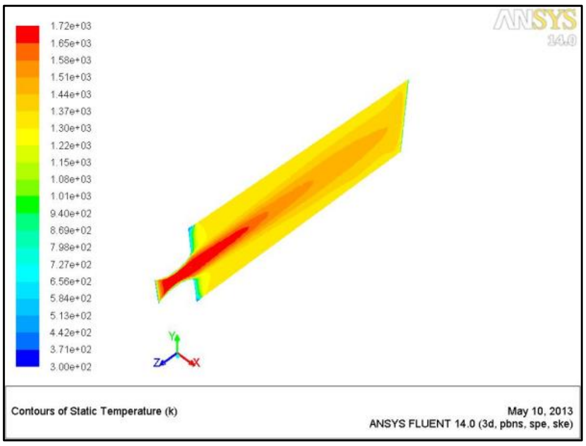


Fig. 5. Static temperature contours changing on a 2D centre plane according to axial distance.

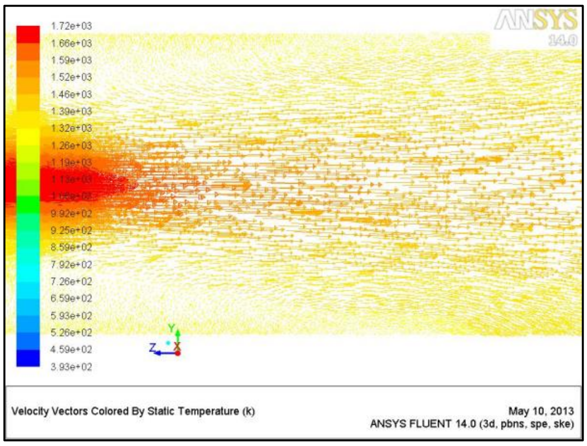


Fig. 6. Velocity vectors containing temperature information changing on a 2D centre plane at the near proximity of the inlet.

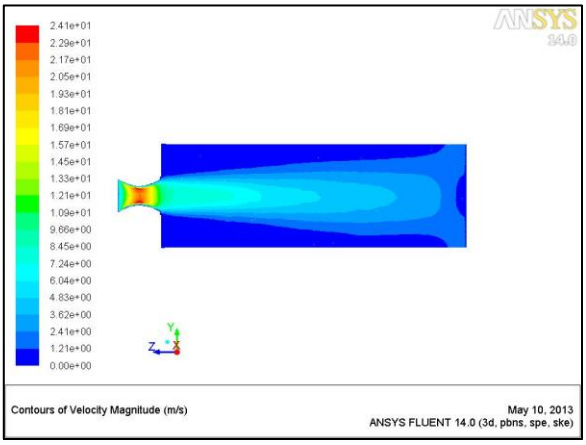


Fig. 7. Velocity magnitude contours changing on a 2D centre plane according to axial distance.

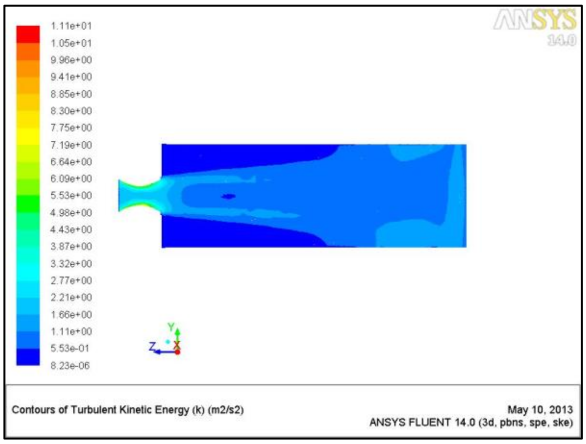


Fig. 8. Turbulent kinetic energy contours changing on a 2D centre plane according to axial distance.

As the fluid enters to the computational domain, a compression effect takes place and this leads to an increase at the initial part of the inlet. As the flow continues to the inlet neck, velocity increases and the static pressure is converted to kinetic energy and hence the value of the static pressure decreases here. When further continuing on the axial direction, static pressure recovers and has its final value.

3D domain in respect of the temperature quantitative results can be confusing since the wall has a dominant value comparing to the axis of the computational domain. The dominant temperature value at the boundaries is about 1300 K. Due to the dead points and relatively poor conduction at the corners, temperature value is about 940 K at the corners. Since the flame temperatures are much higher, these temperatures are expected at the boundaries. Considering the radiation, porous burner can actively transfer the heat via radiation. However the present work doesn't contain boundaries for porous extensions, so the real application will lead lower temperatures at the outer boundaries.

When a 2D plane inserted in the 3D domain in order to inspect temperature distribution contours for the computational domain, one can see that the fluid temperature can reach approximately 1720 K at the axis of the medium. Since the present work doesn't contain any solid boundaries at the center of the volume, this temperature can be regarded as the flame temperature. With the effect of the axial conduction and diffusion this maximum becomes lower as the field of interest advances more through dead points. We can see the effects of flow separation at close proximity of the adjacent edges perpendicular to the inlet. The lowest temperatures can be observed here because of the flow separation. However real world application will differ in this respect because of the axial conduction, i.e. reverse heat transfer.

One can see a nozzle effect because of the inlet structure of the burner when Figure 6 is investigated. The shear layers and recirculation regions can be identified in this figure. By the nozzle effect, the relatively higher velocity in the core makes shear layers stable.

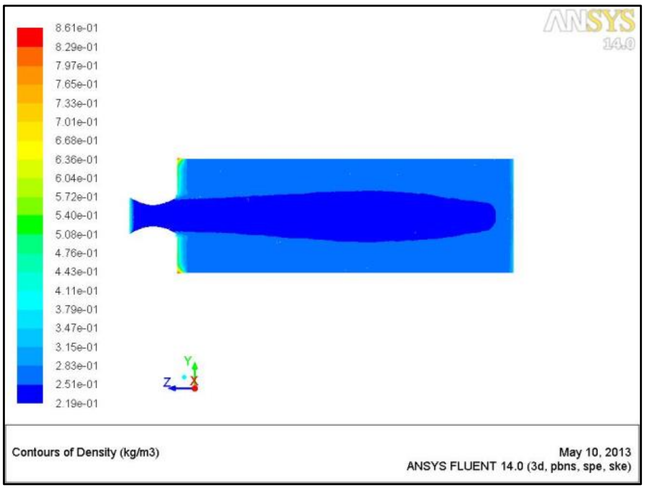


Fig. 9. Density contours changing on a 2D centre plane according to axial distance.

Due to the conversion of the static pressure to the dynamic pressure, in other words to velocity, at the minimum diameter in the inlet, maximum velocity can be observed here in Figure 7. The maximum temperature seems to be around 24.1 m/s. At the first quarter of the volume the velocity decreases to 6.4 m/s and then gets a more moderate value around 2-4 m/s. We can see a dead point at the farthermost point in axial direction.

Turbulence kinetic energy gives information of local fluctuations. A laminar and un-mixing flow would lead to a minimum turbulence kinetic energy pattern. Considering this information, Figure 8 indicates the locations of the shear layers. A relatively low turbulence kinetic energy, namely 2.21 m^2/s^2 can be observed, so we can think that mixing process is not strong. However one should consider that real application won't contain such a structure since there will be secondary boundaries due to the porous structure. Symmetry can also be seen for turbulence kinetic energy. At the inlet, a flow structure similar to the pipe flow can be observed due to the developing flow in the presence of close walls.

Density increases at the close proximity of the walls of the cylindrical body due to the compression in the presence of a solid obstacle. Considering the gas fluid, compression yields change in the density of the fluid. Also maximum enthalpy occurs near the outlet of the cylindrical body.

4. Conclusion

The flow patterns and general flow conditions yielded a favorable situation in respect of a medium designed as a burner. Since the real application combustion volume will have several inner boundaries due to the porous structure, flow separations and irregularities will diminish and/or become smaller in scale.

Heat transfer will be superior with a porous inner structure by generally speaking. Since conduction, convection and radiation will exist, the burner can act a preferable device. Convection is expected to be the least effective mechanism in the porous volume comparing to the radiation and conduction. General geometry of the porous medium model is found to be favorable in respect of its operational flow conditions. Further work can be undertaken in order to achieve a more realistic geometry and numerical combustion can be included into the analysis.

Acknowledgement

The author of this work would like to acknowledge the help and support of Selcuk University and Kirkuk University which provided their financial and infrastructural supports. Also the author would like to thank MSc Eyüb CANLI and Dr. Ali ATEŞ for their aid and guidance.

References

[1] Masoud, Z., Mohammadi, A., 2012, "Numerical Simulation of Combustion in Porous Media", *http//dx.doi.org/10.5772/50386*

[2] Mujeebu M.A., Abdullah, M. Z., Mohamad, A.A., Abu Bakar, M.Z., 2010, "Trends in modeling of porous media combustion", *Progress in Energy and Combustion Science*, 36, 627-650

[3] Altinişik, K., Teberoğlu, Ö., Yalçın, Ş., Tekin, M., Altinişik, A., 2006, "Semi-Spherical Ceramic Foam Burners and Burning Simulation (Part II)", *10th International Research/Expert Conference, Barcelona-Lloret de Mar*, 993-996, Spain.

[4] Al-Turaihi, R.S., Oleiwi, S.H., 2016, "Heat Transfer Of Two Phases (Water – Air) In Horizontal Smooth And Ribbed Ducts", *International Journal of Energy Applications and Technologies*, 3 (2), 41 – 49.

[5] Kent, E.F., 2016, "Laminar Natural Convection In Triangular Enclosures", *International Journal of Energy Applications and Technologies*, 3 (2), 37 – 40.