

Design and Computational Fluid Dynamics Modeling for a Municipal Solid Waste Incineration Process

Halidini Sarakikya¹, Peter Mashingo¹, Fidelis Kilonzo²

¹Arusha Technical College, Arusha, Tanzania

²School of Engineering and Technology, Kenyatta University, Nairobi, Kenya

Email: halidinisarakikya@gmail.com, mashinggopp@hotmail.com, kilonzo.fidelis@ku.ac.ke

How to cite this paper: Sarakikya, H., Mashingo, P. and Kilonzo, F. (2021) Design and Computational Fluid Dynamics Modeling for a Municipal Solid Waste Incineration Process. *Open Journal of Fluid Dynamics*, **11**, 177-191.

<https://doi.org/10.4236/ojfd.2021.114011>

Received: January 4, 2021

Accepted: November 12, 2021

Published: November 15, 2021

Copyright © 2021 by author(s) and Scientific Research Publishing Inc.

This work is licensed under the Creative Commons Attribution International License (CC BY 4.0).

<http://creativecommons.org/licenses/by/4.0/>



Open Access

Abstract

Municipal Solid Waste (MSW) incineration is among the important opportunities to control the pollutions caused by improper dumping and flue gas emissions so as to meet the environmental guidelines. The objective of this study was to design and develop a model for the MSW incineration process. The paper explains the design and development of MSW incinerator involving details of the Computational Fluid Dynamics (CFD) procedures through model design to the post process. CFD model was developed to reveal these features of the incineration process in the MSW incinerator. The model predicts the temperature in combustion chambers as well as at stack outlet of the incinerator. The solid waste characterization was done, and the laboratory analysis shows that, the Higher Heating Value (HHV) of the MSW is 11.65 MJ/kg on dry basis. The elemental composition MSW consists of 54.8%, 5.27%, 34.61%, 2.37% and 0.3% for Carbon, Hydrogen, Oxygen, Nitrogen and Sulphur respectively which are the input parameters for the CFD. The model simulation was positively achieved during combustion process and temperature in combustion chambers ranged between 930 K and 1700 K.

Keywords

Combustion, Temperature, Computational Fluid Dynamics

1. Introduction

The increasing amount of Municipal Solid Waste (MSW) generation and its disposal without environmental effects is now a serious problem which at present has caused a great concern [1]. It is obvious that, incineration of MSW has the maximum overall degree of destruction and control for various types of

waste materials [2]. One of the advantages of incineration is that, it decreases the volume of solid wastes by 80% to 95%, and it is a suitable method of disposal when there is a shortage of dumps which brings the idea of dumping costs [1] [3]. Thermal energy recovery from flue gases to produce steam for electricity generation is also another advantage of incineration. However, there must be an appropriate control of incineration process in order to produce flue gases which cannot pollute the environment [4]. According to Bjorn *et al.*, [5], the prediction of flue gases and the operating temperature in the incinerator with respect to its operating parameters is important information for the optimization of the incineration process.

Incineration of solid waste consists of processes such as drying, pyrolysis, and oxidation, convective, conductive and radiative heat transfer, mass transfer and gas flow through randomly in feed of the heterogeneous materials [6]. These materials have different size, shape, physical and chemical structure, in which all of these properties change continuously during thermal flow processes in the incinerator combustion chamber. According to Parkinson *et al.* [7], Computational Fluid Dynamics (CFD) is among the new methods that can be used to design and run a simulation without actually the need to build a physical model [4] [8]. Computational work has become very important in the current research due to its lower cost and accuracy with very small errors [9]. Therefore, the paper explains the CFD simulation and modeling for the incineration process, including details of the computational fluid dynamics process.

2. Governing Equations

The modeling of the fluid flow is based on a set of coupled conservation equations of mass, momentum, energy and chemical species. These equations are capable of describing both laminar and turbulent flows. In cases of a chemical reacting flow, the system at each point can be completely described by specifying temperature, mass, momentum, energy and the concentration of each species. Analysis of thermal flow in the combustion chamber of a solid waste incinerator could provide crucial insight into the incinerator's performance. The modeling equations used to evaluate thermal flow are summarized below:

1) Mass Conservation (Continuity Equation)

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) = s_m \quad (1)$$

where ρ is the mass density in per unit volume, u is the fluid velocity, S_m is the sources in the system.

2) Momentum Conservation Equation

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \partial F_i + S_m \quad (2)$$

where, u is the velocity, ρ is density of fluid, p is the pressure, n species have indices $i = n$ and τ_{ij} is the viscous stress tensor, F_i is the sum of external forces and

S_m is the source term. The source Term S_m means the mass transfer from solids or liquid phase to the gas phase.

3) Chemical Species Equation

$$\frac{\partial}{\partial t}(\rho Y_i) + \frac{\partial}{\partial x_j}(\rho u_j Y_i) + \frac{\partial \Gamma_{ij}}{\partial x_j} = R_i + S_i \quad (3)$$

where, Y_i is the species mass fraction, n species have indices $i = n$ and Γ_{ij} is the molecular diffusive flux of the species i , R_i is the rate of production of species i due to chemical reaction, and S_i is the source term to explain the species created in reactive flows.

4) Energy Conservation Equation

$$\frac{\partial}{\partial t}(\rho E) + \nabla \times [\mathbf{u}(\rho E + p)] = \nabla \times (\lambda_{eff} \nabla T - \sum_{j=1}^N h_j \Gamma_j + \boldsymbol{\tau} \times \mathbf{u}) + S_h \quad (4)$$

where E is the total energy, p is the static pressure, λ_{eff} is the effective thermal conductivity and h is the total specific enthalpy, the first three terms of the right side of Equation (4) represent heat flux due to conduction, species diffusion and dissipation due to normal shear stress respectively.

3. Materials and Methods

3.1. Determining the Physical Composition of the Waste

Trucks offloading solid waste at Murieta landfill were randomly selected on each day of the analysis. The landfill is located about five kilometers from the city, in Arusha Tanzania. The selected truck was directed to empty its solid waste at the selected area of the study. Waste pickers were employed for analysis, assisted manually to sort the waste into different classifications including food waste (35%), mixed papers (15%), plastic film (10%), frozen fruits (9%), diapers (8%), local newspapers (6%), leather (2%), cardboard (3%), yard waste (5%), metals 0.15% and others 1.03%. Materials that were not classified into any of these fractions were considered to belong to the fraction of others and this included items like medical wastes and inert items. The weights of the different fractions were then obtained and recorded in a data sheet. This procedure was repeated for the organic waste other trucks selected from the different wards of the city for thirty consecutive days and thereafter for two consecutive days in a period of two months. Samples were collected for a period of four months on a weekly basis.

3.2. Proximate Analysis of the Municipal Solid Waste

In determining moisture content of the samples, about 10 g of the sample was brought in from the field and weighed in to a dish that had been dried in an oven and weighed. The uncovered dish was then dried in an oven at 105°C for about two hours. The dish was covered and was transferred to desiccators and weighed as soon as the dish was cooled. The heating and weighing procedure were repeated until successive weight did not differ by more than one milligram. Loss in weights was recorded. The sample procedure was repeated for two other

sub-samples and the average moisture content of the three sub-samples was taken to be the moisture content of the sample.

3.3. Ultimate Analysis of the Municipal Solid Waste

The ultimate analysis of the Municipal Solid Waste (MSW) was estimated using the heat and material balance. Heating value of the solid waste was determined by converting 1 g of the MSW sample into pellets. Pellets were placed in the sample pan of the bomb calorimeter (Gallen Kamp auto bomb) and the energy content of the sample was determined by following proper procedures. CFD requires input parameters of MSW in terms of carbon, hydrogen, nitrogen, oxygen and sulfur. The major components of the MSW are shown in **Table 1** below.

3.4. Model Development and Design

The incinerator was designed by using ANSYS software and the geometry was drawn in solid Edge [4]. As shown in **Figure 1**. The geometry was optimized by using ANSYS Computational Fluid Dynamics [10]. Eulerian multiphase model was used and the energy conservation equation was solved for both solid and gases phases. Heterogeneous and homogeneous reactions were defined in the model to express the detailed combustion kinetics in the incinerator. The design was optimized to suit the characteristics of the MSW through several iterations.

3.4.1. Model Meshing

In the CFD simulations meshing the model is the most important stage and the

Table 1. Analysis of the municipal solid waste (dry base).

| Proximate Analysis (% wt) | Moisture | Volatile Matter | Fixed Carbon | Ash | Higher calorific value (MJ/kg) |
|---------------------------|----------|-----------------|--------------|-------|--------------------------------|
| | 59.75 | 78.89 | 10.52 | 10.53 | 11.65 |
| Ultimate Analysis (% wt) | C | H | O | N | S |
| | 54.80 | 5.27 | 34.61 | 2.35 | 0.30 |

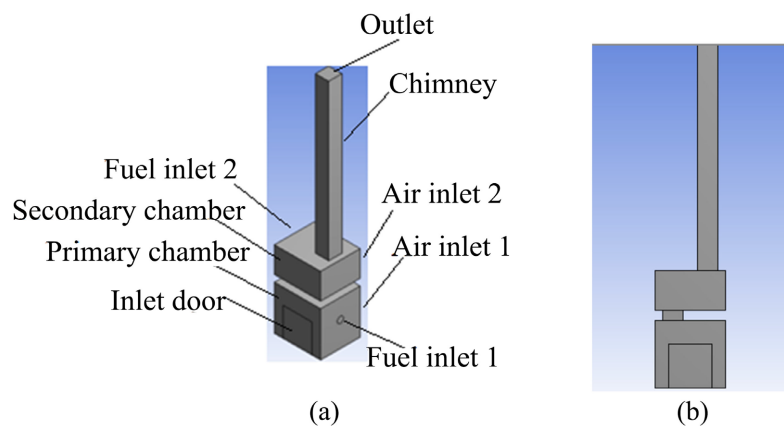


Figure 1. A 3D incinerator (a) Model schematic diagram with boundary conditions (b) Half model axisymmetric at xy-plane.

meshing technique applied will have direct impact on the quality of the mesh. Desired fluid flow equations are solved by the mesh which creates the elements, and the size of the elements or grids will have key impact on computational time. This has direct connection with the cost of simulation. Also, convergence speed and accuracy of the solution depend on the quality and quantity of the mesh elements or grids. During meshing, inflation is applied in areas near the walls and boundary layer mesh. **Figure 2(a)** shows the meshing done without inflation option and **Figure 2(b)** is the detail sizing of the important areas in the Figure.

At the start the meshing process was done by the coarse mesh as shown in **Table 2**. The quality of the mesh was done by applying statistics shown in **Table 3**

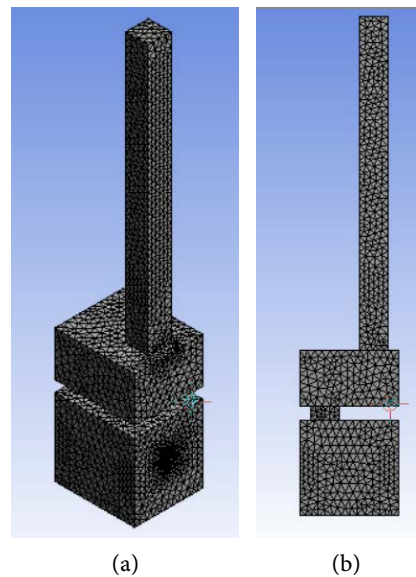


Figure 2. 3D Mesh without inflation option: (a) 3D Geometry mesh (a) 2D geometry mesh.

Table 2. Parameters for mesh sizing.

| Sizing Parameter | Initial Setting | Final Setting |
|------------------------|----------------------|-------------------------|
| Advanced size function | Curvature | Proximity and curvature |
| Relevance center | Coarse | Fine |
| Initial size seed | Active Assembly | Active assembly |
| Smoothing | Medium | High |
| Transition | Slow | Slow |
| Span angle center | Fine | Fine |
| Curvature normal angle | Default (18°) | Default (18.0°) |
| Min Size | Default (2.08910 mm) | Default (0.611870 mm) |
| Max face size | Default (208.910 mm) | Default (61.1870 mm) |
| Max size | Default (417.820 mm) | Default (122.370 mm) |
| Growth rate | 1.20 | 1.20 |

Table 3. Parameter settings for inflation.

| Item | Setting |
|----------------------------|--------------------|
| Use of automatic inflation | Program controlled |
| Inflation option | Total thickness |
| Inflation algorithm | Pre |
| View advanced option | Yes |
| Collision avoidance | Stair stepping |
| Growth rate type | Geometric |
| Use of post smoothing | Yes |
| Number of layers | 5 |
| Growth rate | 1.2 |
| Maximum thickness | 2.0 mm |
| Gap factory | 0.5 |
| Maximum length over base | 1 |
| Maximum angle | 140.0 |
| Fillet ratio | 1 |
| Smoothing iteration | 5 |

Skewness and aspect ratio are among the most important parameters for quality check on the mesh. In order to avoid the solution to easily diverge the maximum skewness should be below 0.98 [5] [7]. Skewness value range varies from zero to one, where the best value is zero and worse is one. To make sure that the skewness is below 0.98, a medium or fine mesh is applied. A fine relevance center as well as other final settings shown in **Table 3** will lower the maximum skewness below the most rigid limit. The aspect ratio is obtained when the longest edge length by the shortest edge is computed. Cells become square and have equal edge length of any shape when the aspect ratio is one. It is recommended that, the change in cell size be regular and should not exceed 20% change from one cell to another [6].

It is important to do mesh inflation as in **Figure 3** in the areas which are close to the walls by adjusting setting shown in **Table 3**. The stings used to avoid collision of the meshing process are stair stepping or layer compression. For the 20 percent growth rate, there are 5 layers and maximum thickness of 2.0 mm. Air inlets and fuel inlets are very small. During meshing sweep method, face sizing and edge sizing have been considered.

The mesh independent test was performed to ensure high quality simulation and high accuracy of the modeling. In order to simulate the temperature profile in combustion chambers of the incinerator, grid independent test study was done for different mesh settings. The mesh sizes for grid A, B, C and D were 0.2 mm, 0.1 mm, 0.03 mm and 0.05 mm respectively. The ortho skew for grid A, B, C, and D was 0.6724, 0.7203, 0.7342 and 0.7499 respectively. In order to avoid

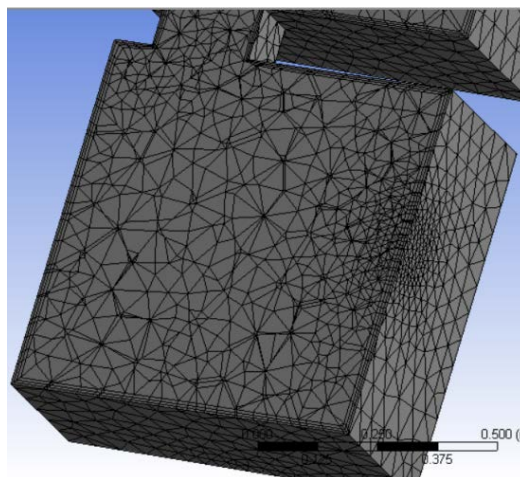


Figure 3. 3D mesh with inflation option.

solution to diverge, the maximum skewness should be below 0.98. Different grid sizes are shown in **Figures 4(a)-(d)** and temperature is compared for each specific grid size. The graph shows that grid C and D have similar characteristics as can be seen in **Figure 5**. The temperature in the combustion chamber ranges between 930 K - 1700 K, which means that the grid was independent.

3.4.2. Modeling Setup and Simulation

It is important to check the mesh quality and see if they are at the acceptable range. One of the quality checks is the orthogonal quality which ranges from 0 to 1, such that the lower values mean that the mesh is of low quality. In this study, the minimum orthogonal quality is 0.3177. Density based and pressure based are the settings selected depending on the type of solver. Solver should be selected for velocity formulation setting, that is absolute and relative, and also for the time between steady and transient. Pressure-based, absolute velocity and steady solver was used in this study. Also, the setting for gravitational acceleration was made at 9.81 m/s^2 in the Y direction. **Table 4** shows the settings used to make the plane surface of the incinerator.

Other important settings for model such as energy, radiation, turbulent viscosity and species were done since these parameters are directly related to the simulation circumstances. The energy equation was turned on because during MSW combustion temperature is expected to change. For realizable k-epsilon, turbulent viscous model was selected applying standard wall function. The discrete ordinate (DO) and non-premix combustion was selected for radiation and species respectively. **Table 5** shows the probability Density Function (PDF) and the boundary condition for the fuel and oxidant. ANSYS-FLUENT interpolates the PDF during solution of the non-premix combustion model.

In the simulation, methane (CH_4) is used as fuel to support the combustion process during MSW incineration. The oxidant consisted of 79 and 21 percent nitrogen and oxygen respectively and the temperature for fuel and oxidant was set at temperature of 315 K. The pressure was set at 101,325 Pa. The PDF table

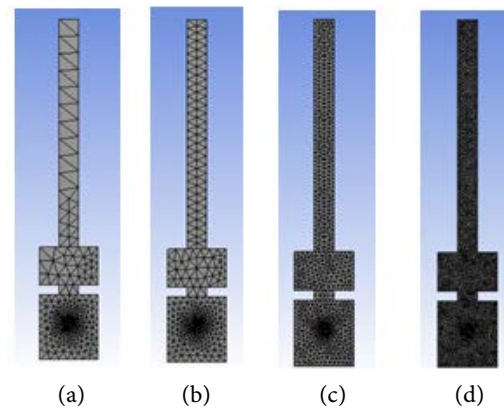


Figure 4. Meshing Independent test. (a) Mesh grid A (b) Mesh grid B (c) Mesh grid C (d) Mesh grid D.

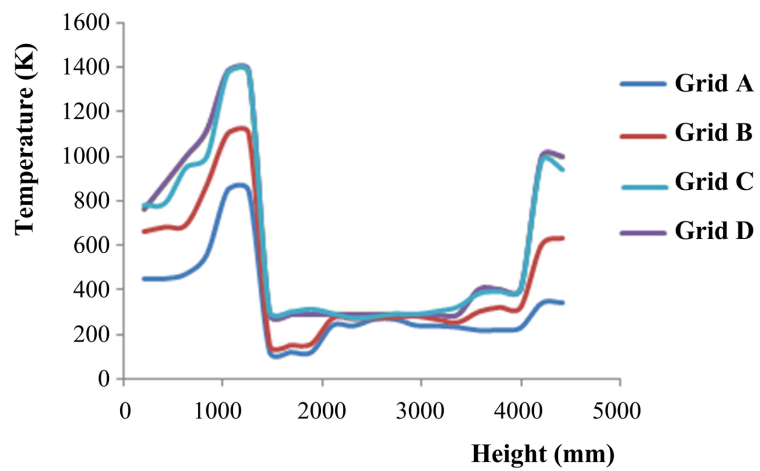


Figure 5. Temperature distribution for grid A, B, C and D.

Table 4. Selected points for 2D-XY-plane surface.

| | | |
|------------|------------|-----------|
| $x0$ (m) | $x1$ (m) | $x2$ (m) |
| -0.0024996 | -0.0024996 | 0.0024996 |
| $y0$ (m) | $y1$ (m) | $y2$ (m) |
| 0 | 0 | 4.043 |
| $z0$ (m) | $z1$ (m) | $z2$ (m) |
| 0 | 0 | 0 |

Table 5. PDF table creation for boundary condition.

| Species | Fuel | Oxidant |
|----------------|---------|---------|
| H ₂ | 0.0527 | 0 |
| N ₂ | 0.0235 | 0.78992 |
| O ₂ | 0.08985 | 0.21008 |
| C | 0.5489 | 0 |
| S | 0.003 | 0 |
| Cl | 0.0005 | 0 |

was calculated and results were displayed by choosing the type of Figure whether to be 2D or 3D and type of parameters to be displayed. For this case the default is 3D Figure having mean temperature, scaled variance and mean mixture fraction. Settings for the air inlets, fuel inlets, chimney and walls represent the boundary condition. In this study, the incinerator model consists of two air inlets, two fuel inlets, one chimney and twelve walls.

The momentum setting of air inlet and fuel inlets for velocity specification method applies the component method. Setting of velocity for air inlet was done in the X and Y direction and the air temperature was set at 315 K. The zero value was applied to species setting in the mean mixture fraction and the unity value was set for mean mixture fraction. The method used for the turbulence specification is intensity under the hydraulic diameter which was set to 5% and hydraulic diameter of 600mm according to model measurements. The setting of the air and fuel inlet velocity was set in meter per second because this is the most important parameters to consider during MSW combustion. “Stationary wall” method was used for the walls setting having no slip shear condition applying thermal heat flux of 0 w/m² and internal emissivity of 1.

3.4.3. Model Solution

Chemical reactions are involved in this model simulation as it is a non-premixed combustion process consisting of heat transfer, turbulent flows and species transport. The simulation applied the Navier-stokes equations which have been solved together with a realizable $k-\varepsilon$ turbulence model. The model simulation was solved by using commercial CFD software ANSYS Fluent 16.1. The Discrete Ordinate (DO) radiation model involving absorption coefficient of the weighted sum of gas (WSGGM) was applied. The original and new setting for the solution in this case is shown in **Table 6**. Presto was used for pressure and the couple method was used for the pressure-velocity coupling scheme with at least square cell-based gradient.

Table 6. Parameter settings for the solution method.

| Parameter | Original Setting | New Setting |
|----------------------------|------------------------|--------------------------------|
| Pressure-velocity Coupling | Simple | Couple |
| Gradient | Green-gauss cell based | Least squares-based cell based |
| Pressure | Standard | PRESTO |
| Momentum | First Order Upwind | Second Order Upwind |
| Turbulent kinetic Energy | First Order Upwind | Second Order Upwind |
| Turbulent dissipation rate | First Order Upwind | Second Order Upwind |
| Pollutant nox | First Order Upwind | Second Order Upwind |
| Energy | First Order Upwind | Second Order Upwind |
| Discrete Ordinates | First Order Upwind | Second Order Upwind |
| Mean mixture fraction | First Order Upwind | Second Order Upwind |
| Mixture fraction variance | First Order Upwind | Second Order Upwind |

The second order upwind is preferred for the momentum, turbulent kinetic energy, turbulent dissipation rate, pollutant no, energy, discrete ordinates, mean mixture fraction and mixture fraction variance during final running of the solution. **Table 6** presents the solution control for flow courant number, explicit relaxation factor (ERF) and under-relaxation factor. Original values of the parameter settings are needed to run the simulation at the start and thereafter to evaluate the results. Simulations involving chemical reaction including combustion, sometimes the original setting can cause divergence error and therefore require more concentration. When there is a divergence, the flow courant number and relaxation factor should be changed depending on the type of error. Enthalpy was the error in this study and after applying the new settings the flow courant number was reduced from 200 to 50. Also, it reduced the relaxation factor for momentum, pressure and density as presented in **Table 7**.

The residual of convergence criteria is used for monitoring simulation and calculation process. The simulation was set up with time steps of $1e^{-3}$ s which led to 3000 iterations to ensure that convergence is reached and at the beginning, the hybrid was selected. Calculations were performed until convergence is reached, that means until when the species residuals were less 10^{-6} to make sure no concentration fields are developing. In this study, the patch is used for Y velocity, which is used to force the flow downward for the exhaust gas recirculation to the chimney. The patch for Y velocity is -9.81 m/s². The second patch is for temperature which is 315 K.

To ensure error free, a check case is performed before any calculation process is started. The mesh quality, models, boundaries and cell zone, materials and the

Table 7. Control parameters for flow Courant number, explicit relaxation factor and under-relaxation factor.

| Parameters | Original Value | New Value |
|---------------------------------|----------------|-----------|
| Flow courant number | 200 | 50 |
| ERF: momentum | 0.75 | 0.3 |
| ERF: pressure | 0.75 | 0.45 |
| URF: density | 1.0 | 0.3 |
| URF: body force | 1.0 | 1.0 |
| URF: turbulent kinetic energy | 0.8 | 0.8 |
| URF: turbulent dissipation rate | 0.8 | 0.8 |
| URF: turbulent viscosity | 1.0 | 1.0 |
| URF: pollutant no | 0.9 | 0.9 |
| URF: energy | 1.0 | 1.0 |
| URF: temperature | 1.0 | 1.0 |
| URF: discrete ordinates | 1.0 | 1.0 |
| URF: mean mixture fraction | 1.0 | 1.0 |
| URF: mixture fraction variance | 0.9 | 0.9 |

solver are checked by the check case. In the current study, the calculations were performed by setting 3000 iterations as indicated in **Figure 6**. During model simulation, the convergence is reached easily when a set of equations are solved. After solving the gas flow equations, the particles route interacting with the flue gas are also solved. Later, the chemical reaction and enthalpy equations for combustion are calculated. These calculations are repeated several times until the flow and temperature is converged.

4. Model Results

Results for the model simulation can be shown either by numeral or as flow fields using the command File → Export → Solution data. Another option for result presentation is by using graphics which are contour plots, vector plots, iso-surface plots or by animation. Vector and contour plots are the most popular methods for the results presentation. In this study, the results shown in **Figure 7** are the contour plots in the graphics. In the Figure, options were set to filled, node values and global range which is a 2D view of the velocity magnitude and temperature profile.

Figure 7(a) shows the temperature distribution in the incinerator. The temperature reached in combustion chambers ranges between 930 K of 1700 K. There is an active combustion and in the primary combustion chamber, the main reactions are exothermic that drive the endothermic reactions in the secondary chamber. As gases flow from the incinerator up to the exit, the temperature gradually decreases due to the heat transfer from the gas to the incinerator walls via convection and radiation. The temperature also is seen to decrease in the right side of the secondary chamber but in the left side the temperature deviation is seen to decrease due to partly increased mixing and rotation of the gases.

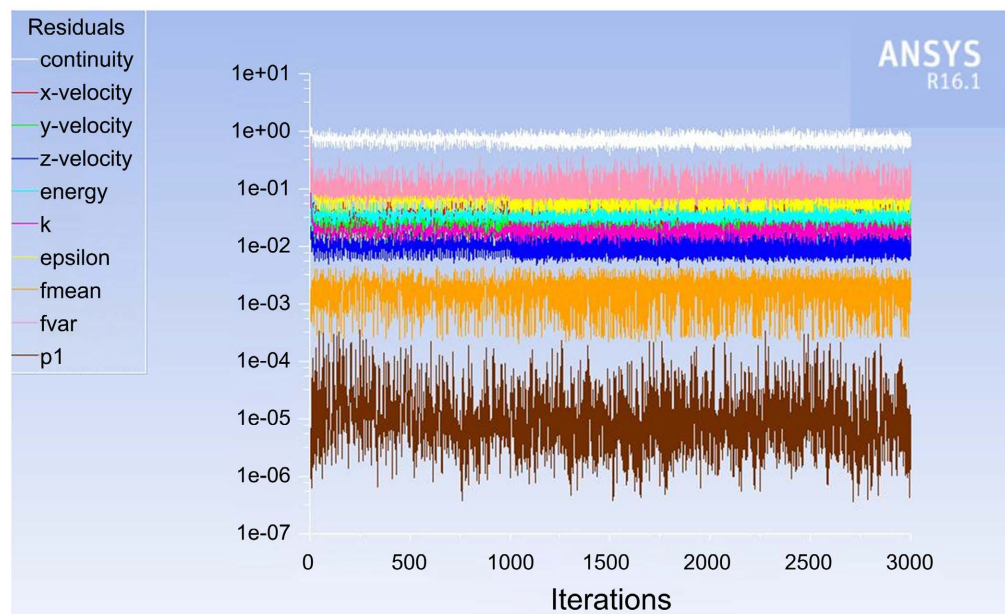
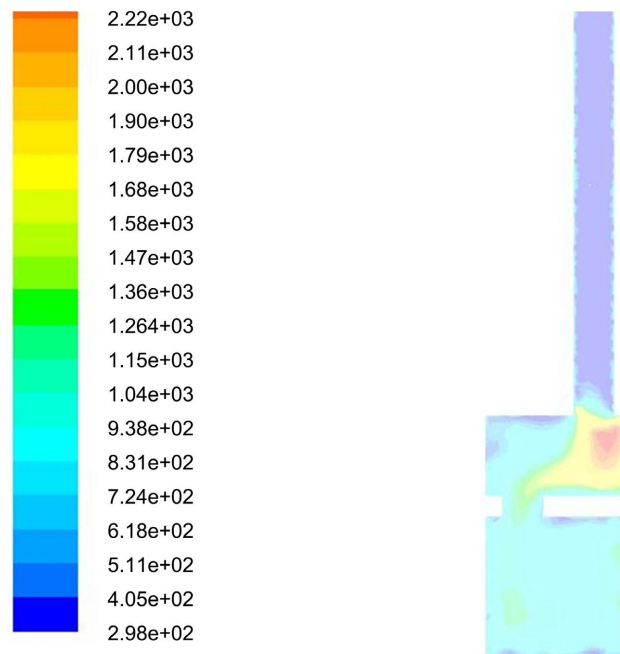
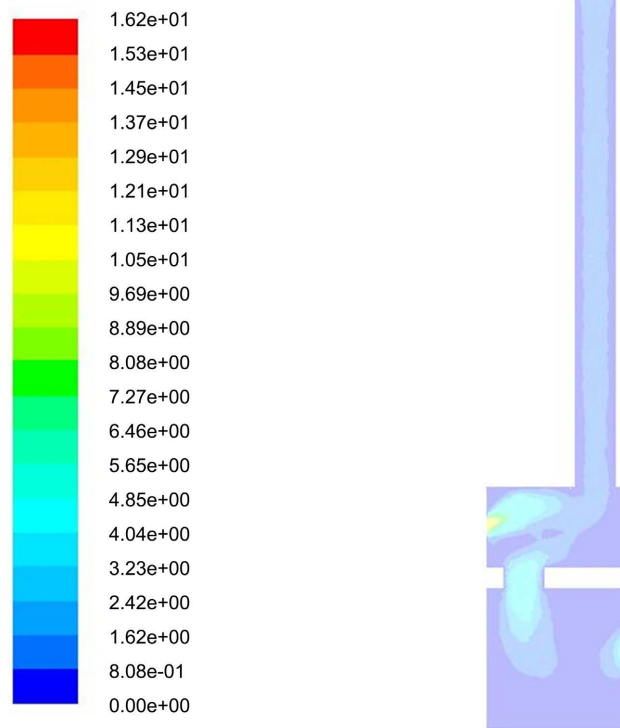


Figure 6. Convergence solution.



(a)



(b)

Figure 7. Contour plots (a) Temperature distribution (b) Velocity magnitude (m/s).

Finally, the temperature of the flue gas leaving the chimney at the outlet is around 405 K to 511 K.

The path lines of the temperature are shown in **Figure 8**. Routes formed by the fuel particles resemble the flue gas path lines but due to difference in their

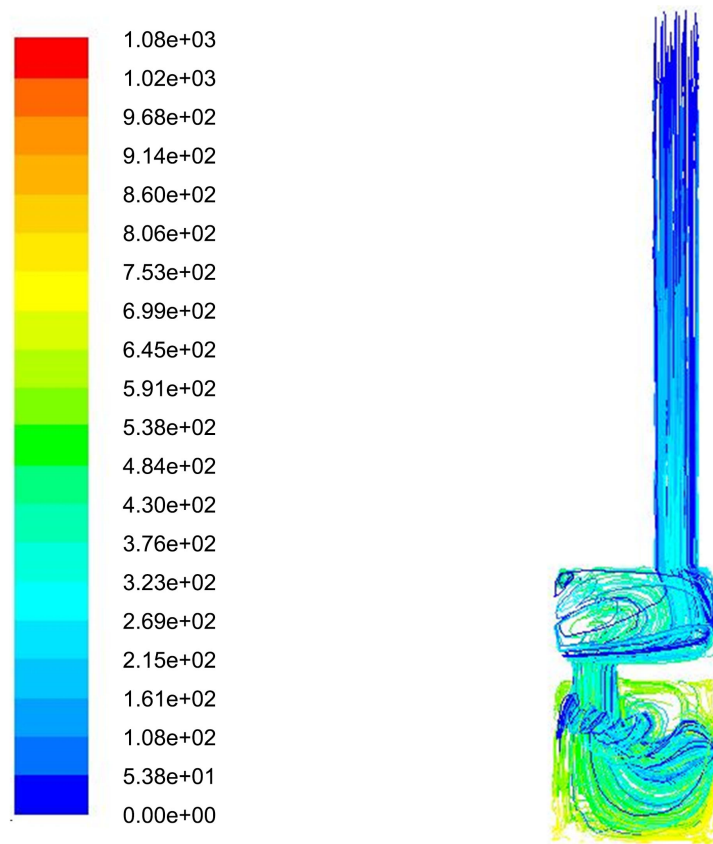


Figure 8. Path lines colored by velocity magnitude (m/s).

density and turbulence, there is no synchronization between them. The flue gases initially circulate in the bottom of the incinerator and thereafter travel up to the stack outlet. In this process, devolatilization and char combustion takes place simultaneously while the fuel particles are moving around the incinerator. The fuel species in the incinerator burns very quickly, but the char is consumed at the slower rate on the fixed grate at the center of the incinerator.

5. Conclusion

This study presents the design and computational Fluid Dynamics modeling for a municipal solid waste incineration process. From the study, it has been observed that the most significant step in CFD modeling is meshing. This means that quality of mesh on any model design has the highest impact for the solution to converge or to produce divergence. Also, choice of solution method and solution control has very big influence on the model and equations that are going to be solved. When the solution divergence occurs, it can be resolved by varying the so called explicit under relaxation factors where the changing rate is about 0.1 in each new simulation. Monitoring of the residual is an important factor in monitoring the iterations steps during convergence or divergence. The accurate design and combustion process require attention in selecting the calculation domain, generating mesh and choosing numerical models as temperature and

various flue gas concentrations are affected by fluid flow, temperature and oxygen concentration distribution. The input air and fuel supplied to combustion chambers were preheated to 315 K and 500 K respectively resulting in incineration process reaching combustion conditions as anticipated. The temperature reached in the combustion chambers ranged between 930 K to 1400 K and the temperature of the flue gas leaving the chimney at the outlet is around 405 K to 511 K.

Acknowledgements

The authors are grateful for the financial support given by Arusha Technical College during this study. They also recognize the support provided by the management and staff of Tanzania Engineering and Manufacturing Design Organization (TEMDO) for their assistance during laboratory work analysis.

Conflicts of Interest

The authors declare no conflicts of interest regarding the publication of this paper.

References

- [1] Huseyin, K., Senem, Y., Lokmen, G. and Goksel, D. (2016) Municipal Solid Waste Characterization according to Different Income Levels: A Case Study. *Sustainability*, **8**, 1044. <https://doi.org/10.3390/su8101044>
- [2] Krizan, P., Matus, M., Soos, L., Kers, J., Peetsalu, P., Kask, U. and Menind, A. (2011) Briquetting of MSW by Different Technologies in Order to Evaluate Its Quality and Properties. *Agronomy Research Biosystem Engineering*, **9**, 115-123.
- [3] Amina, A. (2016) Effectiveness of Solid Waste Management Systems in Local Government Authorities in Tanzania: The Case of Moshi Municipal Council. MSc. Thesis, Mzumbe University, Tanzania.
- [4] Arthur, M., Geoffrey, R., Karoli, N. and Peter, L. (2015) Operating Conditions of a Locally Made Fixed Bed Incinerator. A Case Study of Bagamoyo Tanzania. *International Journal of Environmental Monitoring and Analysis*, **3**, 80-90. <https://doi.org/10.11648/j.ijema.20150302.17>
- [5] Menard, Y., Patisson, F., Sessieq, P., Ablitzer, D., Merz, A. and Seifert, H. (2002) Transient Combustion of Municipal Solid Waste in a Grate Furnace: Modeling and Experiments. *European Conference, the Future of Waste Management in Europe*, Strasbourg.
- [6] Bjorn, B., Viktor, S. and Siegmars, W. (2014) Simulation of Municipal Solid Waste Incineration in Grate Firing Systems with a Particle Based Novel Discrete Element Method. Discrete Element Method to Simulate Waste Combustion. VGB Power Tech.
- [7] Parkinson, A., Balling, R. and Hedengren, J. (2013) Optimization. Methods for Engineering Design. Brigham Young University. Provo, Utah, USA, 208.
- [8] Ahmad, H. (2012) CFD Modeling of Grate Furnace Designs for Municipal Solid Waste Combustion. *Asian Transactions on Engineering*, **2**, No. 3.
- [9] Johari, A., Hashim, H., Met, R., Alias, H., Hassim, M. and Rozainee, M. (2012) Ge-

neralization, Formulation and Heat Contents of Simulated MSW with High Moisture Content. *Journal of Engineering Science and Technology*, **7**, 701-710.

- [10] Ansys, I. (2016) Ansys Fluent User's Guide. Ansys Incorporated, California, USA.