Global Scientific JOURNALS

GSJ: Volume 10, Issue 8, August 2022, Online: ISSN 2320-9186
www.globalscientificjournal.com

MODELING AND SIMULATION OF A REFUSE DERIVED FUEL INCINERATION PROCESS WITH FLUE-GAS CLEANING AND HEAT RECOVERY SECTIONS USING ANSYS FLUENT

M. M. Gaji[1], R. I. Ejilah[2], A. A. Bello[3]

[1]*Department of Linkages, Research and Consultancy, Energy Commission of Nigeria, Abuja*
[2,3]*Department of Mechanical/Production Engineering, Abubakar Tafawa Balewa University, Bauchi*

Email: engrmmgaji@yahoo.co.uk

Abstract

The modeling study involved the use of chemical and elemental compositions of the selected solid fuels in order to check their influence on the combustion process and to validate the experimental data. The geometry of the incinerator used has a maximum dimension of 350mm X 660mm X 914mm. The geometry was meshed and values of meshes were determined using ANSYS fluent software. The meshing geometry was converted to 8,433 cells which form Tetrahedral. The converted 8,433 cells were assigned a solver and boundary conditions. The steam outflow (temperature and pressure) were assigned as outlet and the perforated air inlet holes (at atmospheric temperature and pressure) were assigned as air-inlet. The combustion chamber is cylindrical in shape, incorporated with 6.35mm copper tube with 10 numbers coil. The perforated hole for supplying air to the combustion chamber is located under the grate to supply air for the chamber. The air inlet velocity was 4m/s throughout the simulation because the air flow is operating under natural conditions and the velocity at the exit of the incinerator is ranging between 0 – 297.423 m/s. The major composition of the flue gases were CO, CO_2, NOx and SOx in the amount of waste into the simulation process varied within a range to recreate the same operating conditions as in the experimental tests. The differences in flue gases may be attributed to the surface reactions caused by collision, coalescence and fragmentation between the dispersed phases. Thus, the pilot data showed a good match (similar pattern) with the ranges of variation between 9 – 13% with that of the theoretical simulation data which cut across the samples. These variations showed that the model can be acceptable under these conditions and the model created is valid under the same conditions.

Keywords: ANSYS Fluent Software, Finite-element, Meshing, Geometry, Boundary Conditions

1.0 Introduction

Heat recovery in units for the thermal processing of various types of waste i.e. waste to energy systems as well as in those for biomass combustion can be without any doubt considered as one of the most important parts of these processes (Stehlík, 2008). Design of equipment for utilization of energy contained in flue gas (and/or off-gas) from the thermal treatment of waste i.e. incineration and the placement in the process is one of key factors in these technologies. Heat recovery represents one of subsystems which enable to consider incinerators not only as units for the treatment of waste but as energy sources.

In the design and operation of heat recovery systems, it is necessary to take into account the characteristics of heat transfer equipment and/or heat exchangers and their specific features as well as those of process fluids (Stehlík, 2008). These various complicated process such as combustion, radiations and multiphase flow, must be known to designers (Huai, *et al.*, 2008). The incinerator two chambers are set with main reason that the primary chamber stays at low temperature and staved air in order to gasify the waste and minimize particulates to the secondary chamber (Hester, 2005). The secondary chamber is set to admit oxidant in order to complete burn all gases generated at primary chamber and destroy all incomplete combustion products (Morcos, 1989; Shin, *et al.*, 1998). The gases generated at primary chamber include CO, CO_2, H_2, H_2O, CH_4 and trace of hydrocarbons (Helsen and Bosmans, 2010). The speed and quantity of air inlet at the chambers are used to increase or decrease a residence time in primary or secondary chambers and therefore enhance combustion (Huai, *et al.*, 2008).

Conditions such as oxygen concentration, residence time, temperature and mixing turbulence has a big influence in the formation of pollutants (Mudakavi, 2010). The higher amount of CO in the exit is a sign of incomplete combustion (Kumar, *et al.,* 2014). The efficiency of an incinerator can be gauged by the concentration of effluent gases such as CO_2, O_2, CO, H_2 and NO_x (Ujama, *et al.*, 2013). Poisonous gases released in the effluent can be identified by using CFD techniques (Mor, *et al.,* 2006).

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics which uses numerical analysis and data structural to solve and analyze problems that involves fluid flow (Menard *et al.*, 2002). It is a system which work on a combination of various codes developed (Bjorn *et al.*, 2014). The CFD can be applied to solve and analyze problems which involve heterogeneous and homogeneous reactions. According to Menard *et al.*, 2002, CFD is applied in various complicated processes like combustion, radiation, multiple flow, mass change, velocity change and temperature change. Building a set of governing equations is required for modeling and simulation of these processes (Oran and Boris, 2001). According (Ahmad, 2012), due to availability of efficient computer systems nowadays, the numerical modeling technique such as CFD methods are used in industry as well as in academic. In the current work, process modeling and simulation of produced municipal solid waste combustion based refuse derived fuel – a case study of a single chamber incineration system is presented. ANSYS technique is used to develop the model for the simulation of the incinerator flow conditions.

2.0 Development of a Mathematical Model

Almost all the new research and development methods in combustion technology involve the application of computational fluid dynamics (CFD) in the design. This appears to be the best approach to solving design challenges. Mathematical modeling therefore is seen as an essential part of almost all combustion research programs. Models define our beliefs about how the world functions in mathematical modeling, that believes are translated into the language of mathematics. A wider view of our attempt to understand the physical world, describe it in the language of mathematics, and finally investigate its consequence by means of analytical, graphics or numerical methods is shown in Figure 1. A mathematical model is at best an estimate to the physical objects and these models are built based on a certain conservation principles or empirical observations. A constructed mathematical model can be classified as linear or nonlinear, steady state or dynamic. A numerical model or computer simulations is an approximation to the mathematical model (Goldschmidt and Smolkov, 2006). The importance of mathematical model has been recognized by its ability to solve various problems relating to transport phenomena in exploring the link between the physical objects and the mathematical one for transport processes especially those consisting of momentum, heat and mass transfer (Servedio *et al.*, 2014). Currently, computers have become the important tool to analyze and solve problems concerning mathematical models. Significant advances have been made in their ability to analyze nonlinear systems. As a result of these advances computer aided design (CAD) have become standard tools and is evidenced by the success of several commercial packages including ASPEN PLUSS, ANSYS, FLUENT and MULTPHYSICS. There is also a general purpose mathematical tool such as MATLAB which has been able to solve mathematical functions both symbolically and numerically (Servedio *et al.*, 2014).

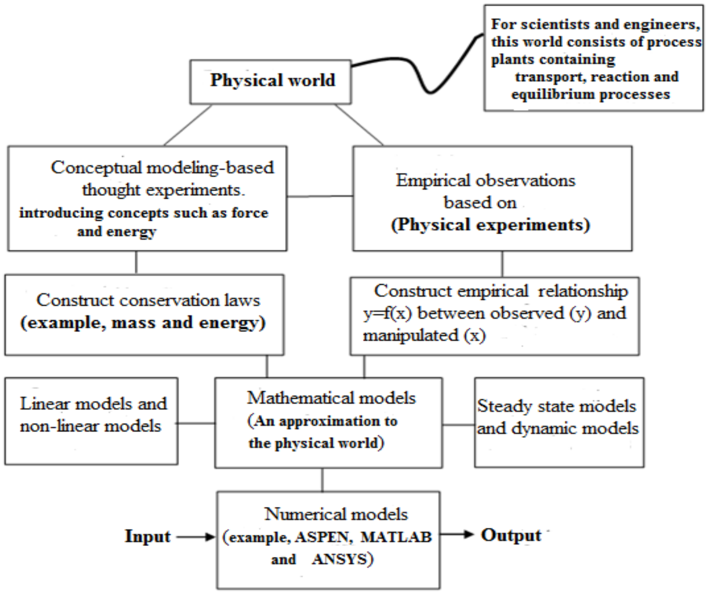

Figure 1: A Scheme of the Mathematical Model (Servedio et al., 2014)

3.0 Governing equations

The governing equation of a mathematical model describe how the values of the known variables (dependent) changes when one or more value of the known (independent) variable change. The mathematical modeling of the fluid flow is based on a set of coupled conservation equations of mass, momentum, energy and chemical species. The governing equations set for a general three-dimensional (3D) fluid flow is called Navier-Stokes equations. 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. The latter is computed from corresponding chemical species conservation equations. Different physical quantities are applied by every equation in a manner that they are considered as dependent variables but the stability of factors inducing then variables should be assured. For the incineration process, dependent variables are such as velocity, temperature and chemical species, which are expressed on mass fractions.

Computational Fluid Dynamic (CFD) methodology is shown in Figure 2 and it consists of three main elements which are;

(i) Pre-processing - create geometry of the problem, generate computational mesh, define the flow parameters, as well as initial and boundary conditions

(ii) Solver - computes the solution of the governing equations.

(iii) Post-processing – analyze the data and display results (data, tables, graphs).

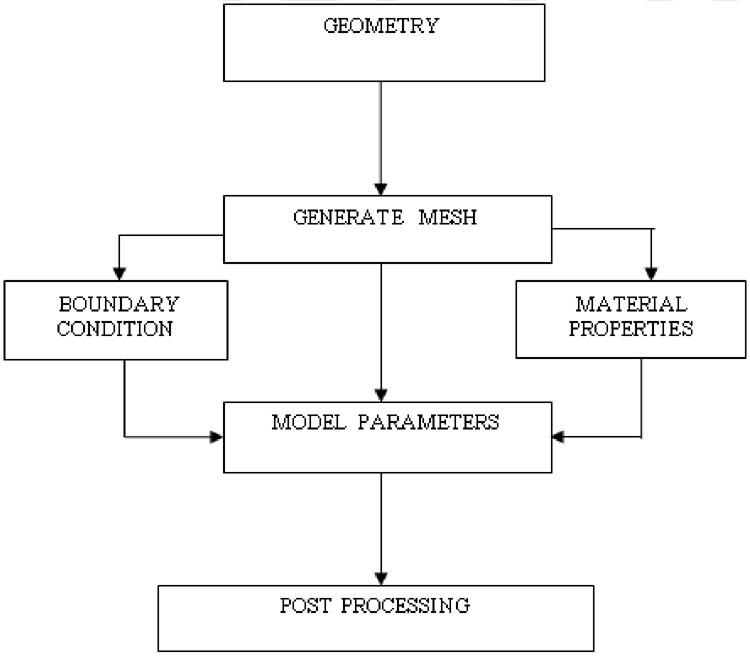

Figure 2: A Computational Fluid Dynamics Procedure (Mtui, 2013)

First of all, a general sketch of the system was prepared by preliminary data and views. After necessary editions, a digital geometry of the object was prepared through Computer Aided Design (CAD) with the help of available platforms, i.e. AutoCAD (Autodesk), SolidWorks, Design Modeler (ANSYS), CATIA etc. The choice of 2-dimensional or 3-dimensional geometry depends upon the available computational resources as well as the system configurations. The geometry (computational domain) was then divided into small computational cells or grids. Surfaces, e.g. inlet, outlet, walls etc., were also defined for better adaption to real operation by appropriate boundary conditions. The mesh was then imported into the solver, e.g. ANSYS (Fluent).

4.0 Model Validation

In order a model to be accepted for its validity and whether it can be used for its accepted purpose, it must prove its ability of performing the simulation work of real procedures to the acceptable standards (Ahmad, 2012; Johari *et al*., 2012; Tian *et al*., 2001). In order to proof that a model is accurate and that it will satisfy all cases intended for, there must a lot of trials for various conditions, sometimes this cannot be achieved. Validation is usually not based on only considering data that was used in the construction of the model but also employs data that was not used in the construction. In other words, validation usually includes testing some of the model's predictions. A model developed for a specific case cannot be expected to handle other cases problems which can come later.

When a model is developed for specific conditions, the testing of its validity will involve proofing its specific case in which the model was developed for. The modeling and simulation of incineration systems emphases on essential steps of modeling and simulation, that gives a relatively simplified picture of what really occurs in the incineration process. Verification of computer simulation models is conducted during the development of a physical model with the ultimate goal of creating correct and reliable outputs. When the model has been studied and is satisfied with its performance, it is tested against its observation from the physical system it represents. In the verification step, the model results are compared with experimental data obtained from physical incinerator constructed. Usually, these are the data from experiments that have been specifically designed to verify the model. Therefore, a model is needed to fit the data not only quantitatively, but also qualitatively in a way that it imitates the general shape of the data as closely as possible. Verification of the model can also rely on the data that was not obtained from the physical model constructed. In this study, model verification was based on the data obtained from the physical model.

5.0 Analysis of combustion parameters and investigation of simulation results and discussions at different contours

The modeling in this study involved the use of chemical and elemental compositions of the selected solid fuels in order to check their influence on the combustion process.

The 3D incinerator geometry used in this study is depicted in Fig. 3. The incinerator geometry has a maximum width of 350 mm, maximum depth of 660 mm and maximum height of 914 mm.

The geometry is meshed and values of meshes are determined using ANSYS fluent software. The geometry of computational model was performed using the SolidWorks 2021. The meshing geometry is converted to 8,433 cells. The cells were then converted to Tetrahedral. The total converted cells were 8,433. The meshed incinerator design is shown in Fig. 4 below.

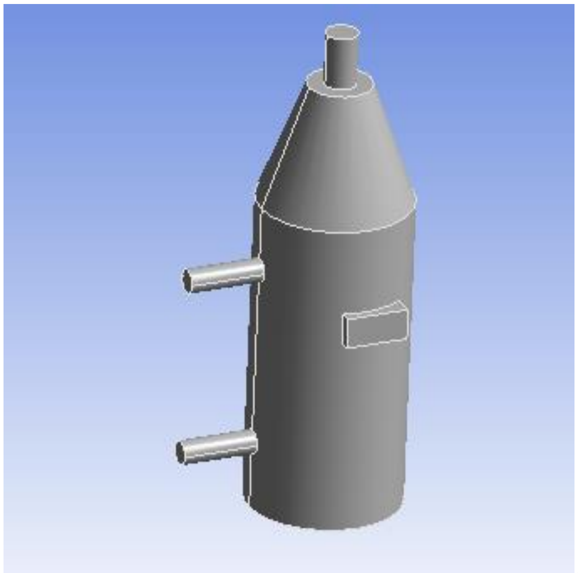

Figure 3: Incinerator Geometry

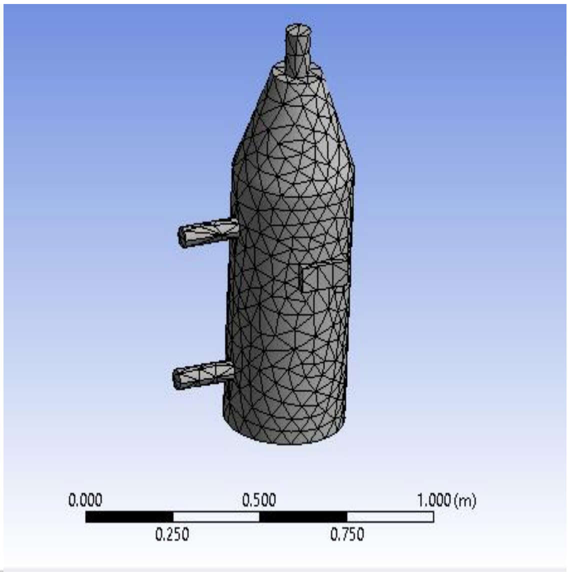

Figure 4: Incinerator Mesh

The converted 8,433 cells were assigned a solver and boundary conditions. The steam outflow (temperature and pressure) were assigned as outlet and the perforated air inlet holes (at atmospheric temperature and pressure) were assigned as air-inlet. The perforated hole for supplying air to the combustion chamber is located under the grate to supply air for the chamber. The air inlet velocity was 4m/s throughout the simulation because the air flow is operating under natural conditions and the velocity at the exit of the incinerator is ranging between 0 – 297.423 m/s.

The air inlet velocity is 4 m/s throughout the simulation, the maximum velocity at the exit of the incinerator is ranging between 0 – 297.423 m/s with average velocity of 297.423 m/s as shown in Fig. 5. The air inlet velocity was 4m/s throughout the simulation because the air flow is operating under natural conditions and the velocity at the exit of the incinerator is ranging between 0 – 297.423 m/s. The formation of gaseous material at the combustion chamber increases the velocity of gases. The O_2 concentration in the combustion chamber increases the velocity and residence time due to excess air supplied (Liang and Ma, 2010).

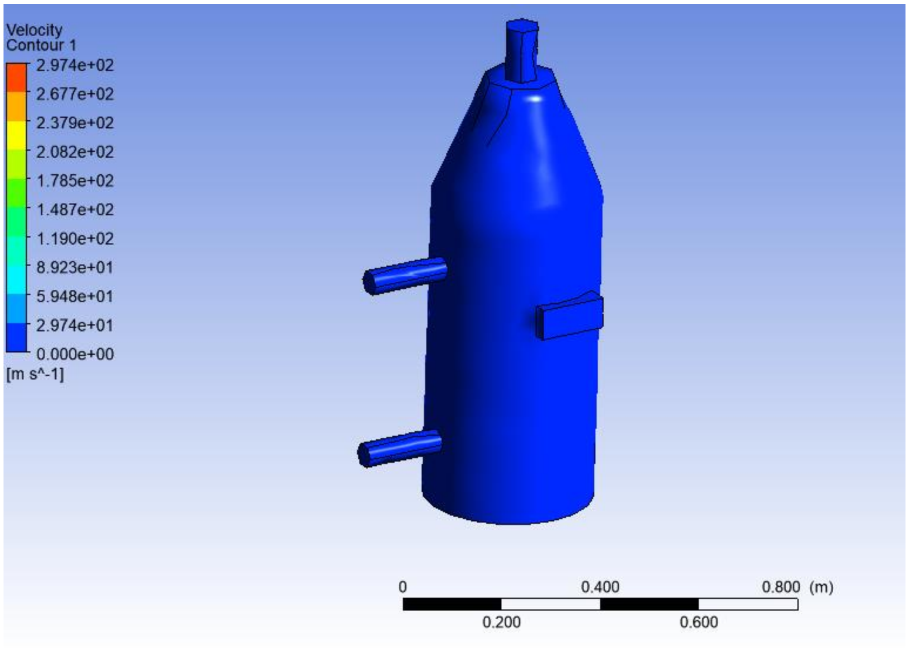

Figure 5: Maximum Velocity at the Outlet

The air inlet velocity was 4m/s throughout the simulation with the input parameters in table 1 and 2 below because the air flow is operating under natural conditions and the velocity at the exit of the incinerator is ranging between 0 – 297.423 m/s. The major composition of the flue gases were CO, CO_2, NOx and SOx in the amount of waste into the simulation process varied within a range to recreate the same operating conditions as in the experimental tests.

For the simulation of the gases, the flue gases appeared to be more diluted than in the experimental study. Regarding the CO and CO_2 contents, there were better for these flue gases which indicated high yield. The differences in flue gases may be attributed to the surface reactions caused by collision, coalescence and fragmentation between the dispersed phases as represented in Figures 6 to 8. Thus, the pilot data showed a good match (similar pattern) with the ranges of variation between 9 – 13% with that of the theoretical simulation data which cut across the samples. These variations showed that the model can be acceptable under these conditions and the model created is valid under the same conditions.

Table 1: Water and steam properties

S/No	Samples	Water Flow Rate (water mass) (kg/s)	Water Temperature, K	Steam Flow Rate (water mass) (kg/s)	Steam Temperature, K
1	100%PL	0.26	301	0.32	401.3
2	75%PL + 25%TE	0.43	301	0.47	428.2
3	50%PL + 50%TE	0.49	301	0.55	474.0
4	25%PL + 75%TE	0.51	301	0.58	485.5
5	100%TE	0.37	301	0.45	448.1
6	100%PA	0.32	301	0.38	411.2
7	75%PA + 25%PL	0.41	301	0.47	430.4
8	50%PA + 50%PL	0.49	301	0.57	448.0
9	25%PA + 75%PL	0.55	301	0.63	474.4
10	100%WO	0.17	301	0.27	378.0
11	75%WO + 25%PL	0.34	301	0.46	410.5
12	50%WO + 50%PL	0.44	301	0.59	424.0
13	25%WO + 75%PL	0.62	301	0.69	440.7

Table 2: Determination the material properties of the following RDF briquettes formed

S/No	Refused Derived Fuel	Mass (kg)	Density (kg/m^3)	Young Modulus (GPa)	Poisson Ratio	Yield Strength (MPa)	Ultimate Tensile Strength (MPa)	Thermal Conductivity (W/(m.K)	Thermal Expansion Coefficient (10^{-5} /°C)	Specific Heat (KJ/kg K)
1	100%Plastic	3.0	920.12	0.91	0.51	27.23	53.20	0.83	12.52	1.69
2	75%Plastic+25%Textile	3.0	789.49	0.86	0.49	22.30	54.23	0.76	8.09	1.04
3	50%Plastic+50%Textile	3.0	714.29	0.85	0.42	20.70	55.50	0.70	7.88	0.90
4	25%Plastic +75%Textile	3.0	638.30	0.79	0.39	16.41	60.45	0.62	6.53	0.88
5	100%Textile	3.0	572.04	8.10	0.43	14.60	63.12	0.35	4.23	1.30
6	100%Paper	3.0	795.13	1.04	0.38	7.20	15.05	0.26	2.03	1.34
7	75%Paper+25%Plastic	3.0	833.33	8.70	0.40	8.91	24.16	0.28	3.90	1.39
8	50%Paper+50%Plastic	3.0	857.14	8.73	0.44	9.40	29.04	0.35	5.14	1.40
9	25%Paper+75%Plastic	3.0	909.09	8.84	0.47	10.82	31.35	0.40	6.04	1.52
10	100%Wood	3.0	610.04	9.12	0.34	41.07	54.09	0.41	3.81	1.76
11	75%Wood+25%Plastic	3.0	666.67	0.88	0.37	38.12	56.02	0.43	5.92	1.79
12	50%Wood+50%Plastic	3.0	731.71	0.91	0.39	36.65	57.80	0.50	6.60	1.80
13	25%Wood+75%Plastic	3.0	833.33	1.02	0.42	30.10	59.02	0.59	7.30	1.80

Table 3: Experimental and Simulation results

		Calorific Value (J/kg)				Calorific Value (J/kg)			
		Experiment				Simulation			
S/N	**Samples**	**CO**	**CO_2**	**NO_X**	**SO_X**	**CO**	**CO_2**	**NO_X**	**SO_X**
1	100% PL	145.76	118.16	112.52	84.58	129.7264	105.1624	1.00E+02	7.53E+01
2	75%PL + 25%TE	184.32	149.41	142.26	106.94	164.0448	132.9749	1.27E+02	9.52E+01
3	50%PL + 50%TE	268.68	217.77	207.37	155.89	239.1252	193.8153	1.85E+02	1.39E+02
4	25%PL + 75%TE	288.46	233.80	222.63	167.35	256.7294	208.082	1.98E+02	1.49E+02
5	100% TE	191.61	155.32	147.89	111.17	170.5329	138.2348	1.32E+02	9.89E+01
6	100%PA	106.21	86.09	81.98	61.32	94.5269	76.6201	7.30E+01	5.46E+01
7	75%PA + 25%PL	158.29	128.30	122.17	91.83	140.8781	114.187	1.09E+02	8.17E+01
8	50%PA + 50%PL	194.71	157.84	150.30	112.97	173.2919	140.4776	1.34E+02	1.01E+02
9	25%PA + 75%PL	210.36	170.50	162.35	122.05	187.2204	151.745	1.44E+02	1.09E+02
10	100%WO	231.18	187.39	178.43	134.12	205.7502	166.7771	1.59E+02	1.19E+02
11	75%WO + 25%PL	191.61	155.32	147.89	111.17	170.5329	138.2348	1.32E+02	9.89E+01
12	50%WO + 50%PL	161.39	130.82	124.59	93.64	143.6371	116.4298	1.11E+02	8.33E+01
13	25%WO + 75%PL	116.64	94.55	90.02	67.67	103.8096	84.1495	8.01E+01	6.02E+01

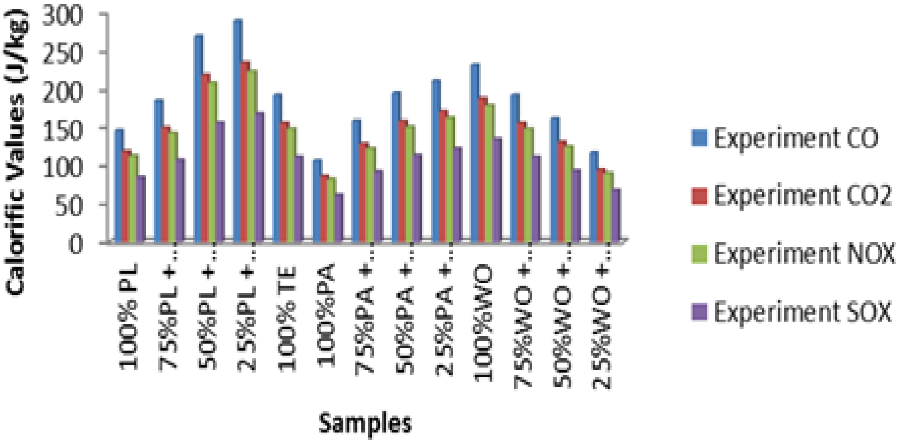

Figure 6: Line Chart of Experimental results of the Flue Gasses

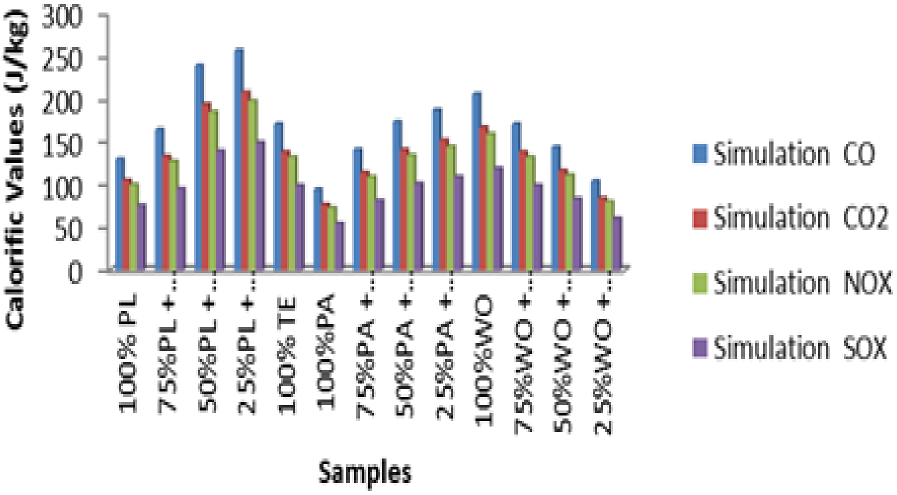

Figure 7: Line Chart of Simulation results of the Flue Gasses

6.0 Conclusion

The design of the Incinerator’s geometric model was performed using the SolidWorks 2021. The input data to the model were generated by the experimental practice to feed in the model for process modeling and simulation where the air inlet velocity was 4 m/s throughout the simulation and the maximum velocity at the exit of the incinerator was ranging between 0 – 297.423 m/s. The major composition of the flue gases were CO, CO_2, NOx and SOx in the amount of waste into the simulation process varied within a range to recreate the same operating conditions as in the experimental tests.

The experimental data was used to determine the modeling process and the validation results show that the simulation of the flue gases appeared to be more diluted than in the experimental study. Regarding the CO and CO_2 contents, the energy evolved ranges between 116.64 J/kg to 288.46 J/kg and 86.09 J/kg to 233.80 J/kg for experimental practice while 94.5269 J/kg to 256.7294 J/kg and 76.6201 J/kg to 208.082 J/kg for simulation respectively which attributed to the surface reactions caused by collision, coalescence and fragmentation between the dispersed phases. The pilot data showed a good match (similar pattern) with the ranges of variation between 9 – 13% with that of the theoretical simulation data which cut across the samples. These variations showed that the model can be acceptable under these conditions and the model created is valid under the same conditions.

References

1. Ahmad, H. (2012). CFD modeling of grate Furnace Designs for Municipal Solid Waste combustion. *Asian Transactions on Engineering (ATE),* volume 02, issue 03,2221-4267.

2. Bjorn, B., Viktor, S., Siegmar, 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.*

3. Goldschmidt, G., Smolkov, M. (2006). Variances in the Impact of Visual Stimuli on Design Problem Solving Performance. *Design Studies*, 27 (5), pp. 549-569.

4. Helsen L, Bosmans A. Waste-to-energy through thermochemical processes: matching waste with process. Symposium on Enhanced Landfill Mining. Houthalen-Helchteren, Belgium; 2010.

5. Hester R. E., (2005). Waste treatment and disposal, Cambridge, UK, Royal Society of Chemistry; 2005.

6. Huai X, Xu W, Qu Z, Li Z, Zhang F, Xiang G, Zhu S, Chen, G. Analysis and optimization of municipal solid waste combustion in a reciprocating incinerator. Chemical Engineering Science. 2008; 63:3100-3113.

7. Johari, A., Hashim, H., Met, R., Alias, H., Hassim, M., Rozainee, M. (2012). Generalization, Formulation and heat contents of simulated MSW with high moisture content. *Journal of engineering Science and Technology,* Vol 7, no_6, 701-710.

8. Kumar, S., Balakrishnan, P., Chandraseelan, R. (2014). Numerical Investigation of Swirl Enhancement for Complete Combustion in an Incinerator. *International Journal of Engineering & Technology,* vol. 6, 0975-4024.

9. Menard, Y., Patisson, F., Sessiecq, P., Ablitzer, D., Merz, A., Seifert, H. (2002).*Transient combustion of municipal solid waste in a grate furnace*: modeling and experiments.

10. Mor S, Ravindra K, De Visscher A, Dahiya R, Chandra A. Municipal solid waste characterization and its assessment for potential methane generation: a case study. Science of the Total Environment. 2006; 371:1-10.

11. Morcos, V. (1989). Energy recovery from municipal solid waste incineration—A review. Heat Recovery Systems and CHP. 1989;9:115-126.

12. Mudakavi J. (2010). Principles and practices of air pollution control and analysis, New-Delhi, India, I.K. International Pvt Ltd; 2010.

13. Oran, E., Boris, J. (2001). *Numerical Simulation of Reactive Flow*, 2nd edition, Cambridge University Press.

14. Servedio, M., Brandvain, Y., Dhole, S., Fitzpatrick, C., Goldberg, E., Stern, C., Van Cleve, J., Yeh, D. (2014). Not just a theory-the utility of mathematical models in evolutionary biology. *PLoS Biol,*12, e1002017.

15. Shin D, Ryu CK, Choi S. (1998). Computational fluid dynamics evaluation of good combustion performance in waste incinerators. Journal of the Air & Waste Management Association. 1998; 48:345-351.

16. Stehlík, P. (2008). Effective utilization of heat in waste and biomass processing. *HEFAT 2008*.

17. Tian W., Wei, X., Wu, D., Shag, H. (2001). Analysis of ingredient and heating value of Municipal Solid Waste. *Journal of Environmental Science,* 87-91.

18. Ujama A, Ebohb F, Chimec T., (2013). Effective utilization of a small-scale municipal solid waste for power generation. Journal of Asian Scientific Research. 2013; 3:18-34.