2-D CFD simulation study of radioactive solid waste incinerator system

Keyur C. Pancholi*[1,2], S. Agarwal[2], T. Vincent[3,1], C.P. Kaushik[2,1]

[1]Homi Bhabha National Institute, Mumbai-400094, India
[2]Waste Management Division, Nuclear Recycle Group
[3]Process Development Division, Nuclear Recycle Group
Bhabha Atomic Research Centre, Trombay, Mumbai-400085, India

ABSTRACT

Low active radioactive solid waste of Category-I nature ($\leqslant$ 20 mR/h) requires volume reduction before its eventual disposal in Near Surface Disposal Facility (NSDF). Conventional incineration is used for management of cellulosic waste which provides a volume reduction factor in the range 30-50. A 2D CFD (Computational Fluid Dynamics) study of an existing diesel fired solid waste incinerator is presented by simulating turbulent flow coupled with heat transfer and reactions to understand the temperature and concentration distributions inside the incinerator chamber during incineration of cellulosic waste. The temperature at a reference thermocouple location obtained from simulation is compared with actual operational data and a good agreement is observed. The results of the present study will form reference data of temperature and concentration distributions in the incinerator chamber for a similar study with plasma-based system planned for combustible waste forms.

Keywords: Incineration, Radioactive Solid Waste, Volume Reduction Factor, 2D CFD study

Introduction

Large quantities of secondary solid wastes with potential radioactive contamination are generated from the operation and maintenance of nuclear power plants, the nuclear fuel cycle facilities, research laboratories, radiation facilities and medical facilities. These wastes have major contribution of Very Low-Level Waste and Low-Level Waste ($\leqslant$ 20 mR/h) [1,2], having more than 50% volume of combustible materials such as cellulose, rubber and plastics [3,4]. These wastes need to be treated and conditioned into a form acceptable for safe storage and disposal. Processes like incineration, compaction or melt densification [4,5] are in practice for the same. Incineration is the most preferred option as a treatment method due to associated advantage of high Volume Reduction Factor (VRF more than 30) [4,5]. Cellulosic wastes are generally treated by diesel fired incinerators.

At present a diesel fired incinerator of 50 kg/h capacity is in use at Solid Waste Management Facility of Waste Management Division of BARC for effective management of potentially contaminated cellulosic wastes. A CFD simulation study of incineration chamber of this system for processing of cellulosic waste using diesel burner has been performed. The study provides useful insights into the concentration and temperature distributions inside the incineration chamber. Temperature at the reference thermocouple location obtained from CFD simulation is compared with actual measurement during experimental runs.

Processing of rubber and plastic wastes by conventional incineration could release toxic gases like dioxins and furans due to lower operating temperatures and hence not preferred [6,7]. Plasma-based incineration is considered as a source of high temperature (>1500 K) which can effectively help in suppressing the formation of dioxins and furans [7-10]. Therefore, experiments with plasma torch as an intense heat source for preheating and waste decomposition are underway. Simulating existing incineration chamber is important before simulating plasma-based incineration chamber as, due to very high temperature gradients, simulations of plasma incineration chamber are expected to be computationally challenging. Also understanding of the temperature distribution in the existing diesel fired incineration chamber will lead to better insights into expected temperature distribution in plasma incineration chamber.

System Description

The system studied is a typical diesel fired incinerator for management of radioactive cellulosic waste. The main component of the system is a combustion chamber. Diesel fired burner is used to preheat the chamber and then the waste box (containing cellulosic waste) is introduced inside the chamber for incineration in presence of adequate air supply. The chamber is a cuboidal geometry with cross-section of 1 m x 1 m. The 2D schematic image of the combustion chamber is shown in Fig.1.

Pilot air and diesel are fed from a side nozzle in segregated mode, and mix while entering the chamber. At the entry of the diesel and air, ignition spark is provided for onset of combustion. Main air for waste combustion is fed from a side nozzle at the bottom. Waste is fed in a carton box. A thermocouple is used to track the progress of combustion process in the chamber by measuring the temperature.

The detailed inlet conditions of air and diesel are given in Table 1. Flow pattern inside the chamber, temperature of gaseous mixture in the chamber and the temperature of the surfaces of the

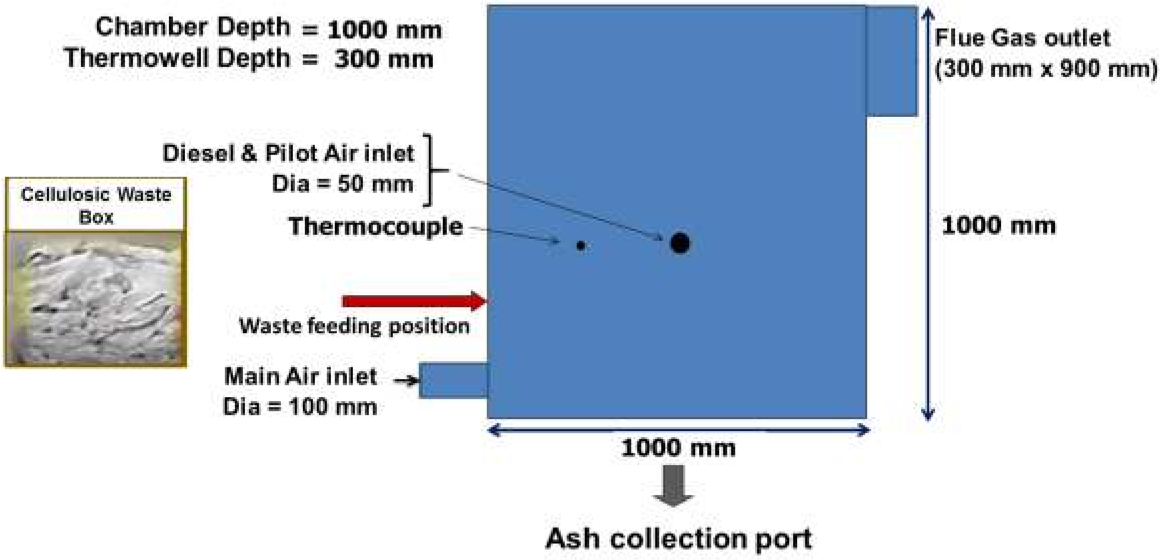


Fig.1: Schematic of primary chamber of incinerator setup

Table 1: Inlet conditions for diesel, pilot air and combustion air

	Air (For waste combustion)	Pilot Air (For diesel ignition)	Diesel
Flow rate	4 m^3/min	0.8 m^3/min	0.2 kg/min
Velocity	4 m/s	4 m/s	2 m/s
Temperature	300 K		
Pressure	100 kPa		

waste box are important parameters which affect cellulose combustion and rate of heat evolved during combustion.

Reaction Kinetics:

Overall reactions of diesel and cellulose combustion are as given below [11]:

Diesel combustion:

$$4C_{12}H_{23} + 71O_2 \rightarrow 48CO_2 + 46H_2O \quad (\Delta H = -44.8\ kJ/g) \quad (1)$$

Two-step cellulose combustion:

Step-1:

$$C_6H_{10}O_5 \rightarrow 5CO + 3H_2 + CH_4 \quad (\Delta H = 0.3\ kJ/g) \quad (2)$$

Step-2:

$$5CO + 3H_2 + CH_4 + 6O_2 \rightarrow 6CO_2 + 5H_2O \quad (\Delta H = -14\ kJ/g) \quad (3)$$

$$C_6H_{10}O_5 + 6O_2 \rightarrow 6CO_2 + 5H_2O \quad (\Delta H = -13.7\ kJ/g) \quad (4)$$

Eq. 1 is used for simulating the diesel reaction in the chamber. The cellulose combustion is two step reaction, in which step-1 is volatile formation by decomposition of cellulose and step-2 is combustion of volatile formed in step-1. However, in the model overall rate of combustion (Eq. 4) is used for simulating the surface reaction of the waste box. For estimating the combustion rate, order of reaction is taken as first order with respect to O_2 concentration and zeroth order with respect to cellulose concentration (cellulose in excess at the surface). To estimate the local surface reaction rate and corresponding local heat generation rate, the surface temperature and local oxygen concentration are used. The kinetic data in literature [11] suggests that, the decomposition reaction is very fast for the temperature greater than 673 K, fast for temperature range of 573-673 K and slow if the temperature is less than 573 K. Further, if temperature is less than 473 K, hardly any decomposition takes place.

Experimental study for cellulosic waste incineration

Actual low active cellulosic waste is incinerated in the incineration chamber (Fig. 1). Preheating of the chamber is done with diesel feed at 12 kg/h. The steady state temperature achieved in the chamber at the reference location, only with diesel combustion, is 573-673 K, which is adequate for onset of decomposition of cellulosic wastes [11]. Only after attaining the adequate steady state temperature, waste cellulosic feed is introduced in the form of a waste box. Combustion of cellulose provides additional heat to the chamber leading to further increase in the temperature. After sometime, temperature starts reducing which indicates that combustion is subsiding due to significant reduction in the combustible mass. At this time, a new box is fed for continuation of combustion and heat addition for sustaining the temperature in the chamber. The two-stage combustion phenomena, volatilization on box surface and oxidation nearby the surface of the box, observed during combustion of box in the chamber is shown in Fig. 2.

Simulations

The present study involves the simulation of diesel fired incinerator for a specific geometry using COMSOL Multiphysics software. The model solves turbulent flow (using standard k-ε model) coupled with heat transfer and reactions. The governing equations solved include continuity equation, momentum equation, energy equation, species transport equation, and equations of k-ε model of turbulence. The simulation procedure is benchmarked for the combustion studies using Sandia Flame D experimental results [12]. Geometry used for validation is shown in Fig. 3 (a).

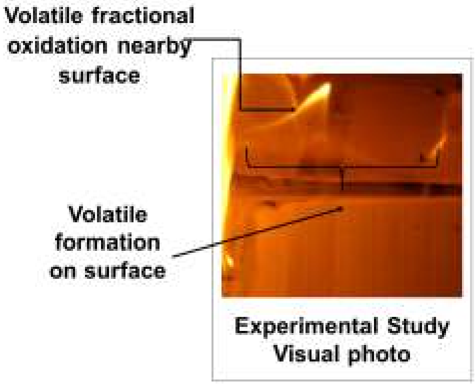


Fig.2: Photograph of waste box processing in combustion chamber

Sandia Flame D is a turbulent piloted methane jet flame and provides a standard and high-quality reference data set for validation of turbulent flame calculations. The experimental setup consists of central main jet surrounded by a pilot jet and a slow co-flow of air outside. The hot mixture from the pilot jet, besides stabilising the main jet, is also responsible for igniting the fuel which is injected from the main jet.

The simulation uses standard k-ε model for turbulence modelling, reacting flow with turbulence chemistry interaction and energy transport equations. The simulation results need to be independent of the grid resolution. This is verified by running simulations with higher resolution grids. Adequate grid resolution is provided near the jet axis and close to the burner where gradients are expected to be large.

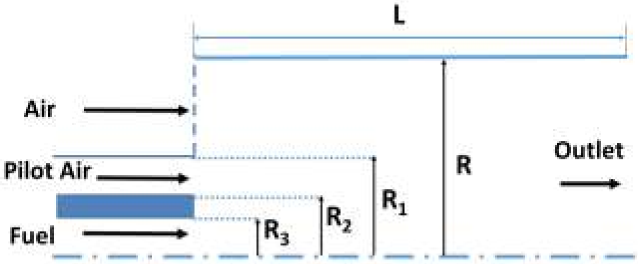
Dirichlet boundary condition is prescribed at the inlets for all variables i.e. velocity, temperature and species mass fractions. At the chamber walls, no-slip, insulated wall and zero-flux boundary conditions are used for momentum, energy transport and species transport, respectively. At the surface of the waste box, no-slip condition along with species flux and heat flux due to surface reaction are specified. Pressure outlet and outflow boundary conditions (zero concentration and temperature gradients) are specified at the outlet. As observed in Fig. 3(b), predicted results are in good agreement with the experimental results of Sandia D Flame with minor under-prediction or over-prediction of product species.

After validation of the CFD model with literature data, the CFD analysis was taken up for the incineration chamber. The complexity of the system requires transient multiphysics (coupled solution of momentum transport, energy transport, species transport along with an appropriate model of turbulence) 3D simulation. During the course of processing, the waste box shrinks as it undergoes combustion. This should also be accounted in the CFD model. However, a CFD model incorporating complete physics will be too complex. Due to the complexities involved, in the first-step of CFD model development, a CFD model with ceratin assumptions has been developed and reported in this study. The model will be gradually taken forward by incorporating more complexities to realistically simulate the actual system.

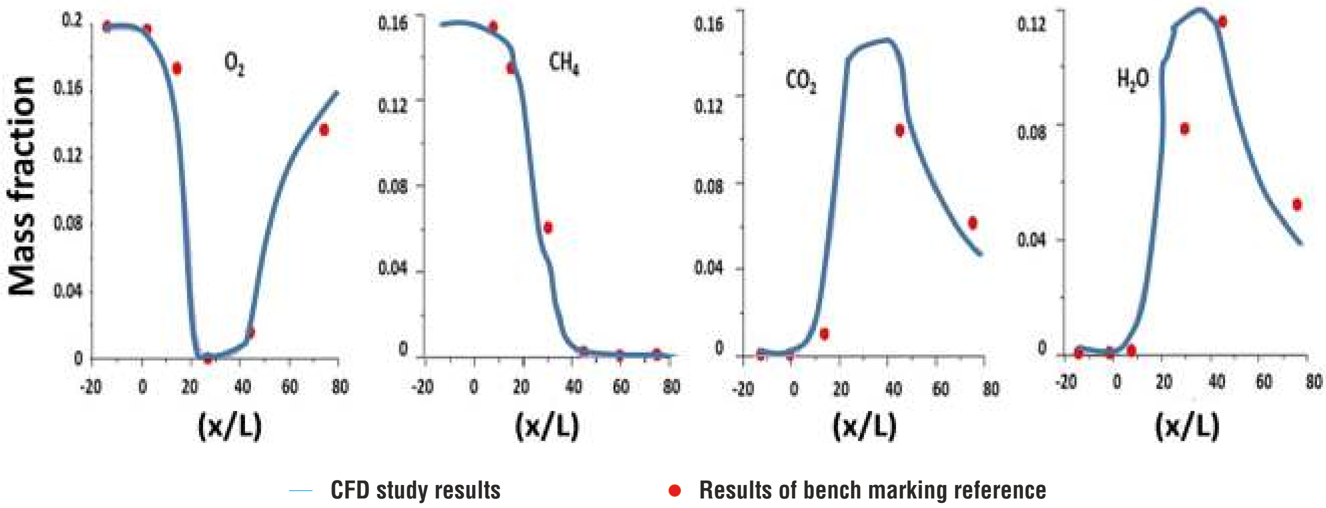
A 2D geometry has been considered for the simulation. The geometry and meshed computational domain are shown in Fig. 4. There are five sub-components in the system, i.e. main incineration chamber, fuel and pilot air inlet nozzles, air supply nozzle, outlet of the flue gases and waste box. Mass feed rate of diesel and air have been kept the same as in the actual plant conditions.

Following assumption are used while carrying out the CFD simulations:

1. 2D CFD steady state simulation is performed maintaining the geometric and kinetic similarity.
2. Walls of the chamber are assumed to be perfectly insulated with no heat loss. Only heat loss from the system is through the outgoing flue gases.
3. Combustion of waste box is considered as the surface reaction. No bulk reaction is simulated for the waste box. Overall combustion reaction as given in Eq. 4 is used.



(a) Geometry used for validation of simulation methodology



(b) Comparison of predicted and experimental mass fraction profiles

Fig.3: Validation of the CFD model using Sandia Flame D experimental results (x/L: relative axial position)

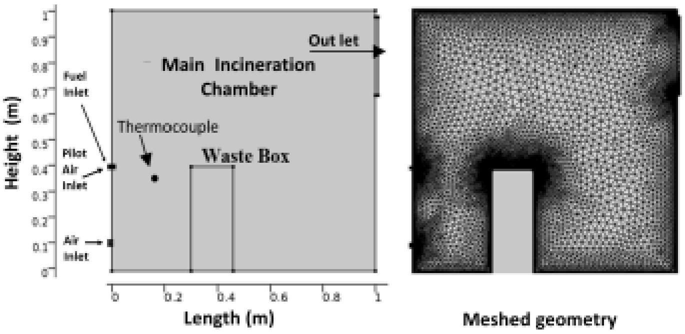


Fig.4: Typical geometry for CFD simulation and meshed computational domain

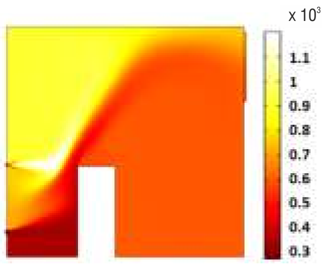
4. Size of the waste box is considered to remain same while the surface reactions at the edges of the waste box continue.

5. Turbulent mixing model has been assumed for diesel combustion as the reaction is spontaneous once favourable temperature is achieved.

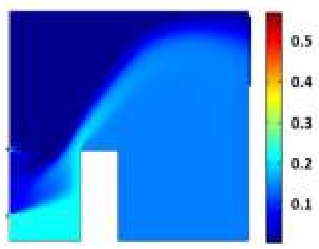
The simulations are carried out in phase-wise manner. Initially, flow patterns of diesel (vapor phase) and air are simulated. The results of this simulation are taken as initial solution for simultaneous solution of flow and diesel combustion to estimate the temperature and species distribution in the chamber. These results are then used as the initial conditions to simultaneously solve flow, diesel reaction and waste box combustion reaction. This methodology of solution is opted to ensure convergence. Typical spatial distribution of temperature and oxygen inside the chamber as predicted by the simulation just after the onset of box processing are shown in Fig. 5(a) and (b), respectively. Temperature profiles on the three faces of the box are shown in Fig. 5(c) for two different conditions, i.e. before onset of the surface reaction and after onset of the surface reaction.

As observed from Fig. 5(a) and 5(b),very high temperature (> 1273 K) and lowest concentration of oxygen are observed in top left corner of the chamber. This is due to rising hot gaseous mixture depleted in oxygen concentration generated in the combustion of diesel. In fact, as is clearly evident from Fig. 5 (a), the maximum temperature in the chamber is observed in the diesel jet. Temperature observed at the front face of the waste box is lower (< 423 K) because of its direct exposure to the room temperature combustion air coming into the chamber, resulting into blanketing effect. Thus, reaction is slow along the front face and so is the consumption of oxygen resulting in higher oxygen concentration in the vicinity of the front face. The top and back faces are not directly swept by the incoming air at room temperature and hence temperatures are higher at these faces (654-702 K), which leads to faster reaction which in turn leads to reduced oxygen concentration in the vicinity of these faces.

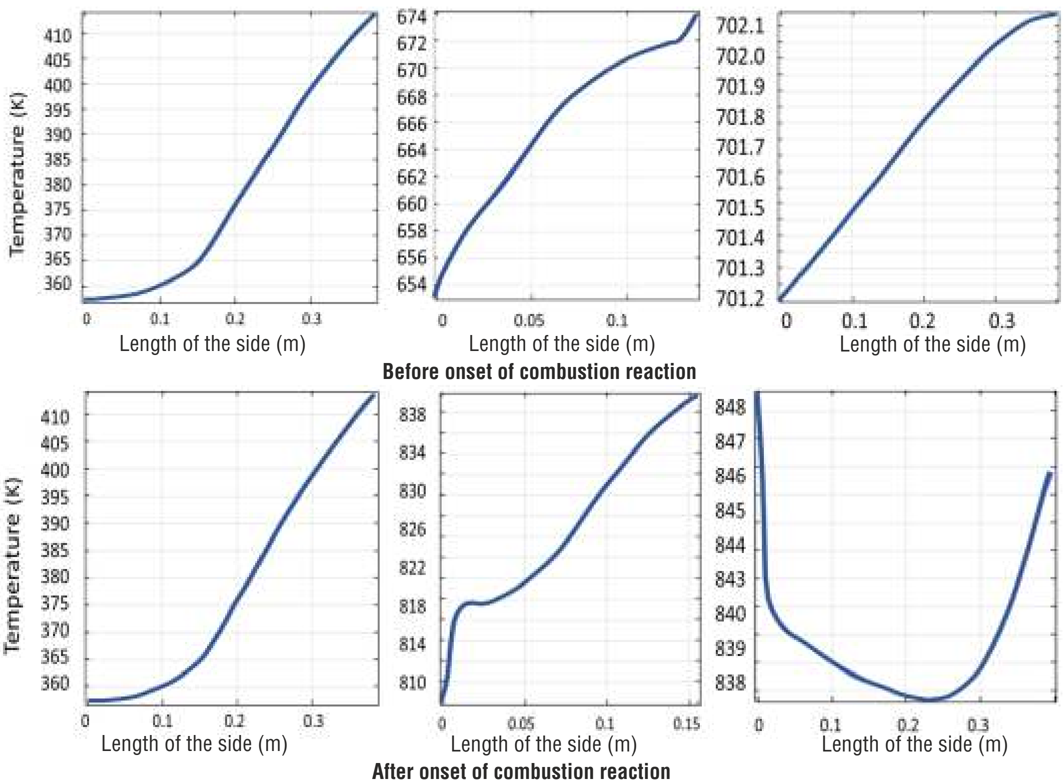
The front face sees average temperature less than 423 K throughout the process, hence even though higher concentration of oxygen is present lower temperature causes very slow decomposition at the front face compared to decomposition at other two faces which have higher temperatures. Temperatures at top and back side faces are more than 800 K during waste



(a) Temperature distribution (in Kelvin) in the chamber just after onset of waste combustion



(b) Oxygen mass fraction profile in the chamber just after onset of waste combustion



(c) Predicted temperature profiles (in Kelvin) on the surface of the waste box before and after onset of decomposition

Fig.5: Concentration and temperature profiles obtained from simulation

processing, i.e. after onset of waste combustion, giving favourable condition for further decomposition and incineration of the waste box.

The comparison of the experimental and predicted temperatures at the reference location, after the onset of decomposition reaction, was also done. The values are found to be 825 K and 840 K, respectively, showing a reasonably good match despite the several assumptions involved in the CFD model.

Conclusions

The study reports a CFD model of a diesel-fired incineration chamber used for radioactive waste processing. The CFD model has been bench-marked with the reported data of Sandia Flame D experiments. The developed model is used to gain an understanding of distribution of temperature and chemical species involved in the reaction inside the incinerator chamber. The visual observation, during waste processing in experimental run, confirm the two-stage combustion phenomenon i.e. volatilization on box surface and oxidation nearby the surface of the box. Simulation result shows that the front face of the waste box sees more oxygen concentration compared to the top and back faces. Results also show that for both before and after onset of waste combustion the front face of the box has lower temperature (<423 K), compared to other two faces (>650 K), due to cooling effect of combustion air. The temperature at the front face is very low to initiate effective combustion reaction. Hence, combustion is predominantly occurs on the top and back side faces of the waste box kept in the chamber. The temperature predicted by simulation compares well with measured temperature at a reference location during an experimental run.

The reported CFD model provides useful insights into the temperature and concentration distributions inside the chamber and will be useful for further model development to understand inherently complex process of incineration. The results of the present study will also form reference data of temperature and concentration distributions in the incinerator chamber for a similar study with plasma-based system planned for combustible waste forms.

Acknowledgement:

Authors acknowledge with gratitude constant encouragement of K. Agarwal and P.K. Wattal (former Directors of NRG, BARC) as well as K. Banerjee (former Associate Director, NRG, BARC). Authors convey heartfelt thanks to K.K. Singh (ChED) and his team for technical contribution towards successful CFD study. Authors thank all team members at SWM/WMD towards contribution in demonstration runs of the system.

Corresponding author*

K.C. Pancholi (keyur@barc.gov.in)

References

[1] L.P. Sant'ana and T. Cordeiro, "Management of radioactive waste: A review", in Proceedings of the International Academy of Ecology and Environmental Sciences, 2016, **6**, 38-43.

[2] International Atomic Energy Agency (IAEA), General Safety Guide GSG-1, "Classification of radioactive waste", Vienna, 2009.

[3] International Atomic Energy Agency (IAEA), Specific Safety Guide, SSG-40, "Predisposal management of radioactive waste from nuclear power plants and research reactors", Vienna, 2016.

[4] K. Raj, K.K. Prasad and N.K. Bansal,"Radioactive waste management practices in India", *Nuclear Engineering and Design*, April, 2006, **236**, 7–8, 914-930.

[5] P.K. Wattal,"Indian programme on radioactive waste management", *Indian Academy of Sciences*, October, 2013, **38**, Part 5, 849–857.

[6] H. Fiedler, "Dioxins and furans (PCDD/PCDF)", *Persistent Organic Pollutants - The Handbook of Environmental Chemistry*, Springer, 2003, **3**, 123-201.

[7] Incineration and Dioxins: Review of Formation Process, Consultancy report prepared by Environmental and Safety Services for Environment Australia, Commonwealth Department of the Environment and Heritage, Canberra, Available at: https://www.environment.gov.au/-protection/publications/incineration-and-dioxins-review-formation-processes 1999.

[8] International Atomic Energy Agency (IAEA),Tech Doc series no 1527,"Application of thermal technologies for processing of radioactive waste", Vienna, 2006.

[9] E.S. Prado, F.S. Miranda, L.G. de Araujo, G. Petraconi and M.R. Baldan, "Thermal plasma technology for radioactive waste treatment: a review", *Journal of Radioanalytical and Nuclear Chemistry*, 2020, 1-12.

[10] S. Nema and K. Ganeshprasad, "Plasma pyrolysis of medical waste", *Current Science*, 2002, 271-278.

[11] A.L. Sullivan and R. Ball, "Thermal decomposition and combustion chemistry of cellulosic biomass", *Journal of Atmospheric Environment,* 2012, **47**, 133-141.

[12] A. Kadar, "Modeling turbulent non-premixed combustion in industrial furnaces using the open source toolbox OpenFOAM (MS thesis)", Delft University of Technology, Netherlands, 2015.