"A CFD Modeling Study of a Bio-Diesel Blend Oil Evaporation Device Operating In the "Stabilized Cool Flame" Regime"

Keval Raval[1] Mr.Trilok D Parmar[2]
[1]P G Student [2]Assistant Professor
[1,2] Mechanical Department, LCIT-Bhandu

Abstract---An innovative modeling approach, based on the fitting parameter concept, has been developed in order to simulate cool flame reactions. The model, based on physico-chemical reasoning coupled with information from available experimental data, is implemented in the CFD code and is validated by comparing numerical predictions to experimental data obtained from feed pressure, find effect of recirculation flow in SCF device. From this work analyze experimental and simulation data, try to find good agreement with this results.

I. I INTRODUCTION

Oil fired furnaces and boilers, diesel engines and gas turbines utilize liquid fuel sprays in order to increase the fuel surface area and thus accelerate the vaporization and combustion rates. Conventional liquid fuel burning technologies inject the fuel into the combustion chamber through a nozzle that atomizes it, producing a spray comprising many droplets, typically the order of a few tens of microns in diameter. The droplets, subjected to the high temperatures of the combustion chamber, are evaporated and burnt in a sequential process. During this procedure, there may arise problems owing to the incomplete mixing of the fuel vapors with the combustion air. The separation of the two phenomena, namely evaporation and combustion, could lead to the alleviation of in homogeneities in the fuel vapour-air mixture. A satisfactory mixing of the gaseous mixture can be thus achieved before initialization of the combustion process. A novel way to accomplish such "separation" is to evaporate the fuel with the use of a process based on the "cool flame" phenomenon.

II. THE "COOL FLAME" PHENOMENON

The phenomenon described as "cool flame" is essentially a low temperature oxidation process during which the fuel is partially oxidized but not burnt [1] and it is mainly observed during the auto ignition process of hydrocarbon fuels. Whenever alkane fuels have to reside partially or fully mixed in an oxidizing atmosphere at high temperatures, ignition can occur in a multistage mode, subsequently following completely different schemes of oxidation. At temperatures below 500^0C, the complex chemical reactions involved result in a two stage ignition process in which "hot" ignition is preceded by a self-quenching temperature pulse referred to as a "cool flame" [2]. During the auto ignition process, the operating kinetic mechanisms change continuously according to the temperature of the air-fuel mixture. It is possible to define low and high temperature mechanisms, in which different oxidizing schemes are effective.

Cool flames refer to the low-temperature (500-800K) oxidative chemical activity during which a hydrocarbon fuel is partially oxidized but not burnt. A competition between termination and branching reactions arises, whenever the former exhibit higher activation energies than the latter [3, 4]. In this case, a Negative Temperature Coefficient (NTC). Region emerges, corresponding to a decrease in the overall reaction rate with increasing temperature, representing a barrier for auto ignition to occur shown in figure 1.1. By exploiting the NTC phenomenon it is possible to "stabilize" the cool flame reactions: heat losses at the system's boundaries are compensated by exothermal cool flame chemical activity and a "stable" thermo-chemical state is achieved, without ignition being observed. Experiments have shown in figure 1.1 that when SCF are realized in open flowing systems, the air-fuel mixture temperature increases up to 200K in the flow direction and essentially stabilizes at the raised level [5, 6]. No "conventional" ignition occurs when SCF operation is achieved and only 2-10% of the fuel's available thermal energy is being "consumed".

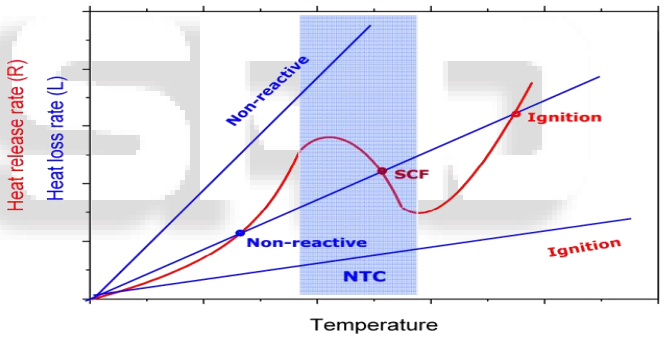

Fig. 1.1: Stabilize Cold Flame and Negative temperature coefficient [6]

III. EXPERIMENTAL SET-UP DESIGN

Experiment start with some basic instruction and preparation. Biodiesel 5% blind with diesel prepared in tank. This blend solution was heated with the help of heater. The temperature was reached to 600 ^{0}C. This heated diesel supply into the injector. Injector operated at pressure 150 bar. Diesel blends at high pressure and temperature supply into the combustion chamber. Heated air supply into the combustion chamber, this heated air produce by blower. When mixture of air and fuel supplied into the combustion chamber, fuel temperature will reach ignition point and generate flame. Flame centre temperature measure with the help of thermocouple.

The SCF reactor was fabricated by use standard dimensions. [1] This part fabricated with S.S material using standard fabrication techniques.

SCF Reactor Length (m)	0.300
External Cylinder Diameter (m)	0.108
Re – circulation tube Diameter (m)	0.080
Re – circulation Disc Diameter (m)	0.054
SCF Reactor Outlet Diameter (m)	0.025
Spray Inlet Diameter (m)	0.015
Peripheral Holes Diameter (m)	0.005

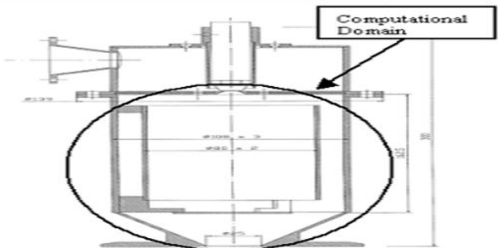

Figure. 2.1: Basic Geometric parameters of SCF reactor [1]

From the experiment, data collect on thermocouple indicator and listed below table 3.1.

Table No. 3.1: Experimental Reading

Reading	Thermocouple	Position (m)	Temperature
1	1	0.25	638
	2	0.15	852
2	1	0.25	647
	2	0.15	827

IV. Computational Fluid Dynamics Methodology

The flow in the combustion is extremely complex due to its turbulent and three dimensional natures and rapidly changing reaction of the flow. In addition, it also exhibits unsteady behavior as a result of the flam let. Considering these complexities, most of the investigators have analyzed the flow SCF reactor using numerical analysis tool Computational fluid Dynamics (CFD), which reduces the time required for the design phase by predicting performance, efficiency, flow and part of composition behavior accurately. CFD – FLUENT is used for the modeling and simulation in this project. CFD – FLUENT is computer software that allows modeling and simulation of flow of fluid and heat and mass transfer in complex geometries. It is capable to complete meshing flexibility, solving flow problems with unstructured meshes that can be generated through the complex geometries. The program structure is shown in Figure 4.1.

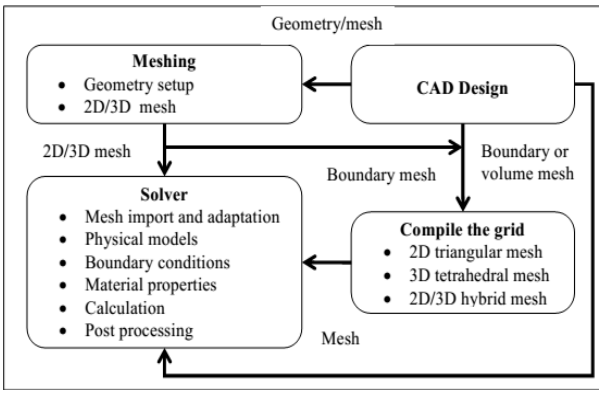

Fig. 4.1: Program Structure.[8]

In CFD analysis geometry creation is a first step to create physical domain. This domain can help for further simulation. So, difficult task to make geometry having fluid domain. Solid works Design modeler is using for geometry creation. It is shown in Figure 4.2

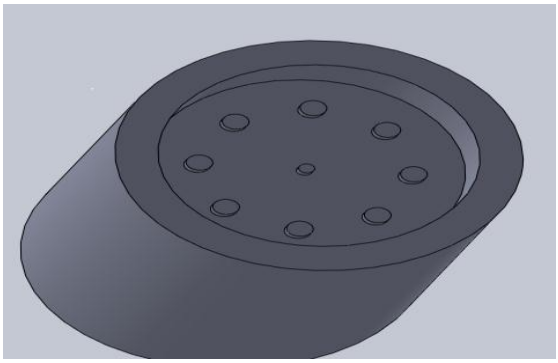

Fig. 4.2: Geometry Modeling

Meshing is a part of modeling. After complete geometry mesh will be applying on geometry. Meshing was required element size, element type and element connectivity in terms of skeewness which can be set in Ansys mesh modeler. Complete step of mesh generation not down element number and node number which can use for grid independency study. Mesh of above figure 4.2 is shown in figure number 4.3.

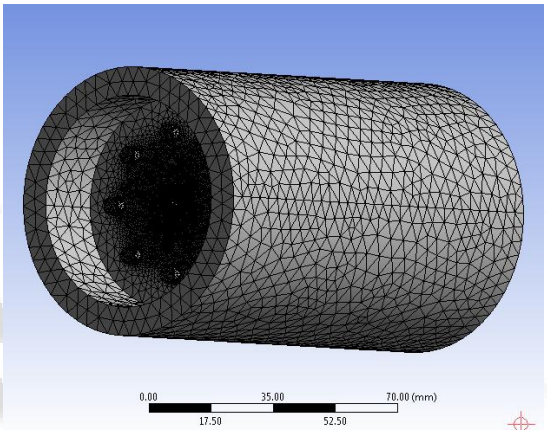

Fig. 4.3: Meshing of geometry

Boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are, therefore, a critical component of your **FLUENT** simulations and it is important that they are specified appropriately. As shown in figure 4.4.

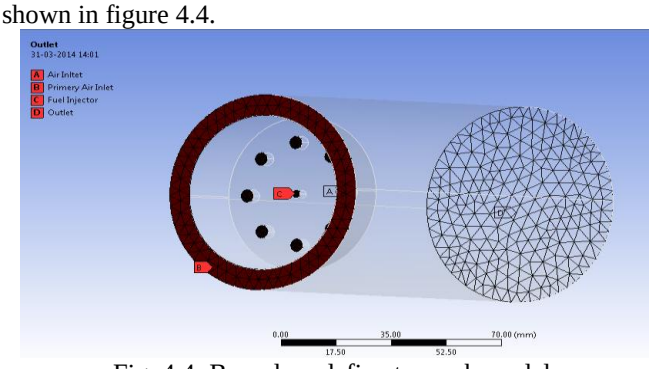

Fig. 4.4: Boundary define to mesh model

In this simulation model set as time steady and applying solver-pressure coupling method. Material- Liquid-Air at 27^0 C and Fuel inject at temperature 600^0C and Physical Model set k-epsilon turbulence with energy model along radiation with p1 model.

V. Result And Discussion

After finished simulation task in FLUENT. Its generating

data this data use for further evaluation purpose and further improvement in results.

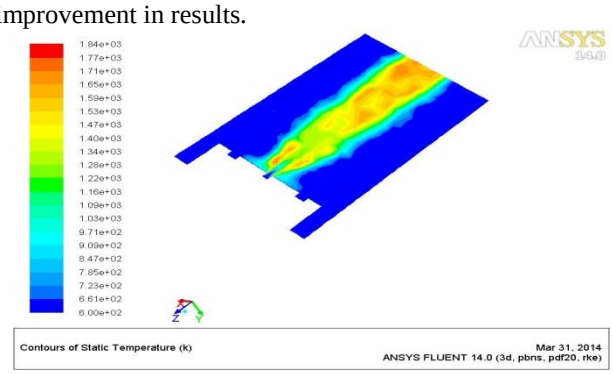

Fig. 5.1: Temperature Contour

From the contour of temperature was shown the temperature value at the symmetric plan. This contour generated from pressure 150 bars and fuel temperature 600K. CFD simulations predict the minimum temperature 600 K and maximum temperature 1840 k after combustion. This is very close to bio-diesel properties. The flame was propagated in linear direction. Shown in figure 5.1.

Above contour was presented overall view of temperature change in computational domain. This is more clearing picture of temperature plot. Graph was plotted the result from centre domain. Flame was started at temperature 600 K and highest at 825 K. which is the cold flame temperature zone. Present in temperature plot.

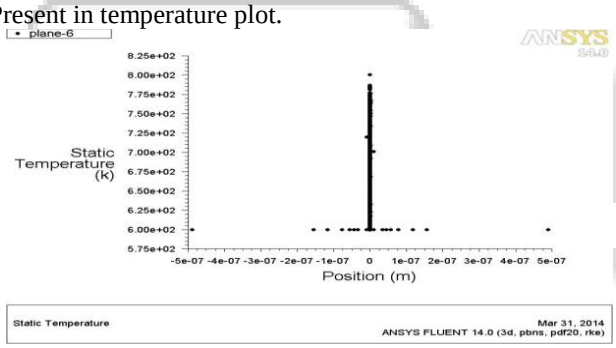

Fig. 5.2: Temperature Plot

CONCLUSION

Validation tests in a re-circulating flow SCF reactor showed that the developed computational model exhibits a cheering performance, yielding results within a satisfactory level of accuracy. As a result, they focus on the low- and intermediate-temperature oxidation phenomena. Computational results obtained by both approaches are compared with available temperature measurements along the reactor's main recirculation section and the taken as a whole agreement is satisfactory.

REFERENCES

[1] Lignola, P.G. and Reverchon, E., "Cool Flames", Prog. Energy Comb. Sci.13, 75-96, (1987).
[2] Harrison, A.J. and Cairnie, L.R., "The Development and Experimental Validation of a Mathematical Model for Predicting Hot-Surface Auto ignition Hazards Using Complex Chemistry", Comb. Fl., 71, 1-21, (1988)
[3] P.G. Lignola, E. Reverchon, Prog. Energy Combust. Sci. 13 (1) (1987) 75-96.
[4] J.F. Griffiths, Prog. Energy Combust. Sci. 21 (1) (1995) 25-107.
[5] K. Lucka, H. Koehne, 5th Int. Conference On Technologies and Combustion For A Clean Environment, Lisbon, Portugal, 1999, pp. 207-213.
[6] L. Hartmann, K. Lucka, H. Koehne, Journal of Power Sources 118 (2003) 286-297.
[7] Fluent 6.2 user's guide, Ansys Corporation. 2002.

