OPTIMIZATION OF GAS FLOW THROUGH DUCT BURNER

M. Thoufiq Ahamed* & K. Senthil Nathan**
* PG Student, Department of Mechanical Engineering, Easwari Engineering College, Ramapuram, Tamilnadu
** Assistant Professor, Department of Mechanical Engineering, Easwari Engineering College, Ramapuram, Tamilnadu

Abstract:

This project has been taken up for looking critically into the supplementary firing in the Duct Burners and improve their operational performance, thereby enhancing the overall efficiency of the Heat recovery steam generator. The problems taken up for critical evaluation are, Gas turbine exhaust gas flow distribution and mixing of these gases with the fuel being fired and also to find methods to reduce adverse impact on the HRSG operation due to these issues. The heat release after combustion and heat pick-up by the heat absorption surfaces of the major components can be significantly improved and also the higher overall efficiency of the HRSG.

Index Terms: HRSG, Duct Burner, Supplementary Firing & CFD

1. Introduction:

1.1 Heat Recovery Steam Generator:

A Heat Recovery Steam Generator or HRSG is an energy recovery heat exchanger that recovers heat from a hot gas stream. The HRSGs are generally fitted in the downstream of a Gas Turbine to recover the Turbine Exhaust Gas (TEG) heat. The HRSG is typically used to produce steam that can be used in a cogeneration process or is used to drive a steam turbine in a combined cycle system. The temperatures in TEG will vary from 350°C to 650oC depending on the class of the upstream Gas turbine. The HRSG is designed with a supplementary firing system to augment its steam output

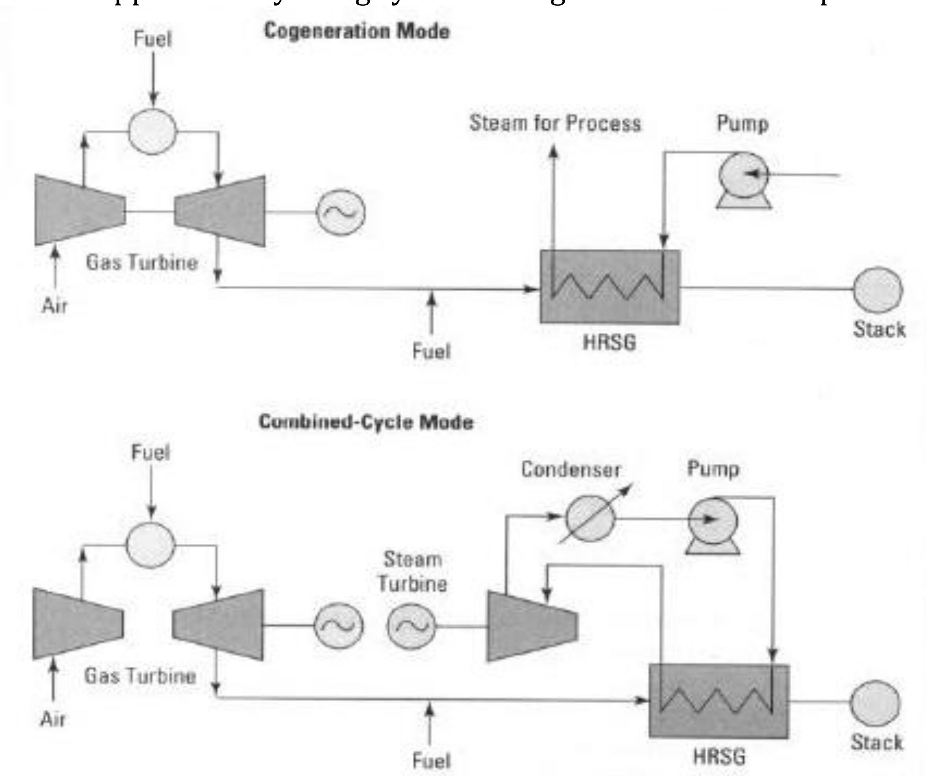


Figure 1: HRSG can be operated either in cogeneration mode and combined cycle mode

Supplementary Firing:

It may be used in combined cycles (in the HRSG) raising exhaust temperatures from about 550 °C (GT exhaust) to 900°C or even 1000 °C. The HRSG can be designed with supplementary firing of fuel after the gas turbine in order to increase the quantity or temperature of the steam generated. More fuel is sometimes added to the turbine's exhaust. This is possible because the turbine exhaust gas (flue gas) still contains some oxygen. Temperature limits at the gas turbine inlet force the turbine to use excess air, above the optimal stoichiometric ratio to burn the fuel

1.2 Computational Fluid Dynamics:

Computational fluid dynamics is a branch of fluid dynamics a cost–effective means of simulating flow by the numerical solution of the governing equations. The governing equations for Newtonian fluid dynamics, namely the Navier-strokes equation, have been known for over 150 years. However, the development of reduced forms of these equations is still an active area research in particular turbulent closure problem of the Reynolds – averaged Navier-stroke equation.

Experimental methods has played as an important role in validating and exploring the limits of the various approximation to the governing equations, particularly wind tunnel and rig tests that provide a cost effective alternative to full scale testing. The flow governing equations are extremely complicated such that analytic solutions cannot be applicable for most practical applications. Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions.

With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

1.3 Duct Burner:

Duct burner is an arrangement in the HRSG through which the Gas Turbine Exhaust (GTE) is further heated to improve the efficiency of steam generation in HRSG. The GTE temperature will be raised from 400-450°C to 800-850°C. Burners are stacked in multiple elevations and fitted in a single large frame (log type) and this assembly is mounted on the HRSG inlet duct end. This Duct Burner will help in increasing the inlet GTE's temperature from 550-600°C to 850-900°C. The GTE will have 10-14% of inherent O2 in it. This oxygen will assist in the combustion of the fuel that is being fired in the Duct burners.

Duct burners use supplementary firing to increase the heat energy of a gas turbine's exhaust(GTE),making it possible to increase the output of a downstream heat-recovery steam generator (HRSG). Grid type systems were designed to reduce the pressure drop and spread the heat out across the duct. The grid design uses an array of fuel manifolds (called runners) to deliver the fuel into the GTE stream and a diffuser/bluff body attachment to stabilize the flame.

As gas turbine (GT) technology advanced, mass flow rates and exit temperatures rose along with combustion efficiency and capacity ratings. In response to the increases, the cross-sectional area of HRSGs grew to increase the tube surface area available for heat recovery and to keep exhaust pressures low enough to maintain the GT's efficiency.

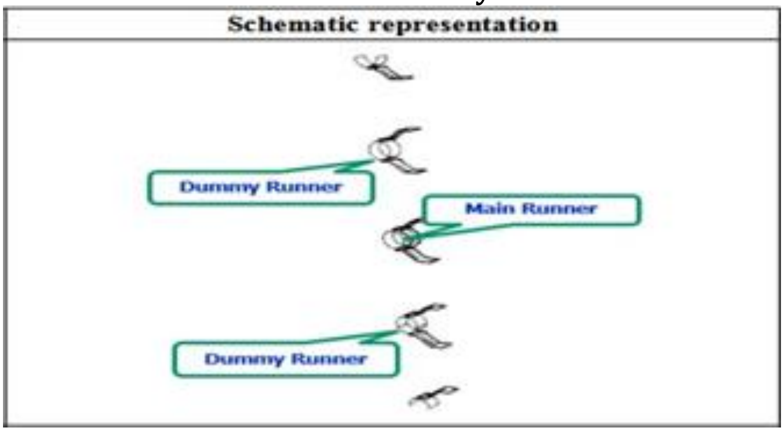
For nearly all duct burners, the GT is the source of combustion air. Accordingly, burner performance depends largely on the flow and composition of GTE gas. In addition to affecting the burner's combustion efficiency, the flow of GTE gas partially determines its structure. Advancements in GT technology have imposed the following conditions on modern duct burners:

- ✓ Higher mass flow rates (from larger turbines).
- ✓ Lower GTE oxygen levels (due to higher turbine efficiency).
- ✓ Higher inlet temperatures (due to higher turbine outlet temperatures).

The turbine's application also has a bearing on duct burner design. For example, if a GT drives a generator, its rotational speed is kept constant to maintain a constant generator frequency. But even at this fixed speed, the turbine's exhaust mass flow rate may change with ambient air density, which is a function of temperature and humidity. Such changes can be minimized by conditioning the air at the GT's inlet.

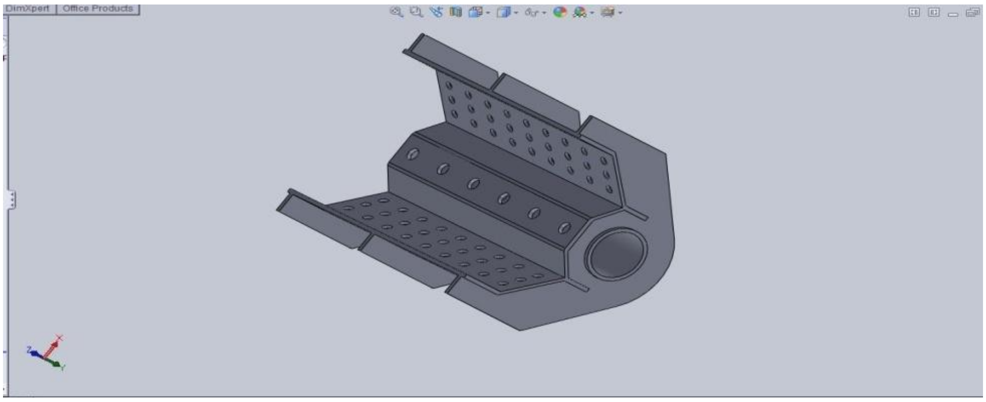
Ideally, the designer of a duct burner would have access to the target turbine's exact conditions entering the plane of the burner for the purpose of optimizing the burner's design. The first step in the burner design process is to understand that GTE mass flow is a function of GT load. Usually, the mass flow drops proportionally until about 50% load and then increases. Given modern plant permit levels and operational conditions, the GT is typically not run at much lower loads.

Schematic Representation of Active and Dummy Runner:

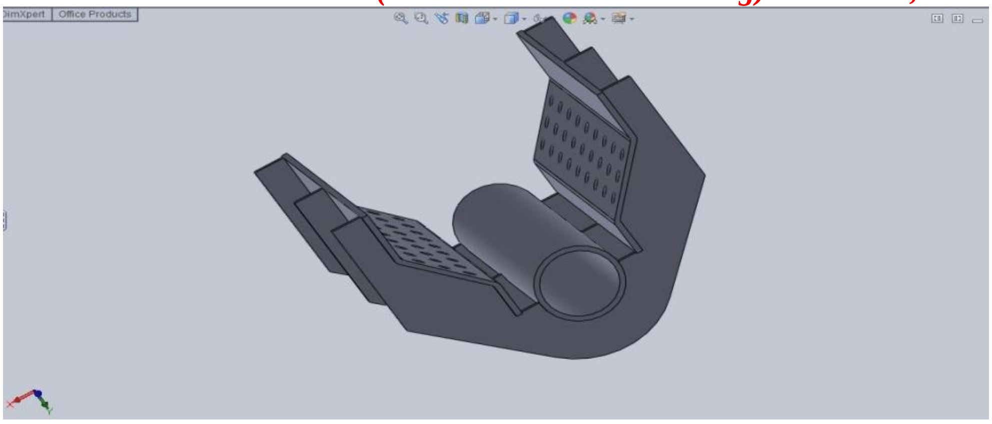


2. Modelling:

The model of duct burners with and without dummy runners was deisgned using by solid works software



(a)



(b)

Figure 2: Solid Works Model of a Runner; (A) Active Runner, (B) Dummy Runner

So, Figure 2 shows the solid work model of active and dummy runner.

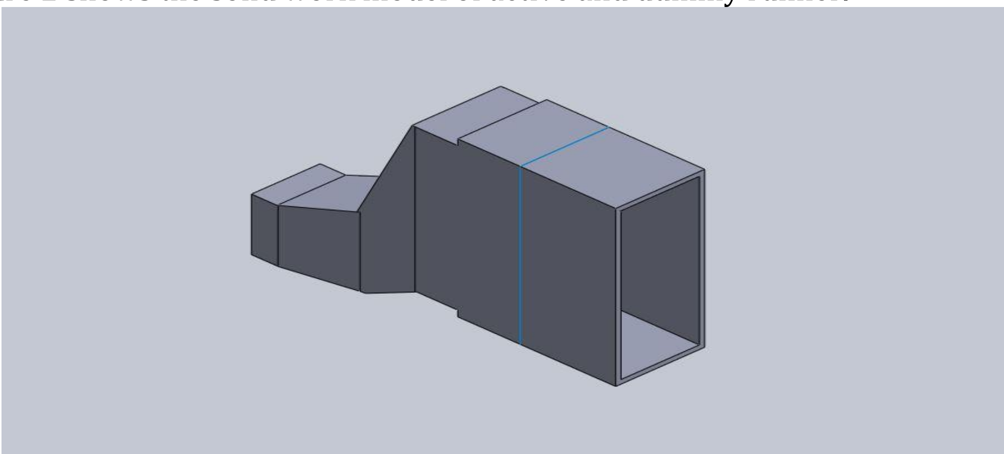


Figure 3**: S**olid works model of duct burner

The solid works model of duct burner is shown in Fig. 3.

3. Elements and Meshing:

The representative geometry of the duct burner is modeled for the two configurations in ansys with solver as Fluent. Meshing of the volumes was done with respect to face by providing size functions from the finer meshes of faces. The active gas runner with gas injection nozzles that had critically the lowest dimension in the entire model is meshed in a finer manner. As the accuracy of the fuel mass flow through the gas hole is of highest concern, hexahedral mesh is selected for this. The hexahedron with 8 vertices, 12 edges and bound by 6 quadrilateral faces is the element. The density of the mesh is sufficiently high in order to capture all the flow features. Hex/wedge element of Cooper type is used for this mesh with Interval size .The dummy runners along with HRSG gas path are less critical compared to active runner nozzle. The faces of these are meshed and size function to mesh these volumes with tetrahedron was selected. The tetrahedron with 4 vertices, 6 edges and bound by 4 triangular faces is the element. Tet/hybrid element of T-grid type is used for this mesh with Interval size of one as shown in Figure 6.The representative elements of tetrahedron and hexahedron that are used in the meshes are indicated in Figure 4. The mesh skewness and grid independent test has been checked for the model in both configurations. For Case -1 the mesh has 4155786 elements and 877621 nodes. For Case -2 the mesh has 4991465 elements and 9823418 nodes.

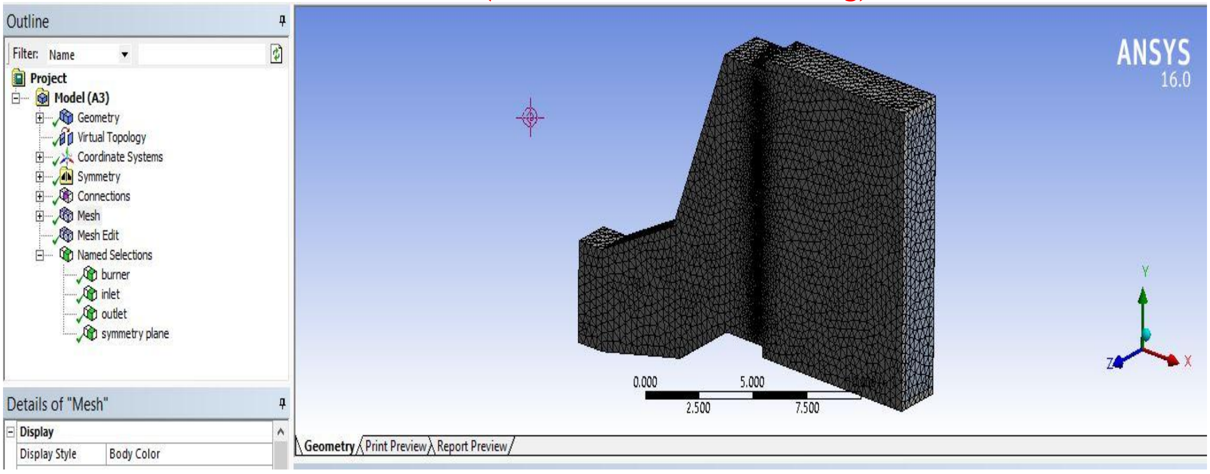


Figure 4: Mesh view of duct burner

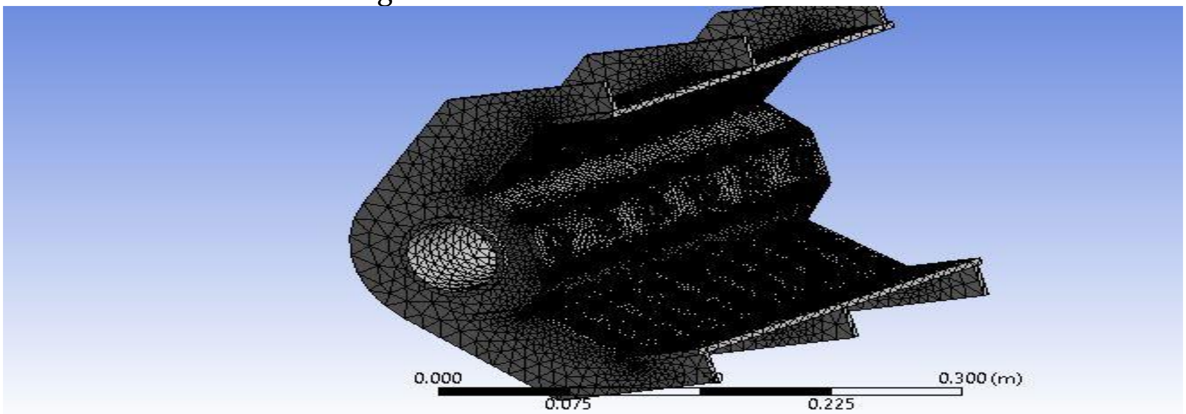


Figure 4.1: Mesh view of active runner

Above Figure 4 & 4.1 Shows the mesh view of duct burner and active runners

3.1 Boundary Condition and Continuum:

Natural Gas as combustion fuel is injected out through the active runner gas pipe and this pipe inlet is selected as 'Mass flow Inlet'. The fuel injection hole in the gas pipe is selected as 'Interior'. The Gas turbine Exhaust flowing around the natural gas burners is selected as 'Velocity Inlet' at the HRSG duct model inlet. All the area surrounding the representative HRSG duct model comprising the active runners, dummy runners and block plate are selected as 'Symmetry'. Symmetry type of condition is used for taking values of properties just adjacent to the solution domain as values at the nearest node just inside the domain thereby the model mesh size is reduced by not requiring to simulate for total cross section. The runner pipe surfaces, the flame stabilizer surfaces and the block plate surfaces are selected as 'Wall' which acts as no participation medium in the flow model. The continuum inside the active runner gas pipe is given as gas for modeling Natural Gas combustion. The continuum inside the HRSG chamber is given as Air for modeling the TEG oxygen for combustion.

4. Computational Fluid Dynamic Analysis in Fluent:

The processing of the model is done in Ansys FLUENT [6-7]. The Fluent processor divides the processing to pre-processing and post-processing and details on the same are elaborated below.

4.1 Pre-Processing of the Model:

The solver mainly utilizes the Navier-Stokes equations which is the basic governing equations for a viscous, heat conducting fluid. It is a vector equation obtained by applying Newton's Law of Motion to a fluid element and is also called the momentum equation. It is supplemented by the mass conservation equation, also called continuity equation and the energy equation. Usually, the term Navier-Stokes equations is used to refer to all of these equations. The following are the details on models selected for combustion flame study. Energy equation for the model is activated to model the fluid flow with along with the associated thermal properties. Further, k-ε – 2 equation model is used to capture the turbulence in natural gas and turbine exhaust gas flow patterns. To capture the combustion of methane in natural gas, species transport is modeled using Eddy- dissipation model.

The Eddy-dissipation model captures the Turbulence-Chemistry interaction for the volumetric reactions. Most fuels are fast burning and the overall rate of reaction is controlled by turbulence mixing. In the non-premixed flames, turbulence slowly mixes the fuel and oxidizer into the reaction zones where they burn quickly. In premixed flames the turbulence slowly mixes cold reactants and hot products into the reaction zones where reaction occurs rapidly. In such cases the combustion is said to be mixing-limited, and the complex and often unknown chemical kinetics can be safely neglected. In this model, the chemical reaction is governed by large eddy mixing time scale. Combustion initiates whenever there is turbulence present in the flow. It does not need an ignition source to initiate the combustion. As the combustion of methane and ethane is non-premixed, the chemical species will burn at the moment they enter in the computation model. The model accommodates inlet diffusion and the diffusion energy source by using the stoichiometric chemical coefficients and rate exponents of methane and ethane reactions with oxygen as indicated in equations. The material and fluids are governed by methane-air and ethane-air mixture in case of Natural gas and in case of Turbine Exhaust Gas, it is air with molecular composition and fluid properties of flue gas.

For the total duct burner cross section, by virtue of the symmetry boundary condition, the total mass flow rate of natural gas being modeled in combustion is 1.6 kg/s. Solution method for the model follows an iterative procedure where the pressure-velocity coupling of the Navier-Stokes equation is done by Semi-Implicit (SIMPLE) algorithm. The spatial discretization for Gradient and Pressure are Green-Gauss cell based and Standard respectively. For the other parameters namely momentum, turbulent kinetic energy, turbulent dissipation rate, Energy and species of methane, ethane, oxygen, carbon-di-oxide, water vapor and nitrogen, first order upwind is used. The boundary conditions of Natural gas and TEG are as indicated below

Boundary Condition:

FLUID TYPE - Turbine exhaust gas flow
VALUE - 3.46 m/s
TEMPERTURE - 784 K

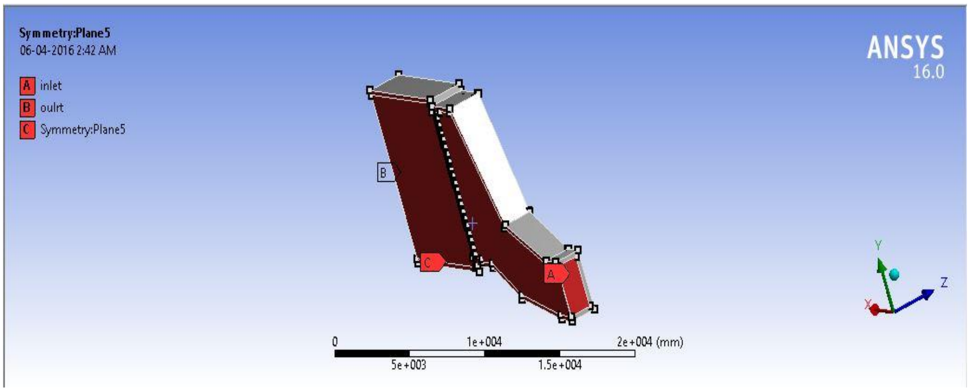


Figure 5: View of duct burner with boundary condition for cfd analysis

5. Conclusion:

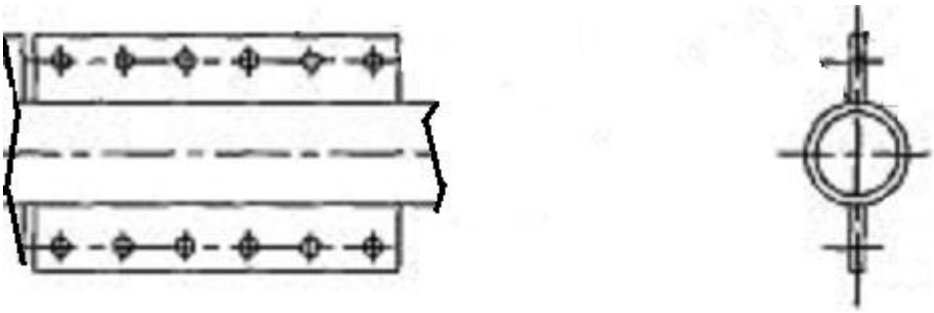
5.1 Flow Behaviour Without Dummy Runner: Flow analysis with only the gas runners in place and by taking out the dummy runner, was done. The flow pattern was non-uniform and haphazard across the burner section. It is implied that by this arrangement, the GTE flow may not contribute for effective combustion of the fuel due to non-uniform flow across the section and no proper concentration of O2 at the gas runner for effective burning of the fuel gas.

5.2 Flow Behaviour With Dummy Runner: Flow analysis with the existing arrangement is done. The flow pattern was uniform across the burner section. It is implied that by this arrangement, the GTE flow was steady with effective turbulence. But the proper mixing of GTE with the fuel gas to contribute for the concentration of O2 to its maximum extent is expected to be non-satisfactory. This will lead to carry-over of some amount of gas along with the GTE flow with burning completely. As a result, more CO is emitted and black smoke be visible at the chimney outlet.

A closer view of the existing dummy runner (part) is shown below.

The dummy runner element is so configured that the GTE flow is steady and uniform. But this configuration may not contribute for the thorough mixing of GTE with the gas, so that complete combustion takes place.

5.3 Flow Behaviour With Dummy Runner of Different Configuration: In order to have steady and uniform GTE flow with good concentration of O_2 with the fuel gas, dummy runner of different configurations were thought of. One of the configurations is shown below



The analysis done with the above configuration showed improvement in mixing pattern of the GTE with the fuel gas, so that better O2 concentration is achieved

6. Future Scope:

From the analysis done, it is inferred that by designing such a configuration of the dummy runner, the flowing conditions can be properly set to achieve

- ✓ uniform and steady flow of GTE
- ✓ Thorough mixing of GTE with the fuel gas
- ✓ Maximum heat generation takes place on combustion of the fuel
- ✓ Maximum heat pick-up in the heat absorption components is effected
- ✓ Maximum Steam output is got

Hence, there is scope for further analysis with different dummy runner configurations to achieve the above benefits and enhance the duct burner performance.

7. References:

1. Afsaneh Soleimani, Simon Schneiderbauer, Stefan Pirker (2015) 'CFD Study of the Gas particle Flow in a Horizontal Duct: The Impact of the Solids Wall Boundary Conditions Procedia Engineering', Volume 102, 2015.
2. Amin Mohammadi Khoshkar Vandani, Mokhtar Bidi, Fatemeh Ahmadi (2015) 'Exergy analysis and evolutionary optimization of boiler blow down heat recovery in steam power plant', Energy Conversion and Management Volume 106.
3. Allouis. C, Amoresano. M, Langella. G, Niola. V, Quaremba. G (2015) 'Characterization of gas turbine burner instabilities by wavelet analysis of infrared images Experimental Thermal and Fluid Science', In Press, Corrected Proof.
4. Emad Ghadirian, Hamid Arastoopour (2015) 'CFD simulation of a fluidized bed using the EMMS approach for the gas-solid drag force Powder Technology', Volume 288.
5. Mahmoud Nadir, Adel Ghenaiet (2015) 'Thermodynamic optimization of several (heat recovery steam generator) HRSG configurations for a range of exhaust gas temperatures Energy', Volume 86.
6. Marco Cavazzuti, Elia Agnani, Mauro A. Corticelli (2015) 'Optimization of a finned concentric pipes heat exchanger for industrial recuperative burners Applied Thermal Engineering'.
7. Pedram Hanafizadeh, Sajjad Falahatkar, Pouria Ahmadi, Mojtaba Mirzakhani Siahkalroudi (2015) 'A novel method for inlet duct geometry improvement of heat recovery steam generators', Applied Thermal Engineering, Volume 89.
8. Sheng Chen, Yiping Fan, Zihan Yan, Wei Wang, ChunxiLu (2015) CFD simulation of gas–solid two-phase flow and mixing in a FCC riser with feedstock injection Powder Technology', Volume 287.
9. Tian-Hu Zhang, Feng-Guo Liu, Xue-Yi You(2014) 'Optimization of gas mixing system of premixed burner based on CFD analysis Energy Conversion and Management', Volume 85, September 2014.
10. Vít Kermes, PetrBe˘lohradský, Jaroslav Oral, Petr Stehlík' Testing of gas and liquid fuel burners for power and process industries Energy', Volume 33, Issue 10.