iJRASET
International Journal For Research in
Applied Science and Engineering Technology

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

**Volume:** 2 **Issue:** IX **Month of publication:** September 2014

**DOI:**

ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

# Analysis of Flue Gas Flow Behavior in Economiser Duct Using Cfd

Anand Kumar S Malipatil[1], Shivaraj[2]

*Department of Thermal Power Engineering, VTU-Gulbarga*

***Abstract--*** ***Energy saving and efficiency are the key issues in power generation system not only from the View point of fuel consumptions, but also for the protection of global environment .flue gas Ducts are the major parts of oil-fired power plant, which are used to exhaust flue gases from Boiler. This work presents an approach for CFD analysis of economizer duct design. The aim Of this work is to analysis and compares the new economizer duct design with traditional Strategies. The most economical solution of this problem seems to distribute gas flow uniformly At inlet of economizer by using vanes in economizer duct, so that effective heat transfer can be Obtained .In the present work commercial software ansys is used for the 3D simulation using its inbuilt K-e Reliable model. Optimization of economiser is done for effective heat transfer with reducing number of tubes required.***

***Keywords: K-Ɛ model, Simulation, Mesh generation, Contours, Vanes***

## I. INTRODUCTION

Economizer, an economizer is a heat exchanger which raises the temperature of the feed water leaving the highest pressure feed water heater to about the saturation temperature corresponding to the boiler pressure. This is done by the hot flue gases exiting the last super heater. The successful design of economizer was invented by Edward Green in 1845..In coal fired power plants the boiler is fitted with economizer and air preheater to recover heat from the flue gases. An increase of about 20% in boiler efficiency is achieved by providing both economizer and air pre-heaters. Providing economizer alone gives only 8% efficiency increases. The Economiser enables the boiler to operate at a much higher rate of efficiency with the result of lowering the fuel usage requirements. This is due to the capture and recycling of waste heat. The recovered heat is used to increase the temperature of the boiler feed water, which in turn reduces the amount of additional heat required for steam production. This has a direct impact by reducing the fuel requirement, hence reducing the running cost. An added benefit is the lowered environmental impact linked to reducing the amount of carbon and the volume of flue gases entering the atmosphere via the boiler chimney. Hence there is need for analysis of economiser to increase the effiency of boiler and also reduce emissions. In coal-fired boilers, an accumulation of fly ash particles will result in a certain amount of metal erosion on economizer tubes.

Unit operation much above maximum continuous rating and design excess air also encourages the process of fly ash erosion. Any reduction of flow area, such as localized fly ash plugging will increase gas velocity and erosion potential by concentrating fly ash in an area adjacent to the plugged area. A regular program of washing during out-of-service periods should be part of normal maintenance procedures. Economizer tube ruptures require immediate attention. Delayed repairs can result in steam damaging adjacent tubes, turning a minor repair job into a major one. Economizer ruptures, left unattended, can also lead to plugging of the economiser and air heater from the water mixing with the fly ash, a mixture that can set as hard as concrete. Another precaution associated with economizer operation is the accumulation of steam in the economizer during the period when pressure is rising. During this period there is no feed water flow through the economizer; nevertheless, even with the economisers location in relatively low pressure zones, steam is generated and becomes trapped and remains so until feed water flows through the economizer. This makes the control of Steam-drum water level difficult and causes water hammer. This can be overcome by supplying feed water constantly, venting the steam out of the economizer or by recirculating boiler water through the economizer.

In 1986 A.P.Mann et al, have analyzed Computational fluid dynamics (CFD) has been successfully used in a number of cases to reduce the operating and maintenance costs

Vol. 2 Issue IX, September 2014

ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

associated with erosion of convection bank tubes and erosion and corrosion of air heater tubes. S.K.DAS et al in 2006 reports Fly ash particles entrained in the flue gas from boiler furnaces in coal-fired power stations can cause serious erosive wear on steel surfaces along the flow path. A.D.Patil et al, in 2011 have developed a methodology which finds optimization of economizer design presented an approach for the optimization of economizer design with finned and bare tube economizer After analyzing with CFD, they have concluded that the most common and reliable economizer is the bare tube. Deendayal yadav et al in 2011 have found that the more tube leakage in power plant found in economizer due to the flue gases using CFD tool by analyzing they found that arrester are useful in reducing erosion rate. Yong SUN et al in 2011 have analyzed that the economizer is a critical component in coal fired power plants. Neihad AI-Rhalidy has focused on a design modification to a number of industrial ducts using CFD analysis considering all flow features relating to the duct system efficiency.

By literature survey, it is found that many have done experiment and analysis on economizer tube failure, its material property and failure and economics of economizer but there are less researchers who have worked on optimal design of economizer. Hence there is need for analysis of economizer to extracting maximum heat from flue gases in order to reduce number of tubes by increasing the feed water temperature.

1. To carry out the CFD simulation of economizer to evaluate flue gas flow behavior on tubes with and without vanes at its inlet.
2. Comparing modified economizer with existing and predicting the outlet temperature of water.

## II. GOVERNING EQUATIONS IN CFD

There are mainly three equations we solve in computational fluid dynamics problem. They are Continuity equation, Momentum equation (Navier Stokes equation) and Energy equation.

Continuity equations often can be expressed in either integral or differential form as shown below.

$$\int_{\sigma} \rho V\, dA + \frac{\partial}{\partial t}\int_{\Omega} \rho\, dA = 0$$

This is a statement of the principle of mass conservation for a steady, one-dimensional
flow, with one inlet and one outlet.

$$\nabla(\rho V) + \frac{\partial \rho}{\partial t} = 0$$

$$\text{Where } \nabla = \frac{\partial}{\partial x} i + \frac{\partial}{\partial y} j + \frac{\partial}{\partial z} k$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho.u)}{\partial x} + \frac{\partial(\rho.v)}{\partial y} + \frac{\partial(\rho.w)}{\partial z} = 0$$

There are thus three different momentum equations that together comprise the Navier-Stokes Equations.

$$\rho\frac{\partial u}{\partial t} + \rho u\frac{\partial u}{\partial x} + \rho v\frac{\partial u}{\partial y} + \rho w\frac{\partial u}{\partial z} = \rho g_x - \frac{\partial p}{\partial x} + \mu\frac{\partial^2 u}{\partial x^2} + \mu\frac{\partial^2 u}{\partial y^2} + \mu\frac{\partial^2 u}{\partial z^2}$$

$$\rho\frac{\partial v}{\partial t} + \rho u\frac{\partial v}{\partial x} + \rho v\frac{\partial v}{\partial y} + \rho w\frac{\partial v}{\partial z} = \rho g_y - \frac{\partial p}{\partial y} + \mu\frac{\partial^2 v}{\partial x^2} + \mu\frac{\partial^2 v}{\partial y^2} + \mu\frac{\partial^2 v}{\partial z^2}$$

$$\rho\frac{\partial w}{\partial t} + \rho u\frac{\partial w}{\partial x} + \rho v\frac{\partial w}{\partial y} + \rho w\frac{\partial w}{\partial z} = \rho g_z - \frac{\partial p}{\partial z} + \mu\frac{\partial^2 w}{\partial x^2} + \mu\frac{\partial^2 w}{\partial y^2} + \mu\frac{\partial^2 w}{\partial z^2}$$

This expression of the energy equation is valid for most applications.

$$\frac{\partial}{\partial t}\left(\rho e + \frac{1}{2}\rho v^2\right) + \frac{\partial}{\partial x}\left(\rho u e + \frac{1}{2}\rho u v^2\right) + \frac{\partial}{\partial y}\left(\rho v e + \frac{1}{2}\rho v v^2\right) + \frac{\partial}{\partial z}\left(\rho w e + \frac{1}{2}\rho w v^2\right) =$$

$$k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial xz^2}\right) - \left(u\frac{\partial p}{\partial x} + v\frac{\partial p}{\partial y} + w\frac{\partial p}{\partial z}\right) +$$

$$\mu\left[u\frac{\partial^2 u}{\partial x^2} + \frac{\partial}{\partial x}\left(v\frac{\partial v}{\partial x} + w\frac{\partial w}{\partial x}\right) + v\frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial y}\left(u\frac{\partial u}{\partial y} + w\frac{\partial w}{\partial y}\right) + w\frac{\partial^2 u}{\partial z^2} + \frac{\partial}{\partial z}\left(u\frac{\partial u}{\partial z} + v\frac{\partial v}{\partial z}\right)\right]$$

$$+2\mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial u}{\partial y}\frac{\partial v}{\partial x} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial v}{\partial z}\frac{\partial w}{\partial y} + \frac{\partial^2 w}{\partial z^2} + \frac{\partial w}{\partial x}\frac{\partial u}{\partial z}\right] + \rho u g_x + \rho v g_y + \rho w g_z$$

The solver chosen for this analysis is pressure-based unsteady solver.

## III. METHODOLOGY

Initial Thinking

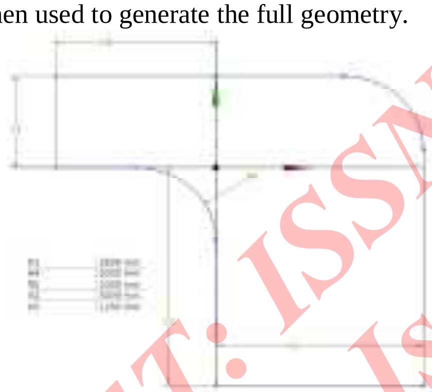
It is very important to understand as much as possible about the problem being simulated in order to accurately define it. This stage involves collecting all the necessary data required for the simulation including geometry details, fluid properties, flow specifications, boundary conditions and initial conditions.

Pre-processor

ISSN: 2321-9653



Build geometry: Depending on whether the problem geometry is one, two or three dimensional, the geometry consists of the flow domain is created using ANSYS Design Modeller.
Define materials: A material is defined by its material constants. Every volume has to be assigned a particular material.
Mesh generation: In this stage, the continuous space of the flow domain is divided into sufficiently small discrete cells. The distribution of which determines the positions, where the flow variable are to be calculated and stored. The variable gradients
are generally more accurately calculated on a fine mesh than on a coarse one. A fine mesh is therefore particularly important in regions where large variations in the flow
variable are expected.
Flow specification: The solver is a specialized programme that solves the numerical
equations based on the data specified in the data file. The results obtained by the solver are written to a result file for examination using the post-processor software.
CFD–solver: All the data defined in the pre-processing steps are fed into the solver 2-D sketches are first drawn and 3-D tools are then used to generate the full geometry.

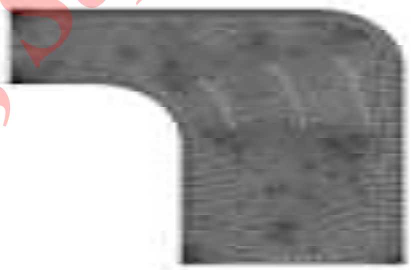


Geometric model of economizer baseline duct

CFD model of economizer baseline duct

CFD meshing of economizer baseline duct

CFD model of economizer duct with straight vanes



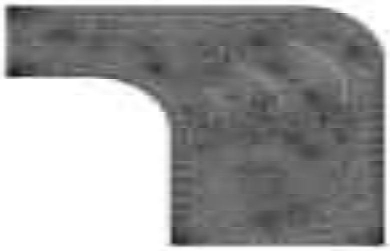
CFD meshing of economizer duct with straight vanes

CFD model of economizer duct with curved vanes

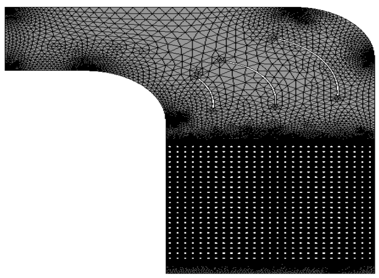
Vol. 2 Issue IX, September 2014

ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)



CFD meshing of economizer duct with curved vanes



CFD meshing of economizer with curved vanes

## IV. BOUNDARY CONDITIONS

Boundary conditions economizer.

| | |
|---|---|
| Flue gas flow | 15.62 kg/s |
| Economizer inlet flue gas temperature | 337 °c |
| Economizer inlet feed water temperature | 120 °c |
| Feed water flow | 6.21 kg/s |

| | |
|---|---|
| Flue Gas outlet pressure | 0 gauge |
| Flue Gas inlet Velocity | 12 m/s |
| Economiser tube wall | No slip &escape |
| Water outlet pressure | 0 Gauge |

The walls are assumed to be adiabatic. In his analysis tube walls are maintained at constant temperature of 120 °c to reduce the complexity of the problem because the total number of 572 tubes needs to be meshed, this will results larger domain mesh size.

Inputs

A CFD simulation for a set of reference input data is performed in this phase.

3.4.1 Flow Configuration

Air is assumed as incompressible gas model. Since air temperature is not going to vary much, constant viscosity/conductivity will be used for air. Turbulence will be modeled using the RNG k-epsilon two-equation model suitable for swirling flows.

Geometry of economizer

Table :Geometry of economizer

| Geometry of Economizer: | |
|---|---|
| Tube | 38.1 mm |
| Tube thickness | 3.66 mm |
| Longitudinal Pitch | 100 mm |
| Number of tubes wide | 26 |
| Number of tubes deep | 22 |

Material Properties

The following flue gas properties have been used.

Table :Flue gas Properties

| **Property** | **Fluid (Flue gas)** |
|---|---|
| Mass flow rate | 15.62 kg/s |
| Specific heat kJ/kg K | 1.12 |
| Density (kg/m$^3$) | 1.337 |
| Viscosity kg/mh | 0.101 |
| Thermal Conductivity kW/mK | 0.00046 |

The following economizer tube flow properties have been used.

Table: economizer tube flow properties

| **Property** | **Material (Carbon steel)** |
|---|---|
| Mass flow rate | 6.31 kg/s |
| Density (kg/m$^3$) | 913 |
| Specific heat kJ/kg K | 0.42 |
| Thermal Conductivity | 1 kcal/m-hr-°c |

## V. RESULT AND DISCUSSIONS

### 5.1 Introduction

ISSN: 2321-9653

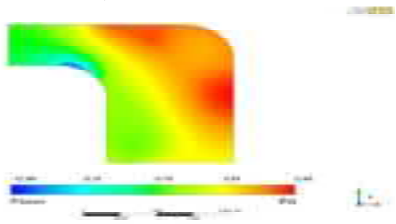
INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

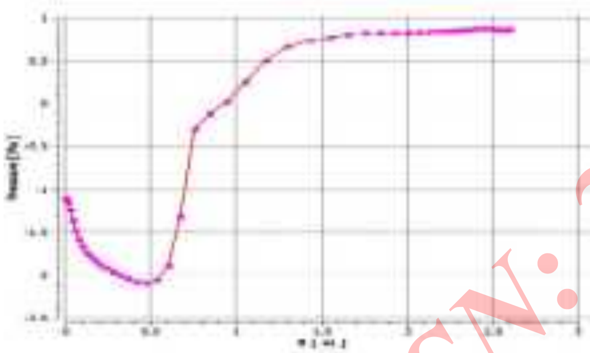
Previous chapter reviews that all models have been created in ANSYS and boundary conditions are applied. In this chapter analysis have been disused.

5.2 Results

5.2.1 CFD simulation of economizer baseline duct

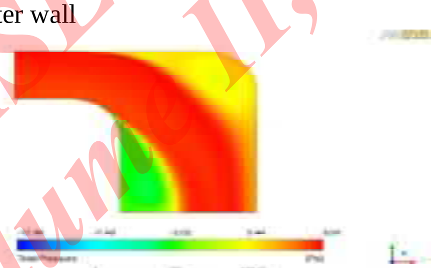
Simulation is done on existing economizer inlet duct with fillet, without vanes and results of pressure contour, total pressure contour; velocity contour and turbulence contour are presented in figures below.

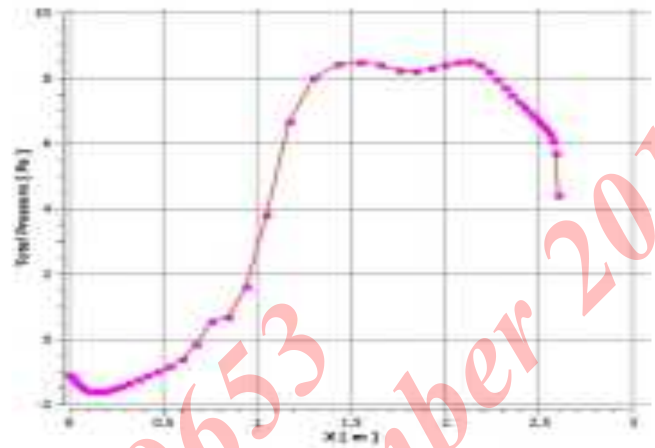




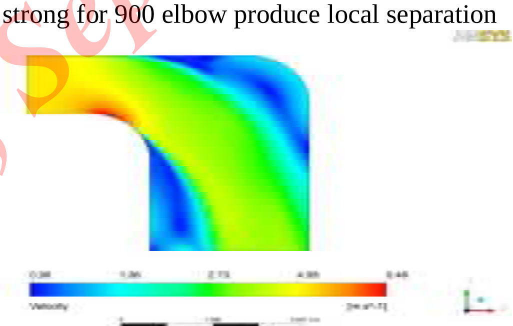
Above figure shows the pressure contours at baseline 90 degree elbow duct. As the fluid flows through the straight pipe and then enter into the elbow section, the pressure which is uniform across the flow in the straight section, must adjust in the elbow to

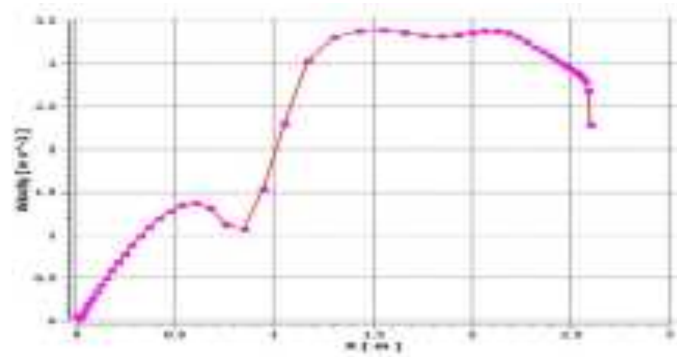
counter the centrifugal force. The pressure is greatest at the outer wall furthest from the center of curvature and least at the inner wall nearest to the centre of curvature. At the inlet of the elbow a low pressure exists in the inner wall and high pressure exists at the outer wall





Above figure shows the initial pressure gradient resulting from the change from straight to curve flow, a cross stream pressure gradient exists in the elbow, at the elbow inlet the boundary layer on the outer wall experiences the effect of an the adverse stream wise pressure gradient which may be sufficiently strong for 900 elbow produce local separation

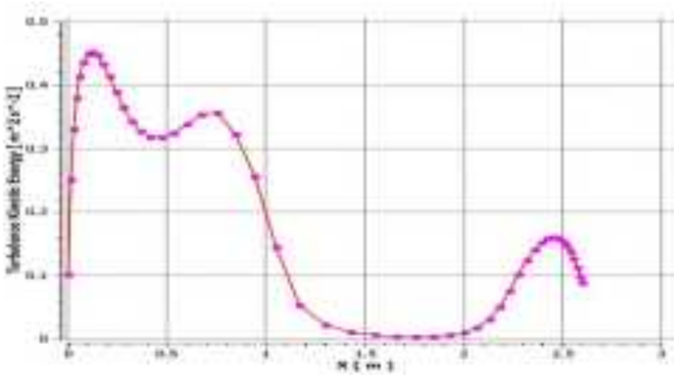




From above figure it shows, as fluid flows through the straight pipe the velocity is uniform and the enters into elbow section the velocity increases near to the inner wall because of curvature.

ISSN: 2321-9653

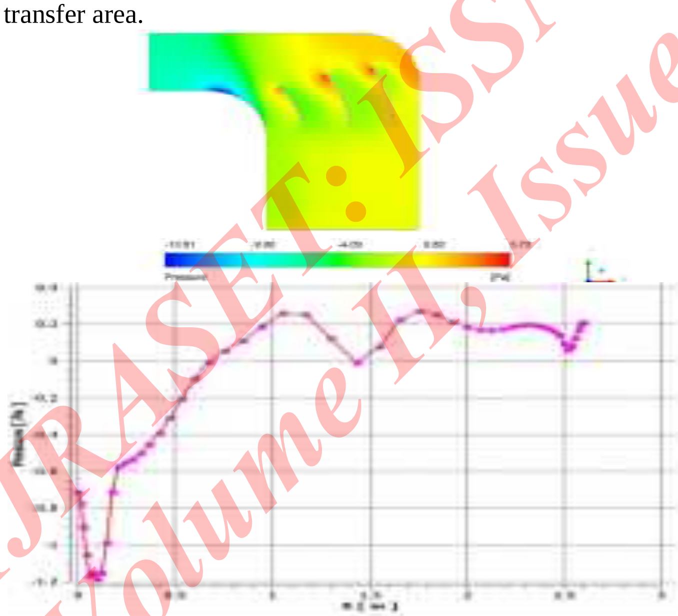




Above figure shows at the inner and outer wall higher turbulence created due to an reverse flow in the duct and the reverse occurs at the exit of the elbow where local pressure gradients of the opposite sign appear as the flow adjust to uniform pressure condition of the downstream.

5.2.2 CFD Simulation of economizer duct with Straight inlet guide vanes

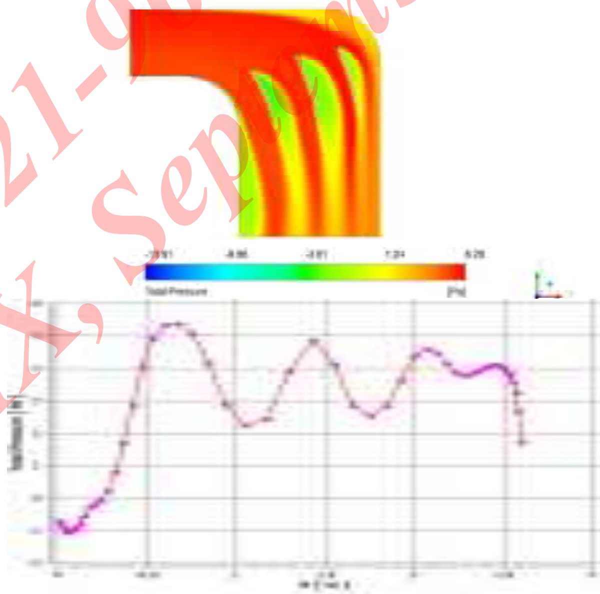
Simulation is done on existing economizer duct without vanes and results of velocity vector and pressure contour are shown in previous chapter. It is clear that velocity profile at the economizer section is not uniform, mass flow distribution of gas over tubes is also non-uniform. Due to this heat transfer across tubes is also ineffective, result of this increases heat transfer area.



Pressure contours of economizer duct with straight inlet vanes

Above figure it clearly shows the pressure drop reduced by 50% compared to baseline duct due to smooth flow. Thus the flow at the entrance of the elbow differs considerably from a fully developed pipe flow. The flow in elbow is influenced by centrifugal force due to its curvature. This centrifugal force is, in principle, balanced by a pressure gradient in the plane of curvature. However, near the wall where the velocity is small, this pressure gradient can no longer be balanced and consequently fluid in the middle of the pipe moves at the outer wall and then turns to move inward along the wall.

The flow on the outer wall and separation at the inner wall make flow very complex. The result is a secondary flow superimposed in the main flow in the plane perpendicular to the main flow.

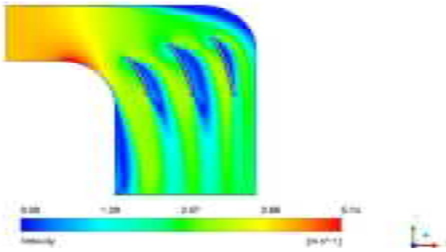


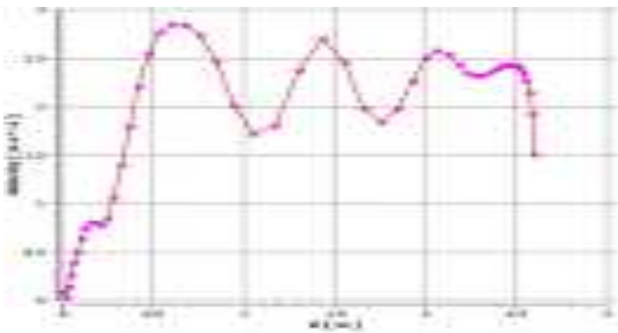
Total pressure contours of economizer duct with straight inlet vanes

Above figure shows ,the pressure is high as fluid enters into the straight pipe and pressure is reduce near the inner and outer wall by placing straight vanes at the middle. Separation of flow takes place by placing vanes at the middle and pressure is uniform as fluid flows over vanes. The pressure again increases near the exit.

ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

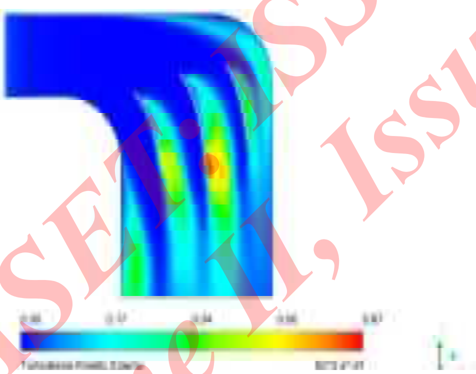


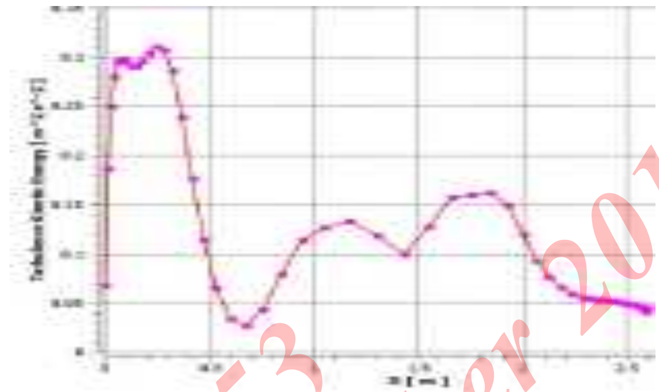


Velocity contours of economizer duct with straight inlet vanes

Above figure shows, the direct effect of secondary flow is to displace the region of maximum velocity to the center towards the outer wall. For the elbow entrance the mean axial velocity profile significantly altered with respect to the fully developed profile in

the straight pipe and the location of the maximum velocity is shifted towards the inner wall of the elbow.





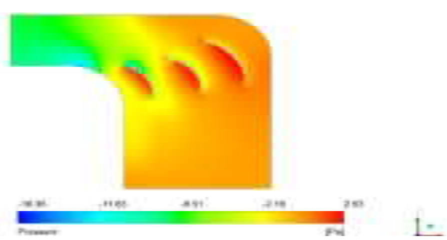
Above figure shows, the flow is uniform in straight pipe and increases towards outer wall. As fluid flows over the vanes at the middle the turbulence is high due to reverse

flow.

5.2.3 CFD Simulation of economizer duct with curved inlet guide vanes

Simulation is done on existing economizer duct with straight vanes and results of velocity vector and pressure contour are shown in previous chapters. It is clear that velocity profile at the economizer section is not uniform, mass flow distribution of gas over tubes is also non-uniform. Due to this heat transfer across tubes is also ineffective, result of this increases heat transfer area. Turbulence levels are reduced compared to baseline model the flow is improved still the flow pattern has to be optimized to improve the performance of the economizer. This is done by adding inlet guide vanes in the flow

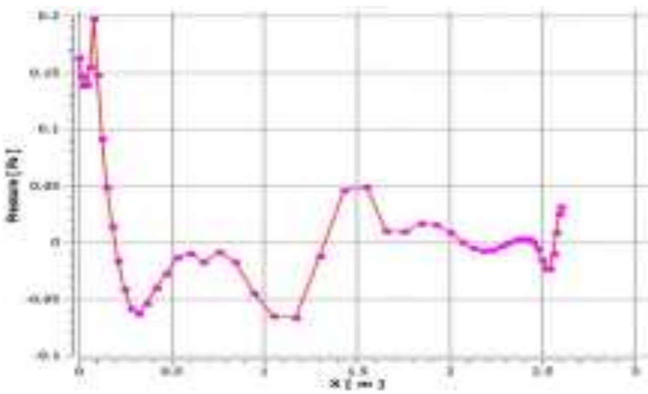
path of the simulation. The results in the form of velocity, pressure and turbulence contours are studied based on fluid mechanics applications, geometry modification are implemented by providing vanes at inlet to the original geometry.



Vol. 2 Issue IX, September 2014

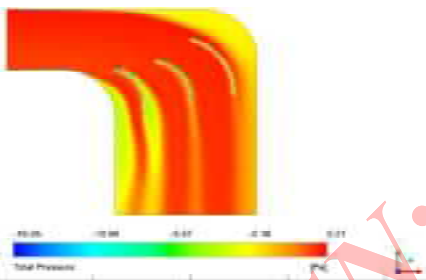
ISSN: 2321-9653

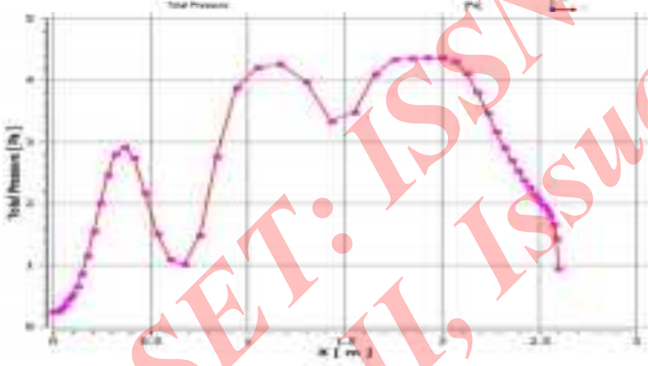
INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)



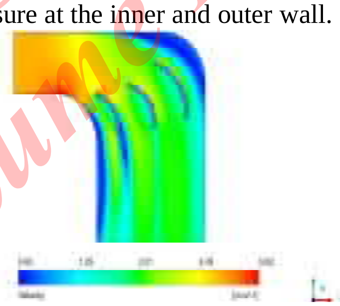
Pressure contours of economizer duct with curved inlet vanes

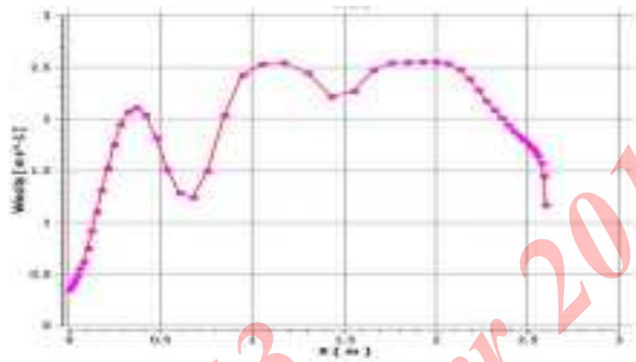
Above figure shows the pressure contours economizer duct modification with inlet vanes fillet radius 0.5m shows lesser pressure drop reduced.





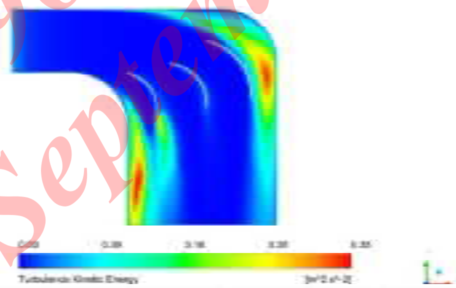
Above figure shows high pressure at the middle and low pressure at the inner and outer wall.

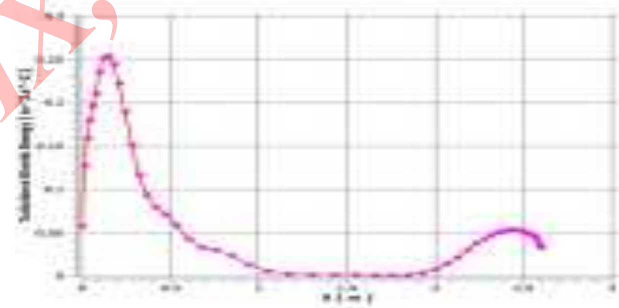




Velocity contours of economizer duct with curved inlet vanes

Above figure indicates that velocity profile at the inlet of economizer is uniform. This results Mass flow distribution of gas over tubes is also uniform.





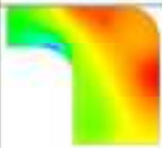
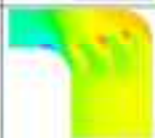
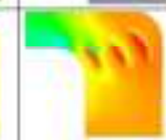
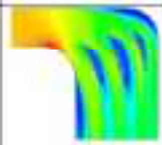
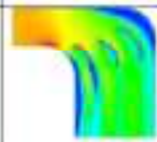
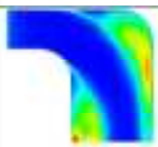
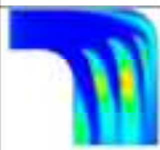
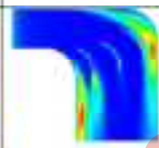
Turbulence contours of economizer duct with curved inlet vanes

Above figure shows more turbulence occurs towards inner and outer wall due to placing of curved vanes at the middle. But there is uniform flow of fluid over the vanes.

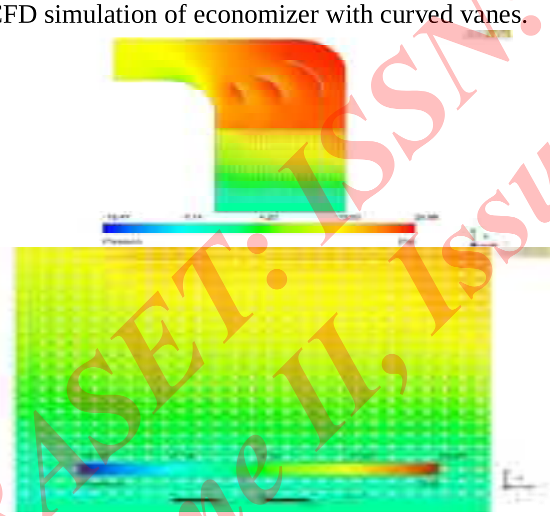
5.2.4 Comparison of Pressure velocity contours with and without inlet vanes

ISSN: 2321-9653



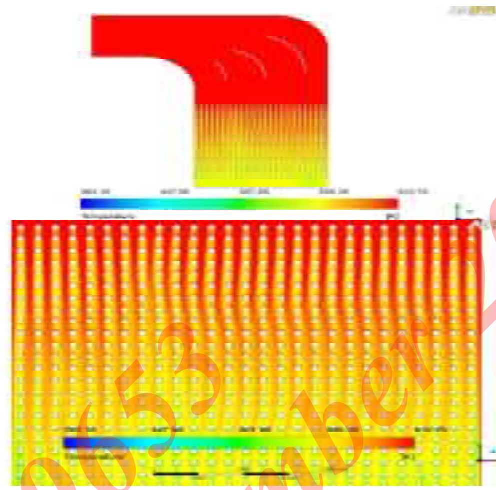
| Variable | Baseline economizer duct | Economizer duct with straight vanes | Economizer duct with curved vanes |
|---|---|---|---|
| Geometry-flue gas domain |  |  |  |
| Pressure contours |  |  |  |
| Velocity contours |  |  |  |
| Turbulence contours |  |  |  |

5.2.5 CFD simulation of economizer with curved vanes.

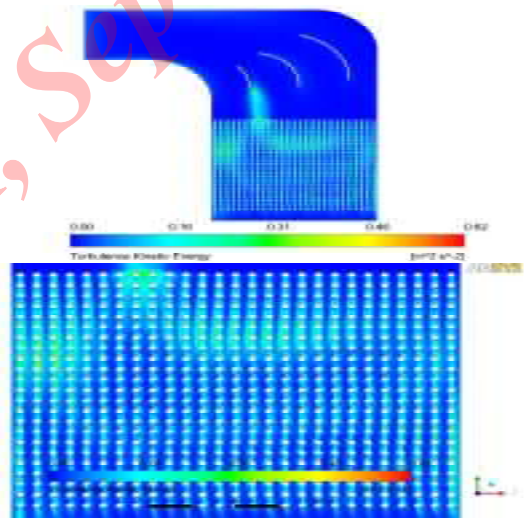


Pressure contours of economizer duct mid plane

Above figure shows pressure is more at entrance of tube section and gradually decrease of pressure take place. More drop pressure occurs at the middle plane.

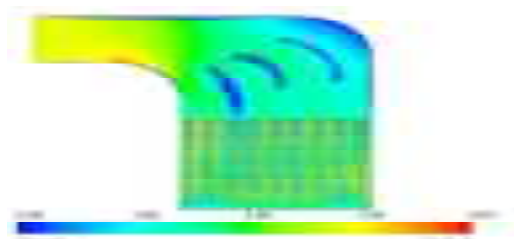


Above figure shows, there high temperature at the inlet section of economizer and gradually drop temperature takes place at the outlet.



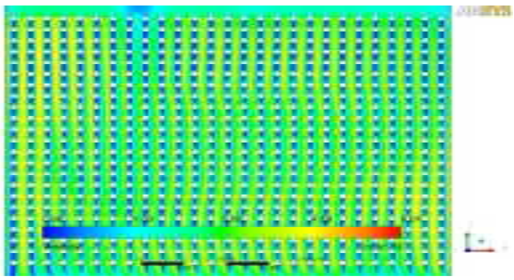
Turbulence contours of economizer duct mid plane-zoomed view

Above figure shows there is uniform flow of fluid over the curved vanes. Turbulence occurs at the entrance of tube section.



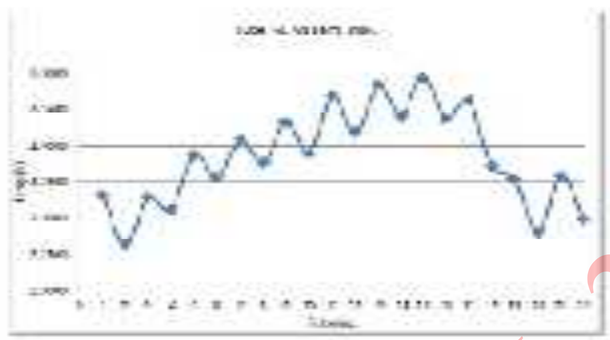
ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

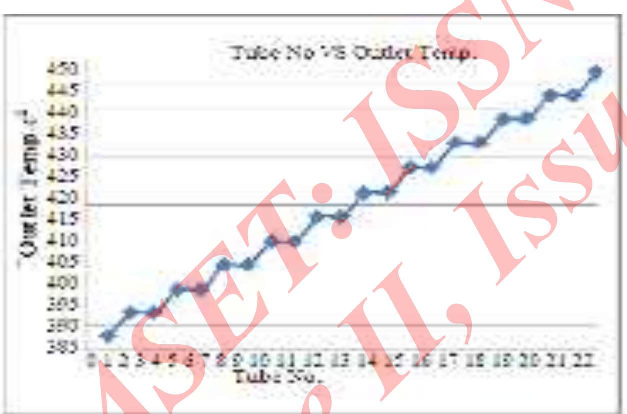


Velocity contours of economizer duct mid plane view

Above figure shows there is less velocity at the inlet and increases towards the outlet.



Temperature rise across the tube




Tube Outlet temperature.

## VI. CONCLUSIONS

Specific issues like the undesirable effects of mal-distribution of heating and heated medium have been understood through applications of CFD codes.

1.Gas flow distribution or heat transfer into the economizer section is improved through use of guide vanes at inlet of economizer duct.

2.Analysis of economizer module was carried out using K-ε model. The model was validated. The results of simulation indicate the uniform flow of gas over tubes after adding the curved vanes at inlet of module. 3.The results were compared with site data and showed good agreement.

4.The results of the CFD analysis can be used in enhancing the heat transfer in design of different type of economizer. The model developed is better equipped to predict the economizer outlet temperature.

5.Future direction would include development of model to predict soot formation over tubes.

6.In days to come, CFD would become integral design tool to predict various operating scenarios of the product and thus improving the effectiveness of design process.

## REFERENCES

1. A.D. Patil, M.T. Sable, S.B.Barme "To optimize economizer design for better performance", International Journal of Applied Research in Mechanical Engineering, Issue 2011, Issn; 1792-4596

2. Deendayal Yadav, Dr. G.V. Parishawad, P.R. Dhamangaonkar, Dr. S.R. Kajale, Dt. M.R. Nandgaonkar, Dr. S.N. Sapali "Effect of Arrestor on erosion in economizer zone and its analysis", AMAE, 2011.

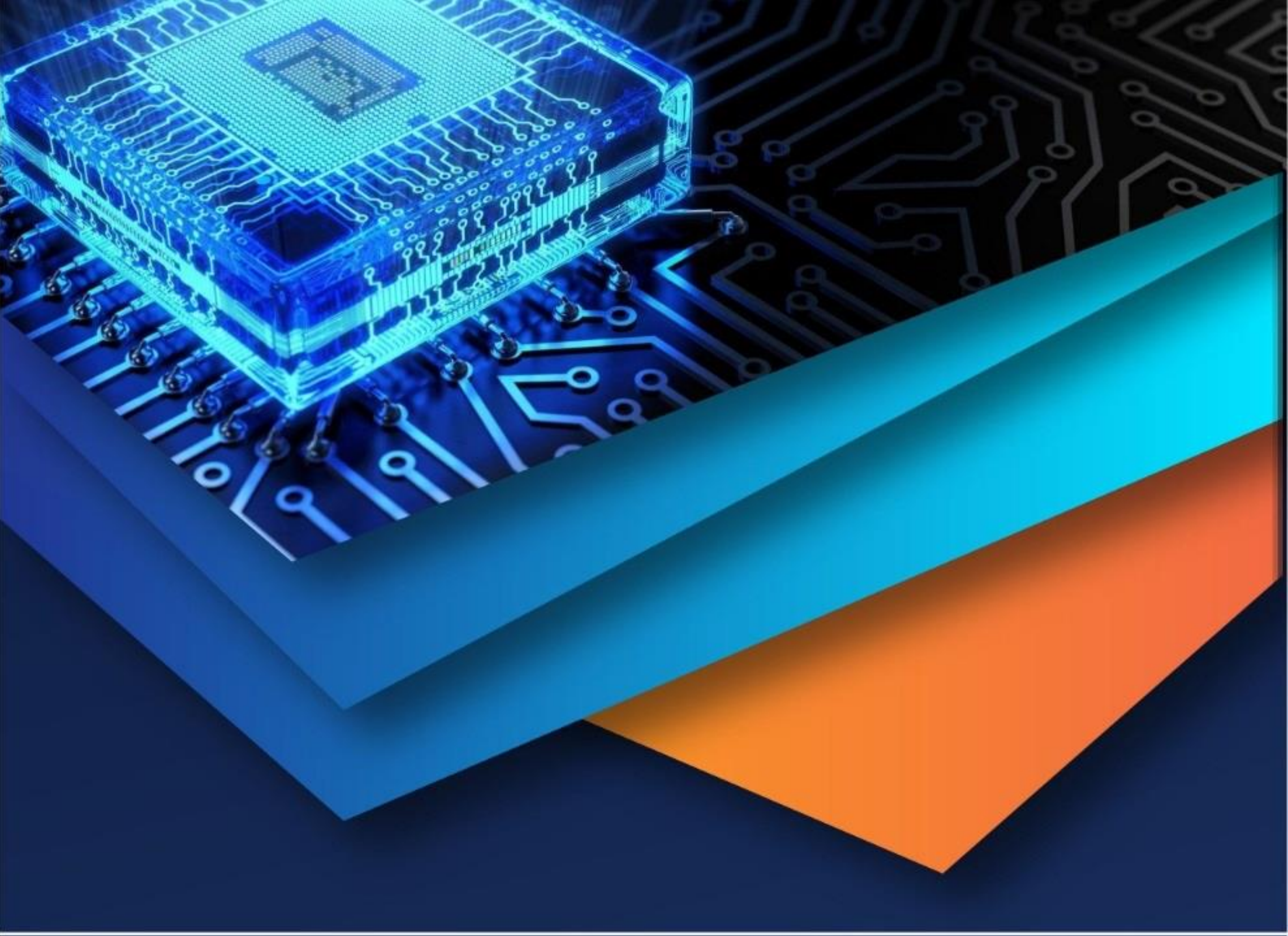
3. Young SUN, Lin Ma, and Colin Fidge Optimisation of Economizer tubing system renewal decisions", Queensland University of Technology, Austrialia, 10 March 2011.

4.Neihad Al-Rhalidy "Design optimization of industrial duct using computational fluid dynamics".

5.S.K. Das, K.M. Godiwalla, S.O. Mehrotra, K.K. Sastry and P.K. Dey "Analytical model for erosion behaviour of impacted fly-ash particals on coal-fired boilet components", Sadhana Vol.31. Part 5, pp 583-595, October 2006.

6. A.P. Mann, T.F. Dixon, F. Plaza and J.A. Joyce "Opportunities for improving the performance and reducing the costs of bagasse- fired boilers", Three Dimensional Furnace computer modeling, 21st Symposium (international) on Combustion,. 21: 265-452, 1986.

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)




10.22214/IJRASET

45.98

IMPACT FACTOR:
7.129

IMPACT FACTOR:
7.429


# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY