DESIGN IMPROVEMENT OF A NON-CONDENSING ECONOMISER USING CFD

ECONOMISER

An economiser is a mechanical device which is used as a heat exchanger by preheating a fluid to reduce energy consumption. For many industrial processes the availability of steam plays a major role. But even the exhaust gas of modern steam boilers still contains a lot of energy that can be used. This is where Economisers are applied as feed water pre-heater and supply the boiler with pre-heated water. As a result the efficiency of the plant can be increased.

SCOPE OF WORK

Economisers are heat exchangers with exhaust gas on the one side and water on the other side. So, the main objective is to increase the heat transfer from the flue gas to water tube.

NON - CONDENSING ECONOMIZER

The most widely used one in a thermal power (in the case of coal-fired boilers) plant is the non-condensing economizer. These are basically heat ex-changer coils, that are finned around in the form of a spiral and are located inside the flue gas duct near the exit region of the boiler. They have the ability to reduces the fuel requirements of a boiler by transferring heat from the exit flue gas to the steam boiler feed water.

CFD APPROACH

- ✓ CAD model development (given by customer)
- ✓ Fluid domain extraction
- ✓ Grid generation
- ✓ solving (boundary conditions and solver settings)
- ✓ Interpreting the results

A three dimensional geometry of typical economiser was developed using solid works CAD tool. As shown in Fig 1, it has a duct arrangement where the flue gas will be sending through and inside which bank of tubes are placed to carry the boiler feed water. Thus heat is transfer to the boiler feed water due to the interaction between hot gas with the feed water tubes

Two different ducts were analysed using CFD technique.

1. Plain duct
2. Duct with baffle arrangement

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

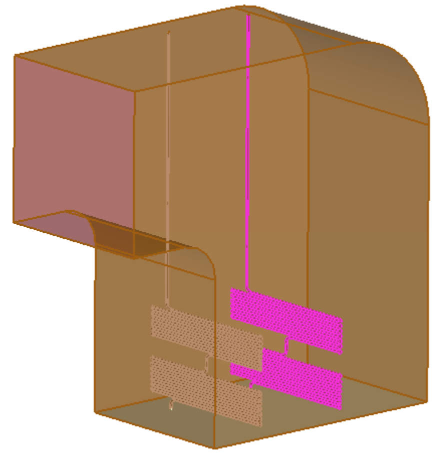

Fig 1 Plain duct with water tubes

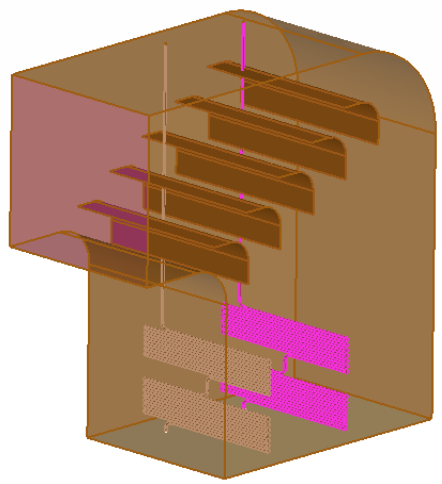

Fig 2 Baffled duct with water tubes

Grid generation: CAD geometry is imported into meshing software where the fluid domain has been extracted and surface meshes were created. Tetrahedral elements were used to discretize the fluid domain where the governing equations will be solved.

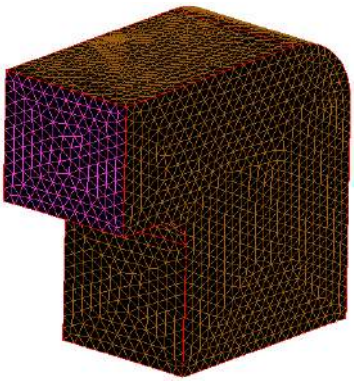

Fig 3 Surface mesh over duct wall

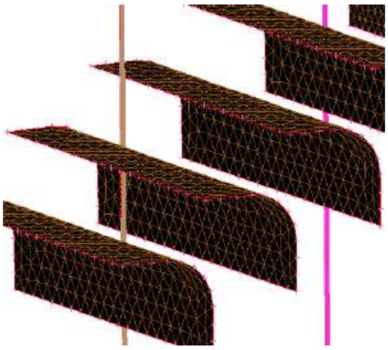

Fig 4 Meshed baffles

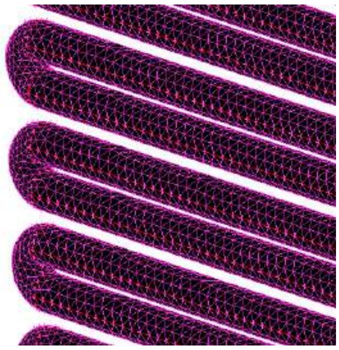

Fig 5 Meshed tubes

BOUNDARY CONDITIONS

The mesh file is then imported into solver interface where the properties of flue gas, inlet and outlet conditions were specified. In order to capture the turbulence nature, an appropriate turbulence model was used.

- Inlet – mass flow rate and temperature of flue gas
- outlet – pressure outlet
- walls – convection and conductive heat transfer
- working fluid – flue gas and water

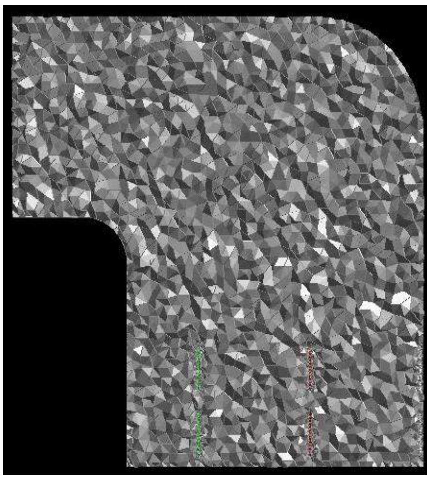

Fig 6 volume mesh

CONTOUR PLOTS AND VECTOR PLOTS

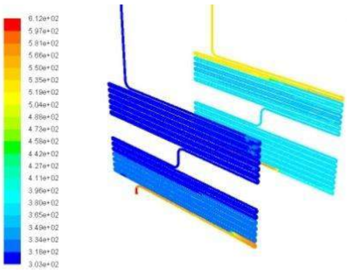

Fig 7 temperature plot -plain duct

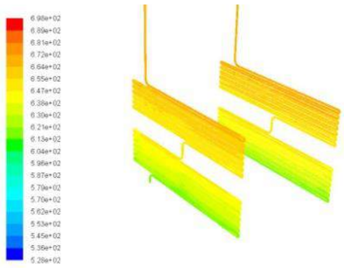

Fig 8 temperature plot– duct with baffles

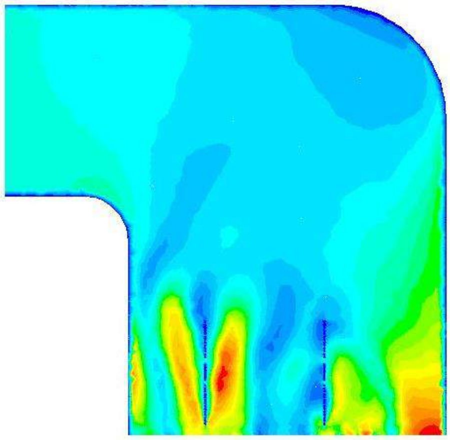

Fig 9 Plain duct

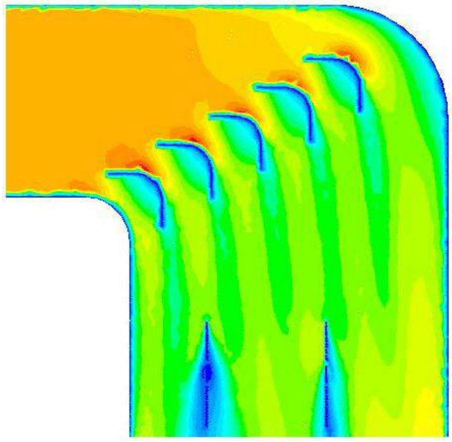

Fig 10 duct with baffles

Above figures shows the comparison of velocity distribution between two different ducts. The baffle arrangement (Fig 8 & Fig 10) provides a uniform velocity distribution which in turn allows the flue gas to flow over almost the entire water tubes. Thus the energy from the flue gas can effectively be transferred to the water tubes which increase the performance of economiser.

POWER OF CFD

CFD provides an insight view at anywhere inside the fluid domain which is not possible to get from experiments. And with the aid of various contour plots and vector plots it is quite comfort to study the flow nature and temperature distribution over the water tubes. So , CFD can comprehensively replace the time consuming , expensive experimental procedures and helps us to research infinite number of design modification with in a very short period.

CONCLUSIONS & FINDINGS

Thus, CFD was comprehensively utlized to understand the velocity ,temperature distribution of existing design. And it has been found that adding baffels would result in uniform and increased level of turbulence intensity (Fig 11) so does the heat transfer rate.

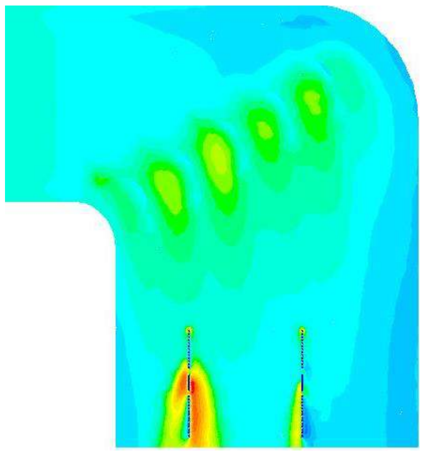

Fig 11 turbulence intensity

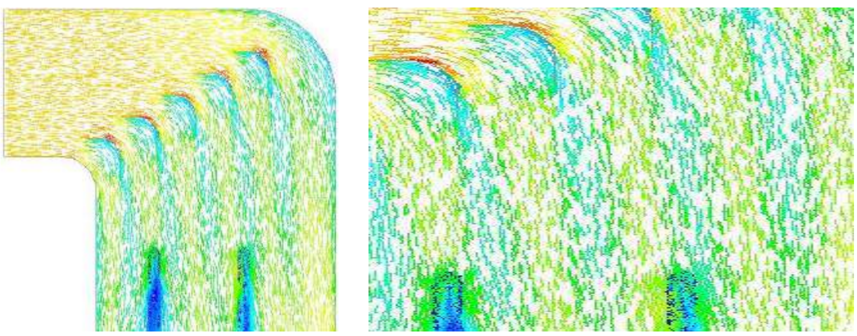

Fig 11 velocity vectors

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com