CASE STUDIES IN APPLICATION OF CFD ANALYSIS TECHNIQUE FOR AUXILIARY POWER SAVINGS

By K.K.Parthiban, Venus energy audit system

Web: www.venus-boiler.com -www.boilerspares.com
Email: venus1@venus-boiler.com -sridhar.sdbesbng@gmail.com

INTRODUCTION

Since the invention of desktop and laptop computers and the various software packages, the work styles of engineers have changed. Not to mention that lifestyle has also changed considerably. Earlier in power industry there used to be a flow modelling laboratory in which prototypes were made to study the flow patterns of air / flue gas duct system. For standardised designs such investment was worth it. As the industrial boilers developed at a faster pace, new boiler companies had entered the market segment which was not catered by the utility equipment manufacturers. Most of the industrial boiler equipment manufacturers do not have the expertise / interest in the area of flow modelling. Many a times the layouts are non standard for various reasons such site constraints and choice of number of fans. Sometimes we end up with an improperly designed duct system with local turbulences and eddies which cause pressure losses. When a turbulence / eddies is not favourable, it can affect the fan performance as well in terms of vibration, excess power consumption.

There is scope for improvement in duct system layout for optimising the energy consumed for pumping combustion air or evacuation of the flue gases from boilers. These case studies are about the use of CFD technique to visualize the flow patterns as existing and after a proposed modification. At least now we need not make trials in a running plant.

What is computation fluid dynamics?

It is fairly new design tool used by several industries for optimum design related to fluid flow. It is a numerical analysis method involving large number of iterations using computers and predicting the fluid flow patterns. In boilers it can be used for burner design, furnace design, duct design, heat transfer, flue gas emission control.

CFD involves the solution of the governing laws of fluid dynamics numerically. The complex set of partial differential equations is solved on in geometrical domain divided into small volumes, commonly known as a mesh (or grid).

Why CFD has become popular?

There are various reasons why CFD has become popular, some of which are:

- CFD allows numerical simulation of fluid flows, results for which are available for study even after the analysis is over. This is a big advantage over, say, wind tunnel testing where analysts have a shorter duration to perform flow measurements.

- CFD allows observation of flow properties without disturbing the flow itself, which is not always possible with conventional measuring instruments.
- CFD allows observation of flow properties at locations which may not be accessible to (or harmful for) measuring instruments. For example, inside a combustion chamber, or between turbine blades.
- CFD can be used as a qualitative tool for discarding (or narrowing down the choices between), various designs. Designers and analysts can study prototypes numerically, and then test by experimentation only those which show promise.

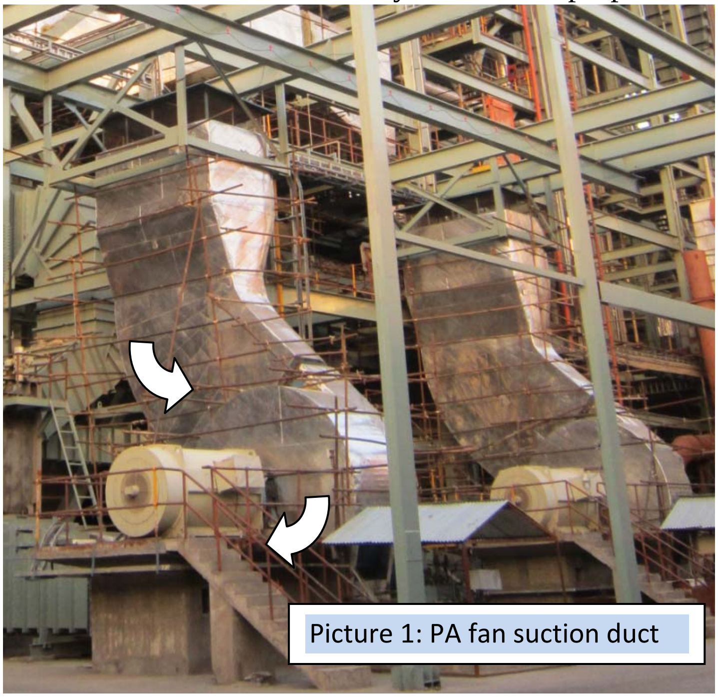
Ducts in boiler system- Scope for application

In the case of Industrial boilers / captive power plants, the boiler capacities are not standardised due to various reasons such as plant layout, plant capacity requirement, fuel combinations to be used. The duct layouts are engineered every time. Honestly the boiler design engineers in some companies do not have practical exposures. They tend to copy / make drawings based on the reference projects. The designs in most of the case are thus not optimised. There are many instances where the duct layout is so awkward, that the fans are not able to handle the additional resistances encountered. An expert engineer should be able to identify the defects in design.

CFD comes handy to visualise the flow pattern and to decide on the corrections to be carried out. We present some cases where we made use of CFD program to explain the picture now and the picture when the suggested modification is implemented.

Case 1 – Wrong fan inlet duct layout

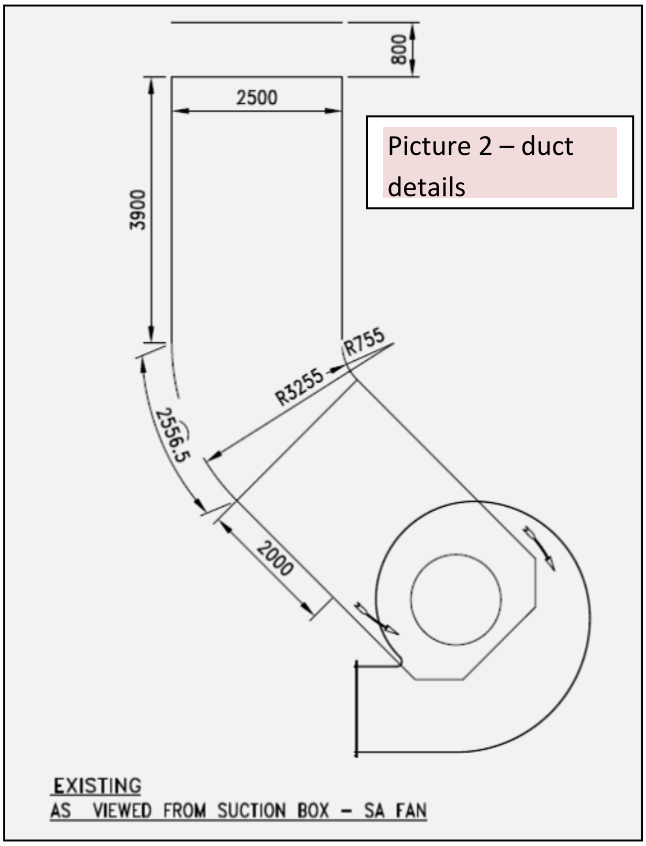
This is a case of a CFBC boiler wherein the PA fan and SA fan inlet duct layouts are not proper. See picture by the side. The fan inlet duct layout is such that the air is imparted with a pre-swirl which is not favourable to that of the impeller and volute casing.



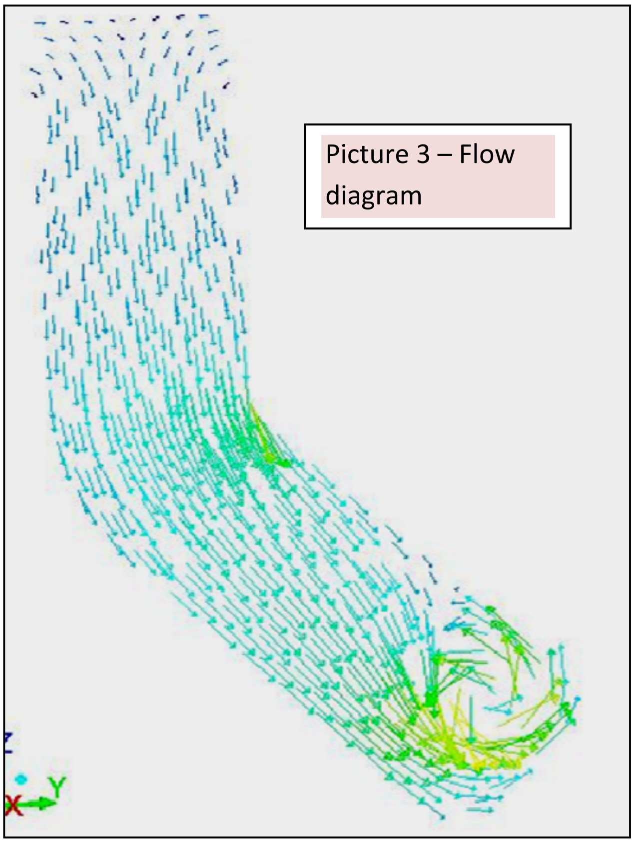
Picture 1: PA fan suction duct

Flow dividers can be added to modify the flow pattern so that the direction of swirl is reversed. A top inlet duct is generally the right way.

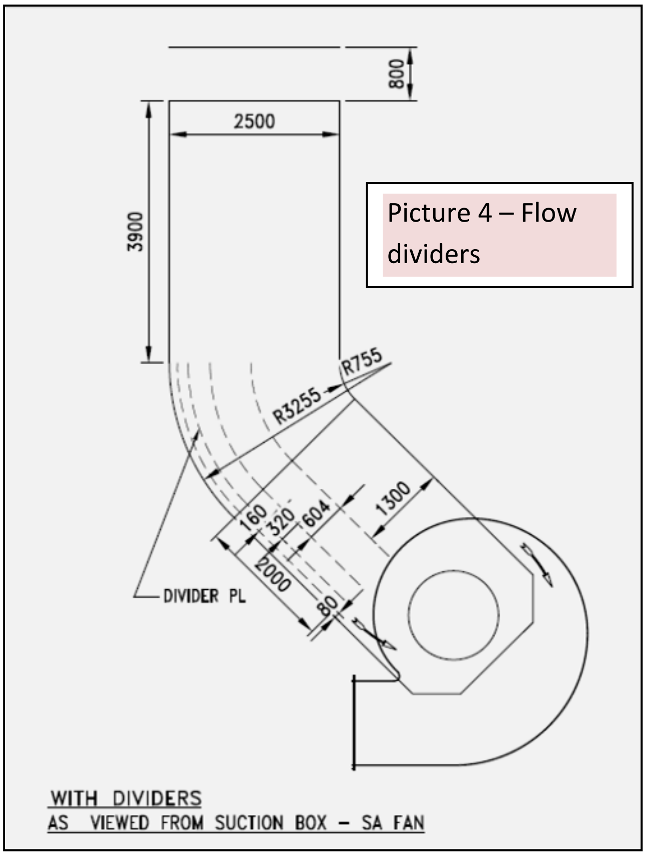
The modelling was done using Catia software. The meshing was generated using gambit. The analysis was done using CFD software. Picture 2 is the present duct configuration and the picture 3 is the flow vectors. We finalised flow dividers as shown in picture 4. The effect can be seen in picture 5. Wrong fan inlet duct design can result in poor performance and vibration.



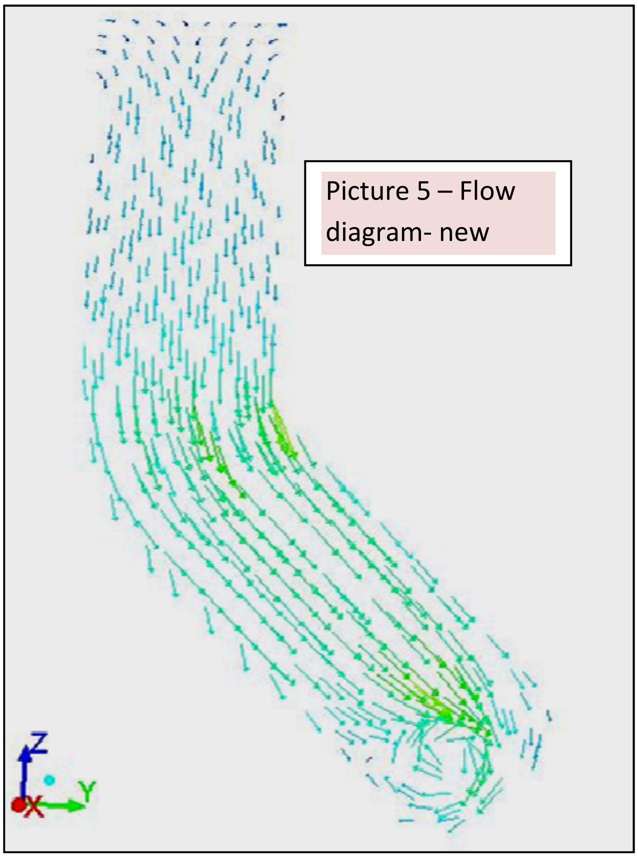
Picture 2 – duct details



Picture 3 – Flow diagram



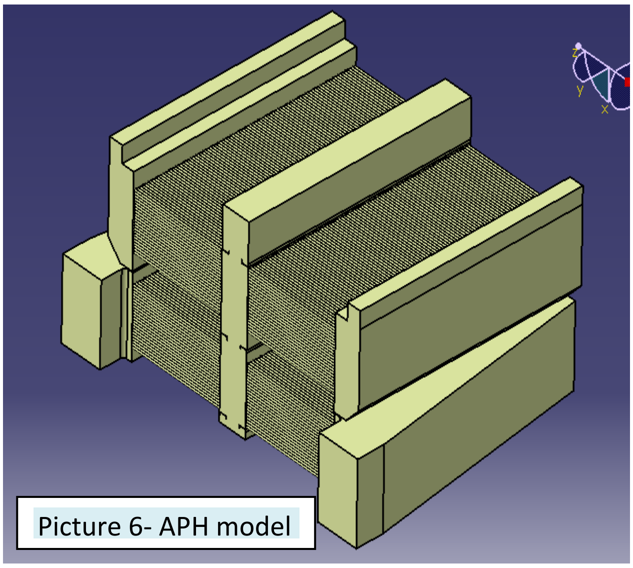
Picture 4 – Flow dividers



Picture 5 – Flow diagram- new

Case 2 – deviation in design performance

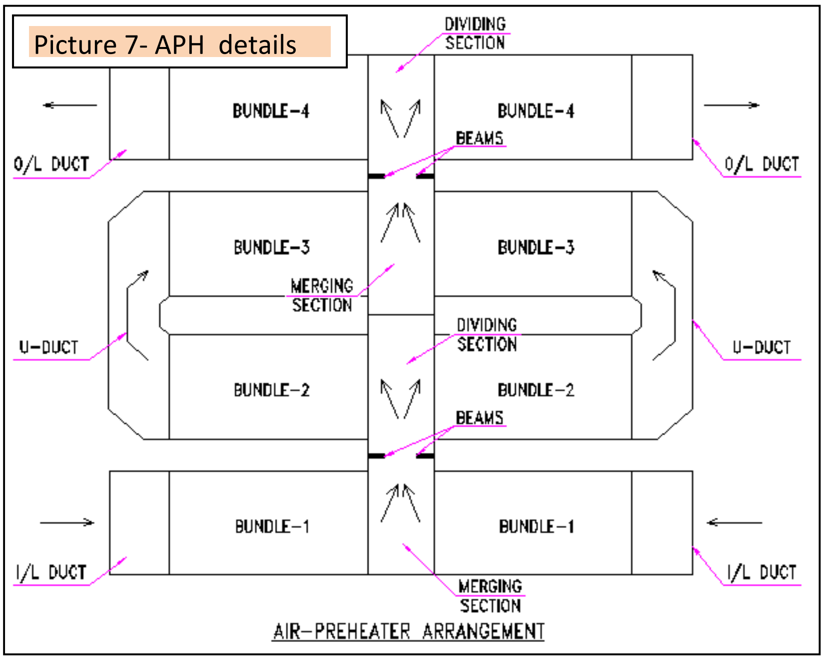
There are two APH in this boiler. One is primary APH (PAPH) and the other is secondary APH (SAPH). Both PAPH and SAPH are identical. The 3D model of PAPH / SAPH is shown in picture 6. Each Airpreheater is divided in to two blocks to reduce the span of the tubes. The air from two PA fans (or SA fans) enters the blocks as separate streams but they mix at the intermediate chambers. When the APH was taken up for fluid flow studies, it was seen that there are two areas of pressure loss. They are



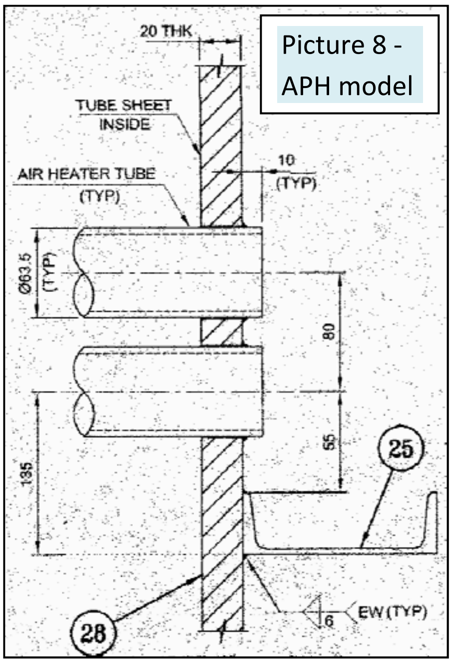
Picture 6- APH model

1. The air streams oppose each other and take a 180 deg turn at the intermediate ducts. Also they divide in two streams in opposite directions at the intermediate duct. There can be additional losses here.
2. The air while passing from block 1 to block 2, goes through narrow passage created by the two stiffening channels of ISMC 250. See picture 7 & 8 below. Similar narrow passage is formed between block 3 & 4. The air velocity in this area is of the order of 20-24 m/s.

See the detail below which explains the above two points.



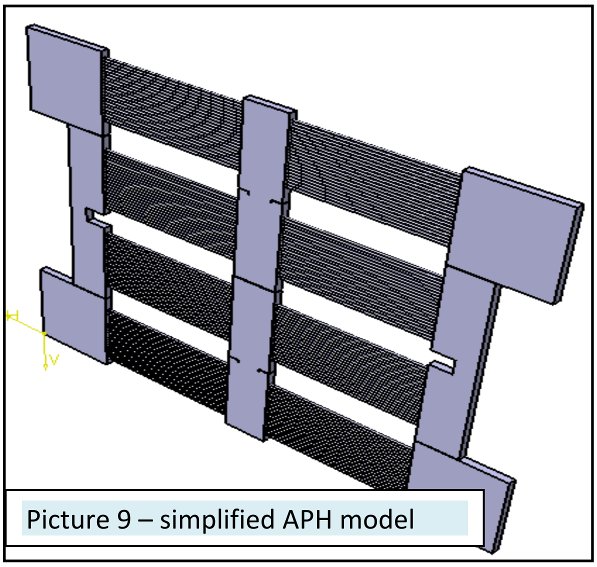
Picture 7- APH details



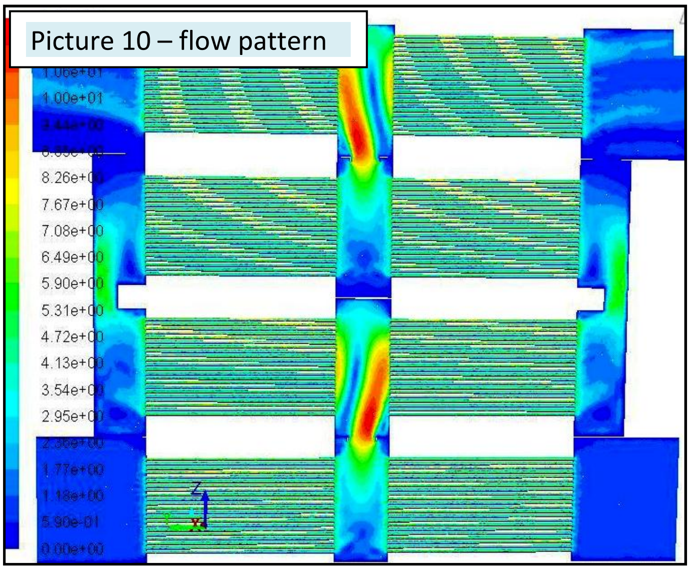
Picture 8 - APH model

Simple modelling

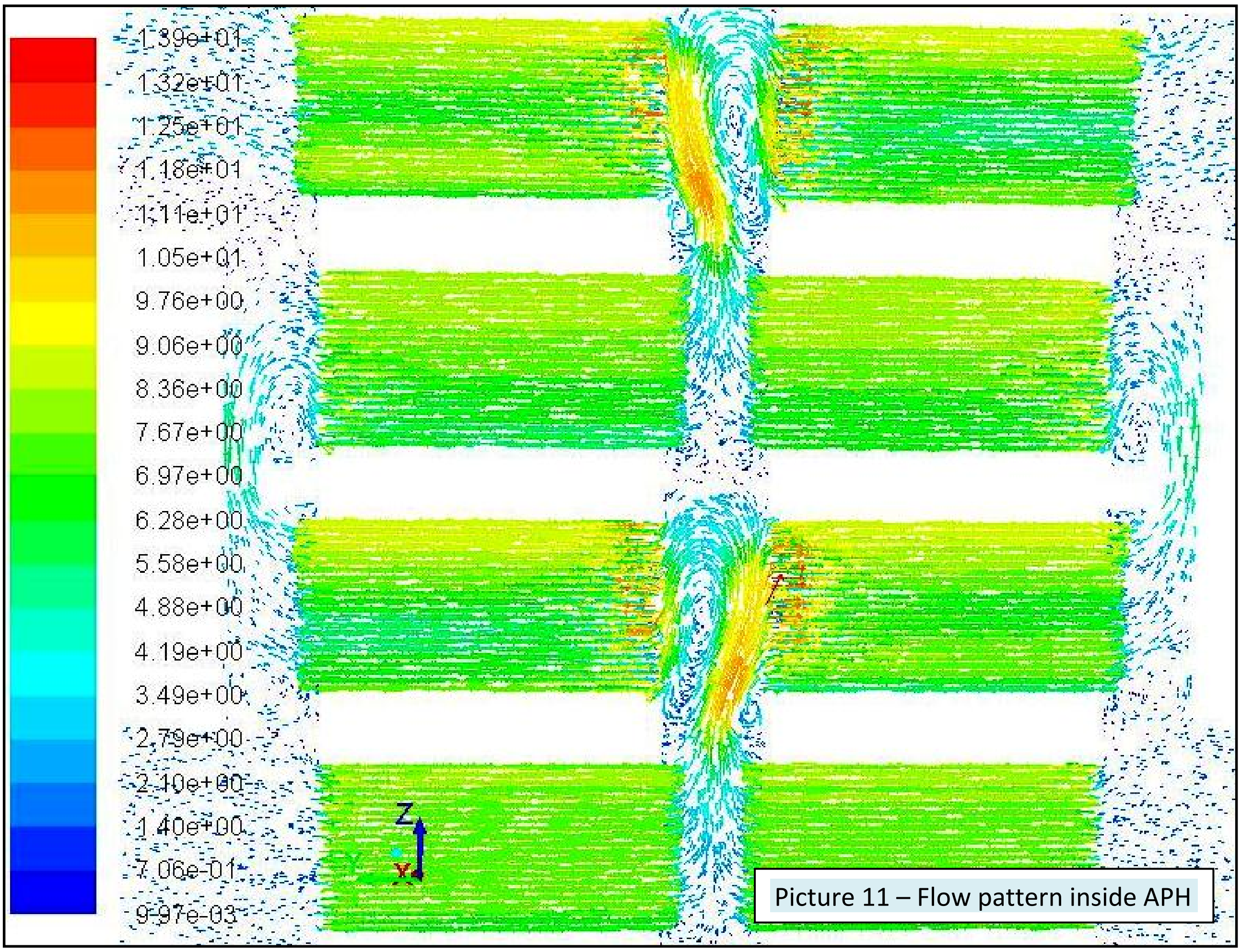
The model had to be simplified for analysis purpose. Only one column of 22 tubes in a block is considered for analysis and flow visualisation. See picture 9 & 10. No heat flux was considered in modelling. The aim was to obtain the flow patterns and as well as flow in each tube of a block on the right side and left side. Picture 11 shows the recirculation zone due to beam stiffener.



Picture 9 – simplified APH model



Picture 10 – flow pattern

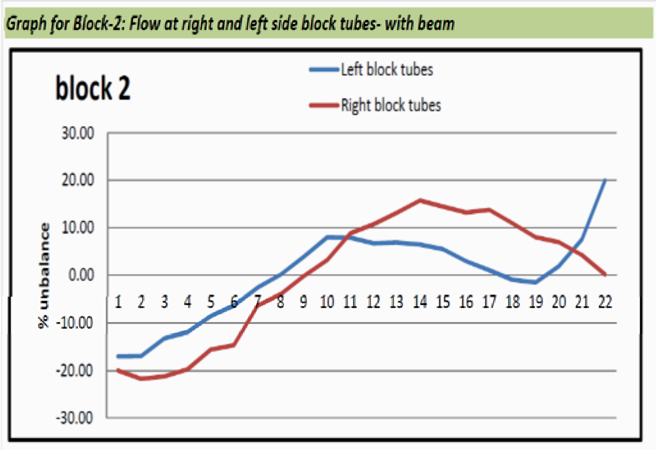


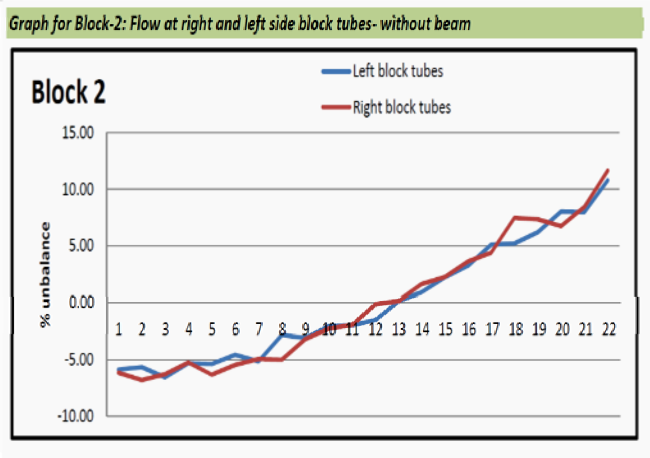
Picture 11 – Flow pattern inside APH

Result

The flow pattern is seen to improve when the beam is not present. The unbalance in flow was found to go up to -20% & + 10% in block 2 & 4 with the stiffener channel. The unbalance comes down to ± 6% in the case without stiffener. See pictures 12 & 13 below.

Picture 12: Flow unbalance with stiffener *Picture 13: Flow unbalance without stiffener*





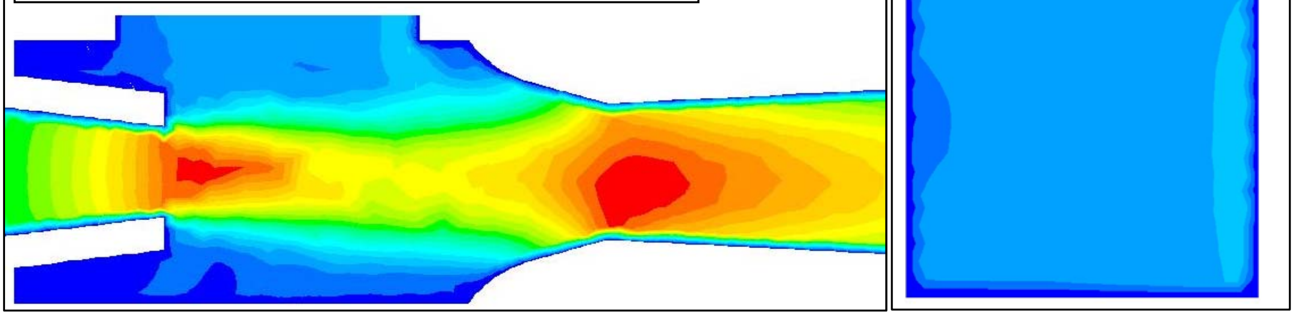
Recommendation

The APH blocks are a solid mass once the tubes are rolled and seal welded in place. The stiffener channel does not contribute to any stiffening. However vertical stiffener plates were suggested for the tube sheet. The modifications were carried out and the draft loss had come down considerably. However the opposing air streams continue to remain as the pressure drop device.

Case 3: Performance improvement of mixing nozzles

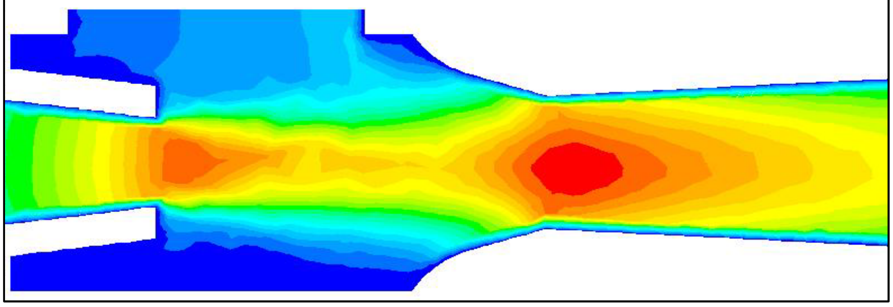
Mixing nozzle is an important component in under bed feeding system of AFBC boiler. This can generate dust nuisance and choking of inlet chute when the coal has more percentage of fines. One of our customers desired to have better vacuum at the venturi. Generally this is possible to achieve with lower throat diameter or by optimising the location of the throat in the mixing nozzle. We took up the case for CFD analysis. See picture 14. The right side picture shows the air sucked from the suction chamber opening from where the fuel falls in to the throat.

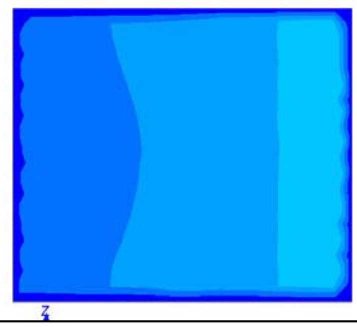
Picture 14: flow picture at mixing nozzle chamber



The picture on the left is the visualisation of the jet effect. Red lines indicate the highest velocity lines. We advanced the cone forward to see the effect in air flow from suction chamber opening. The pictures below show the effect.

Picture 15: flow picture at revised mixing nozzle





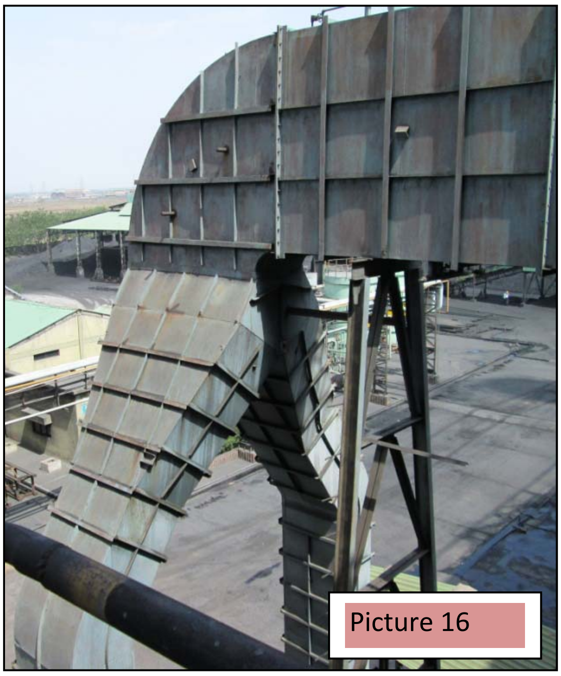
There is more area of dark blue color indicating the suction effect at the venturi inlet. The tables below indicate the free flow of air through the suction chamber inlet area. The flow is found to improve when the venturi cone is advanced in optimum manner. The flow area is divided in to cells and flow values are taken from CFD reports.

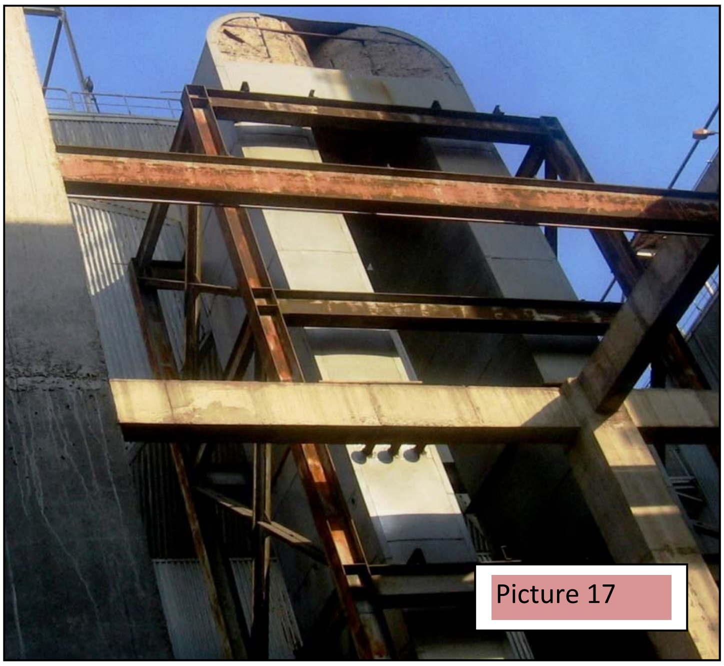
Free Flow at throat inlet from top before modification												
0	5	15	25	35	45	55	65	75	85	95	105	Sum
5	13.3917	13.6809	14.0371	14.3559	14.6395	14.8684	15.244	15.9402	16.8124	18.1358	15.9259	167.032
15	13.2706	13.6055	13.9821	14.3275	14.5876	14.7965	15.091	15.7824	16.7427	18.4332	16.7176	167.337
25	13.0415	13.3692	13.8285	14.2105	14.4455	14.5601	14.7197	15.4135	16.6456	18.7795	19.0695	168.083
35	11.2901	13.1035	13.6409	14.0506	14.2712	14.2441	14.4229	15.1282	16.5024	19.0166	21.8308	167.501
45	10.1511	12.9767	13.5429	13.9787	14.1458	14.1509	14.3076	15.03	16.3937	19.1922	22.2228	166.092
55	12.7024	13.0427	13.6091	14.0271	14.271	14.3573	14.5534	15.2865	16.6025	19.3057	21.8585	169.616
65	12.9371	13.2852	13.7748	14.2049	14.4448	14.681	15.0011	15.7598	16.9851	19.2084	20.595	170.877
75	13.206	13.5918	14.0226	14.4524	14.7301	15.041	15.4904	16.1837	17.3566	19.2067	20.2691	173.55
83.5	13.45	13.7591	14.2066	14.5934	14.9216	15.25	15.6875	16.4562	17.5437	19.0943	19.5459	174.508
91.5	10.6833	13.7736	14.1965	14.5954	14.9412	15.2806	15.8014	16.5721	17.5827	18.9625	13.9473	166.337
Avg	12.4124	13.4188	13.8841	14.2796	14.5398	14.723	15.0319	15.7553	16.9167	18.9335	19.1982	
											Total average	***169.093***

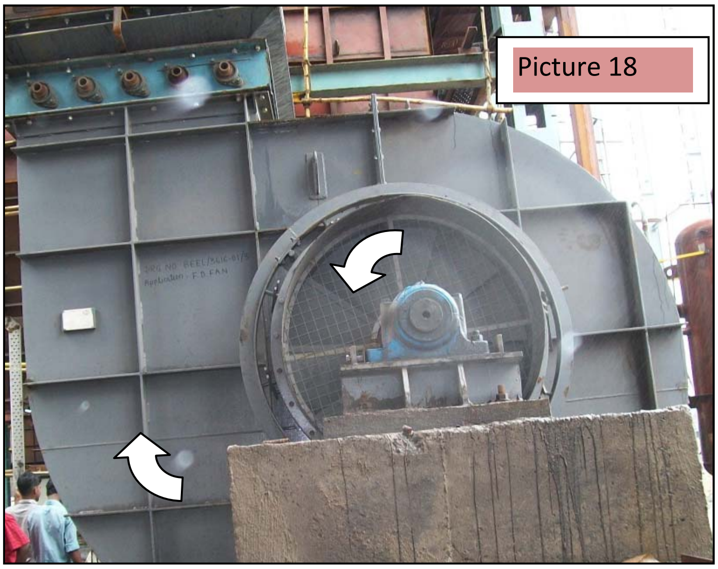
Free Flow at throat inlet from top after modification												
0	5	15	25	35	45	55	65	75	85	95	105	Sum
5	11.769	12.2309	12.902	13.801	14.84	15.819	16.9973	18.3728	20.073	21.694	17.136	175.634
15	11.483	11.9743	12.672	13.638	14.686	15.726	16.8983	18.3047	20.121	22.021	22.62	180.144
25	11.002	11.3995	12.119	13.237	14.398	15.512	16.5816	18.0395	20.038	22.477	23.32	178.122
35	9.2544	10.8481	11.624	12.82	14.238	15.152	16.3943	17.9137	19.993	22.652	24.018	174.907
45	8.1059	10.4664	11.317	12.615	13.959	15.051	16.288	17.7442	19.811	22.765	25.962	174.084
55	10.305	10.5303	11.321	12.626	14.029	15.213	16.3581	17.8105	19.973	22.716	25.71	176.591
65	10.714	11.0412	11.798	12.922	14.24	15.429	16.5781	18.0149	19.952	22.493	24.5	177.681
75	11.226	11.6416	12.398	13.382	14.543	15.709	16.8516	18.1866	19.969	22.182	23.416	179.504
83.5	11.672	12.1222	12.803	13.707	14.801	15.923	17.0333	18.3719	20.037	21.834	22.39	180.695
91.5	8.952	11.8883	12.884	13.772	14.788	15.911	17.0612	18.3503	19.962	21.645	16.319	171.533
Avg	10.448	11.4143	12.184	13.252	14.452	15.544	16.7042	18.1109	19.993	22.248	22.539	
											Total Avg	***176.89***

From the above, one can decide that the modification in the mixing nozzle can result in improvement the suction effect. The customer had gone ahead with procurement of new mixing nozzles with the revised geometry and installed them too. The performance was found to be good.

Some common mistakes made by design engineers in respect of ducting


Picture 16


Picture 17


Picture 18

Some pictures taken in some plants are presented below. Pictures 16 to 18 show come peculiar duct layouts which create unnecessary loading of the fan.

Picture 16 shows the opposing streams of air flow at the inlet of APH. Opposing streams are again to take a 90 deg bend. In picture 17, the ESP outlet duct has two 90 deg bends with flow split, all placed too close. In picture 18 shows a case which was handled by us in a 135 TPH FBC boiler. The FD fan IGV direction was wrong. The IGV at partial opening imparts a swirl which is opposite to that of impeller and the direction of volute. During commissioning the fan showed up vibration and then the fan supplier changed the IGV.

Conclusion

Some boiler users could easily understand the defects when identified and explained. Some boiler users do not appreciate the defects. CFD analytical tools make things simpler to them. It is recommended to look in to such defective layouts at the plant. Many times two fans are operated, since one fan results more inefficiency / does not cater to full air flow. Fan selection may not be wrong. It is defective duct layout, which makes things worse.