DESIGN OPTIMISATION OF INDUSTRIAL DUCTS USING COMPUTATIONAL FLUID DYNAMICS

Dr. Neihad AL-KHALIDY

Vipac Engineers & Scientists, Unit E1 -B Centre court, 25 Paul St. North, North Ryde, NSW 2113, Australia

ABSTRACT

This paper focuses on a design modification to a number of industrial ducts using Computational Fluid Dynamics (CFD) analysis considering all flow features relating to the duct system efficiency. The main objective of this work was to provide improved flow conditions into the top of a gas furnace windbox and along the duct with no re-circulation, minimum pressure losses and low turbulence levels. A geometrically 3D CFD model was assembled to capture the complex air pre-rotation before approaching the blower, air circulation in the blower region and airflow in the duct system to a boiler.

First the CFD model was validated with experimental data at three fan Variable Inlet Vane (VIV) percentage openings. Then, a parametric study was carried out to remove recirculation regions in the duct.

The recommended modifications provided a significant improvement in minimising recirculation regions in the duct and subsequent testing of modified duct system has validated the approach using CFD analytical tools.

INTRODUCTION

The development of powerful computers has made it possible to develop models to describe complex industrial fluid flow system. In this study CFD analysis was used to optimise the design of a number of industrial ducts. The ducts were used to provide air to a gas boiler in a power station. The ducts were historically suffering from a low efficiency due to air circulation inside the duct, high turbulence intensity into the top of a gas furnace windbox and pressure differentials at burner injector inlets.

In this work we analysed two duct systems and provided preliminary design concepts for the redesign of the systems based on a detailed flow analysis and acoustic analysis. This paper focuses on the fluid flow results analysis.

MODEL DESCRIPTION

Views of one of the duct systems can be seen in Figure 1 and Figure 2. The duct comprises a total volume of about 32 m^3 excluding the boiler and inlet diffuser. The duct has a rectangular cross section adjacent to the blower and the wind box. A geometrically 3-dimensional model of that duct was assembled to capture the complex airflow pattern in the system (see Figure 2).

Three dimensional incompressible steady flow computations were carried out using the commercially available code ANSYS-Flotran. This solves discretized forms of the Reynolds average Navier Stokes equations for turbulent flow using the Finite Element Methods. Two turbulence models were investigated namely standard K-ε and RNG models (Launder and Spalding, 1974 & ANSYS-Flotran, 2001). In general, the RNG k-ε model with near wall function has shown a reasonable improvement over the standard K-E model as it accounts for the effect of swirl on turbulence (Yakhot and Orszag, 1986 & 1992). Also, RNG capable of providing more accurate results for flows involving separation and pressure gradients (Al-Khalidy, 2001). The RNG model was used to perform the calculations of the cases presented in this paper. A number of user-defined codes were written to define the boundary conditions at the blower and to calculate the mean values at a number of selected sections inside the domain of analysis.

Figure 1: View of the duct system.

BOUNDARY CONDITIONS AND DISCRETISATION

The following boundary conditions and assumptions were used

- Inlet pressure boundary condition was prescribed at the diffuser inlet, according to different mass flow rate
- The inflow boundary conditions were based on known flow profile and direction. The flow profiles in the ducts were measured using Pitot tubes fitted with pressure transducers. Resultant pressures were recorded (by a data acquisition system) for a number

of incremental depths into the duct and at a range of locations along the duct. The measurements were performed for three different boiler demands 10%, 60% and 100% for two duct systems

- Fan was simulated as a vector flow with varied radial and tangential components based on the boiler demand. The boundary condition of the fan was modelled through user defined functions

- Burner registers modelled as open-ended cylinders with a simple warmer can assembly, and porous plug at register outlet was used to simulate the swirlers by matching the known windbox to boiler pressure drop (OCPL control room information)

- Turbulence inlet quantities (kinetic energy and dissipation rate) were calculated from empirical relationships

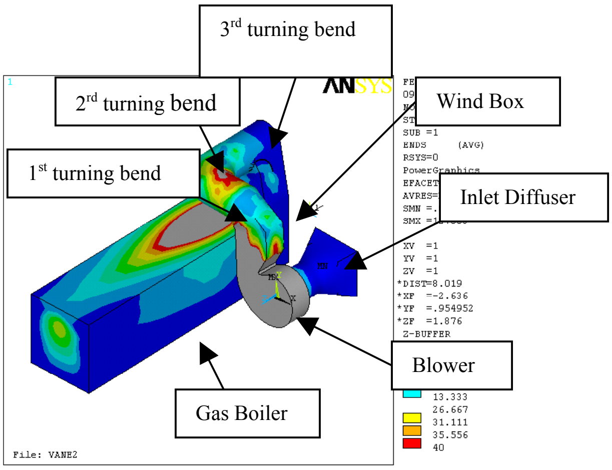


Figure 2: Geometrical Model.

For the current analysis more than 400 000 unstructured cells covered the computational domain. The RNG k- ε model is combined with a wall function scheme to avoid using a very fine mesh near the wall and improve turbulent flow simulation. An average cell size of 8 cm^3 was used.

An iterative procedure was used to estimate the air velocity into three directions, pressure profile and turbulence parameters. For the pressure-velocity coupling ANSYS-Flotran employs a global solver based on the SIMPLE algorithm [Patankar S., 1980]. A second order accurate numerical approach available in ANSYS-Flotran[1] was employed to discretise the advection term in the governing equations. Relaxation parameters were specified to stabilise the numerical solution process and guarantee overall convergence. The normalised residuals of continuity, x-, y-, and z-velocity, k and epsilon were reduced between three and five orders of magnitude.

RESULTS

First the CFD model of the existing duct was validated by comparison with the measured flow profiles for three fan Variable Inlet Vane (VIV) percentage openings. A Pitot tube fitted with a dynamic pressure transducer was used to measure the velocity and pressure fluctuation profiles at various Fan VIV percentage openings. Figure 3 shows a plot of the airflow speed versus VIV% as measured at the centre of the inlet of wind box. Good agreement is found between the CFD results and the measurement data. One can see from Figure 5.B1 the velocity at the centre of the wind box varies between 12 to 15 m/s at 60% VIV opening. The CFD results and measured data also show the same flow characteristics such as re-circulation region location and approximate size at the 3^{rd} bend and recirculation size at the first bend.

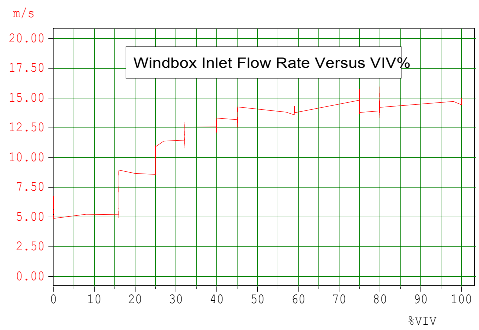


Figure 3: The mass flow rates and other parameters corresponding to the boiler demand %.

In parallel with the CFD analysis, ¼ scale physical modelling was also undertaken. Good agreement was achieved between the CFD results and the experimental data from the physical model. However, CFD analysis provides the ability to quantify flow improvement and to quickly predict the behaviour of several design proposals at less cost. The CFD analysis has offered a comprehensive range of output including velocity distribution, pressure profile, turbulence levels, etc. Then, a parametric study was carried out to minimise recirculation regions in the duct. The following modifications were investigated:

1. Providing improved and additional turning vanes
2. Changing the length of the duct
3. Changing the pressure drop at the throat (entry to the boiler). A porous zone model was developed for this part
4. Proposing a new design

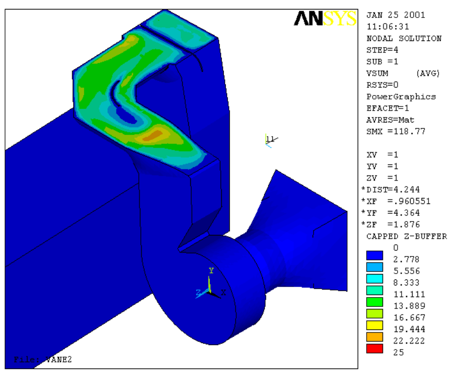


Figure 4: Flow profile at a section at the 1st bend inside the existing dust.

[1] Currently Flotran has a streamline upwind/Petrov Galerkin approach

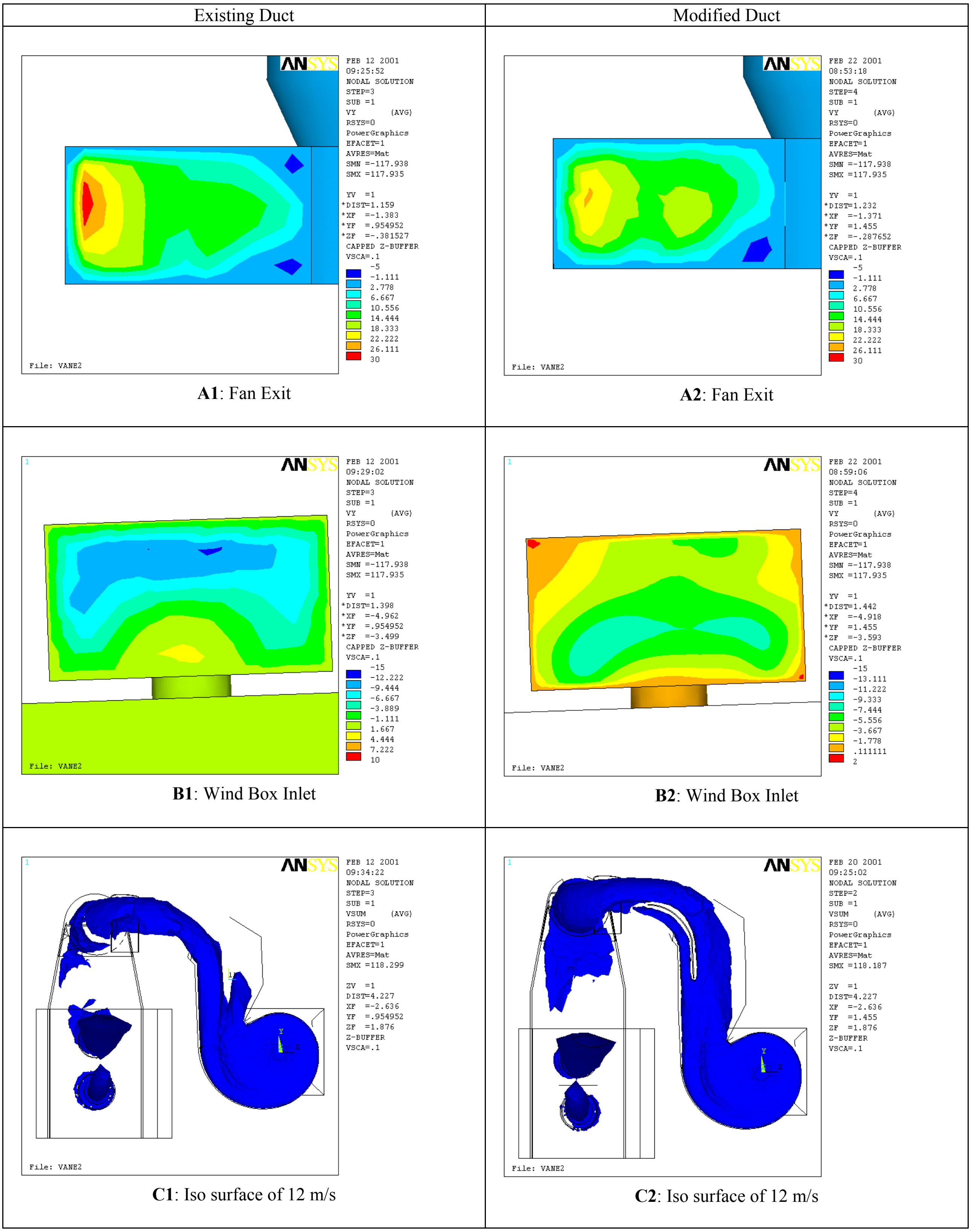


Figure 5: Flow profile for the existing and modified ducts at 60% boiler demand.

The analysis featured more than 25 runs for two duct systems and for three boiler demands. The 60% boiler demand case dominated the design comparisons. Based on the CFD results a modified duct was proposed and manufactured. Figures 4 to 8 show flow characteristics comparison between the modified duct system and the existing one. The modified duct comprises the following:

- ⇨ Extension of the fan exit duct by 1 m (Figure 5.C2 against Figure 5.C1)
- ⇨ One extended turning vane at the 1st bend (Figure 6.A)
- ⇨ Double turning vanes at the 2nd bend (Figure 6.B)
- ⇨ Two turning vanes at the 3rd bend immediately above the windbox entrance (Figure 6.D)

The following conclusions can be achieved from the above figures

- ⇨ The high velocity column which was in the existing duct as defined by the red contour in Figure 5.A1 was diffused as shown in Figure 5.A2
- ⇨ The recirculation regions were minimised at the fan exit (Figure 5.A2 against Figure 5.A1)
- ⇨ The flow profile at the windbox entry was improved (Figure 5.B2 against Figure 5.B1)
- ⇨ General flow profile improvement as shown in Figure 5.C2

The maximum, minimum and average velocity at a number of selected sections for the existing and modified duct system was extracted from the CFD results through user defined functions. The results are reported in Table 1. One can see that the maximum velocity at the windbox inlet was reduced from 15 m/s to 9 m/s while the average velocity is slightly changed.

Contour plots of velocity profiles at the registers and entry to the boiler is shown in Figure 7 for the modified duct system. One can see that the peak velocity in the top burner registers is less than 54 m/s while a peak velocity of 65 was recorded for the existing duct. The CFD results also showed a significant reduction in the pressure drop through the system.

Figure 8 illustrates contours of turbulent dissipation rate (Epsilon) associated with turbulence level. One can see that the rate of the energy dissipation at the 2[nd] bend is reduced significantly and the distribution is more uniform in the modified duct system.

The pressure, maximum velocity, minimum velocity, average velocity and mass flow rate from the CFD model were used to quantitatively determine the effect of changes to the duct and vane structure on the generation of duct related noise. Approximated flow generated noise is calculated based on Bies and Hansen, (1996). In parallel with the CFD design work an acoustic model was also developed using ANSYS software.

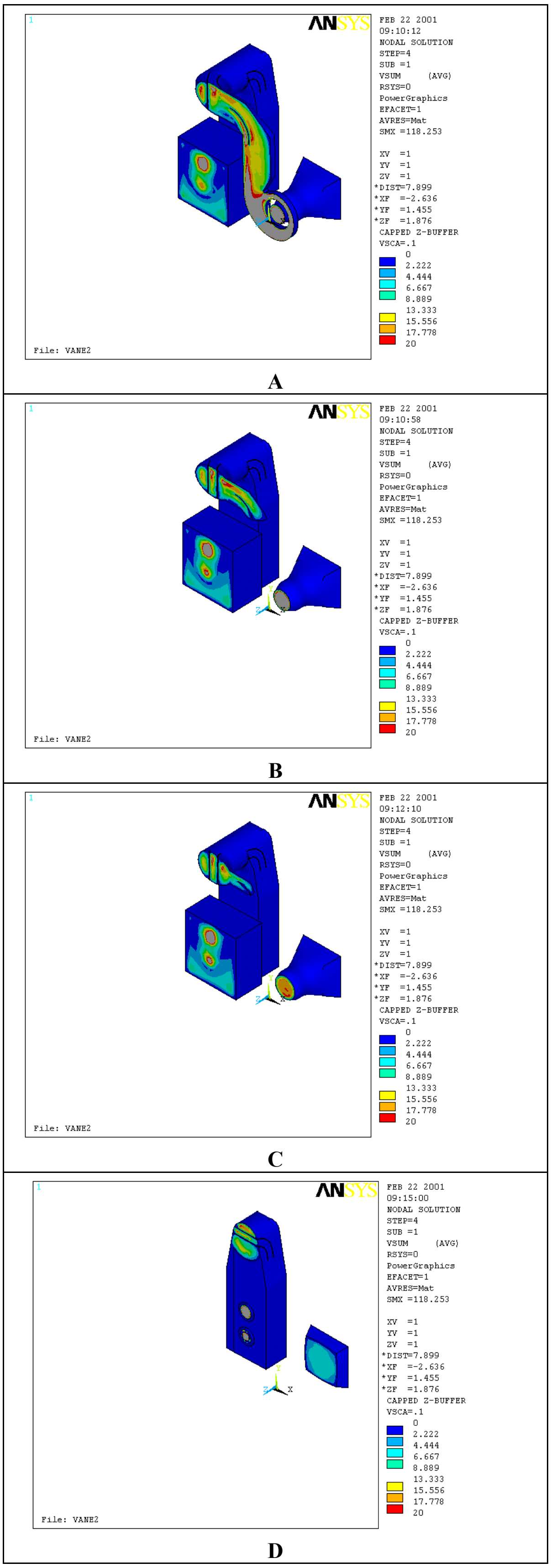


Figure 6: Flow behaviour at different sections inside the modified duct.

For the 60% boiler demand, duct flow noise generation is estimated to reduce by approximately 7 dB as a source with an added benefit of ~ 2 dB to be gained from the shifting of the duct acoustic resonance away from the boiler resonance.

Subsequent testing of modified duct system has validated the approach using CFD analytical tools.

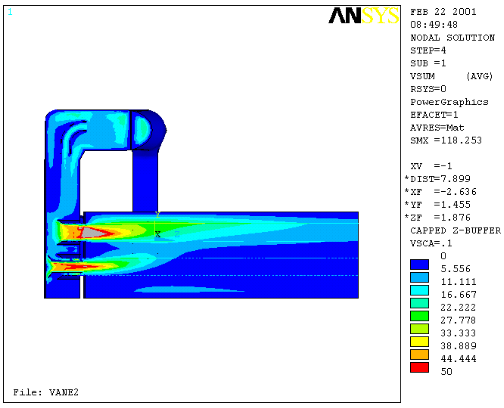
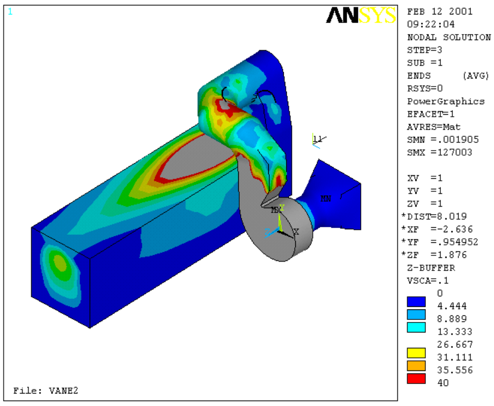


Figure 7: Velocity profile at the register and windbox – modified duct.

Existing Duct



Modified Duct

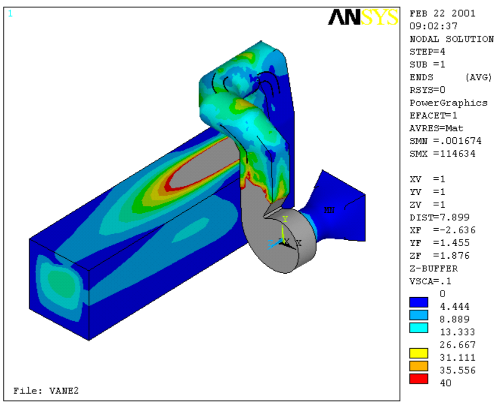


Figure 8: Contours of turbulent dissipation rate.

	Fan Exit	After 1st Turn	Wind box inlet
Modified Duct			
V (Max)	26.7	20	9
V (Min)	-3.1	-1.5	-1.5
V (Average)	9.75	8.8	5.4
Existing Duct			
V (Max)	30	22.2	15
V (Min)	-3	-1.5	-1.5
V (Average)	10.1	9.1	5.5

Table 1: Velocity values at selected sections.

CONCLUSIONS

The experimental and computational investigation provided a satisfactory tool to capture the complex airflow pattern of a number of industrial ducts. CFD analysis provides the ability to quantify flow improvement and to quickly predict the behaviour of several design proposals at low cost. The CFD analysis has offered a comprehensive range of output including velocity distribution, pressure profile and turbulence levels. Subsequent testing of the modified duct system has validated the approach using CFD analytical tools.

ACKNOWLEDGMENTS

The author acknowledges the contribution made to the experimental work by Chris Dimitropoulos

REFERENCES

AL-KHALIDY N. (2001), *P3C Drag Investigation Using Computational Fluid Dynamics*, 504316-TRP-12872-01 Vipac Engineer & Scientists, Commercial-In-Confidence Report, pp. 1-69.

ANSYS-FLOTRAN (2001), *Theory Manual*, USA.

BIES D. and HANSEN C. (1996), *Engineering Noise Control*, 2nd Edition, London SEI 8HN, UK.

LAUNDER B. and SPALDING D. (1974) *The Numerical Computation of Turbulent Flows*, Computer Methods in Applied Mechanics and Engineering, Vol. 3, pp. 269-289.

PATANKAR S., (1980), *Numerical Heat Transfer and Fluid Flow*, Hemisphere, New York.

YAKHOT V. and ORSZAG S. (1986), *Renormalization Group Analysis of Turbulence*, J. Sci. Comput., Vol. 1, No. 3.

YAKHOT V., ORSZAG S., THANGAM S., GATSKI T. and SPEZIALE C. (1992), *Development of Turbulence Models for Shear Flows by a Double Expansion Technique*, Phys. Fluids A, Vol.4, No.7.