Second International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
6-8 December 1999

CFD ANALYSIS OF EROSION OF BIFURCATION DUCT WALLS

M. MANICKAM[1], M. P. SCHWARZ[1] and M. J. MCINTOSH[2]

[1]CSIRO Division of Minerals, Clayton, Victoria, Australia

[2]CRC for Clean Power from Lignite, Mulgrave, Victoria, Australia

ABSTRACT

The flow in the bifurcation duct of a Victorian Power generation boiler plant was numerically modelled. The primary aim was to investigate means for mitigating the erosion of the walls of the bifurcation duct. To achieve this, a model was developed which allowed simultaneous treatment of the gas and particulate phases. The Computational Fluid Dynamics (CFD) code was customised to determine erosion rate caused by particles that hit the duct walls. The model predicted that particles preferentially strike positions that correspond to observed wear patterns namely on the lower half of the duct wall and the lower damper vanes. To reduce duct wall erosion, five modifications to the duct geometry were tested using the CFD model. The focus of these modifications has been to improve particle flow characteristics to mitigate the erosion of the walls of the duct by using extended trailing edges or setting up more baffles in the central zone where the particle traffic is intense. One of the modifications, which involves the introduction of a false wall and extension of trailing edge of the turning vanes, was found to significantly reduce erosion of the duct walls.

INTRODUCTION

The bifurcation duct splits the flue gas flow from the boiler furnace into two streams that pass through parallel flow rotary air heaters. The flue gas carries coarse particles which under certain operating conditions hit the duct walls and other physical surfaces such as damper vanes generating localised high erosion rates that often results in holes formation. It is hard to repair worn surfaces, especially when they occur on duct walls.

The objective is to reduce the occurrence of localised high erosion rates and achieve a reduced and more uniform erosion of the duct wall. In this study a CFD model of the bifurcation duct was developed to investigate the means for mitigating the erosion of the duct walls and evaluate different modifications to the duct geometry.

MODEL DATA

The geometry of the bifurcation is quite complex (see figure 1) but in essence the flow is split into two equal parts through a 90° tee intersection. Turning vanes are incorporated to facilitate the changes in flow direction. Downstream of the turning vanes on each side of the tee is an array of damper vanes, which are normally open.

The gas flow is modelled at the design temperature of 400°C and pressure of 11.25 kPa. The flow at these conditions is 1800 m^3/h.

The particulates that pass through the bifurcation consist of fine ash particles, char and coarser adventitious sand and clay. For the purposes of modelling the wear, it has been assumed that it is the coarser and denser adventitious material that causes duct erosion. Particulate samples taken from the downstream precipitator have been analysed to obtain a size distribution for use in the modelling. Particles are fairly evenly distributed in size, varying from 75µm to 2mm.

NUMERICAL MODEL

Gas flow through the duct was solved using a finite volume method on a non-staggered grid, with the Rhie-Chow algorithm used to prevent oscillations in the pressure field. The turbulent flow field was solved using the Reynolds time averaging for the fluctuating turbulent quantities, which allows splitting of velocity, pressure and other scalars into mean and fluctuating components.

The governing equations of mass and momentum for a steady state and incompressible flow are defined (see Manickam et al., 1998) using the Navier Stokes equations. The velocity-pressure coupling has been effected through the SIMPLEC algorithm (Van Doormal and Raithby, 1984) which provides the basis for updating pressure and correcting velocity components for continuity.

Turbulence is modelled using the standard k-ε turbulence model. The use of the Reynolds Stress turbulence model and Van Leer higher order differencing scheme did not significantly alter the solution.

The flow equations were solved using the general purpose CFD code, CFX-F3D (1995).

In this gas-particle system, where the volumetric flow rate of the gas phase is significantly higher than that of the particulate phase and particle-particle collision rate is therefore small, there is no advantage in treating the particulate phase as continuum. Indeed the representation of the particle phase using the Lagrangian technique is superior since it allows more accurate specification of different characteristics of the particle phase. Hence the Lagrangian technique, in which representative particles with predefined initial starting locations, velocities, directions and compositions are tracked in the computational domain, is employed to model the particle phase. These particles interact with the gas phase through source terms in the mass, momentum and energy equations of the continuum gas phase, a method known as the Particle-Source-In Cell (PSI-Cell) technique (Crowe *et al.*, 1977).

The effect of turbulence in the gas phase on particle motion was modelled using a stochastic method (Gosman and Ioannides, 1981). When particle trajectories hit a wall they are assumed to bounce inelastically with a coefficient of restitution of 0.8.

GRIDDING ISSUES

Complex Geometry

The bifurcation duct model has a complex geometry and is defined using a number of blocks and a non-orthogonal body-fitted mesh. These blocks are 3-dimensional arrays of computational cells and can be joined with each other.

Computation begins at the top of the boiler furnace, assuming a uniform flow field and particle loading across this section, and extends beyond the bifurcation to slightly above the entrance to each rotary air heater as shown by figure 1. Turning and damper vanes (see figure 2) are modelled by the inclusion of dedicated computational blocks shaped to match the curvature of the vanes. The interpolation scheme used for generating the interior computational grids necessitated the use of user FORTRAN routines to accurately define the shape of the curved turning vanes.

A stable numerical solution was achieved using algebraic multi grid (AMG) solver for the pressure correction equation, as the geometry has a complex shape and multiple blocks.

Mesh Sensitivity

In solving the numerical problem, the focus was on accurate prediction of gas and particle flow field in the bifurcation section without unduly increasing the computational size to prohibitively high levels. Initially a typical grid of size 50×18 (defined using multiple computational blocks) was used to represent a horizontal plane in the bifurcation section. The grid was made finer to 70×18 to accurately predict gas and particle flow field variations in a domain where multiple baffles are present. Since further refinements of mesh did not generate any

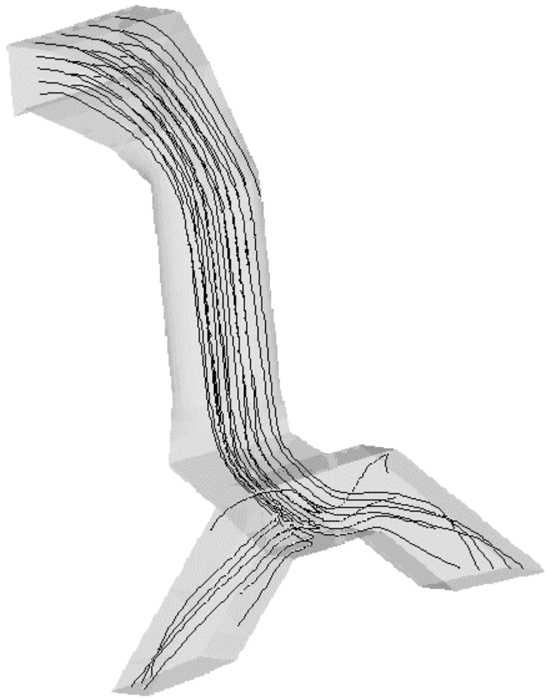

Figure 1: Predicted gas flow field streamlines.

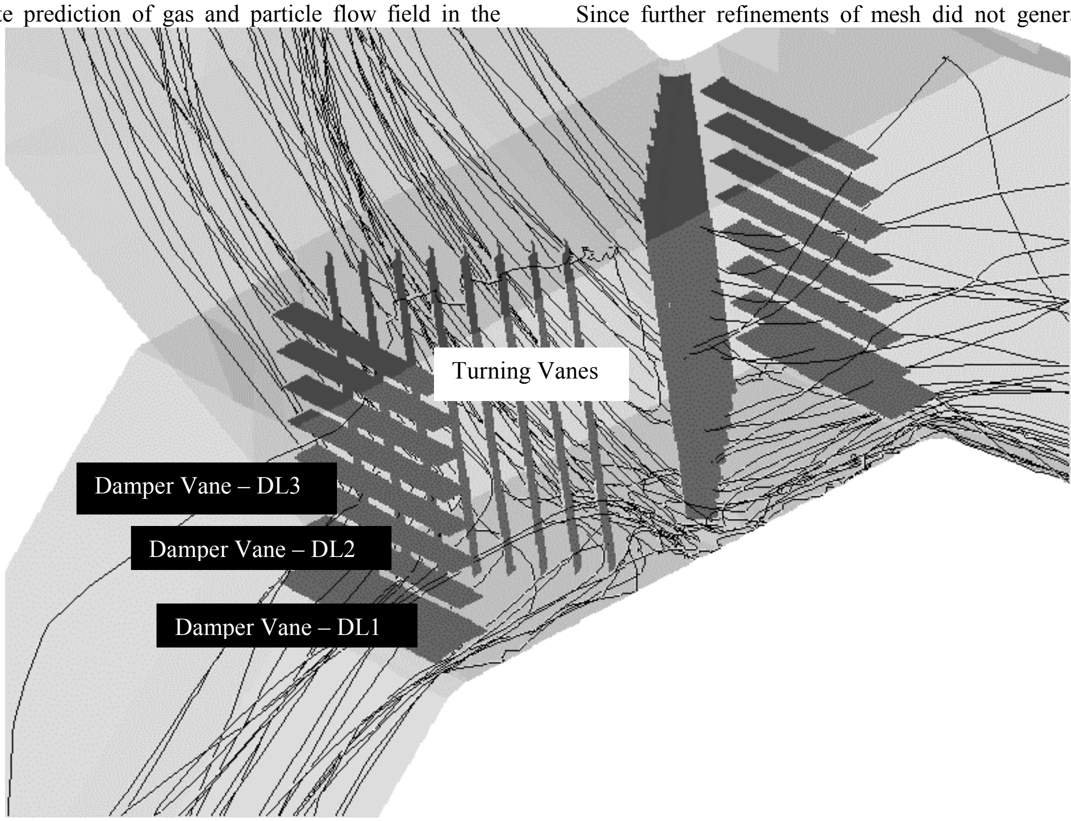

Figure 2: Representative particle tracks - Predicted for the existing design.

significant changes to the solution, a final mesh size of 70 × 18 was used for the model.

The constructed model had 58 blocks of computational cells, subdivided into over 37,000 computational cells.

RESULTS AND DISCUSSION

The predicted flow behaviour is presented in a series of plots that have been generated using the post processor CFX-VISUALISE.

Figure 1 represents the overall flow pattern. Flow velocity streamlines indicate the complexity of the flow field particularly where turning vanes are present. A more detailed view of the flow field is given in figure 2, which also shows computed trajectories of representative particles. The presence of a strong recirculation (not shown) constrains most of the gaseous and particle flow to be close to the floors of the duct.

Figure 2 also shows the surfaces including turning vanes and damper vanes in the bifurcation section as generated using CFX-VISUALISE. The particle flow pattern is nearly symmetric with respect to the centre line of the duct work and indicates that a significant percentage of heavier particles hit and slide along the wall facing the gas stream. The lowest damper vanes also receive many particle hits.

The erosion pattern predicted qualitatively matches the physical observations in figures 3. Figure 3-a shows a section of duct floor and lower damper vanes. The hole observed in the figure is indicative of the erosion suffered by the bottom most damper vane. The wear pattern caused by coarse particles to the duct wall, near the right-hand end of the lowest damper vane, indicates the extent of damage caused by particle erosion. It is hard to repair worn surfaces, especially when they occur on duct walls.

Figure 3-b shows that the wear patterns are similar on both sides of the bifurcation.

The CFD code was customised by writing FORTRAN routines to record information of all the particle hits on the duct walls and vanes. Information gathered includes size of the particle, velocity, location and angle of impingement on the duct walls.

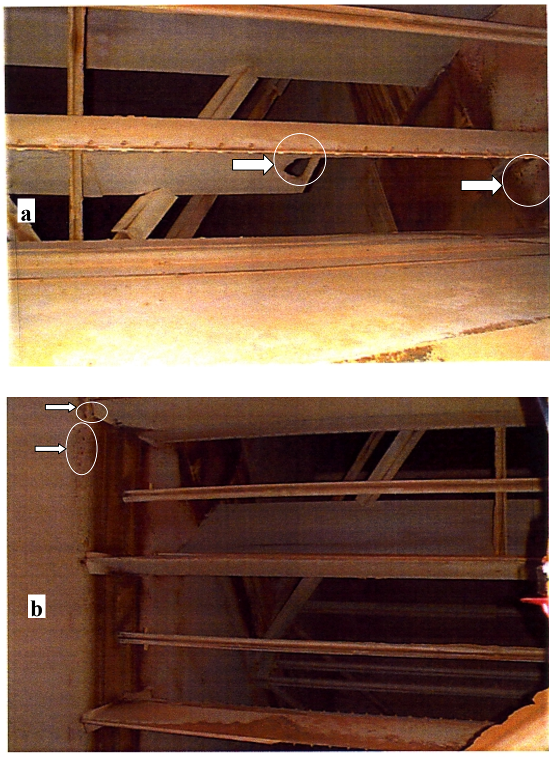

Figure 3: Physical observations of erosion pattern a) on the damper vanes and b) on the duct wall facing the gas flow.

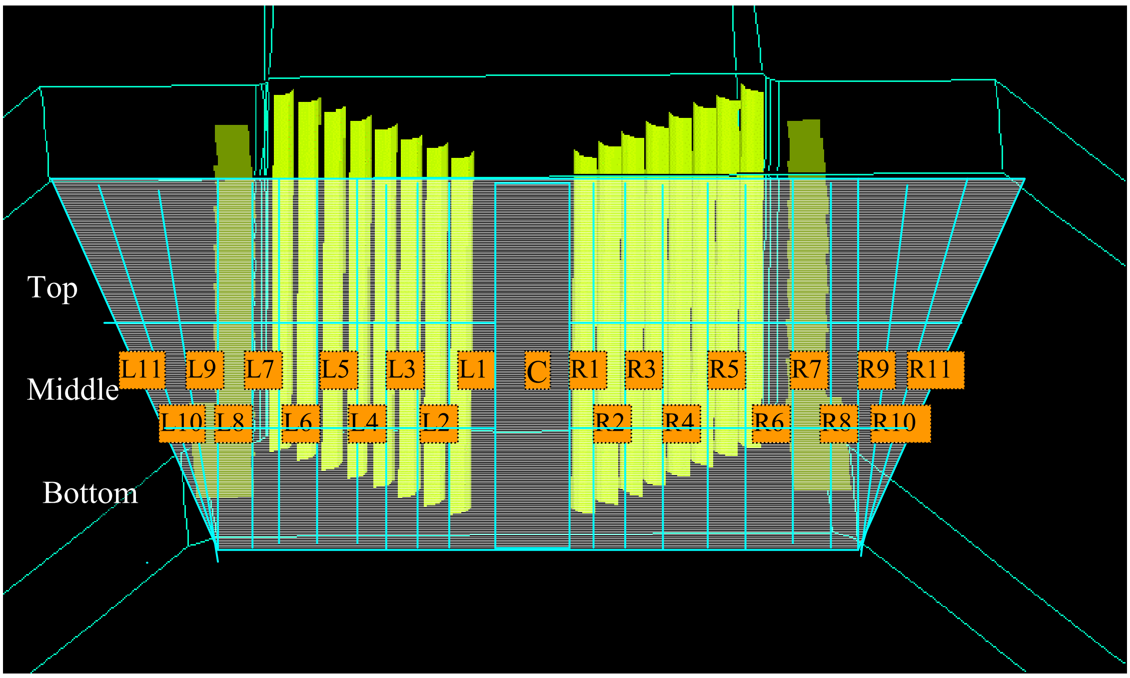

Figure 4: Classification of zones used in analysing the particle hits.

In order to improve the accuracy of the analysis, the number of particle tracks was increased to 3000. The wall at the head of the tee opposing the gas flow was split into various zones marked as L1 to L11, C and R1 to R11 (see Fig 4). The zones L1 to L11 and R1 to R11 are further divided vertically into three zones called Bottom, Middle and Top.

The erosion rate E is defined using Finnie erosion model (1960) as follows:

$$E = \frac{C m_P |v_P|^2}{P} f(\alpha) \qquad (1)$$

where C is an experimental constant
m_p is the mass of the particle
v_p is the velocity of the particle
P is the hardness of the wall,
and

$$f(\alpha) = \sin(2\alpha) - 3\sin^2(\alpha), \quad \alpha \leq 18.4°$$

$$f(\alpha) = \frac{\cos^2(\alpha)}{3}, \qquad \alpha > 18.4° \qquad (2)$$

where α is the angle of impingement.
For the bifurcation duct, C, and P are constant. For each of the zones, grouping the variables m_p, $f(\alpha)$ and v_p, an erosion factor is defined as

$$EF = \frac{\sum m_P f(\alpha) |v_P|^2 \times 10^5}{area} \qquad (3)$$

where the sum is over all particles impacting the zone under consideration.

The erosion factor EF measures the extent of erosion at each zone of the duct wall, by taking into account the particle's size, velocity and angle of impingement. From the data gathered for the particle hits, the erosion factor *EF* was determined.

A plot of variation of erosion factor across the bottom duct wall zones is shown in figure 8. The erosion potential is predicted to be high at the zones L5 to L8 and R5 to R8 as shown in the chart. A maximum erosion factor of 526 was predicted at R8 on the right side and 462 at L6 on the left side.

Significant numbers of the particles are deflected by the turning vanes to hit the zones L8-L10 / R8-R10. An extension of trailing edge of the turning vane might possibly deflect the particles towards the core of the gas stream, avoiding the hit on the duct wall. A high percentage of particles hit the central zone C and bounce back to hit duct walls at L5-L8 / R5-R8, with an impact angle of around 20°. According to the Finnie Erosion model, the erosion rate is a maximum for an impact angle of 18.4°. A possible solution might be to add another turning vane near the Central zone which could deflect the particle into the gas stream instead of hitting the duct wall.

OPTIONS FOR MINIMISING WEAR

Given the complexity of the flow field, erosion of the duct wall is hard to avoid. Introduction of suitable mechanisms that alter the particulate and gas phase flow characteristics can possibly minimise erosion at places in the duct wall where the rate of erosion is at a maximum and substantially delay the formation of holes on the duct wall. The objective is to protect the duct wall from erosion by reducing the level of erosion at places which experience maximum erosion rate and achieving a uniform and minimised overall erosion of the duct wall.

The particles with higher *EF* values are generally larger than 600µm and travel with sufficient momentum that they are not influenced greatly by the gas flow field. The focus should be on setting up of mechanisms to deflect the particles towards the core of the gas stream and to reduce the erosion caused by the impinging particles rather than tuning or modifying the gas flow field. The possible alterations to the current geometry can be:

- Introduction of additional turning vanes close to the duct wall
- Introduction of a false wall at the Central zone of the duct wall and
- Extension of the trailing edge of the turning vanes.

All three modifications involve additional components, which are easily replaceable and are less expensive than repairing an eroded duct wall. These modifications were implemented in the model as the following five variations to the existing model:
Basic geometry variations:
1. Additional turning vanes
2. False wall
3. Extended trailing edge (for the turning vanes)

Combinations of basic geometry variations:

4. Additional turning vanes and extended trailing edge
5. False wall and extended trailing edge

Figure 5 shows the new geometry after introducing an additional turning vane near the duct wall. Figure 6 shows the model geometry after introducing a false wall in the form of a rectangular block in the Central zone. In the Extended Trailing Edge configuration, the trailing edge is extended by adding a flat strip parallel to the axis of the bifurcation section along the main gas stream flow direction as shown in figure 7.

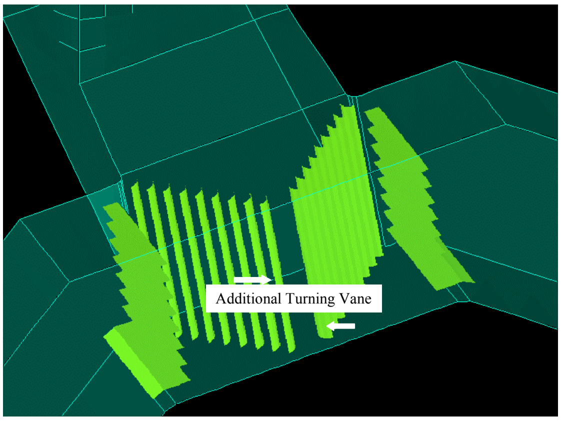

Figure 5: Model geometry for Additional Turning Vanes configuration.

Comparison of Erosion factor - Basic Geometric Variations

Fig 8 compares the erosion factor values for the three basic geometry variations. The erosion is predicted to be less near the vicinity of the central zone C, at the zones L1

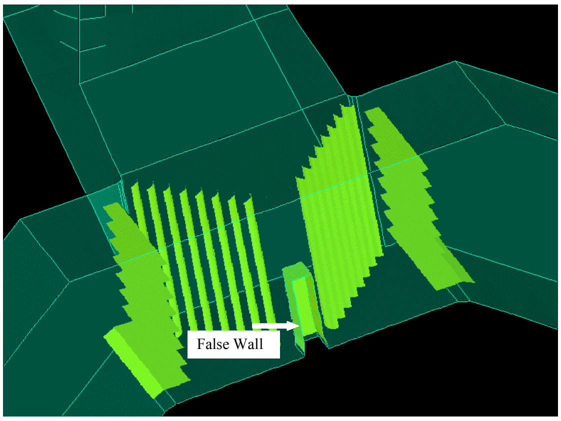

Figure 6: Model geometry for False Wall configuration.

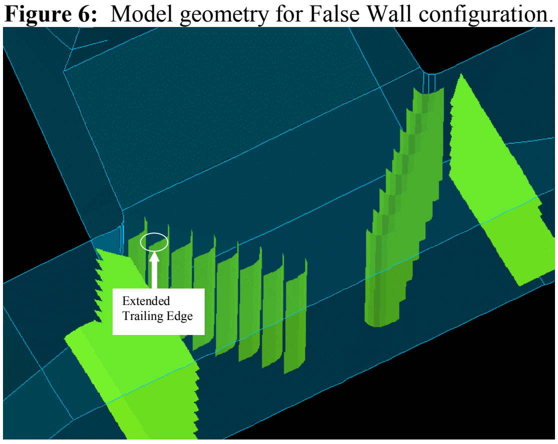

Figure 7: Model geometry for Extended Trailing Edge configuration.

-L4 and R1-R4, for the *Additional Vane* and *False Wall* configurations. For the zones L5-L11 and R5-R11, farther away from the central zone C, the *Extended Trailing Edges* configuration is predicted to generate comparatively less erosion. All the three basic geometry variations failed to reduce maximum erosion significantly or generate a uniform pattern of erosion.

Comparison of Erosion factor – Combinations of Basic Geometric Variations

Fig 9 compares the erosion factor values predicted for the combinations of basic geometry variations. The configuration with *False Wall and extended trailing edges* is predicted to reduce erosion by 69 to 71% at the zones L6 and R8 where the maximum rates of erosion are predicted for the existing configuration. In addition, the erosion of the duct wall for this configuration is fairly uniform. The maximum erosion predicted for this configuration occurs at the zones L3 and R3 which is an increase of 32-43% over the corresponding values predicted for the existing design. However, the maximum predicted erosion rate is reduced by roughly 36%. In addition, the expected erosion of the duct wall at the Middle and Top zone is negligible (not shown).

Comparison of Number of Hits on the Damper Vanes for different Configurations

The number of particle hits on the damper vanes is compared for different configurations as shown in Table 1. Extension of the trailing edges decreased the number of hits on the lowest damper vanes (DL1) by 24 to 46% and almost eliminates the particle hits on the rest of the damper vanes. The reductions achieved in the number of hits is substantial with the False Wall and extended trailing edges configuration where reductions of 38 to 46% are predicted.

CONCLUSIONS

The two phase flow through the bifurcation duct of a Victorian Power generation boiler plant has been numerically modelled. The positions at which larger particles were predicted to hit the duct surfaces were found to correlate with the observed wear. The CFD code was customised to determine erosion rate of all particles which hit the duct walls.

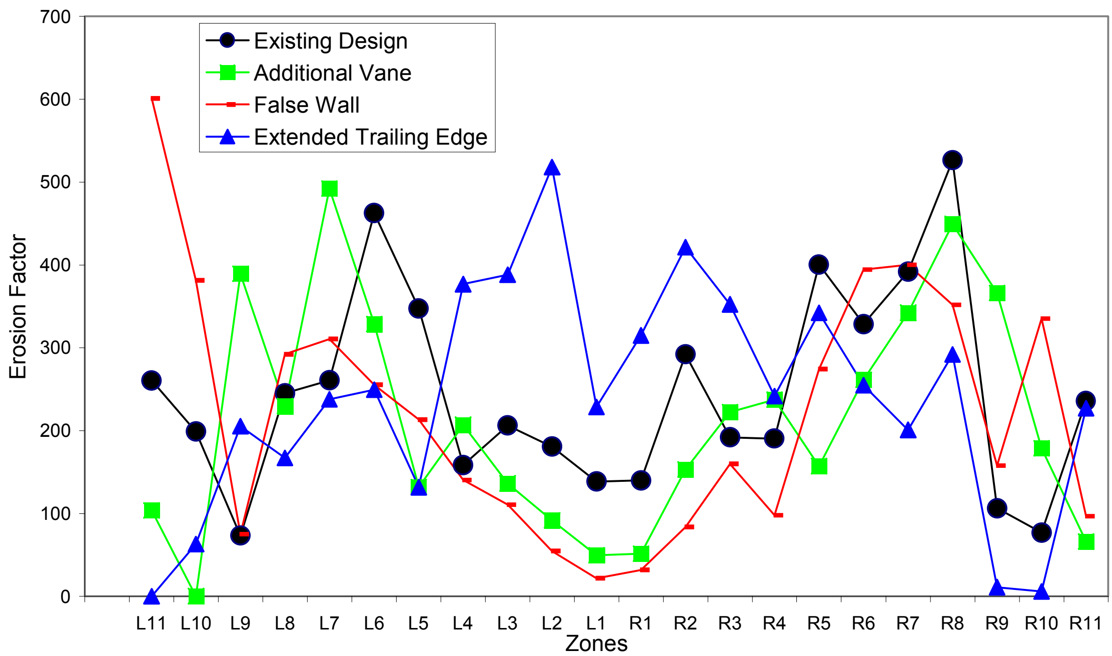

Figure 8: Comparison of erosion factor - Basic geometric variations.

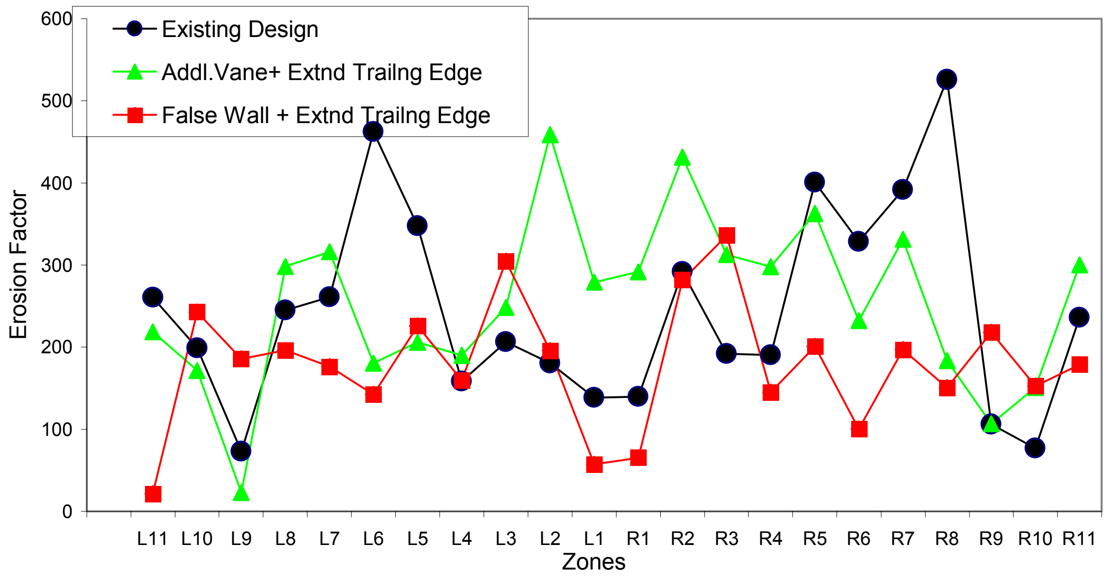

Figure 9: Comparison of erosion factor – Combinations of basic geometric variations.

Configurations	Left Damper Vanes			Right Damper Vanes		
	DL1	DL2	DL3	DR1	DR2	DR3
Existing Design	360	36	33	412	38	34
Additional Vane	373	45	36	389	44	28
False Wall	332	64	34	320	48	33
Extended Trailing edge	230	0	1	274	0	0
Additional Vane + Extended Trailing Edge	267	0	0	314	0	0
False Wall + Extended Trailing Edge	221	0	0	222	0	0

Table 1: Comparison of number of hits on the damper vanes for different configurations.

The gas flow field is complex and the particulate phase contains a considerable number of large particles, which travel with high momentum: erosion of the duct wall is therefore hard to avoid. The objective is to reduce the occurrence of localised high erosion rates and achieve a reduced and more uniform erosion of the duct wall.

Five options for modifying the bifurcation were tested in the CFD model for their influence on expected wear. One of the options, which involves the introduction of false wall and extension of trailing edge of the turning vanes, was found to significantly reduce the maximum erosion rate. The maximum representative erosion factor predicted for this configuration is less by approximately 36% of the maximum value predicted for the existing design and the erosion overall is more uniform. This configuration also reduces the number of particle hits on the lower damper vanes. As a consequence this configuration involving false wall and extended trailing edges would be expected to offer a longer service life than the current configuration.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the support for this work by the Cooperative Research Centre For Clean Power From Lignite which is funded in part by the Cooperative research Centres program of the Commonwealth of Australia.

REFERENCES

CFX, (1995). CFX 4.1 Flow Solver User Guide, Computational Fluid Dynamics Services, Harwell, United Kingdom.

CROWE, C.T., SHARMA, M.P. and STOCK, D.E., (1977). "The Particle-Source-In Cell (PSI-CELL) Model for Gas-Droplet Flows", *Trans. ASME, J. Fluids Engineering*, **99**, 325.

FINNIE, I., (1960), "Erosion of Surfaces by Solid Particles", *Wear*, **3**, 87-103.

GOSMAN, A. D., and IOANNIDES, E., (1981). "Aspects of Computer Simulation of Liquid-Fueled Combustors", *AIAA Paper No*. 81-0323.

MANICKAM, M., SCHWARZ, M. P., and PERRY J. H., (1998). "CFD Modelling of Waste Heat Recovery Boiler", *Applied Mathematical Modelling*, Volume 22 No. 10, 823-840.

VAN DOORMAL, J.P., and RAITHBY, G.D., (1984). "Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows", *Numerical Heat Transfer*, **7**, 147-163.