Journal of Industrial Technology Vol. 14, 2005, 87-96

COMPUTATIONAL FLUID DYNAMICS SIMULATIONS OF FLOWS AND PRESSURE DISTRIBUTIONS IN A 96 MW COMBINED CYCLE DIVERTER DAMPER

Mohd. Zamri Yusoff[1] and Zainul Asri Mamat[2]

[1]Department of Mechanical Engineering, College of Engineering
Universiti Tenaga Nasional
43009, Kajang, Selangor.
[2]Generation and Energy Technology Unit, TNB Research Sdn. Bhd.
(zamri@uniten.edu.my)

RINGKASAN : *Peredam pengalih aliran (diverter damper) selalu digunakan untuk mengalihkan aliran gas samada ke alur keluar altenatif atau memasuki sistem melalui salur masuk altenatif. Untuk janakuasa kitar padu, peredam pengalih aliran digunakan untuk mengarahkan gas ekzos daripada tarbin gas samada ke dalam dandang perolehan haba sisa atau apabila beroperasi dalam kitar terbuka keluar secara terus melalui paip tumpu pirau. Peredam juga membolehkan aliran gas ke dalam dandang di kawal mengikut sukatan. Kertas kerja ini menerangkan simulasi Pengiraan Dinamik Cecair (CFD) yang telah dijalankan untuk mengkaji bentuk aliran di dalam peredam pengalih aliran pada sudut bukaan yang berlainan. Tujuan analisis ini ialah untuk menentukan daya yang dikenakan oleh aliran gas kepada bilah peredam pada sudut bukaan yang berbeza. Data-data taburan tekanan di atas bilah peredam akan digunakan untuk analisis kekuatan struktur. Keputusan simulasi menunjukkan terdapat daya tekanan yang tidak seimbang ke atas permukaan bilah peredam. Aliran masuk yang tidak simetri, telah menyebabkan kewujudan kecerunan halaju dan seterusnya menyebabkan tekanan yang tidak seimbang. Tekanan tertinggi dibentuk di bahagian penjuru bawah kanan bilah peredam berkenaan terutama sekali pada sudut bukaan yang kecil. Keadaan ini mungkin menyebabkan terbentuknya momen kilasan pada bilah peredam.*

ABSTRACT : A diverter damper is commonly used where the gas is either to be directed to alternate outlets or enters the system from alternate inlets. For combined cycle power plant, the diverter damper is used to direct exhaust gas of the gas turbine either into the waste heat recovery boiler or, when running under open cycle mode, exits directly to the by-pass stack. The damper also allows control of the proportion of flow entering the boiler. This paper describes Computational Fluid Dynamics (CFD) simulations that have been performed in order to study the flow pattern inside the diverter damper at various opening angle. The objective is to determine the net force exerted by the exhaust gas on the damper at various opening angle. Data on the pressure distribution on the damper will be utilised for structural loading calculations. Examination of the pressure distribution on the face of the damper showed that there is a non-uniform force across it. The inlet flow is not symmetrical and this causes a velocity gradient across the flow, which results in a pressure gradient. The pressure is highest at the bottom right hand corner of the damper, especially when the opening angle is very small. This may be significant in generating a twisting moment on the diverter damper blade.

KEYWORDS : CFD, diverter damper, combined cycle power plant, failure analysis

INTRODUCTION

Combined cycle power plants are among the most efficient power generating plants today (Franco and Casarosa, 2002). In Malaysia, currently about 40% of the total electrical energy demands are generated by combined cycle power plants (Shafie, 2003). One of the most critical components in a combined cycle power plant is the diverter damper. Diverter damper linked the gas turbine topping cycle with the steam turbine bottoming cycle and allows more flexibility in the operation. With the diverter damper, the gas turbine can operate in open cycle mode if required and the exhaust gas from the gas turbine can be directed to the by-pass stack. This would allow maintenance to be done in the waste heat boiler section, without requiring the whole block to be shut down. The diverter damper, also allows exhaust mass flow regulation that enters the waste heat boiler, which is important, during the plant start up.

The flow characteristic in a diverter damper is complex, being three-dimensional and with multiple recirculating regions (Bell and Nitzhen, 2003). The flow could also be unsteady due to the periodic vortex shedding from the diverter damper blade especially at small angle of opening (Yusoff *et al.*, 2001). The hot gas flows in the diverter also exclude direct flow measurements. In order to study the flow characteristics in the diverter damper, multidimensional modelling such as Computational Fluid Dynamics (CFD) is required.

The present paper describes a study that has been undertaken on a particular type of diverter damper that has failed during service (Mamat, 2000). The purpose of the study is to investigate the flow pattern inside the diverter damper at various opening angle and in particular the aerodynamic pressure loading experience by the damper due to the hot exhaust gas flow. The results obtained from study are used to analyse the possible cause of the failure and to suggest possible modifications.

The diverter damper is subjected to two types of loading. Firstly, the static load due to its own weight which needs to be supported by the shaft assembly. This load presumably will not cause failure. Secondly, the load due to the exhaust gases from the gas turbine. This load can be further divided into two. Firstly, thermal load due to the high temperature of the exhaust gases. This will cause expansion of the damper assembly, resulting in significant magnitude of stresses, if it is not allowed to expand freely. Secondly, the pressure load on the blade of the diverter damper, due to the flow of the exhaust gases. The magnitude and direction of the load will vary based on the flow velocity and angle of the damper. The flow rate, velocity and temperature of the exhaust gas will vary depending on the gas turbine operating characteristics and this will result in dynamic load on the diverter damper blade. Most failures happen due to dynamic loads, rather than static loads. These failures typically occur at stress level significantly lower than the yield strengths of the materials and commonly known as "fatigue failure". There is also a possibility of failure due to vibration. This can occur if the forcing frequency falls close to one of the fundamental frequencies of the diverter damper blade assembly. If this occurs, the structure will resonate and this will lead to failure.

In order to investigate the failure mechanisms of the diverter damper, CFD analysis, stress analysis, fatigue analysis and modal vibration analysis need to be performed. This paper will only focus on the computational fluid dynamics analysis that has been done on the damper, in order to determine the force exerted on the damper due to the exhaust gas flow.

DESCRIPTIONS OF THE DIVERTER DAMPER

The diverter damper comprises a main frame consisting of flanged and a braced box assembly (known as the plenum chamber) fitted with an internal blade assembly and external drive system. The inside of the plenum chamber is insulated with 4 in. of ceramic fibre material, which is then covered with stainless steel cladding sheets. The diverter is installed in the horizontal ducting system and is a double drive type. Torque is applied instantaneously to both blade stub shafts by the actuation system. The limit switches contained within the actuator are used to control the fully open and closed position of the blade. Exploded view of the diverter damper and its driving mechanisms are shown in Figure 1.

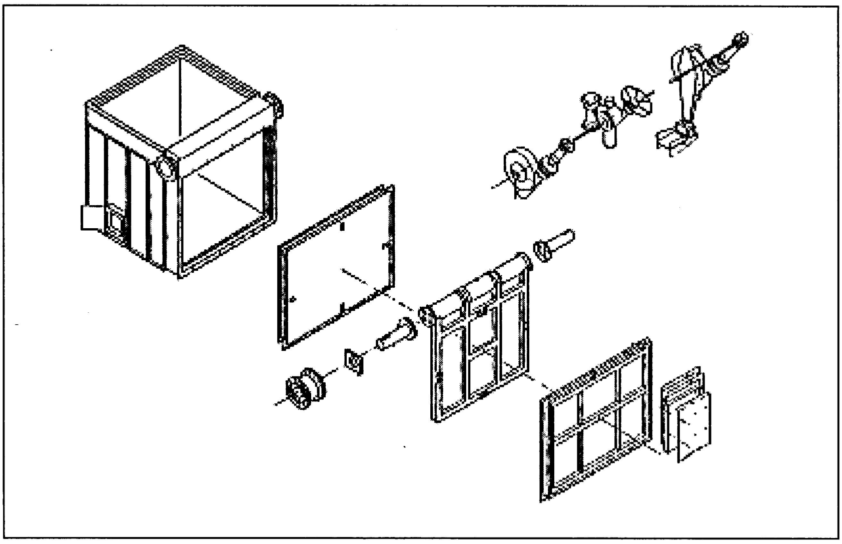

Figure 1. *Exploded view of the diverter damper*

The diverter damper assembly consists of a blade arm structural assembly, on each side of which is independently mounted blade. Bolted around the edge of each blade are two sets of sealing elements. The blades are arranged so that two sets of seals operate when the blade assembly is in the "closed to boiler inlet" position and the other two when the assembly is in the "closed to by-pass" position. Figure 2 shows the schematic diagram of the complete system consisting of diverter damper, the gas turbine exhaust gas outlet, by-pass stack and the Heat

Recovery Steam Generator (HRSG) inlet ducting. In normal operation, the diverter damper can be regulated as follows :

- Between 0° - 15° angle: Sudden opening (severe vibration can be observed in this region).
- Between 15° - 75° angle: Opened gradually depending on the amount of heating required to warm up the boiler.
- Between 75° - 90° angle: Sudden opening.

Table 1 gives the design specifications of the diverter damper (Tenaga Nasional Berhad, 1995). Table 2 gives the plant typical operation data.

Table 1. Design Data of the Diverter Damper (Tenaga Nasional Berhad, 1995)

Duct Details	
Size	5466 mm x 5466 mm (width x height)
Cross Sectional Area	29.877 sq. meters
Face to Face Dimensions	7214 mm maximum
Operating Conditions	
Gas Temperature	621°C
Gas Velocity	38 m/s at base load
Gas Flow Rate	405.4 kg/s at 542°C
Gas Type	Gas Turbine Type Frame 9E Exhaust
Frequency	Daily shut down/up of HRSG
Pressure Drop to Boiler	7 mm H_2O at maximum flow
Pressure Drop to By Pass	30 mm H_2O at maximum flow
Materials	
Frame	Mild steel
Blade Arms	2.25 Cr 1.0 Mo
Blades	1.0 Cr 0.5 Mo
Stub Shaft	AISI 316 S31
Seals	Inconel 625°
Bar Work	1.0 Cr 0.5 Mo
Insulation Cladding	Stainless steel AISI 409, 1.6 mm thick in shaft
Internal Insulation	Fibrefrax, 101.6 mm
Insulation Density	64 kg/m^3

Table 2. Plant Operational Data (Tenaga Nasional Berhad, 1995)

Item	**Operational Data**
Gas Turbine Load	96.2 MW
Temperature of Gas Turbine Exhaust	565°C
Gas Temperature at Inlet of HRSG	527.2°C
Gas Temperature at Exit of HRSG	144.9°C
Gas Pressure at Inlet of HRSG	1.2 mm H_2O gauge
Gas Pressure at Exit of HRSG	-3.2 mm H_2O gauge

COMPUTATIONAL FLUID DYNAMICS SIMULATIONS

The flow was simulated using the CFX5 Computational Fluid Dynamics (CFD) package (AEA Technology, 1999). It uses a coupled solver based on finite-volume discretisation and unstructured grid methodology to solve governing fluid flow conservation equations within the flow domain. The equations solved are the Reynold's Averaged Navier-Stokes (RANS) equations together with the k-ε model to represent turbulence. The standard constants were used in the turbulence model.

Model Assumptions

The geometry representing the gas turbine outlet duct, the diverter damper, boiler duct and by-pass duct, as shown in Figure 2 was created using CFX-Build pre-processor. Following are the assumptions made in performing the analysis :

- *Diverter Damper* - Figure 3 shows the geometry of the damper. It was represented as a solid within the flow domain.

- *Dimensions* - All the dimensions required to build the geometry were obtained from the plant working drawings and some rough estimates from the actual plant. From the basic dimensions a wire frame model is constructed. Then, a B-Rep (Boundary Representation) solid that surrounded the flow domain is generated. A gap of 30.5 cm from the top wall and 3.8 cm from the side-wall were used. The thickness of the damper was set to be 36.5 cm.

- *Isothermal and incompressible flow* - Once the geometry was constructed, a flow domain was set. As the temperature variation within the flow domain is very small (~ 2°C), it is adequate to assume that the flow is isothermal and incompressible.

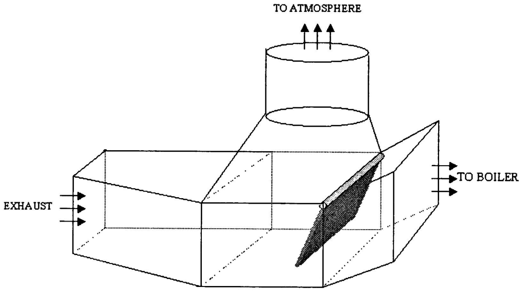

Figure 2. *Schematic of the damper system, showing the exhaust from the turbine, the damper and the exit paths to the boiler and atmosphere.*

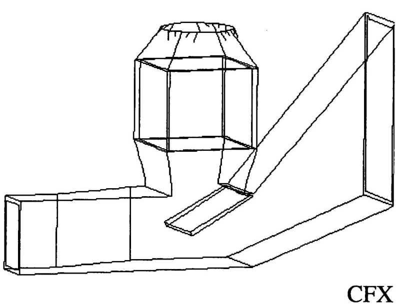

Figure 3. *The Geometry Used in the CFD Calculation (60 Degree Angle of Opening)*

- *Exhaust gas properties* - The gas was assumed to have typical properties of gas turbine exhaust gas at atmospheric pressure and at a constant temperature of 830 K (556°C). The data were obtained from Keenan and Keyes (1984).

- *Silencer* - The silencer, which is located in the square section in the stack outlet, was modeled as a porous media. This was represented by a sub-domain, with a quadratic momentum resistance. Data from the plant indicated that the pressure drop across the silencer is 120 Pa/m for a flow speed of 36 m/s. This was equivalent with a quadratic resistance coefficient of 0.92 kg m^{-4}.

Boundary Conditions

Boundary conditions need to be set at all boundaries, which are wall boundaries, inlet boundary and exit boundaries. All wall boundaries were set to be smooth and adiabatic. This is appropriate as all walls are insulated. The inlet and outlet boundary represent the position of the flow coming and leaving the diverter damper. The boundary conditions at inlet and outlets are :

- Inlet :
  - The mass flow rate of gas was set to the measured value of 404 kg/s.
  - The turbulence level in the flow was assumed to be that of fully developed flow.
- Outlets :
  - At the by-pass stack outlet, an outlet boundary condition was set with the static pressure set to a zero gauge pressure.
  - At the boiler inlet, the opening boundary was set, which allows flow to enter and exit the domain, and the gauge pressure was also set to zero. This was necessary because of the large recirculation zone downstream of the damper.

Mesh Generation

Generating a good mesh for a particular flow situation is extremely important in CFD. In the present analysis, unstructured tetrahedral mesh was used. The mesh was constructed using CFX5. Firstly, a surface mesh was generated over all B-Rep surface boundaries. The maximum edge length for the surface mesh was set at 61 cm. On all walls inflation meshing was set with 5 layers of prisms and a maximum thickness of 30.5 cm. This ensures higher mesh concentration near the walls on which to apply no-slip boundary conditions and to resolve the boundary layer.

High mesh concentration was also set between the walls and the damper. This was achieved by setting a line mesh control along the edges of the damper, with elements of 9.1 cm extending for 9.1 cm and then increasing in size via a geometric factor of 1.3. Figure 4 shows the unstructured mesh generated. Visual inspection of the mesh showed that the mesh is adequate to resolve the flow field. A volume mesh was then generated.

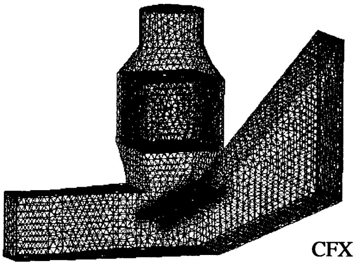

***Figure 4.** The Unstructured Mesh Used (60 Degree Angle of Opening)*

Solver Parameters

The solver parameters used were the default settings in CFX5 with automatic time-step selection. For all cases, 60 time steps were required to achieve converged solution. At convergence the solutions global mass tolerance reduced to almost machine zero and the global momentum balances to a fraction of a percent. The average residuals were typically 0.0001. The range of y^+ values showed that the turbulence boundary conditions had been correctly applied at the walls. The calculations were performed for damper opening angles of 0°, 15°, 30°, 60°, 75° and 90°. Angle 0° represents fully close position and angle 90° represents fully open position.

RESULTS

Figures 5a, 6a, 7a, 8a and 9a show the streamlines plots along the mid-plane of the geometry at opening angles of 0°, 30°, 60°, 75° and 90°, respectively. Figures 5b, 6b, 7b, 8b and 9b show the corresponding angles surface plot of pressure on the damper. It can be observed that under all conditions there always exists a small recirculation zone at the entrance to the silencer in the stack by-pass outlet. The magnitude of the recirculation zone increases when the damper

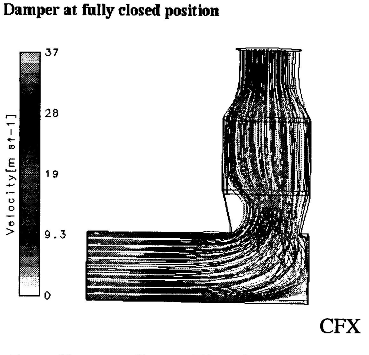

***Figure 5a.** Streamlines Plot for 0 Degree Angle of Opening*

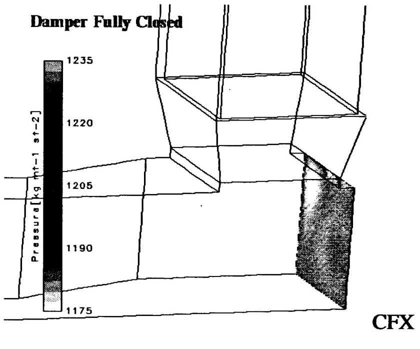

***Figure 5b.** Blade Static Pressure Plot for 0 Degree Angle of Opening*

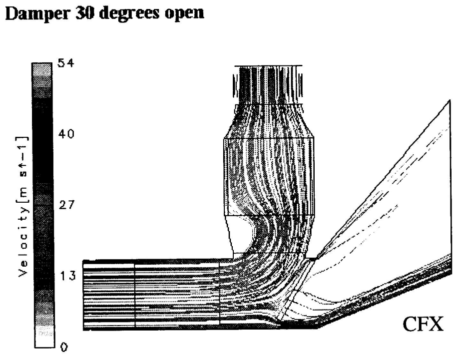

Figure 6a. Streamlines Plot for 30 Degree Angle of Opening

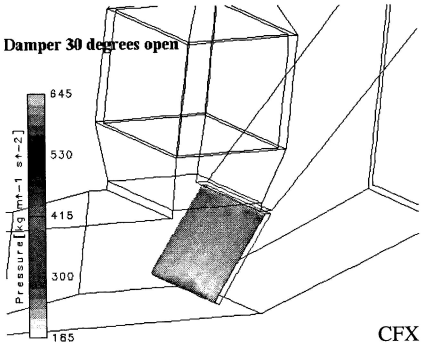

Figure 6b. Blade Static Pressure Plot for 30 Degree Angle of Opening

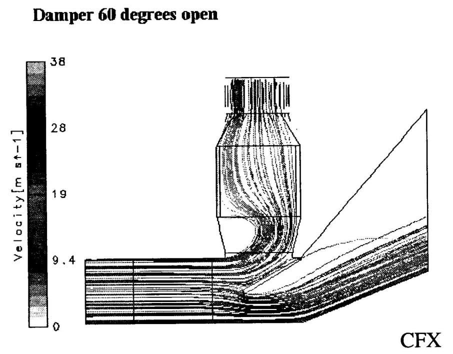

Figure 7a. Streamlines Plot for 60 Degree Angle of Opening

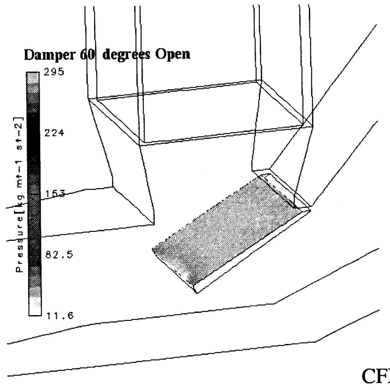

Figure 7b. Blade Static Pressure Plot for 60 Degree Angle of Opening

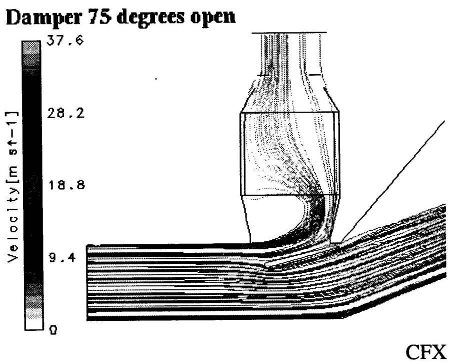

Figure 8a. Streamlines Plot for 75 Degree Angle of Opening

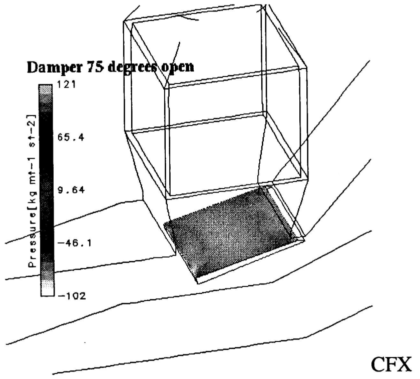

Figure 8b. Blade Static Pressure Plot for 75 Degree Angle of Opening

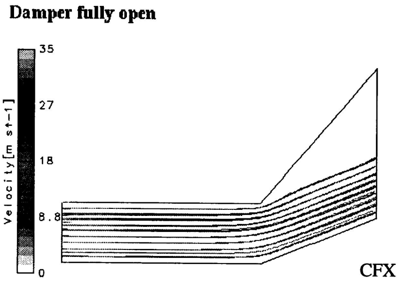

Figure 9a. *Streamlines Plot for 90 Degree Angle of Opening*

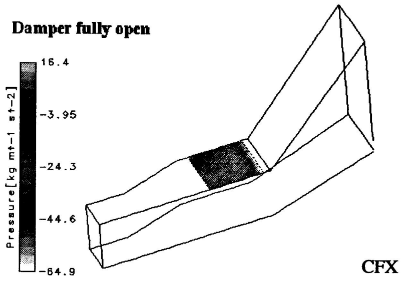

Figure 9b. *Blade Static Pressure Plot for 90 Degree Angle of Opening*

opening angle increases except for the fully open position. Whenever there is flow into the boiler there is a large recirculation zone present behind the damper. This will lead to a significant pressure loading on the diverter damper blade. This recirculation zone disappears when the damper is fully open, but the flow distribution to the boiler is very poor. It is interesting to observe that when the damper is fully closed there is a small recirculation zone at the bottom of the damper. Similar observations have been reported by Bell and Nitzhen (2003).

It can be observed that the pressure distribution on the face of the damper is non-uniform. The inlet flow is not symmetric and this causes a velocity gradient across the flow, which results in a pressure gradient. The pressure is highest at the bottom right hand corner of the damper, especially when it is only slightly open. This may be significant in generating a twisting moment on the blade. In general, the simulations show significant pressure forces acting on the damper. In addition, they show that these forces vary across the plate because of the asymmetry of the geometry. The total normal pressure forces on the plate as calculated by CFX5 are given in Table 3. The data in the table show that, as expected, the net force on the plate is such that it acts to close the damper; that is it acts downwards and towards the boiler. These data are used as inputs into the structural calculations.

Table 3. Total pressure force components on the damper

Angle	Front or Back	Pressure force in horizontal direction (N)	Pressure force in vertical direction (N)
0	F	38,600	0
15	F	24,200	-7000
	B	980	-260
30	F	14,700	-893
	B	1,480	-882
60	F	2,300	-4,200
	B	1,600	-2,600
75	F	352	-1,400
	B	574	-2,000
90	B	0	-860

CONCLUSION

CFD simulation of flows in the diverter damper has been successfully done. The model constructed includes the effect of the damper on the flow and leakage around the top and sides of the damper. In addition, the silencer in the stack has been modeled and inclusion of this was found to be necessary to provide the correct flow conditions in the stack. The results obtained show that the force acting on the damper blade is not symmetric and this may generate significant twisting moment on the blade. The asymmetric distribution is due to the asymmetric shape of the duct leading to the diverter damper, which causes a velocity gradient across the flow and hence results in a pressure gradient. This pressure gradient can be avoided if the inlet ducting has been designed as symmetrical. In addition to providing information on the pressure loading on the damper, the flow field simulations have shown that there is very poor flow distribution downstream of the damper, even when it is fully open. CFD analysis alone is not sufficient to determine the exact cause of the failure. The data obtained from the CFD analysis will be used in the structural and vibration analysis.

REFERENCES

AEA Technology Plc. (UK). (1999). CFX5 Solver User Manual.

Bell, M.B. and Nitzhen, J.A. (2003). Controlling steam production in heat recovery steam generator for combined cycle and enhanced oil recovery operation. *Proceedings of the Power-Gen International Conference*, Las Vegas, Nevada, Dec 9-11.

Franco, A. and Casarosa, C. (2002). On some perspectives for increasing the efficiency of combined cycle power plants. *Applied Thermal Engineering*, **22(13)**; pp1501-1518

Keenan and Keyes. (1984). *Gas Tables*, John Wiley and Sons Inc.

Mamat, Z.A. (2000). Study on WAHLCO heat recovery steam generator diverter damper failure. TNB Research Project No. TNB 148/99 Final Report.

Shafie, H. (2003). National Energy Balance, Malaysia, Quarter 3&4. Ministry of Energy, Water and Communications, Malaysia.

Tenaga Nasional Berhad, (1995). Diverter Damper MS-9000 Service Manual. *Gas Turbine Conversion Project, GEK-10773, Volume No. 1C,* Turbine Nos. 296364 & 296369.

Yusoff, M.Z., Leong, C.W., Boosroh, M.H, Hussein, I. and Mamat, Z.A. (2001). CFD simulation of flow induced vibration in a combined cycle diverter damper. *Proceedings of the Brunei International Conference on Engineering and Technology*, Oct. 8-11.