03.31.2020

BERKELEY PUBLICATION
AND RESEARCH INTERNATIONAL

Journal of Pure and Applied Science Vol. 12 No. 9

A COMPUTATIONAL FLUID DYNAMIC FLOW ANALYSIS: A SIMULATION OF NATURAL GAS DELIVERY SYSTEM LEADING TO VENT AND GAS FLARE CHAMBER

UMAR MU'AZU TADAMA[1], AHMAD ABDURRAZZAQ[2], SOLOMON ANNA AMOS[3] AND KASIDI SIMON[4]

[1, 2 &3]Department of Chemical Engineering Technology, Federal Polytechnic Mubi, [4]Department of Geology Adamawa State University, Mubi. Adamawa State-Nigeria

ABSTRACT

The study investigate numerical simulations that evaluate and predict the fluid flow dynamics acting on a transmission pipe leading to vent and flare stack gas chamber. This simulation used computational fluid dynamic (CFD) package (Star CCM+). Navier-stokes equations were employed to simulate viscous flow within the S-Bend like structural pipe with inlet velocity of flare stack gas. The numerical S-Bend pipe generating a progressive regular flow was made using the flow maker at the inlet, while the grid was enabled to flare the stack gas at the outlet.

Keywords: *Simulation, Fluid, Dynamics, Flow, Analysis, Flare Gas, Pipe, processing.*

Introduction:

Industry flares CO, CO_2, H_2O, Volatile Organic Compounds (VOCs), and NO_X among others, are broadly used in chemical processes industries, upstream sector, refining and related industries that flare gas or vent unwanted gas to relieve pressures via flaring, and release it safely to the environment (Castiñeira and Edgar 2008; Umar et al 2016a). The laboratory estimates of these emissions accrued lots of cost to the overall oil processing expenses, hence computational fluid dynamics (CFD) simulations and related correlations vital

Software to predict and factors influencing flare gas operations (Singh et al., 2014). In this parametric studies of fluid flow and vent flare stack gas is based on CFD (Star CCM+) modelling are used to determine important flare gas operating parameters such as: vent gas inlet velocity, flare outlet pressure, heat content of the flare gas, etc.

This combustion technique oxidises the stack gas to CO_2 and H_2O vapour and by doing so avoids atmospheric contamination that may cause air pollution and Oxone depletion. However these combustion systems are not completely free of complications due to effects of flare performance based on numerous ranges of flow parameters (Umar et al 2016b; Castiñeira and Edgar 2008). This study summarizes the influence of the operating and parametric conditions on flare performance efficiency via employing the use of a commercial CFD (Star CCM+) package.

Potential fluid flow analysis is often carried out to predict the inertia parameters of the incoming stack gas and the behaviour of the structural material itself. Recently computational fluid dynamics (CFD) were developed to supplies detailed flow information in fluid engineering. The S-Bend pipe was made to simulate the delivery of stack gas otherwise known as flare gas to vent and flare facility from both onshore and offshore oil and gas production (Park and Kim 2013). The objectives of this research therefore is to introduce the use of a computational fluid dynamics CFD simulation software package, Star CCM+ for fluid flow analysis, and to simulate a gas flow to flare chamber in S-Bend like structural pipe.

LITERATURE REVIEW

The gas processing plant has an inlet gas with mixed liquidities. The first phase of the process separates the pure gas and then saturated the gas, then desulphurized the gas, water and mercaptan removal, dew point set. The separated gas then enters the unified chamber having passing through the liquidity units, the various separated streams are again channel along for further separation or treatment (Davoudi et al., 2013). The figure one

Vol. 12, No.9
International Journal of Pure and Applied Science

represents the process block diagram and the flare gas route within the process plants

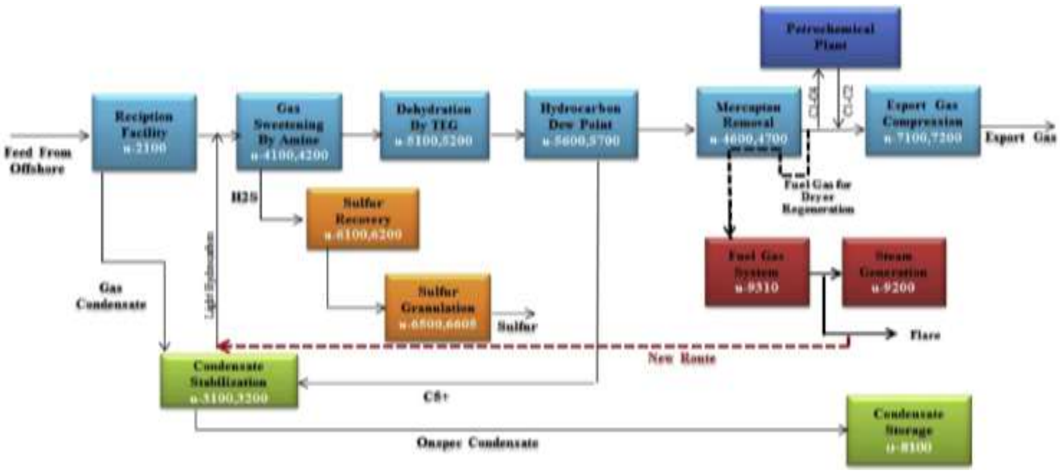

Fig 1: process description flow diagram of gas processing plants. Source: (Davoudi et al., 2013).

Principal gas flaring and air pollutants of interest

The main sources of flaring gas and atmospheric pollutants of concerned from gas processing plants are specified based on types and emissions levels. The high flaring frivol processes such as acid gas, off gas, stabilization gas flaring, regeneration gas, continuous flaring e.t.c of the plants should be given special consideration (Umar et al 2016a; Umar et al 2016b; Davoudi et al., 2013).

Quantification of heating value of flare gas

EPA specification 40 CFR 60.18, stated that flares be designed and operated without visible emissions, except for time periods not more than 5 minutes within two consecutive hours of flaring. Flares are operated with a burning flame present throughout the flaring times as specified by the methods mentioned. Flares shall be utilized with net heating value of combustion gas at 11.2 MJ/NCM (300 BTU/SCF) or more for air or steam assisted flare. With the net heating value of combustion gas at 7.45 MJ/NCM (200 BTU/SCF) or more for non-assisted flare (Hailey 2017). This rule aimed at limiting the emissions of (VOCs) at minimum heating value at the tip of the burner mixed with auxiliary fuel, as well as air or steam to boost complete blending for more than (98%) for VOCs destructions (Hailey 2017).

Fluid Flow profile

When a fluid move through a medium, velocity distribution across the medium as pipe alters to form a profile depending on inner diameter, surface roughness, and Reynolds number. Nonlinear geometry (bends, valves, flow separations) affect the profile drastically, as it causes rotation or swirl to the velocity distribution. The sensors are highly affected by the velocity profile, and have to take into consideration for calibration purposes. A special care must therefore be taken to ensure pipes upstream and inlet meter have same inner diameter. And to avoids obstructions as much as possible, so as not to cause irregularities in the profiles (NMS 2016).

MATERIALS AND METHODOLOGY

This study investigates the flow of fluid in S-Bend structural pipe with diameter (8cm), for turbulent flow. The geometry was developed by using a parametric modeler 3D-CAD of Star-CCM+. All needed region was created and polyhedral mesh were obtained by employing the generalized meshing model. The model was used to extrude the directional polyhedral cells parallel to fluid flow, enabling the mesh suitable for the flowing pipes.
Creating The Geometry: This is achieved by starting the Star-CCM+ software simulation packages, the new file is save as s-bends.sim, the new 3D-CAD was activated and the model was rename to s-bend. The following selection was made to achieve the geometry.

Table 1: Flow specifications

Fluid Type	Gas	Atmospheric Pressure	101,325
Inlet velocity	15 (m/s)	Reynolds number	40,000
Mach number	0.0337	Fluid density	0.85
Diameter of the pipe	0.04 m	Mass flow rate	0.256 kg/s

Flow Regime	Steady state	Gas composition	90% CH_4 10% others
Gas viscosity	1.10	Specific heat capacity	2.22
Temperature	20 degree Celsius	Coefficient of compressibility	0.998

Table 2: Parameters for STAR CCM+

Flow type	Segregated flow
Flow regime	Turbulent
Geometry	3D
Equation of state	Constant density
Time	Steady state
Pressure drop	High inlet pressure, Low outlet pressure
Material	Gas
Turbulence intensity	3%
Scale	0.0025
Atmospheric pressure	101,325 Pa
Prism layer thickness base %	45
Number of prisms	5
Prism layer stretching	1.8
Base size	8.0E-4 m
Reference frame	Lab reference frame

Table 3: Boundary Conditions

Inlet Velocity	15 m/s
Length of pipe	3.52 m
Diameter of pipe	0.04 m
Bends in pipe	2 bends (Upper bend radius=0.32m, Lower bend radius=0.40m)
Heat transfer through pipe walls	No heat transfer
Gravitational effects	No Gravitational effects

Outlet pressure	101,325 Pa
Inlet gas temperature	20 degree Celsius
Turbulence intensity	3%

Fig 2: The developed mesh

Figure 3: The plotted convergence

The convergence plot shows a total of 1800 iterations while as the plot also show that the Tdr reached value of 1e-09 after 1800 iterations.

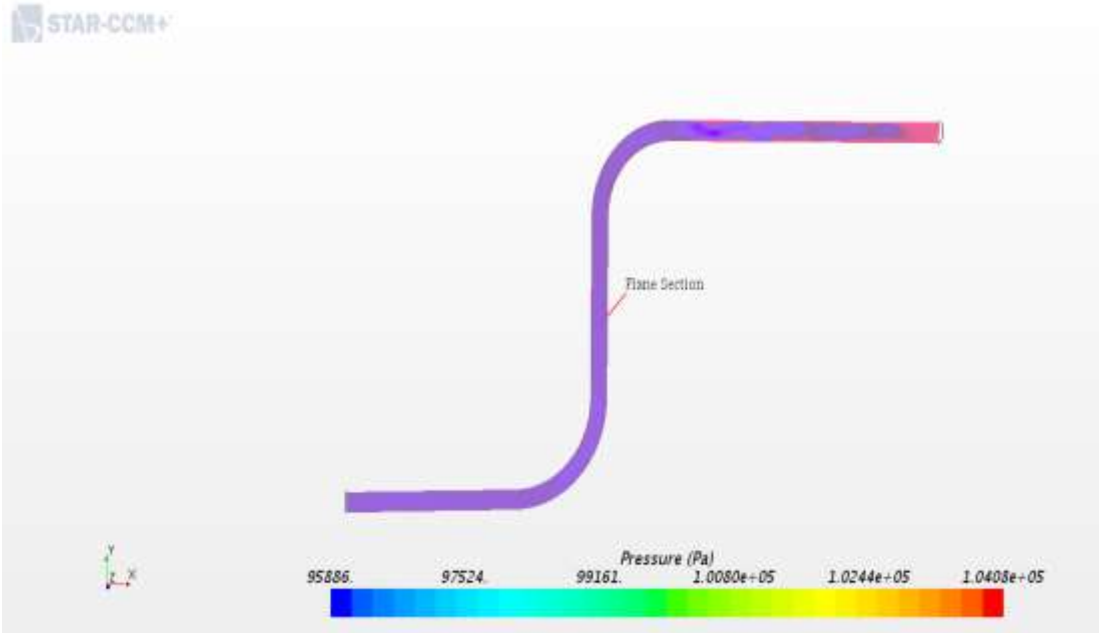

Fig 3: Showing total pressure profile (contours)

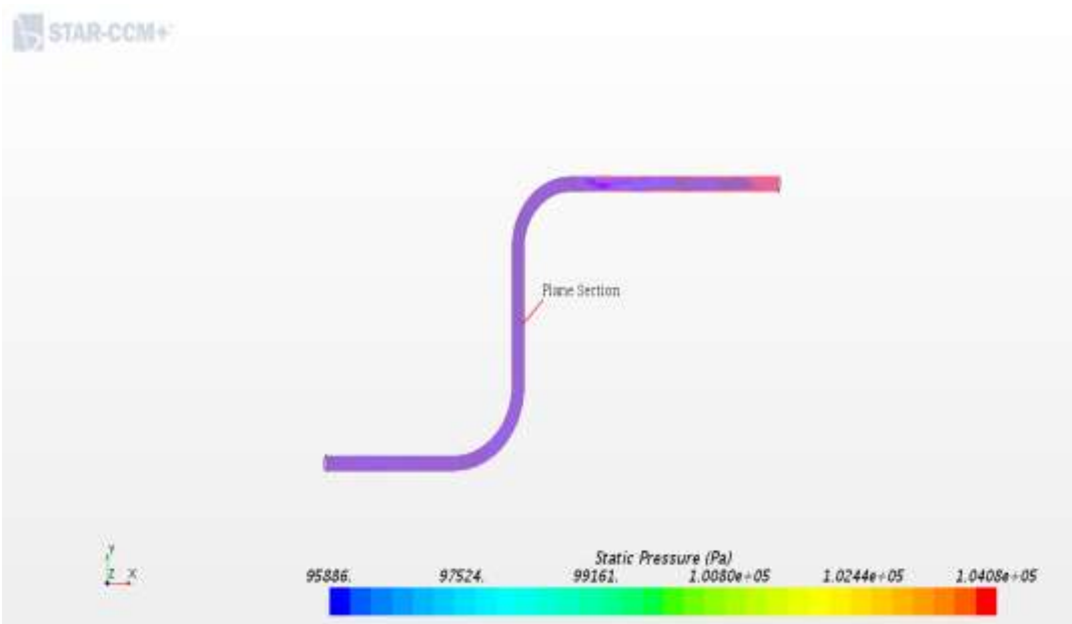

Fig 4 showing static pressure profile (contours)

ISSN: 1399-8466

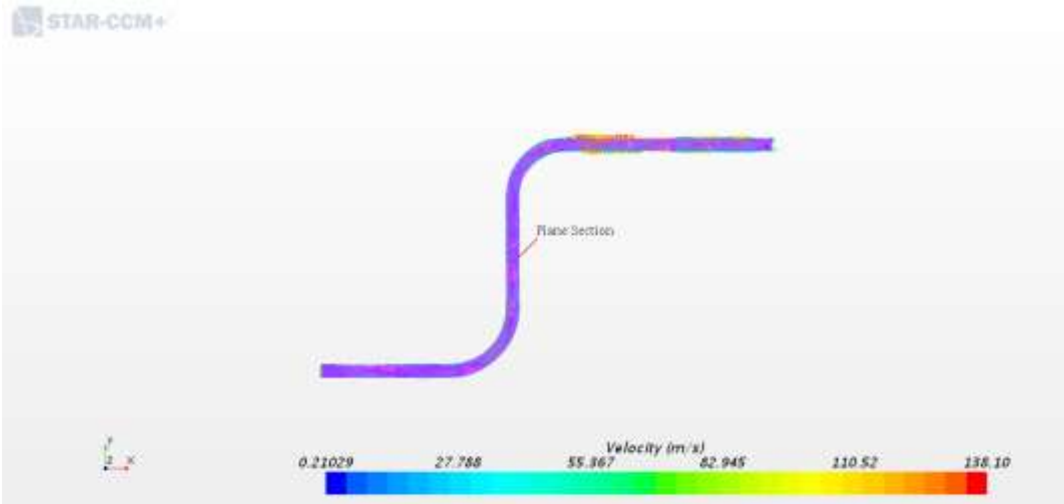

Fig 5: Showing velocity profile (contours)

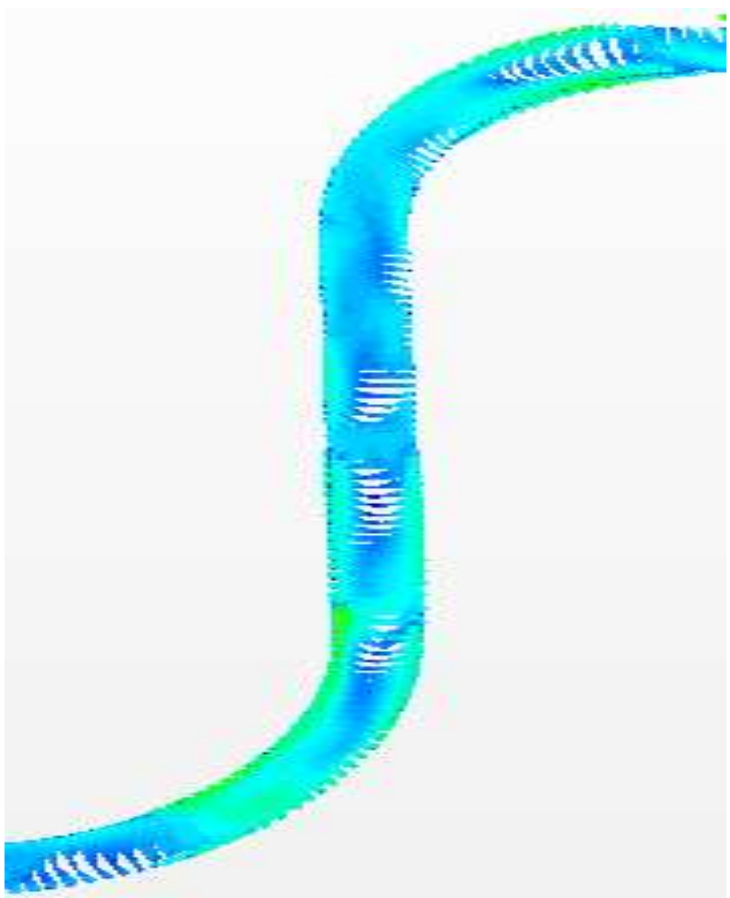

Fig 6: Showing velocity vector for the S bind

The mass flowrate and pressure loss between inlet and outlet

Table 4: Showing mass flowrate and total pressure drop

Mass Flowrate Total:	6.407608e-02
Total Pressure Drop:	2.828609e+03 (Pa)

Total pressure loss in pipe

$$\Delta P_{loss} = \rho h_L$$

$$h_L = F(Re)[\left(\frac{LV^2}{d2}\right)]$$

Assuming Reynold's Number = 40,000
Assuming no gravitational effects
F (Re) = 0.022 (Moody Diagram)

$$h_L = 0.022 + 4950$$

$$h_L = 4950.022\ Pa$$

$$\Delta P_{loss} = \rho h_L$$

$$\Delta P_{loss} = 0.85 * 4950.022$$

$$\Delta P_{loss} = 4207.5187 Pa$$

Combustion takes place at the outlet chamber, the outlet is characterised by high temperature, high pressure, and higher outlet velocity. This characteristic is indicated the simulation diagrams above.
The simulation data:
Length of the oil well = 1850m
Internal diameter of the tubing (D) = 105mm = 0.105m
Density of the crude oil = 720kg/m^3
Flow rate =1500 barrels per day $= \frac{1500\ barrels \times 159\ litres\ \times 0.001 m^3}{24\times 60\times 60} =$ $0.00276 m^3/s$
Velocity = $\frac{Q}{A}$, $where\ A = \frac{\pi D^2}{4}$
Velocity (V) = 0.31870 m/s^2

International Journal of Pure and Applied Science

CALCULATIONS AND RESULTS

Firstly at the vertical section, length (L) =1000, and assuming gravity = 9.81 and

Hydrostatic pressure $= \rho g l$

$$= 720 \times 9.81 \times 1000$$
$$= 7063200 Pa$$

Secondly at inclined angle of 60°, $hydrostatic\ pressure = \rho g l sin\theta$

Hydrostatic pressure $= 720 \times 9.81 \times 850(\sin 60)$

$$= 5199374.037\ Pa$$

Hydrostatic pressure difference between top and bottom $= 5199374.037 - 7063200$

$$= -1863825.963\ Pa$$

The negative sign indicates that it is a vertically ascending flow (against gravity)

Phase slip is the component of a multiphase mixture travelling at different velocities and it is usually influenced by the difference in densities of the fluid mixture and topographical nature of the flow loop (geometry of the pipe).

- If there is no slip, slip velocity $V_R = V_{Crude} - V_{water} = 0$

and the slip ratio $K = \frac{V_{crude}}{V_{water}} = 1.$ The density is calculated as

$$\rho_{in-situ} = \rho_w \times Q_w + \rho_{oil}(1 - Q_w)$$

Where,

ρ_w *is the density of water*, Q_w *is the volumetric fraction of water and* ρ_{oil} *is the density of oil.*

$$\rho_{in-situ} = 1000 \times 0.000828 + 720(1 - 0.000828) = 720.23\ \text{kg}/m^3$$

Since $V_{crude} = V_{water}$ and the density of the mixture is approximately equal to that of the crude oil. Thus, we can deduce that the hydrostatic pressure difference in this case will be the same as that of the initial case.

- Water cut is simply the water volume fraction of the liquid phase. $water\ cut = \frac{30}{100} = 0.3$ Since we assume there is no slip, then water cut and water holdup are the same. (Bailey et al, 2000) Therefore, water and the crude oil will move at the same speed but different flow rate. Water holdup $(\wedge_w) = 0.3$
- The field correlation that is comparable to this case, is that of a homogeneous flow (commingle stream) from a well to a distant production facility (i.e. separator)

Thirdly if we assume there is slip velocity between the phases, then we need to find the velocities of each phase. Such that $V_{water} = \frac{4\times0.000828}{3.142\times(0.105)^2} = 0.0956\ m/s^2$

and for the crude, crude flow rate $Q_{crude} = \frac{70}{100} \times 0.00276 = 0.001932m^3\ /\ s$

$$V_{crude} = \frac{4 \times 0.001932}{3.142 \times (0.105)^2} = 0.22309\ m/s^2$$

- From the calculations we can deduce that, since the velocity of the crude is greater than that of water, the crude oil will flow faster than water. $(V_{crude} > V_{water})$
- Due to the effect of gravity, the deviation may likely results to a lesser slippage than a vertical well. In a vertically ascending flow, the lighter fluid will flow faster than the heavier fluid. Thus the effect will be greater when it is vertical than when the pipe is slightly inclined or deviated. The slip velocity depends mainly on the difference in density between the two fluids, and their holdups. (Shlumberger oil review 2017)

- In the presence of slip velocity, the water holdup $\wedge_w = \frac{V_w}{V_m} = \frac{0.0956}{0.3187} = 0.3$
- Still remain unchanged; therefore it has no any effect on the hydrostatic pressure difference in this case.

CONCLUSION AND RECOMMENDATIONS

Water cut has an impact on the profitability of the oil and gas assets in mature fields.

- It increases the cost of separation, treatment, and disposal. Thereby increasing the total cost of production. (Bailey et al, 2000)
- Water cut may lead to flow assurance challenges, such as corrosion, scaling and hydrate formations.
- Increase in water cut decreases the rate of hydrocarbon production due to water break through affecting the oil producing zones. (Platt et al, 2013)
- When the reservoir experiences pressure depletion, it is an assumption that it can serve as a driving mechanism.

Since the true length of a year on earth is 365.2422.
For water-cut of 30% of 1500barrels per day = 450 barrels per day.
If we assume the entire life time of the well to be 30 years.
The maximum likely water cut over the entire life of the well will be

$= 450 \times 365.2422 \times 30$

$= 4930769.7STB$ Which is equivalent to 5 million stock tank barrels.

Water cut therefore increases the hydrostatic pressure difference between the top and bottom of the well.

REFFERENCES

Bailey, B., Crabtree, M., Tyrie, J., Elphick, J., Kuchuk, F., Romano, C., and Roodhart, L. (2000) 'Water Control'. *Oilfield Review* 12 (1), 30-51

ISSN: 1399-8466

Castiñeira D., and Edgar T. (2008). CFD for simulation of crosswind on the efficiency of high momentumjet turbulent combustion flames. J. Environ. Eng. 134(7): 561–571.

Davoudi M., Rahimpour M.R., Jokar S.M., Nikbakht F., and Abbasfard H. (2013), The major sources of gas flaring and air contamination in the natural gas processing plants: A case study, Journal of Natural Gas Science andEngineering

Hailey D. (2017) Flare Gas Measurement and Recovery of Fuel Feed Gas with Residual Oxygen Calorimetry

National Measurement System (NMS) (2016): good practice guide the calibration of flow meters.

Park Y. S. And Kim W. J., (2013) CFD simulation of hydrodynamic forces acting on subsea manifold Templates at wave zone, proceedings of the 23rd international offshore and polar engineering, Alaska USA June 30-july 5, ISBN 978-1-880653-99-9

Platt, C., Houtzager, F., Piyatad, T., Theeranun, L., McClure, J., Kienast, V., Phaophongklai, W., Watcharanantakul, R., Goh, K. G., and Wachiragorn, P. (eds.) (2013) *IPTC 2013: International Petroleum Technology Conference*. 'High Water-Cut Production Remedy by Inflow Control Solution in Challenging Intermittent Channeled Sands Oil Recovery in Gulf of Thailand Offshore'

Singh K. D., Gangaharan P., Dabade T., Shind V., Chen D., Lou H. H., Richmond P. C., and Li X., (2014) Parametric Study Of Ethylene Flare Operations Using Numerical Simulation, Vol. 8, No. 2, pp. 211–228

Umar M. T., Apanpa R. O. Ibrahim I. S. & Hussaini M. S. (2016a)"Total Petroleum Hydrocarbons: Environmental Health Effects" Proceedings on exploring sub-Sahara African resources for sustainable development in the millennium. Held at ASUU conference hall, University of Jos, Plateau state, Nigeria, West Africa. 28th-29th January, vol. 3(2), pp. 112-128.

Umar M. T., Apanpa R. L. & Mohammad B. A. "Hydrocarbons Pollution: Effects on Living Matter" Proceedings on exploring sub-Sahara African resources for sustainable development in the millennium. Held at

ASUU conference hall, University of Jos, Plateau state, Nigeria, West Africa. 28th-29th January, 2016, vol. 3(2), pp. 19-32.