International Journal of Engineering Applied Sciences and Technology, 2018
Vol. 3, Issue 7, ISSN No. 2455-2143, Pages 66-74
Published Online November 2018 in IJEAST (http://www.ijeast.com)

PREDICTION OF NATURAL GAS FLOW THROUGH A PIPELINE IN 2-DIMENSIONAL CYLINDRICAL COORDINATE

Ekeke, I. C.
Department of Chemical Engineering
Federal University of Technology, Owerri, Imo State, Nigeria

Effiong, E. E.
Department of Chemical Engineering
Federal University of Technology, Owerri, Imo State, Nigeria

Ndukwe, O. C.
Department of Chemical Engineering
Federal University of Technology, Owerri, Imo State, Nigeria

Orga, A. C.
Department of Chemical Engineering
Federal University of Technology, Owerri, Imo State, Nigeria,

Abstract— **A model for predicting compressible natural gas flow through a pipeline is presented. The model involves the two-dimensional Navier-Stokes equation in axisymmetric cylindrical coordinates together with the standard $k-\varepsilon$ turbulence equations. The Soave-Redlich-Kwong equation of state was included as auxiliary equation. The Pressure Implicit with Splitting of Operators applied on a staggered grid was used as the method of solution. Computer simulation was then carried out to determine pressure, density, velocity, temperature, eddy viscosity, turbulence eddy dissipation and turbulence kinetic energy variation within the pipeline system. The model was validated for pressure using Industry data from Shell Petroleum Development Company, Port-Harcourt, Nigeria, and published data from the National Iranian Gas Company. Profiles for pressure, temperature, velocity, density along the pipeline were obtained. Percentage error for the pressure validation was 1.44% for Shell data and 0.77% for National Iranian Gas Company data. These results prove accuracy of the model.**

Keywords— Simulation, Turbulence, Finite Volume, Soave-Redlich-Kwong equation, Navier-Stokes Equation, Axisymmetric cylindrical coordinates, Prediction, Pipeline

I. INTRODUCTION

Pipelines have been considered the best means of transporting natural gas over long distances. The accurate prediction of the flow properties within the pipeline system is very important in ascertaining what occurs within the system.

The Navier Stokes system of equations is normally used to model flow through conduits, rectangular domains and the atmosphere in fluid dynamics. The 3-dimensional set of equations gives very accurate results but under realistic assumptions, the two dimensional case can be employed. The accurate solution of this system of equations is a viable tool in predicting flow properties within the pipeline system. Many authors have presented various methods for carrying out such predictions (Adibi et al, 2018; Effiong et al, 2016; Kessal, 2000; Noorbehesht, 2012; Noorbehesht and Ghaseminejad, 2013; Nouri-Borujerdi, 2011; Nouri-Borujerdi and Ziaei-Rad, 2007; 2010; Siögreen, 1995; Ziaei-Rad and Nouri-Broujerdi, 2008).

The three main numerical solution techniques used to solve the Navier Stokes equation include, the finite difference (FD) method, the finite element method (FE) and the finite volume (FV) method. The FD method has been employed by some authors to compressible flow simulations (Kessal, 2000; Nouri-Borujerdi, 2011; Siögreen, 1995). One major drawback of the finite difference scheme is the fact that it is mainly used for Cartesian coordinates. Application of this scheme to the cylindrical or spherical coordinate system will require coordinate transformation before implementation (Blazek, 2001). This is very time consuming. In addition, it can only be applied on structured grid.

The finite element method, though applicable to both structured and unstructured grid, and to any coordinate system, is best suited to diffusion-dominated problems (Blazek, 2001), but not also well suited to turbulence flows (Bakker, 2002).

The finite volume method can be applied to any coordinate system and to both structured and unstructured grid as well (Bakker, 2002; Blazek, 2001). It also has the advantage that it can handle turbulent situations with better accuracy than the first two schemes. Various authors (Adibi et al, 2018; Effiong et al, 2016; Haris et al, 2014; Noorbehesht, 2012; Noorbehesht and Ghaseminejad, 2013; Nouri-Borujerdi and Ziaei-Rad, 2007; 2010; Nouri-Broujerdi, 2008) have applied the finite volume method in different ways to solutions of fluid dynamics problems.

Nouri-Borujerdi and Ziaei-Rad (2007) simulated the dynamic behaviour of compressible gas flow in pipelines for two dimensional, laminar, viscous, supersonic flows using a finite-volume-based finite-element method applied to unstructured grids. It was found that heating a flowing gas leads to increased pressure loss while cooling decreases the power consumption of a compressor booster station. Furthermore, change in the gas viscosity had significant effects on the flow parameters such as pressure loss and friction factor.

Ziaei-Rad and Nouri-Broujerdi (2008) numerically investigated the compressible flow of gas through a pipe subjected to wall heat flux at unsteady state condition in the entrance region by solving the coupled conservation equations governing turbulent compressible viscous flow in the developing region of a pipe under different thermal boundary conditions. The technique applied was a finite-volume-based finite-element method applied to unstructured grids. The effect of different thermal conditions on the pressure loss of unsteady flow was considered. The results demonstrated that increase in the inflow temperature or pipe-wall heat flux increased the pressure drop or decreased the mass flow rate in the pipe.

A two-dimensional unsteady turbulent compressible high pressure gas flow with a rupture at its centre had been studied numerically using the k-ε model with the finite volume-Galerkin upwind technique (Nouri-Borujerdi and Ziaei-Rad, 2010). The results showed that the downstream pressure would be more affected by the rupture than the upstream pressure.

Natural gas flow in a transmission line had been modelled by Noorbehesht (2012) with continuity, momentum, energy and equations of state in two-dimensional cylindrical coordinates at steady state. Turbulence effects and the ideal gas equation were applied. A pressure-velocity coupling technique, the Semi Implicit Pressure Linked Equations Revised (SIMPLER) algorithm was employed in the finite volume solution of the problem. Results obtained by this method agreed very well with the industrial data from the National Iranian Gas Company.

Noorbehesht and Ghaseminejad (2013) modeled the dynamic behaviour of natural gas in transmission lines using the transient equations of continuity, momentum, energy and the $k-\varepsilon$ turbulence model. The same simulation technique as applied by Noorbehesht (2012) already mentioned was used. The accuracy of this method was verified by comparing the field data with the predictions which showed an error of approximately 4 to 4.5%.

Adibi et al (2016) studied the numerical simulation of natural gas leakages from low pressure pipelines using the finite volume method and the pressure-based algorithm on a structured grid. The results showed that in just 6.4 s after an accidental rupture, released natural gas penetrated to 40m in the vertical direction and 20m in the horizontal direction. The results also indicated that the wind speed was a key factor in the dispersion process.

In this paper, a solution scheme for the simulation of compressible natural gas flow through a pipeline in 2-dimensional axisymmetric cylindrical coordinates is presented. The Navier-Stokes system of equations, the energy equation and the $k-\varepsilon$ turbulence equation were employed. The Soave-Redlich-Kwong (SRK) equation of state was used to close up the system of equations and the finite volume method of solution (the PISO algorithm) was used. To the best of our knowledge, the inclusion of the SRK equation of state with the PISO algorithm in the simulation technique is the distinct difference between the present work and previously published works (Noorbehesht, 2012; Noorbehesht and Ghaseminejad, 2013).

The model has been validated for pressure using real Industry data from Shell Petroleum Development Company, Port-Harcourt, Nigeria, and National Iranian Gas Company data (Noorbehesht, 2012). Simulated profiles for pressure, temperature, velocity, density along the pipeline are presented. This paper is an extension of an earlier published work by Effiong et al (2016) which did not involve validation with industry data.

II. MATERIALS AND METHODS

A. **Materials**

The Industrial Data were obtained from Shell Petroleum Development Company, Port-Harcourt, Nigeria and the published data of the National Iranian Gas Company, Iran (Noorbehesht, 2012).. Table 1 shows these data.

Table 1: Industrial Data from Shell, and National Iranian Gas Companies

S/N	PROPERTIES	SHELL	NATIONAL IRANIAN GAS COMPANY
1	Inlet Pressure (Pa)	6830000	7111000
2	Outlet Pressure (Pa)	3199170	5555000
3	Inlet Temperature (K)	314.90	319.86
4	Outlet Temperature (K)	300.05	307
5	Molecular Weight	16.91	17.74
6	Heat Capacity (KJ/KgK)	2680	2533
7	Viscosity (Kg/m^2)	1 x 10^{-5}	1.34 x 10^{-5}
8	Thermal Conductivity (W/mK)	0.04	0.040033

9	Pipeline Length (m)	18,000	135,000
10	Pipeline Diameter (m)	0.22	1.442

B. Methods

The Model Equations

The basic equations used to model flow in pipelines at steady state and employed in this work are the Navier-Stokes equations presented as equations (1) through (9).

Assumptions

The cross-sectional area of the pipe is constant.

The gas flow is highly turbulent

Conservation of mass or continuity equation (Bird, 2002):

$$\frac{1}{r}\frac{\partial}{\partial r}(\rho r V_r) + \frac{\partial}{\partial z}(\rho V_z) = 0 \qquad (1)$$

Conservation of Momentum equation in terms of τ in the r-directions (Bird, 2002):

$$\frac{1}{r}\frac{\partial(r\rho V_r V_r)}{\partial r} + \frac{\partial(\rho V_r V_z)}{\partial z} = -\frac{\partial P}{\partial r} - \frac{1}{r}\frac{\partial}{\partial r}(r\tau_{rr}) - \frac{\partial}{\partial z}\tau_{zr} + \frac{\tau_{\theta\theta}}{r} \qquad (2)$$

Conservation of Momentum equation in terms of τ in the z-direction (Bird, 2002):

$$\frac{1}{r}\frac{\partial(r\rho V_r V_x)}{\partial r} + \frac{\partial(\rho V_z V_z)}{\partial z} = -\frac{\partial P}{\partial r} - \frac{1}{r}\frac{\partial}{\partial r}(r\tau_{zr}) - \frac{\partial}{\partial z}\tau_{zz} \qquad (3)$$

where

$$\tau_{rr} = -\mu_{eff}\left(2\frac{\partial V_r}{\partial r} - \frac{2}{3}(\nabla . v)\right) \qquad (4)$$

$$\tau_{xz} = -\mu_{eff}\left(2\frac{\partial V_z}{\partial z} - \frac{2}{3}(\nabla . v)\right) \qquad (5)$$

$$\tau_{xr} = -\mu_{eff}\left(\frac{\partial V_r}{\partial z} + \frac{\partial V_z}{\partial r}\right) \qquad (6)$$

$$\tau_{\theta\theta} = -\mu_{eff}\left(2\frac{V_r}{r} - \frac{2}{3}(\nabla . v)\right) \qquad (7)$$

$$\nabla . v = \frac{1}{r}\left(\frac{\partial}{\partial r}\right)(rV_r) + \frac{\partial V_z}{\partial z} \qquad (8)$$

$\mu_{eff} = \mu + \mu_t = effective\ viscosity$ (Blazek, 2001; Noorbehesht, 2012)

Conservation of Energy Equation:

$$\frac{\partial}{\partial z}(\rho c_p V_z T) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho c_p V_r T) = \frac{\partial}{\partial z}\left(\left(\lambda + \frac{c_p\mu_t}{G_T}\right)\frac{\partial T}{\partial z}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(r\left(\lambda + \frac{c_p\mu_t}{G_T}\right)\frac{\partial T}{\partial r}\right) + S_T \qquad (9)$$

Equation of State

The equation of state adopted in this work is the Soave-Redlich-Kwong equation stated as follows (Perry et al, 1997; Soave, 1972):

$$P = \frac{RT}{V-b} - \frac{a(T)}{V(V+b)} \qquad (10)$$

where

$$a(T) = 0.4274\left(\frac{R^2T_c^2}{P_c}\right)\left\{1 + m\left[1 - \left(\frac{T}{T_c}\right)^{0.5}\right]\right\}^2 \qquad (11)$$

$$m = 0.480 + 1.57\omega - 0.176\omega^2 \qquad (12)$$

$$b = 0.08664\frac{RT_c}{P_c} \qquad (13)$$

P_c = Critical Pressure (N/m^2)
T_c = Critical Temperature (K)
R = Gas constant (J/kg.K)
V = Volume of gas (m^3)
ω = Acentric factor

Turbulence_kinetic_energy:

$$\frac{\partial}{\partial x}(\rho V_x k) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho V_r k) = \frac{\partial}{\partial x}\left(\left(\mu + \frac{\mu_t}{G_k}\right)\frac{\partial k}{\partial x}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(r\left(\mu + \frac{\mu_t}{G_k}\right)\frac{\partial k}{\partial r}\right) + S_k \qquad (14)$$

Turbulence_dissipation_rate:

$$\frac{\partial}{\partial x}(\rho V_x \varepsilon) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho V_r \varepsilon) = \frac{\partial}{\partial x}\left(\left(\mu + \frac{\mu_t}{G_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(r\left(\mu + \frac{\mu_t}{G_\varepsilon}\right)\frac{\partial \varepsilon}{\partial r}\right) + S_\varepsilon \qquad (15)$$

where

International Journal of Engineering Applied Sciences and Technology, 2018
Vol. 3, Issue 7, ISSN No. 2455-2143, Pages 66-74
Published Online November 2018 in IJEAST (http://www.ijeast.com)

$S_k = \mu_t g - \rho\varepsilon\ ,\ S_\varepsilon = C_1 g \mu_t \varepsilon k - C_2 \rho \frac{\varepsilon^2}{k}$ and
$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}$
$C_\mu = 0.09, C_1 = 1.4, C_2 = 1.92, G_k = 1.00, G_\varepsilon = 1.3, G_T = 0.85$

Boundary Conditions
The flow domain for a 2D compressible natural gas flow is shown in Figure 1.

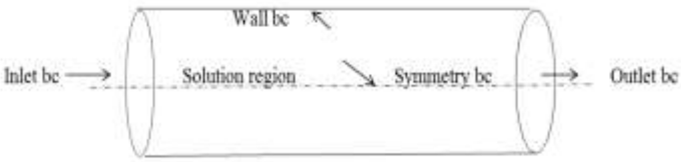

Figure 1: Flow domain and boundary conditions (Noorbehesht, 2012)

The boundary conditions to which the equation (1), (2), (3), (9) and (10), are usually subjected for compressible flows in 2D are as follows:

Steady state condition

- Fluid boundary
- Inlet: P, V_x, V_z, T are defined and ρ is defined according to the equation of state, equation (10).
- Outlet: The general condition of the fluid, which is commonly applied in the finite volume method, is as follows (Ferziger and Peric, 2002):

$\partial T / \partial n = 0$ and $\partial V_{x_n} / \partial n = 0$
where,
n is the normal outward vector of the outlet

- Solid boundary
- No slip condition $V_x = V_{x_w} = 0$
- Constant temperature on the wall: $T = T_w$
- Symmetric boundary condition, $\partial \Phi / \partial n = 0$

Numerical Technique
The finite volume method is the numerical solution method of choice adopted in this work. In employing the finite volume method, the general form of the conservation equations of fluid for the geometry considered in this work for any scalar variable ϕ can be represented as follows:

$$\frac{\partial}{\partial x}(\rho V_x \phi) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho V_r \phi) = \frac{\partial}{\partial x}\left(\Gamma \frac{\partial \phi}{\partial x}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(r\Gamma\frac{\partial \phi}{\partial r}\right) + S_\phi \quad (16)$$

ϕ represents any of the variables, V_r, V_x, T, k *and* ε while Γ represents μ and k. In employing the method, bracketed smaller contributions to the viscous stress terms in the transport equation are hidden (Versteeg and Malalasekera, 2007). Equations 1, 2, 3, 10, 15 and 16 are then integrated over a control volume after which Gauss' divergence theorem is applied to give the following:

$$\int_A n.(\rho V_x \phi) dA + \int_A n.(r\rho V_r \phi) dA = \int_A n.\left(\Gamma\frac{\partial \phi}{\partial x}\right) dA + \int_A n.\left(r\Gamma\frac{\partial \phi}{\partial r}\right) dA + \int_{CV} S_\phi dV \quad (17)$$

The solution region comprising of a grid is then divided into discrete control volumes (CV). The CV surface consists of four (in 2D) plane faces, denoted by lower-case letters corresponding to their directions (e, w, n, s) with respect to the central node, P (Ferziger and Peric, 2002).

The governing equations are then integrated over the CV to obtain a discrete equation on node P. The equation is as follows:

$$(\rho V_x A\phi)_e - (\rho V_x A\phi)_w + (\rho V_r A\phi)_n - (\rho V_r A\phi)_s = \left(\Gamma A\frac{\partial \phi}{\partial x}\right)_e - \left(\Gamma A\frac{\partial \phi}{\partial x}\right)_w + \left(\Gamma A\frac{\partial \phi}{\partial r}\right)_n - \left(\Gamma A\frac{\partial \phi}{\partial r}\right)_s + (S_p\phi_p + S_u)\Delta V_p \quad (18)$$

The integration of the continuity equation gives equation (19):

$$(\rho V_x A)_e - (\rho V_x A)_w + (\rho V_r A)_n r - (\rho V_r A)_s r = 0 \quad (19)$$

In the discretization of the governing equations, central differencing scheme is used for the spatial diffusion terms and upwind differencing scheme in the spatial convection terms. The general discrete equation is then:

$$a_P\phi_P = a_W\phi_W + a_E\phi_E + a_S\phi_S + a_N\phi_N + S_U \quad (20)$$

where

$a_P = a_W + a_E + a_S + a_N + \Delta Fr - S_P,$
$\Delta Fr = (F_e r - F_w r) + (F_n r - F_s r),\ F_e = (\rho U)_e A_e$
$F_w = (\rho U)_w A_w,\ F_n = (\rho U)_n A_n\ ,\ F_s = (\rho U)_s A_s$

Next, the staggered arrangements (Harlow and Welch, 1965) have to be used as shown in Figure 2. As the momentum equations are coupled with pressure term, the Pressure-Implicit with Splitting of Operators (PISO) (Issa et al, 1986) which is an improvement to the Semi-Implicit-Pressure Link Equation (SIMPLE) algorithm (Patankar and Spalding, 1972) was employed in the computations.

The idea of using staggered grid arrangement is to evaluate scalar variables, such as pressure, density, temperature, turbulence kinetic energy and dissipation rate at ordinary nodal points (•) where they are stored, and to calculate velocity components on staggered grids centered around the cell faces of scalar control volume. The reason for using staggered grid in solving pressure-velocity coupling equation is because a uniform pressure field will be obtained if velocity and pressure are stored at the same nodal points.

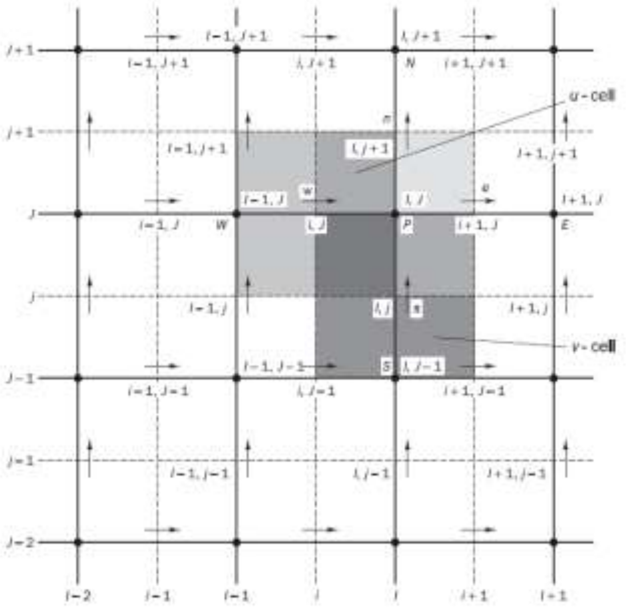

Figure 2: Staggered Structured Grid Arrangement for Pressure and Velocity

Model Validation
To validate the steady state model, the data (Table 1) obtained from Shell Petroleum Development Company, Port-Harcourt, Nigeria, and published data from National Iranian Gas Company (Noorbehesht, 2012) were used. Profiles for pressure, temperature, velocity, density along the pipeline were obtained and are presented in Figures 3 through 9 while the pressure validation is given in Table 2.

III. RESULTS AND DISCUSSION

The results of the simulation are shown in Figure 3 through Figure 9 while Table 2 shows the pressure validation.
The shapes of the pressure distribution curves for Shell and the Iranian Gas Companies (Figures 3 and 4), show a drop in pressure along the pipeline length. This agrees with the fact that for a fluid flowing in a pipe, there is usually a pressure drop along the pipe due to frictional resistance between the fluid particles. The parameters used for the validation were the pressure outlet values of both experimental and calculated or simulated results. The result (from Table 2) for Shell shows that the pressure difference ($|\Delta P_{out}|$) between the experimental and calculated values was 46170Pa. This gave a percentage error of 1.44%. On the other hand, the percentage error for the Iranian data was 0.77%.

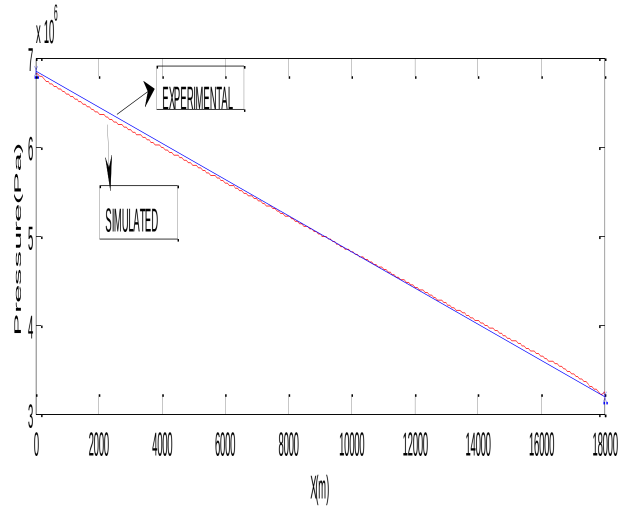

Figure 3: Comparison of simulated pressure distribution along the length of the pipeline with Shell data

These values confirm the fact that the solution scheme is good enough to predict pressure outlet values in a pipeline. A percentage error of 4 – 4.5% was obtained in an earlier publication (Noorbehesht, 2012) where the SIMPLER algorithm and the ideal gas equation were included in the finite volume solution technique.

Table 2: Pressure Validation

	SHELL	IRANIAN
$P_{out_{exp.}}(Pa)$	3199170	5555000
$P_{out_{calc.}}(Pa)$	3153000	5512400
$\lvert\Delta P_{out}\rvert(Pa)$	46170	42600
Percentage error	1.44%	0.77%

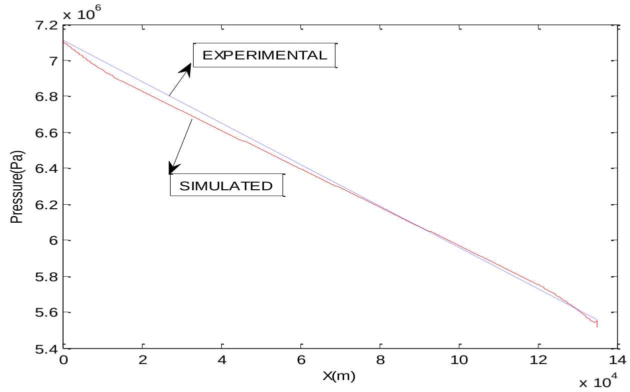

Figure 4: Comparison of simulated pressure distribution along the length of the pipeline with Iranian data.

Figure 5 and Figure 6 depict those of temperature profiles for the two companies. A sharp drop in temperature is observed from the beginning of the pipe up to a distance of about 2000m down the pipeline length for Shell. The curve then becomes constant till the end of the pipeline. This means that the gas attained a constant temperature of about 300K which is most likely the ambient temperature.

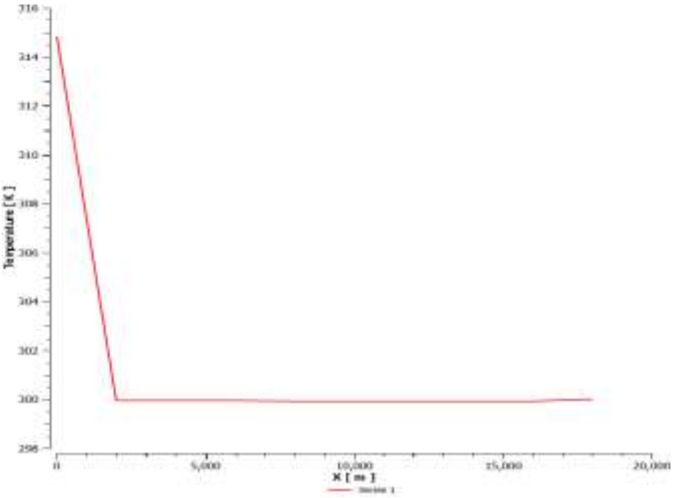

Figure 5 Temperature distribution along the length of the pipeline with Shell data

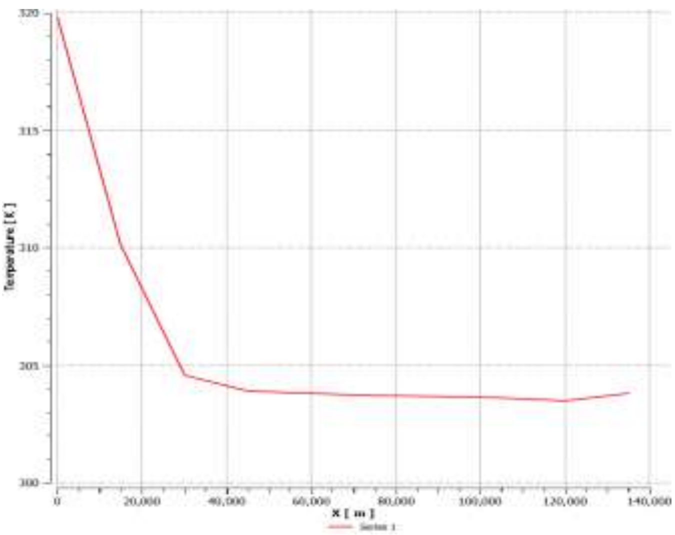

Figure 6: Temperature distribution along the length of the pipeline with Iranian data

The temperature profile for the Iranian Company (Figure 6) has a similar profile to that of Shell from the beginning of the pipe until it decreased gradually from a distance of about 30,000m of the pipe to a point 120,000m along the pipe length. It then increased again until the pipe exit. The gradual decrease observed is as a result of the Joule Thompson effect which is usually experienced by real gases. The Joule Thompson effect is the temperature change of a *real* gas or liquid when it is forced through a valve or porous plug so that no heat is exchanged with the environment (Perry, 2007; Roy, 2002). This phenomenon causes adiabatic expansion. Two factors that can change the temperature of a fluid during an adiabatic expansion include a change in internal energy and the conversion between potential and kinetic energy of the fluid. Since temperature is the measure of thermal kinetic energy (or energy associated with molecular motion), a change in temperature indicates a change in thermal kinetic energy. The internal energy is the sum of thermal kinetic energy and thermal potential energy (Rock, 1983). Hence, even if the internal energy does not change, the temperature can change due to conversion between kinetic and potential energy; this is what happens in a free expansion and usually produces a decrease in temperature as the fluid expands (Pippard, 1957; Tabor, 1991). If work is done on or by the fluid as it expands, then the total internal energy changes. This is what happens in a Joule–Thomson expansion and it can produce larger heating or cooling than is observed in a free expansion. In the present work, the Joule-Thompson effect was proposed for the gradual temperature decrease (from 30,000m to 120,000m along the pipe length) and subsequent increase (from 120,000m to the pipe exit) seen in Figure 6. The temperature decrease (resulting in the cooling of the gas) suggests that heat was transferred from the surroundings to the gas and vice versa. A similar finding was reported for a gas flow through a Norwegian pipeline (.Sund et al, 2015).

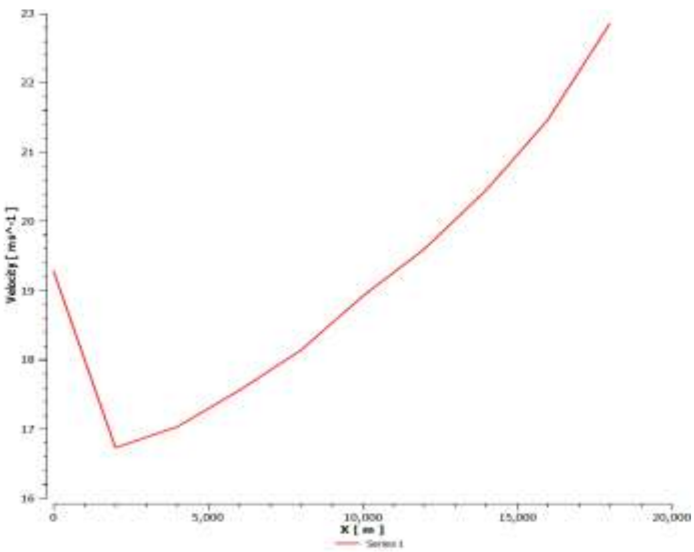

Figure 7: Velocity distribution along the length of the pipeline with Shell data

International Journal of Engineering Applied Sciences and Technology, 2018
Vol. 3, Issue 7, ISSN No. 2455-2143, Pages 66-74
Published Online November 2018 in IJEAST (http://www.ijeast.com)

Figure 7 depicts the velocity distribution for the Shell pipe. It experienced a sharp decrease first, up to a point 2000m down the pipe length, due to the fact that the gas got cooled as evidenced on the temperature plot (Figure 5) and then increased until the gas reached the end of the pipe due to gas expansion. The expansion however, occurred in the pipe during a period of constant temperature as depicted in Figure 5, that is referred to as isothermal gas expansion. This normally occurs when a system is in contact with an outside thermal reservoir (in this case, the surroundings about the pipeline and the pipeline itself) and the change in the system will occur slowly enough to allow the system to continue to adjust to the temperature of the reservoir through heat exchange. In an expansion process, the volume of the system increases. Velocity then increases since the gas molecules gain energy on expansion and move faster. Moreover, an expression which gives a relationship between density and velocity (equation 21) shows that as density decreases, velocity also increases.

$$m^* = \rho u A \qquad (21)$$

where $m^* = mass\ flow\ rate$, $u = velocity\ of\ gas$ and $A = area$

The velocity profile for the Iranian Company, Figure 8, also shows a similar trend to that of the Shell Company. The profile first decreased because the gas first cooled due to a decrease in temperature from the entrance of the pipeline to a distance of about 30,000m down the pipe length. It then increased until the pipe exit because the gas experienced Joule-Thompson expansion through the rest of the pipeline length.

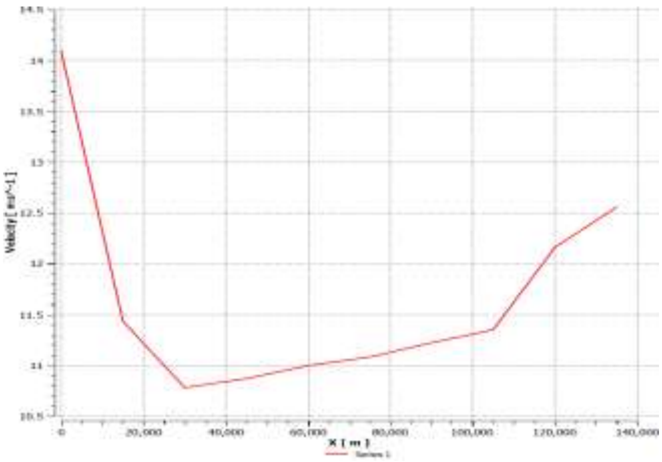

Figure 8: Velocity distribution along the length of the pipeline with Iranian data.

The density profile for the Shell Company, depicted in Figure 9, shows a downward trend. This trend is similar to that of the pressure profile (Figure 3) which is in agreement with the ideal gas equation of state that gives a direct relationship between pressure and density.

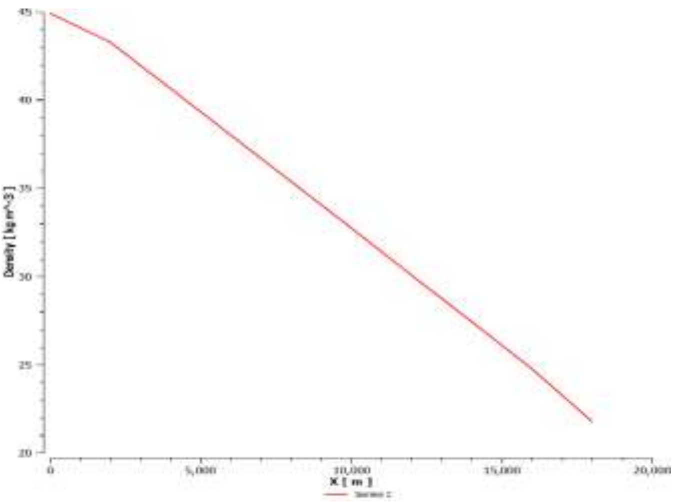

Figure 9: Density distribution along the length of the pipeline with Shell data

IV. CONCLUSION

A model for predicting compressible natural gas flow through a pipeline has been presented. The model was validated for pressure using Industry data from Shell Petroleum Development Company, Port-Harcourt, Nigeria, and published data from the National Iranian Gas Company. Profiles for pressure, temperature, velocity, density along the pipeline have been simulated. Percentage error for the pressure validation was 1.44% with the Shell data and 0.77% with National Iranian Gas Company data. These results show that the model is good enough to predict gas flow properties in the Oil/Gas Industry.

V. NOMENCLATURE

V_r, v	velocity in the radial (r) direction, m/s
V_z, u	velocity in the axial (z)direction, m/s
V_θ	velocity in the azimuthal direction
V	volume
r	radial direction, m
x, z	axial direction, m
T	temperature, K
T_c	critical temperature, K
E	total internal energy, J
k	thermal conductivity, W/m.K
k	kinetic energy of turbulence , J/Kg
P	pressure, N/m^2

P_c critical pressure, N/m^2
C_p constant pressure specific heat capacity, J/KgK
q heat flux, W/m^2
R gas constant, J/kg. K
a, b, m constants
G_k constant for turbulence kinetic energy,
G_ε constant for specific dissipation rate,
G_T constant for specific dissipation rate,
S source term

Greek Characters

μ dynamic viscosity, Ns/m^2
ρ density, kg/m^3
τ shear stress tensor, N/m^2
ε turbulence dissipation rate , m^2/s^3
θ azimuthal direction
ω acentric factor

Subscripts

E, e East
N, n North
S, s South
W, w West
t turbulence

VI. REFERENCE

Adibi, O., Najafpour, N., Farhanieh, B. & Afshin, H. (2018) 'Numerical Simulation of Natural Gas Dispersion from Low Pressure Pipelines', *International Journal of Mechanical and Mechatronics Engineering*, Vol.12 No. 2, pp. 1307 – 6892.

Bakker, A, (2002) *Applied Computational Fluid Dynamics*. © Fluent Inc (Lecture notes). http://www.bakker.org.

Bird, R. B., Stewart, W. E. and Lightfoot, E, N. (2002) *Transport Phenomena* (p. 337). John Wiley and Sons, Inc., New York.

Blazek, J., (2001) *Computational Fluid Dynamics: Principles and Applications*, Elsevier.

Effiong, E. E., Orga, A. C., Ibe, E. C., Ekeke I. C. & Nzebuka, C. G. (2016) 'A Model For The Flow Of Natural Gas Through A Pipeline In Two-Dimensional Cylindrical Coordinates', *International Journal of Current Research in Life Sciences*, Vol.5 No. 11, pp. 614-621.

Ferziger, J. H. and Peric, M (2002) *Computational Methods for Fluid Dynamics*, 3rd edition, Springer, Germany. 3rd edition.

Harlow, F. H. and Welch, J. E. (1965) 'Numerical Calculation of Time-dependent Viscous Incompressible Flow of Fluid with Free Surface', *The Physics of Fluids*, Vol. 8 No. 12, pp. 2182–2189.

Issa, R. I., Gosman, A. D. and Watkins, A. P. (1986) 'The Computation of Compressible and Incompressible Recirculating Flows', *Journal of Computational Physics*, Vol. 62, pp. 66–82.

Kessal, M , (2000) 'Simplified Numerical Simulation of Transients In Gas Networks', *Trans IChemE*, Vol. 78, Part A, pp. 925-931

Noorbehesht, N. (2012) 'Numerical Simulation of Natural Gas Flow in Transmission Lines through CFD Method Considering Effective Parameters', *Journal of Basic and Appied. Scence Reearch*, Vol. 2 No. 8, pp.7473-7487.

Noorbehesht, N. and Ghaseminejad, P. (2013) 'Numerical Simulation of the Transient Flow in Natural Gas Transmission Lines Using a Computational Fluid Dynamic Method', *American Journal of Applied Sciences*, Vol. 10 No. 1, pp. 24-34.

Nouri-Borujerdi, A. and Ziaei-Rad, M. (2007) 'Numerical Prediction of Developing Flow in Gas Pipelines', *5th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics.*

Nouri-Borujerdi, A. and Ziaei-Rad, M., (2010) 'Numerical Modeling of Transient Turbulent Gas Flow in a Pipe Following a Rupture', *Transaction B: Mechanical Engineering*, Vol. 1 No. 7, pp 108-120. © Sharif University of Technology, Tehran, Iran.

Nouri-Borujerdi, A. (2011) 'Transient modeling of Gas Flow in Pipelines following catastrophic failure', *Mathematical and Computer Modeling*, Vol.54, pp 3037–3045.

Patankar S. V. and Spalding, D. B. (1972) 'A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows' *International Journal of Heat and Mass Transfer*, Vol. I5, pp. 1787-1806.

Perry, R. H., Green, D. W. and Maloney, J. O. (1997) *Perry's Chemical Engineers' Handbook*, Seventh Edition (p. 4-21). McGraw-Hill.

Pippard, A. B. (1957) *Elements of Classical Thermodynamics*, Cambridge University Press, Cambridge, U.K.

Rock, P. A. (1983) *Chemical Thermodynamics*, University Science Books, Mill Valley, CA.

Roy, B. N. (2002) *Fundamentals of Classical and Statistical Thermodynamics,* John Wiley & Sons, Inc., New York.

Siögreen B. (1995) 'High Order Centered Difference Methods for the Compressible Navier-Stokes Equations', *Journal of Computational Physics*, Vol. 117, pp. 67-78.

Soave, G. (1972) 'Equilibrium constants from a modified Redlich-Kwong equation of state. Chemical Engineering Science', Vol. 27 No. 6, pp. 1197-1203.

Sund, F., Oosterkamp, A. and Hope, S. M. (2015) *Pipeline Modeling – Impact of Ambient Temperature and Heat Transfer Modeling,* paper presented at the Twenty-fifth International Offshore and Polar Engineering Conference, Kona, Hawaii Big Island, USA.

Tabor, D. (1991) *Gases, liquids and solids,* Cambridge University Press, Cambridge, U.K.

Versteeg, H K and W Malalasekera, W. (2007). *An introduction to computational fluid dynamics the finite volume method, second edition*. Longman Scientific and Technical.

Ziaei-Rad, M. and Nouri-Broujerdi, A., (2008) 'Numerical simulation of turbulent unsteady compressible pipe flow with heat transfer in the entrance region', *The Royal Swedish Academy of Sciences,* pp. 1 – 6.