EFFECT OF SUPERFICIAL VELOCITY ON SEVERE SLUGGING IN A SUBSEA PIPELINE/RISER SYSTEM;

A COMPUTATIONAL FLUID DYNAMIC (CFD) INVESTIGATIVE APPROACH

BY

ANA, EKPENYONG A.

CORPORATE BUSINESS RESEARCH SERVICES

140 IKOT EKPENE ROAD

UYO AKWA IBOM STATE

NIGERIA

Abstract

One basic underlying challenge associated with multiphase transport is slugging. To grasp its fundamentals, a Computational Fluid Dynamics (CFD) analysis is employed to study the consequences of altering pipeline/riser systems' superficial velocities with regards to slugging. The analysis is carried out on six different combinations of air and water superficial velocities. Tested air superficial velocities ranges between 14 m/s and 4 m/s while keeping that of water constant at 2.5 m/s. Tetrahedral mesh of 0.017 cell size is employed in a 0.25m diameter pipeline riser system. The system design contains a 9m horizontally inclined flowline, connected to a 5-meter riser with the help of a 1.25m radius of curvature bend. A steady injection of equivalent volume fraction of water and air into the transient system with an initial bottom slug, induced by the inclination shows zero slugging at high velocities and substantial amount of slugs at lesser velocities.

Keywords: Riser; Slugging; Simulation; Pipeline; CFD; Multiphase flow; Superficial Velocity; ANSYS CFX;

INTRODUCTION

The task of transporting well effluents to the top side through the subsea wellhead sometimes require multiphase flow. This pattern of fluid transport is usually employed to avoid a more costly underwater separation. Slugging hinders high production rate, causes damages to production facilities among other effects. Multiphase flow is characterised by varying pressure and fluid composition. These different parameters make the flow very unstable, therefore giving room for the undesirable phenomenon known as slugging. Although eminent scholars have come up with various means of predicting and mitigating against slugging, however, separator flooding, corrosion, variable flow rate, cavitation and pipe damage arising from slug still pose great challenge (Pots, Bromilow, and Konijin 1987).

Altering the top-side choke has been the conventional solution method for preventing slugging in the oil and gas industry. This approach however, does not address the challenge of production decline. Havre and Dalso (2002) indicated that the automatic control can eliminate slugging effectively, however the procedure was experimented and tested (Godhavn, Strand, and Skofteland 2005), and result showed it could not achieve maximum output.

The aim of this work was therefore to get a good understanding of severe slugging problem in subsea pipeline/riser systems using computational fluid dynamic method. The procedure attempts to answer the following research questions;

- At what superficial velocities of liquid and gas will severe slugging begin?
- How will severe slug react to change in superficial velocity?
- At what maximum, superficial gas velocity will slug no longer be tolerated?

A case study of typical subsea pipeline riser arrangement is considered for this project.

SEVERE SLUGGING PHENOMENON

Severe slugging occurs where there is a vertical inclination or at the base of a vertical pipe (mostly risers). The Severe slugging effect is highly dominated by gravity forces and temperature variation, especially in offshore operation. Studies on severe slugging showed that the phenomenon is a terrain-induced type of flow resulting in equal or greater height of slug when compared with the riser (Schmidt, Brill and Beggs 1980; Schmidt, Brill and Beggs 1981). It was also discovered that the high magnitude of pressure fluctuation and unsteady delivery rates were being induced by slugging (Schmidt, Brill and Beggs 1981). This discovery was confirmed by later work (Taitel 1986). It was also reported that topography or low flowrate could induce severe slugging.

A full cycle of severe slug is categorised into four stages (Taitel1986). They include the liquid build-up, slug production, gas blowout and gas blowdown stages as shown below (figure1).

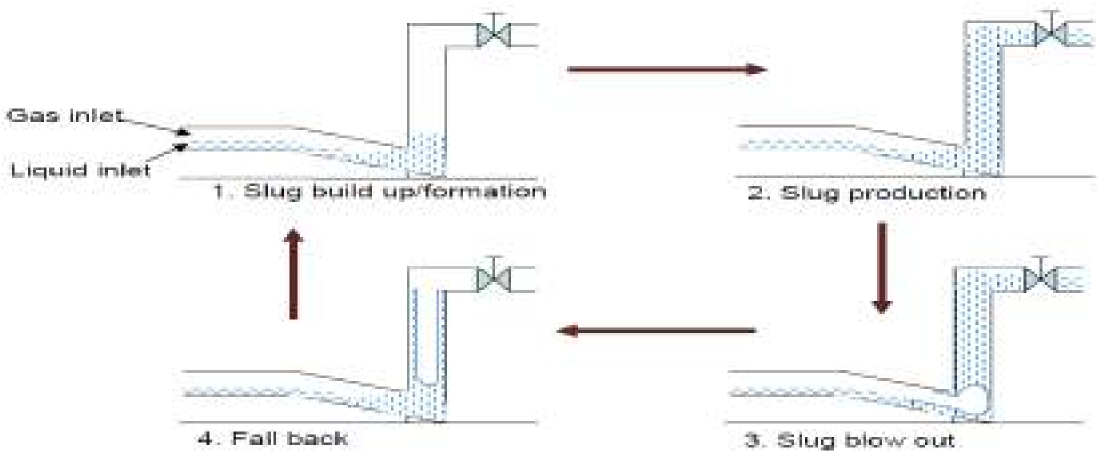


Figure 1: Severe slug Stages/cycle phenomenon adapted from Taitel (1986) and presented by Ogazi (2011)

COMPUTATIONAL FLUID DYNAMICS (CFD)

CFD involves the use of governing mathematical equations such as conservation of mass, energy, momentum, species and body force effects to predict fluid flow, chemical reaction, heat and mass transfer and related processes. CFD is one among three basic approaches for solving fluid dynamics and heat transfer problems. For every CFD study, mathematics, engineering, and computer science must come together (Tu*et al.* 2012). CFD approach is on the trend nowadays compared to experimental and analytical methods, perhaps due to its acceptability in various industries.

The introduction of CFD into the oil and gas industry in the late 20th century brought a different approach to the oil and gas project implementation. CFD can give insight, provide foresight, and enhance efficiency in oil and gas exploration, production, and processing operations (Mohiuddin 2016). The inception of CFD made it possible for engineers to possibly predict key parameters that are required for safe and efficient project execution.

SIMULATIONS

The key simulation tool here is ANSYS software resource commonly known as ANSYS CFX. ANSYS CFX is a commercial computational fluid dynamic tool designed to handle a broad range of fluid dynamic issues. ANSYS CFX has been in existence for over twenty years now (ANSYS 2016). Though engineers have been deploying the tool to address fluid dynamic problems in many engineering fields, it was only introduced into the oil and gas industry recently.

Apart from being efficient for modelling MPF, it can as well be used to handle issues concerning fluid structure interaction, chemical reaction, combustion, heat transfer, radiation, and rotating machinery. Embedded Multiphysics tool, flexible workflow, mesh flexibility, high speed and turbulence modelling are the key features that are worthy to mention.

GEOMETRY

A pipeline geometry of 0.25m internal diameter connected to the vertical riser with the help of a bend of 1.25 radius of curvature is created. The length of the horizontal pipe is 9m while that of the riser is 5m. Just as illustrated on figure 2. It is assumed that the horizontal pipeline is the flowline which transports fluids from the wellheads to the bottom part of the riser, while the vertical pipe is the riser itself which conveys the fluids from the flowline to the topside. The model considers an inlet point on the horizontal section of the flow line. Most times, the velocity at which the fluid leaves the well is considered as the speed of inlet, while the gas injection is done other times to maintain flow velocity. A choke manifold is usually provided at the wellhead section to control the flow rate.

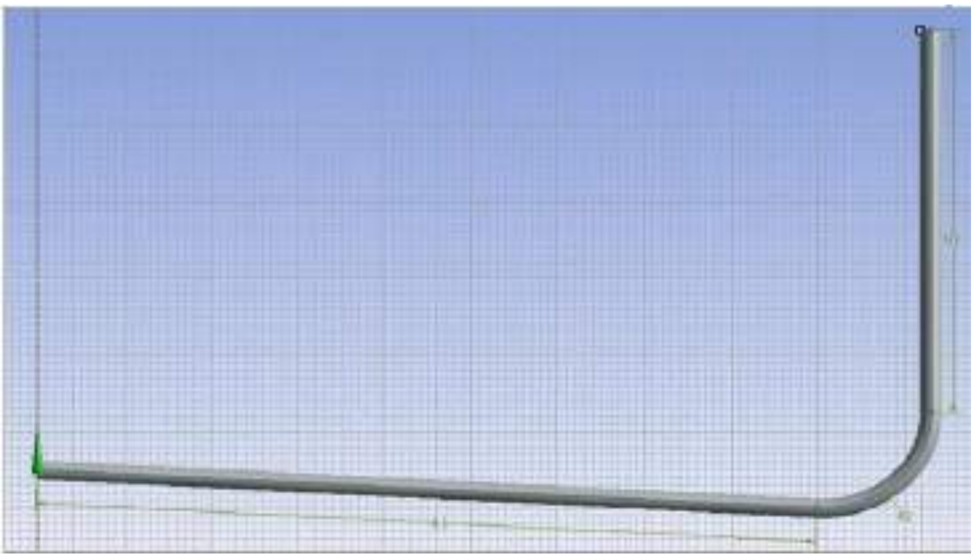


Figure 2: A diagram showing the Pipeline riser system geometry

Meshing and Discretization of Domain

The geometry is split into subdomains also known as cells to enable the governing equation solve in each subdomain (ANSYS 2016). With ANSYS Workbench, the discretization is done using the finite volume method. This approach utilises the conservation equation to get a discrete equation for each cell (Versteeg, H.K., Malalasekera, W., and Versteeg, H. 2007).

Structured grid is considered due to the simplicity of the geometry. One benefit of structured mesh is its ability to produce higher resolution result and better convergence (Hernandez Perez, V. 2008). The riser system model is meshed using ANSYS Workbench meshing tool after which the physical model and boundary conditions are set up using CFX pre. Mesh size of 0.017m is considered to avoid excessive diffusion. See table 1 for grid properties.

Table 1: Table showing grid properties

Mesh Type	Cell Size	No of Cells	Nodes
Hexahedral	0.017m	460265	5E+05

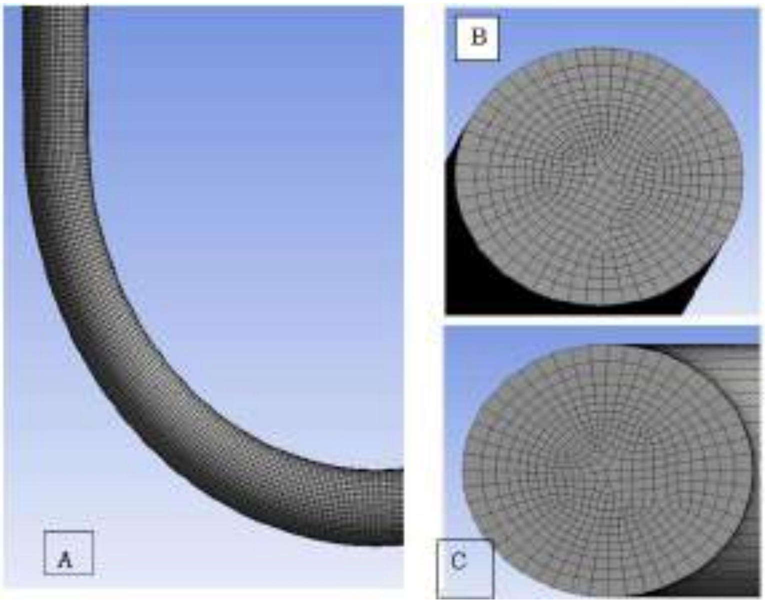


Figure 3: A picture of generated mesh; (a) bend section; (b) Inlet section (c) Outlet section

Fluid Properties

A model of air and water flowing in a pipeline-riser system at different superficial velocities is created. The pipeline-riser system is modelled to contain majorly air as primary fluid before equal volume fractions of air and water are injected through the flowline (horizontal section).

Boundary Conditions

Boundary specification entails the allocation of physical properties to the domain. This is done once the mesh generation is completed. The implication of boundary type and conditions specification is that its gives a representation of the operational scenarios and allocating such boundaries will be based on the physics of the problem. Boundary condition for this problem is chosen based on a typical subsea pipeline-riser arrangement.

*Table 2**:** Table showing the boundary types of boundary conditions*

BOUNDARY TYPE	BOUNDARY CONDITIONS
INLET	Velocity Inlet (Uair or Uwater) Volume Fraction (Vair or Vwater)
OUTLET	Pressure outlet
WALL	Symmetry (No-slip)

Stability Constraint

A time step of 0.02 is used in other to avoid instability which is usually caused by selecting high time steps. Another important condition that must be kept to retain stability is that fluid must not advert through more than a cell in a step time. This condition is necessary because different equations are involved in solving, and each of them assumes fluxes strictly between cells.

CFX Solver

ANSYS CFX solves all the fluid flow equations using the finite volume discretization method (VOF). Typically, the numerical algorithm required by this solution method is made up of three distinct steps (Versteeg, H.K., Malalasekera, W., and Versteeg, H. 2007). First, the domain fluid flow is integrated. Secondly, the integral equations gotten from step one are discretized, and lastly, the resulting algebraic equation is solved via iteration method.

RESULT AND CONCLUSION

The primary condition for slugging is that the volume fraction of liquid must be equal to 1 (i.e. Vwater =1). Tested superficial velocity of air are 14 m/s, 12 m/s, 10 m/s, 8 m/s, and 4 m/s, while that of water is kept constant at 2.5m/s.

The flow line

With the absence of bubbles or any form of intermittency, in the flowline, it can be agreed that the flow pattern along the line is stratified. It is also agreeable that varying the initial gas superficial velocity does not affect the incremental effect of gas phase volume fraction as flow heads towards the bend. The different phase's volume fraction contours for all the tested velocities showed same characteristics along the flowline as shown in in figure 4 bellow.

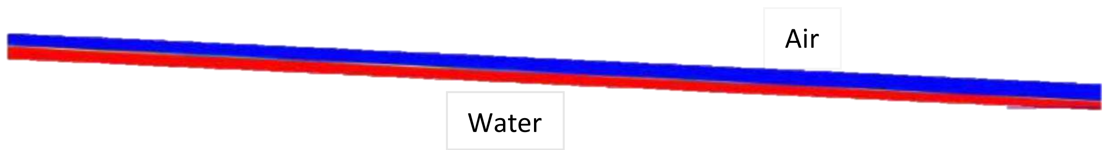


Figure 4: Contours showing liquid volume fraction in the flowline for the tested superficial velocities

The Bend

The animation clips, 3 dimensional views, and volume fraction reports indicate higher liquid content along the bend during flow. It is observed that none of the tested velocities gave room for slug at the bottom of the riser. Furthermore, 0.4 and 0.8 liquid volume fractions were recorded within the bend for minimum and maximum fractions respectively for all the cases.

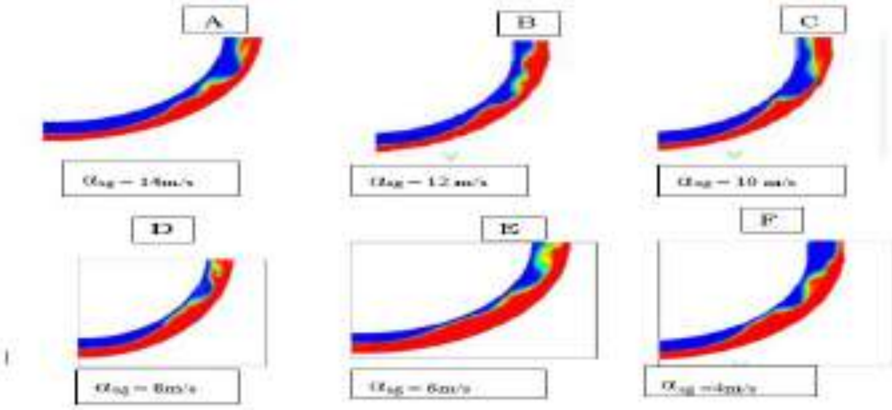
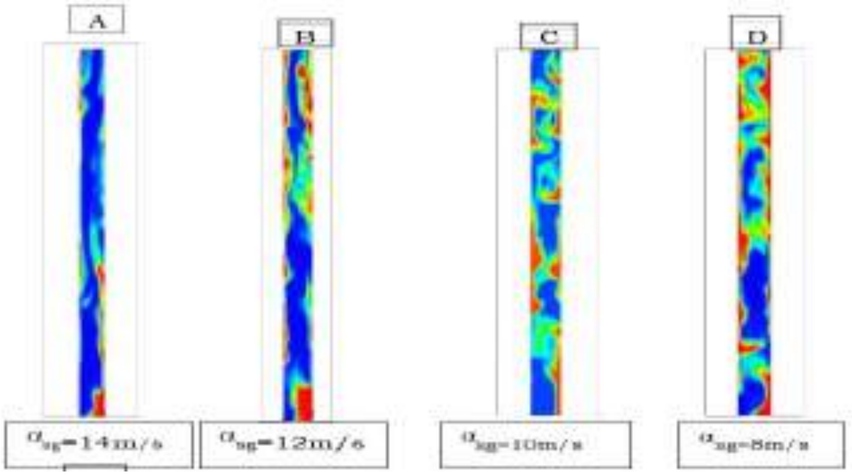


Figure 5: Contours showing the nature of flow along the bends

The Riser

One primary observation made on the flow along the vertical pipe is that the tendency for slugging increases as the inlet superficial velocity of the gas is reduced. The slug length also increases as the superficial gas velocity is reduced. For the flow with 14m/s, the slugging condition was not met. However, there were some near slugging phenomena (up to 0.85 volume fraction of liquid). Such phenomena were short and never rose to the riser outlet. Liquid slugs became realistic from case 4 up to case 6. Summary of activities observed on the vertical riser is provided in Table 3. Also, the different volume fraction contours are provided (Fig. 6).



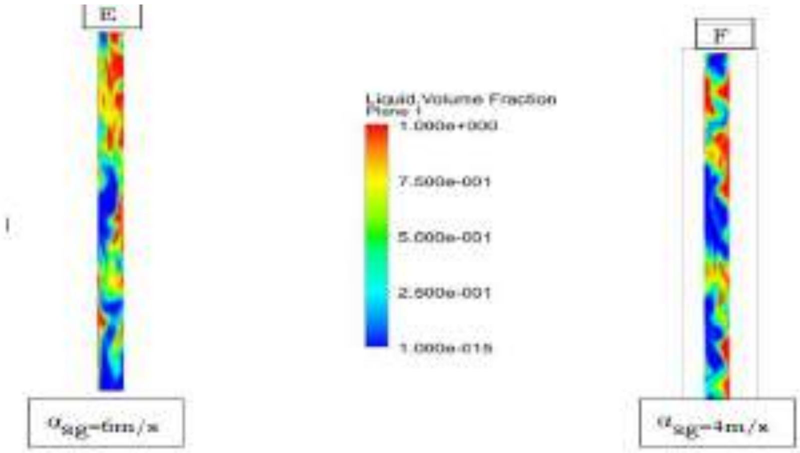


Figure 6: Contours showing liquid volume fraction in the vertical riser for all the tested superficial velocities

Table 3: Pipeline riser system flow activity summary

CASE NO	VSG(M/S)	VSL(M/S)	SLUGGING	SLUGGING POINT	SLUG LENGTH (M)	SLIP RATIO (VSG/YSL)
1	14	2.5	Negative	Riser	0.00	5.6
2	12	2.5	Negative	Riser	0.00	4.8
3	10	2.5	Negative	Riser	0.00	4.0
4	8	2.5	Positive	Riser	0.80	3.2
5	6	2.5	Positive	Riser	0.91	3.3
6	4	2.5	Positive	Riser	1.51	2.4

As table 4 indicates, slugging increases as the air superficial velocity reduces. Judging from the chart below (figure 7), it can be deduced that slug elimination and control goes with increase in slip ratio.

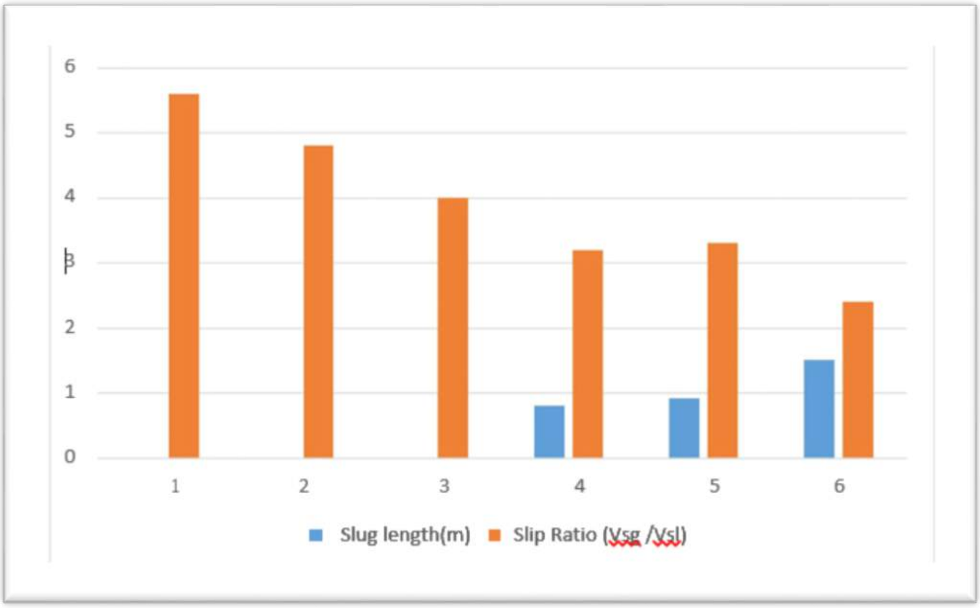


Figure 7: Plot of initial air superficial velocity against the slip ratio of water and air superficial velocities

Conclusions

The study clearly shows that slugging tends to increase along the pipeline riser system as the slip ratio reduces. Increase in air velocity while keeping that of the liquid constant will prevent slugging in the pipeline riser system. This finding confirms the results of Mcquillan and Whalley (1985) as well as Mishima and Hibiki (1996). ANSYS CFX has a high modelling efficiency. Moreover, the interface and embedded numerical models and transient parameters are very efficient for analysing slugging.

Recommendations

The following recommendations are deemed necessary:

1. A detailed comparison between CFX result and field data should be made to check for the level of accuracy that CFX result can render.
2. Mesh independent study should be conducted using CFX to determine the best size required to get the best result on ANSYS CFX.

REFERENCES

ANSYS (2016) *Academic support resources* [online] available from http://www.ansys.com/products/academic/support-resources.

Godhavn, J.-M., Strand, S., and Skofteland, G. (2005) 'INCREASED OIL PRODUCTION BY ADVANCED CONTROL OF RECEIVING FACILITIES'. *IFAC Proceedings Volumes* 38 (1), 567–572

Havre, K. and Dalsmo, M. (2002) 'Active feedback control as a solution to severe slugging'. *SPE Production & Facilities* 17 (03), 138–148

Hernandez Perez, V. (2008) *Gas-liquid two-phase flow in inclined pipes*.

McQuillan, K. W. and Whalley, P. B. (1985) 'Flow patterns in vertical two-phase flow'. *International Journal of Multiphase Flow* 11 (2), 161–175

Mishima, K. and Hibiki, T. (1996) 'Some characteristics of air-water two-phase flow in small diameter vertical tubes'. *International Journal ofMultiphase Flow* 22 (4), 703–712

Mohiuddin, A. (2016) *CFD simulation in the oil and gas industry*. [online] available from <https://www.academia.edu/1160796/CFD_simulation_in_the_oil_and_gas_industry?auto=download> [16 November 2016]

Pots, B. F. M., Bromilow, I. G., and Konijn, M. J. W. F. (1987) 'Severe slug flow in offshore Flowline/Riser systems'. *SPE Production Engineering* 2 (04), 319–324

Schmidt, Z., Brill, J. P., and Beggs, H. D. (1980) 'Experimental study of severe slugging in a Two-Phase-Flow pipeline - Riser pipe system'. *Societyof Petroleum Engineers Journal* 20 (05), 407–414

Schmidt, Z., Brill, J. P., and Beggs, H. D. (1981) 'Experimental study of Two-Phase normal slug flow in a Pipeline-Riser pipe system'. *Journal of EnergyResources Technology* 103 (1), 67

Taitel, Y., Shoham, O., Barnea, D., and Dukler, A. E. (1985) 'Gas-liquid flow in inclined tubes: Flow pattern transitions for upward flow'. *ChemicalEngineering Science* 40 (1), 131–136

Versteeg, H. K., Malalasekera, W., and Versteeg, H. (2007) *An introduction tocomputational fluid dynamics: The finite volume method*. 2nd edn. Harlow,England: Pearson Education