International Journal of Civil Engineering Research.
ISSN 2278-3652 Volume 5, Number 3 (2014), pp. 203-210

http://www.ripublication.com/ijcer.htm

Experimental and Numerical Study of Flow in Prismatic and Non-prismatic Section of a Converging Compound Channel

Abinash Mohanta[1], Bandita Naik[2], K.C. Patra[3] and K.K. Khatua[4]

*[1,2,3,4]Department of CIVIL Engg, National Institute of Technology,
Rourkela, Odisha, India.*

Abstract

Rivers have fascinated engineers and scientists for decades while providing water supply for domestic, irrigation, and industrial consumption or transportation and recreation use. As a result of topography changes along the open channels, designing the converging compound channel is an essential. Water surface prediction is an important task in flood risk management in urban area. In this paper based on the principle of the momentum balance, a one dimensional method is investigated to predict the water surface elevations in non-prismatic compound channels. The numerical method is then applied to calculate water surface elevation in non-prismatic compound channel configurations, the results of calculations show good agreement with the experimental data. In this paper a complete three-dimensional and two phase CFD model for flow distribution in a converging compound channel is investigated. The finite volume method (FVM) with a dynamic Sub grid-scale was carried out for convergence condition. The volume of fluid (VOF) method was used to allow the free-surface to deform freely with the underlying turbulence. The accuracy of the model was analyzed with observed data from experimental studies of a converging compound channel as the qualitative reference and the computed results of the present study were validated. The predicted results for the flow characteristics are in reasonable agreement with the experiment data.

Keywords: Experimental model, Numerical model, FVM method, VOF method, prismatic and non prismatic section, converging compound channel.

1. Introduction

Prediction of conveyance capacity in open channel flows is complex and requires adequate modeling of flow features such as secondary circulation cells and, specifically for over-bank channels, the momentum exchange that occurs at the main channel/floodplain interface. One of the significant characteristic attributes of flow in an open-channel bend is its secondary flow and therefore the helical motion that is the main reason of the winding river morphology and the tendency to create a succession of shoals and deeps along its way. Due to the existence of secondary flow, flow characteristics in channel bends are much more complicated than those in straight channels. In other words, close to the inner wall and also at the channel bed, pressure gradient exceeds centrifugal force and conveys water in a transverse direction towards the inner wall. At the free surface, centrifugal force drives the flow to the outer wall. This kind of flow is known as the secondary flow (Lien et al., 1999). Super-elevation, secondary flows and their tending to redistribute the mean velocity, permuting the boundary shear stress, bank erosion and shifting, flow separation (that its presence coming together with vortex bar formation decreases the channel width and conveyance capacity), and bed migration in mobile boundary channels have made the study of the non-prismatic open channels of a high interest in the field of river engineering.

A step has been taken to do numerical analysis on a non prismatic compound channel flow .The work will help to simulate the different flow variables in such type of complex flow geometry. Booij (2003) and VanBalen et al. (2008) modeled the flow pattern at a mildly-curved 180º bend and assessed the secondary flow structure using large eddy simulation (LES). Lu et al. (2004), Bodnar and Prihoda (2006) and Omid Seyedashraf, AliAkbar &Milad Khatib Shahidi(2012) applied a three-dimensional numerical model to simulate secondary flows, the distribution of bed shear stress, the longitudinal and transversal changes of water depth and the distribution of velocity components at bend using the standard k-ϵ turbulence model. B. K. Gandh, H.K. Verma and Boby Abraham (2010) determined the velocity profiles in both the directions under different real flow conditions and investigated the effects of bed slope, upstream bend and a convergence / divergence of channel width of velocity profile. Ahmed Kassem; Jasim Imran and Jamil A. Khan analyzed from the three-dimensional modeling of negatively buoyant flow in a diverging channel with a sloping bottom and modified the k-e turbulence model for the buoyancy effect. Anthony G. Dixon (2012) simulated CFD software with fluid flow interactions between phases. Other studies have been also conducted by researchers in this area (e.g. Ervine and Jasem (1995), Jasem (1990), James & Brown (1977), Elliott (1990), Bousmar (2002) and Bousmar et al. (2004a) Bahram Rezaei (2006)).

In the present work, an effort has been made to investigate the velocity profiles for prismatic and non-prismatic section of a convergent compound channel by using a commercial computational fluid dynamics(CFD) code, namely FLUENT. The CFD model developed for a real open-channel was first validated by comparing the velocity profile obtained from it with that obtained by actual measurement in the same channel using preston tube. The CFD model has been the used to analyze the effects of upstream bend, convergence of channel width and bed slope, and to study the variations in velocity profiles along the horizontal and vertical directions.

2. Experimental Setup

Experiments was conducted in non-prismatic compound channels with varying cross section built inside a concrete flume measuring 15m×.9m×0.5m at National Institute of Technology Rourkela Hydraulic laboratory. The width ratio of the channel is $\alpha \leq 1.8$and the aspect ratio is $\delta \geq 5$. The converging angle of the channel is 12.38°.Converging length of the channel is 0.84m.The channel is made up of cement concrete. Water will be supplied through a Centrifugal pumps (15 hp) discharging into a RCC overhead tank. In the downstream end there will be a measuring tank followed by a sump which will feed to over head tank through pumping thus completing recirculation path. Fig.1 shows the schematic diagram of experimental setup and dimensions of channel with test section respectively.Fig.2 shows the plan view of two different experimental sections. Water was supplied to the flume from an underground sump via an overhead tank by centrifugal pump (15 hp) and recirculate to the sump after flowing through the compound channel and a downstream volumetric tank fitted with closure valves for calibration purpose. Water entered the channel bell mouth section via an upstream rectangular notch specifically built to measure discharge in the laboratory channel. An adjustable vertical gate along with flow straighteners was provided in upstream section sufficiently ahead of rectangular notch to reduce turbulence and velocity of approach in the flow near the notch section. At the downstream end another adjustable tail gate was provided to control the flow depth and maintain a uniform flow in the channel. A movable bridge was provided across the flume for both span wise and stream wise movements over the channel area so that each location on the plan of compound converging channel could be accessed for taking measurements. The broad parameters of this channel such as aspect ratio of main channel (δ), width ratio (α).

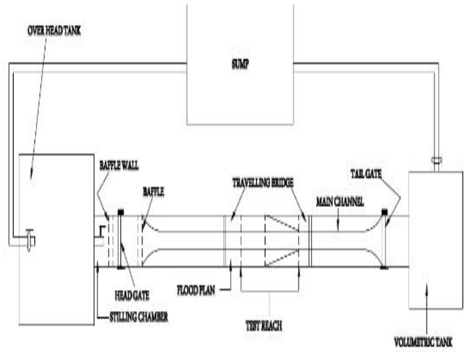

Fig. 1: Plan view of Experimental Set up

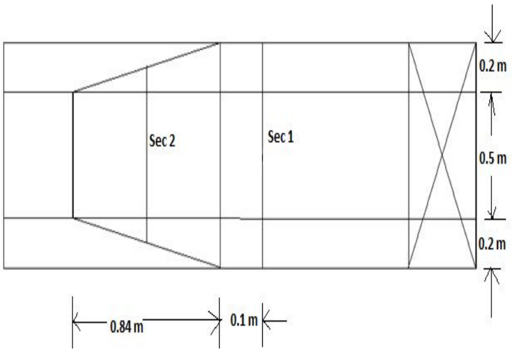

Fig. 2: Plan view of two different experimental section

3. Validation of CFD Model

For validation of CFD simulation, the velocity profile across the width of a channel is measured by a preston tube and compared with the numerical results. A long converging water conveying non prismatic channel has been selected for this purpose. The channel width is divided into cells of 0.05 m size for measurement of velocity at the centre of each cell by Preston tube. For CFD simulation, the flow domain is initially discretized with hexahedral elements of face length equals to 0.01 m for analysis. Flow is assumed to be steady, turbulent and three-dimensional. Experimental velocity profile was measured by preston tube and velocity contour was drawn by SURFER and then validated with CFD prediction.

Discharge evaluated from the two velocity profiles using velocity-area integration method are as under:

Discharge from Preston tube measurement data = 0.051 m3/s

Discharge from CFD simulation data = 0.063 m3/s

4. Results and Discussion

Results have been shown to demonstrate the performance of commercial generic CFD package when applied to open channel flows. In cases where the channel geometry is more complex and varies along the channel, the secondary flow is more as a result of the geometry than the turbulence. Thus the usefulness of CFD and applicability of the models for the problem under consideration depends very much upon the type of geometry and particularly on the nature of dominant forces.

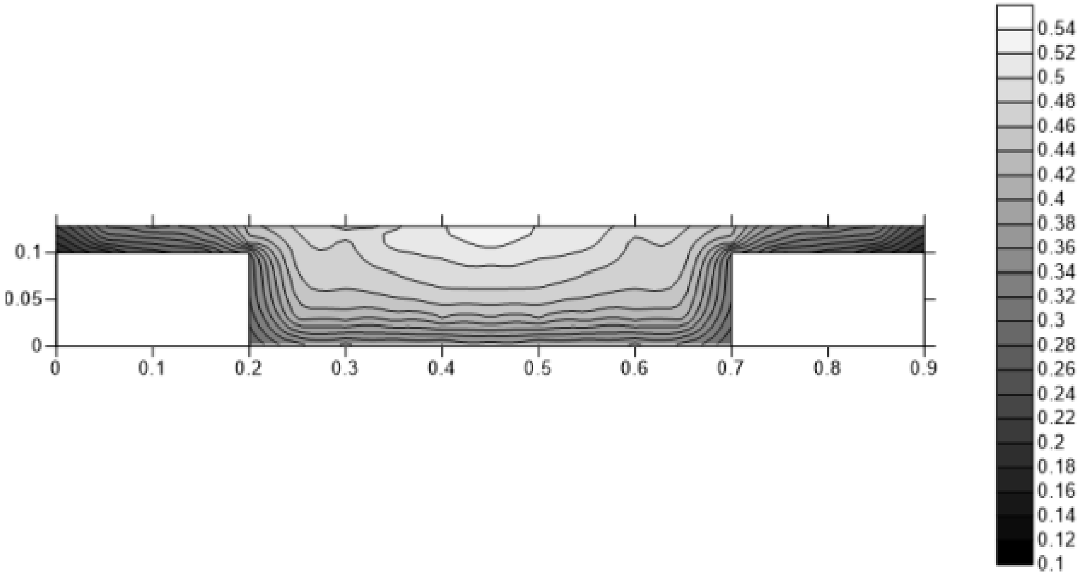

Fig.3 (a) Contour of Sec 1 of experimental results.

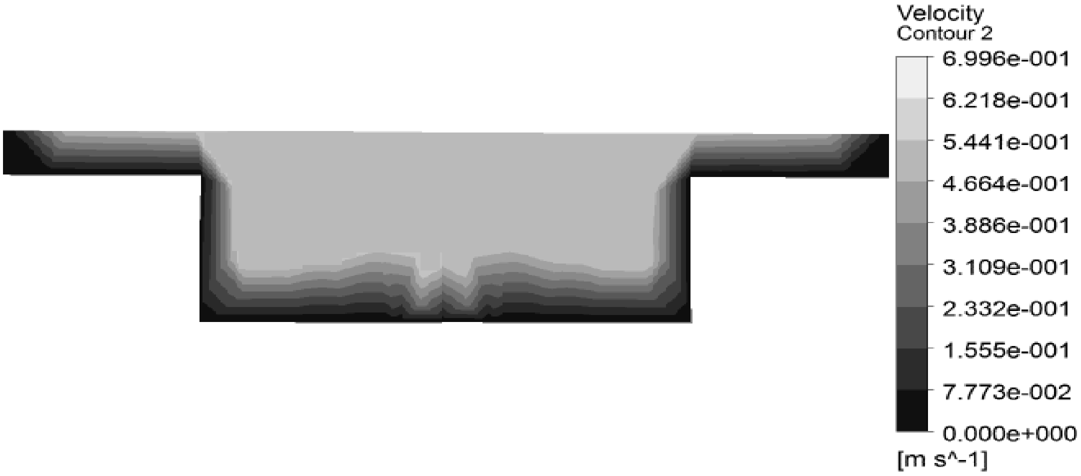

Fig.3 (b) Contour of Sec 1 by CFD simulation.

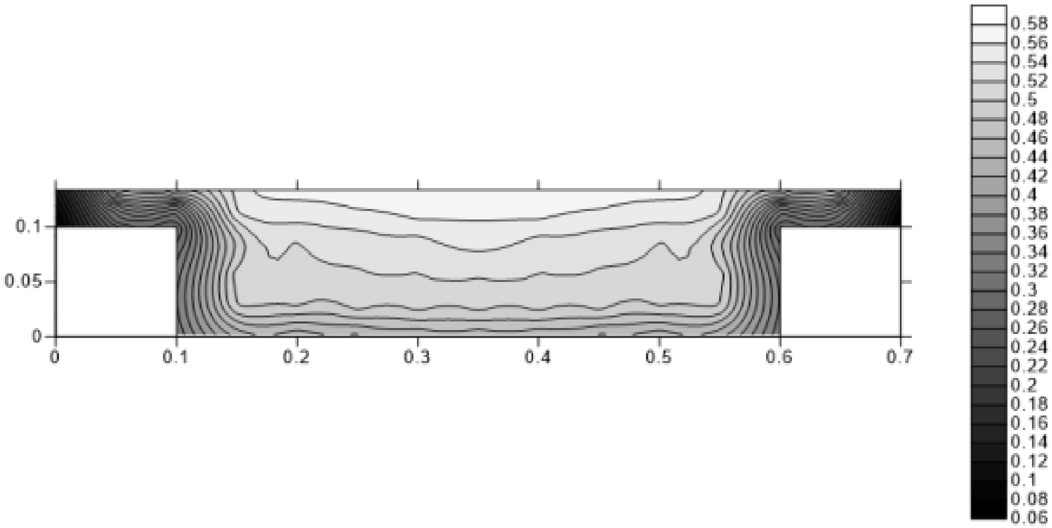

Fig. 4 (a) Contour of Sec 2 of experimental results.

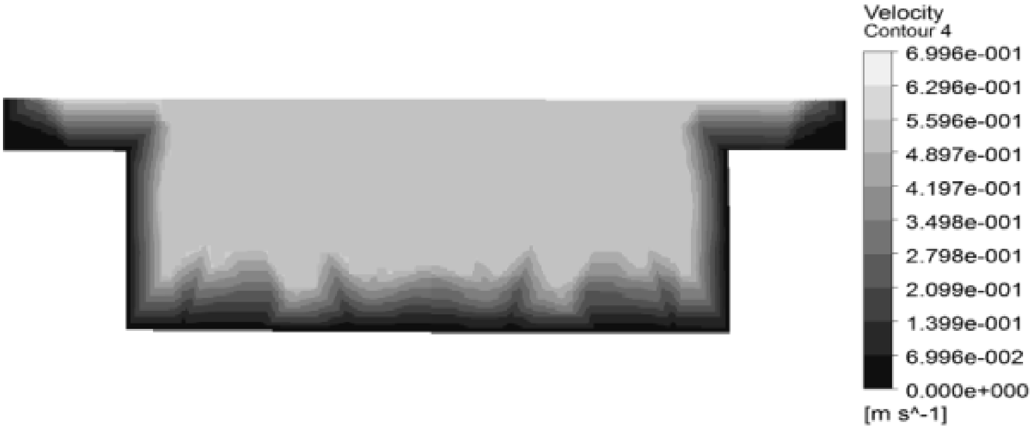

Fig. 4 (b) Contours of sec 2 by CFD simulation.

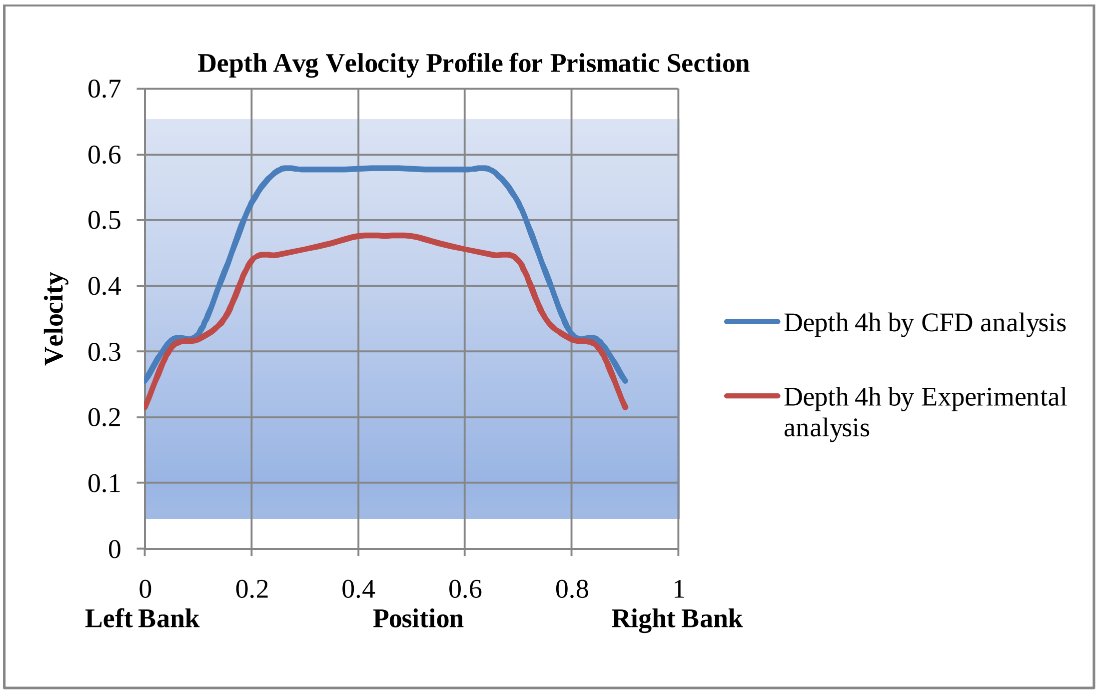

Fig. 5: Depth average velocity profile for sec 1(by both experimental and Computational results)

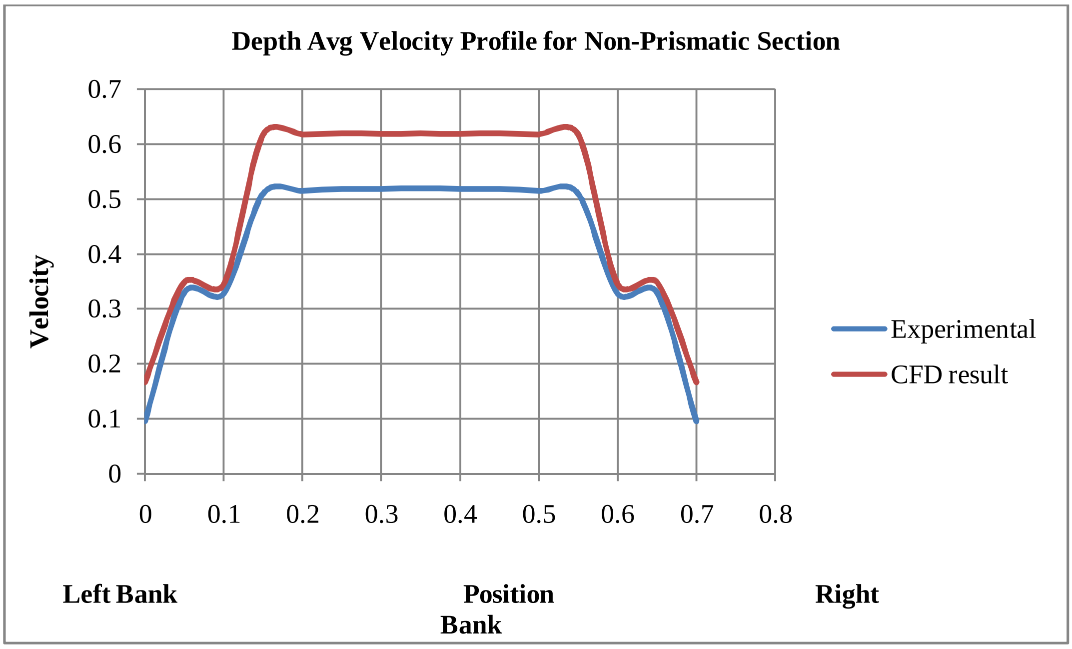

Fig. 6: Depth average velocity profile for section 2(by both experimental and Computational results).

5. CONCLUSIONS

1. Fig.3 (a) and Fig.4 (a) shows the velocity contours for sec-1 & sec-2 obtained by experimental results. Fig.3 (b) and Fig.4 (b) shows the velocity contours for sec- 1 &2 by Numerical analysis. Fig.5 and Fig.6 shows the both experimental and Computational depth average velocity profile for sec-1 & sec-2. 2. The results show that the CFD predictions accurately predict the velocity and depth average velocity for LES turbulence model.3. In both the cases velocity was over predicted when smooth walls were applied as expected. In this paper the flow velocity profile in non prismatic compound channel has been numerically modeled. The CFD model has been validated by comparing the results with actual measurement carried out with Preston tube.

References

[1] ANSYS, Inc. (2010, November*), ANSYS Workbench User's Guide,* Canonsburg, Pennsylvania, United States of America.

[2] Ahmed Kassem, Jasim Imran, and Jamil A. Khan (2003), Three-Dimensional Modeling of Negatively Buoyant Flow in Diverging Channel, **10.1061**/*ASCE* ***0733-9429-2003/129/12936***

[3] Bakker, A. (2008, February 3), Computational Fluid Dynamics Lectures: *Lecture 14*, Multiphase flow, Retrieved from The Colorful Fluid Mixing Gallery: http://**www.bakker.org/dartmouth06/engs150/**

[4] Bodnar, T. and Prihoda (2006), Numerical simulation of turbulent free-surface flow in curved channel. Journal of Flow Turbulent Combust **76**, **429-442**

[5] Booij (2003), Measurements and large eddy simulation in some curved flumes, *Journal of turbulence- 4,* **8-16**

[6] Bousmar, D., Wilkin, N., Jacquemart, J.H., Zech, Y., (2004), Overbank flow in symmetrically narrowing floodplains, *J. Hydraul. Eng., ASCE,* **130(4), 305-312**.

[7] Bousmar, D., Denis, B. &Zech, Y. (2004), Coherent flow structures in a converging compound channel, *Proc. River Flow 2004 Conference*, Naples, Italy, 1, **423-430**.

[8] Bousmar, D., and Zech, Y., (2002), Discussion of two-dimensional solution for straight and meandering overbank flows, *J. Hydraul. Eng*., **128(5), 550-551**.

[9] B. K. Gandhi, H.K. Verma and Boby Abraham(2010)Investigation of Flow Profile in Open Channels using *CFDIGHEM-2010,* **21-23**, AHEC, IIT Roorkee, India

[10] Ervine, D.A &Jasem, H.K. (1995), Observations on flows in skewed compound channels, *J. Wat. Marit. En. ICE,* **112, 249-259**.

[11] Fluent Inc. (2006), *FLUENT 6.3 User's Guide*, Lebanon, New Hampshire, United States of America.

[12] Ferziger, J. H., and Peric, M. (2002), Computational methods for fluid dynamics, 3rd Ed., Springer, New York.

[13] *Gambit 2.2* tutorial guide.

[14] Knight, D.W. &Shiono, K. (1990), Turbulence measurements in a shear layer region of a compound channel, *J.Hydr. Res., IAHR*, **28(2), 175-196**.

[15] Khazaee, M. Mohammadiun (2012), Effect of flow field on open channel properties using numerical investigation and experimental comparison, pp.**617-628**.

[16] Lien, H.C., Hsieh, T.Y., Yang, J.C. and Yeh, K.C. (1999), Bend flow simulation using 2D depth–averaged model, *Journal of Hydraulic Engineering-ASCE* **125, 1097-1108.**

[17] Lu, W.Z, Zhang, W.S, Cui, C.Z. and Leung, A.Y.T. (2004), A numerical analysis of free surface flow in curved open channel with velocity–pressure-free-surface correction, *Computational Mechanics*- **33, 215–224.**

[18] Omid Seyedashraf, Ali Akbar Akhtari, Milad Khatib Shahidi (2012), Numerical Study of Channel Convergence Effects on Flow Pattern in 90 Degree Bends, *9th International Congress, IUT*, Isfahan, Iran.

[19] Ramamurthy, A., Han, S., and Biron, P. (2013), Three-Dimensional Simulation Parameters for 90° Open Channel Bend Flows, *Journal of Computing in Civil Engineering-ASCA* **27(3), 282-291**.

[20] Rasool Ghobadian, Kamran Mohammadi (2011), Simulation of subcritical flow pattern in 180° uniform and convergent open-channel bends using SSIIM 3-D model, *Water Science and Engineering*, **4(3) 270-283**

[21] Rezaei, B. (2006), Overbank flow in compound channels with prismatic and non-prismatic floodplains, *PhD Thesis, Univ. of Birmingham*, U.K.