Proceedings Paper:

Polansky, J, Wang, M and Faraj, Y (2015) Numerical modelling of gas-liquid flow phenomena in horizontal pipelines. In: Proceedings. 12th International Symposium on Experimental Computational Aerothermodynamics of Internal Flows (ISAIF12), 13-16 Jul 2015, Genova, Italy. .

Proceedings of the 12th International Symposium on Experimental
Computational Aerothermodynamics of Internal Flows
13–16 July 2015, Genova, Italy

ISAIF12-041

Numerical modelling of gas-liquid flow phenomena in horizontal pipelines

Jiri Polansky[1], **Mi Wang**[1], **Yousef Faraj**[1]

1 Institute of Particle Science & Engineering
University of Leeds, LS2 9JT, Leeds, UK
j.polansky@leeds.ac.uk, m.wang@leeds.ac.uk, y.faraj@leeds.ac.uk

Abstract

Gas-liquid flows are omnipresent in industrial and environmental processes. Examples are the transportation of petroleum products [1, 2], the cooling of nuclear reactors [3, 4], the operation of absorbers [5], distillation columns [6], gas lift pumps [7] and many mores. Different input parameters induce topologically different flow patterns with different flow character and behaviour [7, 8] . The present study concentrate to adiabatic incompressible two-phase flow in horizontal pipeline with separated character [9, 10] ($U_{gas} < 10m/s$ and $U_{liquid} < 0.2m/s$) such as stratified wavy flow regime including typical multiphase instability (Kelvin-Helmholtz instability) [11, 12]. The Proper Orthogonal Decomposition (POD) [13], introduced by Lumpay (1967) [14] was used to extract synthetic information essential to understand and to model flow dynamics phenomena. POD in this study are used to identify flow structure in the horizontal pipeline specially under transient of separated flow regimes. The snapshot matrix are reconstruct for specific flow sections and regimes. Present decomposition method, in this case used to analyse CFD data, are originally testing and developing for future using to analyse experimental data obtained by process tomography system [15].

Keywords: multiphase flow, horizontal pipeline, CFD, LES, OpenFoam, Proper orthogonal decomposition.

Introduction

Considering a gas-liquid two phase flow, the liquid and gas are regarded as the continuous and dispersed phases respectively. Gas-liquid flows are commonly observed in many industrial processes such as oil and gas, chemical, pharmaceutical and nuclear industries. The relative distribution of the gas and liquid phases can take many different configurations depending on the process conditions, such as the flow rates of the gas and liquid. The configuration of the gas and liquid phases is known as the flow regime Wallis (1969) [16]. The flow regime describes the pattern of the inner structure of the flow and important hydrodynamic features such as volume fraction, phase and velocity distributions. Two phase flow regimes are often determined subjectively using direct methods such as the eyeballing method, high speed photography method and the radial attenuation method (Deng, et al. 2001) [17]. Empirical flow regime maps such as the Baker chart for horizontal flow [18] and and Hewitt and Roberts (1969) [19] for vertical flow are commonly used for approximate and rapid identification of the flow regime under specific operating conditions. However due to their approximate and subjective nature these techniques are not able to identify the prevalent multiphase flow regime with the required degree of accuracy.

In order to better understand the fluid dynamic nature of Gas-liquid multiphase flows this paper focuses on flow regimes and pressure drop identification using approaches based on Computational Fluid Dynamics and . Proper Orthogonal Decomposition techniques. Physical, mathematical and numerical models of horizontal flow regimes are developed and presented.

Model description

Physical and mathematical model

Physical model of the study phenomena is define as; horizontal unsteady, adiabatic multiphase turbulent flow of two incompressible, immiscible and Newtonian fluids. The transport properties of both phases are considered as a constant. The temperature dependences of viscosity, surfaces tension and other thermophysical properties are neglected for present study.

Mathematical model is based on the system of Navier-Stokes equations for the turbulent flow of incompressible fluids. For the turbulence model, the Large Eddied Simulation model is adopted. To find a suitable turbulence model in computations of the unsteady separated turbulent flow is extremely subtle and difficult. Mesh refinement are used to model the flow near the pipeline wall. The numerical model is solved using the Runge-Kutta method in the form of finite volumes. Coupled implicit scheme with the second order accuracy and default under relaxation factors was applied.

Numerical model

The basic characteristics of CFD model is apparent from Table 1., the Computational domain and calculation grid are shown on Figure 2. Tree-dimensional non-structured computational mesh includes 1e6 cells. Inlet components of the velocity, gas void friction and exit static pressure values are taken as a boundary conditions.

The present numerical model concentrates on gas-liquid flow in a horizontal pipeline with typical flow parameters for separated flow regime, see Figure 1. CFD solver interFoam, introduced in code OpenFoam ver.2.3. [20] is used to identify significant flow parameters such as velocity, turbulence intensity, pressure and gas volume friction profiles and free surface dynamics behaviour, such as wave propagation. In our case, the solver interFoam is recompiled and optimised [21] for 2 incompressible, isothermal immiscible fluids using a Volume of Fluid phase-fraction based interface capturing approach [22]. Multi-dimensional limiter for explicit solution MULES is used for a part of the calculations, in addition the new Predictor-Corrector Semi-Implicit variant of MULES, introduced in OF ver.2.3., is tested for numerical stability and convergence of modelling acceleration.

Proper Orthogonal Decomposition

Proper Orthogonal Decomposition (POD) [22, 23] finds applications in computationally and experimental processing large amounts of high-dimensional data with the aim of obtaining low-dimensional descriptions [24, 25]. The snapshot approach suggested by Sirovich (1987) [26] was applied. Using POD, time independent basis functions were extracted from the data and the governing equations of the numerical solver were projected onto the basis functions to generate reduced-order models. In the reduced-order models (ROMs) [27, 28] the large number of partial differential equations were replaced by a much smaller number of ordinary differential equations. These reduced-order models were applied to several reference cases; Liquid mass flow rate equal 100 kg/sm^2, gas mass flow rate between 1 and 10 kg/sm^2.

From mathematical point of view, finite dimensional representations of a function is sought in terms of a basis modes which allows a linear approximation to be constructed:

$$U_m = \sum_{j=1}^{M} a_j(x)\phi_j(t). \tag{1}$$

Assume a set of representative datasets (snapshots), u^k , and the basis functions are chosen such that they best describe a typical member of the sample data. The basis functions should be chosen so that they maximise the averaged projection of our ensemble of functions onto the basis functions. To estimate set of POD bases functions, Python library MODRED [29] was used.

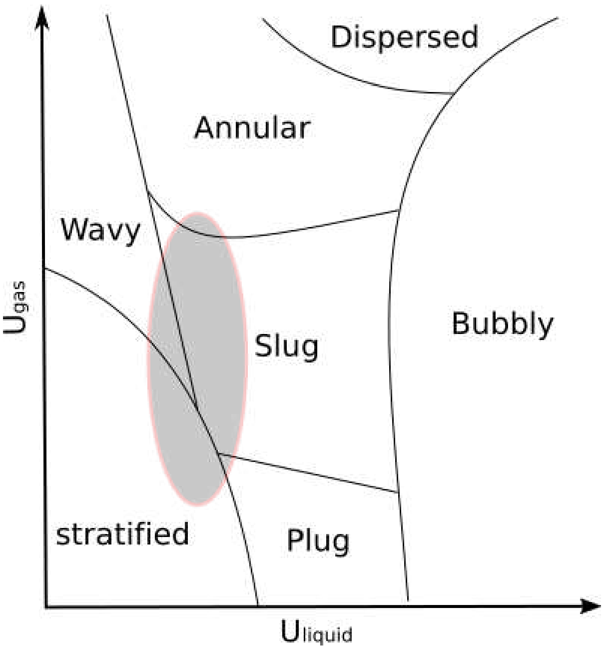

Figure 1 Horizontal flow regimes map, range of present CFD model application

Table 1 CFD model description

Parameter	Value
Solver	interFoam
No. of cells	1e6
CPU time	260 h
Heat transfer	No
Gas compressibility	No
Turbulence model	LES
Superficial velocity:	$U_{gas} < 10m/s$ $U_{liquid} < 0.2m/s$
Phases transport properties	Air & Water
Inlet boundary condition	mass flow rate of each phase
Outlet boundary condition	static pressure
Diameter, length	50 mm , 3.5 m

Results

Pressure losses

Industry sector is mostly interested in the pressure losses of specific flow pattern, transient behaviour as a potential risk of pipeline damage and so on. In order to understand the dynamic nature of gas-liquid multiphase flows the publication focus to unsteady behaviour identification by CFD simulation. Lockhart and Martinelli correlation was used for pressure friction drop and void fraction comparison with CFD. The estimated pressure drop for separated horizontal flow regimes are shown on shown in Figure 3. In the area of interest, water superficial velocity below 0,2 m/s and gas superficial velocity below 10 m/s

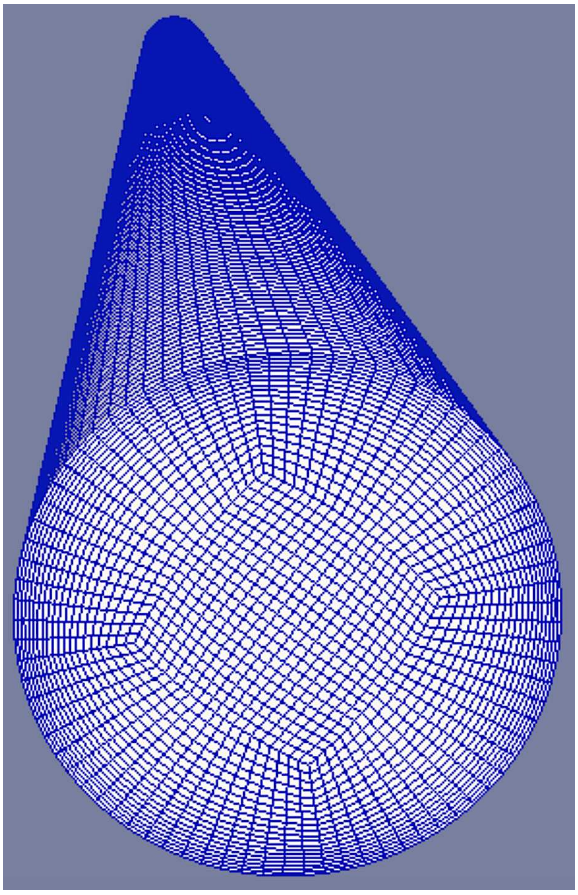

Figure 2 Computational model of horizontal pipeline

the results agreement is about 20%. The accuracy of CFD pressure estimation dependent on the length of CFD record (typically about 2-3 s) and monitoring plans position to boundary condition (typically more then $l_{plane}/d > 40$). Specially short distance from inlet boundary can negatively affect the averaged value of pressure drop, because the fully develop multiphase flow could be expected for more then $l/d > 100$. The requested length of CFD domain is dependent on type of boundary conditions. The using of known velocity and concentration profiles (obtain by theoretical or experimental investigation), can positively affects the convergence of CFD calculation and required length .

Flow visualisation

Unsteady multiphase simulation is performed in order to generate a database of snapshots of matrices for proper orthogonal decomposition analyses. As a secondary effect of simulation is multiphase flow visualisation. Flow visualisation here could serve for CFD model validation as well. The CFD data are tested for accuracy of pressure loss predictions and the flow regime prediction correctness. The Figure 4 show us slightly wavy stratified flow regime, Figure 5 transient regime between Slug and annular flow and Figure 6 fully developed annular flow. The flow regime estimated by CFD correspond with the horizontal flow patterns map [9].

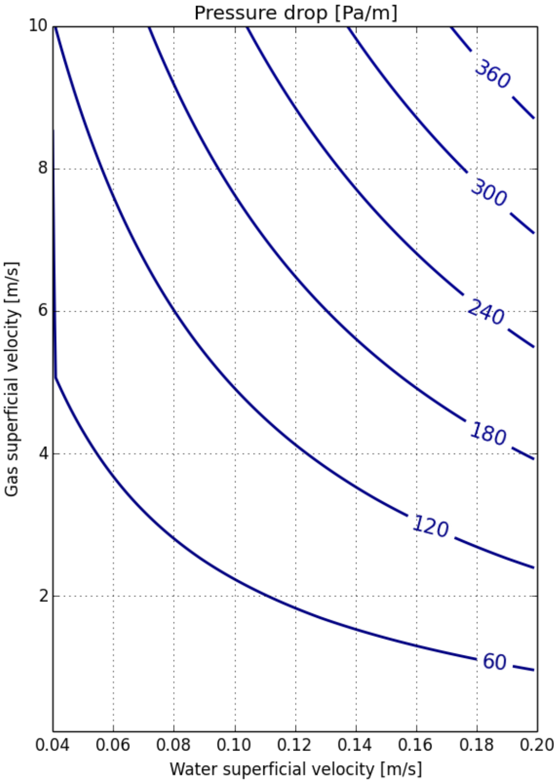

Figure 3 Pressure losses, correlation and CFD estimation

By results visualisation, we are also able to study typical multiphase flow instability initialisation and propagation, see Figure 6, wave flow regime.

POD post-processing

Different flow regime can be characterised by different POD modes, see Figure 8. To estimate fist dominant bases function allow us recognise, with certain probability, the flow regime base on signal records from multiphase flow measurement.

The main aim of present CFD simulation is creating and filling the database by characteristic POD bases functions for different flow conditions, see Figure 9. Highlight blocks in the scheme present current state and study contribution.

In advance, the CFD data allow us to correlate typical measurement parameters: concentration and disperse phase velocity, with other significant parameters such as continuous phase velocity, turbulence intensity, vorticity, pressure losses and so on.

Conclusions

The CFD model based on interFoam solver was developed, tested and used for numerical simulation of gas-liquid horizontal flow. The application of model is limited by separated flow character. The area of solver using is demonstrated on Figure 1. The characteristic of the CFD

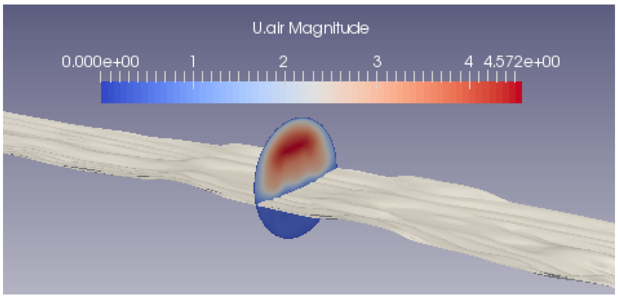

Figure 4 Free surface of stratified flow regime, velocity field

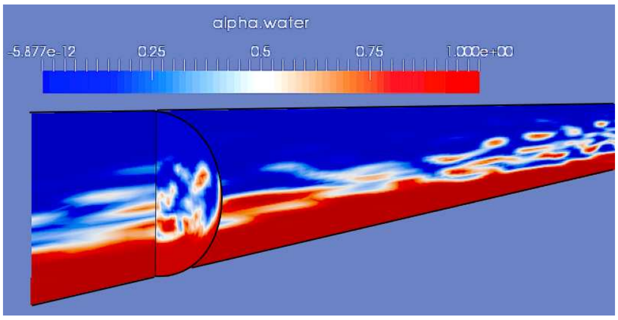

Figure 5 Transient flow, semi-annular flow regime

model based on interFoam solver is shown on Table 2. For dispersed types of flow regime simulation is more suitable twoPhaseEulerFoam OpenFoam solver, which is based on Euler-Euler approach, see model description on Table 3. In this solver, both the phases are described using the Eulerian conservation equations and each of the phases are treated as a continuum in this approach.

Present results demonstrated the influence of boundary conditions on quality of results and convergence. The optimal way of CFD calculation is to use the known velocity and concentration profiles from experimental investigation. Otherwise, the length of calculation domain and CPU time could be enormously high.

Present study contribute the Databases of typical POD function, see Figure 9, specially for separate flow regimes. The CFD model and database of functions could be validate by multiphase flow experimental modelling, such as process tomography, ultrasound velocimetry, PIV and so on.

Acknowledgements

This research was supported by European Association of National Metrology Institutes, project ENG58, Multiphase flow metrology in oil and gas production .

References

[1] FREESE T. L.& Associates, Inc., The Transportation of Petroleum and Derived Products in the American Market, 2006

[2] Energy, transport and environment indicators, Eurostat, 2014, ISBN 978-92-79-41256-1

[3] COPPOLANI P., HASSENBOEHLER N., JOSEPH J., La chaudiere des reacteurs a eau sous pression, ISBN: 2-86883-741-7 Paris, 2004

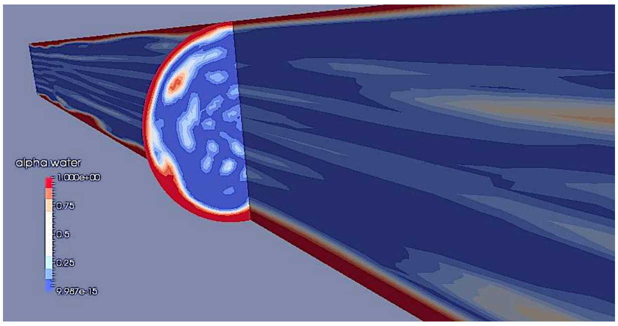

Figure 6 Fully developed annular flow regime

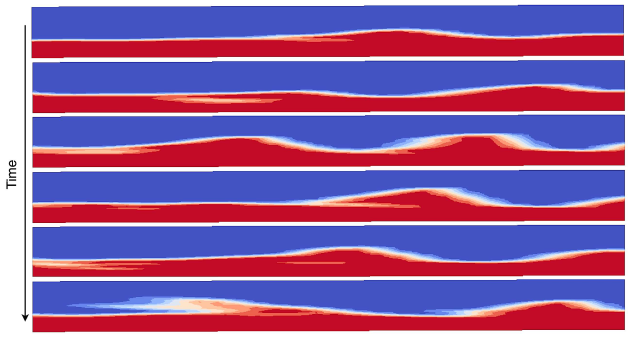

Figure 7 Wavy flow regime, wavy formation and propagation

Table 2 Characteristics of interFoam solver

Parameter	Value
Number of phases	2
Gas compressibility	No
Miscibility	No
Heat transfer	No
Turbulence model	LES, RANS
Suitable flow conditions:	$U_{liquid} < 20m/s$ $U_{gas} < 20m/s$
Suitable for flow regime:	Stratified & Wavy Annular (with limitation) Slug & Plug

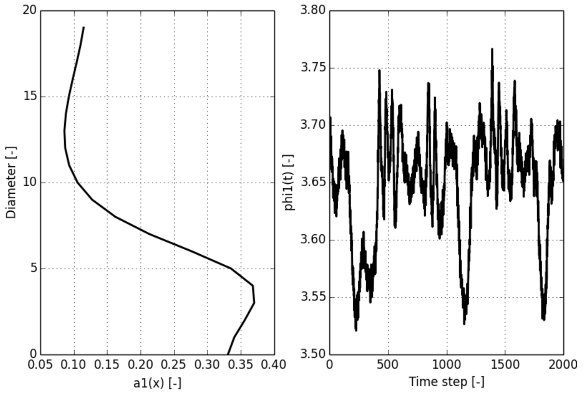

Figure 8 POD bases functions, 1st mode, stratified flow regim

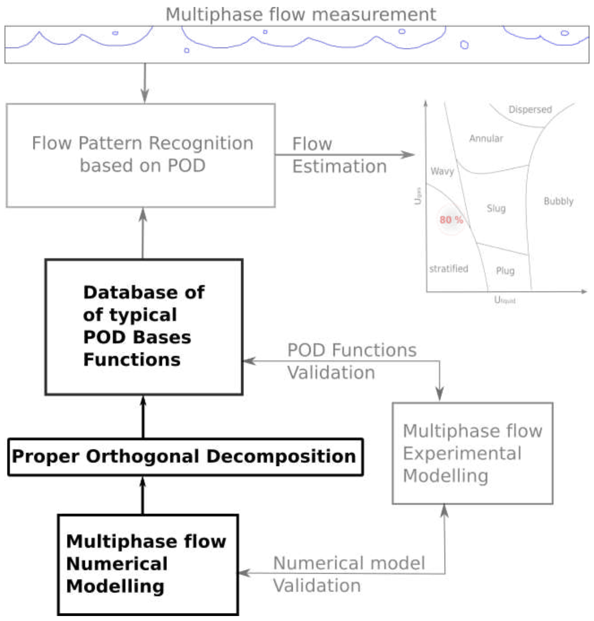

Figure 9 Scheme of Flow pattern recognition based on POD

Table 3 Characteristics of twoPhaseEulerFoam solver

Parameter	Value
Number of phases	2 (one phase dispersed)
Gas compressibility	No
Miscibility	No
Heat transfer	Yes
Turbulence model	LES, RANS
Suitable flow conditions:	$U_{liquid} > 20m/s$
	$U_{gas} > 20m/s$
Suitable for flow regime:	Annular (with limitation)
	Bubbly & Dispersed

[4] DELHAYE J.M., Thermohydraulique des réacteurs, ISBN 13: 9782759810949, Paris, 2008

[5] CORA J., ORDONEZ S. DIAZ E., Mass Transfer Operations: Absorption And Extraction, Encyclopedia of Life Support Systems

[6] HALVORSEN I.J., SKOGESTAD S., Distillation Theory, Norwegian University of Science and Technology, 2000

[7] TAKACS G., Gas Lift Handbook, ISBN-10: 0878148051, 2004

[8] THOMAS J. HANRATTY, Physics of Gas-Liquid Flows,University of Illinois at Urbana-Champaign, ISBN 978-1-107-04120-2

[9] BERTOLA V, Modelling and experimentation in Two-Phase Flow, *Springer-Verlag Wien New York in 2003, ISBN 978-3-211-20757-4 ISBN 978-3-7091-2538-0 (eBook) DOI 10.1007/978-3-7091-2538-0*

[10] HEWITT G.F., Empirical and Phenomenological Models for Multiphase Flows II. Horizontal Flows, Department of Chemical Engineering Imperial College London, UK, 2015

[11] YADIGAROGLU G., Instabilities in two-phase flow, Swiss Federal Institute of Technology, Zurich, 2015

[12] PRASSER H.M. , Basic model for two-phase flows, Swiss Federal Institute of Technology, Zurich, 2015

[13] BONNET J.P., COLE D.R., Stochastic Estimation and proper orthogonal decomposition: Complementary techniques for identifying structure, Experiments in Fluid 17 1994

[14] LUMLEY J. L. The structure of inhomogeneous turbulent flows. In Atmospheric Turbulence and Radio Wave Propagation (ed. A. M. Yaglom & V. I. Tartarsky), p. 166. Nauka, Moscow. 1967

[15] RAMSKILL N.P.; WANG M. Measurement and recognition of gas-liquid multiphase horizontal flow regimes using electrical resistance tomography in: 6th World Congress in Industrial Process Tomography, pp.159-167. International Society for Industrial Process Tomography. 2010.

[16] WALLIS G.B., One-dimensional Two-phase Flow. New York : McGraw Hill, 1969. p. Chapter 1.

[17] DENG, X, et al. 2001. The design of a dual plane ERT system for cross correlation measurement of bubbly gas/liquid flow. Measurement Science and Technology. 2001, Vol. 12, pp. 1024-1031.

[18] BAKER D., Simultaneous Flow of Oil and Gas, Oil and Gas J, 53, pp 183-195 1954.

[19] HEWITT G.F., ROBERTSON D.N., Studies of Two-Phase Flow Patterns by Simultaneous X-ray and Flash Photography, Rept AERE-M2159, UKAEA, Harwell, 1969.

[20] OpenFOAM The Open Source CFD Toolbox, User guide, ver.2.3.1, 2014

[21] OpenFOAM, The Open Source CFD ToolboxProgrammerâs Guide, ver.2.3.1, 2014

[22] ADRIAN R.J., On the role of conditional averages in turbulence theory, In J. L. Zakin & G. K. Patterson, editor, Turbulence in Liquids, pages 323â332, 1977

[23] TRIGGVASON G., SCARDAVELLI R. , ZALESKI S., Direct Numerical Simulations of GasâLiquid Multiphase Flows, The volume-of-fluid method pp. 95-132 , ISBN:9780511975264

[24] LAURENT C., BERGMAN M., Post-processing of experimental and numerical data, POD: an overwiew, von Karman Institute for Fluid Dynamics, Lecture Series, 2003

[25] VOLKWEIN, Proper Orthogonal Decomposition: Theory and Reduced-Order Modelling, University of Konstanz, lecture note, 2013

[26] SIROVICH L., Turbulence and the dynamics of coherent structures, Quarterly of Applied Mathematics XLV, No.3., pp.561-571, 1987

[27] BRENNER T.A., Practical Aspects of the Implementation of Reduced-Order Models Based on Proper Orthogonal Decomposition Texas A&M University, PhD theses, 2011

[28] GORDEYEV S., POD,LSE and Wavelet decomposition: Literature Review, University of Notre dame

[29] BELSON B.A., TU J.H., ROWLEY C.W., A parallelized model reduction library, Princeton University