The use of Computational Fluid Dynamics (CFD) for slugging prediction in a two-phase geothermal production system

Andrew Marsh[1], Kava Crosson-Elturan[2], Karl Spinks[3], Michael Lee[4]

[1] Upflow, PO Box 61, Taupo 3351, New Zealand

[2] Matrix Applied Computing, PO Box 56-316, Auckland 1446, New Zealand

[3] Ngati Tuwharetoa Geothermal Assets, PO Box 334, Kawerau 3169, New Zealand

[4] Contact Energy, PO Box 10742, Wellington, 6143, New Zealand

andrew.marsh@upflow.nz

Keywords: *CFD, two-phase, production system, slugging, simulation, process design, separator*

ABSTRACT

Slugging is a flow phenomenon that occurs when gas and liquid phases coexist in a pipeline under certain conditions, causing pressure fluctuations and operational challenges. In geothermal production systems, slugging can compromise stable operation and safety of the power plant and separation equipment. It is important to predict and avoid slugging conditions in two-phase geothermal pipelines.

In this paper, we present a Computational Fluid Dynamics (CFD) approach for slugging prediction in a planned two-phase delivery pipeline for the Ngati Tuwharetoa Geothermal Asset (NTGA) Geothermal Field in Kawerau, New Zealand. We use CFD to model the pipeline geometry and the multiphase flow conditions and examine the results. A weakness in the results presented is that the pipeline being modelled has not yet been built, so the accuracy of model predictions has not yet been validated with field data

We also present a study of two-phase flow into a separator and how this can lead to structural vibration. The focus was on adjusting the incoming pipework to reduce unsteady loads from the high liquid fraction. Achieving smooth, steady liquid entry helps prevent wall splashing and cyclic loading. The results were calibrated to movements measured in the operating separator.

One of the historic challenges of setting up a CFD model for slugging prediction is managing the size of the model and the calculation time for such large and complex systems. Our results show that, with the assistance of cloud-computing, a CFD approach to slugging prediction is now feasible and cost-effective. Our premise is that the CFD approach can provide useful insights for the design and operation of two-phase geothermal pipelines.

1. INTRODUCTION

Geothermal energy is a renewable source of power that can provide electricity and heat for various applications. However, developing and operating geothermal resources poses several technical challenges, including the design and construction of two-phase geothermal fluid delivery pipelines. These are pipelines that transport a mixture of steam and water from the production wells to the power plants or to other downstream end users. The flow regime of the two-phase fluid in the pipeline depends on various factors, such as the pressure, temperature, mass flow rate, fluid chemistry, and pipe geometry. Predicting the flow regime is essential for estimating the pressure drop, pipe support loadings and route for the pipeline, as well as for avoiding operational problems such as slugging and water hammer.

1.1 Risks of unsteady flow

Slugging is a flow phenomenon that occurs when large pockets of liquid accumulate amongst the gas flow. Slugging can cause severe fluctuations in the pressure and flow rate at the outlet of the pipeline, which can damage equipment and affect the performance of the power plant. Moreover, slugging can also occur during the start-up and shut-down of the pipeline, when the fluid velocity is low and the liquid fraction is high. In these situations, the liquid can accumulate at the low points of the pipeline and form a slug that can be pushed by the gas as the flow increases, creating a surge that damages supporting structures and/or exceeds the design pressure of the pipe.

1.2 Designing two-phase pipelines and separators

Designing and building two-phase geothermal pipelines requires careful consideration of the terrain, the fluid properties, and the operational conditions to avoid or mitigate the risk of slugging. However, conventional methods for predicting the flow regime and the slugging behaviour of two-phase fluids are often based on empirical correlations (Harrison, 1975, Mandhane, 1974 and Spedding, 1980) or simplified models that may not capture the complexity and variability of geothermal systems. There is a need for more advanced and reliable tools that can provide a detailed and accurate representation of the multiphase flow dynamics in geothermal pipelines.

Similarly, separator designs are based on empirical methods devised 40-60 years ago (Bangma, 1961, Lazalde-Crabtree, 1984) with very little changes noted since then (Jung, 1989 and Zarrouk, 2014, Rivera-Diaz, 2021). While the development of these methods have enabled considerable progress and predictability in these designs (McLellan, 2021), they are often extrapolated to the conditions of interest and are based upon simplified flow patterns and therefore not always applicable.

2. COMPUTATIONAL FLUID DYNAMICS

2.1 What is CFD?

Computational Fluid Dynamics (CFD) is a numerical method that can solve the governing equations of fluid motion and thermodynamics for any geometry and boundary condition. CFD uses finite volumes to simulate the behaviour of fluids in complex systems by dividing the domain into small cells and solving the equations of conservation of mass, momentum, and energy for each cell. By doing this, CFD can obtain detailed information about the flow velocity, pressure, temperature, and other variables at any point in the domain.

CFD has been applied to geothermal problems involving two-phase flow increasingly in recent time as the computational power has become more accessible (Purnanto, 2013, Siwach, 2015 and Osato, 2016).

2.1 How does CFD work out two-phase flow?

CFD can model two-phase flow where there are free surfaces by using a multiphase model that tracks the interface between the phases. One of the most common multiphase models is the Volume of Fluid (VOF) model, which assigns a fraction of each cell to each phase and solves a transport equation for each phase fraction. The VOF model can capture the shape and motion of the free surface, as well as the effects of surface tension, interfacial momentum transfer, and phase change.

2.3 Advantages

The advantages of using CFD for two-phase flow are that it can provide high-resolution data on the flow variables, such as pressure, temperature, velocity, and phase distribution. Additionally, the method can handle complex geometries, transitions and boundary conditions of cross-country two-phase lines to ensure that they do not produce excessive support forces and produce flow conditions compatible with downstream separation facilities.

2.4 Disadvantages

The disadvantages are that full line models require a lot of computational resources and time, and that the accuracy of the results depends on the accuracy of the sub-models and parameters used in the simulation.

3. NTGA TWO-PHASE DELIVERY STUDY

3.1 Background

During preliminary design for a two-phase delivery system at Kawerau field, NTGA recognised that uphill topography combined with low enthalpy could potentially result in slugging flow. Existing pipelines conveying similar fluid in the area tend to produce long wavelength slugs, highlighting the elevated potential for slug flow along the planned pipeline path.

The primary concern relates to the large diameter pipeline being used to keep pressure drop low over the 3 km total delivery length from well to separator, in combination with a high liquid fraction from adding brine to the flow, and the uphill topography for a new 1.6km section of pipeline delivering to the separator (Figure 1).

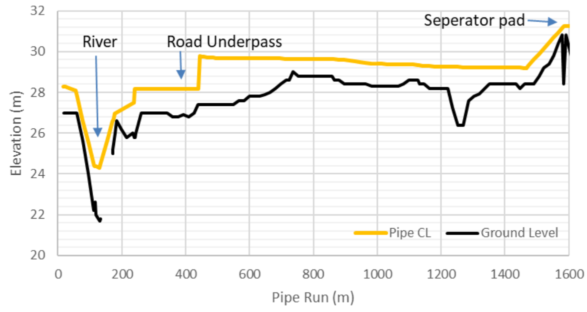

Figure 1: Topography of the new pipeline section.

3.2 Objective

The objective of the CFD study was to determine the flow regime, particularly the risk of slug flow and the nature of any hazards, for the planned pipeline. Visualisations of the flow regime provided by CFD simulations under various process conditions would complement the ongoing design process in terms of evaluating options and help to quantify the risk by providing another point of reference to compare to the 1-D process model based on empirical methods.

3.3 Methodology

The methodology included testing a 120 m straight pipe model at various inclinations to calibrate the simulation and compare with expected slug flow, then modelling a two-phase pipeline from where it branches with the existing line along the proposed pipe route to the new separator entry.

Successive models increased the liquid fraction at a constant total flow rate and, separately, tested higher flow rates at the nominal liquid fraction to determine how close the operating conditions were to the slugging condition.

Finally, a 40 m slug of liquid was introduced into the model to understand whether a slug from upstream of the new pipeline can trigger a sustained flow regime change or whether it would dissipate.

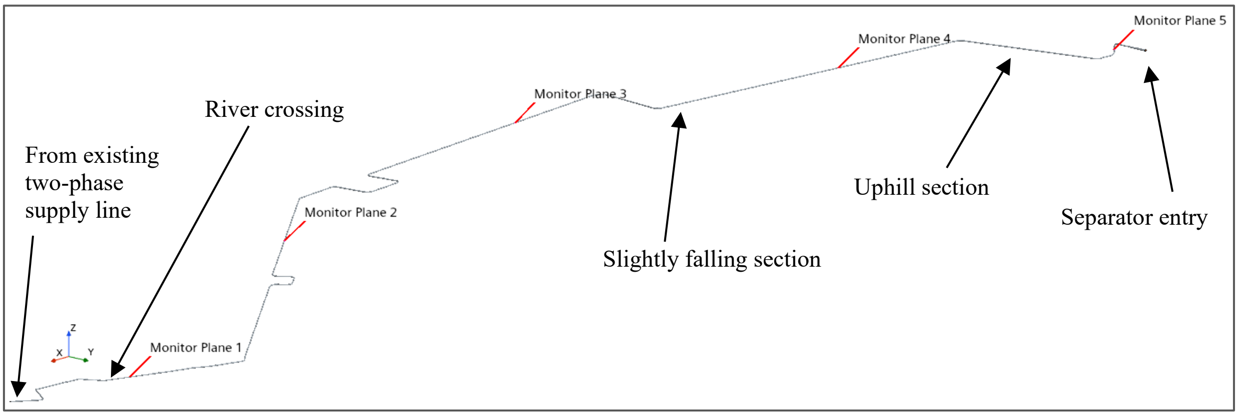

Figure 2: Full length of 1.6 km line modelled.

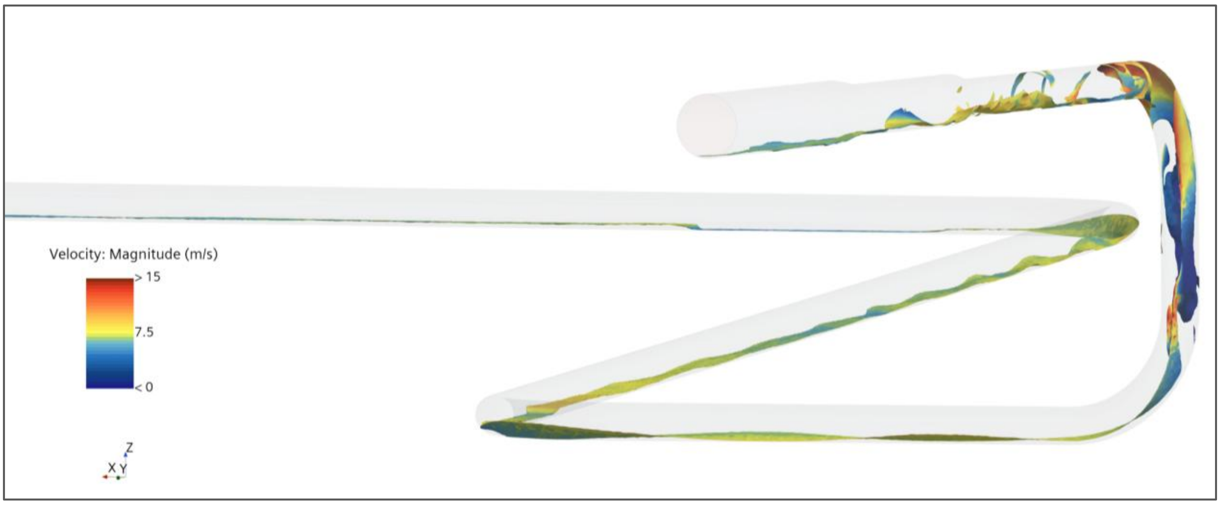

Figure 3: Final section of line leading into the separator showing the interface surface between gas phase and liquid phase, coloured by the velocity. Stratified flow leads into an unstable wavy-annular flow in the vertical riser.

The pipe diameter is 882 mm (36”) with an expanding diffuser section before the separator. The final iteration of the model had a total length of 1.6 km (Figure 2) and a total fluid volume of 1000 m^3.

Animations and data extracted from the model for pressure, liquid build-up and cross-sectional phase distributions were collected to support conclusions.

3.4 Results

The results of the study indicated that mainly stratified flow can be expected with small waves at a few-second intervals (Figure 3). This can be disturbed by bends sometimes to create fast-moving liquid, swirling over the upper surface of the pipeline. The slug introduced at the beginning of the system dissipated along the pipeline showing that the conditions will tend to stabilise the flow regime during upsets.

The study concluded that the planned elevation profile and operating conditions of the new pipeline do not significantly disrupt the flow regime. The risk of poor operation and damage is not zero, but the study supports adding this new piece of pipeline to the system, as it does not seem to particularly add to the risk. It is reasonable to expect it to perform as designed, subject to taking the usual commissioning precautions to confirm this.

4. SEPARATOR STUDY

4.1 Background

The Contact Energy Tauhara Power Station experienced vibration in its Intermediate Pressure (IP) separator during abnormal operating conditions while undertaking commissioning activities, requiring a change to the design operating conditions. One of the consequences were abnormal vibrations in the fluid handling system, pipework and IP separator

4.2 Objective

This study aimed to investigate the cause of these vibrations and propose solutions to restore normal operating conditions without compromising the safety and integrity of the separator and associated pipework.

4.3 Methodology

The study used CFD to simulate the flow regime and assess the forces applied by the geothermal fluid to the piping, separator, foundation and the dynamic response of the structure to these forces.

CFD simulations used Star-CCM+ software to model the fluid dynamics within the IP separation system (Figure 4). The fluid volume included the high-pressure (HP) level control valves (LCVs), inlet manifold, riser, separator, loop seal, brine tank outlet, and steam outlet. Various steady-state scenarios were simulated to mimic known operating points and provide insights into the fluid flow regimes and their impact on vibration levels.

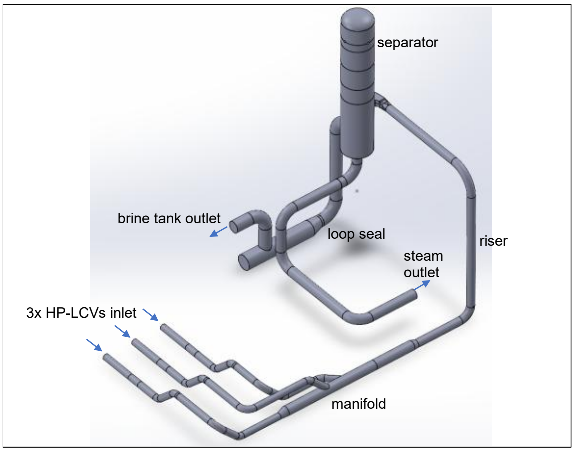

Figure 4: Separator component fluid volumes included in the CFD model scope.

The CFD model was meshed, and boundary conditions were applied at inlets and outlets. Sensitivity checks were conducted on turbulence, mesh size, time step, inlet type, energy/phase transfer, and simulation duration to ensure accurate results. The initial results were compared to actual vessel movement to arrive at a calibrated model. The simulations were evaluated by viewing animations of the liquid-steam bounding iso-surface and analysing the resulting force components applied to the separator vessel surfaces using the Fast Fourier Transform (FFT) method, comparing these to measured data (e.g. Figure 5).

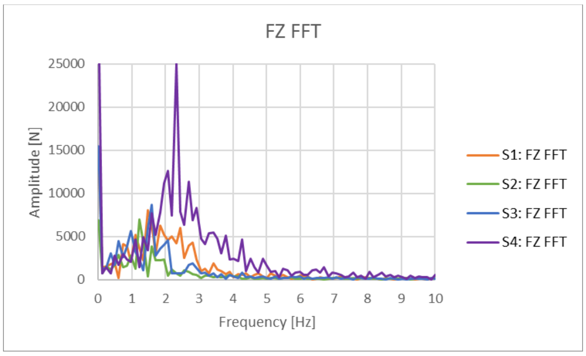

Figure 5: Example of comparative FFT analysis of fluid forcing applied to the separator structure.

This study also utilised a more advanced method, Fluid-Structure Interaction (FSI) to couple the forces imparted by the fluid to the structural response of the separator vessel and foundation (Figure 6). This method enabled the prediction and correlation of vibration amplitudes, and ultimately the stress amplitudes at every point in the vessel.

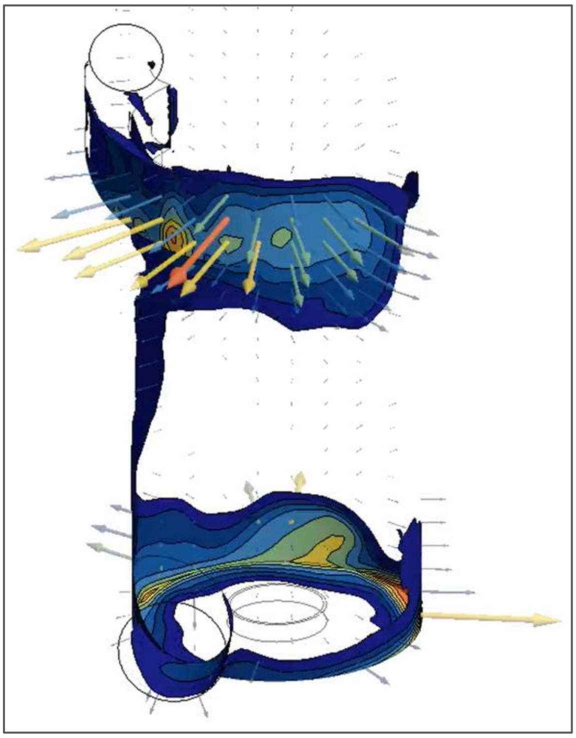

Figure 6: Diagram representing liquid rotation (blue surfaces) and the resulting force (arrows) applied to the wall of the separator drum.

4.4 Results

The simulations revealed that the current inlet pipework configuration contributed to the vibrations in the IP separator. The flow regime transitioned from stratified to wavy-annular in the vertical riser, leading to intermittent liquid flow into the separator. This intermittent flow regime resulted in oscillatory forces applied to the separator structure, intensifying at higher flow rates and lower pressures (Figure 7).

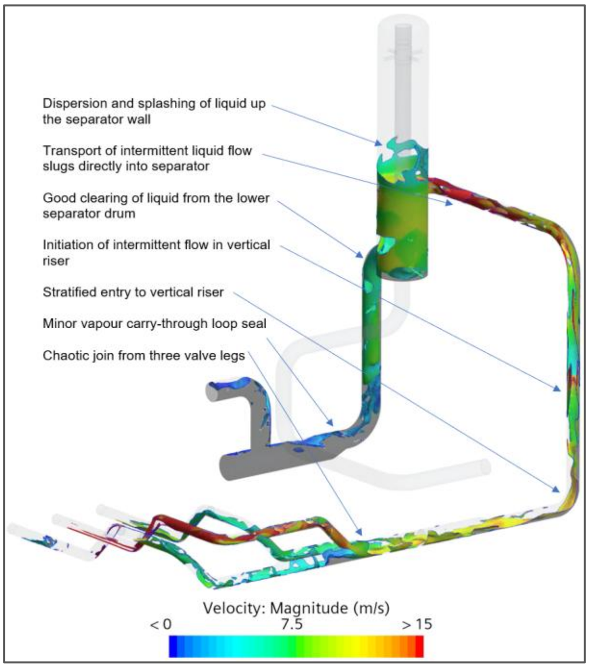

Figure 7: Features of two-phase flow in the separator.

Several modifications to the inlet pipework were evaluated to identify potential solutions (Figure 8). The most promising modification involved routing the three individual runs downstream of the level control valves vertically and merging them into a large manifold at the height of the separator entry. This configuration disrupted the intermittent flow in the risers, allowing a less variable flow to enter the separator without undermining effective droplet separation conditions inside the separator vessel. The results indicated that modifications could reduce the average excitation force magnitude by up to 80% and shift the excitation frequency away from the structure's natural frequency.

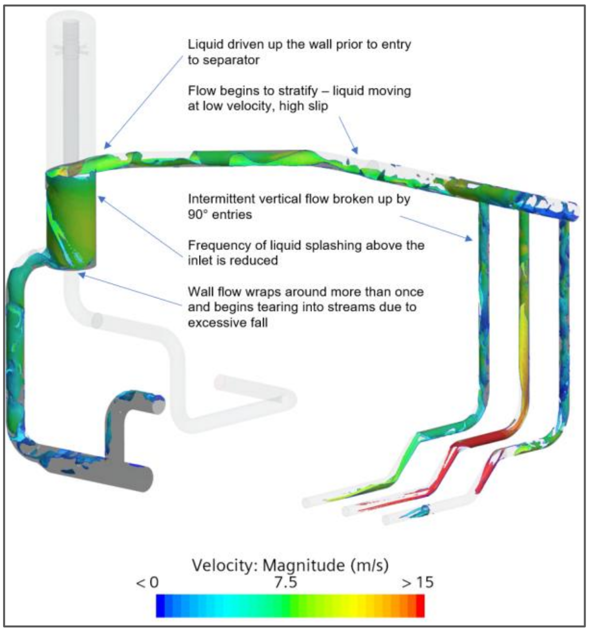

Figure 8: Modified entries alter the flow regime entering the separator to reduce forces.

Other modifications also showed potential utility but required less extensive pipework and support modifications. The study concluded that modifications to both the inlet pipework and the separator design were effective at achieving vibration reduction, however, in this instance, other structural solutions were implemented to resolve the issue and reduce vibration to an acceptable long term operational level.

The CFD study for the Tauhara IP separator provided valuable insights into the causes of excessive vibrations and proposed effective modifications to mitigate these issues. Despite this being a large and complex fluid and structural interaction model, it was possible to run dozens of scenarios allowing a wide array of solutions to be explored and evaluated.

5. FURTHER APPLICATIONS OF TWO-PHASE CFD

Computational Fluid Dynamics (CFD) is a powerful tool for simulating and analysing the behaviour of two-phase flows in complex geometries and operating conditions. However, traditional CFD methods are often limited by the computational cost and memory requirements of solving the governing equations for large domains and long time scales. Recent advances in CFD technology, such as the use of cloud computing, have enabled the development of models for very large two-phase system volumes, including long pipelines, that can capture the spatial and temporal variations of the flow regime, pressure drop, and phase distribution. These models can be used for design optimisation, performance evaluation, and troubleshooting of two-phase systems in various applications, including new and existing geothermal power plants.

Cloud computing offers several benefits for CFD applications, such as scalability, flexibility, reliability, and cost-effectiveness. By using cloud computing, CFD users can access high-performance computing resources on demand, without the need to invest in expensive hardware or infrastructure. Cloud computing also allows CFD users to run multiple simulations in parallel, reduce the turnaround time, and increase the accuracy and resolution of the results.

Below are several examples of other potential applications for using CFD to assist in geothermal plant design and operations.

5.1 Steam Field Design

It is probably not realistic or worthwhile to complete a steam field design from scratch using CFD simulations as part of the workflow. Routing of pipelines and sizing may iterate many times over early in a design and the overhead of maintaining a 3D model and simulating a range of operating conditions would likely slow the process. Therefore, a good 1-D process simulator based off well-established empirical correlations for pressure drop and flow regime is still the most viable tool for early design.

Once early iterations have been completed, a preferred arrangement been settled upon and problematic areas identified, then a CFD model (or multiples) may be usefully completed to increase confidence in the design, or to problem solve difficult features, such as constrained pipeline corridors, abrupt changes in elevation, or complex geometry. This could include a fluid structure interaction where the flow model applies loads to a structure (e.g. pipeline supports, or a vessel nozzle or base) to validate specifications for the mechanical design. This could mitigate against damage due to occasional slug loading or potentially pre-empt vibration issues.

5.2 Acid pH-mod and NCG injection systems design

The capabilities of CFD in gas-liquid and liquid-liquid systems extends its capabilities into increasingly important realms of chemical treatment systems for inhibition of mineral scaling, such as acid dilution into cooled and concentrated geothermal brines. Dissolution of geothermal gas mixtures into condensate or brine streams to facilitate reinjection and inhibition of scaling from the brine is also a growing area of interest and study.

Tackling such problems requires a multiphysics approach, incorporating two-phase flow, heat transfer, transport of dilute substances, gas dissolution and chemical reactions to faithfully resolve the system. The multiphysics approach allows these domains to be solved in parallel and cross-referenced, to refine results of each physical process according to the others. Through advances in multiphysics approaches in recent years it is possible to apply commercial tools to these problems in a practical manner and directly design tailored solutions to the complex geothermal context. For this reason, Upflow has invested in the COMSOL Multiphysics package for its consulting, innovation and research workstreams.

5.3 Troubleshooting and investigation

Another useful application for deploying a CFD model is to investigate flow and vibration issues in a plant once in operation. It is often difficult to visualise the nature of the problem in these systems since there is only sound and movement to signal what is going on. Inspections inside the pipeline might indicate patterns of flow imprinted on the walls but these can be misleading. Where excessive vibration or a failure has occurred, then a CFD visualisation of the two-phase flow condition can offer the insight and evidence needed to narrow down on probable cause.

5.4 Future paradigms

Since applying CFD as the only modelling tool is not yet the best approach (due to computational intensiveness and modelling time) a hybrid approach may be best.

There are several ways to go about this. An extensive study of CFD results compared to the established empirical relationships could highlight where the predictions align and diverge. Where there is good cause, new correlations may result. This would hopefully improve predictability of 1-D models and flow regime maps, especially where they have been extrapolated outside the original test conditions (to larger diameter pipelines for example).

Extending the study to some common geometries such as bends, vertical legs, junctions, expansion loops, may extend the predictive ability of 1-D models, allowing them to provide even more confidence. Vertical legs at the entry to cycle separators are a particular area where the conventional design could be improved. The standardisation of new geometries that perform better in a range of conditions would be a very useful reference for steam field designers.

Beyond this there are Reduced Order Models (ROMs) that can take a range of detailed CFD results and extend them out to imply results to other models. This is a sophisticated version of what is described above.

Ultimately, increasing computing power and engineer's skillsets may enable rapid design iteration using 3D models and full physics simulations from the outset to simplify and lend more confidence to the designer's workflow.

6. CONCLUSION

This paper explores the growing usefulness of Computational Fluid Dynamics (CFD) in the geothermal sector, highlighting how historical challenges in its application are being addressed by modern tools and computational resources. CFD has historically faced hurdles, such as size limitations and reliance on simplified geometries, limiting its utility in complex design scenarios. However, advancements in computing power, hybrid modelling approaches, and multiphysics approaches have expanded its applicability. The case studies presented here illustrate how CFD can be instrumental in troubleshooting operational challenges, refining design paradigms, and enhancing predictability in flow regime mapping. These examples demonstrate that CFD can play a crucial role in advancing geothermal design, contributing to the sector's growing success by enabling more accurate modelling and facilitating the development of innovative solutions for diverse system conditions.

ACKNOWLEDGEMENTS

The authors wish to thank Ngati Tuwharetoa Geothermal Assets (NTGA) and Contact Energy for their support for these studies and in publishing this paper.

REFERENCES

Bangma, P. (1961). *The Development and Performance of A Steam-Water Separator for use on Geothermal Bores.* United Nations Conference on New Sources of Energy.

Harrison, R. F. (1975). *Methods for the analysis of geothermal two-phase flow* (Doctoral dissertation, University of Auckland).

Jung, D. B. (1989). *An Overview of Geothermal Separators.* Geothermal Resources Council, Transactions, vol 13.

Lazalde-Crabtree, H., (1984). *Design approach of steam-water separators and steam dryers for geothermal applications* Geothermal Resource Council Bull., 11–20.

Mandhane, J.M., Gregory, G.A., Aziz, K., (1974). *A flow pattern map for gas–liquid flowing horizontal pipes* Int. J. Multiphase Flow vol 1, pp. 537–553.

McLellan L., Koorey K. (2021) *Steam Separator Selection for a Geothermal Power Station* NZGW Proceedings

Osato K., Sato M., Kasai K. (2016*) Coupled Analysis Using 3D-CFD and Chemical Model for Corrosion and Scaling of Two Phase Flow Wellbore/pipeline* NZGW Proceedings

Purnanto, M. H., Zarrouk, S. J., & Cater, J. E. (2013). *CFD modelling of two-phase flow inside geothermal steam-water separators*. IPENZ Transactions, 40, 1-10.

Rivera-Diaz, A., & Koorey, K. (2021). *Review of Steam-Water Separator Performance Testing.* Proceedings World Geothermal Congress 2020+1. Reykjavik.

Siwach S., Jermy M. and Sellier M. (2015) *Design Of Pentane Vaporizers – A Cfd Approach* NZGW Proceedings

Spedding, P.L. and Nguyen, V.T. (1980). *Regime Maps for Air-Water Two-Phase Flow* Int. J. Multiphase Flow, vol. 35, pp. 779–793.

Zarrouk, S. J., & Purnanto, M. H. (2014). *Geothermal steam-water separators: Design overview.* Geothermics 53, 236-254.