Proceedings World Geothermal Congress 2026
Calgary, Canada, June 8 – 11, 2026

Application of Two-Phase Computational Fluid Dynamics modelling (CFD) to Geothermal Surface Facilities

Kevin Koorey[1]* and Kava Crosson-Elturan[2],
MB Century[1]*, Matrix Applied Computing Ltd[2]
*kkoorey@mbcentury.com

Keywords: Two-phase flow. Computational Fluid Dynamics (CFD)

ABSTRACT

The application of two-phase Computational Fluid Dynamics (CFD) modelling to geothermal surface facilities design presents a significant advancement over empirical methods derived from limited experimental and field data and limited piping and plant configurations. This study explores the utilization of CFD to simulate the complex interactions between liquid and vapor phases within geothermal pipelines, separators, and other equipment. By modelling these two-phase flows, we can predict phase distribution, pressure drop, forces on pipework and other parameters under various operational conditions and plant configurations. Performance improvements can be identified and tested via simulation. Additionally, the study investigates the role of CFD in analysing incidents and dynamic system behaviours, providing insights into potential failure modes and safety improvements. Various modelling methods, including Volume of Fluid (VOF), Eulerian multiphase (EMP), Lagrangian multiphase (LMP) and fluid-structure interaction (FSI) approaches are discussed, along with their respective advantages and limitations. Case studies of applied CFD in geothermal applications are summarised. The results demonstrate that CFD modelling can provide valuable insights into the design and operation of geothermal surface facilities, leading to improved plant performance and safety. Open questions in facilities design where CFD could be beneficial are discussed.

1 INTRODUCTION

The use of geothermal energy often requires the two phase (steam and hot water) from production wells to be transported/piped and separated. Two-phase mixtures also occur with carried over water and scrubbing water in steam mains and mixing non-condensable gas (NCG) in reinjection lines. Flashing of hot water will cause two phase conditions in various locations. Analysing these flows is possible with Computational Fluid Dynamics (CFD). This gives a valuable look inside geothermal systems which are difficult to observe otherwise due to the extreme environments found within pipelines and equipment.

There are also applications where CFD has been applied to flow within the geothermal reservoirs, working fluid flow in binary power plants, cooling towers and drilling operations.

2 COMPUTATIONAL FLUID DYNAMICS (CFD)

CFD uses a numerical approach to approximate the Navier-Stokes equations, simulating the flow of fluids. Numerous physical processes relevant to fluid handling systems such as long pipelines, separators and flash tanks can be modelled.

Given a complete set of operational flow parameters (e.g. temperature, brine volume fraction and mass flow), and the geometry of the internal fluid volume of the equipment, a CFD analysis can be undertaken. Generally speaking, the geometry must be simplified to avoid details which would both slow the computation and simultaneously contribute little to the results of interest. Once simplified geometry is available, it can be imported into a CFD program such as Star-CCM+ and meshed. Meshing is the process of dividing the geometry up into small cells, within and across which energy and momentum are conserved. Because of the simplicity of these cells, the conservation equations (also known as the Navier-Stokes equations) can be solved locally in an iterative process, which when the results stop changing, can be considered "converged" for the system as a whole. Thus, from simple conservation laws applied to small, simple cells, complex flow patterns can emerge.

The simplest example of CFD is inviscid flow, in which the flow field is typically free of any vortices. The next level up in complexity is viscous laminar flow. This allows the inclusion of stationary boundaries ("walls") which is

obviously necessary for modelling any fluid system of engineering interest. Flow features which are smaller than the cell size cannot be captured. Phenomena such as turbulence, which, in reality, include microscale vortices, can instead be included using additional mathematical treatments such as Reynolds-averaged Navier-Stokes (RANS) approaches, in which energy (and hence momentum) is dissipated in regions where turbulence is predicted to occur. Similarly, other phenomena such as multiphase flow (steam + brine mixtures) can be included as required. By including the relevant physics models in the simulation, complex flows such as the turbulent multiphase flow found in geothermal systems can be modelled.

The study of two-phase systems has been limited due to the added complications of the two-phase free surface interface, where mass (and correspondingly energy) transfer can cause complex flow behaviour. Geothermal applications involve steam and water often with the complexity of phase change, (boiling and condensation) and flow regime changes. While the latter can be captured by practical CFD models even at the whole-line scale, modelling phase changes is much less numerically stable and hence more expensive to model. Similarly, small droplet transport can be crudely interpreted using coarse grid VOF methods, but more advanced methods such as Lagrangian particles can be used to track breakup and droplet transport more accurately.

Despite current limitations, it remains far more cost-effective to simulate lines using CFD before they are built (than to simply build and test) in order to identify unforeseen flow regime risks such as slugging and large wave interference.

3 GENERAL MODELLING

When modelling a fluid system with CFD, a few points should be observed.

- The quantities of interest must be converged (i.e. stop changing with successive numerical iterations), but other quantities may still be varying without materially affecting the results
- Accepting results without performing sensitivity studies just because they are giving results with favourable outcomes to the engineering questions posed is called "confirmation bias" and should be avoided. Rigorous quality assurance is required, especially when undertaking simulations on novel systems, or involving complex physical processes.
- Boundary conditions are used to define how the fluid enters and interacts with the system's boundaries. The definition of the flow is typically particularly non-physical at inlets, so the inlets (and outlets) should be placed far enough away from the region of interest so as not to influence those results.
- The flow upstream of the region of interest should be fully developed. This point is really an extension of the prior point, because if the inlet boundary conditions are far enough upstream of the region of interest, the flow will naturally be more fully developed and less sensitive to those conditions.
- Simulation should usually only be undertaken after engineering questions about the system have been well posed. This is because the choice of physical models to be included will depend on what is to be ascertained by the model results. For example, The size of carryover droplets from separators in important rather than just the total carryover flow. (Rivera-Diaz & Koorey, 2021).

4 GEOTHERMAL SYSTEMS APPLICATIONS

4.1 Two phase pipeline flow with moderate velocities and separation of steam and water phases.

Two phase production lines connect wellheads in the steam field with downstream steam processing equipment, with the primary aim being to provide stable high-quality steam to steam turbines. These lines can be many kilometers long, traversing uneven terrain, which can lead to risky flow regimes. For example, uphill sections can provide the opportunity for the flow to back up and slug. Long straight sections of pipe can lead to wave formation. These waves are unstable in the sense that the taller they get, the faster they move. The faster they move, the more likely they are to catch up with other smaller, slower waves, snowballing into a wave so large that mor than half of the pipe height may be filled. Such a wave will travel at nearly the superficial steam velocity, and can cause damage when they reach bends. It is often presumed that such damage is caused by slugging, even in the absence of any other evidence to indicate that a full-bore slug has formed. This is possibly because of the nature of these production lines - we cannot see what is going on inside of them. CFD provides a glimpse into these systems and can predict the flow regimes commonly assumed to exist in pipe networks, such as stratified, ripple, wave, slug flow and annular regimes (Figure 1)

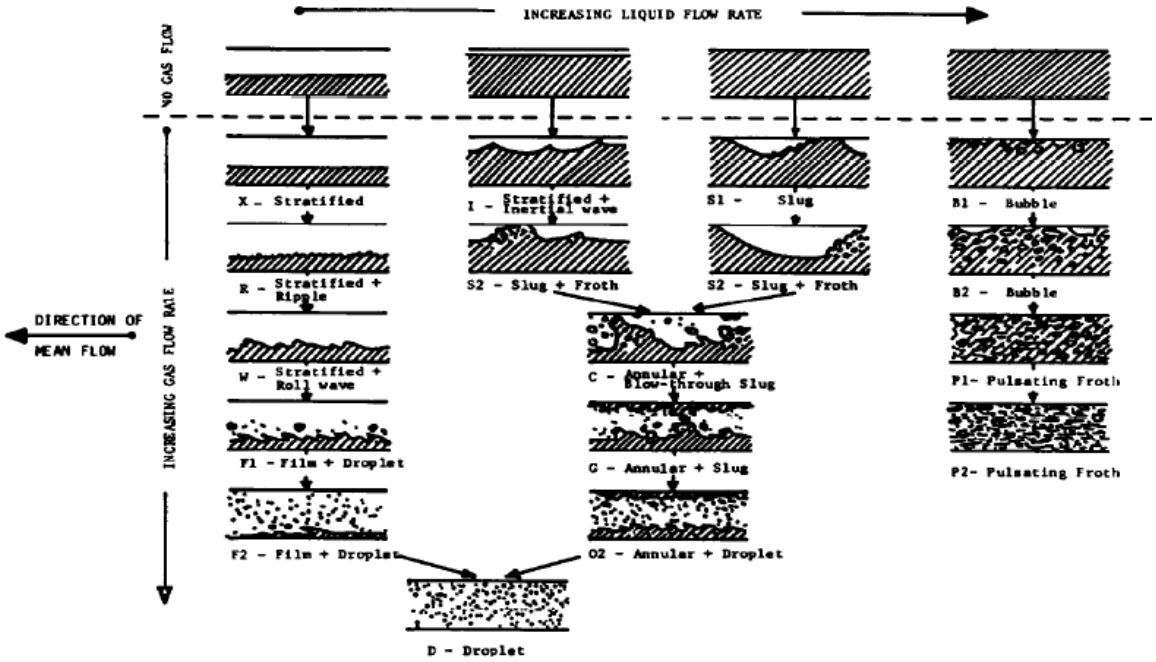

Figure 1 Two-phase flow regimes (Spedding & Nguyen, 1980)

Over longer distances, pressure changes of greater than ~1 bar mean that the flow conditions change significantly enough to violate the assumption of constant flow properties. As pressure drops through a pipeline, the following changes occur:

- Brine flashes into steam, increasing the volume flow rate of steam and hence the flow velocity
- The density of steam decreases, with the same effect as the preceding point
- The viscosity of the steam decreases, which somewhat mitigates the increase in viscous losses due to the accelerating steam flow

At pressure drops of less than 1 bar, the steam volume flow rate and viscosity vary by less than 10%. Generally, a properly sized two-phase production line will have pressure losses of about 1-2 bar per kilometer, meaning that a 1 km section can be modelled under a constant-properties assumption, which greatly simplifies the simulation. Over longer distances or segments with higher pressure drops, the flow properties change too significantly for such an assumption to hold. Two solutions to this problem are: 1) to break up the simulated domain into < 1 km sections or 2) to define a fluid model with properties which vary with pressure. However, in the second case, flashing of the brine into steam is still not considered. Although it can be in theory, it has been found to be impractical to do so on the kilometer scale typical of these models.

It is assumed that the flow in pipelines is generally dominated by coherent, strongly coalesced brine rather than misting flows. However, the presence of entrained mist in the steam flow would in theory change the steam flow properties by the rule of mixtures in such a way as to increase the viscosity and density of the mixed flow. These effects can be modelled using the VOF method without free surface sharpening, but have not yet been validated.

Distinct steam voids in the mostly water filled pipe can be modeled but modeling void collapse and Condensation Induced Water Hammer (CIWH) will require inclusion of the thermal properties of steam and water among other difficulties (Xiong, et al., 2024)

4.1.1 Applications

Some examples of where CFD can be applied to two-phase line flow include:

- Study of the transient (surge/slug) in pipelines with slope and elevation changes. As previously mentioned, the risk of slug formation in topographical depressions can be assessed using CFD.
- Flow in the vertical pipe risers upstream of separator inlets, and how these affect separator performance and loads on inlet nozzles. Careful design of riser geometry has been found to reduce separator forces induced by wave action by up to 80%.

- Dynamics of flow around bends and obstacles. Local velocity and impacts that may affect corrosion, erosion and scaling. The transient force on elbows can be simulated and used to design more robust foundations and carriages. This can be expanded to even include the flexible pipe / foundation interaction using fluid-structure interaction approaches available in CFD packages such as Star-CCM+.

4.2 Carryover flow and scrubbing water in steam lines

In contrast to the flows common to production pipelines, droplet heavy flows can be found in other parts of a typical geothermal system. In such situations, the Eularian Multiphase model (EMP) can be useful, as it provides a method for bi-directional dispersion between two distinct phases, including bubble and droplet size distribution and coalescence models. This model can also be coupled with the Lagrangian Multiphase model (LMP) if discrete particle flow is significant to the engineering question being posed, e.g. impingement erosion effects. A third model, the Fluid Film model, can be coupled to those two, providing more accurate on-wall flow estimations with bidirectional mass exchange between all three models (impinging deposition and atomization via wave / edge stripping).

Transport equations in such contexts require droplets in the gas and liquid transport via film. Exchange of liquid between the two occurs by atomization of the film and by droplet deposition. (Morud, 2005). Surface tension is an important parameter (Verdin, Thompson, & Brown, 2014).

To capture such phenomena, a fine mesh is typically required at the pipe wall. Custom CFD modelling has been developed by spray nozzle vendors using test data (Spraying Systems Co.) Star-CCM+ is also capable of modelling sprays and can also track their downstream deposition rates to monitor the effectiveness of e.g. automated cleaning systems.

4.2.1 Applications

- Improvements to condensate drain pots (Jung, 1995) and carryover removal from steam lines.
- Local velocity profiles that may affect corrosion, erosion and scaling.
- Optimization of scrubbing water spray design.
- Mist eliminators and scrubber vessels. (Grabo, 2016)
- Novel steam scrubbing devices.

4.3 Two-phase separators

Two-phase separators are designed to separate water and steam to provide stable high-quality steam to turbines. The most common design involves cyclonic action, which has the advantage of being able to separate even small brine droplets out of the steam flow (Lazalde-Crabetree, 1984), (Zarrouk & Purnanto, 2014). A spiral entry into a cylindrical chamber provides a simple arrangement which spins the mixture fast enough to separate the higher density water to the outside of the cylinder. It is effectively like applying increased gravity to the flow in the radial direction - heavier particles "sink" more rapidly. Thus, in theory, the faster the flow the better. However, in practice, higher inlet speeds generally lead to separation beak down when the water film on the separator wall reaches the top of the vessel (Foong, 2005).

Detailed modelling of separation efficiency would likely require consideration of the transition from two-phase flow with moderate velocities to liquid droplets and thin liquid films. This includes the need to model both bulk separation and the formation and transport of fine droplets.

Separator efficiency models have been limited to modelling the steam flow with particle tracing. (Pointon, Mills, Seil, & Zhang, 2009), (Purnanto, Zarrouk, & Cater, 2013)

Accurate complex modelling would require calibration with measured performance data.

4.3.1 Applications

The application of CFD to separator design is general but could include e.g. incremental improvements, novel separator arrangements, the effects of defects and internal details, and optimised sizing, and inlet pipework arrangements.

The effects of changes to the upstream flow regime can be studied. This can include separation performance and physical loading on the vessel from unstable flow.

4.4 Water flashing and Atomization

Discharging saturated separated hot water to a lower pressure to generate steam or discharge to waste is common in geothermal plants. When flashing occurs upstream of separators generated droplet size can be important (Misa, Harwood, & Morris, 2020). CFD would generally not be a good fit for this due to the extreme nature of the thermodynamics at play, which tend to be correspondingly unstable numerically. This can make solving such problems impractical.

Away from the discharge point, the two-phase fluid will coalesce and form a two-phase mixture the CFD can model. Velocities after flash valves are much higher than the two-phase production lines.

4.4.1 Applications

Applications of CFD to flashing systems include flows to discharged silencer manifolds, and investigations involving the transient performance of downstream systems subjected to e.g. opening of the water dump valves.

4.5 Gas mixing (CO2 injection)

Gas mixing systems are being explored to alleviate the need for acid injection and additionally to provide a potential carbon sink, mitigating anthropogenic climate change. The goal of such systems is to keep bubbles small (high total surface area) and prevent the injected gas from coalescing. High local turbulence and mixing can promote these goals. Although gas dissolution into the brine is the goal, and such models are available in Star-CCM+, that process is complex, hard to model accurately, and is furthermore extremely difficult to validate. However, simply ensuring a fine enough distribution of bubble sizes for a sufficient duration of brine contact is more easily achievable.

5 CASE STUDIES

5.1 Long Pipeline with Changes in Slope

When a pipeline goes uphill, the flow dynamics can change such that slugging becomes more likely for a given flow rate and water volume fraction. In this study, an uphill section of line's flow regime was investigated. Prior to the uphill section, a long straightaway allows travelling waves to form.

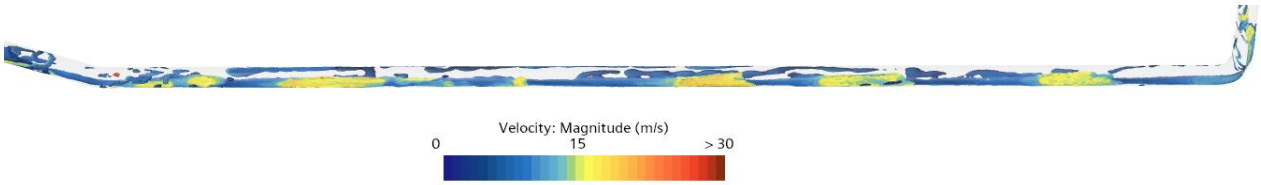

Figure 2 Wave formation in long horizontal section

Once the waves hit the elbow entering the uphill section, a helix forms, followed by chaotic mixing and partial re-suspension of the brine in the faster-moving steam. This is visible as orange blobs in the image below.

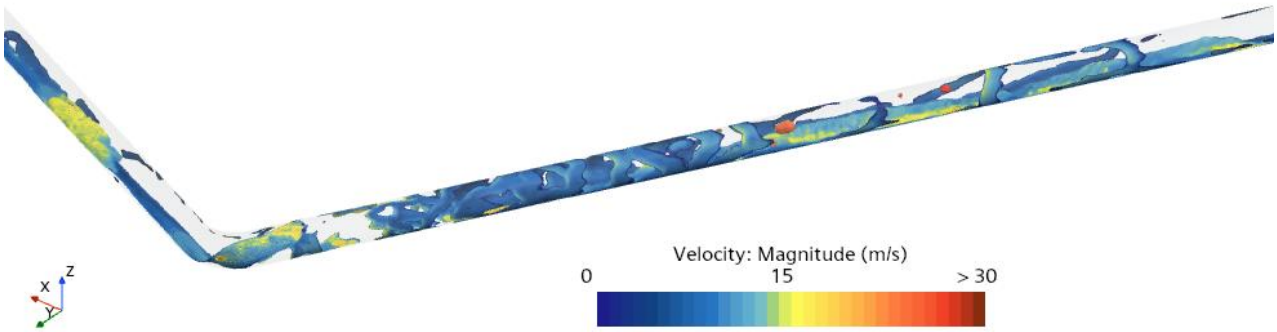

Figure 3 Waves transitioning to chaotic helical mixing and re-entrainment in uphill section

Wave flow re-establishes in the uphill section after the disruption caused by the elbow settles. However, the lateral momentum of the helical ramping at the elbow is retained in the waves, as they were observed to "snake"

up the uphill section. This corresponded to observed “beach marks” in the pipe. The helical ramp formed by the elbow was also visible on the passivation layer within the pipe.

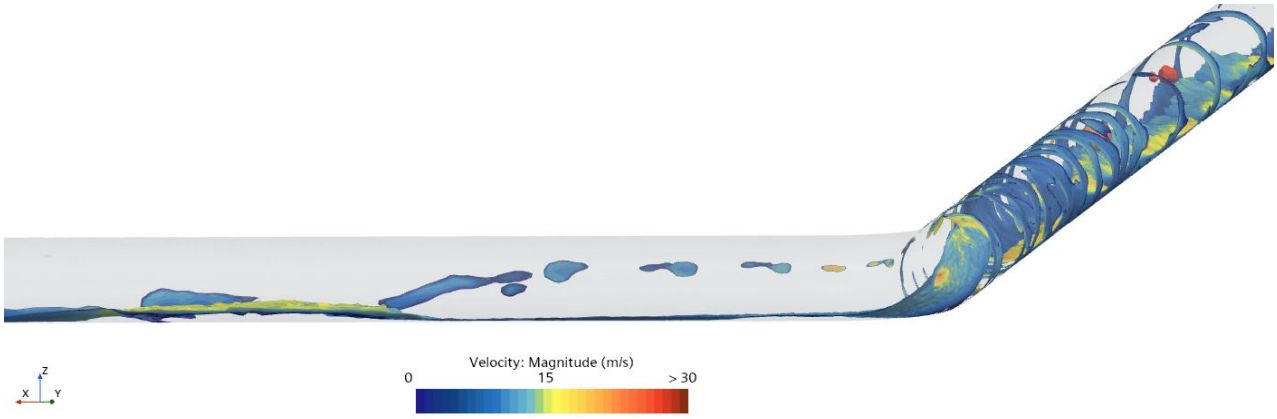

Figure 4 Alternate view of uphill section

5.2 Compact Slug Diffusor

Long straight sections of pipeline can be prone to formation of high-speed waves. These waves can cascade into one another, gathering momentum as they travel down the pipe. Smaller waves are slower, owing to the larger steam cross-section above them, while larger waves are conversely faster. Thus, larger waves tend to catch up to smaller ones, and when they do the two can combine into an even larger wave. In this study, a long pipeline was subjected to constant wellhead conditions. The resulting wave formation was observed, while recording total force on each elbow (plus 1.5D up and downstream) in the model.

By tracking the forces on each elbow, insight was gained into the nature of the forcing on the pipeline. In this case, the wave action was very unsteady, with a larger set of waves hitting the elbow roughly every 8 seconds. Sometimes a very large hit was recorded, which corresponded visually to two or more waves combining just before hitting the elbow. Despite the small time period practicably modelled, it is expected that longer simulations could chaotically yield monster waves/slugs such as those theorized to cause pipeline damage.

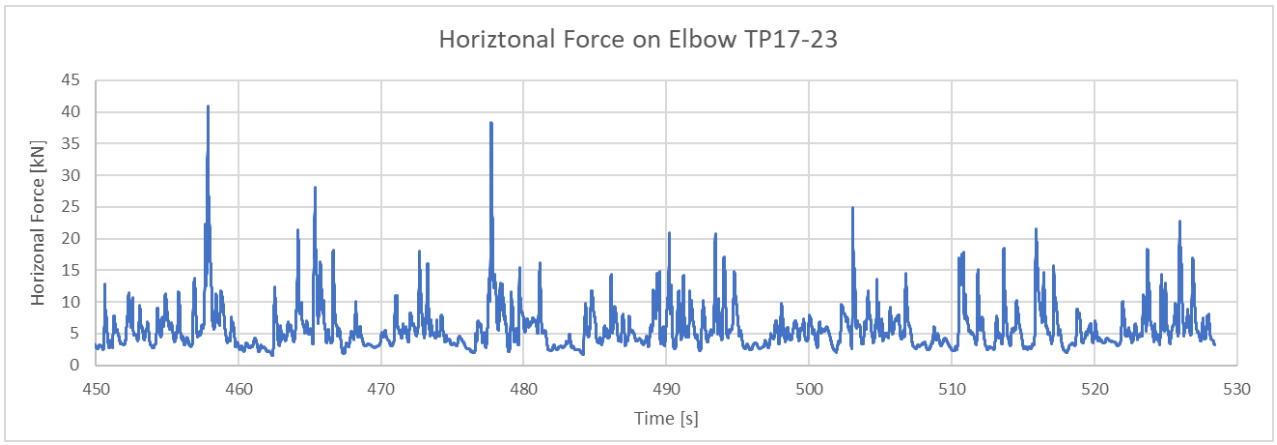

Figure 5 Force time history on the worst elbow

This wave combination effect can be imagined to be worse the longer a straight section of pipeline is. That is, there is more time for smaller waves to be caught up in larger ones, accelerate, and so reinforce the wave accumulation trend. When a large wave hits an elbow, it ramps up and around the circumference of the pipe in a helical fashion. Due to this flattening and spreading out of the wave, it effectively has more surface area across which the steam can shear, which transfers kinetic energy from the steam into the brine. Thus, with each elbow in an expansion loop, the brine gathers more momentum, with the final elbow in the loop generally being hit the hardest.

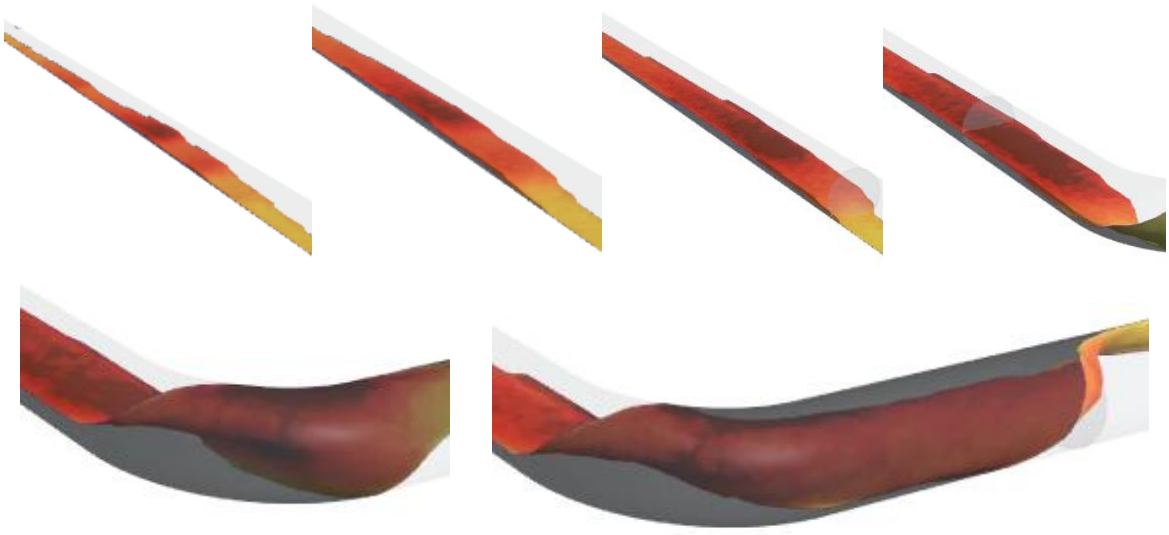

Figure 6 Image sequence showing merging of waves just before hitting an elbow

In order to mitigate the effect of waves coming off of long straights and hitting elbows, a few compact in-line diffusor designs were modelled. Traditionally (in the oil & gas industry) slug diffusors are large arrays of parallel pipes. These designs take up a lot of space, and could be accurately described as fields of pipework. In this instance, space was very limited, so a more compact design was sought.

A full-bore slug was introduced upstream of the various designs. Once the slug passes into and through the diffusors, it also has to then re-accelerate into the smaller nominal piping diameter. It was this point that ultimately led to the failure of any of these designs to reduce downstream elbow loading, as the slug essentially just reformed at the other end of the diffusor. It may be that a longer diffusor section could be more effective, but that was not explored in this study.

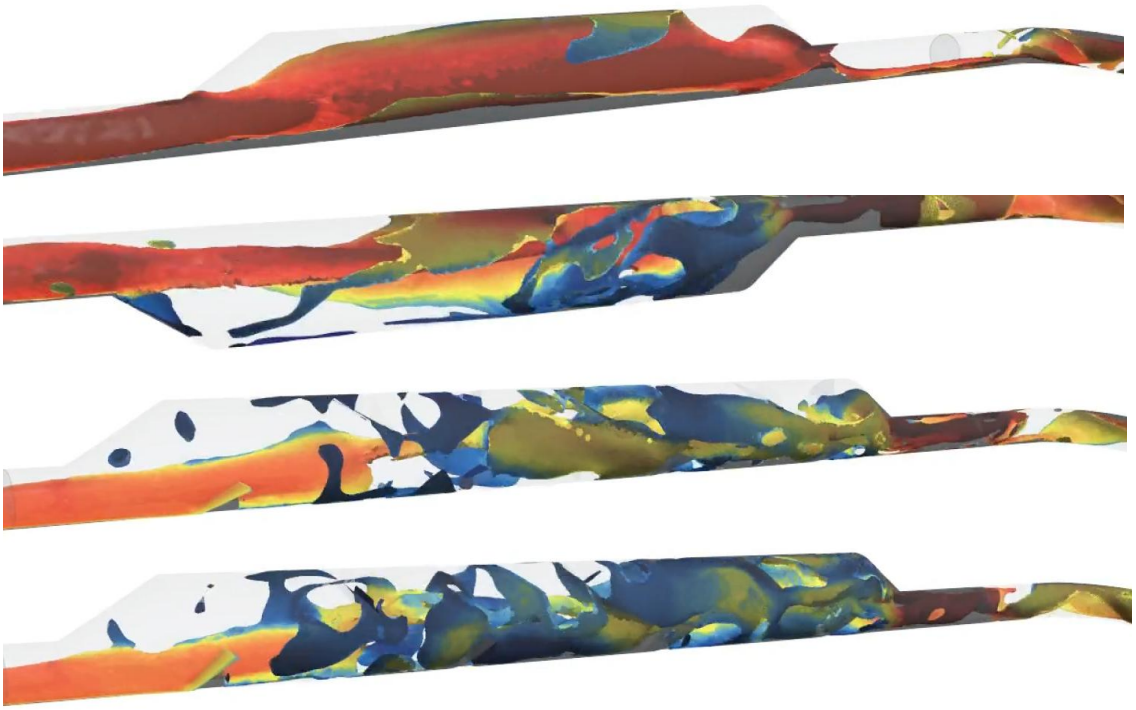

Figure 7 Diffusor flow regime passing a slug (darker shading indicates underwater)

Downstream elbow loading was recorded for each design. Although the peak slug passing force on the first and second elbows was generally reduced, the force on the subsequent elbows was increased significantly, relative to having no diffusor at all. It is theorised that this is because passing the elbows re-energised the slug via higher steam shearing exposure as previously described.

5.3 CO2 Injection and Mixing

Reinjection of CO2 into geothermal flows is an active field of research. Motivators include beneficial acidification and climate change offsets. In this study, a stream of CO2 is injected into a geothermal pipeline upstream of an arrangement of mixing baffles. The baffle-induced turbulence helps break down the stream of CO2 into smaller bubble sizes, which in turn aids dissolution. The numerical approach used is the Eularian Multiphase model (EMP) with bubble physics including breakup and coalescence considered. In this approach, the individual bubbles are not explicitly modelled, rather, the size distribution is tracked and altered by the varying degrees of local turbulence throughout the domain. Final dissolution was not included in this simulation but is an available physical model.

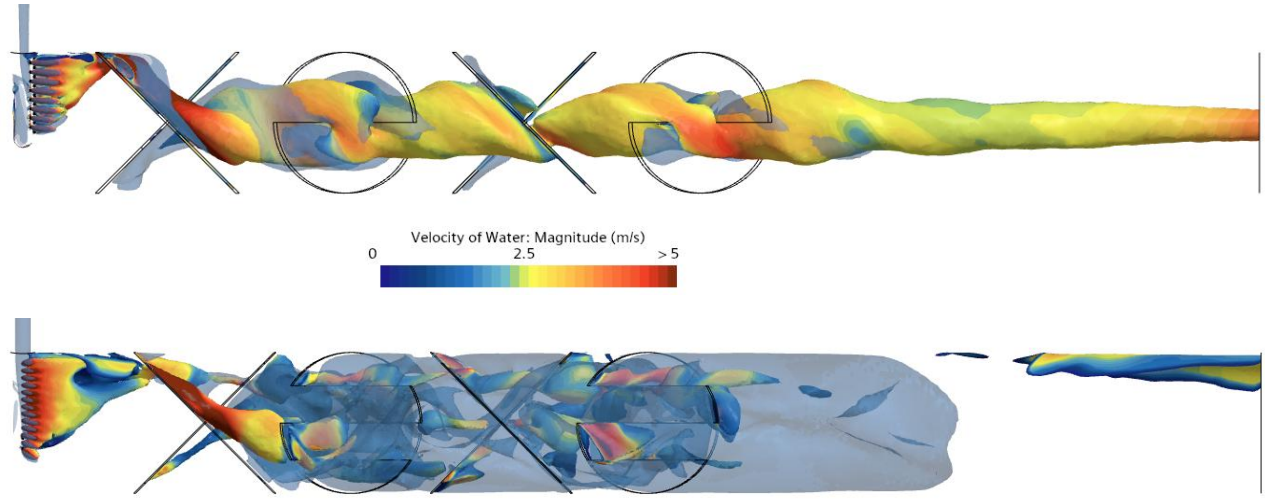

Figure 8 Isosurface of CO2 concentration coloured by velocity and turbulent energy (transparent blue)

5.4 TOPP2 Line Dynamics

A study investigated a proposed 1.6 km long geothermal line (Marsh, Crosson-Elturan, Spinks, & Lee, 2025).The study found that the line wasn't expected to produce slug/surge flow, primarily exhibiting stratified and calm wave flow regimes. A short riser at the separator produced unstable wavy-annular flow.



Figure 9 Velocity of the brine free surface in the final leg of the NTGA line from (Marsh, Crosson-Elturan, Spinks, & Lee, 2025)

An introduced 40 m full-bore slug was found to dissipate in the system rather than carry on, indicating that the new line would be likely to have a calming effect on introduced upsets.

6 CONCLUSION

CFD modeling of the geothermal two-phase pipeline flows is a powerful tool which can be used to validate new designs and troubleshoot issues in operational plant. There is potential to use CFD for complex modelling of steam-water separation, flashing and recondensing flows, and other geothermal plant equipment.

7 ACKNOWLEDGEMENTS

The authors wish to thank our clients who allowed sharing of the modelling examples and the support of our employers for this paper.

8 REFERENCES

Foong, K. (2005). Design Concept for a More Efficient Separator. *World Geothermal Congress*. Antalya.

Freestone, D. H., Lee, K. C., & Hole, H. (1983). *Utlisation of Geothermal Energy : Two Phase Flow Studies*. Auckland: New Zealand Energy Research & Development Commttee.

Grabo, M. A. (2016). *Numerical Modeling of Flow in a Mist Eliminator For Geothermal Power Plants*. Faculty of Industrial Engineering, Mechanical Engineering and Computer Science University of Iceland.

Hu, X., Qian, S., Yamauchi, K., & Okochi, H. (n.d.). CFD Prediction of Gas-Liquid Separation Efficiency of Geothermal Steam-Water Cyclone Separator Using a Verified Turbulence Model. *ASME 2017 Fluids Engineering Division Summer Meeting*. Waikoloa.

Jung, D. B. (1995). Drip-Pot Applications. *Geothermal Resources Council 1995 Annual Meeting*. Reno.

Lazalde-Crabetree, H. (1984, September). Design Approach of Steam-Water Separators and Steam Dryers for Geothermal Applications. *Geothermal Recources Council Bulletin*.

Marsh, A., Crosson-Elturan, K., Spinks, K., & Lee, M. (2025). The use of Computational Fluid Dynamics (CFD) for slugging prediction in a two-phase geothermal production system. *47th New Zealand Geothermal Workshop*. Rotorua.

Misa, T., Harwood, K., & Morris, C. (2020). Investigation and Application of Flashing Flow Nozzles to Mitigate Geothermal Turbine Life Reduction at Te Mihi Power Station. *Proceedings 42nd New Zealand Geothermal Workshop*. Waitangi.

Morud, J. (2005). Dilute Gas-Liquid Flows with Liquid Films on Walls. *Fourth International Conference on CFD in the Oil and Gas, Metallurgical & Process Industries* . Trondheim.

Nguyen, V.-T., & Sagar, H. (2024). Computational Fluid Dynamics Modeling and Experiments of Two-Phase Flows. *fluids*.

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. *New Zealand Geothermal Workshop*. Auckland.

Pointon, A. R., Mills, T. D., Seil, G. J., & Zhang, Q. (2009). Computational Fluid Dynamic Techniques for Validating Geothermal Separator Sizing. *GRC Transactions, Vol. 33*, 943 - 948.

Purnanto, M. H., Cater, J. E., & Zarrouk, S. J. (2012). CFD Modelling of Two Phase Flow inside Geothermal Steam-Water Separator. *New Zealand Geothermal Workshop*. Auckland.

Purnanto, M. H., Zarrouk, S. J., & Cater, J. E. (2013). CFD Modelling of Two-Phase Flow Inside Geothermal Steam-Water Separators. *IPENZ Transactions Vol 40*.

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

Spedding, P. L., & Nguyen, V. (1980). Regime Maps for Air Water Two Phase Flow. *Chemical Egineering Science Vol 35*, pp. 779-793.

Spraying Systems Co. (n.d.). *Optimizing Spray Performance with Computational Fluid Dynamics*. www.spray.com.

Verdin, P. G., Thompson, C. P., & Brown, L. D. (2014). CFD modelling of stratified/atomization gas–liquid flow in large diameter pipes. *International Journal of Multiphase Flow Volume 67*, 135-143.

Wikipeda. (2025). *Computational fluid dynamics*. Retrieved from https://en.wikipedia.org/wiki/Computational_fluid_dynamics

Witlox, H. W., Haper, M., Oke, A., Bowen, P. J., Kay, P., Jamois, D., & Proust, C. (2009). Two-Phase Jet Releases and Droplet Dispersion: Scaled and Large-Scale Experiments, Droplet-Size Correlation Development and Model Validation. *IChemE SYMPOSIUM SERIES NO. 155*.

Xiong, F., Zhao, J., Tu, R., Chen, X., Zhang, Z., Guo, W., . . . Zhou, Y. (2024). Improved CFD simulation of water hammer caused by condensation with an enhanced phase change model. *Progress in Nuclear Energy Vol 173*.

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

Zhao, H. D., Lee, K. C., & Freeston, D. H. (2000). Geothermal Two-Phase Flow in Horizontal Pipes. *Proceedings of the World Geothermal Congress* (pp. 3349-3353). Kyushu: International Geothermal Association.