



energies



Article

Field-Validated Two-Phase CFD Modelling of a Utility-Scale Geothermal Steam Ejector: Operational Boundaries and Performance Scaling

Ximena Guardia Muguruza, Christine Groves, Yonatan Afework Tesfahunegn, Gudrun Arnbjorg Saevarsdottir and Maria Sigridur Gudjonsdottir

Special Issue

Advanced Geothermal Energy Production and Utilization

Edited by
Dr. Hadi Jabbari



<https://doi.org/10.3390/en19184478>

Article

Field-Validated Two-Phase CFD Modelling of a Utility-Scale Geothermal Steam Ejector: Operational Boundaries and Performance Scaling

Ximena Guardia Muguruza ^{*}, Christine Groves , Yonatan Afework Tesfahunegn ,
Gudrun Arnbjorg Saevarsdottir  and Maria Sigridur Gudjonsdottir 

Department of Engineering, Reykjavik University, Menntavegur 1, 102 Reykjavik, Iceland; yonatant@ru.is (Y.A.T.); gudrunsa@ru.is (G.A.S.); msg@ru.is (M.S.G.)

^{*} Correspondence: ximena21@ru.is

Abstract

Supersonic ejectors offer a promising solution for extending low-pressure well life and increasing total power output in geothermal plants by entraining low-pressure fluid using a high-pressure primary flow. While steam supersonic ejectors are widely used in industrial applications such as refrigeration, their deployment in geothermal power generation remains largely unexplored, leaving a critical gap in field-validated numerical models for utility-scale two-phase systems. To address this, this study presents a 3D Computational Fluid Dynamics (CFD) framework validated against industrial-scale field tests conducted at the Theistareykir Geothermal Power Plant in Iceland (connecting wells ThG-11 and ThG-15). Four RANS turbulence models (Standard $k - \epsilon$, RNG $k - \epsilon$, Realizable $k - \epsilon$, and $k - \omega$) were evaluated in ANSYS Fluent using a homogeneous Eulerian wet-steam formulation. The Realizable $k - \epsilon$ model demonstrated superior accuracy, achieving the lowest absolute error (6.4%) against field data. While non-equilibrium thermodynamic relaxation caused a systematic 8.9–12.6% overprediction in primary motive flow, secondary entrainment predictions closely tracked physical performance, with entrainment ratio errors reaching 0.00% under stable operation. Crucially, the model identifies operational boundaries: while field data places the physical backflow limit at an inlet pressure ratio of 2.2 (9.9 bar difference), numerical divergence near zero-entrainment establishes a conservative modelling threshold at 2.6. By defining these physical and numerical limits while quantifying nozzle-sizing safety margins, this work provides a verified benchmark for scaling up CFD models for full-capacity geothermal ejector networks.

Keywords: supersonic ejector; numerical models; Computational Fluid Dynamics; experimental validation; geothermal applications; two-phase flow



Academic Editor: Marcin Sosnowski

Received: 2 September 2026

Revised: 14 September 2026

Accepted: 17 September 2026

Published: 21 September 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article

distributed under the terms and

conditions of the [Creative Commons](https://creativecommons.org/licenses/by/4.0/)

[Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

1. Introduction

An ejector is a static device that combines a high-pressure primary fluid with a low-pressure secondary fluid, resulting in a mixed flow at an intermediate outlet pressure [1]. Ejectors with converging–diverging nozzles can achieve sonic conditions in the nozzle’s throat and supersonic conditions at the nozzle’s outlet due to the expansion of the flow in the diverging section [2].

In a geothermal power plant, ejectors can utilise the high pressure of a geothermal well to induce flow from a low-pressure geothermal well and elevate its pressure to meet the

plant's operating requirements [3]. This not only extends the functional life of low-pressure wells but also enhances the facility's overall power output.

The analysis of two-phase flow and non-equilibrium condensing phenomena in supersonic steam ejectors is a well-established topic in fluid mechanics, supported by extensive Computational Fluid Dynamics (CFD) documentation across various industrial sectors [4,5]. Numerical and experimental research has thoroughly investigated supersonic ejectors for applications in vapour compression and expansion work-recovery refrigeration [4,6,7], thermal power plants and steam turbine condenser air-extraction systems [5,8], multi-effect distillation with thermal vapor compression (MED-TVC) seawater desalination [5,6,9], fuel cells [6,10], chemical processing [6,11], and aeronautical bleed-air management [11,12]. While low-temperature geothermal heat is occasionally cited alongside solar and industrial waste heat as a potential driving heat source to activate ejector refrigeration or combined power/cooling cycles [11,13,14], the direct CFD modelling and fluid dynamics analysis of two-phase supersonic steam ejectors applied specifically to high-enthalpy geothermal power station equipment remain largely unexplored. This represents a highly targeted application, critical for geothermally active regions such as Iceland and applicable to similar high-enthalpy reservoirs worldwide, where optimising low-pressure resource recovery and secondary steam entrainment is essential.

Computational Fluid Dynamics (CFD) serves as a primary tool for evaluating complex ejector internal flows, which are characterised by interacting compressible gas dynamics, shock-train structures, and phase-change behaviour [11,15]. While two-dimensional (2D) axisymmetric models are widely used for circular cross-sections to balance numerical accuracy and computational economy [4,12,16,17], full three-dimensional (3D) domain formulations are essential for geometries with non-axisymmetric features, complex boundary interfaces, or off-design conditions [8,11,14,16,18–20]. Among commercial platforms, ANSYS Fluent is predominantly employed for ejector simulations [4–6,8,10,12,14–29], alongside platforms such as ANSYS CFX, PHOENICS, and OpenFOAM [11,18].

Numerical formulations generally rely on either density-based or pressure-based solvers [4,11,14,15,17,18,24,30]. Density-based algorithms solve mass, momentum, and energy conservation equations simultaneously as a fully coupled system, deriving static pressure via the equation of state [31,32]. This simultaneous coupling excels in capturing strong localised gradients, expansion fans, and shock-wave structures while minimising numerical oscillations [31,33]. Conversely, pressure-based solvers use pressure–velocity coupling to enforce mass conservation sequentially [34,35]. Modern pressure-based coupled solvers have evolved to reliably handle compressible flow regimes, offering enhanced convergence stability at lower operating pressures, though they require careful validation when capturing severe adverse pressure gradients or shock-boundary-layer interactions [36,37].

Regarding working fluids, water vapour and steam represent the most frequently analysed media, moving from ideal gas approximations toward specialised wet-steam formulations that account for non-equilibrium condensation [5,7,10,11,14–16,18–20,22,24,25,28,30,38,39]. Alternative media include synthetic refrigerants (e.g., R134a, R245fa, CO₂) [4,7,8,11,13,15,17,18,20,25–27], as well as air, nitrogen, and natural gas [11,12,18,20].

To model turbulence in variable-density flows, Reynolds-Averaged Navier–Stokes (RANS) formulations (often incorporating Favre averaging), are standard practice [33,35,40,41]. While no single turbulence closure model performs universally across all operating conditions [11,18,42], the Shear Stress Transport (SST) k – ω model is widely recognised for resolving shock locations and stream-mixing interfaces [14,18,19]. The Realizable k – ϵ model offers strong performance in predicting primary jet expansion angles and recirculation zones [9,11,14,16,18,29], whereas standard or RNG k – ϵ models may exhibit larger discrepancies under off-design shock conditions [11,18,27,43]. Higher-order models, such as the

Reynolds Stress Model (RSM), require greater computational effort and pose convergence challenges [11,14].

Despite this extensive background in general ejector CFD, the literature on Computational Fluid Dynamics modelling of ejectors for connecting high- and low-pressure geothermal wells remains scarce, limited primarily to a few preliminary studies [44–46]. Gutsol [44] proposed a simplified two-dimensional (2D) CFD model to evaluate potential well candidates for ejector applications, though without any physical or experimental validation. Andal [45] advanced this by developing a three-dimensional (3D) model in ANSYS CFX for a proposed supersonic steam ejector at the Theistareykir Geothermal Power Plant in Iceland; however, this work remained purely theoretical as no physical pilot system was built or tested for this design.

To bridge the gap between theoretical modelling and physical implementation, a preliminary, non-peer-reviewed master's thesis by Groves [46] explored the initial design parameters for a pilot-scale ejector tested at the Theistareykir geothermal power plant in November, 2024. While this work established early attempts at 3D modelling, the simulations were restricted to highly simplified, idealised geometries that deviated significantly from the eventual physical installation.

To move beyond idealised concepts and unvalidated designs, this study presents the first comprehensive, field-validated 3D two-phase CFD framework for an operational, utility-scale geothermal steam ejector, using ANSYS Fluent. Utilising the finalised, as-built geometry of the physical system tested at the Theistareykir Geothermal Power Plant in November 2024, we performed new CFD simulations and compared them against eight distinct physical field-test data points. By comparing localised numerical vectors directly with real-world thermal and pressure fluctuations, we analyse the complex shock-wave structures, phase distribution, and internal hydrodynamics of the two-phase flow. Crucially, this paper explicitly defines the thermodynamic operational limits, specifically, the critical pressure-ratio threshold at which suction loss occurs, and provides concrete primary-sizing safety factors necessary for commercial scale-up. This establishes a reliable, field-tested benchmark for confidently scaling and standardising ejector technology across global geothermal power networks.

In that sense, the subject of this research is a utility-scale, 3D two-phase steam ejector system designed to entrain low-pressure geothermal fluid (supplied by well ThG-15 at 8.1 bar_g with an 80% liquid mass fraction) using high-pressure primary steam (supplied by well ThG-11 at up to 30 bar_g) [47]. Unlike laboratory-scale or single-phase ejectors, this study models an operational industrial unit tested at the Theistareykir Geothermal Power Plant in Iceland [47]. The objective is to evaluate the predictive accuracy and operational boundaries of an Eulerian wet-steam CFD formulation under real-world, high-enthalpy geothermal field operating conditions.

The main innovation presented in this work lies in the parameter selection and calibration within the ANSYS Fluent CFD environment for complex two-phase geothermal steam flows. By evaluating RANS turbulence closures alongside a homogeneous Eulerian wet-steam model, this framework enables the accurate determination of the flow's operational boundaries and dynamic control ranges. Specifically, this paper defines the critical pressure-ratio threshold at which backflow mode occurs and provides concrete primary nozzle sizing safety margins necessary for commercial scale-up across global geothermal facilities.

2. Materials and Methods

2.1. Numerical Methodology

2.1.1. Governing Equations

Computational Fluid Dynamics (CFD) is a numerical technique used to solve partial differential equations that describe fluid flow and heat transfer processes [48]. It achieves this by dividing the fluid domain into many small, finite volumes. This approach allows for the computation and visualisation of local fluid properties, including velocity, pressure, temperature, and turbulence characteristics [18]. In the present study, the flow is considered steady, viscous, turbulent, and free of body forces. The compressible Reynolds-averaged Navier–Stokes (RANS) equations serve as the governing framework for the various turbulence models employed. The behaviour of the flow within the ejector is described by the conservation equations of mass (Equation (1)), momentum (Equation (2)), and energy conservation equations (Equation (3)) [14]. Equation (1) represents the steady-state conservation of mass for a compressible flow, where ρ is the fluid density and u_i is the velocity component in the i -th direction [14]. Under steady-state conditions, the local rate of change of density with respect to time is zero, reducing the general continuity equation to the divergence of the mass flux [2].

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

Equation (2) describes the conservation of momentum for a turbulent flow, where p denotes pressure, and μ the dynamic viscosity [14]. The term on the left-hand side represents the convective transport of mean momentum. The right-hand side captures changes in momentum resulting from pressure gradients, viscous stresses, and Reynolds stresses, thereby ensuring momentum conservation.

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j}(-\rho u'_i u'_j) \quad (2)$$

Equation (3) expresses the conservation of energy where c_p is the specific heat at constant pressure, T is temperature, λ is thermal conductivity, $u'_i T'$ is the turbulent heat flux, and $\mu\phi$ is the viscous dissipation function [14]. The left-hand side describes the convective transport of thermal energy, while the right-hand side accounts for heat conduction, turbulent heat transport, and viscous heating, thus ensuring energy conservation.

$$\frac{\partial}{\partial x_i}(\rho c_p u_i T) = \frac{\partial}{\partial x_j} \left(\lambda \frac{\partial T}{\partial x_j} - \rho c_p u'_i T' \right) + \mu\phi \quad (3)$$

To solve these equations, ANSYS FLUENT 2024 R1 [49] was used. The accuracy of the CFD solution depends on the computational setup for the proposed ejector settings in this paper, including the domain definition, mesh resolution, turbulence model, solver settings, discretisation schemes and boundary conditions. These aspects are discussed in detail in the following sections.

2.1.2. Domain

The internal geometry of the supersonic ejector assembly during the field tests executed in November 2024 at the Thestareykir Geothermal Power Plant, including the primary convergent–divergent nozzle, suction chamber, constant area mixing section (CAMS), and diffuser, is schematically illustrated in Figure 1. To respect IP and confidentiality constraints while providing clear structural context, absolute dimensions are omitted.

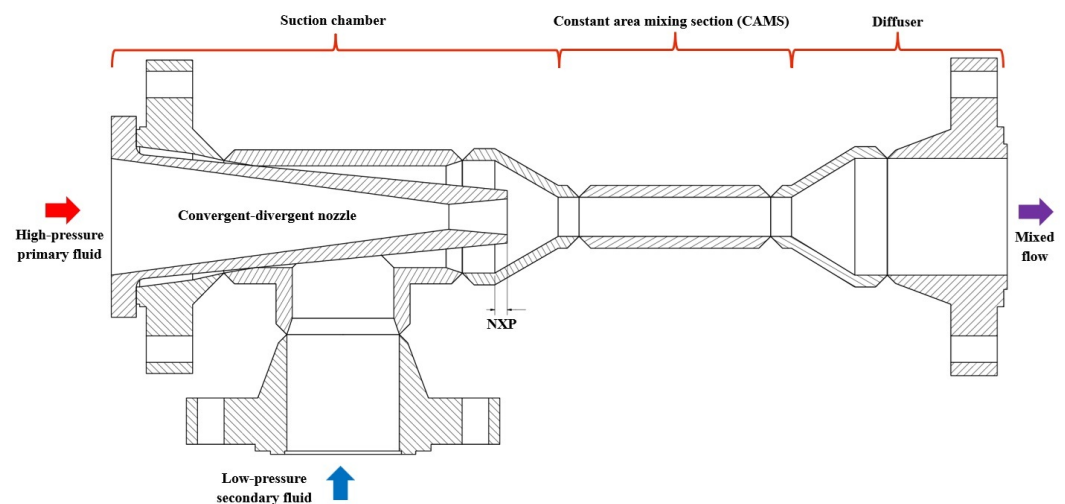


Figure 1. Cross-sectional schematic of the supersonic ejector assembly illustrating key geometric components, flow pathways, and the Nozzle Exit Position (NXP). Absolute dimensions are omitted due to proprietary restrictions.

The field-scale ejector's internal dimensions were used to develop a three-dimensional model in ANSYS FLUENT 2024 R1. The resulting 3D model is presented in Figure 2.

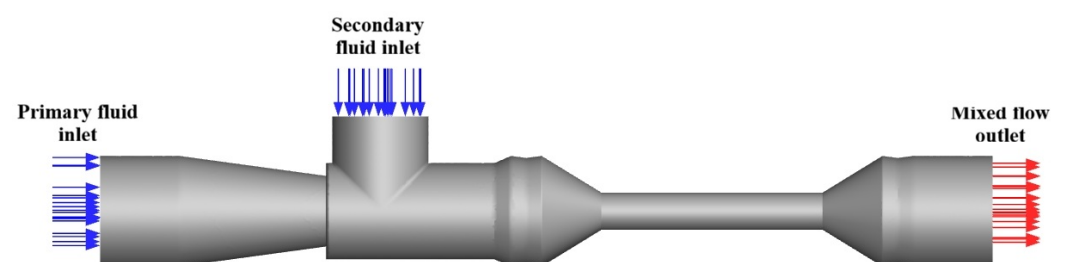


Figure 2. Three-dimensional model of the inside geometry of the ejector used for the two-phase simulations in ANSYS FLUENT 2024 R1.

2.1.3. Mesh Generation

CFD requires that the geometry is divided into a numerical grid, known as the mesh. Ansys Meshing was used to generate these meshes for this study. The geometry was discretised into three resolutions (coarse, medium, and fine) to perform a grid convergence study. Table 1 shows the element size and number of elements for each mesh resolution.

Table 1. Geometric statistics for the three mesh levels evaluated.

Parameters	Coarse	Medium	Fine
Element size [m]	0.01	0.005	0.0025
Number of elements	1,416,131	1,659,401	3,148,023
Min. Orthogonal Quality	0.0077	0.0096	0.0096
Max. Aspect Ratio	284	152	152

To accurately capture the high-velocity gradients and boundary-layer physics, inflation layers were applied to all wall boundaries, supplemented by localised face and edge sizing in the nozzle throat and mixing sections. Figure 3 shows the medium mesh generated for the model, where inflation layers at these locations can be visualised.

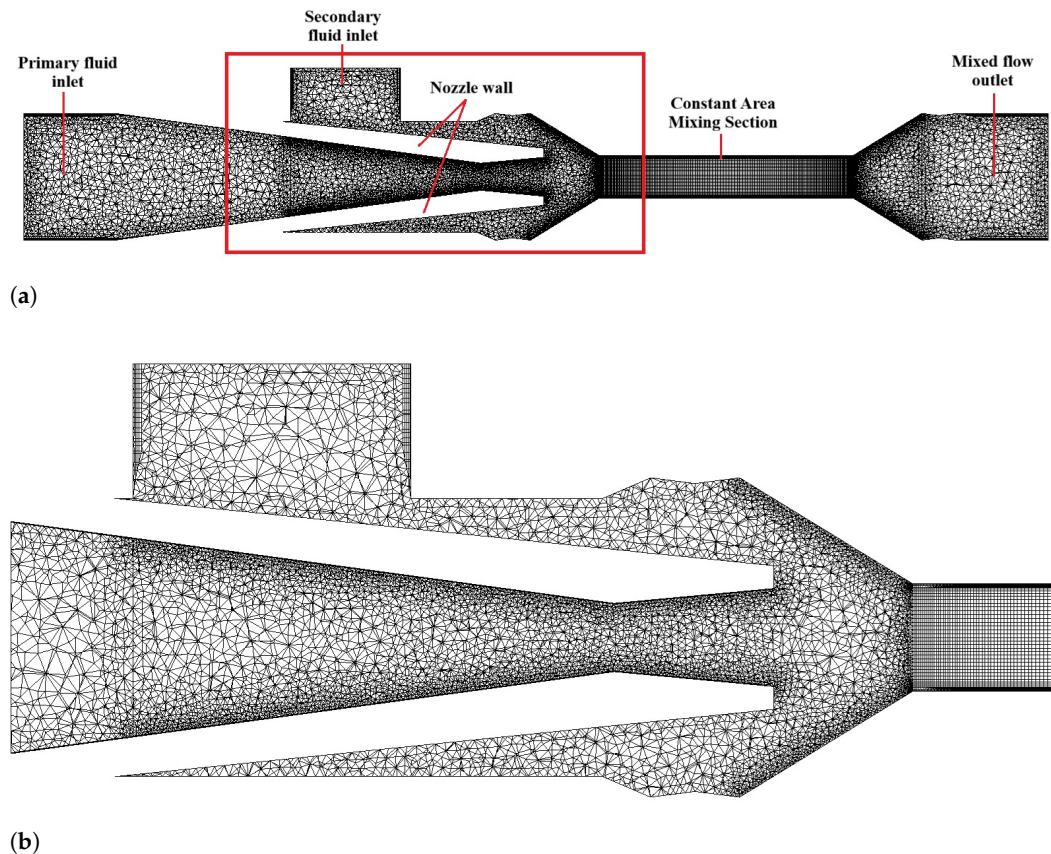


Figure 3. Section views of the medium mesh: (a) full-length longitudinal section showing the red-highlighted region of interest; (b) detailed view of the nozzle and mixing chamber geometry highlighted in red in (a).

The geometric quality of each mesh was assessed using Orthogonal Quality and Aspect Ratio metrics, with the summary statistics provided in Table 1. According to the Ansys Discovery User Manual (Version 2025 R2) [50], orthogonal quality ranges from 0 (worst) to 1 (best), with values approaching 1 indicating ideal orthogonality. For stable fluid simulations, a minimum orthogonal quality above 0.1 is generally recommended, while average values above 0.7 signify high-quality meshing [50].

Although the complex internal geometry of the ejector introduced localised minimum orthogonal quality values (0.0077 to 0.0096) within the outer corners of the mixing chamber and the non-critical periphery of the Constant-Area Mixing Section (CAMS), these elements constituted a negligible fraction of the overall grid. The vast majority of the computational domain maintained high orthogonality (>0.7), comfortably exceeding recommended average thresholds. Furthermore, these low-quality cells were strictly isolated away from high-gradient transport zones, including the primary nozzle core, the shear mixing layer, and the throat shock region. Grid independence evaluation confirmed consistent physical trends across mesh refinements, with the medium mesh providing robust numerical convergence across all tested turbulence models without encountering the numerical stiffness or iterative failures observed on the ultra-fine grid configurations.

Regarding the aspect ratio, according to the Ansys Discovery User Manual (Version 2025 R2), the optimal aspect ratio for triangles and quadrilaterals is 1, although values up to 20 are considered acceptable [50].

In the present meshes, the maximum aspect ratios were 284 for the Coarse mesh and 152 for both the Medium and Fine meshes. These values, while exceeding the ideal free-stream recommendation of 20, are well within the acceptable range for inflation layers

and boundary layer resolution. The high-value aspect-ratio distribution is appropriately localised within the inflation layers of the nozzle's converging section to resolve near-wall gradients, while the core flow maintains a high-quality, uniform structure with aspect ratios well below the recommended limits.

2.2. Experimental Methodology and Data Selection

The experimental supersonic ejector was designed using the 1D analytical model proposed by Andal [45] and comprises a primary convergent–divergent nozzle integrated with a mixing chamber. The assembly was engineered to operate with a primary saturated steam mass flow rate of up to 2.9 kg/s supplied by well ThG-11 (representing approximately one-tenth of the well's total capacity) at an estimated enthalpy of 2739 kJ/kg and a nominal inlet pressure of 30 bar_g. The low-pressure secondary stream was fed from well ThG-15 as a two-phase mixture with a 20% vapour quality, an enthalpy of 1131 kJ/kg, and an average inlet pressure of 8.1 bar_g. The ejector discharged the combined fluid into the plant's steam gathering system at an outlet pressure of 9.8–9.9 bar_g, maintaining an operational pressure margin above the plant's standard baseline of 9.5 bar_g [47].

The internal physical schematic of the ejector assembly is shown in Figure 1, while the overall field experimental setup and process instrumentation layout are depicted in Figure 4.

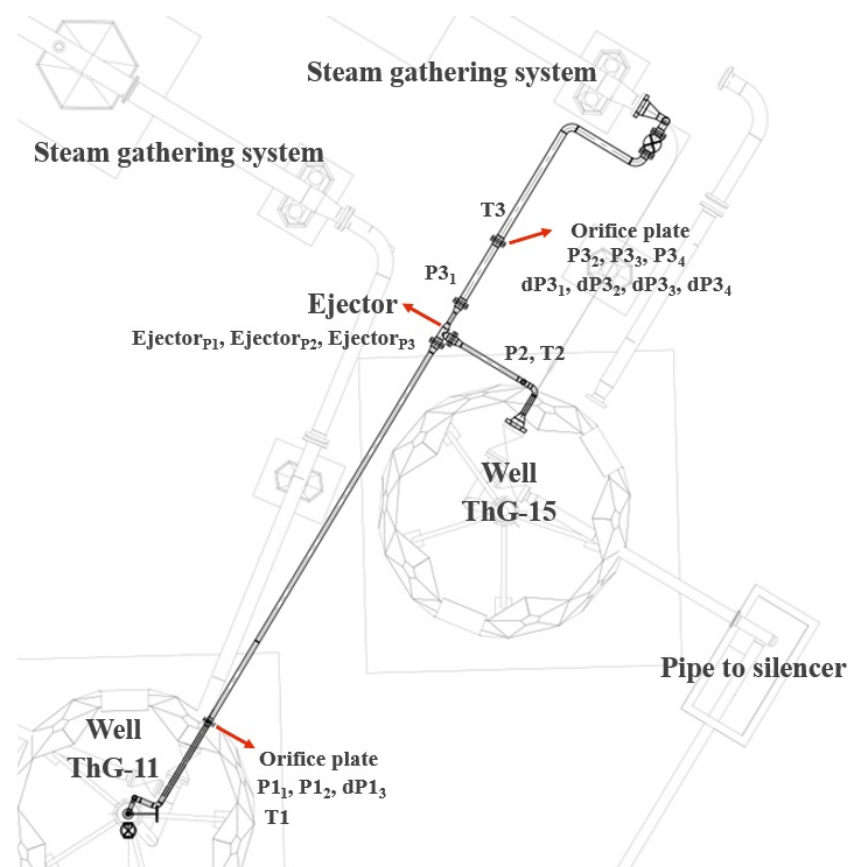


Figure 4. Schematic of the experimental setup illustrating the locations of the geothermal wells, ejector assembly, and instrumentation sensors. Obtained from Guardia et al. [47].

As shown in Figure 4, high-precision instrumentation was distributed across the upstream, internal, and downstream sections of the ejector assembly for field validation. Located 10.5 m upstream of the primary inlet, primary mass flow ($\dot{m}_1$) and localized fluid density were determined using a custom orifice plate ($\beta = 0.7$, $D = 66.01$ mm) coupled with a BD SENSORS DPT 200 differential pressure transmitter (0–20 bar, $\pm 0.075\%$

F.S.O.) and a WIKA CPG1500 static pressure gauge (0–40 bar, $\pm 0.05\%$ F.S.), with primary temperature (T_1) monitored near the wellhead [47]. Along the secondary suction line directly upstream of the suction chamber inlet of the ejector, a static pressure sensor (BD SENSORS DMP 331/333, 0–10 bar, $\pm 0.1\%$ F.S.) and temperature sensor (T_2) captured the incoming low-pressure two-phase properties [47]. Internally, three static pressure taps (Ejector_{p1}, Ejector_{p2}, and Ejector_{p3}) along the inner walls of the suction chamber and Constant Area Mixing Section (CAMS) tracked localised pressure recovery and shock-train compression profiles [47]. Downstream of the ejector outlet (2.5 m from the outlet), there was a static pressure transmitter (P_{31}), a secondary calibration orifice plate with Tek-Bar 3110B/3120B multi-channel differential pressure transmitters (0–145 psid/psig, $\pm 0.075\%$ F.S.), and a temperature sensor (T_3) to record the total mixed outlet mass flow ($\dot{m}_9$) and backpressure [47].

These physical measurements map directly onto the Computational Fluid Dynamics (CFD) boundary conditions and validation targets. The measured static pressure (P_1), temperature (T_1), and specific enthalpy (h_1 , determined via the Liquid Injection Tracer method) defined the thermodynamic state for the primary pressure and mass inlet boundary conditions [47]. Similarly, the secondary pressure (P_2), temperature (T_2), and enthalpy (h_2) established the secondary pressure inlet boundary condition, while the downstream static pressure (P_{31}) specified the pressure outlet boundary condition [47]. For numerical validation, wall pressure measurements from sensors Ejector_{p1–p3} were compared directly against simulated axial wall pressure distributions to evaluate shock-train placement, while the derived secondary entrained mass flow ($\dot{m}_5$) was benchmarked against the integrated cross-sectional mass flow from the CFD solution [47].

Primary and mixed outlet mass flow rates were measured directly using the calibrated upstream and downstream orifice plate assemblies, whereas the entrained secondary mass flow was derived indirectly via mass balance [47]:

$$\dot{m}_5 = \dot{m}_9 - \dot{m}_1 \quad (4)$$

Standard uncertainty propagation yielded an expanded relative uncertainty ($k = 2$) of $\pm 13.14\%$ for the derived secondary mass flow rate under peak entrainment conditions [47]. However, because the absolute uncertainty derived via mass balance ($u(\dot{m}_5) = \pm 0.138$ kg/s) remains relatively constant across operational regimes, the corresponding relative uncertainty bound ($\pm u(\dot{m}_5)/\dot{m}_5$) expands significantly as secondary flow rates diminish. At low-flow and off-design operating points ($\dot{m}_5 \approx 0.28$ – 0.34 kg/s), this relative uncertainty bound widens to 40–50%, demonstrating that the larger numerical discrepancies observed during reduced flow conditions remain comfortably within the experimental measurement uncertainty.

During testing at the Theistareykir Geothermal Power Plant in November 2024 [47], the suction pressure from well ThG-15 was maintained relatively constant at an average of 8.1 bar_g, while the primary supply pressure from well ThG-11 was systematically throttled from 28.5 down to 8.5 bar_g. This operational sequence generated a comprehensive empirical dataset across primary-to-secondary pressure differentials ranging from 9.9 to 26.3 bar [47]. From this testing campaign, eight operating data points were selected for numerical simulation and CFD validation, as highlighted in Figure 3 and Table 2.

The full experimental dataset containing primary and secondary pressures alongside the resulting entrained mass flow rates is summarised in Figure 5, with the eight selected numerical simulation points denoted by triangular markers.

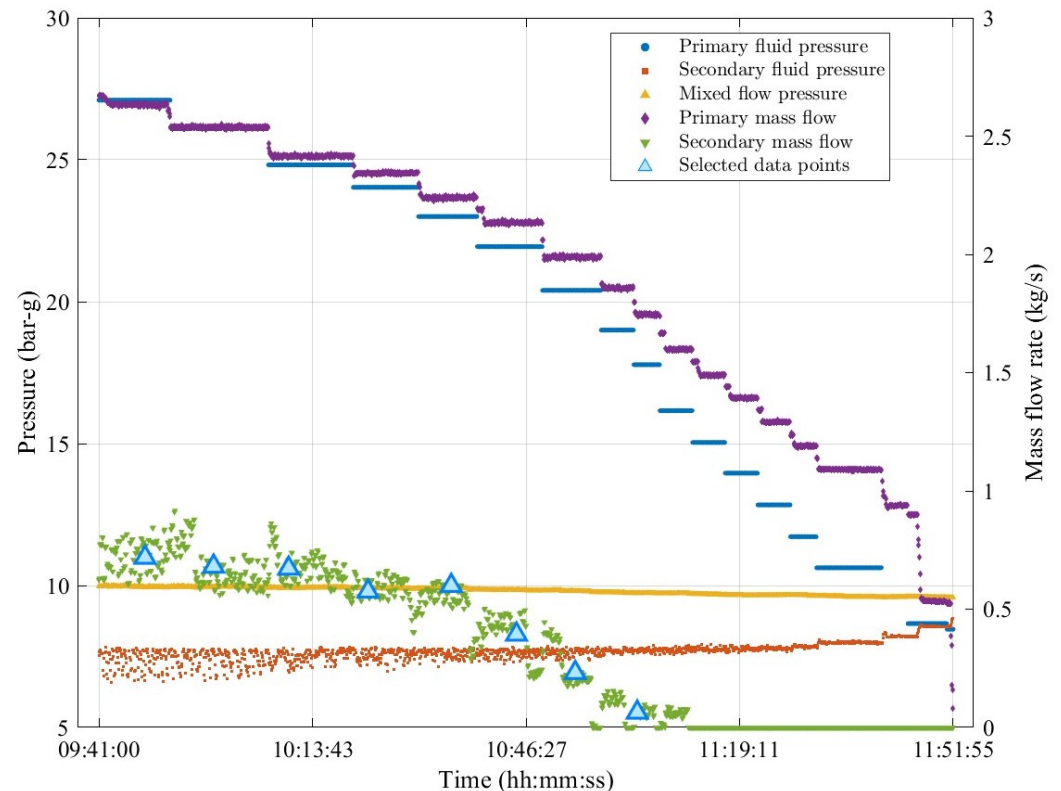


Figure 5. Field experimental data collected at the Theistareykir Geothermal Power Plant, with triangular markers indicating the operating points selected for CFD validation.

2.3. Boundary Conditions

The simulation setup utilises three boundary conditions: two inlets (primary and secondary) and one outlet (see Figure 2). As shown in Table 2, each inlet boundary was specified using measured pressure and temperature values obtained from the field data points selected for the analysis.

The fluids were modelled using the ANSYS FLUENT 2024 R1 wet-steam model, with the ejector material specified as steel using standard material properties. For the primary inlet, the fluid was modelled with a liquid mass fraction of 0.03–0.04 ($x = 0.96$ – 0.97), directly matching experimental conditions. However, a physical simplification was required for the secondary inlet: while field measurements from well ThG-15 indicated a heavily liquid-dominated fluid with a liquid mass fraction of 0.80–0.81 ($x \approx 0.20$), the secondary inlet boundary condition in the simulation was set to a liquid mass fraction of 0.20 ($x = 0.80$). This value represents the maximum liquid mass fraction ($\beta \leq 0.20$) physically and mathematically allowable within the wet-steam formulation of the solver [4,51].

Use of this boundary condition was necessary to maintain numerical stability and physical validity within the Eulerian wet-steam framework. Mathematically, the wet-steam model in ANSYS Fluent relies on classical non-equilibrium homogeneous nucleation and droplet growth theories [4,52]. These formulations explicitly assume a vapour-continuous phase containing finely dispersed liquid droplets with zero interphase velocity slip and negligible droplet-droplet interaction [4,51]. Forcing the solver to process the actual liquid-dominated mixture ($x \approx 0.20$, 80% liquid mass fraction) fundamentally violates the dispersed-phase assumption. At $x \approx 0.20$, real-world flows transition into a turbulent, unstable liquid–steam mixture where interphase momentum slip, liquid accumulation, and heavy drag dominate, causing immediate continuity and pressure–velocity coupling divergence in the solver [4].

Consequently, specifying the secondary inlet at $x = 0.80$ serves as an optimal numerical and physical approach for three main reasons:

1. It operates at the maximum physical limit of liquid loading that the Eulerian wet-steam equations can process without numerical breakdown [4,51].
2. In supersonic ejectors, secondary flow entrainment is predominantly driven by shear stress and momentum transfer between the primary jet core and the continuous vapour phase of the secondary flow [4]. In high-liquid geothermal streams, most of the liquid mass forms a slow-moving film along the pipe walls or heavy mist that contributes very little to the momentum driving secondary suction. A value of $x = 0.80$ isolates the high-velocity vapour core that actually drives entrainment, while maximising secondary moisture loading within the solver's mathematical limits [4].
3. While adopting a single-phase dry gas assumption ($x = 1.00$) avoids divergence, dry gas models completely neglect non-equilibrium phase change, latent heat release, and local speed-of-sound modifications [4]. Studies show that dry gas models overestimate Mach numbers by over 21% and entrainment ratios by up to 12% [4]. Setting the secondary inlet at $x = 0.80$ retains the vital thermal damping, condensation shock structure, and phase-change mass/energy source terms provided by the wet-steam model [4].

This necessary numerical simplification affects downstream mixing predictions, particularly near low-entrainment limits, and highlights a critical boundary condition in which baseline wet-steam formulations diverge from complex, two-phase geothermal wellhead dynamics.

Table 2. Selected field data and corresponding numerical boundary conditions.

Time	09:48:05	09:58:35	10:10:05	10:22:15	10:35:00	10:45:00	10:54:00	11:03:30
Primary Fluid Pressure [bar _g]	27.10	26.13	24.82	24.03	23.00	21.94	20.40	17.78
Primary Fluid Temperature [K]	503.40	501.49	498.82	497.16	494.94	492.57	488.97	482.37
Primary Fluid Liq. Frac [%]	0.04	0.03	0.03	0.03	0.03	0.03	0.03	0.03
Secondary Fluid Pressure [bar _g]	6.66	7.75	7.56	7.46	7.44	7.60	7.78	7.62
Secondary Fluid Temperature [K]	441.77	447.32	446.37	445.87	445.77	446.59	447.44	446.69
Secondary Fluid Liq. Frac [%]	0.20	0.20	0.20	0.20	0.20	0.20	0.20	0.20
Mixed Flow Pressure [bar _g]	9.93	9.95	9.92	9.92	9.91	9.84	9.80	9.73
Mixed Flow Temperature [K]	456.94	457.02	456.88	456.87	456.85	456.58	456.40	456.12
Mixed Flow Liq. Frac [%]	0.17	0.17	0.17	0.17	0.17	0.17	0.17	0.17

2.4. Discretisation Criteria

ANSYS FLUENT 2024 R1 utilises the finite volume method, which discretises the governing equations by integrating them across each control volume [52]. The two-phase ejector simulations employ a steady-state formulation based on fictitious time (pseudo-transient marching). In this approach, the solver advances the solution in pseudo-time until the steady state is reached [51].

Spatial discretisation of the convective flux terms for all momentum, energy, and phase transport equations was achieved using a Second-Order Upwind (SOU) scheme, coupled with an explicit scheme for pseudo-time marching. For high-speed compressible flows featuring shock-boundary layer interactions, the SOU formulation provides multi-dimensional directional bias and essential Total Variation Diminishing (TVD) properties. This prevents non-physical dispersion oscillations and unbounded gradient overshoots across severe pressure discontinuities, a common limitation of higher-order schemes such as QUICK in supersonic regimes, while maintaining formal spatial accuracy across the computational domain.

To account for non-equilibrium phase transition, the wet-steam model in ANSYS FLUENT 2024 R1 was utilised. This model employs a homogeneous Eulerian framework that enables phase change and compressible two-phase flow simulation [30]. The liquid mass fraction at each inlet was specified as a boundary condition within this framework, as detailed in Section 2.3.

ANSYS FLUENT 2024 R1 offers two distinct numerical formulations: the pressure-based solver (PBS) and the density-based solver (DBS). Historically, density-based solvers were the standard choice for high-speed compressible flows where density, energy, and momentum are strongly interdependent, such as in hypersonic regimes [14,18,52]. However, the PBS solver employs either a segregated or a coupled solution approach in which the governing equations are solved with pressure–velocity coupling, offering exceptional flexibility across a broad range of flow regimes from incompressible to moderately compressible flows [52]. Recent comparative benchmarks demonstrate that the pressure-based solver with a coupled pressure–velocity formulation is equally accurate for compressible supersonic ejector flows while providing significantly higher numerical stability and faster convergence [14,15,18,20]. Under this Coupled PBS algorithm, the momentum and pressure-based continuity equations are solved vectorially and simultaneously while evaluating density updates [18,20]. Because the flow through the ejector is supersonic ($\text{Mach} < 2.2$) rather than hypersonic, the Coupled PBS was selected for all simulations.

This pressure-based approach offers critical advantages when paired with the wet-steam formulation. Within multiphase wet-steam formulations, the pressure-based solver avoids the severe time-step restrictions and convergence instabilities common to density-based solvers when handling steep phase-change source terms [4,18]. Consequently, numerical evaluations confirm that the pressure-based solver yields reliable entrainment ratios and realistic thermodynamic fields under condensing conditions [18,26].

The physical dynamics of the geothermal steam ejector further support this solver selection. In supersonic steam ejectors, phase change and non-equilibrium condensation release latent heat into the continuous vapour phase, suppressing excessive flow expansion and capping maximum flow velocities to a moderate supersonic range ($\text{Mach} \approx 1.6\text{--}1.9$, compared to $\text{Mach} > 2.2$ predicted by dry-gas approximations) [5]. The pressure-based solver is specifically recognised for maintaining high numerical stability and physical accuracy within this low-to-moderate supersonic regime ($\text{Mach} < 2.2$) [4,52].

Finally, high-resolution spatial discretisation and mesh refinement ensure that complex compressible flow structures are accurately captured. Implementing a second-order upwind spatial discretisation scheme minimises numerical diffusion and false dissipation across steep gradients [7,18]. When paired with high-density mesh refinement in the mixing section and constant area mixing section (CAMS), this high-order discretisation enables the solver to sharply capture both aerodynamic shock phenomena, including oblique shock trains, compression diamonds, and normal shocks, and non-equilibrium condensation shocks [5,7,12,18,23].

2.5. Verification Summary

The numerical verification procedure followed a four-step framework to ensure solution accuracy and code reliability prior to physical validation:

1. **Grid Convergence:** A systematic grid refinement study evaluated three mesh resolutions (coarse: 1.42×10^6 , medium: 1.66×10^6 , fine: 3.15×10^6 elements) with a constant refinement ratio ($r = 2.0$). Grid independence was confirmed on the medium mesh, where primary mass flow varied by $< 0.4\%$ relative to the fine grid.

2. Near-Wall Grid Quality: Wall y^+ distributions were verified across the domain ($30 < y^+ < 300$ across critical momentum-exchange zones) to ensure validity of the Standard Wall Functions under high Reynolds number conditions ($Re > 10^6$).
3. Iterative Convergence: Rigorous convergence criteria were enforced, requiring continuity and RANS equation residuals to fall below 10^{-3} and energy residuals below 10^{-6} , combined with pseudo-transient marching and monitored mass flow balance across all ports.
4. Turbulence Closure Evaluation: Four RANS closures (Standard $k - \epsilon$, RNG $k - \epsilon$, Realizable $k - \epsilon$, and $k - \omega$) were systematically tested under identical boundary conditions to quantify sensitivity to turbulence formulation.

2.6. Convergence Criteria

To validate the simulation results, residuals must demonstrate convergence. Residuals indicate the degree to which conservation equations are satisfied at each iteration and should decrease over time. Convergence is achieved when the solution no longer changes from iteration to iteration [51]. Typically, residuals below 10^{-3} indicate qualitative convergence, meaning the main flow features are captured, while energy residuals should approach 10^{-6} when using PBS [51]. Residuals were continuously monitored, and simulations were only considered valid if convergence was achieved.

To address the convergence challenges arising from domain complexity, strong compressibility, and non-equilibrium condensation inherent to the wet steam model, conservative under-relaxation factors (URFs) were implemented (0.1 for momentum and energy, and 0.05 for pressure). These parameters limit the step size between successive iterations, mitigating abrupt variations in the solution matrix and preventing numerical divergence [48].

Because a time-independent, steady-state formulation was employed, these reduced URFs served purely to control iterative updates across pseudo-time steps without dampening physical time-dependent phenomena. Although the continuity and wet-steam scalar transportation equations presented the highest numerical stiffness, stable convergence across all cases was successfully achieved within a fixed run of 1000 iterations. Full numerical convergence was confirmed by ensuring that key physical monitors, specifically the outlet mass flow rate and throat static pressure, flatlined completely into asymptotic equilibrium, and all discrete residuals dropped to tight tolerance levels well prior to the final iteration.

3. Results

3.1. Mesh Independence Study

A mesh independence study was conducted across three mesh refinements (coarse, medium, and fine) to ensure that the numerical solutions were independent of grid resolution. Table 3 presents the simulation results for the three mesh refinements for four different turbulent models (Standard $k - \epsilon$, RNG, Realizable $k - \epsilon$, and $k - \omega$).

Based on the mesh independence study presented in Table 3, the primary fluid flow predictions were almost identical across all three mesh levels, indicating that this variable is largely insensitive to refinement beyond the coarsest grid. For the secondary fluid flow and entrainment ratio, the coarse and medium meshes produced similar results, suggesting that the medium mesh captures the essential flow physics without the excessive numerical dissipation associated with the coarse mesh. In contrast, the fine mesh exhibited simulation errors and convergence difficulties for several turbulence models (e.g., RNG $k - \epsilon$ and $k - \omega$), making it unreliable for production simulations. Furthermore, the computational cost of the medium mesh was lower than that of the coarse mesh, offering improved efficiency without compromising solution quality. Therefore, the medium mesh represents the most appropriate choice, balancing grid independence, numerical robustness, and computational time.

Table 3. Mesh independence study: Primary fluid flow, secondary fluid flow, and entrainment ratio for Standard $k - \epsilon$, RNG $k - \epsilon$, Realizable $k - \epsilon$, and $k - \omega$ turbulence models. Refinement ratio $r = 2.0$ between successive mesh levels.

Coarse Mesh				
Turbulence Model	Standard $k - \epsilon$	RNG $k - \epsilon$	Realizable $k - \epsilon$	$k - \omega$
Primary fluid flow [kg/s]	2.55	2.47	2.47	2.49
Secondary fluid flow [kg/s]	0.23	0.71	0.75	0.38
Entrainment Ratio (ER)	0.09	0.29	0.31	0.15
Iterations completed [%]	100	100	100	100
Medium Mesh				
Turbulence Model	Standard $k - \epsilon$	RNG $k - \epsilon$	Realizable $k - \epsilon$	$k - \omega$
Primary fluid flow [kg/s]	2.55	2.47	2.44	2.49
Secondary fluid flow [kg/s]	0.24	0.62	0.64	0.33
Entrainment Ratio (ER)	0.09	0.25	0.26	0.13
Iterations completed [%]	100	100	100	100
Fine Mesh				
Turbulence Model	Standard $k - \epsilon$	RNG $k - \epsilon$	Realizable $k - \epsilon$	$k - \omega$
Primary fluid flow [kg/s]	2.55	2.51	2.45	2.49
Secondary fluid flow [kg/s]	0.21	0.88	0.44	—
Entrainment Ratio (ER)	0.08	0.35	0.18	—
Iterations completed [%]	100	84	100	45

Note: “—” indicates missing data due to convergence failure.

3.2. Experimental Validation of Turbulence Models

The accuracy of the selected medium mesh was validated against field data from the 10:35:00 data point, which had the mean primary pressure across all selected data points presented in Figure 3. The mesh was evaluated using four turbulence models: Standard $k - \epsilon$, RNG $k - \epsilon$, Realizable $k - \epsilon$, and $k - \omega$.

The Entrainment Ratio (ER), defined as the ratio of secondary to primary fluid mass flow rates [26], was selected as the primary metric for convergence because it acts as a sensitive indicator of mass balance accuracy in both fluid flows. Quantitative comparisons between the CFD results and field measurements are detailed in Table 4 and Figure 6.

Table 4. Validation of the medium mesh: Comparison of CFD predictions against field test data (recorded at 10:35:00 h) for four turbulence models. Percentage errors relative to field data are shown in parentheses.

Variable	Field Data	Standard $k - \epsilon$	RNG $k - \epsilon$	Realizable $k - \epsilon$	$k - \omega$
Primary fluid flow (kg/s)	2.24	2.55 (13.8%)	2.47 (10.3%)	2.44 (8.9%)	2.49 (11.2%)
Secondary fluid flow (kg/s)	0.60	0.24 (60.0%)	0.62 (3.3%)	0.64 (6.7%)	0.33 (45.0%)
Entrainment Ratio	0.27	0.09 (66.7%)	0.25 (7.4%)	0.26 (3.7%)	0.13 (51.9%)
Absolute average error	—	46.8%	7.0%	6.4%	36.0%

From Table 4 and Figure 6, it can be observed that, while all models captured the general flow physics, the Realizable $k - \epsilon$ model was identified as the most robust and accurate for this application. This model consistently yielded the lowest average absolute error (6.4%) among all turbulence models. Specifically, it achieved errors of 8.9% for primary fluid flow, 6.7% for secondary fluid flow, and 3.7% for the Entrainment Ratio. The RNG $k - \epsilon$ model also performed well, with an average error of 7.0%, closely following the Realizable $k - \epsilon$ model. In contrast, the Standard $k - \epsilon$ and $k - \omega$ models exhibited significantly higher average errors of 46.8% and 36.0%, respectively, indicating their inadequacy for this flow configuration.

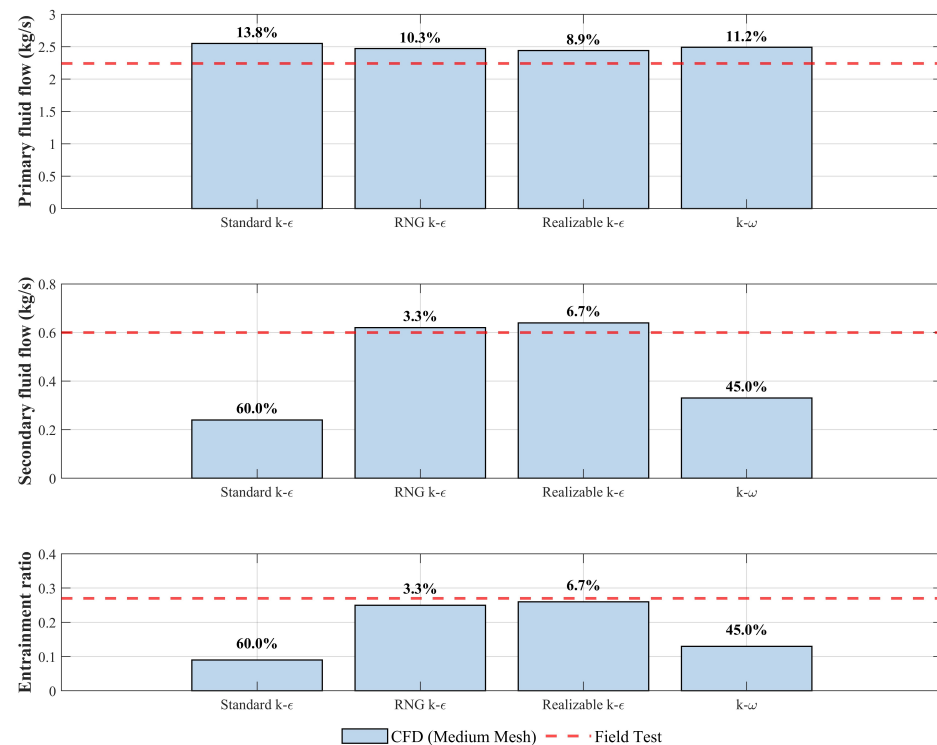


Figure 6. Validation of the medium mesh: Comparison of turbulence models (Standard $k-\epsilon$, RNG $k-\epsilon$, Realizable $k-\epsilon$, $k-\omega$) against field data (10:35:00 h in Figure 5) for primary fluid flow, secondary fluid flow, and entrainment ratio. Percentage errors to the measured values are shown above each bar.

Given that the Realizable $k-\epsilon$ model provided the best balance of accuracy and computational reliability, it was selected as the standard model for the remainder of this study. To ensure its validity in conjunction with Standard Wall Functions, the wall y^+ distribution was analysed. As illustrated in Figure 7, the majority of the high-velocity flow resides within the valid log-law region required for wall function accuracy ($30 < y^+ < 300$) [48]. While localised y^+ values below the ideal limit ($y^+ < 30$) occur in low-velocity stagnation zones, these deviations are confined to areas with negligible momentum exchange and do not significantly impact global mass flow or Entrainment Ratio predictions.

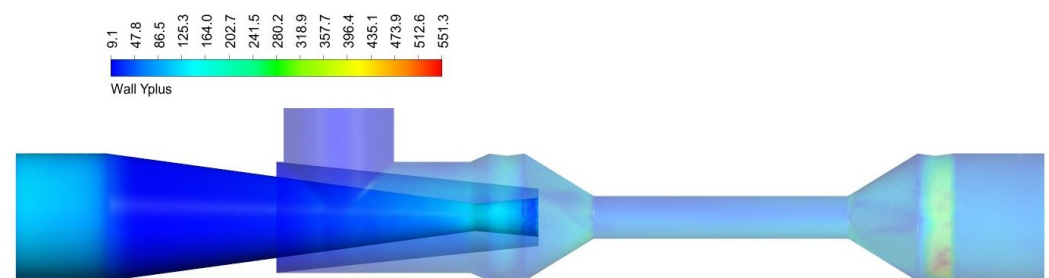


Figure 7. Distribution of y^+ values for the 10:35:00 data point (medium mesh, Realizable $k-\epsilon$ model). Outer surfaces are rendered with partial transparency to reveal internal wall regions.

In contrast, certain regions at the edges of the Constant Area Mixing Section (CAMS) and within the diffuser exhibited y^+ values exceeding the conventional limit of 300. This is considered acceptable for the current supersonic ejector study, given the high Reynolds numbers ($Re > 10^6$) involved. In such high-speed flows, the logarithmic region of the turbulent boundary layer extends significantly further than in subsonic applications, often remaining valid up to $y^+ \approx 10^3$ or higher. Since the Entrainment Ratio demonstrated stability

across the tested mesh densities, these elevated y^+ values are confirmed to be within the range where the wall functions effectively bridge the viscosity-affected near-wall region.

As the primary fluid expands through the nozzle, the boundary layer remains fully turbulent and well-developed. The placement of the first interior cell within this extended log-law region ensures that the Standard Wall Functions accurately calculate the wall shear stress and heat transfer.

3.3. Mass Flow

The medium refinement mesh with the Realizable $k - \varepsilon$ turbulence model was used for all two-phase simulations of the Theistareykir field selected data (see Figure 5). The mesh and simulation settings remained unchanged between simulations; only the boundary conditions were adjusted to match those of the various selected data points. Table 5 and Figure 8 present the main CFD results, along with relative percentage errors for the primary and secondary fluid flows and the resulting Entrainment Ratio, compared with the experimental measurements.

Table 5. Comprehensive validation using Realizable $k - \varepsilon$ model: mass flow and entrainment ratio with error analysis.

Time	Primary Fluid Flow			Secondary Fluid Flow			Entrainment Ratio		
	CFD (kg/s)	Field Test (kg/s)	Error (%)	CFD (kg/s)	Field Test (kg/s)	Error (%)	CFD	Field Test	Error (%)
09:48:05	2.87	2.63	9.13	0.69	0.72	4.17	0.24	0.27	12.50
09:58:35	2.81	2.54	10.63	0.62	0.68	8.82	0.22	0.27	18.52
10:10:05	2.66	2.41	10.37	0.60	0.67	10.45	0.23	0.28	17.86
10:22:15	2.56	2.35	8.94	0.72	0.58	24.14	0.28	0.25	12.00
10:35:00	2.44	2.24	8.93	0.64	0.60	6.67	0.26	0.27	3.70
10:45:00	2.34	2.14	9.35	0.42	0.39	7.69	0.18	0.18	0.00
10:54:00	2.18	1.99	9.55	0.18	0.23	21.74	0.08	0.12	33.33
11:03:30	1.96	1.74	12.64	0.09	0.06	50.00	0.05	0.03	40.00

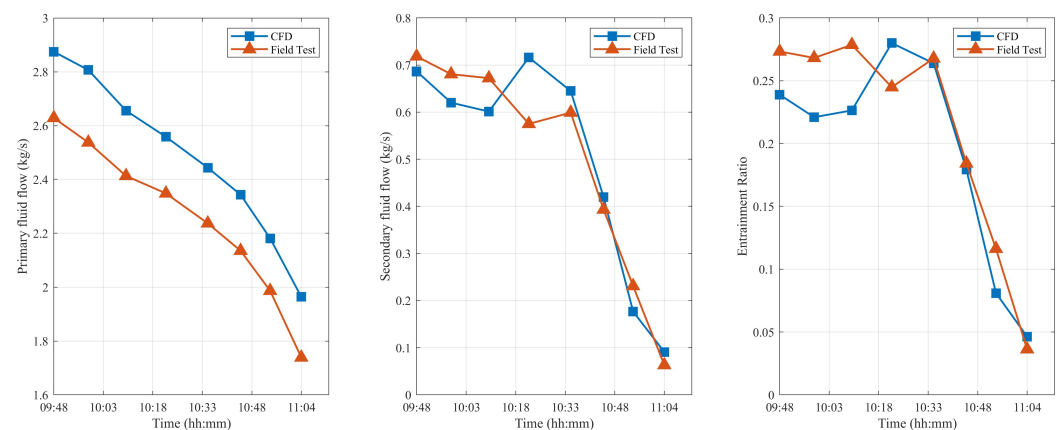


Figure 8. Comparison between simulated and the field test results for primary fluid flow, secondary fluid flow and entrainment ratio for selected data points shown in Figure 5.

Across all simulated cases, the primary mass flow rates yielded by the CFD model consistently overpredicted the values recorded during the field tests. The relative errors for the primary fluid flow ranged from a minimum of 8.93% (at 10:35:00 h) to a maximum of 12.64% (at 11:03:30 h). The systematic overprediction of the primary mass flow rate is fundamentally due to non-equilibrium thermodynamic relaxation and subcooling effects during rapid expansion of the primary fluid in the supersonic nozzle, which are not properly accounted for in the numerical model [53].

In the ejector system tested in the geothermal power plant, the high-pressure primary steam expands almost instantaneously at extreme expansion rates in the nozzle. Because

of this high velocity, the vapour cannot maintain a normal equilibrium state; it becomes supersaturated and enters a temporary, unstable metastable state where it continues to behave as a pure gas even though its temperature has dropped below the condensation threshold [53]. As the fluid travels further, it undergoes spontaneous homogeneous nucleation, where the vapour abruptly reverts to equilibrium through the rapid formation and growth of microscopic liquid droplets out of the pure gas phase [53].

In the numerical setup, a steady-state, pressure-based solver utilising the Eulerian wet-steam model was employed, specifying a boundary condition constraint of $\log_{10}(\text{Droplets Per Unit Volume})$ equal to 0 m^{-3} at the primary fluid inlet to reflect the initial high-quality vapour phase (3–4% liquid fraction). Viscous heating was kept active, paired with standard wall functions modified for pressure gradient and thermal effects. Because standard commercial RANS codes rely on simplified theories that struggle to capture the precise timing of droplet formation and the resulting size distribution at high expansion rates [53], the solver slightly underestimates the onset of this phase change. This mismatch causes the model to overestimate the true fluid density at the sonic choking point. Because mass flow scales linearly with density at the throat, these modelling limitations result in a consistent overprediction compared to the field data.

Conversely, the secondary mass flow rates predicted by the simulation demonstrated much closer absolute agreement with the experimental data. The maximum baseline accuracy was achieved for the 09:48:05 h data point with a minor error of 4.17%, whereas the peak relative deviation reached 50.00% at 11:03:30 h. It is critical to contextualise this upper error bound; the absolute numerical values modelled for the low-entrainment data points (10:54:00 and 11:03:30 h) remain highly aligned with the field measurements. However, because these mass flow rates are close to zero, even minor absolute discrepancies yield a mathematically magnified relative percentage error.

It is also important to note that these discrepancies in the data points with secondary fluid entrainment close to zero can be attributed to the physical limitations of the steady-state Eulerian wet-steam model when resolving highly stratified or moisture-rich liquid-vapour regimes near the critical operating limit of a supersonic ejector.

Field-test data indicated that for the specific geometry of the ejector tested at the Theistareykir Geothermal Power Plant, the minimum physical pressure difference required between the primary and secondary fluid flows to successfully initiate suction is 9.9bar, which equates to a physical inlet pressure ratio ($P_{\text{primary}}/P_{\text{secondary}}$) of 2.2 (results from field tests at the Theistareykir Geothermal Power Plant, November 2024). As detailed in Table 2, the data points at 10:54:00 and 11:03:30 h operated at pressure ratios of 2.6 and 2.3, respectively, placing them immediately adjacent to this physical threshold. While physical suction persists down to 2.2, the CFD solver encounters numerical divergence and heightened relative errors near zero-entrainment at a higher ratio of 2.6. Consequently, to ensure highly reliable numerical predictions of secondary mass flow rates and avoid localised model divergence in future geothermal ejector designs, operational pressure ratios should ideally be maintained above 2.6 as a conservative modelling safety margin.

Regarding the overall entrainment ratio, the strongest agreement between the CFD simulations and field data occurred at 10:45:00 h, where both calculated values rounded to 0.18. This yielded a nominal reported error of 0.00% (with an unrounded relative difference of $\approx 1.5\%$), which occurs as individual discrepancies in primary flow (9.35%) and secondary flow (7.69%) partially offset each other within the ratio. Crucially, the observed deviations across the operational envelope remain within the expanded experimental measurement uncertainty ($\pm 13.14\%$), validating the capability of the numerical framework to capture real-world performance trends. Conversely, the maximum discrepancy was observed at

11:03:30 h with a 40.00% mismatch, reflecting the heightened sensitivity of secondary fluid entrainment modelling under choked or near-stalled conditions.

3.4. Mach Number

Figure 9 displays the Mach number profile for the two-phase industrial-scale ejector tested at Theistareykir, using the boundary conditions of the 10:35:00 h data point, which had the mean primary pressure across all selected data points presented in Figure 3.

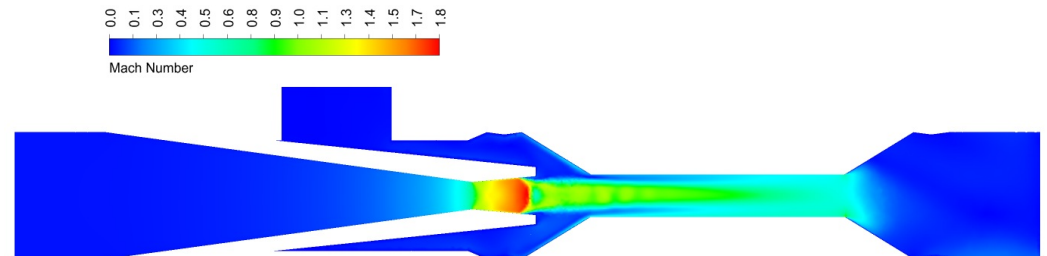


Figure 9. Mach number profile for the tested ejector using boundary conditions of the 10:35:00 h data point from Figure 5.

The Mach number profile shows that the ejector reaches sonic conditions (Mach 1) at the nozzle throat, then transitions to supersonic flow, with a maximum Mach number of about 1.8 near the nozzle outlet. After this, the sonic velocity decreases in the mixing section before increasing again as both fluids combine, just before entering the CAMS. Inside the CAMS, another transition from supersonic to subsonic conditions occurs, continuing until the ejector outlet.

3.5. Velocity

Figure 10 displays the velocity vectors for the industrial-scale ejector tested at Theistareykir, using the boundary conditions of the 10:35:00 h data point, while Figure 11 shows the velocity profile of the ejector for the same selected data point.

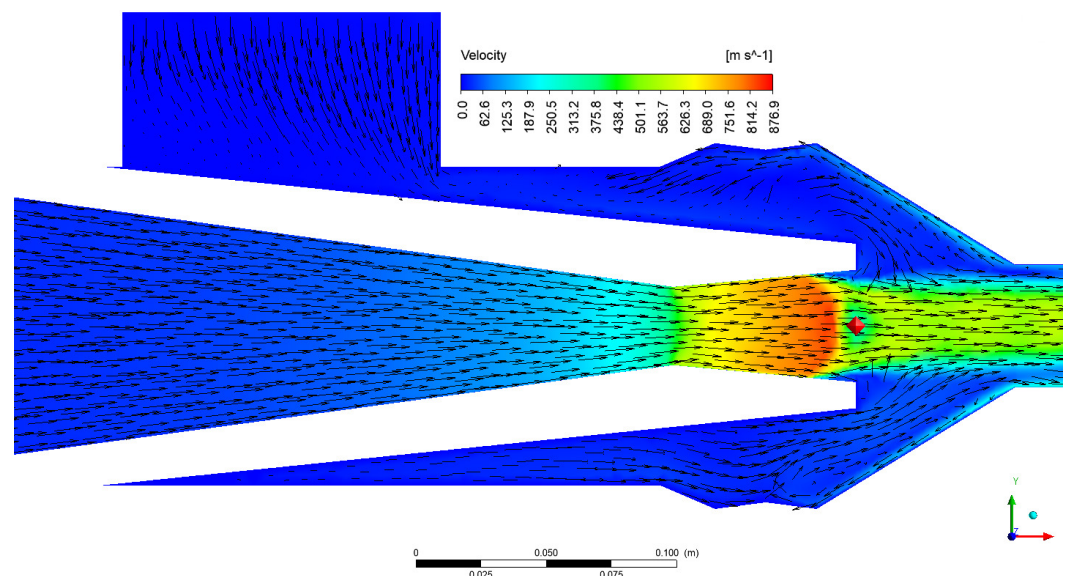


Figure 10. Velocity vectors for the tested ejector using boundary conditions of the 10:35:00 h data point from Figure 5. The red octahedron represents the outlet of the nozzle.

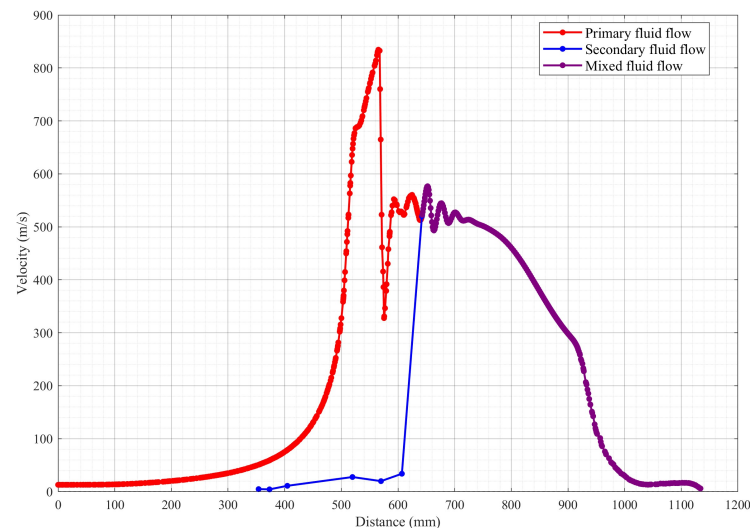


Figure 11. Velocity profile for the tested ejector using boundary conditions of the 10:35:00 h data point from Figure 5.

From the velocity vectors in Figure 10, it can be seen that the primary fluid flow exiting the nozzle expands into the CAMS entrance, but it does not collide with the converging section of the suction chamber, helping to avoid recirculation in that area. However, the internal wall design of the suction chamber creates vortices before the converging section, which can increase pressure in those regions, hindering the entry of the secondary fluid flow. From the analysis of the vortices' behaviour, having the nozzle's exit position (NXP) closer to the CAMS could decrease the pressure inside the CAMS and increase the entrainment ratio [27].

The velocity profile in Figure 11 shows that the primary fluid flow reaches a maximum velocity of 834.5 m/s in the diverging section of the nozzle and 330.9 m/s at the nozzle's outlet (red octahedron shown in Figure 8, while the secondary fluid flow enters the ejector at a very low velocity (4.8 m/s). This creates a highly turbulent flow when the two fluids mix, leading to exergy losses.

3.6. Pressure

Figure 12 illustrates the static pressure distribution for the industrial-scale ejector tested at Theistareykir, while Figure 13 plots the pressure profile along the central axis. Both visualizations utilise boundary conditions derived from the 10:35:00 h data point.

As shown in Figure 12 and Figure 13, the primary fluid flow undergoes a significant static pressure drop from 22.99 bar_g to 2.05 bar_g as it expands through the nozzle. This expansion creates a low-pressure region that entrains the secondary fluid flow, which enters at an average static pressure of 7.38 bar_g.

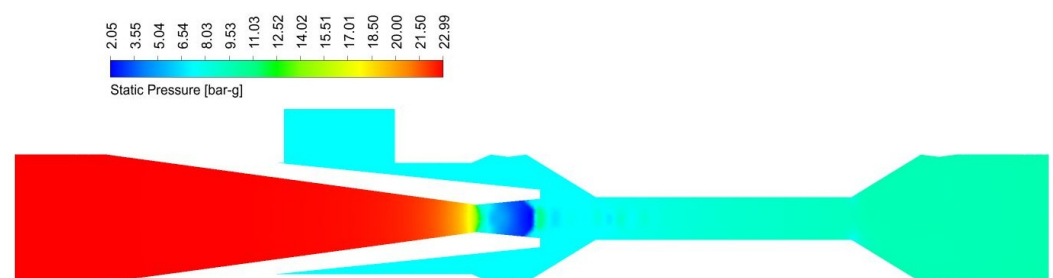


Figure 12. Static pressure for the tested ejector using boundary conditions of the 10:35:00 h data point from Figure 5.

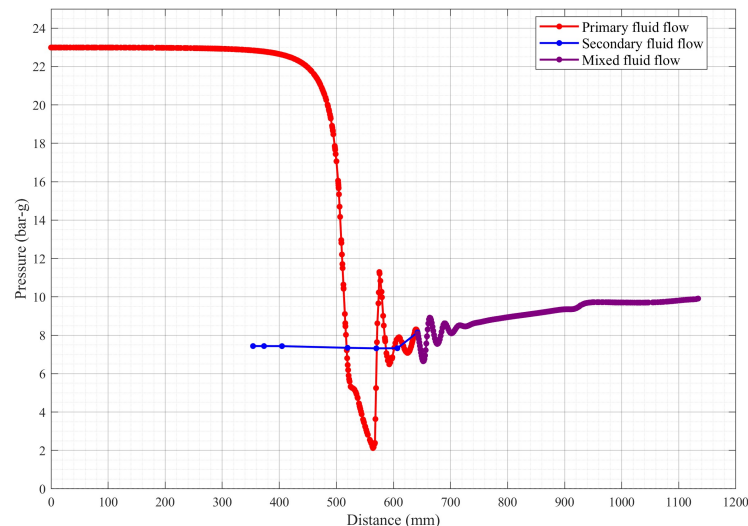


Figure 13. Static pressure profile of the central axis of the tested ejector using boundary conditions of the 10:35:00 h data point from Figure 5.

Mixing begins in the converging section of the suction chamber, where the two fluid flows interact, resulting in a uniform pressure distribution. The mixed flow reaches a peak static pressure of 9.91 bar_g just before entering the constant-area mixing section (CAMS).

Inside the CAMS, the flow decelerates through a shock train, with a minimum static pressure of 6.64 bar_g when primary and secondary fluids start to mix and with a mixed fluid static pressure of 9.39 bar_g at the exit of the CAMS. Further pressure recovery occurs in the diffuser, where the flow slows subsonically, increasing the mixed fluid flow static pressure to 9.91 bar_g at the exit of the ejector, which is greater than the power plant operating pressure of 9.5 bar_g. This means that, according to the CFD simulations, the mixed fluid flow can be entrained into the power plant system.

3.7. Enthalpy

Figure 14 displays the enthalpy of the fluid through the industrial-scale ejector tested at Theistareykir for the boundary conditions of the 10:35:00 h data point.

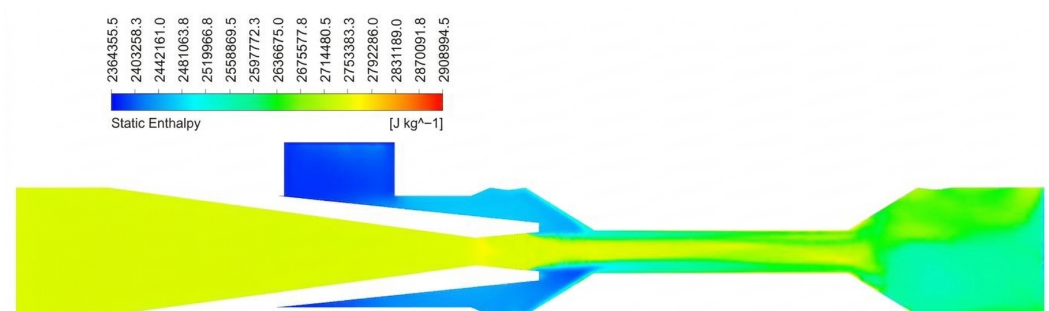


Figure 14. Enthalpy for the tested ejector using boundary conditions of the 10:35:00 h data point from Figure 5.

The high enthalpy of the primary fluid flow is reduced in the nozzle's diverging section as thermal energy is converted to kinetic energy, but a further reduction occurs when the primary and secondary fluid flows combine in the mixing chamber. An intermediate enthalpy is observed within the CAMS, and an increase in the diffuser is observed due to the pressure recovery of the mixed fluid flow. It is also important to notice that there is an intermediate enthalpy region in the converging section of the suction chamber after the entrance of the

secondary fluid and before the CAMS. This intermediate enthalpy can be attributed to the recirculation and pressure increase observed in Figure 10 and discussed in Section 3.5.

4. Discussion

The field-validated numerical framework developed in this study provides critical insights for engineering firms and geothermal power plant operators tasked with designing and deploying supersonic ejectors to combine high- and low-pressure geothermal wells. By demonstrating that the Realizable $k - \varepsilon$ model can predict the system's baseline performance with minimum relative errors of 8.93% for the primary fluid flow and 4.17% for the secondary fluid flow, this work establishes a robust numerical foundation to standardise and de-risk the design process of supersonic ejector systems, bypassing the need for expensive trial-and-error field prototyping.

However, industrial and commercial deployment must be reconciled with the model's localised physical divergence and limitations. Design engineers must specifically account for two major operational constraints when utilising this model:

- Primary mass flow overprediction margin: Across all operating points, the model consistently overpredicted the primary mass flow rate by 8.93% to 12.64% due to non-equilibrium thermodynamic relaxation and delayed homogeneous nucleation at the nozzle throat, as explained in Section 3.3. Design engineers must incorporate a safety factor into their nozzle throat sizing to prevent underestimating the primary fluid capacity.
- Model breakdown near zero entrainment: As detailed in Table 5, while the model tracks the experimental entrainment ratio with high precision during stable operating periods, relative errors spike drastically (up to 50.00% for secondary flow and 40.00% for entrainment ratio) as secondary mass flow approaches zero when the primary-to-secondary pressure ratio drops to 2.6 or less.

It is crucial to recognise that these elevated percentage errors arise primarily from mathematical scaling at small absolute flow magnitudes rather than massive physical divergence. For instance, at 11:03:30, the absolute deviation between the CFD prediction (0.09 kg/s) and field data (0.06 kg/s) is a mere 0.03 kg/s, which is the smallest absolute mass flow discrepancy recorded across the entire testing campaign. Because secondary fluid flow is derived via overall mass balance, fixed instrumental measurement tolerances generate a balance-derived relative experimental uncertainty ($\pm \delta \dot{m} / \dot{m}$) [47] that expands significantly as flow rates diminish. This causes the expanded uncertainty bound to expand from $\pm 13.14\%$ at peak flow to 40–50% in low-flow regimes, placing these low-flow model deviations comfortably within the experimental uncertainty bound.

Concurrently, these spikes reflect the inherent physical limitations of steady-state Eulerian wet-steam formulations when attempting to resolve highly dynamic, liquid-vapour phase separation and non-equilibrium condensation near critical operational thresholds. Rather than interpreting these near-zero entrainment predictions as exact quantitative performance metrics, design engineers and plant operators should treat these high relative error regions as a clear, robust numerical signal that the physical system is transitioning into an unchoked, backflow-prone operating regime.

Ultimately, operators can confidently utilise this CFD model as a predictive tool for the operational performance of a supersonic ejector system in geothermal power plants, defining the exact primary pressure thresholds at which the ejector ceases to perform useful work. By acknowledging these inherent boundary limitations and maintaining conservative engineering margins for high-moisture wellheads, this model serves as a powerful utility for scaling up commercial-scale two-phase flow geothermal ejector networks. When upscaling from the tested pilot unit with 1/10th of mass flow of well ThG-11 to full-capacity networks, high confidence in the derived scaling laws is supported by the fact that super-

sonic momentum transfer is primarily governed by inlet pressure ratios and Mach number distributions rather than absolute throughput. However, full-scale deployment requires addressing key scaling dynamics:

- Reynolds number effects: Upscaling increases the Reynolds number, which reduces the relative boundary layer displacement thickness along the mixing tube walls and decreases relative friction losses compared to the pilot scale.
- Mixing length and phase dynamics: Preserving geometric similarity, specifically the Area Ratio between the Constant Area Mixing Section and the primary nozzle's throat area, the length-to-diameter ratio of the Constant Area Mixing Section and Nozzle Exit Position (NXP), is essential for complete momentum exchange, while local expansion rates ($d\mu/dt$) must be controlled to manage non-equilibrium condensation in wet-steam regimes.
- Manufacturing tolerances and roughness: Larger throat dimensions diminish sensitivity to absolute machining tolerances, but maintaining low relative surface roughness remains critical to prevent shock-induced boundary layer separation in the supersonic diffuser.

5. Conclusions

A two-phase CFD model of an industrial-scale ejector was developed and validated against field-scale operational data from the Theistareykir geothermal power plant. A grid independence study determined that a medium-refinement mesh achieved a stable, grid-independent solution while optimising computational efficiency.

To identify the most reliable turbulence treatment, four models were evaluated: Standard $k - \epsilon$, RNG $k - \epsilon$, Realizable $k - \epsilon$, and $k - \omega$. The Realizable $k - \epsilon$ model demonstrated superior predictive accuracy, yielding the lowest error for the entrainment ratio compared to the measured field data. Consequently, it was selected as the optimal model for subsequent multi-point operational validation.

Validation across diverse data points revealed that the Realizable $k - \epsilon$ wet-steam model is highly sensitive to localised thermodynamic transitions. The primary fluid flow was predicted with a consistent overestimation error ranging from 8.93% to 12.64% across all cases. This systematic deviation is fundamentally tied to non-equilibrium thermodynamic relaxation and delayed spontaneous homogeneous nucleation at the nozzle throat. To account for this, engineering firms must incorporate a proportional sizing safety factor for the primary fluid entrainment when designing the converging–diverging nozzle.

Conversely, the secondary fluid flow and entrainment ratio exhibited greater variance under certain operational regimes. The Realizable $k - \epsilon$ model achieved its highest baseline accuracy for secondary fluid flow at 09:48:05 with a minor error of 4.17%. For the overall entrainment ratio, the strongest agreement occurred at 10:45:00, where both calculated values rounded to 0.18 for a nominal error of 0.00% (an unrounded relative difference of $\approx 1.5\%$), as primary (9.35%) and secondary (7.69%) flow discrepancies partially offset each other. Conversely, the maximum discrepancy occurred at 11:03:30 with a 40.00% mismatch, reflecting the sensitivity of entrainment modelling under choked or near-stalled conditions.

Operational phases approaching zero secondary entrainment exhibited high relative errors (up to 50.00% for secondary flow and 40.00% for entrainment ratio). These spikes arise primarily from relative percentage scaling over small absolute mass flow offsets (0.03 kg/s) combined with expanded experimental measurement uncertainty at low flow rates. Numerically, these high relative discrepancies and associated solver divergence serve as a distinct indicator that the model is approaching the zero-entrainment boundary ($P_{\text{primary}}/P_{\text{secondary}} \leq 2.6$). While physical backflow in the field initiates at a lower ratio of 2.2 (9.9bar pressure difference between the primary and secondary inlets), this 2.6 ratio represents the critical numerical stability limit where the steady-state Eulerian model loses predictive fidelity.

Overall, the simulations successfully characterised the device's complex internal physics, confirming the presence of supersonic flow in the tested ejector system. The high-pressure motive fluid from well ThG-11 effectively entrains the secondary fluid from well ThG-15, producing a mixed discharge flow with an intermediate pressure exceeding the power plant's operational threshold.

These findings validate the Realizable $k - \varepsilon$ wet-steam model as a robust tool for simulating internal flows in supersonic geothermal ejectors, even when handling secondary fluids with high liquid mass fractions, as demonstrated in this study. Consequently, this model can guide the design and optimisation of ejectors that combine high- and low-pressure geothermal wells, ultimately enhancing energy recovery in geothermal power plant operations.

Author Contributions: Conceptualization, X.G.M.; methodology, X.G.M.; software, X.G.M.; validation, X.G.M.; formal analysis, X.G.M.; investigation, X.G.M. and C.G.; writing—original draft preparation, X.G.M. and C.G.; writing—review and editing, Y.A.T., G.A.S. and M.S.G.; supervision, Y.A.T., G.A.S. and M.S.G.; project administration, M.S.G.; funding acquisition, M.S.G. All authors have read and agreed to the published version of the manuscript.

Funding: Financial assistance for this project was sourced from the Rannís Technology Development Fund (Grant Number 2112948-0611), the VOR Reykjavik Energy Research Fund, the Landsvirkjun Energy Fund (Project Number OLV-2026-06), the Rannís Student Innovation Fund, the Reykjavik University Infrastructure Fund, and the Sustainable Institute and Forum (SIF) from Reykjavik University.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The data presented in this study are available on request from the corresponding author.

Acknowledgments: We appreciate the contributions of the employees at the Landsvirkjun Theistareykir geothermal power plant, who helped collect the field test data used to validate the numerical model evaluated in this manuscript. During the preparation of this manuscript, the authors used Grammarly, Writefull, and Gemini to assist with language editing and writing refinement. The authors reviewed and edited the content as needed and take full responsibility for the content of the publication.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations

The following abbreviations and symbols are used in this manuscript:

CAMS	Constant Area Mixing Section (mm)
CFD	Computational Fluid Dynamics
ER	Entrainment Ratio
Ma	Mach Number
NXP	Nozzle Exit Position (mm)
c_p	Specific heat at constant pressure ($\text{J} \cdot \text{kg}^{-1} \cdot \text{K}^{-1}$)
D	Section diameter (mm)
h	Specific enthalpy ($\text{kJ} \cdot \text{kg}^{-1}$)
$\dot{m}$	Mass flow rate ($\text{kg} \cdot \text{s}^{-1}$)
p	Pressure (bar_g)
T	Temperature (K)
u_i	Streamwise velocity component in the i -th direction ($\text{m} \cdot \text{s}^{-1}$)
$u'_i T'$	Turbulent heat flux ($\text{W} \cdot \text{m}^{-2}$)

v	Velocity ($\text{m} \cdot \text{s}^{-1}$)
y^+	Non-dimensional wall distance (y^+ value)
λ	Thermal conductivity ($\text{W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$)
μ_i	Dynamic viscosity ($\text{kg} \cdot \text{m}^{-1} \cdot \text{s}^{-1}$)
$\mu\phi$	Viscous dissipation function
ρ	Density ($\text{kg} \cdot \text{m}^{-3}$)

References

- McGovern, R.K.; Narayan, G.P.; Lienhard, J.H. Analysis of reversible ejectors and definition of an ejector efficiency. *Int. J. Therm. Sci.* **2012**, *54*, 153–166. [\[CrossRef\]](#)
- White, F.M. *Fluid Mechanics*, 8th ed.; McGraw-Hill Education: New York, NY, USA, 2016.
- Guardia, X.; Andal, J.M.; Lárusson, R.; Sævarsdóttir, G.; Tesfahunegn, Y.A.; Júlíusson, E.; Chauhan, V.; Sveinsson, K.E.; Guðjónsdóttir, M.S. Connecting high- and low-pressure geothermal wells using an ejector: Analysis of first field tests at the Theistareykir Geothermal Power Plant. In Proceedings of the Proceedings World Geothermal Congress 2023, Beijing, China, 17–21 April 2023; pp. 1–11.
- Yazdani, M.; Alahyari, A.A.; Radcliff, T.D. Numerical modeling of two-phase supersonic ejectors for work-recovery applications. *Int. J. Heat Mass Transf.* **2012**, *55*, 5744–5753. [\[CrossRef\]](#)
- Yang, Y.; Zhu, X.; Yan, Y.; Ding, H.; Wen, C. Performance of supersonic steam ejectors considering the nonequilibrium condensation phenomenon for efficient energy utilisation. *Appl. Energy* **2019**, *242*, 157–167. [\[CrossRef\]](#)
- Kong, F.; Kim, H. Analytical and computational studies on the performance of a two-stage ejector–diffuser system. *Int. J. Heat Mass Transf.* **2015**, *85*, 71–87. [\[CrossRef\]](#)
- Croquer, S.; Poncet, S.; Aidoun, Z. Turbulence modeling of a single-phase R134a supersonic ejector. Part 2: Local flow structure and exergy analysis. *Int. J. Refrig.* **2016**, *61*, 153–165. [\[CrossRef\]](#)
- Croquer, S.; Poncet, S.; Galanis, N. Comparison of ejector predicted performance by thermodynamic and CFD models. *Int. J. Refrig.* **2016**, *68*, 28–36. [\[CrossRef\]](#)
- Tang, Y.; Liu, Z.; Li, Y.; Zhao, F.; Fan, P.; Chua, K.J. Mixing process of two streams within a steam ejector from the perspectives of mass, momentum and energy transfer. *Appl. Therm. Eng.* **2021**, *185*, 116358. [\[CrossRef\]](#)
- Wang, C.; Wang, L. Investigation of Fluid Characteristic and Performance of an Ejector by a Wet Steam Model. *Entropy* **2023**, *25*, 85. [\[CrossRef\]](#)
- Aidoun, Z.; Ameur, K.; Falsafioon, M.; Badache, M. Current Advances in Ejector Modeling, Experimentation and Applications for Refrigeration and Heat Pumps. Part 1: Single-Phase Ejectors. *Inventions* **2019**, *4*, 15. [\[CrossRef\]](#)
- Schillaci, E.; Vera, J.; Oliet, C.; Vemula, J.B.; Duponcheel, M.; Bartosiewicz, Y. Numerical modeling of air ejectors covering supersonic, subsonic and closed-port operations. *Energy* **2024**, *302*, 131627. [\[CrossRef\]](#)
- K., G.; Sundaram, S.S. Numerical analysis on ejector performance with varying geometric parameters of chevrons. *Int. J. Refrig.* **2025**, *178*, 180–191. [\[CrossRef\]](#)
- Xiao, J.; Wu, Q.; Chen, L.; Ke, W.; Wu, C.; Yang, X.; Yu, L.; Jiang, H. Assessment of Different CFD Modeling and Solving Approaches for a Supersonic Steam Ejector Simulation. *Atmosphere* **2022**, *13*, 144. [\[CrossRef\]](#)
- Muhammad, H.A.; Abdullah, H.M.; Rehman, Z.; Lee, B.; Baik, Y.J.; Cho, J.; Imran, M.; Masud, M.; Saleem, M.; Butt, M.S. Numerical Modeling of Ejector and Development of Improved Methods for the Design of Ejector-Assisted Refrigeration System. *Energies* **2020**, *13*, 5835. [\[CrossRef\]](#)
- Pianthong, K.; Seehanam, W.; Behnia, M.; Sriveerakul, T.; Aphornratana, S. Investigation and improvement of ejector refrigeration system using computational fluid dynamics technique. *Energy Convers. Manag.* **2007**, *48*, 2556–2564. [\[CrossRef\]](#)
- Yan, J.; Wen, N.; Wang, L.; Li, X.; Liu, Z.; Li, S. Optimization on ejector key geometries of a two-stage ejector-based multi-evaporator refrigeration system. *Energy Convers. Manag.* **2018**, *175*, 142–150. [\[CrossRef\]](#)
- Besagni, G.; Cristiani, N.; Croci, L.; Guédon, G.R.; Inzoli, F. Computational fluid-dynamics modelling of supersonic ejectors: Screening of modelling approaches, comprehensive validation and assessment of ejector component efficiencies. *Appl. Therm. Eng.* **2021**, *186*, 116431. [\[CrossRef\]](#)
- Li, Y.; Shen, S.; Niu, C.; Mu, X.; Zhang, L. The effect of variable motive pressures on the performance and shock waves in a supersonic steam ejector with non-equilibrium condensing. *Int. J. Therm. Sci.* **2023**, *185*, 108034. [\[CrossRef\]](#)
- Talebiyan, M.A.; Nili-Ahmadabadi, M.; Ha, M.Y. Performance enhancement of supersonic ejectors via aerodynamic cavities: A numerical parametric study. *J. Mech. Sci. Technol.* **2025**, *39*, 259–274. [\[CrossRef\]](#)
- Bartosiewicz, Y.; Aidoun, Z.; Desevaux, P.; Mercadier, Y. Numerical and experimental investigations on supersonic ejectors. *Int. J. Heat Fluid Flow* **2005**, *26*, 56–70. [\[CrossRef\]](#)

22. Sharifi, N.; Boroomand, M.; Sharifi, M. Numerical assessment of steam nucleation on thermodynamic performance of steam ejectors. *Appl. Therm. Eng.* **2013**, *52*, 449–459. [\[CrossRef\]](#)
23. Ariaifar, K.; Buttsworth, D.; Al-Doori, G.; Malpress, R. Effect of mixing on the performance of wet steam ejectors. *Energy* **2015**, *93*, 2030–2041. [\[CrossRef\]](#)
24. Tang, Y.; Liu, Z.; Li, Y.; Huang, Z.; Chua, K.J. Study on fundamental link between mixing efficiency and entrainment performance of a steam ejector. *Energy* **2021**, *215*, 119128. [\[CrossRef\]](#)
25. Sun, W.; Ma, X.; Zhang, Y.; Jia, L.; Xue, H. Performance analysis and optimization of a steam ejector through streamlining of the primary nozzle. *Case Stud. Therm. Eng.* **2021**, *27*, 101356. [\[CrossRef\]](#)
26. Ding, Z.; Wang, L.; Zhao, H.; Zhang, H.; Wang, C. Numerical study and design of a two-stage ejector for subzero refrigeration. *Appl. Therm. Eng.* **2016**, *108*, 436–448. [\[CrossRef\]](#)
27. Yu, M.; Zou, L.; Yu, J. Numerical investigation on the effect of area ratio on an adjustable ejector for heat pump dryer based on dynamic mesh. *Energy* **2024**, *308*, 132876. [\[CrossRef\]](#)
28. Guo, D.; Wang, J.; Zhu, Z. Numerical investigation on performance optimization of steam ejector considering full geometric parameters and working conditions. *Case Stud. Therm. Eng.* **2024**, *59*, 104552. [\[CrossRef\]](#)
29. Zhang, X.; Zhang, H.; Kong, F.; Zhang, C.; Xin, G.; Zhao, H.; Zhang, G. Numerical simulation of blade-type adjustable steam ejector. *Appl. Therm. Eng.* **2024**, *238*, 122199. [\[CrossRef\]](#)
30. Charton, H.; Perret, C.; Phan, H.T. Analysis of Supersonic Flows inside a Steam Ejector with Liquid–Vapor Phase Change Using CFD Simulations. *Thermo* **2024**, *4*, 1–15. [\[CrossRef\]](#)
31. Besagni, G.; Inzoli, F. Computational fluid-dynamics modeling of supersonic ejectors: Screening of turbulence modeling approaches. *Appl. Therm. Eng.* **2017**, *117*, 122–144. [\[CrossRef\]](#)
32. Atmaca, M.; Ezgi, C. Three-dimensional CFD modeling of a steam ejector. *Energy Sources, Part A Recover. Util. Environ. Eff.* **2022**, *44*, 2236–2247. [\[CrossRef\]](#)
33. Ramesh, A.S.; Sekhar, S.J. Experimental and numerical investigations on the effect of suction chamber angle and nozzle exit position of a steam-jet ejector. *Energy* **2018**, *164*, 1097–1113. [\[CrossRef\]](#)
34. Yang, X.; Long, X.; Yao, X. Numerical investigation on the mixing process in a steam ejector with different nozzle structures. *Int. J. Therm. Sci.* **2012**, *56*, 95–106. [\[CrossRef\]](#)
35. Varga, S.; Oliveira, A.C.; Ma, X.; Omer, S.A.; Zhang, W.; Riffat, S.B. Comparison of CFD and experimental performance results of a variable area ratio steam ejector. *Int. J. Low-Carbon Technol.* **2011**, *6*, 119–124. [\[CrossRef\]](#)
36. Ariaifar, K.; Buttsworth, D.; Al-Doori, G.; Sharifi, N. Mixing layer effects on the entrainment ratio in steam ejectors through ideal gas computational simulations. *Energy* **2016**, *95*, 380–392. [\[CrossRef\]](#)
37. Sriveerakul, T.; Aphornratana, S.; Chunnanond, K. Performance prediction of steam ejector using computational fluid dynamics: Part 1. Validation of the CFD results. *Int. J. Therm. Sci.* **2007**, *46*, 812–822. [\[CrossRef\]](#)
38. Ji, M.; Utomo, T.; Woo, J.; Lee, Y.; Jeong, H.; Chung, H. CFD investigation on the flow structure inside thermo vapor compressor. *Energy* **2010**, *35*, 2694–2702. [\[CrossRef\]](#)
39. Ding, H.; Zhao, Y.; Wen, C.; Wang, C.; Sun, C. Energy efficiency and exergy destruction of supersonic steam ejector based on nonequilibrium condensation model. *Appl. Therm. Eng.* **2021**, *189*, 116704. [\[CrossRef\]](#)
40. Varga, S.; Soares, J.; Lima, R.; Oliveira, A.C. On the selection of a turbulence model for the simulation of steam ejectors using CFD. *Int. J. Low-Carbon Technol.* **2017**, *12*, 233–243. [\[CrossRef\]](#)
41. Wang, X.D.; Dong, J.L. Numerical study on the performances of steam-jet vacuum pump at different operating conditions. *Vacuum* **2010**, *84*, 1341–1346. [\[CrossRef\]](#)
42. Petrovic, A.; Svorcan, J.; Pejcev, A.; Radenkovic, D.; Petrovic, A. Comparison of novel variable area convergent-divergent nozzle performances obtained by analytic, computational and experimental methods. *Appl. Math. Model.* **2018**, *57*, 206–225. [\[CrossRef\]](#)
43. Chen, W.; Chong, D.; Yan, J.; Liu, J. The numerical analysis of the effect of geometrical factors on natural gas ejector performance. *Appl. Therm. Eng.* **2013**, *59*, 21–29. [\[CrossRef\]](#)
44. Gustol, A. Ejector for Geothermal Well Lifetime Increase. *Geotherm. Resour. Counc. Trans.* **2018**, *42*.
45. Andal, J.M. Geojector: Extracting Geothermal Fluid from a Low-Pressure Geothermal Well. Master's Thesis, Reykjavik University, Reykjavik, Iceland, 2023.
46. Groves, C.J. Numerical Modeling of Supersonic Ejectors for Combining High- and Low-Pressure Steam for Geothermal Applications. Master's Thesis, Reykjavik University, Reykjavik, Iceland, 2024.
47. Guardia, X.; Juliusson, E.; Sveinsson, K.E.; Einarsson, H.; Saevarsdottir, G.; Tesfahunegn, Y.A.; Gudjonsdottir, M.S. Entraining low-pressure geothermal fluid for power generation using a supersonic ejector: A case study from Theistareykir power plant. *Geothermics* **2026**, *142*, 103811. [\[CrossRef\]](#)
48. Zikanov, O. *Essential Computational Fluid Dynamics*; John Wiley & Sons: Hoboken, NJ, USA, 2010.
49. Ansys Inc. *ANSYS Fluent*, Version 2024; Ansys Inc.: Canonsburg, PA, USA, 2024.
50. Ansys Inc. *Ansys Discovery User Manual*; Ansys Inc.: Canonsburg, PA, USA, 2025.

51. Ansys Inc. *Ansys Fluent User's Guide (Release 2025 R1)*; Ansys Inc.: Canonsburg, PA, USA, 2025.
52. Ansys Inc. *ANSYS Fluent Theory Guide*; Ansys Inc.: Canonsburg, PA, USA, 2024.
53. Bakhtar, F.; Barrett, J.R. Classical Nucleation Theory and Its Application to Condensing Steam Flow Calculations. *Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci.* **2005**, *219*, 1315–1333. [[CrossRef](#)]

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.