Two-Phase Flow Analysis and Design of Geothermal Energy Turbine

S Rane and L He
Department of Engineering Science, Southwell Laboratory, University of Oxford, Osney Mead, Oxford, OX2 0ES, United Kingdom.
Email: sham.rane@eng.ox.ac.uk

Abstract. A two-phase geothermal turbine generates mechanical energy by means of a total flow system. Literature references are available for experimental investigation of such flow machines but there is no validated numerical model for prediction of flow field and flashing for optimal design of the turbine’s flow channels. This paper presents a test reaction-turbine design study with two-phase flow using CFD. An Eulerian-Eulerian multiphase model with the Thermal Phase-Change criteria for two-phase heat, mass and momentum transfer has been used. Phase change validation has been conducted on published experimental data on a converging-diverging nozzle flow at various operating conditions. The stationary nozzle model was then extended to curved nozzle turbine and the results of flow and power has been validated with the data available in literature. Liquid water and vapour distribution due to flash evaporation was predicted by the analysis. Two-phase turbine model so established has been used further to re-design and optimise turbine nozzle by evaluating its cross section, area ratio, throat design, specific torque variation and curvature. Two new channel geometries, square and blended, were compared with the base circular channel and it was found that a blended channel offers on an average 6% higher specific power at 4623rpm. For the same rotor diameters and feed water flow rate, the power output could thus be improved by 4.5%. Isentropic efficiency in the range of 10 to 19% was estimated. Final design will be prototyped and tested in the laboratory for validation of design and deployment in eastern Africa.

1. Introduction
Geothermal hot brine deposits are a natural source of high temperature and high pressure fluid. Across the globe, techniques and systems to utilize the potential of geothermal wells are being explored. Output electrical energy or mechanical energy is found to be the preferred means of extracting the thermal energy contained in the geothermal well. At sites such as in Iceland, Africa, America, Italy, Australia several projects are under-way. As a part of the Combi-Gen consortium established in 2017, a thermodynamic system for combined power generation and pure water co-generation has been aimed for. This is due to the demands at sites in East Africa, Ethiopia, and Kenya where geothermal wells can be tapped but the local population is in greater demand for drinkable water, as is the power requirement. In the context of mechanical power generation from geothermal fluids, an impulse type steam turbine has been a classical approach. The systems employing binary fluids such as organic fluids have found use in low temperature heat sources due to their flammability at higher temperature. In high temperature source wells, the geothermal water is initially flashed and steam so produce is used to drive the turbine. The total-flow concept which directly uses the brine as the turbines working fluid is the most efficient means of energy conversion but suffers from the backdrop of heavy mechanical erosion of the turbine, nozzle and mineral deposition which leads to in-operation of the system in due course. Austin et al. [2]
have presented the Total flow concept for power recovery for application to the Salton Sea geothermal bed. Geothermal water was allowed to expand from a pressure of 2200 psia at a temperature of 300 °C to 400 psia. Thermal energy was converted to kinetic head by expansion through a converging-diverging nozzle. The jet was then used to drive an impulse turbine. Similar concepts can be found in literature that have used multi-phase nozzles. Elliot [6] has reported tests with water and nitrogen mixtures and with single stage and two stage impulse turbines. A special two-phase nozzle with mixer housing and liquid injection tube was designed to vary the mass ratio of liquid to vapour in the two-phase flow investigation. Abuaf et al. [1] conducted an extensive experimental study on two-phase flow in a converging-diverging nozzle. The test data is available at various operating conditions in the form of pressure measurements across several points along the length of the nozzle. Vapour void fraction measurements have also been documented in this report and the data is highly suitable for conducting a computational model validation. A closed loop system to heat, pressurize and control the water flow and condenser pressure and temperature was used in these experiments. More recently Date et al. [4] have presented experimental results of a curved nozzle total flow reaction turbine for low temperature feed water application. Patent of this reaction turbine design has been registered by Fabris in 1993.

Two-phase CFD modelling of the BNL nozzle [1] has been analysed using the Thermal Phase-Change model in Liao and Lucas [9, 10]. They have extensively reported the influence of various modelling parameters, closure models and grid dependency. Results have been reported in the form of pressure drop along the nozzle, vapour void fraction along the nozzle and across several transverse planes and compared with the BNL test data. A good match of the overall mass flow rate through the nozzle, pressure drop and average void fraction at the exit were reported. In their simulations, the liquid side heat transfer coefficient was formulated based on Jacob number and Peclet number while the vapour side heat transfer was formulated using two resistance approach across a spherical bubble surface whose diameter was specified using Bubble number density. Grid refinement reported in this study showed an optimum node count of 744,800 for the nozzle geometry and the numerical solver used was ANSYS CFX in steady state. Water data for both liquid and vapour phase was specified using the IAPWS-IF97 equations. SST k-Omega turbulence model was used. Several test points and operating conditions were validated in this study. Particularly, discrepancies in the distribution of radial void fraction were reported. This literature provides a foundation for the modelling approach used in the current study. In order to evaluate the radial void fraction discrepancies in detail, Janet et al. [9] studied the influence of nucleation model using a Bubble number transport equation instead of directly fixing a Bubble number density as was done by Liao and Lucas [10]. Small improvements in prediction were observed but were not satisfactorily validated with the BNL measurements. In addition to Bubble number transport formulation, Janet et al. [9] also considered the bubble breakup and coalescence in their model.

In this paper, the preliminary modelling of two-phase flow using ANSYS CFX solver has been presented first and further extended to analyse the flow regimes inside a test turbine. An Eulerian-Eulerian multiphase model has been used in the numerical solver. Two-phase heat, mass and momentum transfer has been modelled using the Thermal Phase-Change model. Phase change validation study has been conducted on published experimental data on a converging-diverging nozzle flow at various operating conditions (1, 9, 10). Satisfactory results of flash boiling of liquid water to vapour were achieved by the model and quantitative data was validated with literature. The model was then extended to test turbine presented in Date et al. [4, 5]. Turbine efficiency and performance, water liquid and vapour distribution and flash evaporation boundaries were predicted by the analysis. The CFD model so established has been used further to re-design and optimize the two-phase turbine for Combi-Gen project. The curved nozzle channel re-design is based on evaluation of specific torque variation, pressure distribution and a higher specific power output.

2. Two-Phase CFD Modelling

In the Eulerian-Eulerian approach of multiphase modelling, the primary and secondary phases are treated as interpenetrating continua. Phasic volume fraction $\alpha_q$ represents the volume of a computational
cell occupied by $q^{th}$ phase and the conservation transport equations of mass, momentum, energy and other scalars are satisfied by each phase individually.

**Conservation of mass:**

Continuity equation for the $q^{th}$ phase is

$$\frac{\partial}{\partial t} (\rho_q \bar{V}_q) + \nabla \cdot (\rho_q \bar{V}_q \bar{F}_q) = \sum_{p=1}^{n} (\dot{m}_{pq} - \dot{m}_{qp}) + S_q$$  \hspace{1cm} (1)

Here, $\bar{V}_q$ is the velocity of phase $q$ and has three Cartesian components. $\dot{m}_{pq}$ is the mass transfer term from $p^{th}$ phase to $q^{th}$ phase. $\dot{m}_{qp}$ is the mass transfer term from $q^{th}$ phase to $p^{th}$ phase. $S_q$ is any additional mass source for the phase $q$.

**Conservation of momentum:**

Momentum conservation for phase $q$ is

$$\frac{\partial}{\partial t} (\rho_q \bar{V}_q) + \nabla \cdot (\rho_q \bar{V}_q \bar{F}_q) = -\dot{p}_q + \alpha_q \bar{V}_q + \nabla \cdot (\alpha_q \rho_q \bar{V}_q \bar{F}_q) + \sum_{p=1}^{n} \dot{F}_{pq} + \dot{m}_{pq} \bar{V}_p - \dot{m}_{qp} \bar{V}_q + \sum_{p}^{n} \bar{F}_p$$  \hspace{1cm} (2)

Where, $\bar{F}_q$ is the $q^{th}$ phase stress-strain tensor with components that are function of shear and bulk viscosity. $\dot{p}_q$ is the pressure shared by all phases, $\mu_q$ and $\lambda_q$ are the shear and bulk viscosity of phase $q$. $\sum \dot{F}_{pq}$ represents all external body forces such as lift, virtual mass, turbulent dispersion etc. $\dot{F}_{pq}$ is the interaction force between phases such as drag and interphase momentum exchange. $\bar{V}_p$ and $\bar{V}_q$ are the interphase velocities.

**Conservation of energy:**

Enthalpy conservation equation for phase $q$ is

$$\frac{\partial}{\partial t} (\rho_q h_q) + \nabla \cdot (\rho_q h_q \bar{V}_q) = \frac{d \rho_q}{dt} + \nabla \cdot (\alpha_q \rho_q h_q \bar{V}_q) - \bar{q}_q + S_q + \sum_{p=1}^{n} (Q_{pq} + \dot{m}_{pq} h_p - \dot{m}_{qp} h_q)$$  \hspace{1cm} (3)

$$Q_{pq} = h_{pq} A_i (T_p - T_q)$$

Here, $h_q$ is the specific enthalpy of phase $q$. $\bar{q}_q$ is heat flux, $S_q$ is enthalpy source, $Q_{pq}$ is the intensity of heat exchange between phases $p$ and $q$. $h_{pq}$ is the interphase enthalpy i.e. vapour enthalpy in case of evaporation. $h_{pq}$ is the heat transfer coefficient at the phasic interface, $k_q$ is the thermal conductivity of phase $q$, $Nu_p$ is Nusselt number and $d_p$ is the vapour bubble diameter of phase $p$.

### 2.1. Thermal Phase-Change Model

Within the Eulerian multiphase framework, the Thermal Phase-Change model defines the mass transfer rates entirely based on the interphase heat transfer and overall heat balance at the phasic interface. At the water-liquid and water-vapour interface heat balance gives

$$Q_w + Q_v = 0$$

From the interface to water-liquid phase,

$$Q_w = h_{w} A_i (T_{sat} - T_w) - \dot{m}_{ww} \cdot H_{ws}$$  \hspace{1cm} (4)

From the interface to water-vapour phase,

$$Q_v = h_{v} A_i (T_{sat} - T_v) + \dot{m}_{vv} \cdot H_{vs}$$

Sub-script $s$ is selected depending on evaporation or condensation.
\[ \dot{h}_w A_i (T_{sat} - T_w) - \dot{m}_{wv} H_{ws} + \dot{h}_v A_i (T_{sat} - T_v) + \dot{m}_{wv} H_{vs} = 0 \]

\[ \dot{m}_{wv} = \frac{\dot{h}_w A_i (T_{sat} - T_w) + \dot{h}_v A_i (T_{sat} - T_v)}{H_{ws} - H_{vs}} \]

The interphase mass transfer in equations (1, 2 and 3) are obtained from equation (4). Equation (5) and (6) are used to select the enthalpy for source in equation (3).

If \( \dot{m}_{wv} > 0 \) (Boiling),

\[ H_{ws} = H_w(T_w) \]
\[ H_{vs} = H_v(T_{sat}) \]

If \( \dot{m}_{wv} < 0 \) (Condensation),

\[ H_{ws} = H_w(T_{sat}) \]
\[ H_{vs} = H_v(T_v) \]
\[ L = H_v(T_{sat}) - H_w(T_{sat}) \]

As seen from equation (4), heat transfer coefficients \( \dot{h}_w, \dot{h}_v \) and interfacial area \( A_i \) are required to be determined in the Thermal Phase-Change model. These inputs are prescribed by specifying the interface Nusselt numbers and then calibrating it such that vapour dryness fraction within a range of an isentropic process (expansion in the nozzle) is achieved.

3. BNL Nozzle Analysis

Abuaf et al. [1] conducted an extensive experimental study on two-phase flow in a converging-diverging nozzle. A closed loop system to heat, pressurize and control the water flow and condenser pressure and temperature was used in these experiments. This section describes the CFD model of the symmetric nozzle and application of Thermal Phase-Change model to estimate the flash boiling and vapour void fraction distribution in the test section. Two-phase CFD modelling of the BNL nozzle has been analysed using the Thermal Phase-Change model of ANSYS CFX solver in [9, 10]. Same solver settings, water-liquid and vapour fluid properties using IAPWS-IF97 database and numerical scheme has been applied in the current study in order to achieve consistent comparison. Figure 1 shows the hexahedral computational mesh used for the 3D nozzle domain. Grid is refined closer to the throat region.

![Figure 1. Circular converging-diverging nozzle and 3D computational mesh of BNL tests.](image1)

![Figure 2. Axial pressure and vapour void fraction for Run309.](image2)
Inlet boundary is defined using measured pressure, temperature and saturated liquid condition. Outlet boundary is defined using measured pressure. Pressure and temperature initial conditions are specified using a geometric function and water saturation equation according to IAPWS-IF97 coefficients. The initialization control functions help in stability of the steady state solver. About 15,000 iterations of the solver are computed to achieve residual imbalance below 1e-04 for all the transport equations. Three grid refinements are computed to achieve mass balance results within 0.5% accuracy.

3.1. Results and Validation

Results from the BNL nozzle experiments [1], the CFD results presented in literature [10] and current computational model results have been compared here in order to validate the two-phase model. Results of pressure and vapour void fraction variation along the nozzle length have been quantitatively compared. Phase velocity and temperature variation in the domain has also been presented. Experimental run BNL309 and BNL296 have been computed in current study since the operating conditions of these tests are in the range of the test turbine operating condition under consideration in the Combi-Gen project. Table 1 presents the grid independency results evaluated for BNL309. Mesh sizes and boundary layer statistics have been reported here. As the grid is refined from Mesh 1 to Mesh 3, the intake liquid mass flow rate through the nozzle tends to increase.

| Mesh | Node Count | First layer | Growth Rate | y⁺ | Mass flow | Mass Imbalance |
|------|-------------|-------------|-------------|-----|-----------|----------------|
| 1    | 264776      | 20          | 4           | 37.45 | 8.3699 | 0.99          |
| 2    | 322299      | 20          | 5           | 38.35 | 8.3813 | 0.59          |
| 3    | 637120      | 10          | 10          | 38.22 | 8.3928 | 0.48          |

Mass imbalance is calculated between the intake and exit planes of the nozzle and it is seen that from Mesh 1 to Mesh 3 the imbalance reduces from ~1% to less than ~0.5% which was considered to be an acceptable grid refinement level. Table 2 presents the operating conditions and results of the nozzle flow for the two test sets evaluated in the current study. In comparison to the measurements (m_exp), the CFD results reported by Liao and the current computations are under estimating by about 5%. Both the CFD models of Liao (m_Liao) and current study (m_ThPC) are closely predicting the flow at both the operating conditions.

| Run No | Pin  | Pout | Tin  | Psat | m_exp | m_Liao | m_ThPC |
|--------|------|------|------|------|-------|--------|--------|
| BNL309 | 555.9| 402.5| 422.25| 464.8| 8.79  | 8.4    | 8.38   |
| BNL296 | 764.9| 432.6| 421.95| 461  | 13.1  | 12.4   | 12.63  |

Figure 2 is a plot of pressure variation and water vapour void fraction distribution along the centerline of the nozzle for BNL309. The nozzle inlet pressure is 555.9 kPa and saturation temperature is 422.25 ºK. Pressure steadily drops from 555.9 kPa to about 380 kPa in the converging section of the nozzle with minimum pressure occurring at the throat. Downstream of the throat the pressure recovers to about 402 kPa and then remains close to this magnitude in the diverging section of the nozzle. The nozzle exit and condenser pressure in this test was 402.5 kPa. Pressure variation follows close to that of measurements and the CFD results reported by Liao. In the converging section of the nozzle water remains in liquid phase. At the throat the flash boiling phenomenon initiates and in the diverging section of the nozzle phase change from liquid to vapour continues with vapour volume fraction increasing to about 75% at the nozzle exit. Vapour void fraction follows close to that of measurements and CFD results reported by Liao. Similar validation of pressure and vapour void fraction was achieved in case of BNL296 operating conditions.
Figure 3. Contours of liquid and vapour volume fraction.

Figure 3 is a contour plot of water liquid and vapour volume fraction distribution in the longitudinal plane of the nozzle. Radial variation of the vapour void fraction is observed. At the throat flashing initiates with high vapour concentration at the nozzle centerline and gradually reducing radially outwards. In the diverging section the vapour void fraction continues to increase to a value of 75% at the nozzle exit. Figure 4 is a comparison of local saturation temperature ($T_{\text{satCEL}}$) according to IAPWS-IF97 equation, local water liquid temperature and local water vapour temperature in the longitudinal plane of the nozzle. Liquid at inlet is subcooled by about 6 degree and hence in the converging section of the nozzle there is no flashing of the liquid. Just upstream of the throat when flashing initiates due to fall in saturation temperature, liquid temperature begins to drop. In the diverging section of the nozzle liquid temperature continues to drop and the exit temperature is about 418 ºK. Results presented in Table 2 and the local pressure, temperature and vapour void fraction distribution thus validates the two-phase formulation of the Thermal Phase-Change CFD model. Further details of the analysis have been reported in Rane et al. [11, 12].

4. Two-Phase Reaction Turbine Analysis

Date et al. [4] have presented experimental performance of a curved nozzle two-phase reaction turbine. Their curved nozzle design was based on the curvature calculation procedure described in Fabris [7] with variable circular cross section shown in Figure 5 such that pressure drop along the length of the diverging section is assumed to be linear and the change in relative velocity of the fluid along the length of the nozzle is also assumed to be linear.

Figure 5. Two-Phase Fabris turbine design [7] and the CDF model of test turbine [4, 11, 12].

Two-phases tend to separate within the curved nozzle due to the large lateral acceleration and by designing the nozzle curvature using these assumptions, the lateral components of Coriolis acceleration,
centripetal acceleration relative to motion and centripetal acceleration with respect to turbine axis can be balanced. Date et al. [4] have reported test results from two operating feed water temperatures, for first test the average feed water temperature was maintained around 97 °C under local atmospheric pressure, and for the second test the average feed water temperature was maintained around 117 °C at 400 kPa pressure. For both the tests the initial condenser pressure was maintained around 6 kPa. The maximum power output of the turbine was estimated to be around 1330 W with an isentropic efficiency of around 25%. The turbine operation was not continuous due to limits of the condenser to handle higher mass flow rates which eventually raised the 6 kPa back pressure with time. The test lasted for approximately 150 seconds of acceleration and further 100 seconds of deceleration of the turbine. This test turbine CFD model has been described in detail in Rane et al. [11, 12]. In this section the results that are important for re-design of the turbine nozzle have been discussed.

4.1. Base Turbine design

Thermal Phase-change, two phase model established in the BNL nozzle study is also applied in case of the test turbine analysis. There are two main differences as compared to the BNL nozzle model. a. Turbine has a rotor. A moving reference frame approach is used to apply the rotational speed to the nozzles. b. The turbine has two symmetrically positioned curved nozzles. In order to reduce CFD domain, a 180 degree sector of the rotor is modelled with periodic boundaries. Inlet to the nozzle is defined by the feed water pressure of 400 kpa and temperature of 117 °C. The exit of the turbine is open to a flash tank connected to condenser coil which during the start of the experiments is vacuum conditioned to 6 kPa pressure. The CFD model uses a smaller flash tank domain and all boundaries are set at measured pressure and saturated temperature for backflow conditions. CFD calculations were performed over a range of turbine operating speed from 1561 to 4623 rpm. This was the speed at which highest power output has been reported in [4].

![Figure 6](image1.png)  
**Figure 6.** Pressure distribution at 4623rpm on the curved nozzle scaled to a maximum of 6kPa.

![Figure 7](image2.png)  
**Figure 7.** Evolution of pressure and vapour mass fraction along the axis of the nozzle.

Figure 6 presents the distribution of pressure on the nozzle surface and in a mid-plane close to the throat of the curved nozzle. The contour plot is scaled between 3 to 6 kPa so that pressure variation downstream of the throat can be highlighted. On the downstream of the throat a severe drop in local pressure is observed to about 2.93 kPa. Only a small difference in pressure is observed between the leading and trailing surface of the nozzle thus indicating a low torque from the radial pressure gradient. Figure 7 presents the evolution of pressure along the axis of the curved nozzle in turbine midplane. The inlet boundary to the nozzle is specified as 400 kPa. From inlet to the nozzle throat a steady increase in the fluid pressure is observed. Even though this is a converging section of the nozzle, the pressure rises due to rotational effect of the turbine. The peak pressure upstream of the nozzle goes up to 670 kPa and then severely drops down close to 3 kPa downstream of the nozzle throat. Using this information, the nozzle section could be redesigned in order to avoid the pressure rise which could improve the specific power output. Figure 7 also presents the variation of vapour mass fraction along the axis. From inlet to the
throat, feed water remains in liquid state with no phase transition. At the throat the inception of water vapour due to flash boiling is observed. From the throat region and in the diverging section of the nozzle there is steady increase in vapour fraction. At nozzle exit the mass fraction is about 10%. A vapour quality of about 12.8% has been reported in the measurements and the CFD model prediction is close to the results. Figure 8 presents the distribution of vapour mass fraction in mid-plane, exit-plane and close to the throat. In the converging section the pressure is high, liquid is subcooled and hence flashing does not occur. Just on the downstream of the throat a severe drop in local pressure is observed to about 2.93 kPa and this initiates the flash boiling as local saturation temperature is lower than the liquid temperature. The vapour distribution in this region is observed only close to the nozzle wall and not in the core of the flow. Further downstream due to rotation of the nozzle and the curvature, the flash boiling continues on the trailing surface of the nozzle.

![Figure 8. Vapour mass fraction distribution at 4623rpm on the curved nozzle surface.](image8)

![Figure 9. Cumulative Torque variation at 4623rpm on the curved nozzle.](image9)

Heavier liquid phase is present on the high pressure leading surface. This nature of distribution of vapour mass fraction and flashing regime is highly dependent on the turbine rotational speed. Figure 9 presents the distribution of cumulative torque along the nozzle channel. Cumulative torque at any specific channel length indicates the net torque output if the nozzle was clipped to that length. It is calculated by first extracting the specific torque variation along the nozzle channel. From inlet to about 30 degree, the rotor torque is positive and increasing. After this the nozzle curvature and surface area variation is such that the specific torque becomes negative. This results into a decrease in the net torque, up to the throat section. After the throat section the specific torque again becomes positive and the cumulative torque starts to increase up to the nozzle exit. The base turbine design has a tangential exit section and pressure acting on the trailing surface results into a negative specific torque. This further reduces the cumulative torque and net torque output from this turbine. The nature of torque variation presented in Figure 9 is observed at all speeds of the turbine.

![Figure 10. Comparison of feed water flow rate between measurements and CFD model.](image10)

![Figure 11. Comparison of turbine power between measurements and CFD model.](image11)
Figure 10 presents the variation of turbine feed water flow with speed and the results have been compared with measurements [4]. It is seen that the CFD model is accurately predicting the nozzle flow and the deviation is in the range of 0.5 – 7.5%. The measurement error band was reported as 7%. This data could be further used to carry out design changes in the turbine to improve the specific power. Figure 11 presents the variation of turbine power with speed and the results have been compared with measurements. It is seen that the CFD model is over predicting the nozzle torque and hence the turbine power at operating speeds below 4000 rpm in the range of 30-55%. At 4623 rpm turbine power deviation in the range of 4.5% is observed. The power estimate from CFD model could be used for a comparative study between various nozzle channel designs close to this operating speed. Further CFD solution parameters and geometrical parameters are required to be analysed in order to understand the causes for this rotor torque deviation.

4.2. Turbine Re-design
Based on the analysis of the circular cross section channel, one of the important re-design factor was incorporation of a distinct leading and trailing surface into the channel geometry. This would result into an increase in the specific torque variation along the nozzle. One method of introducing distinct leading and trailing surface is by using a square cross section. The disadvantage with square cross section is that it increases the flow area by 21.4% for the same dimension as that of the diameter of the corresponding circular cross section. Hence, two versions of the channel section were proposed as shown in Figure 12. Figure 12b is a square cross section nozzle channel with same area variation function as that of the base CR – circular cross section channel. Figure 12c is a blended nozzle channel where up to the throat section the profile is circular and downstream of the throat it gradually blends to a square profile up to the nozzle exit. Area variation function is maintained same as that of the CR channel except in the short nozzle length where the section blends from circular to square.

Figure 12. Channel geometries: a) Base Design – CR, b) Square – SQ and c) Blended - BSQ
The two re-designed nozzle channels were analysed with the same numerical setup of the two-phase turbine model. The computational grid density was equivalent and results of feed water flow rate, turbine power and specific power output were evaluated over an operating speed range of 1561 to 4623 rpm.

Figure 13. Comparison of turbine performance with CR, SQ and BSQ channel designs.
Figure 13 shows a comparison of the performance of the three turbines. In case of feed water flow rate, with SQ design, since the area increased by 21.4% over the entire channel, the flow increased by 20.7% over the full speed range. With BSQ design, there was no increase in the feed water flow rate as compared to CR channel design (<1%). This showed that the throat area has significant control over the flow through the turbine. Figure 13b compares the power output from the three geometries. In
comparison to the CR channel, the SQ channel which has a distinct leading and trailing surface resulted into a significant 23% higher power output over the speed range. This increase in power was also associated with the 20% higher flow through this design and cannot be entirely attributed to the change in geometry. In comparison the BSQ channel estimated a 4.5% increase in the power output for the same feed water flow. The turbine specific power is compared in Figure 13c. The SQ channel with higher flow and power showed an equivalent specific power as compared to the CR channel, thus with no benefit. While the BSQ channel design showed an improvement of 4.5% in specific power as compared to the base CR channel. At 4623rpm, the specific power of BSQ channel was 6% higher than the CR channel.

5. Conclusions
A two phase CFD model for calculation of flash boiling flows in nozzle and turbines was applied using the Thermal Phase-Change formulation in ANSYS CFX solver. The BNL nozzle experimental data and other CFD literature results on the same test case were used in the model setup to validate the formulation. The two-phase CFD model so established has been adapted with moving reference frame, periodic boundary conditions, etc., to be used for the design and development of reaction turbine.

- Base turbine flow at various operating speed was well estimated, within 0.5 – 7.5% deviation from measurements. Nozzle torque and hence turbine power estimate was within 4.5% at 4623rpm. Turbine performance, liquid and vapour distribution and flash boiling boundaries were predicted by the analysis. Nozzle exit vapour dryness fraction was estimated in the range of 11 – 13%.

- Two new channel geometries, square and blended, were compared with the base circular channel and it was found that a blended channel offers on an average 6% higher specific power at 4623rpm. For the same rotor diameters and feed water flow, the power output could be improved by 4.5%.

- Isentropic efficiency of the base turbine varied from 5% at 1561rpm to a maximum of 19% at 4623rpm. CFD model estimated the Isentropic efficiency in the range of 10 to 19% due to the power over-estimate at lower speed.

The CFD model so established will be used further to design and optimize the two-phase turbine exit, nozzle curvature, throat area and location.

Acknowledgements
The research is sponsored by UK Engineering and Physical Sciences Research Council. Authors would also like to thank Dr Abhijit Date, RMIT, Australia, for providing the reference turbine test data for development and validation of the CFD model.

References
[1] Abuaa N, Wu B J C, Zimmer G A and Saha P 1981 A study of Nonequilibrium flashing of water in a converging-diverging nozzle. Brookhaven National Laboratory BNL-NUREG/CR-1864. Volume 1.
[2] Austin A L, Higgins G H and Howard J H 1973 The total flow concept for recovery of energy from geothermal hot brine deposits. Lawrence Livermore Laboratory UCRL-51366.
[3] ANSYS CFX User Guide. 2019.
[4] Date A, Vahaji S, Andrews J and Akbarzadeh A 2015 Experimental performance of a rotating two-phase reaction turbine. Applied Thermal Engineering 76 475-483.
[5] Date A, Khaghani A, Andrews J and Akbarzadeh A 2015 Performance of a rotating two-phase turbine for combined power generation and desalination. Applied Thermal Engineering 76 9-17.
[6] Elliot D G 1982 Theory and Tests of Two-phase Turbines, Jet Propulsion Laboratory 81-105. DOE/ER-10614-1.
[7] Fabris G 1993 Two-phase Reaction Turbine, United States Patent 5,236,349.
[8] Janet J P, Liao Y and Lucas D 2015 Heterogeneous nucleation in CFD simulation of flashing flows in converging-diverging nozzles. International Journal of Multiphase Flow 74 106–117.
[9] Lee W H 1979 A Pressure Iteration Scheme for Two-Phase Modelling. Technical Report LA-UR-79-975. Los Alamos Scientific Laboratory, Los Alamos, New Mexico.
[10] Liao Y and Lucas D 2015 3D CFD simulation of flashing flows in a converging-diverging nozzle. Nuclear Engineering and Design 292 149–163.
[11] Rane S, He L and Ma H 2019 CFD modelling and analysis of Two-Phase Geothermal Energy Turbine in Project CombiGen. Int Conf on Innovative Applied Energy, Oxford. ISBN: 978-1-912532-05-6. Paper 257.
[12] Rane S and He L 2019 Modelling of flash boiling in two phase geothermal turbine. Proceedings of the 16th UK Heat Transfer Conference, Nottingham, Paper 005.