Turbulent contribution to heat loss in cavity receivers
William R. Logie, Ehsan Abbasi-Shavazi, Graham Hughes, and John D. Pye

Citation: AIP Conference Proceedings 1850, 050003 (2017); doi: 10.1063/1.4984407
View online: http://dx.doi.org/10.1063/1.4984407
View Table of Contents: http://aip.scitation.org/toc/apc/1850/1
Published by the American Institute of Physics

Articles you may be interested in
Sensitivity analysis of heliostat aiming strategies and receiver size on annual thermal production of a molten salt external receiver
AIP Conference Proceedings 1850, 030047 (2017); 10.1063/1.4984390

Review and future perspective of central receiver design and performance
AIP Conference Proceedings 1850, 030052 (2017); 10.1063/1.4984395

Analysis and improvement of the cavity structure of steam receiver of 1MWe solar tower power plant
AIP Conference Proceedings 1850, 030051 (2017); 10.1063/1.4984394

Solar tower enhanced natural draft dry cooling tower
AIP Conference Proceedings 1850, 030050 (2017); 10.1063/1.4984393

Experimental investigations on radiative properties of two kinds of open-cell porous ceramic materials
AIP Conference Proceedings 1850, 030049 (2017); 10.1063/1.4984392

Preliminary analysis of the PreFlexMS molten salt once-through steam generator dynamics and control strategy
AIP Conference Proceedings 1850, 030048 (2017); 10.1063/1.4984391
Turbulent Contribution to Heat Loss in Cavity Receivers

William R. Logie¹,a), Ehsan Abbasi-Shavazi¹, Graham Hughes² and John D. Pye¹,b)

¹Solar Thermal Group, Research School of Engineering, The Australian National University, Canberra
²Fluid Mechanics Section, Department of Civil and Environment Engineering, Imperial College, London

a)Corresponding author: will.logie@anu.edu.au
b)Lead investigator: john.pye@anu.edu.au

Abstract. For the prediction of convective heat loss from solar concentrating receiver cavities a number of empirical correlations exist. Geometry and the inclination angle determine the degree to which natural convection can infiltrate the cavity and remove stably stratified hot air out through the aperture. This makes the task of defining characteristic lengths for such Nusselt correlations difficult, neither does their use offer insight as to how one might reduce heat loss through the use of baffles, air curtains or small aperture-to-cavity-area ratios. Computational Fluid Dynamics (CFD) can assist in the design of better cavity receivers as long as the rules upon which it rests are respected. This paper is an exploration of the need for turbulence modelling in cavity receivers using some common linear eddy viscosity closure schemes. Good agreement was obtained with the CFD software OpenFOAM® 3.0.1 for a deep cavity aperture but it under-predicted a shallow cavity. The experiments used for validation were in the Grashof region $Gr \approx 10^6$, well below the region for transition to turbulence between $10^8 < Gr < 10^9$.

INTRODUCTION

There are a number of works in literature providing numerical and empirical means for predicting convective heat loss from cavities [1]. Previous laboratory work performed in the Geophysical Fluid Dynamics Laboratory (GFDL) at the Australian National University (ANU) has tackled the problem both experimentally and numerically [2] and the good agreement served as validation for larger explorations and the development of a correlation to be applied at the scale of common solar thermal cavity receivers [3]. While this work has served the wider community for validation of similar explorations there remains some uncertainty about the circumstances of agreement; questions surrounding scalability, the occurrence of turbulence inside the cavity and its impact if any on heat loss remain.

Paitoonsurikarn [3] illustrated that the numerical prediction of convective heat loss from Taumoefolau’s [2] model receiver was relatively low when compared with a broad selection of experimentally derived correlations but when scaled up to larger cavities an acceptable agreement was shown [1]. In addition he found no difference in numerical heat loss results for the model receiver using the Spalart and Allmaras model [4] or no turbulence closure at all, although Taumoefolau observed the appearance of turbulence exiting the aperture of the model receiver by way of ‘Synthetic Schlieren’ imaging [5]. More recently Yuan et al. [6] found good numerical agreement with Taumoefolau’s experiments but significant discrepancy with a large cubical cavity experiment, and performed an exhaustive exploration of turbulence closure schemes and model assumptions but was unable to account for the difference.

To broaden the experimental base for comparison, the ANU ran a further test campaign in the GFDL with two newly constructed model receivers of similar dimension [7]; one cavity had an aspect ratio of unity (AR1) and the other was twice as deep as it was wide (AR2). As illustrated in Figure 1, the results have proven to differ in the share of heat loss from convection, but also in the effect of cavity inclination and the wall temperature distribution (non-isothermal) [8]. Interestingly, one of the sources in Figure 1, Jilte [9], used the work of Taumoefolau for numerical validation and went on to simulate a plethora of larger cavity shapes and develop the correlation plotted. One reason Jilte’s correlation differs from that of Taumoefolau and Paitoonsurikarn could be the explicit use of the standard $k-\epsilon$ [10, 11] model which, in addition to modelling the generation of turbulence kinetic energy and dissipation due to the mean velocity gradients (shear), also has terms for the generation and dissipation of turbulence kinetic energy due to buoyancy (unstable density gradient).
The aim of this paper is to evaluate the standard Reynolds Averaging Navier-Stokes (RANS) eddy-viscosity turbulence closure schemes in pure natural convection and in the presence of stratification inside the cavity. The two-equation standard $k$-$\epsilon$ (with buoyancy terms) and Menter’s Shear Stress Transport (SST) $k$-$\omega$ [12] models and the one-equation model of Spalart and Allmaras have been chosen due to their common adoption in practical engineering applications. Solutions for Abbasi-Shavazi et al. [7] are sought on two grids with different orders of discretisation accuracy. The first is more representative of a practical engineering approach using an unstructured mesh including the surrounding field up to a spherical atmospheric boundary and second-order upwind spatial discretisation, while the second uses a structured mesh only resolving the cylindrical cavity to the aperture where the same atmospheric boundary condition is applied and solutions are discretised with second-order central differencing. While this approach introduces modelling error (surrounding field information is discarded), a more isolated and highly resolved understanding of what these turbulence models are doing inside the cavity sheds light on their contribution to heat loss.

**Linear Eddy-Viscosity Concept**

It is mainly the large-scale turbulent motion which transports momentum and heat. The large eddies interact with the mean flow thereby extracting kinetic energy from the mean motion. The rate at which this occurs is determined by the large-scale motion, which then interacts with the mean flow and through ‘vortex stretching’ passes the energy on to smaller scales to be dissipated, a process called energy cascade. Molecular (shear) viscosity determines the scale at which dissipation occurs. At high Reynolds numbers where local isotropy prevails, the rate of dissipation is equal to the molecular viscosity times the fluctuating vorticity.

While modern computer power has enabled the Direct Numerical Simulation (DNS) of an ever increasing set of problems, contemporary Computational Fluid Dynamics (CFD) software still relies heavily on the Reynolds-averaged Navier-Stokes (RANS) equations for the timely calculation of turbulent problems. This saving in time comes from the decomposition of instantaneous velocity, pressure and scalar quantities such as temperature into their time-averaged and fluctuating quantities. The time over which this averaging is performed is large enough to make the calculated flow time-invariant. This introduces an apparent stress term to the RANS equations, the so-called ‘Reynolds stress’, which looks like a symmetrical stress tensor but seeks to correlate momentum arising from fluctuating velocities to the mean flow. It has shear components representing the interaction between fluctuating velocity directions and principal components representing their impact on pressure. Even though it looks like a stress, the largely unknown physics behind it make it very different from viscous stress, which can be related directly to the other flow properties by constitutive equations. The Reynolds stress arises from the flow itself, and the scales giving rise to it are generally the ones we are interested in. This awareness has led to the view that the Reynolds stresses can be modelled by a linear
The constitutive relationship between the so-called eddy or turbulent viscosity $\mu_t$ and the mean rate-of-strain in the fluid. This is then fed back into the mean momentum equations by replacing molecular (shear) viscosity with an effective summation of the two $\mu_{\text{eff}} = \mu + \mu_t$, and similarly to the energy equation by way of heat transfer eddy-diffusivity:

$$\alpha_t = \frac{1}{\rho} \frac{\mu_t}{Pr_t} \quad \alpha_{\text{eff}} = \alpha + \alpha_t,$$

with the turbulent Prandtl number $Pr_t$ representing the non-dimensional ratio between them.

The turbulent viscosity is obtained in a different manner depending on the level of modelling employed. The most versatile and easiest to calculate of the linear eddy-viscosity concepts involves solving additional transport equations for turbulent kinetic energy $k$ and a dissipation variable $\epsilon$ or $\omega$ such that:

$$\mu_t = C_\mu \frac{k^2}{\epsilon} \quad \mu_t = C_\mu \frac{k}{\omega}.$$  \hspace{1cm} (2)

These transport equations contain the linear constitutive relationship for which a number of empirically determined model constants must be introduced (e.g. $C_\mu = 0.09$). Like the Navier-Stokes momentum equations they contain terms for their rate of change, convection and diffusion, but also terms representing the rate of their generation and destruction. The rate $P_k$ at which turbulent kinetic energy $k$ is extracted from mean flow is determined by turbulent viscosity $\mu_t$ and the mean rate-of-strain tensor:

$$P_k = \mu_t \left( \frac{\partial U_i}{\partial x_j} \right)^2.$$  \hspace{1cm} (3)

In most flows only the impact of fluctuating velocities on turbulent motion need be considered. Scalar quantities such as temperature or species are treated as passive quantities whose impact on mean flow is negligible. When buoyancy is present, however, there is an additional fluctuating quantity $\rho$ in the momentum equation which in turn gives us an additional production term in the turbulent kinetic energy equation [13]. This additional exchange of energy between the mean flow and turbulent kinetic energy $G_k$ can go in both directions. Buoyancy reduces the turbulent kinetic energy $k$ in stably stratified flows and increases it when the stratification is unstable according to Eq. 4 - likewise any horizontal density gradient is always unstable:

$$G_k = g_i \left( \frac{\mu_t}{Pr_t} \frac{\partial \rho}{\partial x_i} \right).$$  \hspace{1cm} (4)

where $g$ is the gravity vector.

While this additional source term has been available in the standard $k$-$\epsilon$ model since Fluent® 6.3 and probably earlier, it was only recently added to the standard set of turbulence models in OpenFOAM® 3.0.1. Aside from the analogous Generalised Gradient Diffusion Hypothesis (GGDH) sometimes found within the Reynolds Stress Transport (RST) set of turbulence models, no other standard RANS turbulence model considers buoyancy-affected flow as such, although the exact circumstances invoking its use are not well documented.

**METHOD**

Given the significant temperature differences to be modelled (>> 15 K [14]) and very low speeds expected (Ma<< 0.3) the incompressible perfect gas law is used for buoyancy in the OpenFOAM® 3.0.1 buoyantSimpleFoam solver as per Eq. 5:

$$\rho = \frac{p_0}{R_{\text{air}} T},$$  \hspace{1cm} (5)

where $p_0$ is constant (scalar) atmospheric pressure, $R_{\text{air}}$ is the specific gas constant for air and $T$ is the temperature field with which thermo-physical properties are calculated by way of the Sutherland law for viscosity $\mu$ and JANAF tables for specific heat $c_p$.

Two meshes having symmetry about the plane defined by inclination angle were generated using native OpenFOAM® 3.0.1 utilities. The ‘engineering approach’ mesh was made with snappyHexMesh, a boundary-fitting and surface-aligned-layer-insertion hexahedra-dominant mesh generator which can handle the more complex combination of the cylindrical cavity geometry blended to the far field spherical atmospheric boundary condition. Given the
simple geometry of a cylindrical cavity a finely resolved structured mesh was generated for the comparison of turbulence models inside the cavity with blockMesh, which decomposes the domain into hexahedral blocks with edges defined by straight lines, arcs or splines. Illustrations of the two meshes can be seen in Figure 2.

![Unstructured mesh with surround field and atmospheric boundary](image1.png)

(b) Closeup of snappyHexMesh showing dominance of hexahedra

(c) View of cylindrical cavity blockMesh without surround field

**FIGURE 2.** Symmetrical meshes generated for the AR1 (aspect ratio of 1) experiments from Abbasi-Shavazi et. al [7] with native OpenFOAM® 3.0.1 utilities for (a)(b) the practical engineering approach and (c) the turbulence comparative study - illustrative plume shown for 0° cavity inclination

In both cases mesh resolution was refined at wall boundary layers for low-Reynolds-number (low-Re) near-wall treatment ($y^+ \leq 1$) and within the viscous sub-layer region of wall boundary layer a mesh growth ratio of 1.3 was enforced, while the growth of cell size from the outer boundary layer into far field varied for each mesh: for the unstructured mesh the cell size was doubled at each 5th cell interval away from the wall whereas for the structured mesh no such doubling was performed. The number of cells ranged from $1.71 \times 10^6$ to $1.83 \times 10^6$ in the unstructured surround-field mesh and from $1.41 \times 10^6$ to $1.89 \times 10^6$ in the structured cavity-only mesh, depending on which aspect ratio was being captured (deeper cavity required more cells). No Grid Convergence Index [15] was calculated however a monotonic convergence (no oscillation) of the cavity wall heat flux was observed from the coarse unstructured mesh to the fine structured mesh at all inclination angles.

Modelling the atmospheric boundary condition presents a challenge as one wants the buoyant air to leave the domain without hindrance, and likewise the air entering to replace it should do so quiescently. In Figure 2(a) the spherical outer boundary patch and in Figure 2(c) the left front semicircle patch was modelled in such a way as to allow the buoyant hot air to leave the cavity against a standard atmospheric pressure $p_0$ and in turn allow quiescent ambient air to enter. This was achieved in OpenFOAM® 3.0.1 by applying the prghTotalPressure boundary condition to the so-called ‘pseudo-hydrostatic’ pressure field $p_{prgh}$:

$$p_{total} = p_0 + \frac{p_0}{R_{air}} T_0 g h$$

(6)

$$p = p_{total} - 0.5 \rho |\mathbf{U}|^2$$

(7)

$$p_{prgh} = p - \rho g h$$

(8)

where $gh$ is the dot-product of the gravity and position vector, $p$ is the static pressure field\textsuperscript{1} and $0.5 \rho |\mathbf{U}|^2$ is dynamic pressure according to Bernoulli’s principle. Velocity $\mathbf{U}$ at the boundary is then calculated from pressure with the pressureInletOutletVelocity boundary condition. A summary of boundary conditions are outlined in Table 1 of the Appendix whereby it must be noted that the laws of the wall implemented for $v_\tau$, $\alpha_t$ and $\omega$ are valid for either low-Re or high-Re treatment and those implemented for $k$ and $\epsilon$ are for explicit low-Re treatment.

The model receiver casing and front rim was modelled adiabatically (zero-gradient temperature) and fixed-value temperatures were applied to the inside cavity wall with a linear spatial interpolation between measurements using the

\textsuperscript{1}It is worth noting that initialising simulations with a static pressure including incompressible piezometric head improves chances of convergence significantly. This was performed with the funkySetFields and funkySetBoundaryField utilities found in the add-on package swak4Foam (Swiss Army Knife for FOAM).
griddata tool in SciPy\(^2\). Steady-state solutions were sought using the SIMPLE algorithm \([16]\) and:

- preconditioned conjugate gradient (PCG) linear solver for pseudo-hydrostatic pressure,
- generalised geometric-algebraic multi-grid (GAMG) linear solver for momentum, energy and turbulence fields,
- relaxation factors 0.7 for pseudo-hydrostatic pressure and energy, and 0.3 for momentum and turbulence, and
- convergence criteria \(1 \times 10^{-3}\) for pressure, \(1 \times 10^{-4}\) for momentum and \(1 \times 10^{-5}\) for energy.

**RESULTS**

In the observation of heat loss from cavities it is common to talk of two zones existing inside the cavity depending on geometry and inclination angle. If the aperture for either reason is lower than a portion of the cavity then a pocket of two to three times less dense air develops inside this region with a degree of stratification, often referred to as the stagnant zone. Below that is referred to as the convective zone and the more or less horizontal layer between is the shear layer. Dense air is drawn through the lower portion of the aperture into the cavity until it encounters the shear zone where it rolls, mixing with the stratified air above before flowing out of the upper portion of the aperture and forming a plume. The character of the roll is largely determined by the shape of the cavity. In the case of the cylindrical cavity this roll follows the curvature of the cylinder wall. Due to the stabilising effects of large density gradients (thermocline) there is no counter-rolling eddy in the stagnant zone \([17]\).

The convective heat loss numbers for solutions on the unstructured mesh with surround field using the standard \(k-\varepsilon\) and SST \(k-\omega\) turbulence closures\(^3\) are plotted at seven discrete inclination angles in Figure 3 alongside results from Abbasi-Shavazi et al. \([7]\). In the case of the AR1 cavity agreement is well outside experimental error for both turbulence models. For high inclination angles at and above \(45^\circ\) the choice of turbulence closure does not impact heat loss and the difference equates to \(\approx 25\) W, and for inclination angles below \(45^\circ\) this constant difference persists for the \(k-\omega\) model but not the \(k-\varepsilon\). Buoyancy-affected turbulence increases the convective heat loss with the \(k-\varepsilon\) model to within \(\approx 8\) W of experiment at \(0^\circ\) inclination. Numerical estimation of heat loss for the AR2 cavity is generally within experimental error. Again there is a constant offset between the \(k-\omega\) model and experiment and a difference in estimation between turbulence schemes at low inclination angles, but this time it results in over-estimation by the \(k-\varepsilon\) model within error.

![Convective heat loss vs Inclination angle](image)

(a) AR1: agreement is well outside experimental error and \(k-\varepsilon\) estimates higher convective heat loss due to buoyancy generation terms in the generation of turbulence

(b) AR2: agreement is generally inside experimental error with \(k-\omega\) consistently under-predicting experiments and \(k-\varepsilon\) over-predicting at low inclination angles due to buoyancy

**FIGURE 3.** Comparison of numerical solutions using standard \(k-\varepsilon\) and SST \(k-\omega\) turbulence models on the surround-field unstructured mesh with experimental results from Abbasi-Shavazi et al. \([7]\)

---

\(^2\)http://docs.scipy.org/doc/scipy/reference/generated/scipy.interpolate.griddata.html

\(^3\)For the ‘practical engineering approach’ the standard \(k-\varepsilon\) was chosen for its wide applicability and buoyancy source terms and the SST \(k-\omega\) was chosen for its popularity and merit gained in complex three-dimensional problems.
Observing the full set of chosen turbulence closure schemes and a solution without turbulence closure (laminar) on the structured cavity-only mesh in Figure 4, the numerical solutions appear not to change significantly. This in itself is an interesting observation which would suggest that it is not necessary to include the surround field if all it is resolving is quiescent fluid flow. At 0° inclination the convective heat loss predicted by both meshes is nearly identical but as inclination increases the unstructured mesh predicts numbers lower than the structured mesh. This is due to the diminishing size of the convective zone and lower resolution of the unstructured mesh; an artefact also observed by the higher prediction using the $k-\epsilon$ at inclination angles 45° and 65° using the structured mesh. Agreement between experimental and numerical numbers in the case of the AR2 cavity are largely within experimental error but the tendency of $k-\epsilon$ to over-predict leads to results on the upper limit of experimental error.

A visualisation of temperature stratification and eddy-viscosity is provided in Figure 5 for the AR1 cavity at an inclination angle of 30° as modelled by the $k-\epsilon$ and SST $k-\omega$ model. The ratio of eddy-viscosity to molecular dynamic viscosity (viscosity ratio) in Figure 5(a) for $k-\epsilon$ shows values as high as 17, whereas the maximum viscosity ratio in Figure 5(b) for $k-\omega$ is 4. In Figure 5(a) unstable density gradient dominates the generation of eddy-viscosity along the lower cavity wall and into the stagnant zone before stratification destroys it, whereas in Figure 5(b) eddy-viscosity is only resolved in the shear layer. The proximity of increased effective viscosity to heat transfer boundary layers increases advection, as we have seen in the numbers presented in Figures 3 and 4.

**CONCLUSION**

The scale of the experiments undertaken are undoubtedly in the laminar region of fluid flow. The results obtained with the CFD toolbox OpenFOAM® 3.0.1 confirm that although there is some resolution of eddy-viscosity, its impact on convective heat loss is small at this scale. In regards to the agreement between CFD and experiments, the following can be observed:

1. Without buoyant-affected turbulent modelling there is no impact of turbulence on convective heat loss estimation. The SST $k-\omega$ and Spalart and Allmaras numbers are nearly identical to those of the laminar case; nearly being the operative word, as for the case of AR2 cavity at 0° inclination the laminar prediction is actually higher than both SST $k-\omega$ and Spalart and Allmaras, implying that RANS modelled turbulence might have a negative
(a) Standard $k$-$\epsilon$ (with shear and buoyant generation of turbulence kinetic energy): the unstable density gradient along the lower inclined cavity wall produces a viscosity ratio as high as 17 but as the air reaches the back cavity wall, stratification is having a significant dampening effect; it damps any generation of turbulence due to shear

(b) SST $k$-$\omega$ (with shear generation of turbulence kinetic energy): the shear zone between the air entering the lower portion of the aperture and that exiting through the upper portion generates a maximum viscosity ratio of 4 but at a distance from walls as to have no effect on convective heat loss

FIGURE 5. Illustrations of temperature stratification and eddy-viscosity $\mu$, modelled with (a) the standard $k$-$\epsilon$ and (b) the SST $k$-$\omega$ turbulence closure scheme

impact on convective heat loss prediction at this scale. Indeed, it looks like modelling without RANS turbulence closure gives the best prediction in the case of AR2.

2. Numerical RANS prediction of shallow cavities (AR1) tends to under-predict heat loss. Whether this trend persists at larger scales is not known, although it is noted that in Yuan et al. [6] a similar under-prediction in the case of a large cubical cavity (AR1) was observed.

3. Turbulence does not seem to explain why we get poor agreement for the AR1 cavity but very good agreement for the AR2 cavity. The numerical solutions employed do not explain the general problem satisfactorily.

4. As we increase the size of cavity a transition to turbulence will occur. The above findings have no bearing on this situation.

It should be noted that all turbulent models used are so-called high-Reynolds-number turbulence schemes which assume isotropic turbulence and are to be used in conjunction with the law of the wall. This has the potential effect of overestimating turbulence in and near boundary layers, however this impact is negligible as we do not observe significant turbulence in the above simulations. Currently there are no low-Reynolds-number versions of the $k$-$\epsilon$
model with buoyancy generation terms [18] implemented in OpenFOAM® 3.0.1 which dampen this effect. An additional cause for concern is the definition in the linear eddy-viscosity concepts of the turbulent Prandtl number as constant. Short of more involved modelling, e.g. the Abe-Kondoh-Nagano 4-equation low-Reynolds-number $k − \epsilon$ model [19, 20], the impact of turbulent Prandtl numbers diverging from this constant cannot be explored.

Contrary to the common CFD practice of generating a mesh with sufficient surround-field domain around the cavity, it does not seem necessary to model the near and far field when only natural convection is observed. Everything of interest happens inside the cavity.

ACKNOWLEDGEMENTS

This work was partly supported by the Australian Renewable Energy Agency (project 1-UFA006) and with the computational resources of the Australian National Computational Infrastructure (NCI)

REFERENCES


---

4There are low-Reynolds-number versions without buoyancy generation terms which could be easily modified.
### APPENDIX

**Table 1.** Boundary Conditions for the fields relevant to buoyantSimpleFoam in OpenFOAM® 3.0.1 and the standard $k$-$\epsilon$, SST $k$-$\omega$ and Spalart and Allmaras turbulence models implemented with low-Re-number near-wall treatment

<table>
<thead>
<tr>
<th>Field</th>
<th>Atmosphere</th>
<th>Walls</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U$ m/s</td>
<td>pressureInletOutletVelocity</td>
<td>fixedValue value uniform (0 0 0)</td>
</tr>
<tr>
<td>$p_{gh}$</td>
<td>prghTotalPressure</td>
<td>fixedFluxPressure value uniform 1e5</td>
</tr>
<tr>
<td>$p^*$</td>
<td>value uniform 1e5</td>
<td>calculated</td>
</tr>
<tr>
<td>$T$</td>
<td>inletValue uniform 294</td>
<td>value nonuniform $f(x,y,z)$</td>
</tr>
<tr>
<td>$\nu_t$ m$^2$/s</td>
<td>turbulentIntensityKineticEnergyInlet</td>
<td>value uniform 0</td>
</tr>
<tr>
<td>$\alpha_t$ kg/(m.s)</td>
<td>turbulentMixingLengthDissipationRateInlet</td>
<td>value uniform 0</td>
</tr>
<tr>
<td>$k$ m$^2$/s$^2$</td>
<td>turbulentMixingLengthFrequencyInlet</td>
<td>value uniform 0</td>
</tr>
<tr>
<td>$\epsilon$ m$^2$/s$^3$</td>
<td>turbulentMixingLengthFrequencyInlet</td>
<td>value uniform 0</td>
</tr>
<tr>
<td>$\omega$ s$^{-1}$</td>
<td>turbulentMixingLengthFrequencyInlet</td>
<td>value uniform 0</td>
</tr>
<tr>
<td>$\tilde{\nu}$ m$^2$/s</td>
<td>turbulentMixingLengthFrequencyInlet</td>
<td>value uniform 0</td>
</tr>
</tbody>
</table>

*Initialised with the funkySetFields utility from swak4Foam.
† Initialised with the funkySetBoundaryField utility from swakFoam.
‡ Initialised with griddata linear interpolation in SciPy from patch face coordinates.