

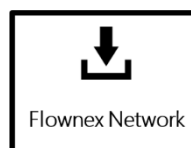


PREDICTING COMBUSTOR WALL TEMPERATURES USING THERMAL FLUID NETWORK MODELLING

This case study shows how a Flownex thermal–fluid network model was used to predict the temperature distribution along the conical bluff body of an academic combustor. The model is validated using experimental measurements, including the cold-flow velocity and the cone-tip surface temperature under the attached-flame condition.



UNIVERSITY OF
CAMBRIDGE



GAS TURBINES

GAS TURBINES

CHALLENGE:

The main problem was that reliable measurements of wall temperature on the combustor cone are generally unavailable. Without these data, setting realistic thermal boundary conditions for CFD simulations is difficult, and assuming uniform or arbitrary wall temperature risks large errors when flame–wall interactions significantly affect flame behaviour.

BENEFITS:

Flownex is a system-level thermal-fluid-network tool that can simulate both fluid flow and heat transfer (convection, conduction, radiation) across connected components. Because it uses simplified 1D / network models rather than full 3D CFD, Flownex runs much faster and requires far less computational effort, enabling practical estimation of wall-temperature distributions when experiments are not available or when full CFD with conjugate heat transfer would be prohibitively expensive.

SOLUTION:

To address the challenge, two coupled networks were built in Flownex: a mass-flow network representing the gas path from the combustor inlet (post-swirler) through to exhaust, and a heat-transfer network representing the cone wall and its interactions with the hot gases and the surrounding atmosphere. Under the attached-flame assumption, the simulation produced a chamber inlet velocity of 14.33 m/s, closely matching the experimental value of 14.4 m/s, confirming correct mass-flow setup. When applied heat from the flame at the cone tip and including convection, conduction, and radiation, the model predicted a cone- tip wall temperature of 603.14 K, which agrees well with the measured value. The full temperature distribution along the cone wall shows a physically reasonable trend, rising toward the flame attachment region.

INTRODUCTION

Computational fluid dynamics (CFD) has become a central tool for investigating combustion phenomena in gas turbine combustors. High-fidelity approaches such as direct numerical simulation (DNS), large-eddy simulation (LES) and appropriately modelled Reynolds-averaged Navier–Stokes (RANS) calculations allow detailed analysis of flow, mixing and flame dynamics. However, the predictive capability of these simulations depends critically on the correct definition of boundary conditions. Among them, the thermal boundary conditions at solid walls are particularly important, especially when the flame is attached to or approaches the combustor walls or flame holders. In such cases, wall heat losses can strongly influence local flame structure, stabilisation mechanisms and pollutant formation, and therefore have a direct impact on the reliability of CFD results.

In practice, accurate combustor wall temperature distributions are rarely available. Experimental measurements of wall temperature in reacting, confined flows are technically challenging, often requiring specialised diagnostics such as thermographic phosphors or intrusive probe techniques, and are not routinely performed for every operating condition or configuration. As a result, CFD practitioners frequently resort to simplified assumptions such as isothermal or adiabatic walls, or prescribe uniform wall temperatures based on limited information. These simplifications may lead to significant discrepancies between simulations and experiments, particularly in configurations where flame–wall interaction and wall heat transfer play a key role.

Thermal–fluid network models offer a complementary route to obtain more realistic wall temperature boundary conditions in a computationally efficient and cost-effective manner. Such models represent the combustor and surrounding system as a network of 0D/1D components with coupled mass, momentum and energy conservation, allowing the prediction of wall and metal temperatures under specified flow and thermal loading conditions. When properly configured and validated, they can provide spatially resolved wall temperature estimates that are more representative than ad-hoc assumptions, while requiring orders-of-magnitude less computational effort than full conjugate CFD.

In this case study, Flownex is used to develop a thermal network model of an academic gas turbine model combustor, with the specific objective of predicting the temperature distribution along its central cone wall. The combustor geometry and operating conditions correspond to a well-studied atmospheric-pressure swirl flame configuration that has been extensively investigated using advanced laser diagnostics, including time-resolved wall temperature measurements, thereby providing a suitable reference for assessing the network-model predictions.

SYSTEM DESCRIPTION

The system modelled in this case study is the atmospheric-pressure gas turbine model combustor investigated by Yin et al. (Experimental Thermal and Fluid Science, 2017, DOI: 10.1016/j.expthermflusci.2016.11.004). The combustor has been extensively studied in both experimental and numerical work, making it a well-established reference configuration for validating thermal boundary predictions.

GAS TURBINES

The schematic of the combustor is shown in Figure 1. The combustor consists of a cylindrical plenum, a swirl generator, a converging nozzle with a central bluff body, a square combustion chamber with quartz walls, and a conical exhaust section. A perfectly premixed methane–air mixture with an equivalence ratio of 0.7 and a thermal load of 20 kW enters the plenum, which has a diameter of 78 mm. Upstream of the plenum, a sonic nozzle isolates the combustor acoustically from the supply system. From the plenum, the mixture passes through a 12-vane radial swirl generator that imparts strong angular momentum before it accelerates through the converging nozzle of diameter 27.85 mm. At the nozzle centre sits a 34-mm-tall conical bluff body with a base diameter of 29 mm, providing the

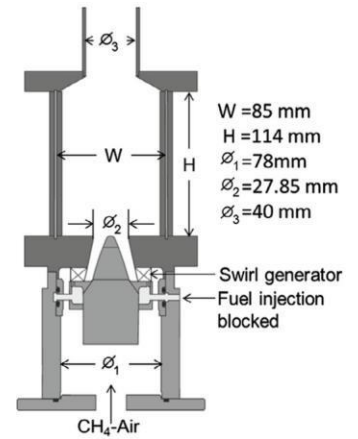


Figure 1: Schematic of the model combustor.

primary flame-holding surface.

The resulting flow at the nozzle exit reaches an average velocity of 14.4 m/s, corresponding to a Reynolds number of approximately 28,000, and produces a swirl number of about 0.6 based on velocity measurements just above the exit. After leaving the nozzle, the mixture enters the combustion chamber, which has a square $85 \times 85\text{ mm}^2$ cross-section and a height of 114 mm. The chamber walls are constructed from quartz to allow optical access, and combustion products are discharged through a conical outlet into a 40-mm-diameter exhaust duct.

At the target operating condition, this combustor exhibits bi-stable behaviour, alternating between two flame shapes: a V-shaped flame attached directly to the bluff-body tip, and an M-shaped lifted flame. These two states lead to markedly different heat-transfer conditions, with experimentally measured surface temperatures on the bluff-body tip differing by up to 100 K during transitions. Such sensitivity to flame shape underscores the necessity of accurately predicting wall temperatures when applying boundary conditions for CFD simulations. Consequently, the present work uses a thermal network model to estimate the temperature distribution along the cone wall of the bluff body, providing more realistic thermal boundary inputs for subsequent combustion modelling.

OBJECTIVE OF SIMULATION

The objective of this simulation is to determine the temperature distribution along the wall of the central conical bluff body when the combustor operates in the attached V-flame condition. Since the cone surface experiences significant heat transfer when the flame anchors directly at the bluff-body tip, obtaining an accurate temperature profile is essential for defining realistic wall boundary conditions in subsequent CFD combustion studies.

To achieve this, a thermal–fluid network model is developed in Flownex to replicate the heat transfer processes acting on the bluff body. The predicted cone-wall temperature is then validated against the experimentally measured surface temperatures reported in the reference study, which provide high-resolution temperature data for the attached flame case. These measurements serve as the benchmark for assessing and tuning the heat-transfer model setup.

In addition to this thermal validation, the simulation also verifies the accuracy of the mass-flow configuration. The experimentally reported cold-flow velocity at the nozzle exit is used to confirm that the mass-flow rate, inlet conditions, and associated pressure boundary setup in the Flownex

GAS TURBINES

model are correctly defined. Ensuring that the flow rate matches the experiment is necessary because the convective heat transfer to the bluff body strongly depends on the local flow field.

Overall, the simulation aims to produce a reliable prediction of the wall temperature distribution on the conical bluff body under the attached flame condition by ensuring that both the thermal and flow boundary conditions in the network model are physically consistent and validated against experimental data.

FLOWNEX MODEL

Identified by a blue flow path, the mass-flow portion of the network depicted in Figure 2 represents the gas flow path from the outlet of the swirl generator, through the combustion chamber where the conical bluff body is located, and finally to the far field (ambient atmosphere). This establishes the correct flow regime and enables a realistic prediction of convective heat transfer to the cone wall.

To capture spatial variations in velocity near the combustor inlet, the pipe section connecting the swirl generator to the combustion chamber is divided into ten pipe elements rather than being modelled as a single segment. This segmentation increases resolution in the predicted velocity distribution, which is important because the local flow behaviour strongly influences the heat transfer to the bluff body.

After the mass-flow network is established, a dedicated heat-transfer network, identified by orange lines, is constructed to model the thermal behaviour of the cone wall. This thermal network is fully coupled to the mass-flow network, ensuring that the convective heat transfer coefficients are driven by the local flow velocity produced by the fluid solution. The conical bluff body is discretised into ten wall segments, enabling the simulation to resolve the temperature distribution along its height rather than producing a single, averaged wall temperature.

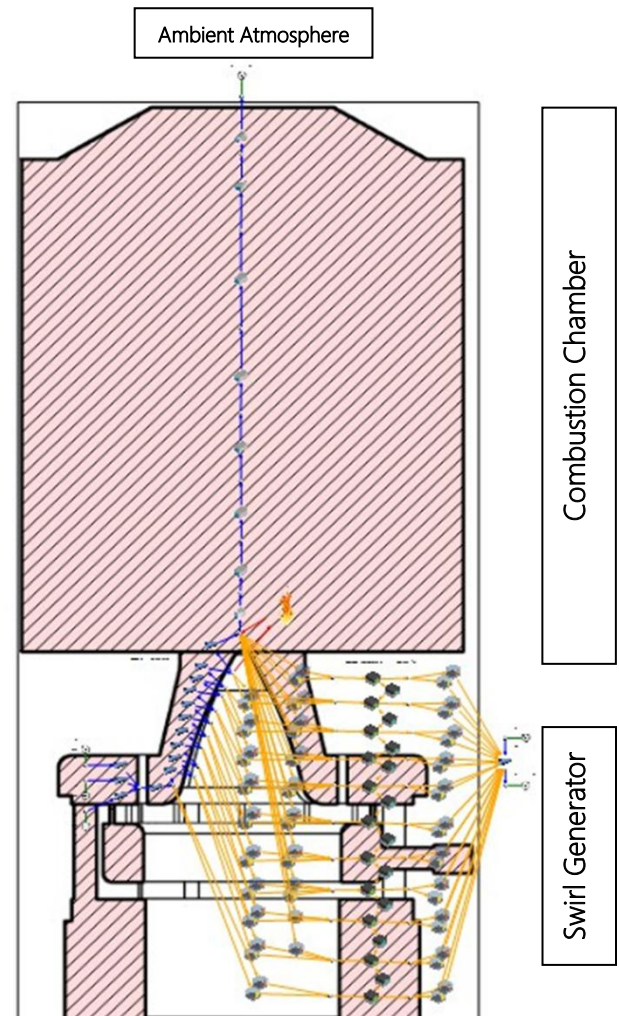


Figure 2: Mass-flow and heat-transfer network.

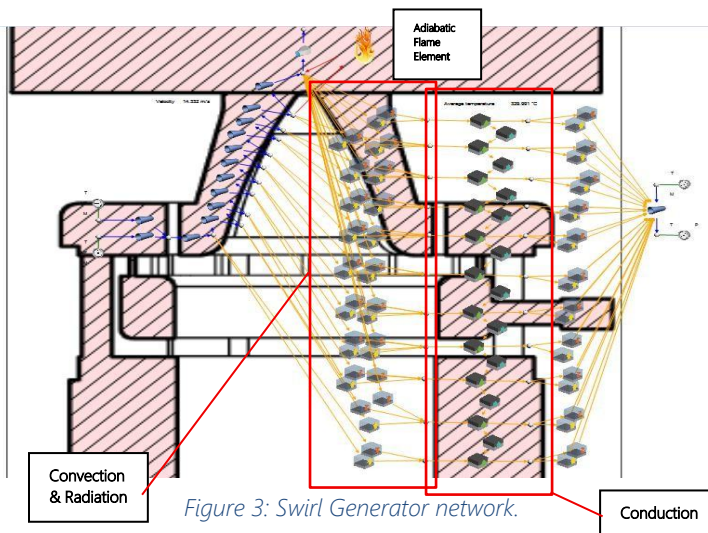


Figure 3: Swirl Generator network.

The cone is assumed to be hollow. It is assumed that other walls of the combustor have negligible influence on the heat transfer process on the cone wall. In Figure 3 all relevant heat-transfer modes are included: convection and radiation from the combusting flow to the cone surface, conduction between adjacent wall segments (including cross-conduction effects), and convection and radiation

GAS TURBINES

from the outer cone surface to the surrounding atmosphere. This ensures that the energy balance on each wall segment reflects both internal and external heat-transfer processes. The parameter settings follow the paper by Du Toit and Theron (Proceedings of ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, 2016, DOI: 10.1115/GT2016- 57685)

To represent flame–wall interaction, an adiabatic flame element is added (Figure 3) to the model. Based on experimental observations, the attached V-flame is assumed to anchor at the cone tip, meaning that the highest incident heat flux is applied to the first segment of the cone. This boundary condition is important for reproducing the experimentally observed steep temperature gradient near the tip of the bluff body.

DESCRIPTION OF SIMULATION

The Flownex simulation operates by solving the coupled fluid-flow and heat-transfer behaviour of the combustor under the attached V-flame condition. Once the network topology is defined, Flownex iteratively solves the conservation equations for mass, momentum, and energy across all components. The mass-flow network determines the pressure distribution, flow velocities, and convective heat-transfer coefficients, while the thermal network computes the transient and steady-state temperature field on the bluff-body cone.

Beginning with the mass-flow solution, Flownex calculates the flow rate through the system based on the pressure difference between the combustor inlet and the far-field boundary. The pipe sections downstream of the swirl generator—including the ten-segment discretisation leading to the chamber—allow the solver to establish a spatially varying velocity field rather than a uniform average value. This is important because the convective heat transfer to the cone wall depends strongly on the local flow speed. The simulation uses the experimentally reported cold-flow exit velocity from the nozzle to validate the mass-flow setup and ensure that the predicted velocity regime is consistent with the operating conditions of Yin et al. (Experimental Thermal and Fluid Science, 2017, DOI: 10.1016/j.expthermflusci.2016.11.004).

Once the velocity and flow properties are converged, the thermal network computes heat transfer to and from each of the ten cone-wall segments. The model treats the cone as a hollow component, which allows heat to be transferred between segments through conduction while ignoring solid-core thermal mass. Each wall segment receives heat from the combustor gases through convection and radiation. These heat-flux terms depend directly on the temperatures and flow velocities computed in the fluid solution. Heat is then redistributed along the cone through conduction—including cross-conduction to adjacent segments—and finally discharged to the surrounding atmosphere by external convection and radiation. As a result, each cone segment reaches a temperature that reflects the integrated effect of local gas heating, conductive transport, and external cooling.

The flame interaction is represented by imposing an adiabatic flame zone near the cone tip. Based on experimental observations, the attached flame anchors at the tip of the bluff body, so the flame element is positioned to supply the appropriate heat load to the first cone segment. This creates the strong temperature gradient expected in the V-flame regime. The flame is set according to the methane-air operating condition reported by Yin et al., ensuring consistency between the simulation conditions and the experimental configuration used for validation.

During the simulation, the mass-flow solution and the heat-transfer solution are fully coupled: changes in wall temperature affect the local convective heat-transfer coefficients, and changes in

GAS TURBINES

flow velocity influence heat flux to the wall. Flownex iterates between these calculations until the overall solution converges. The predicted temperature distribution along the cone is then compared with the experimentally measured values to evaluate the adequacy of the heat-transfer modelling approach.

RESULTS

The Flownex network model was first validated against the measured cold-flow velocity reported by Yin et al. (Experimental Thermal and Fluid Science, 2017, DOI: 10.1016/j.expthermflusci.2016.11.004). The simulation predicts an average flow velocity of 14.33 m/s at the chamber inlet, which is in close agreement with the experimental value of 14.4 m/s. This confirms that the mass-flow setup and pressure boundary conditions reproduce the correct flow regime, ensuring that the resulting convective heat-transfer coefficients around the cone are physically consistent with the operating conditions.

The predicted wall temperature at the cone tip under the attached V-flame assumption is 603.14 K, which aligns well with the experimentally measured surface temperatures at the corresponding location. Since the cone tip is the region of strongest flame–wall interaction, this agreement indicates that the imposed flame heat load and the combined convection–radiation model capture the correct magnitude of thermal exposure on the bluff body.

Figure 4 presents the simulated temperature distribution along the cone wall. The temperature values shown are obtained from the conduction components of the thermal network, where each value corresponds to the temperature at the mid-point of its respective cone-wall segment. This assumption is appropriate because each segment represents a uniform thermal element, and the mid-point temperature is the most representative value for plotting a continuous profile. The temperature increases progressively from the cone base toward the tip,

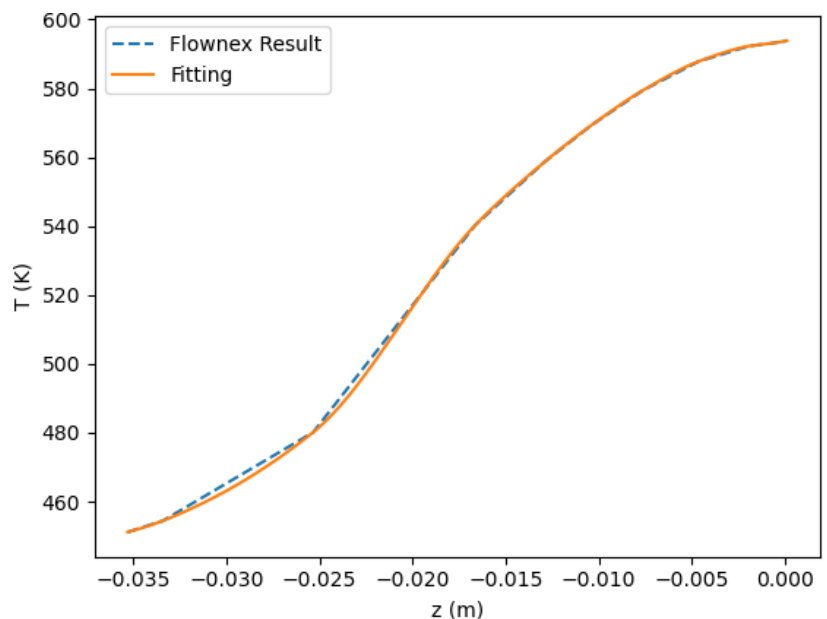


Figure 4: Temperature distribution along the cone wall.

reflecting the increasing influence of flame heating. The steeper gradient near the tip is consistent with the experimentally observed behaviour in the attached V-flame regime, where the flame imparts the highest heat flux to the uppermost region of the cone.

Overall, the simulation results show strong consistency with experimental observations in both flow and thermal behaviour. This confirms that the developed Flownex model is capable of accurately predicting the cone-wall temperature distribution and can therefore be used to provide reliable thermal boundary conditions for subsequent CFD combustion simulations.

CONCLUSION

In this work, a Flownex thermal–fluid network model was developed to predict the temperature distribution along the conical bluff body of an academic swirl combustor operating under the attached V-flame condition. By accurately reproducing the experimentally measured cold-flow velocity and cone-tip temperature, the model demonstrated that a system-level network approach can provide realistic estimates of combustor wall temperatures, which are essential for defining appropriate thermal boundary conditions in CFD simulations. The temperature profile along the cone wall showed good qualitative and quantitative agreement with the expected thermal behaviour, capturing the strong increase in temperature toward the flame attachment point. This confirms that the coupled mass-flow and heat-transfer modelling framework is capable of representing the dominant heat-transfer mechanisms acting on the bluff body.

However, it is important to emphasise that the present network model relies on several simplifying assumptions. While these assumptions allowed the model to be implemented efficiently and yielded encouraging agreement with experimental data, they also limit the accuracy and generality of the current results, highlighting the need for further refinement.

Future development of this work should involve validating the network model using additional datasets, including experiments or simulations such as large-eddy simulations and using different fuels to assess the model performance under varied combustion environments. It may also be necessary to extend the heat-transfer modelling to include other chamber walls if they are found to contribute significantly to the overall thermal behaviour. Such improvements will allow the network model to evolve into a more robust and versatile tool for supporting high-fidelity combustion studies.