



**HAL**  
open science

# LES investigations of turbulent heat transfer structures with exponential power escalation

Elie Roumet, Raksmy Nop, Nicolas Dorville, Marie-Christine Duluc

► **To cite this version:**

Elie Roumet, Raksmy Nop, Nicolas Dorville, Marie-Christine Duluc. LES investigations of turbulent heat transfer structures with exponential power escalation. 9th European Thermal Sciences Conference (Eurotherm 2024), Jun 2024, Lake Bled, Slovenia. pp.012012, 10.1088/1742-6596/2766/1/012012 . hal-04616128

**HAL Id: hal-04616128**

**<https://cnam.hal.science/hal-04616128>**

Submitted on 18 Jun 2024

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons Attribution 4.0 International License

PAPER • OPEN ACCESS

## LES investigations of turbulent heat transfer structures with exponential power escalation

To cite this article: Elie Roumet *et al* 2024 *J. Phys.: Conf. Ser.* **2766** 012012

View the [article online](#) for updates and enhancements.

### You may also like

- [Nano-CdS confined within titanate nanotubes for efficient photocatalytic hydrogen production under visible light illumination](#)  
Lizhen Long, Xiang Yu, Liangpeng Wu et al.
- [Second-order kinetic energy-conservative analysis of incompressible turbulence under a collocated grid system](#)  
Hiroki Suzuki, Yutaka Hasegawa, Masaya Watanabe et al.
- [LES of Thermally Stratified Turbulent Channel Flow at Low Prandtl Number](#)  
Waqar Raza, Ajiv Alam Khan and Syed Mohd Yahya



The Electrochemical Society

Advancing solid state & electrochemical science & technology

**DISCOVER**  
how sustainability  
intersects with  
electrochemistry & solid  
state science research



# LES investigations of turbulent heat transfer structures with exponential power escalation

Elie Roumet<sup>1,2</sup>, Raksmy Nop<sup>1</sup>, Nicolas Dorville<sup>1</sup> and Marie-Christine Duluc<sup>2</sup>

<sup>1</sup> Université Paris-Saclay, CEA, Service de Thermo-hydraulique et de Mécanique des Fluides, 91191, Gif-sur-Yvette, France

<sup>2</sup> Conservatoire National des Arts et Métiers, Lafset, 75003, Paris, France

E-mail: [elie.roumet@cea.fr](mailto:elie.roumet@cea.fr)

**Abstract.** The BORAX-type accident is a key scenario in the safety design of pool-type experimental nuclear reactors. It considers a reactivity insertion large enough to initiate an exponential power escalation with a period as small as tens of milliseconds, necessitating effective heat removal by the coolant system. This study presents a computational analysis of single-phase transient heat transfer under thermal-hydraulic conditions relevant to such reactors, focusing on turbulent channel flow between planar fuel plates with highly subcooled water. Our investigation uses Large Eddy Simulation (LES) performed with TrioCFD, the open-source CFD code developed by the French Atomic Energy Commission (CEA). Several modeling options are tested: incompressible flow with temperature as a passive scalar, incompressible flow with varying viscosity, and quasicompressible flow with temperature-dependent viscosity and mass density. To evaluate the accuracy of our physical models, all simulations were performed under uniform conditions, using the same experimental parameters, mesh, and numerical strategies. The specific examined scenario involves a highly subcooled turbulent channel flow at moderate pressure, experiencing an exponential power increase. We ensure detailed turbulence resolution at the Batchelor scale in the wall normal direction close to the heating wall. This communication presents a comparative study of LES simulations of the single-phase flow under exponential power excursion at the wall. Subsequent analyses focused on the impact of different velocity-temperature couplings on the prediction of wall heat transfer, incorporating comparisons with experimental infrared thermometry measurements. Early in the transient phase, temperature peaks are confined within turbulent streaks, as reported in the existing literature. However, as the exponential power transient unfolds, the viscous layer and streak structures destabilize, leading to a more dispersed distribution of hot spots. Preliminary findings suggest a transition in heat transfer modes from turbulence-driven to conduction during the final stages of the single-phase transient.

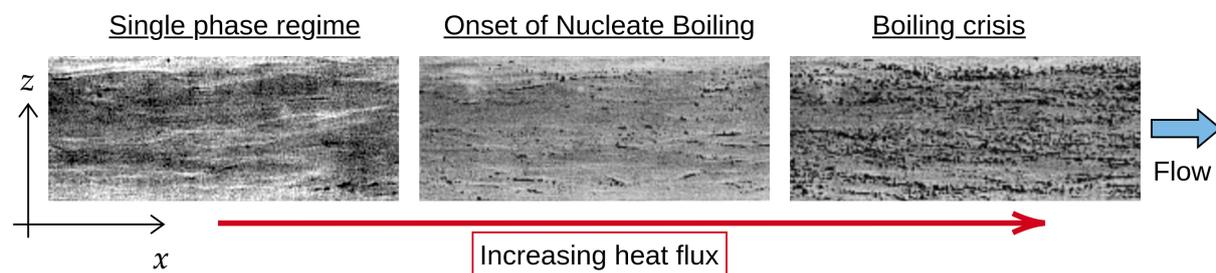
## 1. Introduction

In nominal operating conditions, the fluid within the core of a research reactor is in single phase. During a BORAX-type scenario, an accidental reactivity insertion can induce an exponential increase in power, characterized by a time constant,  $\tau$  of about tens of milliseconds. Due to the large subcooling levels, most of the time of the transient is spent in single phase. However, the rapid escalation of power leads to a concentration of energy deposition in the two-phase regime towards the end of the transient. For the fastest power escalations, nucleation onset nearly coincides with the formation of extensive vapor blankets indicative of boiling crisis. The



emergence of dry patches reduces both heat transfer efficiency and density reactivity feedback, which is linked to volume of vapor in the core. Therefore, an accurate prediction and a better physical understanding of the boiling crisis under an exponential power excursion is a key issue for nuclear safety.

Experimental investigations employing frontlit shadowgraphy techniques have illustrated the dynamic interactions between bubbles and turbulent wall streaks. Figure 1 shows the thermal features during the different stages of a power transient. In these images, turbulent thermal streaks appear as white, while bubbles are depicted in black. We observe that most of the active nucleation sites appear within these turbulent streaks. During the boiling crisis, irreversible dry patches emerge within the hot spots, having the characteristic shape of turbulent thermal streaks.



**Figure 1.** Frontlit shadowgraphy visualization of the heated wall during a 5 ms power transient at 1.2 MPa, 163 K subcooling and  $Re = 30.000$  adapted from Chavagnat et al.[1]

Recently, Nop et al.[2] introduced a mechanistic energy-based model predicting the transient subcooled flow boiling crisis. Central to this model is the concept of ‘mantle thickness’ ( $\delta$ ), a theoretical parameter representing a characteristic energy layer that achieves saturation during power transient. This highlighted the need for insights into the evolution of the thermal boundary layer in response to transient heat inputs.

Using the TRUST/TrioCFD software, we study the dynamics of heat transfer near the wall through Large Eddy Simulation (LES) analyses. The primary objective is to investigate the behavior of the thermal boundary layer at the onset of nucleate boiling, thereby contributing to a more comprehensive understanding of the thermal phenomena underpinning the onset of boiling crises during a transient heat input.

## 2. Methodology

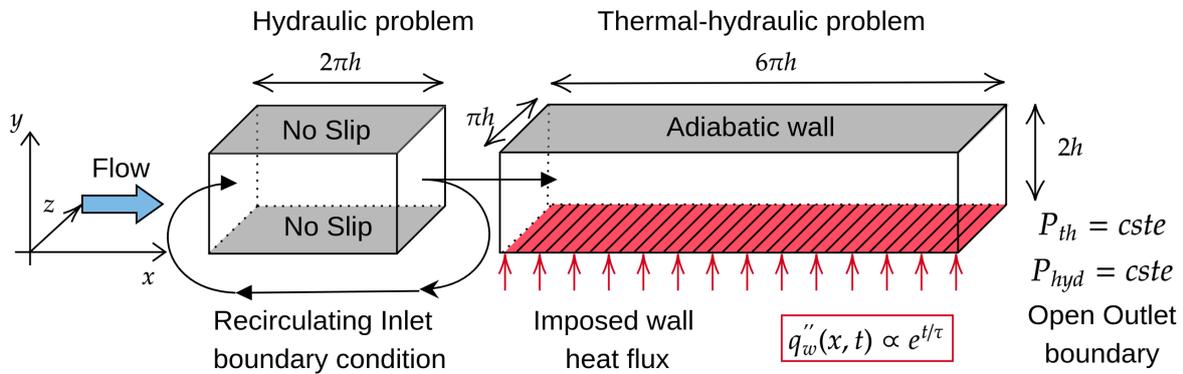
We present several Large Eddy Simulations (LES) focusing on the transient wall heating within a turbulent channel. The simulations explore different configurations to investigate the interplay between thermal and fluid dynamic phenomena under various conditions:

- Incompressible flow with constant fluid properties: In this configuration, the temperature is considered as a passive scalar, with no influence on the fluid dynamics.
- Incompressible flow with varying viscosity: This setup introduces a coupling between the temperature and velocity fields by allowing the fluid viscosity to vary with temperature.
- Quasi-compressible flow with varying viscosity and mass density: This approach aims at capturing the intricate interactions resulting from density variations, particularly their effects on mass conservation and momentum within the flow field.

Each configuration is designed to incrementally increase the complexity of the thermal-fluid coupling, enabling a detailed examination of how temperature variations influence the flow dynamics under different assumptions about the fluid thermophysical properties.

### 2.1. Numerical problem solved by the TRUST/TrioCFD code

The simulation setup is illustrated in figure 2, the mean flow is in the  $x$  direction and a recirculating box is placed at the domain inlet. We solve a hydraulic problem in the recirculating box and a thermalhydraulic problem in the numerical test section. Both the recirculating box and the domain are periodic in the spanwise direction ( $z$ ). The channel half width is denoted  $h$ . The global size  $2h \times 6\pi h \times \pi h$  was defined in order to minimize the influence of the computational domain on the solutions using independence mesh studies of thermal flows by Luesma-Rodriguez et al.[3]. In the present investigation, the mesh dimensions are specified as a height of 2 mm, a width of 3.15 mm, and a streamwise length of 1.89 cm. This size selection in the streamwise direction is intended to accurately model the experimental heater setup and to adequately capture the thermal boundary layer development.



**Figure 2.** Schematic description of the numerical problem

For the Navier-Stokes equations (1-2), the inlet boundary is a fully developed turbulent velocity profile obtained from the recirculating box, ensuring a realistic turbulent flow entering the domain. At the outlet, the boundary is open with a specified pressure condition. The no-slip condition is applied at both the bottom and top boundaries.

For the heat equation (3), the inlet boundary condition is an imposed temperature, modeling the isothermal inlet condition of the fluid. The outlet is considered as an open boundary for temperature, allowing heat to leave the domain freely. The top boundary is treated as an adiabatic wall. At the bottom boundary, a specified time dependant but spacially uniform heat flux is imposed. The set of equations being solved is:

$$\left\{ \begin{array}{l} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \\ \frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = \nabla \cdot (\mu \nabla \mathbf{v}) - \nabla (P_1^*) \\ \rho c_p \left( \frac{\partial T}{\partial t} + \mathbf{v} \cdot \nabla T \right) = \nabla \cdot (\lambda \nabla T) + \frac{dP_0}{dt} \end{array} \right. \quad \begin{array}{l} (1) \\ \text{where } P_1^* = P_1 + \frac{2}{3} \mu \text{div}(\mathbf{v}) \quad (2) \\ \text{where } P_0 = \rho(T)RT \quad (3) \end{array}$$

The TRUST/TrioCFD software uses a staggered grid system in conjunction with a projection technique to decouple velocity and pressure fields. This mitigates the issue of checkerboard pressure oscillations and ensures that the velocity field satisfy both the momentum and continuity equations, leading to more physically accurate simulations. All simulations presented in this work leverage finite volume difference discretization on a hexahedral mesh with 30 million elements. A comprehensive review of the TRUST/TrioCFD code is highlighted in the article by Angeli et al.[4] (2015). The following tables summarize the numerical methods and mesh

characteristics used in the simulations runs.

**Table 1.** Numerical methods and schemes.

Discretisation	Finite Volume Differences
Pressure Solver	General Conjugate Gradient
Diffusion	Implicit (second order)
Momentum Convection	Centered (second order)
Scalar Convection	QUICK (third order)
Time scheme	Runge Kutta 3

**Table 2.** 30M hexahedral mesh characteristics

Size	$y^+$	Dimensionnal unit
$\Delta y_w$	$0.5y^+$	$2.13\mu m$
$\Delta y_{fs}$	$1.86y^+$	$7.95\mu m$
$\Delta z$	$4y^+$	$16\mu m$
$\Delta x$	$8y^+$	$32\mu m$

## 2.2. Physical characteristics and turbulence model

With the high subcooling level of 100 K and the exponential heat input characterized by a time constant  $\tau$  of 5 ms, significant temperature gradients are expected at the vicinity of the wall. Under these transient conditions, viscosity can be reduced by up to 50%, while other properties such as conductivity, heat capacity and mass density vary by around 10%. This highlights the need for a modeling approach that effectively captures the interactions between the temperature and momentum fields. The quasi-compressible model aims to investigate the specific effects of the flow field positive divergence near the wall, due to mass density variation.

Gravitational effects on the flow are considered negligible, supported by a Richardson number ( $Ri = g\Delta\rho L/\rho U^2$ ) much less than 0.01. The equation of state is a stiffened gas fitted on IWAPS97 data for liquid water properties. Properties in metastable states above the saturation temperature are assumed to be constant and equal to those at saturation.

The mesh design incorporates refinement in the wall-normal direction up to the Batchelor scale, targeting accurate representation of thermal gradients and boundary layer dynamics at the wall (noted  $y_w$ ). In the freestream region (noted  $y_{fs}$ ), the mesh is stretched by a factor of 4 in the streamwise direction and by a factor of 2 in the spanwise direction.

For turbulence modeling, the Wall-Adapting Local Eddy-viscosity (WALE)[5] model is selected, with a fixed turbulent Prandtl number of  $Pr_t = \nu_t/\alpha_t = 0.9$  to couple the velocity and temperature fields subgrid models. The WALE model is chosen for its capability to address the complexity of shear flows.

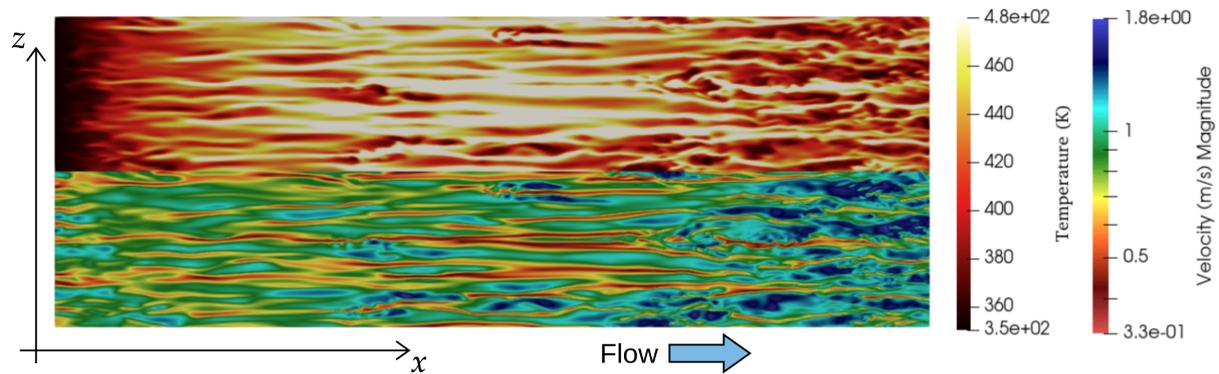
## 3. Simulation results

The image depicted in figure 3 presents a cross-section in the  $(x, z)$  plane, capturing the velocity and temperature fields in the vicinity of the wall moments before the Onset of Nucleate Boiling of a quasi-compressible simulation. We observe that the temperature and velocity fields exhibit a strong correlation. Additionally, the turbulent patterns maintain a consistent shape throughout the channel, albeit with a reduced scale towards the outlet where the viscosity is lowered due to heating.

### 3.1. Comparison with experiments under transient heat input

The experimental setup uses infrared thermography at a high frame rate of 3125 frames per second to capture flow boiling phenomena within a dedicated loop. Details of the boiling loop, experimental apparatus and postprocessing techniques are published in the work of Richenderfer et al.[6](2018).

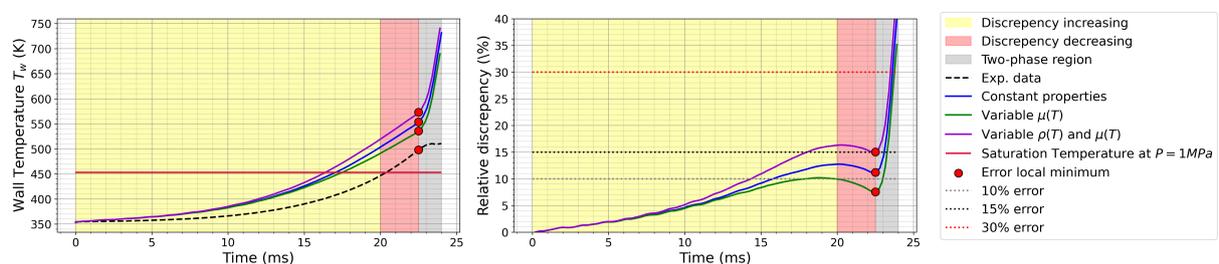
This comparison is based on the data presented in Nop et al.[7](2022), which involves a test section featuring a channel height of 2 mm and a heater measuring 20 mm in length and 7 mm in width. The experiments were conducted at a fixed operating pressure of 1 MPa, with the liquid bulk temperature stabilized at 80°C, leading to a subcooling level of 100K. Water is flowing with a mean velocity of 1.8 m/s, which corresponds to a bulk Reynolds number of 20 000 and



**Figure 3.** Wall normal slice at  $y^+ = 10$  of the temperature and flow velocity close to ONB

a friction Reynolds number of 300. The bottom plate of the channel is subjected to a uniform power transient  $q_w''(t) \propto e^{t/\tau}$ , where  $\tau = 5$  ms.

By replicating the experimental conditions in our simulations, we can compare the heat transfer during a power transient obtained from numerical predictions and those derived from experimental observations. The following figure 4 presents the space average of the wall temperature  $T_w$  plotted over time during power transient. The dashed line is the average experimental wall temperature. Note that the significant discrepancies observed after  $t = 23$  ms, exceeding 30%, are due to the onset of boiling on the heater surface, as noted experimentally. Phase change does not fall within the scope of this work.



**Figure 4.** Left: Wall temperatures  $T_w$  comparison during the transient  
Right: Relative discrepancy of simulations  $\frac{T_{sim} - T_{exp}}{T_{exp}}$

The left graph in figure 4 indicates that the simulations consistently predict higher wall temperatures than observed, suggesting an underestimation of the heat transfer in the fluid. The right graph demonstrates a gradual increase in the discrepancy of the simulated wall temperatures during the transient up to approximately 20 ms. During this period, turbulent structures are weakly affected by the temperature increase and thus efficiently transfer heat from the vicinity of the wall to the bulk of the flow. Thus, the observed discrepancy may stem from inaccuracies in modeling the turbulent heat flux, attributable to the assumption of a constant turbulent Prandtl number. Despite this, all simulations maintain a deviation lesser than 15% with experimental data in the initial phase of the transient.

Interestingly, simulations using the quasi-compressible model show the lowest level of accuracy compared to others, while those adopting the incompressible model with variable viscosity yield the most accurate results. The discrepancies seen in the quasi-compressible simulations could be due to offsetting errors present in the incompressible simulations. In the quasi-compressible

approach, only the mass density and viscosity are considered as temperature-dependent. Since the observed discrepancies align with the physical changes in thermal conductivity and specific heat capacity, using an accurate equation of state is key to improve the modeling process.

As the single phase concludes, particularly between 20 *ms* and 22.5 *ms*, the mean wall heat flux escalates above  $4\text{MW}/\text{m}^2$ , and the gap between simulation outcomes and experimental results diminishes to a minimum. This pattern could indicate a shift in the primary heat transfer mechanism, transitioning from turbulent heat transfer to transient thermal diffusion.

This inference is drawn from the observation that under such extreme transient heat flux conditions, the decrease in near-wall viscosity results in smaller turbulent structures, therefore reducing the efficiency of heat transfer from the wall vicinity to the bulk of the fluid. This suggests that, at this critical juncture, the system ability to manage heat is increasingly reliant on the direct thermal diffusion across the thermal boundary layer, rather than on the convective efforts of turbulence.

#### 4. Conclusion

This work presents several Large Eddy Simulations of liquid water subjected to an exponential heat input. A comparative study of various modelling hypotheses is carried out in order to assess the fidelity of simulations. Overall, all simulations exhibits a consistent behavior and fall within  $\pm 15\%$  of the experimental wall temperature during the single phase regime. The variable viscosity simulation performs best, highlighting the importance of taking into account the viscosity dependance to temperature.

Yet, there are systematic discrepancies during the first part of the transient for all simulations that might be related to subgrid models or offsetting errors. Further work will investigate new simulations implementing tetrahedral meshes refined isotropically at the batchelor scale near the wall. This will ensure that the intricate balance between convection and conduction processes taking place in the buffer layer will be directly resolved by deactivating both thermal and velocity subgrid models. In addition, we might implement fully quasicompressible simulations with all properties depending on the temperature and implement an equation of state valid for metastable superheated water.

Several new configurations of transient heat input will be investigated by varying the mass flux, pressure, subcooling, power escalation time and geometry of the channel.

#### Acknowledgments

The authors would like to acknowledge the insightful discussions and assistance provided by the SALOME and TRUST support teams.

The computations were made possible through access to the HPC resources of TGCC under the allocation A0152A13031 from GENCI.

#### References

- [1] Chavagnat F, Nop R, Phillips B, Dorville N, Duluc M-C and Bucci M 2022 *NURETH-19*, Brussels, Belgium
- [2] Nop R, Duluc M-C, Dorville N, Kossolapov A, Chavagnat F and Bucci M 2021 *Int. J. of Thermal Sciences* **168** 107042
- [3] Lluesma-Rodríguez F, Hoyas S and Perez-Quiles M J 2018 *Int. J. Heat Mass Transfer* **122** 983–992
- [4] Angeli P E, Bieder U and Fauchet G 2015 *NURETH-16*, Chicago, Illinois (US)
- [5] Nicoud F and Ducros F 1999 *Flow, Turbulence and Combustion* **62** 183–200
- [6] Richenderfer A, Kossolapov A, Seong J H, Saccone G, Demarly E, Kommajosyula R, Baglietto E, Buongiorno J and Bucci M 2018 *Exp. Therm. Fluid Sci.* **99** 35–58
- [7] Nop R, Chavagnat F, Duluc M-C, Dorville N, Kossolapov A and Bucci M 2022 *NURETH-19*, Brussels, Belgium