

Thermal Hydraulic Model: Key Results and Findings

Nolan Son

Contents

Executive Summary.....	3
Project Scope	4
Governing Equations and Assumptions	5
Numerical Methods	8
Results and Discussion	9
Model Limitations and Future Work	11
Conclusion.....	12

Executive Summary

This project develops a one-dimensional transient thermal model to simulate heat transfer along a discretized solid domain with convective cooling to an ambient environment. The model applies finite-difference methods to solve the energy balance at each node, incorporating conduction between nodes and convection to an external fluid at a specified ambient temperature.

The simulation successfully captures physically realistic temperature decay from a prescribed inlet temperature toward the ambient temperature, validating both the numerical formulation and boundary condition enforcement. Results demonstrate sensitivity to key parameters such as convective heat transfer coefficient, node spacing, and ambient temperature, providing insight into system-level thermal behavior.

This model serves as a foundational framework for more advanced thermal-fluid simulations, with clear pathways for extension into transient behavior, internal heat generation, and coupled multi-physics analyses relevant to energy systems and reactor thermal design.

Project Scope

This project seeks to model axial heat transfer within a one-dimensional pipe segment containing a flowing liquid coolant. The pipe has a defined length, and the time scale is defined to capture transient behavior. Only axial flow was modeled, ignoring any radial heat transfer, to simplify the model and to understand how the temperature evolves down the length of the pipe, a more valuable insight, given boundary conditions at the start and end of the pipe segment.

Governing Equations and Assumptions

This project is governed by an energy balance in the form of:

$$m_{node} C_p \frac{dT_i}{dt} = (\dot{m} C_p \Delta T) + (q''' V_{node}) - (h A_{surf} (T_i - T_\infty))$$

Where:

- m_{node} = mass of fluid at the current node
- C_p = fluid heat capacity
- $\frac{dT_i}{dt}$ = temperature differential at a given time step
- q''' = volumetric heat generation rate
- V_{node} = the volume at the current node
- h = heat transfer coefficient
- A_{surf} = surface area of the current node
- T_i = temperature at a given time step
- T_∞ = ambient temperature

This equation enforces the conservation of energy in this heat transfer problem. The first term represents the sensible heat change, from the temperature gradient between the fluid and surroundings. The second term represents the heat generated, expected in a pipe that is part of a reactor system. The volumetric heat generation is kept constant and assumed at a reasonable rate for a nuclear reactor. The third term represents the convective heat transfer between the surface area of the pipe and the ambient temperature conditions.

This project made key assumptions to simplify the model:

- Plug flow scheme.

A plug flow scheme, therefore axial mixing, is assumed. This removes any need to estimate radial gradients that might be present in a pipe, keeping the model 1-dimensional.

- Constant thermophysical properties

The properties of the liquid are assumed to be held constant. This eliminates the need to dynamically update these properties as the temperature evolves within the pipe.

- No axial conduction

It is assumed that connecting nodes have no conductive heat transfer between segments. This simplifies the energy balance to reduce significant terms.

- No pressure drop

Pressure drop would become a more significant term as the pipe length increases.

Keeping the pipe length low, pressure drop can be neglected without meaningful loss of accuracy.

- Single phase liquid flow

The fluid is considered to only be a liquid, ensuring the flow scheme is simplified for the model.

- Constant heat transfer coefficient

Heat transfer coefficient is dynamic as temperature changes, but at low temperature changes, most liquids will not exhibit significant changes, meaning that holding a constant coefficient should not vary results by a noticeable amount.

The intention of all assumptions is to isolate thermal hydraulic behavior, rather than thermodynamic nuances.

The following boundary conditions were established:

- Set inlet temperature
- Natural outflow at outlet

These conditions assume a known temperature at an inlet, and a natural progression of the fluid through the pipe. A known temperature at inlet is common in industry and can establish a starting point for the model easily.

Numerical Methods

This project used multiple methods of numerical integration.

Spatial discretization was used by separating pipe length into differential nodes. By using finite control volumes, each segment can be used as a point, giving valuable insight into how the temperature evolves down the length of the pipe. Differential nodes then had individual masses and volumes calculated, so sensible heat transfer and heat generation from the reactor could independently be solved for at each node.

Time integration was implemented via Euler's Method. A time step can be defined within the program, as well as a start and end time, allowing precise time manipulation to investigate transient or steady-state behavior. The model will move forward by one time step, calculate the temperature at the given node and time, and continue to loop until the end time has been reached.

Boundary conditions are enforced within the program, having the 0th node set as the inlet temperature. The program originally would treat the 0th node as the first step in time, meaning that the temperature would be calculated at a time step after the initial conditions, giving unrealistic behavior. The boundary conditions are enforced before the spatial and time loops to ensure only the 0th node is affected.

Results and Discussion

Transient simulations were performed to evaluate the axial temperature response of the modeled pipe under varying inlet conditions and heat transfer parameters. The model produced stable, physically consistent solutions across the simulated time domain, demonstrating expected trends in temperature evolution and energy transport. The temperature profile of a trial with water as the assumed fluid is shown below:

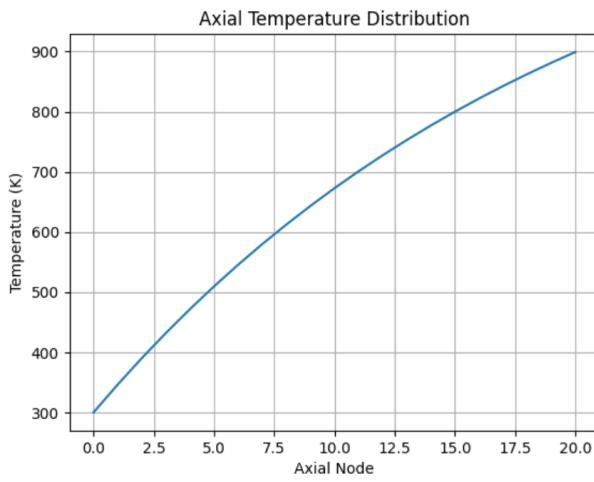


Figure 1: Pipe Temperature Profile (Water)

Axial temperature profiles show an increase along the flow direction due to internal heat generation. At the assumed conditions, transient behavior is present with no indication of being close to steady-state. The model has value in that it can focus on transient behavior, steady-state behavior, or a mix of both, depending on the user's selection of time step or thermophysical properties of the fluid.

Increasing the convective heat transfer coefficient resulted in reduced peak wall and fluid temperatures, consistent with enhanced heat removal to the surroundings. Similarly, changes in inlet temperature propagated downstream with a time delay proportional to the local flow velocity, confirming correct implementation of plug-flow behavior. A similar case with an increased heat transfer coefficient is shown below:

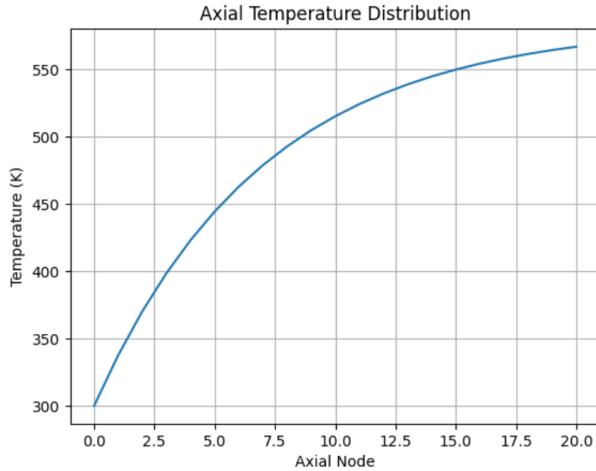


Figure 2: Pipe Temperature Profile (Water, increased HEx coeff.)

Energy balance checks were performed by comparing the net energy added through internal heat generation with the sum of energy convected out of the system and heat lost to the surroundings. The results demonstrated good agreement, with small numerical discrepancies attributable to spatial and temporal discretization. These checks provide confidence that the governing equations, boundary conditions, and numerical implementation are internally consistent.

Overall, the results demonstrate that the model captures the dominant thermal-hydraulic physics expected of a 1-D transient system and responds predictably to changes in operating conditions. This behavior indicates that the framework is suitable for further extension to more complex system-level analyses.

Model Limitations and Future Work

The current model intentionally simplifies several physical phenomena to focus on core thermal behavior. The flow is treated as one-dimensional plug flow, neglecting radial temperature gradients, axial conduction, and multidimensional effects. Fluid properties are assumed constant, and pressure drop and momentum conservation are not explicitly modeled. As a result, the model does not currently capture flow-rate feedback or coupling between thermal and hydraulic behavior.

Heat transfer to the surroundings is represented using a simplified convective heat loss model with prescribed heat transfer coefficients. While sufficient for demonstrating trends, this approach does not account for spatially varying heat transfer regimes or detailed wall conduction effects.

Future work would include coupling the energy equation with momentum conservation to model pressure drop and flow transients, incorporating temperature-dependent fluid properties, and extending the framework to represent multi-component systems. These enhancements would allow the model to support more realistic system-level transient analyses and provide a foundation for coupling with neutronic or control system models.

Conclusion

A one-dimensional transient thermal-hydraulic model was developed to simulate axial temperature evolution in a flowing system with internal heat generation and convective heat loss. The model successfully captures expected physical trends, demonstrates numerical stability, and satisfies energy conservation checks, providing confidence in its formulation and implementation.

While intentionally simplified, the model serves as a flexible and extensible framework for systems-level thermal-hydraulic analysis. Its modular structure and Python-based implementation make it well suited for future expansion to include additional physics, automation, and validation activities. Overall, this work demonstrates a practical approach to building and analyzing thermal-hydraulic models that balance physical fidelity with computational efficiency.