

Development of a Dynamic Model of a Biomass Boiler Using Flownex



Prepared by:

Pierre Mikhail Bosch
BSCP1E002

Department of Mechanical Engineering
University of Cape Town

Supervisor:

Associate Professor Wim Fuls

February 2025

Submitted to the Department of Mechanical Engineering at the University of Cape Town in partial fulfilment of the academic requirements for a Masters of Science degree in Mechanical Engineering

Key Words: Thermal inertia, transient control, level tracking, analytical, numerical modelling, boiler, Flownex

The copyright of this thesis vests in the author. No quotation from it or information derived from it is to be published without full acknowledgement of the source. The thesis is to be used for private study or non-commercial research purposes only.

Published by the University of Cape Town (UCT) in terms of the non-exclusive license granted to UCT by the author.

Abstract

The textile, distillery, and pharmaceutical commercial enterprises are among a large group of industries requiring process steam in their production processes. The shift towards renewable energy sources is extended to industrial scale steam generators, whose designs need to allow flexibility in firing renewable and carbon neutral fuel sources such as biomass.

The present work focusses on developing a dynamic model of a unique hybrid water-tube—fire-tube boiler. The numerical model has been developed using Flownex, a one-dimensional thermohydraulic simulation software with the aim of presenting credible insights into transient performance and controllability of the boiler of interest.

This study develops stand-alone numerical models of the economiser, air heater, fire-tube evaporator, and water-cooled furnace as the four main heat exchangers in the boiler. Each heat exchanger is modelled sequentially using simplified analytical methods in Mathcad followed by detailed numerical implementations in Flownex. They are then verified against available maximum continuous rating (MCR) data before being integrated into a whole-boiler flow network.

Limitations in availability of site data and availability of high-fidelity CFD furnace validation, meant implementing assumptions such as the inclusion of localised convection effects during calibration with the available site measurement data. The fine tuning of the model during calibration extended to fluid absorptivity and cooling air ratio parametric studies in search of the best calibration point which agreed reasonably well with site measurements from steady state test results.

The impact of thermal inertia from the boiler's solid steel heat exchanging surfaces is demonstrated via a comparison of the results of uncontrolled transient runs between the realistic full inertia configuration and a low-inertia configuration of the Flownex model. The full inertia model captured the thermal mass of all the steel in the system including the finned tubes of the economiser, tubes of the air heater, furnace waterwall tubes, fire-tubes, solid steel staybars and evaporator shell.

A key transient indicating parameter is the water-level inside of the evaporator. Much effort was therefore spent discretising the evaporator water volume to provide accurate level-tracking during transient simulations in the Flownex simulation environment. The transient operational scenarios investigated included a stepped steam demand profile and stepped fuel moisture profile with active boiler pressure and level control. The methodology and results establish a foundation for supplementary control optimisation investigations for future studies.

Despite some data limitations for validated transient studies, this work presents a high fidelity controllable dynamic model of the hybrid boiler, which after validation can be used to emulate real world controllable boiler operations.

Declaration

I know the meaning of plagiarism and declare that all the work in the document, except for that which is properly acknowledged, is my own. This thesis/dissertation has been submitted to the Turnitin module (or equivalent similarity and originality checking software) and I confirm that my supervisor has seen my report and any concerns revealed by such have been resolved with my supervisor.

Signed by candidate

Pierre Mikhail Bosch

14 February 2025

Acknowledgements

First and foremost, I would like to express my sincere gratitude to my supervisor, Associate Professor Fuls, for his invaluable guidance throughout the course of this project. His extensive industry expertise in power generation and boilers, along with his deep knowledge of Flownex, provided critical insights that greatly contributed to the successful completion of this research.

I also extend my appreciation to the ATProM academic team, including Professor Pieter Rousseau, Dr. Leon Malan, and Mr. Priyesh Gosai, for their constructive feedback and guidance during the monthly colloquium presentations. Their input played a pivotal role in refining the content and direction of this work.

A special thanks to John Thompson Boilers for partially funding this study and for granting me the opportunity to conduct focused research on one of their boilers. Their generosity in providing access to detailed workshop drawings, maximum continuous rating (MCR) data, and site measurements was instrumental in deepening the scope of this research. In particular, I would like to acknowledge Ettiene de Villiers for engaging with me on how this research could add value to their company, Jacques Matolla for his invaluable discussions and feedback on preliminary findings, and Chris Knutsen for his additional insights into site measurement data and the techniques employed in the field.

I am also grateful to merSETA and TRAJECTS for their financial support, which enabled the successful execution of this project.

Finally, I would like to express my heartfelt appreciation to my partner, mother, father, and brother for their unwavering support throughout this journey. Their encouragement, patience, and belief in me were essential in seeing this project through to completion.

Table of Contents

List of Figures	vi
List of Tables.....	xiii
List of Nomenclature.....	xiv
1. Introduction	16
1.1 Background	16
1.2 Purpose and Problem Statement.....	17
1.3 Goals and Objectives.....	17
1.4 Constraints and Project Boundaries	18
1.5 Report Roadmap	19
2. Literature Review	21
2.1 Fire-tube Boilers.....	21
2.2 Water-tube Boilers.....	22
2.3 Three Pass Hybrid Water-Tube–Fire-Tube Boiler	24
2.4 Plant of Interest Description: TORRIPAC.....	24
2.5 Boiler Modelling Methods	27
2.6 Dynamic Modelling of Power Plants	32
2.7 Summary of findings	39
3. Theory	40
3.1 Heat Transfer Theory	40
3.2 Furnace Modelling	47
3.3 Flownex Theory.....	50
4. Model Development & Verification.....	72
4.1 Analytical Model Development (Mathcad).....	73
4.2 Steady State Numerical Model Development (Flownex)	85
4.3 Inclusion of Thermal Inertia	110
4.4 Steady State Model Calibration and Fine-tuning using Site Data.....	114
4.5 Controller Development	129
4.6 Overview of Integrated Whole-Boiler Model	138
5. Model Application.....	140

5.1	Steady State Operating Conditions.....	140
5.2	Grid and Time Step Sensitivity	140
5.3	Thermal Inertia Studies.....	145
5.4	Transient Studies with Active Control.....	147
6.	Conclusions and Future Work.....	153
6.1	Summary & Conclusions	153
6.2	Project Success.....	155
6.3	Recommendations for Future Work	157
7.	List of References.....	158
Appendix A.	Economiser Verification Details.....	160
Appendix B.	Air Heater Verification Details	182
Appendix C.	Fire-Tube Evaporator Verification Details	218
Appendix D.	Water-Cooled Furnace “Example” Calculation.....	245
Appendix E.	Steady State Calibration and Fine Tuning Details (Parametric Studies)	261
Appendix F.	Off Design Points.....	292

List of Figures

Figure 1: HRT boiler architecture [3]	21
Figure 2: Flue gas flow patterns of a fire-tube boiler (a) two-pass, (b) three-pass, (c) four pass [2]	21
Figure 3: 620 MWe pulverised coal-fired boiler general arrangement [5].....	22
Figure 4: 105 t/h bagasse- coal-fired boiler for cogeneration [6]	23
Figure 5: 3D CAD Model of the 4ton/hr TORRIPAC Unit.....	25
Figure 6: TORRIPAC Process Flow Diagram (PFD).....	26
Figure 7: Process Flow Diagram (PFD) of a generic boiler layout, adapted from [7]	28
Figure 8: Generic heat exchanger model illustrating convective & radiative heat transfer phenomena and energy flow paths [7]	31
Figure 9: Wet back fire-tube boiler geometry based on the Johnston PFTA 500-4 Boiler [1]	36
Figure 10: Flownex model of boiler tube heat transfer segment [1]	36
Figure 11: Complete Flownex fire-tube boiler model [1]	38
Figure 12: Boiler steam demand input profile (left), level response (middle), and pressure response (right) [1]	38
Figure 13: Finned-tube arrangement illustrating the far field velocity at the inlet and the bundle velocity at the minimum flow cross-section (between the finned tubes)	44
Figure 14: Combustion zone incoming and outgoing process streams.....	47
Figure 15: Energy balance on whole-boiler control volume.....	48
Figure 16: Implicit Pressure Correction Method (IPCM) algorithm adapted from [15]	51
Figure 17: Flownex Charts and Lookup Tables in the Libraries window.....	52
Figure 18: "Coal-fired flue gas" custom fluid in ATProM database	52
Figure 19: Carbon steel – AISI 1010 {Master Database} (left) vs custom low inertia steel {Project Database} (right)	53
Figure 20: Multiple view functionality	54
Figure 21: Excel Workbook component in Flownex	55
Figure 22: Quick Script editor (left) & Script editor (right)	55
Figure 23: Example of Data Transfer Link property connections between components.....	56

Figure 24: Sensitivity Analysis set-up for furnace calibration study.....	56
Figure 25: Action Set-up for pressure controller test.....	57
Figure 26: Simple PID input & output parameters	58
Figure 27: Flow Resistance component connecting two nodes in Flownex.....	59
Figure 28: Boundary Condition & Flow Node visualisation in Flownex with input properties shown	61
Figure 29: Example of Two-Phase Tank implementation for a horizontal cylindrical volume using height fraction specifications for branch connections	63
Figure 30: Flownex Level Tracking capability illustration copied from Flownex Library Manual [18]	63
Figure 31: Pipe component connecting two nodes in Flownex.....	64
Figure 32: Tube roughness selection in Flownex for Pipe component.....	64
Figure 33: Pipe database in Flownex containing industry standard pipe specifications.....	65
Figure 34: Finned-Tube Heat Exchanger component in Flownex.....	65
Figure 35: Flownex Finned-Tube Heat Exchanger circuiting convention taken from Flownex Library Manual [18].....	66
Figure 36: Flownex discretisation scheme for Finned-Tube Heat Exchanger taken from Flownex Library Manual [18].....	66
Figure 37: Finned-Tube Heat Exchanger tube arrangement convention, Schematic taken from Flownex Library Manual [18]	67
Figure 38: Finned-Tube Heat Exchanger overall Flow regime convention, Schematic taken from Flownex Library Manual [18]	67
Figure 39: Convection connection configurations and input property options	67
Figure 40: Example of CHT implementation in Flownex showing conduction layers for convective heat transfer in fire-tube evaporator	68
Figure 41: User input properties for Fluid Radiation component	71
Figure 42: High level methodology outline.....	72
Figure 43: Economiser general arrangement	74
Figure 44: “H-type” fin geometry and far-field and free-flow areas (modular basis)	74

Figure 45: Economiser successive approximation algorithm used in Mathcad analytical model.....	76
Figure 46: Air heater general arrangement	77
Figure 47: Discretised air heater arrangement – air flow (0-3) and flue gas flow (A-D)	77
Figure 48: Air heater far-field and free-flow areas	78
Figure 49: Air heater successive approximation algorithm used in Mathcad analytical model.....	79
Figure 50: Fire-tube evaporator general arrangement	80
Figure 51: Close-up view of saturated liquid surrounding the fire-tubes	81
Figure 52: Fire-tube evaporator successive approximation algorithm used in Mathcad analytical model	81
Figure 53: Water-cooled furnace general arrangement and fluid flow paths.....	82
Figure 54: Adiabatic Flame Temperature (AFT) energy balance in combustion zone control volume	83
Figure 55: Economiser Flownex model (left), Quick Script inputs & outputs (right).....	85
Figure 56: Finned-Tube Heat Exchanger inputs for fin-side (left) & tube-side (right).....	86
Figure 57: True TORRIPAC economiser tube arrangement	86
Figure 58: Flownex tube arrangement convention	86
Figure 59: TORRIPAC economiser assembly flipped 90 degrees to show counter-flow regime	87
Figure 60: Flownex convention for overall flow regime for cross-flow over tube banks.....	87
Figure 61: Flue gas composition employed in Flownex 2 model based on an assumed fuel composition	88
Figure 62: Sub-heat exchanger (HX1) illustrating conformance to the Flownex tube arrangement convention	89
Figure 63: Air heater Flownex model (left), Quick Script inputs & outputs (right)	90
Figure 64: Finned-Tube Heat Exchanger inputs for HX1 fin-side (left) and tube-side (right).....	91
Figure 65: Fire-tube evaporator Flownex model	93
Figure 66: Right view of true volume shape (modelled in SolidWorks)	94
Figure 67: First three entries for Polygon Table of Two-Phase Tank in Flownex	94
Figure 68: Right view visual representation of Flownex Polygon implementation.....	94

Figure 69: Water level meter installed onto evaporator of TORRIPAC	95
Figure 70: Flownex Two-Phase Tank level sensitivity results	95
Figure 71: Inputs for downcomer Pipe component in Flownex evaporator model	96
Figure 72: Two-Phase Tank branch height specification	97
Figure 73: Composite Heat Transfer – convective property inputs.....	99
Figure 74: Complete fire-tube evaporator Flownex model showing key inputs & results.....	100
Figure 75: ATProM “Combust” excel workbook in Flownex showing some links between the input and output cells.....	104
Figure 76: Water-cooled furnace Flownex model	105
Figure 77: Flue gas enthalpy correction - Iterative Script.....	107
Figure 78: Flow Resistance inputs for heated air stream	108
Figure 79: Furnace Flownex Model (combustion side) indicating mass source boundary conditions and fixed mass flow elements	108
Figure 80: Furnace Flownex model (radiation side)	109
Figure 81: Model update from Flow Resistance in stand-alone furnace model (left) to detailed Pipe geometries in integrated model (right)	111
Figure 82: Components used to capture solid material thermal inertia	112
Figure 83: Furnace model update: waterwall steel thermal inertia.....	113
Figure 84: Integrated whole boiler model for TORRIPAC (uncalibrated)	115
Figure 85: Calibration step 1 - results (left) and algorithm (right).....	118
Figure 86: Calibration step 2 - AFR calibration targeting O ₂ volume fraction	119
Figure 87: Calibration step 3 – Fuel flow calibration to reduce tank energy source to within 1% of combustion heat	120
Figure 88: Sensitivity Analysis set-up for C-factor parametric study.....	121
Figure 89: Furnace radiation calibration @10% cooling air and radiation with fluid emissivity =0.6 & absorptivity=1.0	122
Figure 90: Effective furnace convection area approximation	123
Figure 91: Furnace convection addition to Flownex model with CHT components.....	123

<i>Figure 92: Furnace radiation calibration @10% cooling air and radiation with fluid emissivity =0.6 & absorptivity=1.0</i>	124
<i>Figure 93: Furnace radiation calibration @10% Cooling air and radiation with fluid emissivity =0.6 & absorptivity=0.6</i>	124
Figure 94: Furnace radiation calibration @10% cooling air and radiation with fluid emissivity =0.6 & absorptivity=0.6 Plus convection @ $h=25 \text{ W/m}^2\text{K}$	125
Figure 95: Furnace radiation calibration @25% cooling air and radiation with fluid emissivity =0.6 & absorptivity=0.6 Plus convection @ $h=25 \text{ W/m}^2\text{K}$	125
Figure 96: Absolute %error vs. C-factor for 25% cooling air, furnace radiation (emissivity=absorptivity=0.6) & furnace convection @ $h= 25 \text{ W/m}^2\text{K}$	126
Figure 97: Final calibration results – normalised weighted error vs. C-factor	127
Figure 98: Box & Whisker Plot illustrating where model temperatures fall within the standard deviation of the site measurements taken over the 1-hour testing period.....	128
Figure 99: Vibrating grate and box ram operation, [19].....	129
Figure 100: Comparison between real TORRIPAC control system (left) and Flownex control system (right).....	131
Figure 101: Pressure controller tuning @ $dt=1s$ - Pressure Switch	132
Figure 102: Pressure controller tuning @ $dt=1s$ - Gains.....	132
Figure 103: Pressure controller tuning @ $dt=1s$ - Pressure Tracking	132
Figure 104: Pressure controller tuning @ $dt=1s$ – Controller Output (fuel flow)	132
Figure 105: Pressure controller tuning @ $dt=1s$ - Two-Phase Tank pressure & level response	133
Figure 106: Pressure control verification for 250 kPa set-point changes ($dt=1s$)	134
Figure 107: Time step sensitivity for pressure controller	134
Figure 108: Pressure control verification for 250 kPa set-point changes ($dt=5s$)	135
Figure 109: Level controller tuning - Level Switch	136
Figure 110: Level controller tuning - Gains.....	136
Figure 111: Level controller tuning – Level Tracking	136
Figure 112: Level controller tuning – controller output (feedwater flow)	136
Figure 113: Level controller tuning - Two-Phase Tank pressure & level response.....	136

Figure 114: Level control verification for 10 mm set-point changes.....	137
Figure 115: Complete integrated controllable TORRIPAC boiler Flownex model	139
Figure 116: Grid & Time Step Sensitivity Case Study: Combined radiative and convective heat transfer to the left furnace waterwall	141
Figure 117: Wall thickness grid sensitivity effect on exit steam quality (0-120s)	142
Figure 118: Wall thickness grid sensitivity effects on exit steam quality (9.5-15s).....	142
Figure 119: Wall thickness grid sensitivity effect on average outer tube wall temperature (9.5-15s)	142
Figure 120: Wall thickness grid sensitivity effect on average inner tube wall temperature (9.5-15s)	142
Figure 121: Time step sensitivity effects on waterwall steam exit quality (0-400s)	143
Figure 122: Effect of the Alpha Crank-Nicolson factor at large time steps	143
Figure 123: Time step sensitivity effects on waterwall steam exit pressure (9.9-10.3s)	144
Figure 124: Fuel flow input profile.....	146
Figure 125: Two-Phase Tank pressure response for fuel step changes (Full Inertia & Low Inertia configuration).....	146
Figure 126: Two-Phase Tank level response for fuel step changes (Full Inertia & Low Inertia configuration).....	146
Figure 127: Steam extract input profile	147
Figure 128: Two-Phase Tank pressure response for steam step changes (Full Inertia & Low Inertia configuration).....	147
Figure 129: Two-Phase Tank level response for steam step changes (Full Inertia & Low Inertia configuration).....	147
Figure 130: Boiler steam pressure response to fuel moisture changes	148
Figure 131: Boiler level response to fuel moisture changes.....	148
Figure 132: Boiler steam pressure response to fuel mass flow changes.....	148
Figure 133: Boiler feedwater response to level changes.....	148
Figure 134: Flue gas temperatures upstream of furnace exit	149
Figure 135: Flue gas temperatures	149

Figure 136: Combustion & radiation heat transfer rates	149
Figure 137: Convection heat transfer rates	149
Figure 138: Boiler steam pressure response to steam demand profile	150
Figure 139: Boiler level response to steam demand profile.....	150
Figure 140: Fuel & gas mass flowrates	151
Figure 141: Feedwater flowrate referenced to steam extraction flowrate	151
Figure 142: Fuel & air temperatures.....	151
Figure 143: Water temperatures	151
Figure 144: Flue gas temperatures upstream of furnace exit	152
Figure 145: Flue gas temperatures	152
Figure 146: Combustion & radiation heat transfer rates	152
Figure 147: Convection heat transfer rates	152
Figure 148: Furnace tube wall temperatures	152
Figure 149: Fire-tube wall temperatures.....	152

List of Tables

Table 1: Economiser verification results - fluid exit temperatures and heat transfer rates	87
Table 2: Economiser flue gas exit temperature [K] %error pairwise comparison	88
Table 3: Economiser water exit temperature [K] %error pairwise comparison	88
Table 4: Economiser heat transfer rate [kW] %error pairwise comparison	88
Table 5: Air heater verification results - fluid exit temperatures and heat transfer rates	91
Table 6: Air heater air temperature [K] %error pairwise comparison	92
Table 7: Air heater flue gas exit temperature [K] %error pairwise comparison	92
Table 8: Air heater heat transfer rate [kW] %error pairwise comparison	92
Table 9: Evaporator verification results – flue gas temperature, tube wall temperature, and evaporator heat transfer	101
Table 10: Evaporator flue gas exit temperature [K] %error pairwise comparison	102
Table 11: Evaporator fire-tube wall temperature [K] %error pairwise comparison	102
Table 12: Evaporator heat transfer rate [kW] %error pairwise comparison	102
Table 13: Heat transfer rate [kW] %errors between Flownex 2 and Flownex 1 models	102
<i>Table 14: Ultimate Bagasse Analysis</i>	<i>106</i>
<i>Table 15: Ultimate Bagasse Analysis (updated 2021)</i>	<i>106</i>
<i>Table 16: Fluid stream compositions applied for the MCR load case</i>	<i>106</i>
Table 17: Steel inertia (mass)	113
Table 18: Fluid inertia (mass)	114
Table 19: Raw Data from 2.8 tonne/hr Test Case on TORRIPAC	116
Table 20: Global inputs and assumptions for whole boiler calibration initialisation	117
Table 21: Pairwise comparison for confidence rankings of site measurements	126
Table 22: Site measurements vs. model for 70% (2.8 tonne/hr) load case	128
Table 23: Pressure controller tuned values (dt=1s)	134
Table 24: Pressure controller tuned values (dt=5s)	135
Table 25: Level controller tuned values	137

List of Nomenclature

General symbols

$\dot{Q}$	Heat Transfer Rate	U	Overall heat transfer coefficient
UA	Overall conductance (overall heat transfer coefficient with area)	t	Time or thickness
C_{min}	Minimum heat capacity (between two separate fluid streams)	C_{max}	Maximum heat capacity (between two separate fluid streams)
C_r	Heat capacity ratio between two streams (minimum: maximum)	L_{char}	Characteristic length
h	Local convective heat transfer coefficient / Fluid enthalpy	k	Thermal conductivity
h_0	Total enthalpy	$d_{o,tube}$	Tube outer diameter
M	Molar mass	X	Molar fraction
$\dot{m}$	Mass flowrate	S_L	Longitudinal fin pitch
S_T	Transverse fin pitch	HX	Heat Exchanger
EC or ECON	Economiser	AH	Air Heater
EV or EVAP	Evaporator	FURN	Furnace
K_p	Proportional gain	K_I	Integral gain
K_D	Differential gain	A	Area
SA	Surface Area	Re	Reynolds number
Pr	Prandtl number	Ra	Raleigh number
Gr	Grashof number	Nu	Nusselt number
f	Friction factor	g	Gravitational acceleration
V	Volume	v	Velocity

Greek symbols

ε	Effectiveness/Surface roughness/ Emissivity	ρ	Density
μ	Dynamic viscosity	ν	Kinematic viscosity
σ	Stefan-Boltzmann constant	δ	Partial differential
τ	Time constant	β	Volume expansion coefficient
α_{CN}	Alpha Crank-Nicolson Factor	Δt	Time step size/ change in time

Acronyms and Abbreviations

ATProM	Applied Thermofluid Process Modelling Research Unit	CFD	Computational Fluid Dynamics
MCR	Maximum Continuous Rating	PFD	Process Flow Diagram
HRT	Horizontal Return Tubular (boiler)	PPSD	Power Plant Simulator & Designer
NTU	Number of Transfer Units	AFT	Adiabatic Flame Temperature
LMTD	Log Mean Temperature Difference	ROS	Ratio of Surfaces
SCOF	Straightened Cross-Section of Flow	UFA	Ultimate fuel analysis
UBA	Ultimate Bagasse Analysis	AFR	Air-fuel-ratio
HMI	Human Machine Interface	FET	Furnace Exit Temperature
HHV	Higher Heating Value	CV	Calorific Value
DCS	Distributed Control System	CHT	Composite Heat Transfer
SR	Split Ratio	O&M	Operations and Maintenance
LPG	Liquified petroleum gas	OD	Outer diameter
PID	Proportional-Integral-Derivative	UC	Unburnt Carbon
PA	Primary Air	SA	Secondary Air

Subscripts

rad	radiation	conv	Convection
W	Wall	fg	Flue gas
CA	Cooling air	HA	Heated air
COMBA	Combustion air	fg	Flue gas
fw	Feedwater	amb	Ambient

1. Introduction

The Applied Thermofluid Process Modelling Research Unit (ATProM) specialises in modelling complex thermofluid energy systems relevant to both global needs as well as within the South African context. As such, ATProM has developed relationships with South African industry leaders in energy production such as Eskom and John Thompson Boilers with the aim of modelling their various technologies to assess the effectiveness and efficiency of existing designs with an emphasis on providing necessary insights for improvements on new designs.

The present work deals with developing a dynamic simulation model for an existing unique hybrid biomass fired boiler part of John Thompson's TORRIPAC range to quantify its performance and its ability to respond to fluctuating operating conditions. The Flownex Simulation Environment software used in the present work is a one-dimensional thermofluid network solving software which has been identified as an excellent tool for the job due to its computational efficiency and high-fidelity heat transfer and fluid flow components.

1.1 Background

The worldwide shift towards renewable energy technologies and fuel sources is not limited to large scale power generation only. Industrial steam generators (boilers) also have far-reaching applications for pure process steam generation for various industries, including textiles, pharmaceuticals and distilleries to name a few. It is increasingly important that these industrial boilers also make strides in their designs to accommodate the firing of renewable or carbon-neutral energies such as biomass.

The nature of energy and process steam demand is an erratic one, where various industries have different steam requirements at different times depending on the nature of factory operations. A factory for example could be using the process steam at a certain operational rate but then need to ramp production up/down as efficiently as possible. This means that the boiler supplying the steam would need to be able to respond to the demand change in the most time, cost, and energy efficient way possible.

The reality of transient boiler operation is that different boiler designs have different thermal inertia depending on their scale and the mass of their heat exchanging materials (usually carbon steel). This means that the effectiveness of a boiler in moving from one steady load case to the next needs to be informed by optimised control systems based on the transient heat transfer and fluid flow characteristics of the boiler which again is a function of the thermal inertia of the system. The

thermal inertia and optimised boiler controls is also increasingly important for the overall control philosophy and safety interlocks of boiler systems.

Steady state models, including CFD models are generally used to evaluate the efficiency of boilers and to verify their ability to meet a few common discrete load cases such as maximum continuous rating (MCR) and half load. These models however are unable to capture the transient effects of common operational scenarios which are imminent in real life applications. This is where the development of numerical models with robust transient capabilities are crucial to assess both the feasibility of boiler designs as well as to optimise their performance for a range of common operational scenarios.

1.2 Purpose and Problem Statement

The purpose of this project is to develop a dynamic model for a biomass fired boiler using Flownex. This research pertains specifically to a hybrid water-tube fire-tube boiler which is part of John Thompson's TORRIPAC range.

Prior to this research, a dynamic model of the TORRIPAC did not exist. Consequently, a means of predicting its transient performance and the fidelity of the existing control philosophies is not possible. The development of the dynamic model in the present work will therefore enable various control and transient studies, which can form the starting point for optimised design and control philosophy improvements in future studies.

The task was to sequentially develop standalone models for each heat exchanger starting with simple analytical methods followed by numerical implementation in Flownex for each of the four heat exchangers, and finally the integration into a single whole boiler Flownex model which can be calibrated with available site measurements. Thereafter the calibrated model can be used as starting point for transient investigations providing credible insights into the boiler's flexibility and transient performance.

1.3 Goals and Objectives

The main objective of the present work is to develop a robust, fully integrated, dynamic boiler model for a 4T/hr TORRIPAC unit complete with simplified active control loops. The realisation of this objective can form the basis for detailed control optimisation studies through which a fully integrated boiler simulator can be developed to investigate optimum transient boiler performance. The simulator could then be used as a tool to test new control philosophies for very specific transient scenarios. The TORRIPAC boiler consists of four main heat exchangers: economiser, air heater, fire-tube evaporator and water-cooled furnace.

The lower-level goals needed to accomplish the main objective are itemised as follows:

- Develop simplified verifiable analytical models for each heat exchanger using Mathcad software.
- Develop detailed numerical Flownex models for each heat exchanger verifiable with the analytical model and a set of maximum continuous rating (MCR) data obtained from John Thompson.
- Develop a methodology to calibrate and fine tune the integrated whole boiler model using site measurement data.
- Demonstrate the effect of solid material thermal inertia on the transient response of the TORRIPAC through a comparison of transient results from a low inertia configuration of the model and a full inertia model including accurate thermal masses of all solid material in the boiler system.
- Demonstrate the suitability of the model to be used for dynamic simulations containing simplified control loops.

1.4 Constraints and Project Boundaries

The scope of the research involves dynamic modelling of an integrated whole boiler. The system control volume was limited only to the four heat exchangers in the boiler system. Auxiliary equipment such as feedwater pumps, supply air fans, forced draft fans and grit collectors were not included. Instead, suitable boundary conditions were employed at the boundary of the boiler control volume to emulate conditions such as feedwater pump exit, furnace draught pressure, etc.

Detailed 3-dimensional CFD was not employed for validation of the furnace model as this is the topic of a separate study being investigated by John Thompson. Instead, some educated assumptions were made to incorporate convection effects inside of the furnace in order to better calibrate the furnace model with site data. Since no CFD results are available, the exact convection characterisation around the large nose in the furnace could not be validated, but the assumptions were made with valid thermodynamic theory based on typical industry values.

The project employed combustion models from the ATProM research library which have been proven valid and generic for any fuel composition. This was in the form of an Excel component connected into the Flownex model. As such, the combustion model and theory are not presented in detail in this report, though it was studied and tested analytically for personal verification.

Simple pressure drop models (Darcy-Weisbach) were used for connecting piping and flow resistances. The natural circulation flowrate does not have a large impact on the heat transfer in the boiler. Additionally, no data exists to validate the true circulation rate. As such the circulation rate

was fixed to be equal to the main steam flowrate in the stand-alone evaporator model and then set equal to a multiple of 10 in the integrated model to mitigate the possibility of dry-out in the furnace walls. Care was taken to include accurate pipe geometries and specifications in the integrated model true to the workshop drawings purely for the capture of the steel mass of the waterwalls.

Due to incomplete information in the available MCR data as well as the site test data, assumptions had to be made in the progression of the model development through verification of the standalone heat exchanger models with MCR data and the integrated boiler calibration with site data.

The economiser and air heater model verification with site data were straight forward. On the other hand, the MCR data lacked a furnace exit condition as well as information about the fuel (bagasse) composition and consequent air-fuel-ratio which were used to generate the MCR data. This meant that the analytical and Flownex stand-alone models for the fire-tube evaporator and water-cooled furnace could not be directly verified with the MCR data since the furnace exit condition links the flue gas sides of these two centralised heat exchangers.

With the focus directed toward dynamic modelling, fine details of the combustion and radiation phenomena in the furnace, including view factors, and highly empirical back scatter and view factor correlations were not included. Instead, a simplified direct radiation model was employed and calibrated using site data as a strong function of an effective view factor (later referred to as “C-factor”). The idea being that for accurate inputs for the furnace surface area, and typical gas emissivity value inputs, together with an over estimation of the wall emissivity, that the furnace radiation contribution could be tuned with a single factor to target site test conditions.

It will become apparent in the calibration section (4.4) that more insight through high-fidelity CFD or further detailed plant data could have better informed the calibration. But this is not in the project scope.

The existing TORRIPAC control system was not modelled in detail. The suitability of the model to emulate controllable transient operational scenarios is demonstrated through a simplified 2-loop PID control implementation but the exact expected response times could not be validated due a lack of transient site data. This should mark a starting point for further studies on optimised controller design and transient performance. No claims will be made about the accuracy of the transient analysis due to a lack of validation data.

1.5 Report Roadmap

The format of this report begins with a literature review in chapter 2, introducing the two-main boiler architectures, namely fire-tube boilers and water-tube boilers to provide necessary insights into the TORRIPAC boiler which is a synergy of both designs. The TORRIPAC plant description is

presented next providing context for the theory presented in the next chapter, and to form a solid foundation for the detailed model development and verification. Common boiler heat exchanger modelling techniques used in literature are discussed, followed by a review of the progress in the development of dynamic boiler models. A review of a dynamic Flownex model for a fire-tube boiler developed by P. Klein et al. [1] draws the literature review to a close.

Chapter 3 begins by introducing the relevant heat transfer theory in section 3.1 as used in the development of the heat exchanger models in the present work. This includes Nusselt number correlations as well as the applicable effectiveness number of transfer units (e-NTU) methods. Section 3.3 provides a brief description of the over-arching mass, momentum, and energy balance solving algorithm employed in the Flownex software. The reader is familiarised with various user interfaces, tools, and techniques offered within the Flownex software to aid in their understanding of the details involved in the later chapters. This also includes the theory behind the relevant Flownex flow components (*Nodes, Boundary Conditions, Pipes*, etc.) which are later used in the development of the various heat exchanger models and the final integrated whole boiler model.

Chapter 4 demonstrates the natural progression in the modelling methodology starting with the analytical Mathcad model development in section 4.1, followed by the numerical Flownex model development with accompanying MCR verification in section 4.2. These two subsections of chapter 4 demonstrate a diligent methodology which allows the ability for one to fall back on previous simplified models to verify and fault find when increasing the complexity in the integrated model. Section 4.3 details the methods employed to include the solid steel thermal mass of all the heat exchanging materials. The steady state model calibration and fine-tuning using site test data is presented in section 4.4. Controller development details follow in section 4.5 with the closing section (4.6) illustrating the fully integrated whole boiler model from a step back, highlighting free parameters needed in the model calibration and illustrating its functionality in investigating transient events and recording data through line graphs.

The model application in chapter 5 presents results from uncontrolled thermal inertia studies as well as transient studies with active control proving the model's suitability in emulating two real world operational scenarios implemented through a fuel moisture input profile and steam demand input profile. Also included in this chapter is a brief case study demonstrating the importance and implications of grid and time step sensitivity studies in the context of dynamic modelling.

Finally, chapter 6 rounds up the report with main conclusions reflecting on the success of the project with relation to the project goals as well as recommendations for future studies.

2. Literature Review

2.1 Fire-tube Boilers

Fire-tube boilers are designed such that combustion gases pass through tubes, which are immersed in a large volume of water. The combustion flue gas transfers heat through the walls of the tubes to heat up the surrounding water, ultimately converting it to steam. F. J. Gutiérrez Ortiz [2] explains in his paper, “Modeling of fire-tube boilers” that these types of boilers can be characterised by the number of passes the flue gas makes along the length of the boiler, with each pass through a new set of tubes involving a reversal of the flow direction from the preceding pass. The large water volume surrounding the fire-tubes is slow to reach operating pressure and temperature, but F. J. Gutiérrez Ortiz [2] explains that the characteristically high thermal inertia contained in the water is able to support rapid steam demand changes without large variations in the main steam pressure and without the need for rapid firing response.

The first horizontal return tubular (HRT) boilers were externally fired with the entire shell being enclosed in brickwork to contain the flame, as per Figure 1. Modern designs have enlarged the diameter of the return gas passes such that the firing grate is housed inside the shell. This eliminated the need for a brickwork enclosure. In this setup the hot flue gasses flow from the furnace cavity to the reversing chamber at the rear of the boiler and then forward again through an array of smaller diameter fire-tubes [2]. The smaller diameter tubes on the subsequent passes after the large furnace shell promote an increased rate of heat transfer due to the high flow velocities through the smaller cross-sectional area of the tubes. Figure 2 illustrates typical flue gas flow patterns for modern fire-tube boilers in accordance with the number of passes.

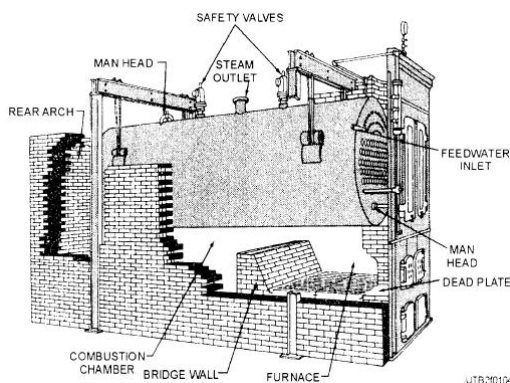


Figure 1: HRT boiler architecture [3]

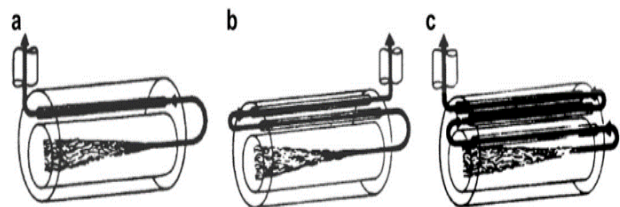


Figure 2: Flue gas flow patterns of a fire-tube boiler (a) two-pass, (b) three-pass, (c) four pass [2]

For applications with limited floor space, there exists the vertical fire tube-boiler which has a similar construction to its horizontal counterpart. The vertical fire tube boilers are also made of a round

steel shell closed by tube sheets on either end. Water surrounds the vertical fire tubes which make a single pass through the boiler from the furnace at the bottom through to the chimney [2].

D. Aguilar Vizcarra, D. Esenarro, and C. Rodriguez [4] explain that the two main types of boilers are water-tube boilers and fire-tube (pyro tubular) boilers, where the former is mainly used in large steam power plants which generate large quantities of steam at high pressures. Fire-tube boilers on the other hand are used in thermal processes where the steam pressure requirements are low and consequently the steam temperatures are also low [4].

2.2 Water-tube Boilers

Water-tube boilers circulate water through tubes which are heated by combustion gasses. The tubes are welded together to form membrane-type waterwalls which receive water via downcomer pipes from a steam drum. This boiler type is most common for large-scale power generation since they include additional convective heat exchangers downstream of the combustion zone to superheat the steam from the drum such that it can be used to drive a turbine coupled to a generator. These boilers also include a feedwater pre-heater known as an economiser and may also include reheaters. The architecture of a coal-fired 620 MWe water-tube boiler can be seen in Figure 3.

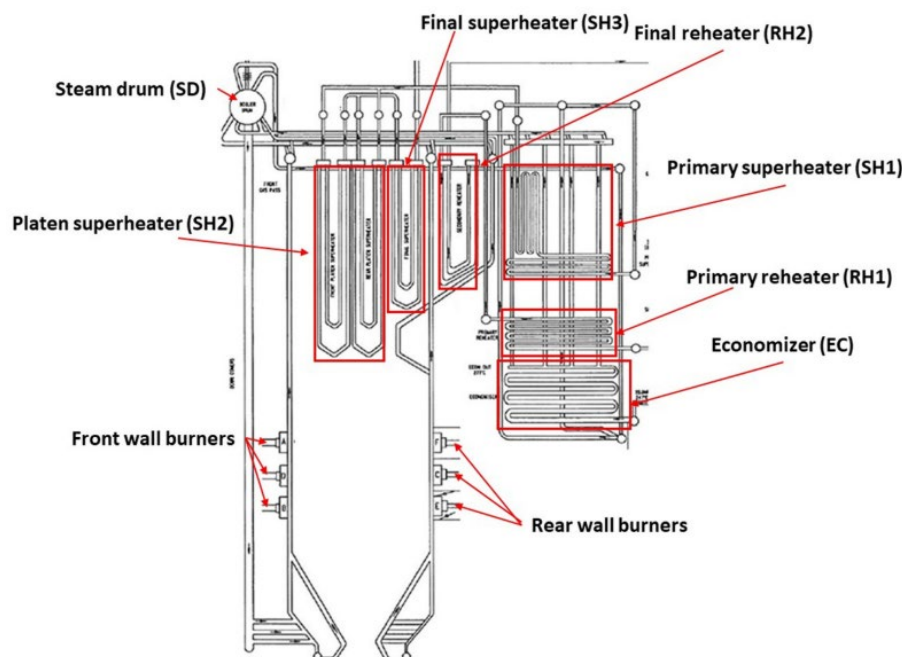


Figure 3: 620 MWe pulverised coal-fired boiler general arrangement [5]

The economiser and reheaters increase the boilers thermal efficiency by recovering heat from the flue gas in the back pass that would otherwise have been wasted. The reheater also improves steam quality such that condensation does not occur, thereby mitigating damage to the turbine blades. Note that the pulverised coal-fired boiler illustrated in Figure 3 uses burners to heat the water in the water-wall tubes.

Water-tube boilers can also fire solid fuels such as solid coal, wood pellets, and bagasse. This can be achieved by injecting combustion air into the water-tube furnace from below the grate on which the solid fuel is propelled. The architecture for a solid-fuel firing water-tube boiler is similar to the pulverised coal boiler but the burners are replaced by a grate as illustrated for a 105 tonne per hour coal-bagasse cogenerating boiler in Figure 4.

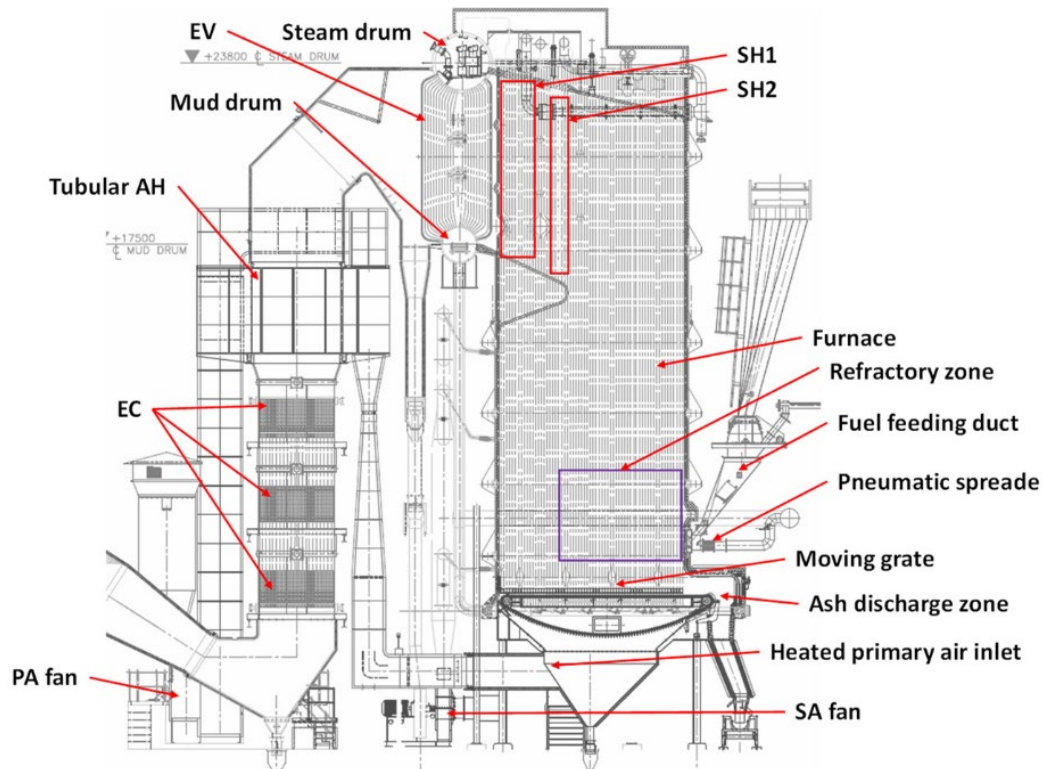


Figure 4: 105 t/h bagasse- coal-fired boiler for cogeneration [6]

2.2.1 Natural Circulation Boilers

The natural circulation process occurs when subcooled water mixes with saturated liquid water from the bottom of the steam drum of a water-tube boiler. This mixture flows down the downcomer pipes of the steam drum to the bottom of the boiler where it enters the membrane wall tubes which enclose the furnace as detailed by P. Rousseau, R. Laubscher, and B. T. Rawlins [7]. The saturated vapour on the other hand exits the steam drum at the top as it flows towards the superheaters. The natural circulation phenomenon therefore occurs due to the difference in steam and water density between the water-wall and downcomer pipes resulting in a continuous loop of re-circulation driven by natural convection [7].

2.2.2 Once-Through Boilers

Once-through boilers are another type of water-tube boiler which does not contain a steam drum and consequently there is no circulation loop. The water is fed at high pressure into the boiler tubes

where it heats up and is gradually converted to steam in a single pass from the bottom to the top of the waterwalls. P. Rousseau, R. Laubscher, and B. T. Rawlins [7] explain that although there is no steam drum, there are small separators and collector vessels through which the water passes. This water is pumped back through the economiser and waterwalls at low-load conditions such as start-up scenarios. This type of boiler is typically found in supercritical power plants.

2.3 Three Pass Hybrid Water-Tube–Fire-Tube Boiler

D. Aguilar Vizcarra, D. Esenarro, and C. Rodriguez [4] modelled a vertical hybrid water-tube and fire-tube boiler thereby investigating a technological innovation. The authors report on the differences between 4 vertical boilers of which the first three are existing technologies and the fourth is the three-pass hybrid boiler which is the topic of their research. The differences between these boilers were quantified in terms of the temperature of the flue gas emitted to the environment which can be related to the thermal efficiency of each boiler.

The flue gas temperatures for each boiler as reported by D. Aguilar Vizcarra et al. [4] were as follows; 500°C for a diesel II fired vertical single-pass hot water boiler, between 250°C and 300°C for a diesel oil fired vertical two-pass boiler, 300°C for a two-pass (tubeless boiler), and 185°C for the innovative three-pass boiler firing diesel oil.

Modelling techniques are not presented in [4]. Instead, the authors report on results obtained from real-life operation of each boiler. The results from the three-pass hybrid boiler with heat transfer surface areas equivalent to boilers of one and two passes, for the same boiler capacity indicated higher utilisation levels of the combustion heat and therefore optimised the thermal efficiency. Efficiencies of 92.42% with liquefied petroleum gas (LPG) to generate steam and 90.24% with diesel II to generate steam were reported by D. Aguilar Vizcarra et al. [4].

2.4 Plant of Interest Description: TORRIPAC

The boiler of interest for this research is a unique hybrid boiler, part of John Thompson's TORRIPAC range, which combines a water-tube and fire-tube design. Aside from the water-cooled (water-tube) furnace and fire-tube evaporator, the architecture includes an air heater and economiser. This system does not contain a superheater since this product range is marketed to small scale industries which require process steam for their production processes rather than utility scale electrical power generation. Superheaters could be coupled with turbines or screw expanders to generate electrical power. The present research deals only with a saturated steam boiler where the process steam can be utilised by the factory.

A 4 ton/hr TORRIPAC unit with architecture shown in Figure 5 has been commissioned by John Thompson for a Rum distillery in Grenada in 2019. Plant data from testing of the Grenada commissioned boiler has been used to calibrate the Flownex model in section 4.4. Design data in the form of a process flow diagram containing temperatures, pressures and mass flows for a maximum continuous rating (MCR) load case was also in hand from John Thompson. This MCR data is used to develop and verify the analytical heat exchanger models as presented in section 4.1 and the stand-alone Flownex heat exchanger models in section 4.2.

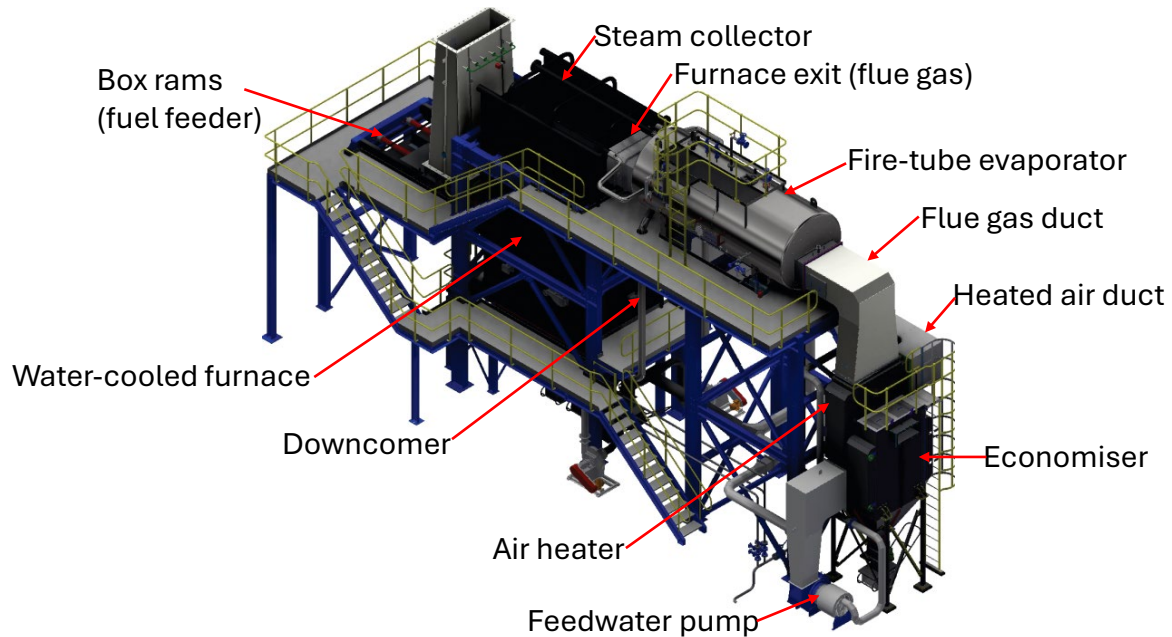


Figure 5: 3D CAD Model of the 4ton/hr TORRIPAC Unit

Other important documentation obtained from John Thompson which form part of the collection of literature include: A full set of workshop drawings for the individual heat exchangers, assemblies and site commissioning drawings for the full integrated TORRIPAC, the O&M manual, and the internal control philosophy documentation.

The workshop drawings were required to extract important geometries needed in the model development. The O&M manual together with the workshop drawings aided understanding of the interfaces between various heat exchangers and other ancillary equipment. The control philosophy documentation provided useful insights needed in identifying key control loops such that a simplified control system could be implemented in the Flownex model as detailed in section 4.5.

The process flow diagram in Figure 6 illustrates the fuel, air, steam, and flue gas streams through the 4 ton/hr TORRIPAC unit.

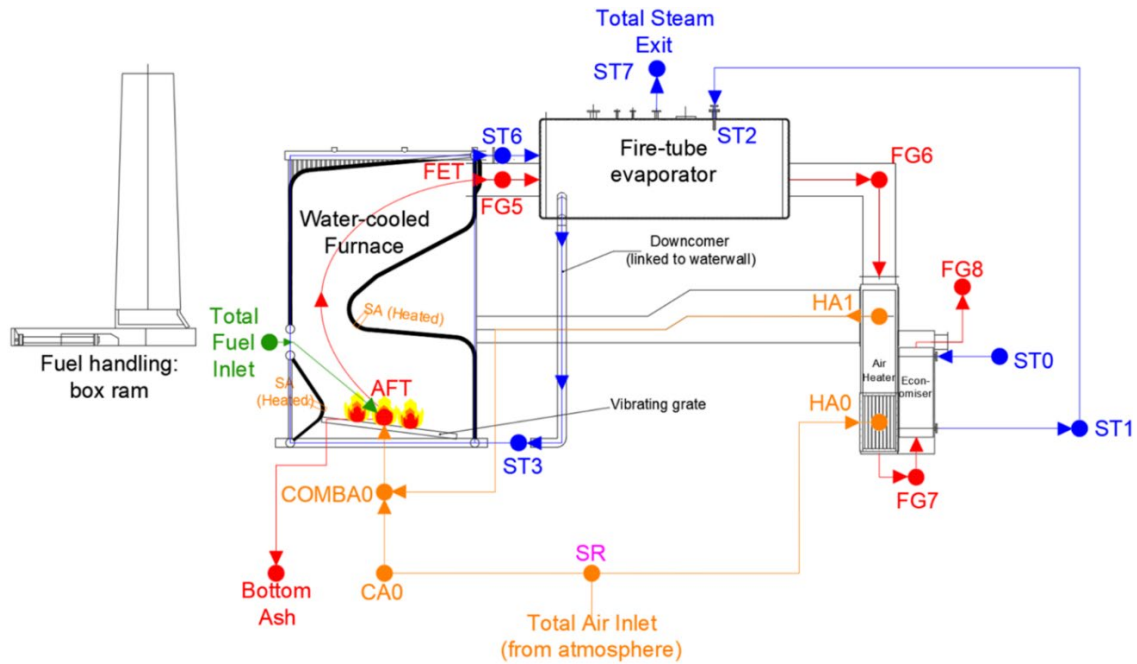


Figure 6: TORRIPAC Process Flow Diagram (PFD)

The biomass fuel being fired is sugar cane bagasse (sugar cane byproduct) and it enters the water-cooled furnace via a box ram feeder through a port in the front furnace wall, where it slides down the lower front wall onto the vibrating grate. The rear wall of the furnace is bent in towards the centre of the furnace cavity forming a prominent nose which forces the combustion gas to flow over the incoming fuel.

The grate is zoned into 3 parts with respect to the combustion air supply with dampers at the front, middle and rear below the grate. Cooling air (CA0) is supplied from ambient conditions via cooling air fans into the front zone. This air is at ambient conditions to aid the cooling of the grate material, but it also enters the furnace cavity through nozzles in the grate and is therefore involved in the combustion process. The remainder of the primary air is first heated in the air heater (HA0 to HA1) before being transported to the middle and rear zones of the grate. Secondary air, also part of the heated air stream is injected into the freeboard of the furnace through nozzles in the lower front wall and the rear wall nose as shown in Figure 6. The secondary air flow is controlled by the front lower secondary air damper and the rear nose secondary air damper.

The simplified assumption of instantaneous combustion is only concerned with the mixing of the cooling air and heated air at COMBA0 from a process modelling standpoint. I.e. the secondary air nozzles are disregarded by assuming all the heated air is mixed with the cooling air before entering the furnace as the total combustion air (COMBA0).

The combustion process transfers radiative heat to the water-walls of the furnace, while the hot flue gas at the adiabatic flame temperature (AFT as per Figure 6) radiates heat to the water-walls as it rises up to the furnace exit (FET and FG5 as per Figure 6). As the flue gas exits the water-cooled

furnace, it enters the fire-tube evaporator at FG5 as it makes a single pass through the fire tubes to the inlet of the air heater (FG6).

Convective heat transfer from the flue gas in the evaporator tubes enables natural convection to heat the water in the evaporator. The combined radiative and convective heat transfer causes a phase change, producing the required pressure and saturated steam flow for factory use. Waste heat recovery downstream of the evaporator (FG6 to FG8) sees the flue gas heat the air in the air heater (HA0 to HA1) and preheat the feedwater in the economiser (ST0 to ST1).

On the steam side, preheated feedwater from the economiser (ST1) enters the evaporator, where natural circulation is facilitated by downcomers connecting the evaporator and furnace water-walls. Saturated process steam (ST7) is extracted from the top of the evaporator for factory use.

Note that the plant of interest contains ancillary equipment which is not indicated in the PFD in Figure 6. On the steam side, this includes a condensate tank with condensate feed pumps, a water softening unit with a chemical dosing pump and a feedwater tank complete with feedwater pumps.

The flue gas handling system downstream of the economiser includes a grit collector, bag filter, induced draft (ID) fan and stack. All these ancillaries as well as the fuel handling system upstream of the vibrating grate fall outside the scope of the present research. The plant control volume from a thermodynamic lens is limited to the information presented in Figure 6.

2.5 Boiler Modelling Methods

Three-dimensional Computational Fluid Dynamics (CFD) is highly effective in modelling complex heat transfer and flow characteristics through discretisation of all the flow paths and heat exchangers. Due to the high computational expense of CFD models, it is often more feasible to apply integrated whole-boiler process models to analyse biomass and coal-fired boilers as explained by P. Rousseau, R. Laubscher, and B. T. Rawlins [7].

Thermofluid modelling of boiler systems is a popular subject in many literature sources including textbooks and technical papers but [7] identify incomplete information which distorts the bigger picture. For this reason P. Rousseau et al. P. Rousseau et al. [7] present a thorough overview of the various modelling techniques presented in literature, clearly detailing the methodologies and simplifying assumptions (and their implications) as well as applying these techniques to case study scenarios to assess their validity.

P. Rousseau et al. [7] discuss the adoption of a one-dimensional process modelling software called APROS by J. Kuronen, M. Hotti, and S. Tuuri [8] in their modelling of a 750 MWe once-through ultra-supercritical boiler. This model was empirically validated and the intention of the model developed

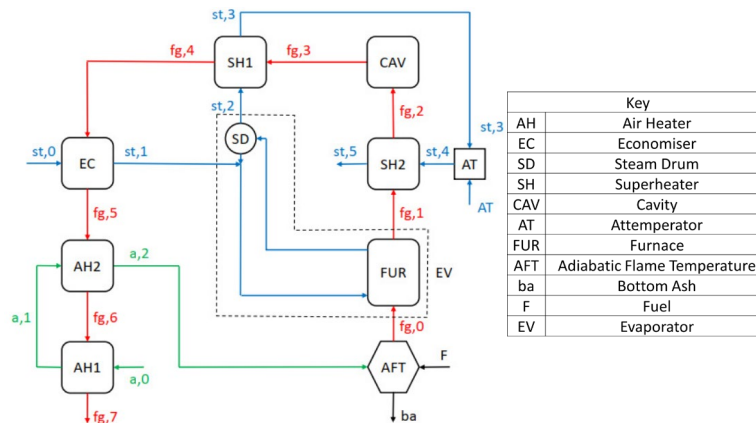
by J. Kuronen et al. [8] was for it to be used as a design tool, used for testing dynamic scenarios for proof of concept of flexible boiler operation.

P. Rousseau et al. [7] comment on the popularity of using a lumped parameter furnace analysis among many scholars in developing models for coal-fired power generating plants due to a combination of complex phenomena (combustion dynamics gas-solid interactions, fluid dynamics, and radiation heat transfer). The Gurvich/Blokh method is a popular choice and will be briefly discussed later. These authors suggest that various control strategies and boiler retrofitting can be done via the 1D method due to the computational efficiencies of the methods.

2.5.1 One Dimensional Process Networks

The entire boiler system is made up of the air, steam and flue gas networks, each of which contains several heat exchanger components and may also include turbomachinery such as feed pumps and forced draft fans as auxiliaries. A mass, energy, and momentum equation for each component in each network is required to model the heat transfer and mass flows.

P. Rousseau et al. [7] present a detailed step-by-step setup of the balance equations for a generic power plant containing all the necessary heat exchangers as illustrated in the process flow diagram of Figure 7.



temperature (FET). P. Rousseau et al. [7] provide the definition of the adiabatic flame temperature to be the “maximum theoretically possible temperature of the flue gas mixture (flame ball) at the outlet of the combustion zone.” It is customary to include an assumed fraction of unburnt carbon boilers which fire solid fuel such as coal, wood pellets, bagasse and other biomass fuels [7].

2.5.2 Furnace Heat Transfer Models

P. Rousseau et al. [7] report on the two commonly used semi-empirical zero-dimensional furnace models. The first method is the projected method (Gurvich/Blokh) and the second is the direct method (Hottel).

Although furnaces have complex geometries consisting of several surfaces each of which may have varying temperature profiles, P. Rousseau et al. [7] explains that zero-dimensional nature of these models assume radiative heat transfer is the dominant mechanism which is applied to all furnace surfaces which themselves are assumed to be a single homogeneous medium which radiates, absorbs and emits to a single temperature.

The projected method and the direct method therefore employ a single equation to calculate the total radiative heat transfer projected from the flue gas to the surrounding surfaces, but these two methods differ in the calculation of the rate of radiant heat transfer.

2.5.2.1 Projected Method (Gurvich/Blokh):

P. Rousseau et al. [7] describes that the emissions from the furnace walls are considered negligible in the projected method due the relatively low wall temperature (i.e. slightly above the saturated water temperature) as compared to the high gas temperatures inside the furnace. The total rate of radiant heat transfer from the flue gas to the wall in the projected method is a function of an effective gas temperature, total rounding projected area and the Stephan-Boltzmann constant. This method also considers the furnace effective emissivity and an area-weighted furnace efficiency factor which accounts for the fouling of the surfaces and an angular coefficient (view factor) of the different surfaces.

Many empirical factors are needed for this method. P. Rousseau et al. [7] reports empirical values and calculation methods for these factors. For example, the angular coefficient value for membrane-type wall is a function of the tube arrangement and are usually in the range of 0.9-1.0. Angular coefficients for refractory covered walls and for the furnace exit plane are 1.0. Various fouling factors for different fuels types are also reported by [7] of which 0.6 for grate fuels, 0.1 for refractory and 1.0 for furnace exit are noteworthy in the context of the present research which deals with grate firing of bagasse.

The sheer quantity of empirical factors which have been derived for very specific boilers makes its implementation somewhat risky in the absence of it being validated with computationally expensive 3D CFD modelling.

2.5.2.2 Direct Method (Hottel):

Unlike the projected method, the direct method takes the temperature and emissivity of the surfaces surrounding the furnace into account [7]. P. Rousseau et al. [7] explains that the wall emissivity, gas and particle absorptivity are evaluated at the wall temperature, with wall emissivities usually ranging between 0.8-0.85. Although this method includes view factors between all the surfaces, wall emissivities and the emission and absorption due to the participating medium, it does neglect scattering [7]. The direct method accounts for radiative heat loss from the furnace via a radiation loss factor. A key benefit of accounting for the wall temperature in the direct method according to [7] is that the conductive and convective thermal resistances of the various surfaces can be explicitly included in the formulation of the direct method.

2.5.3 Thermal Radiation of Gas-Solid Suspension

P. Rousseau et al. [7] presents the “standard model”, “low particle load model”, and “high particle load model” as the three workable approaches to calculating the effective emissivity and absorptivity of the flue gas and particle suspension. All three models utilise an empirical correlation for the mean beam length which describes the geometry of the heat transfer surfaces. This correlation is a function of the furnace volume and a total furnace surface area which includes the virtual furnace exit area [7].

These methods are not within the project scope due to the adoption of a calibration methodology using a fixed flue gas emissivity value as will be detailed in section 3.3.8.7. However, the impacts of the models from the findings of P. Rousseau et al. [7] are rather presented in the furnace modelling section (3.2).

2.5.4 Radiative-Convective Heat Exchanger Model

Radiative and convective heat transfer mechanisms are involved in varying ratios in the heat transfer from the flue gas to the various heat exchangers within a boiler control volume. P. Rousseau et al. [7] explains that the part of the heat transfer owed to each mechanism for a single heat exchanger is a function of the gas temperature and local velocities informed by the tube arrangement.

P. Rousseau et al. [7] devises a generic one-dimensional heat exchanger model incorporating both convection and radiation, where the ratio between these two heat transfer mechanisms are determined based on the local operating conditions. With accurate geometry relations, the generic

model can be employed for many tubular heat exchangers in a boiler control volume. A schematic of this exchanger model developed by P. Rousseau et al. [7] can be seen in Figure 8. below.

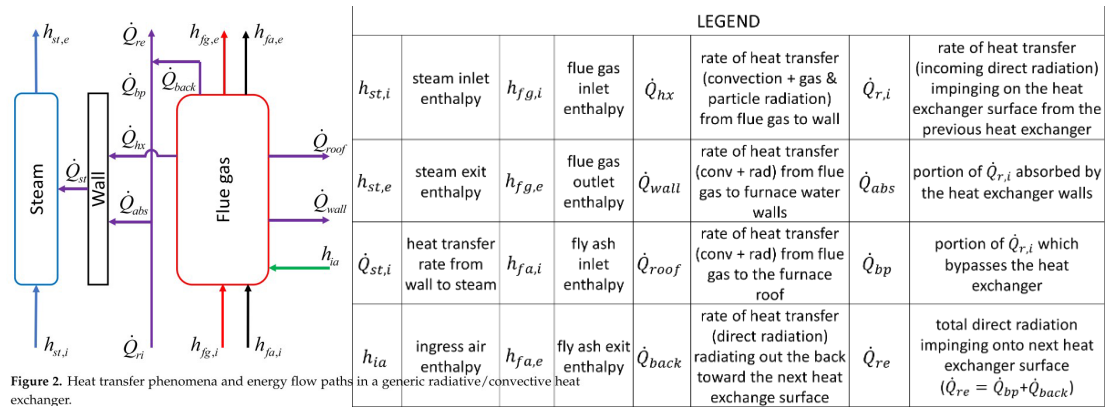


Figure 8: Generic heat exchanger model illustrating convective & radiative heat transfer phenomena and energy flow paths [7]

2.5.5 Model Application for Industrial Biomass-Fired Boiler

The methods explained in sections 2.5.1 to 2.5.4 were applied by P. Rousseau et al. [7] to an industrial scale bagasse fired boiler located in Southern Africa.

P. Rousseau et al. [7] applied both the “standard model” and the “high particle load model” to this boiler and compares the results to experimental site data. The “standard model” will be called SM and the “high particle load model” will be called HPLM from here on. Notable results from [7]’s case study indicated transfer temperatures from both SM and HPLM agreed within 0.4% at the economiser outlet and within 3.3% at the superheater outlet when compared to the experimental data. Another important outcome from this case study was that the difference between the SM and HPLM for this biomass boiler were much smaller than when they were applied to a coal fired power plant. It was therefore inferred that the HPLM is also applicable for cases of low particle loading which is characteristic of biomass fired boilers.

P. Rousseau et al. [7] reports on an important difference between the coal fired plant and the biomass fired plant regarding the dominant heat transfer mechanism which occurs after the evaporator; convection in the biomass case and radiation in the coal case. [7] goes on to explain that the dominant convection in this region in the biomass boiler can be attributed to a significantly lower furnace exit temperature and lower entrained particle concentrations in the case of biomass.

Some outcomes regarding the comparison of the direct method and projected method in the case of the biomass fired boiler as per [7] were the furnace and evaporator underprediction of 3% for the direct method and the 1.7% overprediction at the same locations using the projected method. Further downstream, the projected method underpredicted the first superheater temperatures by 3.9% and overpredicted the second superheater by 2.6%, while the direct method overpredicts the

first superheater by 8.7% and the second by 9.2%. It was concluded by the authors that the projected method provides better results overall.

2.6 Dynamic Modelling of Power Plants

2.6.1 Dynamic Simulation Models as a Tool for Flexibility in Dispatchable Power Generation

One of the most common focusses in modelling thermal powerplants is optimised thermal efficiency. Design for optimum thermal efficiency is usually done via steady-state simulation models for a limited number of discrete load scenarios which include maximum continuous rating and half load. However, due to the world wide shift towards renewable energies, it has become increasingly important to be able to assess the flexibility in operating conditions of existing power plants as well as to consider this in the design of new power plants as explained by F. Alobaid, N. Mertens, R. Starkloff, T. Lanz, C. Heinze, and B. Eppe [9].

Even with the shift towards renewable energy technologies, flexibility in power generation is critical (in both existing plants and new plants) to ensure that the fluctuating energy demand can be met. Intermittence between low and high energy demands occur seasonally, i.e. high demand in cold seasons due to increased heating demand and lower energy demand in warmer seasons. This intermittence can also be present between day-time and night-time within a single day. That is, higher demand during the day where many industries and businesses are operating at full capacity.

F. Alobaid et al. [9] explains that dynamic simulation models are a cost-efficient tool which can be used to improve the flexibility of dispatchable power. These models are therefore critical to assess transient operations which may include load changes and start-up operations. These fluctuations in energy demand puts a strain on power generation plants since the plants need to be able adjust their operating conditions to cater for different load scenarios. According to F. Alobaid et al. [9] dynamic models serve as a valuable tool for analysing this aspect and predicting how plants should adjust their operating conditions to meet energy demands efficiently.

F. Alobaid et al. [9] reports on five major approaches to maintaining power supply security and improving flexibility of future power generation facilities. One of these approaches speaks to the ability of end users to vary load demand. This is known as demand response, which is an approach which can aid the mitigations of short term grid imbalances according to [9]. The second approach as per [9] is to improve the existing electricity supply infrastructure. F. Alobaid et al. [9] goes on to explain that ramp rates and minimum load limits which hinder supply flexibility may be combatted by retrofitting existing power plants with optimised system controls.

Although the main elements in a thermal power plant model would consist of heat exchangers, pumps, pipes and drums to name a few, it also requires various control systems which are essential to obtain a realistic dynamic response [9]. The dynamic simulations are an important tool to investigate system malfunctions and can provide insight as to how to bring the system back to acceptable operating conditions to meet the energy demand.

Before a dynamic model can be produced, one must first produce a steady-state model as the starting point. Steady state simulations are highly effective for evaluating the fluid properties, mass and energy balances, and thermal efficiency of the working fluid across a range of discrete operating conditions. However, F. Alobaid et al. [9] highlights that these simulations do not provide any useful information about transient processes. The authors add that dynamic simulation is in fact the preferred modelling technique for the proposal stage of a power plant project.

2.6.2 Dynamic Modelling Approaches

F. Alobaid et al [9] expresses that modern simulation programmes combine a graphical user interface with detailed models for flow, thermodynamics and heat transfer. F. Alobaid et al [9] goes on to mention that the balance equations of mass, energy, momentum and species are the basis of the mathematical models incorporated into the simulation tools. F. Alobaid et al. [9] emphasises that the complexity of the balance equations and the chosen solution algorithms depend on whether the problem is steady state, quasi-static or dynamic, the latter needing to incorporate the time derivatives.

2.6.3 Dynamic Modelling and Optimal Sizing of Industrial Fire-Tube Boilers for Various Demand Profiles

Due to ability of firing a wide range of different fuel sources, fire-tube boilers are commercially used as low to medium pressure steam sources in a wide range of industries as explained by M. Tognoli, B. Najafi, and F. Rinaldi [10]. High thermal inertia is characteristic of these types of boilers due to the large mass of water in the shell. M. Tognoli et al. [10] emphasises that capturing the thermal inertia of the boiler is critical in a dynamic model especially for intermittent load scenarios.

M. Tognoli et al. [10] explains that the relevance and need for dynamic models comes into its own for small-medium scale industries where these boilers are often oversized and employ a simple on/off control philosophy. By considering the transient behaviour with a demand profile in mind, a significant capital cost reduction can therefore be achieved via dynamic modelling [10].

F. J. Gutierrez Ortiz [2] explains in their journal article, “Modeling of fire-tube boilers” that accurate numerical modelling of firetube boilers is a complex activity which not only involves the partial differential balance equations and additional algebraic equations, but also needs to account for non-linearities which arise as a result of radiation heat transfer and convection coupled with empirical correlations. F. J. Gutierrez Ortiz [2] further explains that specific restrictions and interlocks needed for safe operation of a thermal power plant need also be accounted for in the numerical model. F. J. Gutierrez Ortiz [2] therefore propose a comprehensive fire-tube boiler model with much attention paid to reducing its complexity and cutting down the computational time. A brief description of [2]’s modelling techniques are described next.

F. J. Gutierrez Ortiz [2] assumes that reactions take place instantaneously in the reaction chamber which has been modelled as a series of continuous stirred tank reactors. The tube walls have also been discretised in the direction of flue gas flow. The heat transfer processes are convection and radiation between the flue gas and tube walls, conduction through the tube walls and convection from the tube walls to the water. F. J. Gutierrez Ortiz [2] assumes ideal gas behaviour for the flue gas and neglects conduction in the flue gas phase. Additionally, the axial component of the radiation is neglected.

Design parameters such as boiler geometry and operational parameters such as flowrates of the feedwater, fuel, and air form the inputs to the model, while performance parameters such as temperatures and pressures of the various fluid streams, flue gas composition, water level and overall boiler efficiency are outputs from the model [2].

Similar to P. Rousseau et al. [7], F. J. Gutierrez Ortiz [2]’s model is based on mass, energy, and momentum balance equations coupled with constitutional equations but applied to a fire-tube boiler of which the two main flow networks are; fire/gas side and the water/steam side.

Assumptions on the fire/gas side include; instantaneous combustion, gray emitting, absorbing and isotropically scattering material as the medium in the furnace, and uniform surface temperature for the furnace wall which [2] reports to be consistent with experimental results with detailed mathematical models of the furnace. The model also assumes constant fuel consumption and air-fuel-ratio (AFR) under nominal conditions, but these parameters can be variable once the system is under control [2]. It should be noted that [2] uses the LHV of the fuel to capture the heat flux as opposed to the adoption of the HHV in the work of P. Rousseau et al. [7].

M. Tognoli et al. [10]’s work focusses on the dynamic modelling methodologies presented by F. J. Gutierrez Ortiz [2] in which the mass and energy balances for both the water and flue gas side are considered where [10] incorporate these methods in a finite element method (FEM) to capture the

heat transfer effects on the flue-gas side. M. Tognoli et al [10]’s work investigates five geometrical configurations representing boilers designed by their industrial partner (ICI Caldaie S.p.A).

The Proportional-Integral-Derivative (PID) control strategy implemented by M. Tognoli et al. [10] involves a plant linearisation at the nominal condition coupled with tuning of the PID for all five configurations. M. Tognoli et al [10] then evaluate the performance and controllability of each configuration for a set of four different steam demand profiles which address a wide range of customer requirements. The outputs of this evaluation included a cumulative average efficiency, cumulative pressure deviations and minimum and maximum pressure level.

2.6.3.1 Two-Phase Dynamic Models

F. Alobaid et al. [9] comments on literature sources which present various approaches to modelling the two-phase flow in a thermal power plant. These thermal hydraulic model approaches focus on either a mixture flow model or a two-fluids model. The latter can be extended to a 2, 3, 4, 5, 6 and 7 equation flow model each adding a layer of complexity but also having their own benefits. The two-fluids model requires two sets of mass, energy and momentum balance equations, that is 1 set of balancing equations for each phase. F. Alobaid et al. [9] highlights significant computational cost, mathematical complexity and uncertainty in the modelling of effects at the interphase boundary as a few drawbacks of this approach. The interaction between phases at the interphase boundary often relies on many empirical assumptions.

Implementing a single set of conservation equations (mass, momentum and energy) to compute three distinctive fluid variables (mass flux, local pressure and temperature or enthalpy) via the mixture flow model can therefore mitigate the drawbacks encountered in the two-fluid model [9]. F. Alobaid et al. [9] does add that the two-phase model is preferable in applications where there may be non-equilibrium thermodynamic phenomena.

F. Alobaid et al. [9] comments that there is a third option which they have not reviewed in detail in their paper. This is the lumped parameter model which simplifies the system to a series of control volumes (lumps) which are interconnected via thermal resistors and capacitors. This is the preferred approach for a simplified implementation of transience in a system and has the benefit of significantly reduced computational expense, albeit at the expense of less accuracy relative to the mixture flow and two-fluid model.

2.6.4 Dynamic System Model of a Fire-tube boiler using Flownex

P. Klein et al. [1] developed a dynamic model for a 4-pass wetback fire-tube boiler. Radiative heat transfer dominates in the first pass (furnace pass) where the combustion of the fuel and air streams takes place. Convective heat transfer then dominates as the hot flue gas makes the remaining 3

passes through the system. The geometry of this boiler as illustrated in Figure 9 is based on the Johnston PFTA 500-4 boiler which is rated for 7.6 T/hr of saturated steam at pressures up to 25 bar at maximum load. This dynamic model was developed using the Flownex software package.

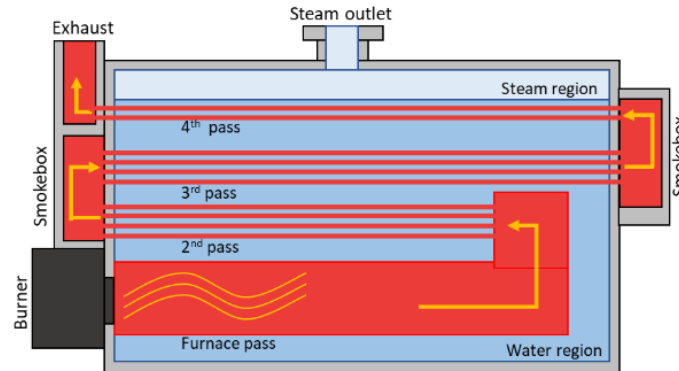


Figure 9: Wet back fire-tube boiler geometry based on the Johnston PFTA 500-4 Boiler [1]

Although literature can be found demonstrating the ability of Flownex to simulate transient boiler operation, P. Klein et al. [1] highlights that these simulations have focussed on large water-tube boilers for large scale electricity generation. Published work on dynamic Flownex models in the context of fire-tube boilers is scarce.

P. Klein et al. [1]’s “Boiler tube heat transfer segment model” is illustrated in Figure 10.

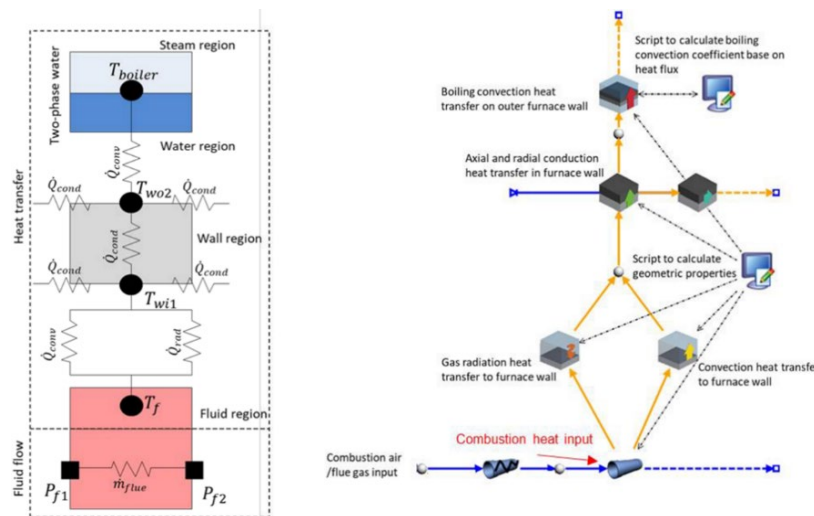


Figure 10: Flownex model of boiler tube heat transfer segment [1]

This models gas radiation to the furnace walls, convection from the flue gas to the pipe wall, convection from the pipe wall to the surrounding water. This compound effect of various heat transfer mechanisms is analogous to P. Rousseau et al. [7]’s generic radiative-convective heat exchanger methodology as discussed in section 2.5.4 but P. Klein et al. [1]’s Flownex implementation includes additional axial and radial conductive mechanisms.

The pipe component in P. Klein et al. [1]’s implementation employed the Darcy-Weisbach relation in equation (2.1) to characterise the pressure drop behaviour.

$$\frac{\Delta P}{L} = f \frac{\rho}{2} \frac{V^2}{d} \quad (2.1)$$

The gas radiation component employs the Leckner correlation which considers the partial pressures of water (P_{H_2O}) and carbon dioxide (P_{CO_2}) as well as the tube wall temperature (T_w), gas temperature (T_f) and the mean beam length (L_b) in the calculation of the emissivity in equation (2.2) and absorptivity in equation (2.3).

$$\varepsilon_f = \varepsilon_{H_2O} + \varepsilon_{CO_2} - \Delta\varepsilon \left(p_{H_2O} L_b \frac{T_w}{T_f}, p_{CO_2} L_b \frac{T_w}{T_f} \right) \quad (2.2)$$

$$\alpha_f = \alpha_{H_2O} + \alpha_{CO_2} - \Delta\varepsilon \left(p_{H_2O} L_b \frac{T_w}{T_f}, p_{CO_2} L_b \frac{T_w}{T_f} \right) \quad (2.3)$$

The Dittus-Boelter correlation is used for flue gas side convective heat transfer coefficient with geometry obtained from the pipe component. This correlation as presented in equation (2.4) was the most accurate during testing in the work of P. Klein et al [1].

$$h_{conv} = \frac{k_f}{d} 0.023 Re^{0.8} Pr^{0.3} \quad (2.4)$$

The surface heat flux (q) and boiler shell side pressure (P_b) were used as part of the Gorenflo boiling heat transfer calculation as per equation (2.5). This heat transfer coefficient was modified to include buoyancy effects of the surrounding boiling fluid via a substitution of equation (2.5) into (2.6).

$$h_{boil} = 5580 \left(1.73 P_b^{0.27} + P_b^2 \left(6.1 + \frac{0.68}{1 - P_b^2} \right) \right) \left(\frac{q}{20000} \right)^{0.9 - 0.3 P_b^{0.15}} \quad (2.5)$$

$$h_{boil}^* = h_{boil} \left(1 + \frac{1}{2 + \frac{q}{1000}} \right) \quad (2.6)$$

A two-phase tank was used to model the boiler shell with branch connection heights of the incoming and exiting pipes specified such that sub-cooled water and saturated steam vapour are extracted at the correct locations to verify an accurate water level.

Scripts components were used to calculate geometric properties and heat transfer coefficients employing the correlations presented in the equations above.

After a mesh independence study, P. Klein et al. [1] found that 15 segments (10 flame and 5 non-flame) were sufficient for the furnace pass and 5 segments were sufficient for the second third and fourth passes. Figure 11 shows the complete fire-tube package boiler model where the flame elements include the addition of heat input.

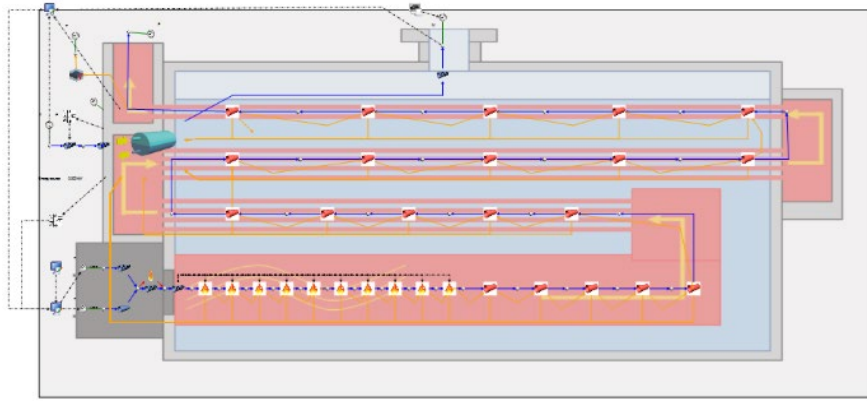


Figure 11: Complete Flownex fire-tube boiler model [1]

The exit flue gas temperatures and boiler efficiencies from the Flownex model showed good accuracy when compared to the manufacturers measured data. P. Klein et al. [1] attribute the slight error between their model results and the manufacturers data sheet to the fact that they used methane instead of specifying the exact natural gas composition used by Johnston during testing.

Simple control was included in the model via a PID controller to regulate the fuel flow based on a specified pressure set-point and a separate PID controller to control the feedwater flow based on the boiler water-level. The measured error as per (2.7) below is the output for each PID controller.

$$Output(t) = K_p e(t) + K_i \int_0^t e(\tau) d\tau + K_D \frac{de(t)}{dt} \quad (2.7)$$

The active control loops were tested through a transient simulation using a steam demand input profile which shifts the demand up and down in increments of 2 tonnes per hours every hour over a 4 hour duration as illustrated in the left graph of Figure 12. The resulting boiler level and pressure responses can be seen in the middle and right graphs in Figure 12.

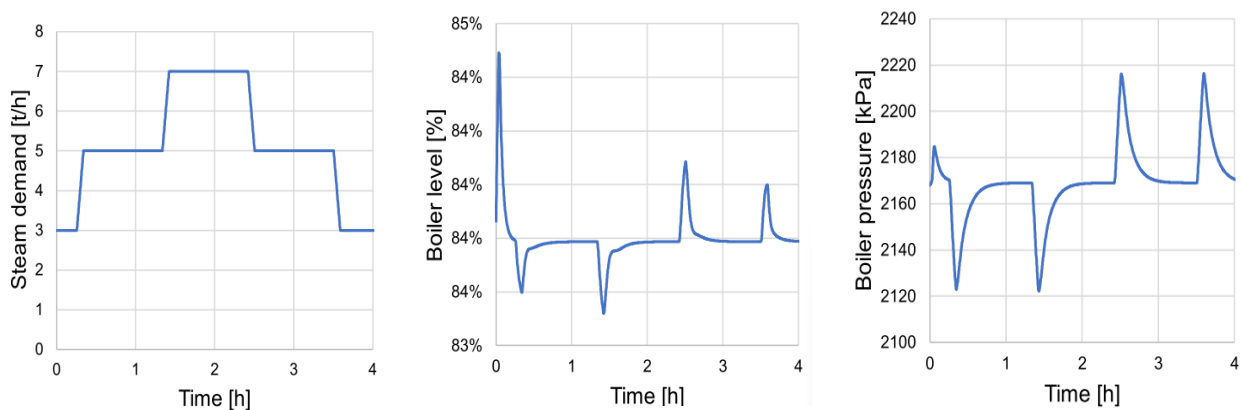


Figure 12: Boiler steam demand input profile (left), level response (middle), and pressure response (right) [1]

P. Klein et al. [1] assume a constant 15% air-fuel-ratio during the transient investigation illustrated in Figure 12 but qualify that future studies could de-couple the air and fuel flows. The stabilisation of the boiler level and pressure after each steam demand change indicates controller success.

2.7 Summary of findings

The architectures and working principles of the various boiler types detailed in sections 2.1 to 2.3 provided good insights for the present work considering the unique hybrid design of the boiler of interest. At a high level, fire-tube boilers use combustion gasses flowing inside of tubes to heat a large volume of surrounding water while water-tube boilers see hot flue gas transferring heat from a large furnace cavity to the water flowing vertically upward inside a membrane type wall. From a dynamic lens it is important to understand that the fire-tube boilers have a large thermal inertia component by virtue of the large mass of water in the drum, while water-tube boilers of a similar steaming rate would have a lower thermal inertia especially in the case of the once-through type.

The boiler modelling methods in section 2.5 suggest adopting a one-dimensional whole boiler network modelling approach for reliable computationally efficient solutions. These approaches include the simultaneous solving of the mass, momentum and energy balance equations for all fluid streams in the boiler system including the air, fuel, flue gas and water streams. Many authors include radiative and convective effects in their modelling of heat transfer to pipe components.

The literature highlights radiation as the dominant mechanism in the furnace region of both the fire-tube and water-tube boiler types with convection dominating downstream of the furnace.

Of the two commonly used radiation models, the projected method is the one that involves the use of a myriad of empirical factors and constants which have been generated from tests on very specific boilers and therefore it may not be reliably implemented for unique boilers. It includes radiative interactions between the flue gas and the furnace wall as well as interactions between the various gas species within the flue gas in its calculation of an effective gas emissivity and absorptivity to ultimately characterise the effective radiative heat transfer. The direct method on the other hand includes effects such as back-scatter and accounts for the surrounding wall emissivity and temperature in its formulation.

Dynamic models have been identified as a tool for flexibility in dispatchable power generation and is preferred over steady-state models when conducting feasibility studies before the manufacture of new boiler designs. The literature emphasises the need for reliable dynamic boiler models to be used for optimising control strategies. These models can therefore provide designers increased confidence in the ability of boiler to adapt efficiently to intermittent load changes, fuel quality variations and other common transient operational scenarios.

Various dynamic modelling methodologies have been presented, with the work of P. Klein et al. [1] being most relevant to the present study as it details a methodology for a dynamic boiler model with a two-loop control system in the Flownex simulation environment.

Although P. Klein et al. [1]'s study was conducted on a pure fire-tube boiler, the same control philosophy of controlling the boiler pressure via the fuel flow and controlling the level via the feedwater flow is implemented in the present work. The difference is that the present work does not use an adjustable valve opening to control the feedwater flow. Instead, the feedwater flow in the present work is simply controlled via a data-transfer link from the level controller output to the feedwater pipe component whose flowrate is fixed inside of the pipe component. This is detailed in section 4.5.

The TORRIPAC boiler which is modelled in the present study is a unique hybrid design encompassing both water-tube and firetube elements. As such, literature pertaining to dynamic modelling of a similar type of boiler is not available.

3. Theory

This chapter summarizes the key fundamentals and correlations that were studied and used in order to develop the boiler model presented in the present work.

3.1 Heat Transfer Theory

3.1.1 Effectiveness Number of Transfer Units and Nusselt Number

The effectiveness number of transfer units (ϵ -NTU) method is the preferred method for cases where the temperatures are unknown and for rating of existing heat exchangers. ϵ -NTU exist for very specific geometries and flow regimes including parallel-flow, counter-flow and cross-flow. The effectiveness (ϵ) is a measure of how effectively a heat exchanger transfers heat ($\dot{Q}$) in relation to the maximum theoretically possible rate of heat transfer ($\dot{Q}_{max}$) as shown in equation (3.1).

$$\epsilon = \frac{\dot{Q}}{\dot{Q}_{max}} \quad (3.1)$$

where :

$$\dot{Q}_{max} = C_{min} \cdot \Delta T_{max},$$

$$C = \dot{m} \cdot c_p,$$

C_{min} = minimum heat capacity of the two streams

ΔT_{max} = maximum temperature difference between the two streams

The Number of Transfer Units (NTU) is a dimensionless number which acts as a measure of the heat transfer capacity of a heat exchanger in relation to the heat capacity of the fluid streams of the heat exchanger. It is evaluated as a function of the overall heat transfer coefficient, heat transfer surface area as well as the minimum heat capacity rate of the two fluid streams as seen in equation (3.2).

$$NTU = \frac{U \cdot A}{C_{\min}} \quad (3.2)$$

where :

$$U = \text{Overall heat transfer coefficient} \left[\frac{W}{m^2 K} \right]$$

$$A = \text{Heat transfer surface area} [m^2]$$

$$C_{\min} = \text{Minimum heat capacity rate of the two fluid streams} \left[\frac{W}{K} \right]$$

The effectiveness and Number of Transfer Units of a heat exchanger are coupled via correlations which have been developed for certain heat exchangers based on the type of heat exchanger and its consequent flow regime.

In the development of analytical models for the heat exchangers in the present research, ϵ -NTU correlations for cross-flow over tube banks were important. There are three main correlations. Equation (3.3) applies for a cross-flow heat exchanger where both fluids are unmixed, equation (3.4) applies to cross-flow heat exchangers in which the fluid with the minimum heat capacity is unmixed and the fluid with the maximum heat capacity is mixed, while equation (3.5) applies for cross-flow regimes where the minimum heat capacity fluid is mixed and the maximum heat capacity fluid is unmixed. C_r in the equations below represent the ratio of the minimum heat capacity to the maximum heat capacity of the two fluid streams. These cross-flow correlations were applicable to the economiser and air heater in the boiler of interest in the present research.

$$\epsilon_{CF,unmixed} = 1 - \exp \left[\frac{NTU^{0.22}}{C_r} \cdot (\exp[-C_r \cdot NTU^{0.78}] - 1) \right] \quad (3.3)$$

$$\epsilon_{CF,C_{\min}unmixed,C_{\max}mixed} = \left(\frac{1}{C_r} \right) \cdot (1 - \exp[-C_r \cdot (1 - e^{-NTU})]) \quad (3.4)$$

$$\epsilon_{CF,C_{\min}mixed,C_{\max}unmixed} = 1 - \exp \left[- \left(\frac{1}{C_r} \right) \cdot (1 - \exp[-C_r \cdot NTU]) \right] \quad (3.5)$$

Finally, equation (3.6) is the ϵ -NTU correlation for a single stream heat exchanger which was applicable to the fire tube evaporator in the present research. This is applicable to the evaporator since the water is a saturated mixture inside the evaporator, effectively having an infinite heat capacity since all the heat input to the water is being used for a phase change.

$$\epsilon = 1 - e^{-NTU} \quad (3.6)$$

The Nusselt number (Nu) is the dimensionless number which characterises the convective heat transfer between the fluid streams and the solid heat exchanger material in connection with the fluid conduction and can be generically expressed as shown in equation (3.7).

$$Nu = \frac{h \cdot L_{char}}{k} \quad (3.7)$$

where:

$$h = \text{Local convective heat transfer coefficient} \left[\frac{W}{m^2 K} \right]$$

$$L_{char} = \text{Characteristic length} [m]$$

$$k = \text{Thermal conductivity of the fluid} \left[\frac{W}{mK} \right]$$

The Nusselt number is derived from empirical correlations based on the specific flow regime and geometries. The Nusselt number correlations used in the present work are presented in section 3.1.2.

Note that equation (3.7) can be written for each fluid involved in the heat exchanger such that each fluid has its own distinct local heat transfer coefficient (i.e. h_1 and h_2). The overall heat transfer coefficient (U) in equation (3.2) therefore encompasses the local convective heat transfer coefficient of each fluid as well as a conductive resistance of the heat exchanger wall material. The overall heat transfer coefficient can therefore be expressed as shown in equation (3.8).

$$U = \frac{1}{\frac{1}{h_1} + R_{foul.1} + R_{wall} + R_{foul.2} + \frac{1}{h_2}} \quad (3.8)$$

where

$$R_{foul.1} = \text{Thermal resistance due to fouling on side 1} \left[\frac{m^2 K}{W} \right]$$

$$R_{wall} = \text{Wall thermal resistance} \left[\frac{m^2 K}{W} \right]$$

$$R_{foul.2} = \text{Thermal resistance due to fouling on side 2} \left[\frac{m^2 K}{W} \right]$$

The wall thermal resistance is calculated from the pipe geometry and thermal conductivity of the material (carbon steel in the present work) using equation (3.9).

$$R_{wall} = \frac{\ln \left(\frac{d_{o,tube}}{d_{i,tube}} \right)}{2\pi k_{tube} L_{tube}} \quad (3.9)$$

The fouling resistance on side 1 and 2 refer to the thermal resistance due to fouling on the inside and outside of the pipe respectively. Fouling could be due to waterside scaling on the insides of pipes in the economiser for example. Fouling due to particulate deposits from the flue gas could also be built-up on various heat exchanging surfaces including on the inside of the firetubes, air heater tubes and outside of the finned-tubes in the economiser.

Fouling factors are empirical values which have been recorded and may increase because of higher surface roughness's and ageing of heat exchanging materials. VDI [11] reports fouling resistances from Weierman RC (1982) for combustion gases on finned surfaces of which coal is 0.89-8.85 m²K/kW and light oil ranges between 0.36-0.7 m²K/kW. Biomass is a low ash fuel which would have a similar fouling characteristic to that of a light oil.

Typical water side fouling resistances as per the Tubular Exchanger Manufacturers Association (TEMA) standards found in VDI [11] - for treated boiler feedwater between 50°C and 115°C are 0.17 m²K/kW for water velocities less than 1 m/s and 0.088 m²K/kW for velocities greater than 1 m/s.

3.1.2 Heat Transfer in Banks of Finned Tubes in Cross-Flow

Heat transfer in banks of finned tubes is a complex calculation which requires information on the geometry, arrangement, fluid/physical properties and velocity of the fluid flowing between the finned tubes. Text from the section, "M1 heat Transfer to Finned Tubes" in the VDI Heat Atlas [11] highlights the difficulty in controlling complex inflow fluid patterns and localised turbulence due to surface roughness characteristic of these types of heat exchangers, whose designs can vary widely for different applications. As such there is not a universal solution applicable to all scenarios.

Due to the complexities described above, the authors of [11] suggest using an empirical Nusselt number correlation based on experimental data to solve the heat transfer in banks of finned tubes in cross-flow. The authors put forward the following two correlations derived from industrial data: equation (3.10) for an inline-tube arrangement and equation (3.11) for a staggered-tube arrangement. It is important to note that the characteristic length used in these Nusselt number correlations is the outer diameter of the tubes as alluded to by the "d" suffix on the Nu and Re variables in equations (3.10) and (3.11) below.

$$Nu_d = 0.22 Re_d^{0.6} ROS^{-0.15} Pr^{\frac{1}{3}} \quad (3.10)$$

$$Nu_d = 0.38 Re_d^{0.6} ROS^{-0.15} Pr^{\frac{1}{3}} \quad (3.11)$$

The Ratio of Surfaces (ROS) as per equation (3.12) is the ratio of the full surface area of a finned tube. That is, the sum of all the exposed fin surface area and tube surface area divided by the surface area of a bare tube (i.e. finless tube).

$$ROS = \frac{SA_{finned-tube}}{SA_{bare-tube}} \quad (3.12)$$

The Reynolds number for cross-flow over a finned-tube bank as per equation (3.13) is a function of the velocity in the bundle, the characteristic length, as well as the average fluid properties

(kinematic viscosity or density and dynamic viscosity) between the inlet and outlet. The bundle velocity is the maximum fluid velocity at the smallest cross-section, i.e. the cross-sectional area between the finned tubes. This velocity is found by multiplying the inlet far-field fluid velocity (minimum velocity) by the Straightened Cross Section of Flow (SCOF), which is simply the ratio of the far-field inlet area to the free-flow area between the finned-tubes as per equation (3.14).

$$\text{Re}_{\text{bundle}} = \frac{v_{\text{bundle}} L_{\text{char}} \rho_{\text{avg}}}{\mu_{\text{avg}}} = \text{Re}_{d_{\text{o,tube}}} = \frac{v_{\text{bundle}} d_{\text{o,tube}} \rho_{\text{avg}}}{\mu_{\text{avg}}} \quad (3.13)$$

$$\text{SCOF} = \frac{A_{\text{far-field}}}{A_{\text{free flow}}} \quad (3.14)$$

The inlet (far-field) fluid velocity as per equation (3.15) can simply be obtained by knowing the fin-side mass flowrate, and the inlet (far-field) area, along with the fluid density at the inlet. The bundle velocity then follows as explained above and shown in equation (3.16) with Figure 13 as visual aid.

$$v_{\text{far-field}} = \frac{\dot{m}_{\text{fin-side}}}{\rho_{\text{inlet}} A_{\text{far-field}}} \quad (3.15)$$

$$v_{\text{bundle}} = v_{\text{far-field}} \text{SCOF} \quad (3.16)$$

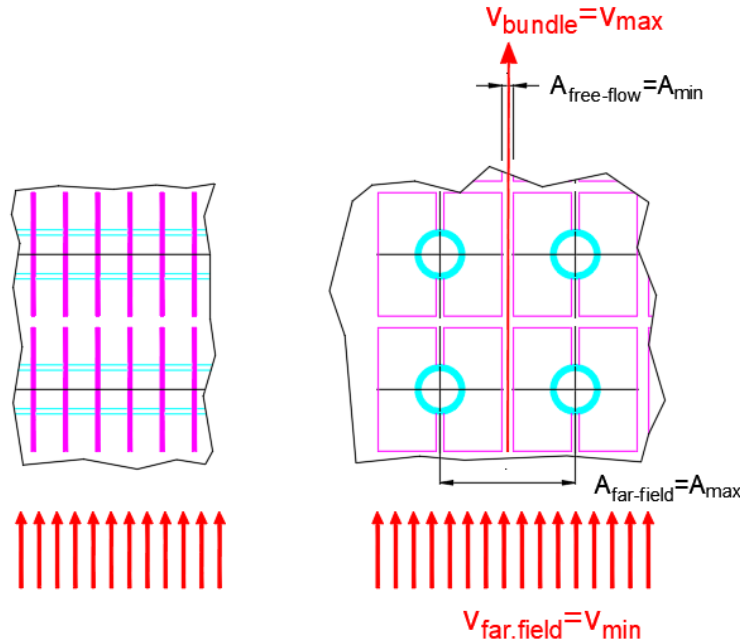


Figure 13: Finned-tube arrangement illustrating the far field velocity at the inlet and the bundle velocity at the minimum flow cross-section (between the finned tubes)

Since the SCOF is a ratio of areas, it should be noted that Figure 13 above illustrates the areas on a modular basis, i.e. using a single unit element of a finned-tube gas cross section. This is equivalent to taking the ratio of the overall flow dimensions (open duct area for A_{max} and free flow area calculated as the A_{max} minus the cross-sectional area obstructed by all the finned-tubes).

The Gnielinski pipe flow correlation as per equation (3.17) is widely used in industry for tube side convection heat transfer calculations since it is valid for a wide range of conditions with improved accuracy in the transition region between laminar and turbulent regimes ($2.3 \cdot 10^3 \leq \text{Re} \leq 10^4$) [12].

$$Nu_G = \frac{\left(\frac{f}{8}\right) \cdot (\text{Re}_{d.i.tube} - 1000) \cdot \text{Pr}}{1 + 12.7 \cdot \left(\frac{f}{8}\right)^{0.5} \cdot \left(\text{Pr}^{\frac{2}{3}} - 1\right)} \quad (3.17)$$

The Gnielinski correlation for tubular flow incorporates the Darcy-Weisbach friction factor and evaluates the Reynolds number using the tubes inner diameter as the characteristic length.

The Colebrook-White correlation for friction factor calculations for flow in the transition region between laminar and turbulent regimes is shown in equation (3.18) below as per [13].

$$f = \left[-2 \log \left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re} \sqrt{f}} \right) \right]^{-2} \quad (3.18)$$

where,

$\varepsilon = \text{surface roughness [m]}$,

$D = \text{diameter [m]}$

Zeghadia, et al. [13] affirm that Swamee and Jain's (1976) adaption of the Colebrook-White correlation is a widely used explicit approximation shown in equation (3.19) and is therefore used in the present work with the Gnielinski pipe flow correlation in the development of the relevant analytical models.

$$f = \left[-2 \log \left(\frac{\varepsilon/D}{3.7} + \frac{5.74}{\text{Re}^{0.9}} \right) \right]^{-2} \quad (3.19)$$

3.1.3 Natural Convection for Horizontal Cylinders

Boiling heat transfer in a liquid can occur via several different mechanisms which relate to the degree of superheat of the tube wall in relation to the fluid saturation temperature as well as the heat flux between the heating surface and the surrounding fluid. At low wall superheats and low heat flux, the boiling regime is natural/free convection. As the wall superheat and heat flux increase the regime changes to nucleate boiling where bubbles start forming.

In the context of the present work, it was determined at the analytical model development stage that the heat flux was just outside the free convection region, i.e. onset of nucleate boiling for the

MCR condition and well within the free convection range for the 70% MCR calibration point. Additionally it is shown in section 4.1.3 that it is the forced convection by the flue gas on the inside of the firetubes that dominates the overall heat transfer coefficient, making the addition of nucleate boiling correlations or other buoyancy correction factors counterproductive.

Natural convection for heating or cooling of horizontal cylinders is relevant for the fire tubes inside the evaporator drum. It requires correlations with the Rayleigh number and average Nusselt number using the cylinder diameter as the characteristic length. The Rayleigh number as per equation (3.21) is defined as the product of the Grashof Number and Prandtl number. For the case of natural convection driven purely by a temperature difference between the tube outer wall and the surrounding fluid, the Grashof number is defined as per equation (3.20) below.

$$Gr = \frac{g \cdot D^3}{\nu^2} \beta (T_s - T_\infty) \quad (3.20)$$

where,

g = gravitational acceleration $\left[\frac{m}{s^2} \right]$

D = diameter (characteristic length) $[m]$

ν = kinematic viscosity $\left[\frac{m^2}{s} \right]$

β = volume expansion coefficient $\left[\frac{1}{K} \right]$

$(T_s - T_\infty)$ = temperature difference between surface and surrounding fluid

$$Ra_D = Gr Pr \quad (3.21)$$

For Rayleigh numbers in excess of 10^7 , Churchill and Chu propose the Nusselt correlation as shown in equation (3.22) as per [14].

$$Nu_{nat, conv} = \left[0.6 + \frac{0.387 Ra_D^{\frac{1}{6}}}{\left[1 + \left(\frac{0.559}{Pr} \right)^{\frac{9}{16}} \right]^{\frac{8}{27}}} \right]^2 \quad (3.22)$$

Once the relevant Nusselt numbers have been calculated they can be substituted along with the characteristic length (L_{char}) and fluid thermal conductivity (k) into equation (3.7) to solve for the local heat transfer coefficient (h). The local heat transfer coefficient computed for each fluid stream are then substituted into equation (3.8) to compute the overall heat transfer coefficient (U).

The final steps involve computing the number of transfer units as per equation (3.2), substituting it into applicable ϵ -NTU correlation to determine the heat exchanger effectiveness, followed by the calculation of the heat transfer rate using equation (3.1).

3.2 Furnace Modelling

The furnace modelling methodology is easiest explained by considering an energy balance in the combustion zone. This control volume sees sensible energies in the form of incoming fuel and air at their respective inlet temperatures. The mixing conditions allow heat to be released in the form of a chemical combustion reaction, after which the control volume loses heat by virtue of the exiting fluid streams. As per the methodology of Rousseau et al. [7], the exiting fluid streams include exhaust flue gas, fly ash, bottom ash, as well as a small amount of unburnt carbon.

In the present work, an 80/20 split between bottom ash and fly ash is assumed. This assumption is in line with typical industry observations of grate firing of solid bagasse. As implied in the name, the bottom ash falls through the grate or is propelled off the grate at the base of the furnace, while the fly ash travels together with the flue gas all the way through the back pass. Both portions of the ash are assumed to leave the combustion zone at the adiabatic flame temperature in the present study. The unburned carbon on the other hand is a slight deviation from the usual assumption of instantaneous and complete combustion but Rousseau et al. [7] qualifies that small amounts of unburned fuel is common for solid fuels. The fluid streams entering and exiting the combustion zone can be seen in Figure 14.

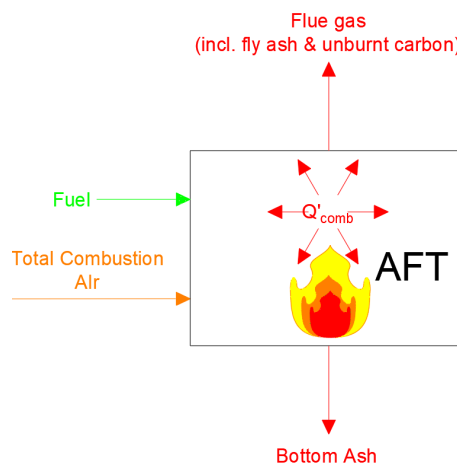


Figure 14: Combustion zone incoming and outgoing process streams

The furnace model, being the central component of the integrated boiler network involves the mixing of the distinct air and fuel fluid stream and combustion chemistry. It also requires expected performance of equipment both upstream and downstream of the furnace in order to inform the fluid conditions of the air, fuel, and flue gas streams. One therefore requires a whole-boiler energy balance to be implemented in tandem with the furnace model development.

The whole boiler energy balance is conducted to determine the fuel requirement for a specified steam demand. The fuel requirement is also highly sensitive to the fuel quality since the fuel composition informs the choice of air-fuel-ratio (AFR) that will ensure complete combustion (with exceptions of the unburnt carbon) to release the necessary energy for the desired steaming rate. This energy balance of the heat entering and exiting the control volume is illustrated in Figure 15.

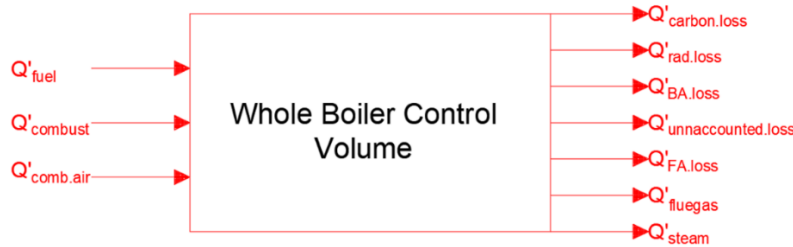


Figure 15: Energy balance on whole-boiler control volume

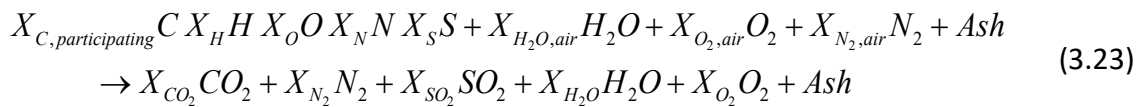
The key output from such an energy balance is the Adiabatic Flame Temperature (AFT). It represents the flue gas temperature before any heat transfer has occurred to the surrounding walls, but all losses have been considered.

3.2.1 Combustion Model

The combustion methodology employed in the present work originated from a pre-developed model in the ATProM library. It generally follows methods presented by Rousseau et al. [7], which begins with conducting combustion species mass balances for the reactants (fuel and humid air) and product species (exhaust gas).

Since the combustion modelling is not a prime focus in the present work, only the main high-level equations are presented here.

The high level molar chemical reaction written with reactant species on the left and product species on the right is formulated in equation (3.23) below.



The reactants species include carbon (C), hydrogen (H), oxygen (O), nitrogen (N), sulphur (S), and moisture (H₂O), where the air constituents are distinguishable via the “air” subscripts. “X” represents the molar weighted compositions of each species from the as-received ultimate fuel analysis (UFA). Notably the first term on the reactants side of the equation excludes the assumed small portion of unburned carbon. The products on the right-hand side of the equation consist of carbon dioxide (CO₂), Nitrogen (N₂), sulphur dioxide (SO₂), water (H₂O) and oxygen (O₂).

If the fuel flowrate is known, then the mass flowrate of any constituent (i) is simply calculated using the molar composition and molecular mass of the constituent, with the fuel flowrate and mean fuel molecular mass as illustrated in equation (3.24).

$$\dot{m}_i = X_i M_i \frac{\dot{m}_{fuel}}{M_{fuel}} \quad (3.24)$$

The actual air-fuel-ratio (AFR) can now be computed as per equation (3.25), where $\dot{m}_{air}$ is the humid air mass flowrate and $\dot{m}_{fuel}$ is the as-received fuel mass flowrate.

$$AFR = \frac{\dot{m}_{air}}{\dot{m}_{fuel}} \quad (3.25)$$

The total combustion heat is calculated from the Higher Heating Value (HHV) of the fuel, which is a function of the as-received fuel composition. Because moisture and ash do not participate in the combustion, its presence has the effect of reducing the mass fractions of the reacting elements. The net effect is a reduction of the HHV value.

3.2.2 Direct Radiation Method for Boilers

The projected method (Gurvich/Blokh) and direct method (Hottel) were discussed at a high level in section 2.5.2 as two common furnace modelling approaches.

The direct method accounts for the furnace wall temperature and emissivity with a simpler implementation which does not rely on the scarce availability of empirical factors relating to highly specific boiler architectures, fuel types and firing techniques. This is therefore the preferred model which is used in the present work. Its implementation in determining the rate of radiant heat transfer in a furnace is simply given in its standard form in equation (3.26) as per [7] below.

$$\dot{Q}_{rad} = \frac{\varepsilon_w}{\alpha_{gp} + \varepsilon_w - \alpha_{gp}\varepsilon_w} \sigma A_{rad} (\varepsilon_{gp} T_g^4 - \alpha_{gp} T_w^4) \quad (3.26)$$

where,

A_{rad} = effective radiation area [m^2]

T_w = effective furnace wall temperature [K]

ε_w = wall emissivity

α_{gp} = effective gas and particle suspension absorptivity

ε_{gp} = effective gas and particle emissivity

T_g = gas temperature [K]

σ = Stefan-Boltzmann constant $\left[\frac{W}{m^2 K^4} \right]$

It is important to note that the area in equation (3.26) is an effective radiation area which includes view factor effects.

Although P. Rousseau et al. [7] present the details involved in the calculation of the gas-particle suspension emissivity and absorptivity via the standard, low and high particle models, these methods still rely heavily on empirical values which can be a function of things such as the particle size. Data regarding these fine details are not available for the TORRIPAC and fall outside the project scope.

All three particle load models include the calculation of a mean beam length characterising the furnace geometry. The TORRIPAC furnace geometry is unique with a large rear wall nose which protrudes over the grate as will be illustrated in later chapters. The calibration results and fine-tuning methodology explained in section 4.4 will shed some light on possible consequences of this unique architecture.

In summary, a constant gas-particle emissivity of 0.6 was used, representing a low-ash content fuel such as biomass. The gas-particle absorptivity was varied as part of a parametric study during the calibration with site measurement data. These details are discussed further in sections 3.3.8.7 and 4.4.

3.3 Flownex Theory

Flownex is a one-dimensional thermofluid, numerical network software capable of accurately solving both steady-state and dynamic process networks through high fidelity solution algorithms which have been extensively validated and verified against multiple Nuclear standards [15]. This section briefly summarises the high level “implicit pressure correction algorithm” employed by the software, specific theory of common flow components used in the present work, libraries, as well as important information on the user interface necessary for action set-ups and fluid network control.

3.3.1 Implicit Pressure Correction Method – IPCM & Governing Equations

The general mass conservation/continuity equation is given in equation (3.27), where the first terms represents the rate of change of density at a specific point in space, accounting for temporal density changes which would occur in compressible flow regimes. The second term represents the rate of change of mass flux in the x-direction (i.e. for one-dimensional flow).

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho V) = 0 \quad (3.27)$$

The conservative form of the general conservation of momentum for one-dimensional pipe flow is given in (3.28). Where the first term on the left side is the unsteady time-dependant change in momentum and the second term on the left side is the convective transport of momentum arising from changes in velocity and density in the x-direction. The terms on the right hand side of equation (3.28) from left to right are: the pressure gradient force in the x-direction, volume based gravitational force accounting for elevation changes, and frictional losses due to viscous effects in a pipe, where f is the Darcy-Weisbach friction factor and D is the hydraulic diameter of the pipe/channel.

$$\frac{\partial(\rho V)}{\partial t} + \frac{\partial(\rho V^2)}{\partial x} = -\frac{\partial p}{\partial x} - \rho g \frac{\partial z}{\partial x} - \frac{f \rho |V| V}{2D} \quad (3.28)$$

The energy conservation for 1-D flow expressed in terms of total enthalpy ($h_0 = h + \frac{1}{2}V^2$) is given in equation (3.29) where the first term on the left hand side represents the total volume based change in energy with the effects of internal energy and gravitational potential energy accounted for. The second term on the left hand side represents the transport of convective energy due to the 1-D flow in the x-direction, where ρV is the mass flux and $h_0 + gz$ represents the specific total energy carried by the flow. The terms on the right-hand side represent the volume-based rate of heat transferred into the system and the work done by the system respectively.

$$\frac{\partial(\rho(h_0 + gz) - p)}{\partial t} + \frac{\partial(\rho V(h_0 + gz))}{\partial x} = \dot{Q}_H - \dot{W} \quad (3.29)$$

The state of the art implicit pressure correction solution algorithm has been summarised in Figure 16 adapted from the steps provided in the Flownex Theory Manual [15].

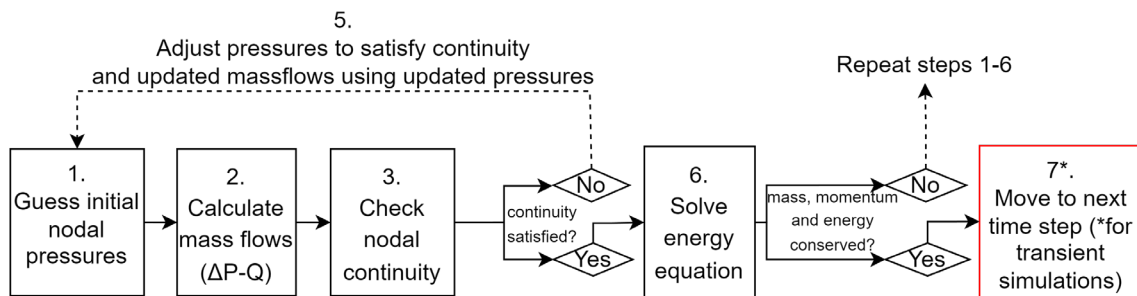


Figure 16: Implicit Pressure Correction Method (IPCM) algorithm adapted from [15]

The implicit nature of this algorithm has advantages of user controllability through tuning of relaxation parameters, number of iterations, and convergence criteria. It is unconditionally stable

with capabilities of solving networks with different types of fluids, with drawbacks of potential inaccuracies at large time steps and slow solver times [15].

3.3.2 Charts and Lookup Tables

Flownex has built-in charts and look up tables which can be found in the “Charts and Lookup Tables” libraries window as shown in Figure 17 below.

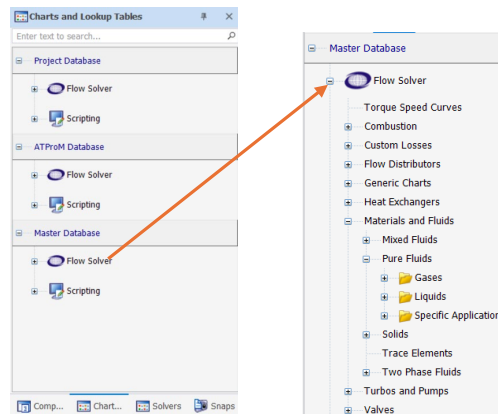


Figure 17: Flownex Charts and Lookup Tables in the Libraries window

As shown in Figure 17, the three databases are; Project database, ATProM database and Master database. Material data, fluid properties and efficiency charts a few items included in the databases.

It is convenient to note here that for a connected flow network, Flownex requires that the same fluid be specified. This means that in the case of boilers, which combusts a mixture of air and fuel to produce flue gas, one needs to create a custom fluid which contains all possible constituents of all three distinct fluid streams. It is important to use the correct type of fluid such that fluid transitions maintain the correct curve enthalpy. ATProM has already developed a custom “Coal fired flue gas” fluid as a gas mixture with Nitrogen (N_2) as the background fluid as shown in Figure 18.

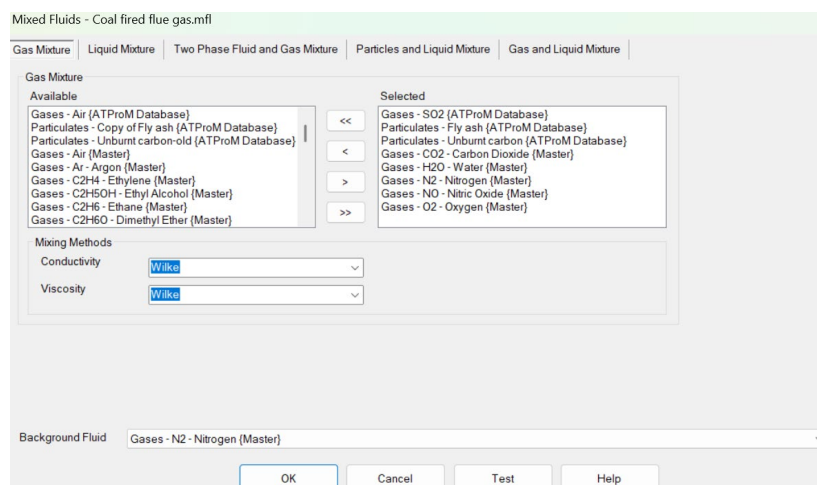


Figure 18: "Coal-fired flue gas" custom fluid in ATProM database

The other constituents included are SO₂, “Fly ash”, “Unburnt Carbon”, CO₂, H₂O, N₂, NO, and O₂. The SO₂, “Fly Ash”, and “Unburnt Carbon” were developed with fluid property lookup charts established by ATProM, while the other constituents were simply copied over from the Master Flownex database. Note that although this custom fluid was developed for coal fired flue gas, it contains the same constituents as other biomass fuels. With all the fluid properties defined for each constituent in the mixture, the Wilke mixing method was used to determine the mixture properties for conductivity and viscosity. Also, the “Fly Ash”, and “Unburnt Carbon” are assumed to behave like fluids in terms of their energy content by assuming they are well mixed and very fine, though their densities remain constant.

Charts and look-up tables can be customised easily by the user copying a chart into the project database so that it is editable. An important place where this was needed in the present work was in the creation of a custom low inertia steel. The built-in AISI 1010 steel from the Master Database has a density of 7832 kg/m³, where polynomial defined scripts are used for the conductivity and specific heat capacity as shown on the left-hand side of Figure 19. The custom (low inertia) steel material as shown on the right-hand side of Figure 19 has been specified with a negligibly small specific heat capacity of 10⁻⁶ kJ/kg.K with all other properties fixed as per the original AISI-1010 steel.

Property	Carbon Steel - AISI 1010.sol (Left)	Carbon Steel - Low Inertia.sol (Right)
Type	Bulk	Bulk
Density	Constant, 7832 kg/m³	Constant, 7832 kg/m³
Cross	Script, Polynomial 1	Script, Polynomial 1
Conductivity	Script, Polynomial 1	Script, Polynomial 1
Specific Heat	Script, Polynomial 1	Constant, 1E-06 kJ/kg.K
Youngs Modulus	Constant, 206000000 kPa	Constant, 206000000 kPa
Poisson's Ratio	Constant, 0.29	Constant, 0.29
Particle	Table	Table

Figure 19: Carbon steel – AISI 1010 {Master Database} (left) vs custom low inertia steel {Project Database} (right)

The idea with the custom, low inertia steel is that it can be specified as the material for all the heat exchanger components in a low inertia configuration of the model. Comparing transient runs between the full inertia (realistic) configuration and the reduced inertia configuration should provide insights on the effects of the solid material thermal inertia effects.

3.3.3 Multiple Views

For complex flow networks with many flow connections and data transfer links, Flownex has the functionality to create multiple views of the same component. This is achieved by simply copying a specific existing component on the drawing canvas such as a *Node* or *Pipe* and then right-clicking at

a different location on the drawing canvas (or even on the canvas of a different drawing page) and selecting “paste view”. By doing this Flownex has created a separate view of a specific component which still has all the input and result data attached to it. The new view has all the same functionality as the original instance of the component and can therefore be connected to other flow elements and or transfer data to other components.

Multiple views can visually simplify complex networks by allowing the user to build small modular flow networks independently on separate drawing pages, after which an integrated model can be connected on a separate drawing page as shown in the connection of the economiser and air heater in Figure 20 below.

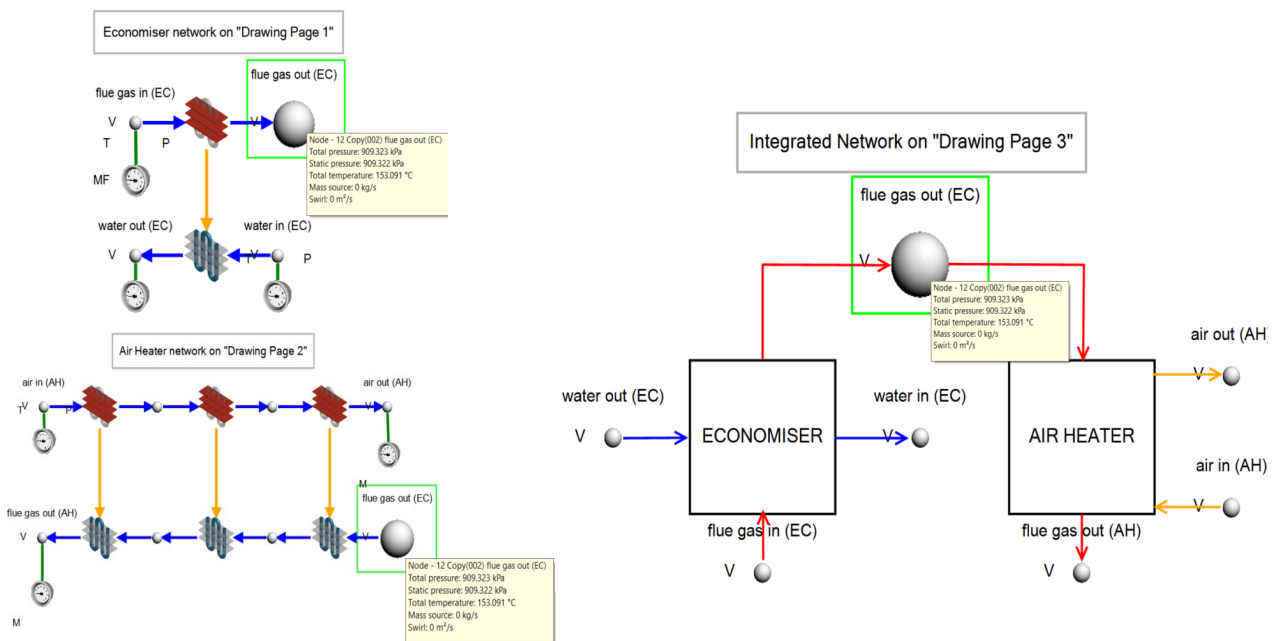


Figure 20: Multiple view functionality

Note that once a new view has been pasted, a “V” will be shown in all instances of the component including the original instance where the component was added to the project for the first time.

3.3.4 Excel Workbooks, Scripting and Data Transfer Links

Excel workbooks are a convenient way to store key-input parameters and constants such that they can be used to perform sequential sets of calculations which can then be implemented elsewhere in a particular flow network. As per the general user manual [16], any number of properties can be defined for the excel workbook since these properties are defined in the cells of the excel property grid. Figure 21 shows the excel workbook component in Flownex as well as the inputs and outputs.

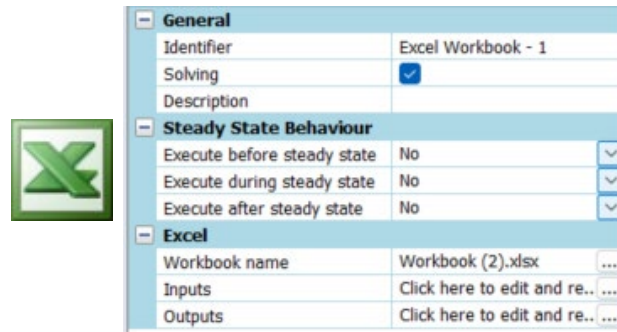


Figure 21: Excel Workbook component in Flownex

The excel workbook component was used in the present work to perform combustion calculations. As will be detailed later in section 4.2.4, the excel workbook conveniently stores the fuel composition, stoichiometric constants, molar masses, formation enthalpies etc, while receiving user inputs for the fuel and air flows, ultimately sending values such as the flue gas composition and combustion heat to the outputs of the excel workbook to be used by other flow components.

The script component allows the user to customise the model behaviour through programming logic in C#. This component can be chosen to be a normal script or a quick script. The latter requires a minimum amount of coding by the user since it contains an easy user interface where inputs and outputs can be inputted by the user into a table. The tabled inputs then form as callable arguments which the user can include in their coded functions shown on the left-hand side of Figure 22 [16]. The normal or iterative script as shown on the right-hand side of Figure 22 is more code-heavy requiring manual initialisation of parameters by the user in line with the correct C# syntax.

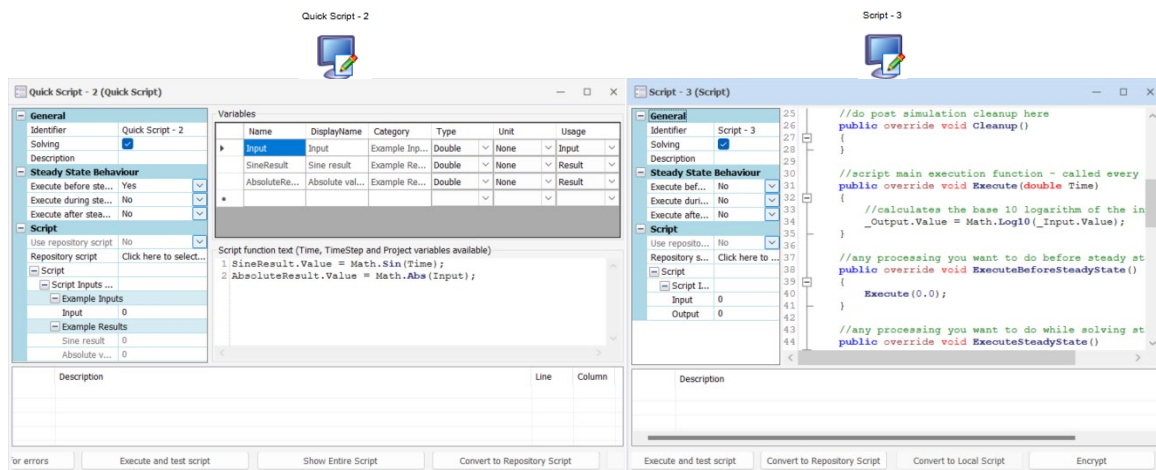


Figure 22: Quick Script editor (left) & Script editor (right)

The quick script was used often in the present research to calculate heat transfer coefficients using specific Nusselt number correlations which do not come built-in to the software. An iterative script was also used in the present work to correct the flue gas enthalpy as explained in section 4.2.4.

Data Transfer Links allows transferring of properties between various components such as excel workbooks, scripts, and flow components [16]. These links are the dashed lines in Figure 23 below.

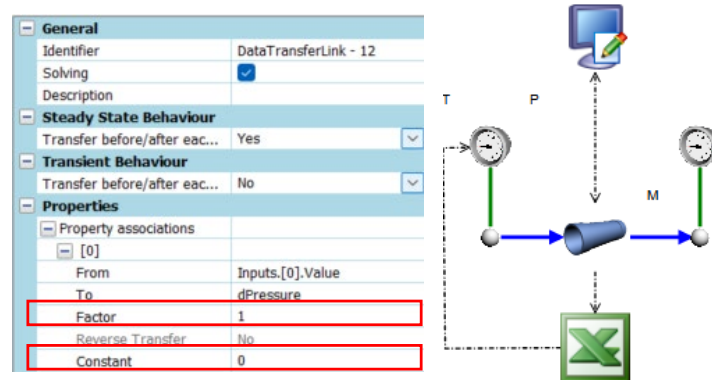


Figure 23: Example of Data Transfer Link property connections between components

Note in the above figure, the property transfer from the pipe to the excel workbook, from the excel workbook to the inlet boundary condition and the bi-directional transfer between the pipe and the script component. Any number of properties can be transferred by a single link. The data transfer link can also be specified to scale the transferred property by a factor or to add a constant.

3.3.5 Sensitivity Analyses

Flownex can address the inherent uncertainties, such as measurement errors and design uncertainties of thermofluid systems through parametric studies and Monte Carlo analyses as part of its sensitivity analysis functionality [16].

The Monte Carlo analysis can vary all nominated variables concurrently with random selections from a user prescribed range and statistical distribution (e.g. normal or triangular distribution) [16]. This can be a computationally expensive method. The automatic parametric study is much simpler and was therefore used in the present work. Parametric studies vary only a single variable at a time within a user specified range providing more computational efficiency [16].

An example of the user interface for the sensitivity analysis set-up is shown in Figure 24 below for a furnace radiation calibration employed in the present work.

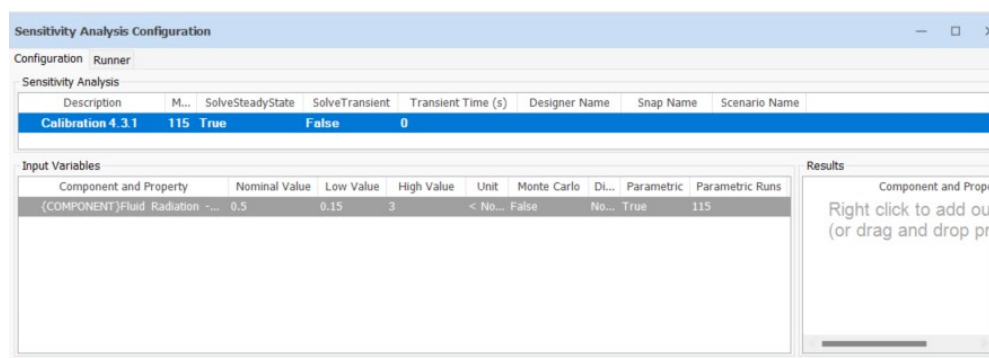


Figure 24: Sensitivity Analysis set-up for furnace calibration study

For the furnace radiation calibration, the “C-factor” from the *Fluid Radiation* component was the input variable being varied between the extremes of 0.15 and 3. Note that the results tab in Figure

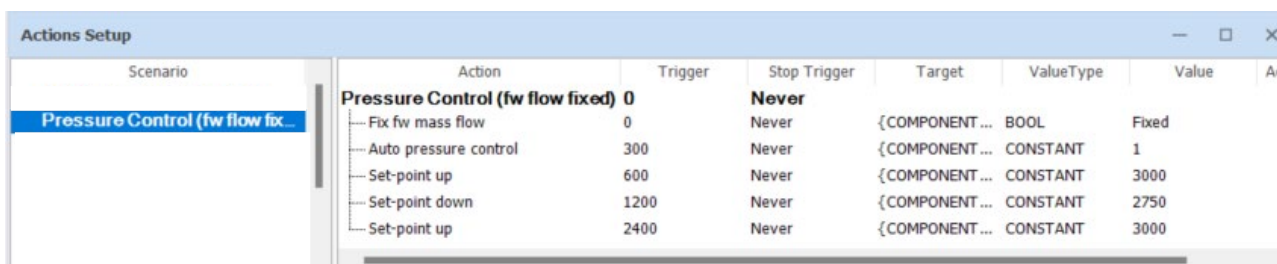
24 is empty because an excel parameter table was used for data logging at each parametric run for the various temperatures, pressures and heat transfer rates throughout the boiler system. The parameter table was set-up by simply dragging and dropping a specific result (such as a temperature) from a component and specifying the table option such that the results for each parametric run would be logged sequentially after steady-state.

3.3.6 Dynamic Modelling – Action Set-ups

Flownex can solve a network under steady state conditions or transient conditions. The steady state simulation mode automatically sets all temporal terms in the governing equations to zero, such that there is no energy or mass accumulation in the flow network. It's important to note that the effect of thermal inertia is a transient result and will therefore not have any influence on a steady state simulation.

The dynamic simulation features in Flownex are robust in that there is lots of room for user input in terms of initialising the system either through specified initial conditions or from a steady-state condition [16]. The dynamic features offer flexibility in time-step size with the allowance for user specified open loop events at different time steps. Open loop actions include; fixing of variables, controller mode switching (on-off) as well as changes in controller gains and set-points as highlighted in the general user manual [16].

An example of an action set-up for a transient run to test the pressure controller in the present work can be seen in Figure 25 below.



Scenario	Action	Trigger	Stop Trigger	Target	ValueType	Value	Av
Pressure Control (fw flow fixed)	Fix fw mass flow	0	Never	{COMPONENT...	BOOL	Fixed	
	Auto pressure control	300	Never	{COMPONENT...	CONSTANT	1	
	Set-point up	600	Never	{COMPONENT...	CONSTANT	3000	
	Set-point down	1200	Never	{COMPONENT...	CONSTANT	2750	
	Set-point up	2400	Never	{COMPONENT...	CONSTANT	3000	

Figure 25: Action Set-up for pressure controller test

For this specific action set-up, the mass flowrate through the feedwater pipe component was fixed at time zero, followed by an action to set the pressure controller into auto mode with a trigger at 300 seconds. The remaining actions changed the set-point of the pressure controller at later time steps.

3.3.7 DCS and Controller Theory

The Distributed Control System (DCS) is a library which when integrated with components from the Flownex library is able to simulate complex control systems in the Flownex environment [17].

Detailed plant simulators can be created in the Flownex environment by also incorporating the visualisation library to develop plant operator user interfaces also known as human machine interfaces (HMI) [17].

The control features in the DCS are vast, so this section briefly describes the functionality and theory of the Simple PID analogue component which was used in the present work.

The simple PID requires user inputs for the controller output limits (high and low), the process value which is the measured value that could correspond to something like mass flowrate, or pressure and the set point which represents the target value that the controller should track. The manual value is the controller output which the user can set when the controller is in manual mode. Once the controller is in automatic mode, the manual value will be continuously updated according to the output of the controller. The time dependant controller output is governed by the addition of the proportional, integral and differential terms as per equation (3.30).

$$Output(t) = P_{out} + I_{out} + D_{out} \quad (3.30)$$

All the controller parameters as well as the expressions for the proportional, integral and differential terms are illustrated in Figure 26.

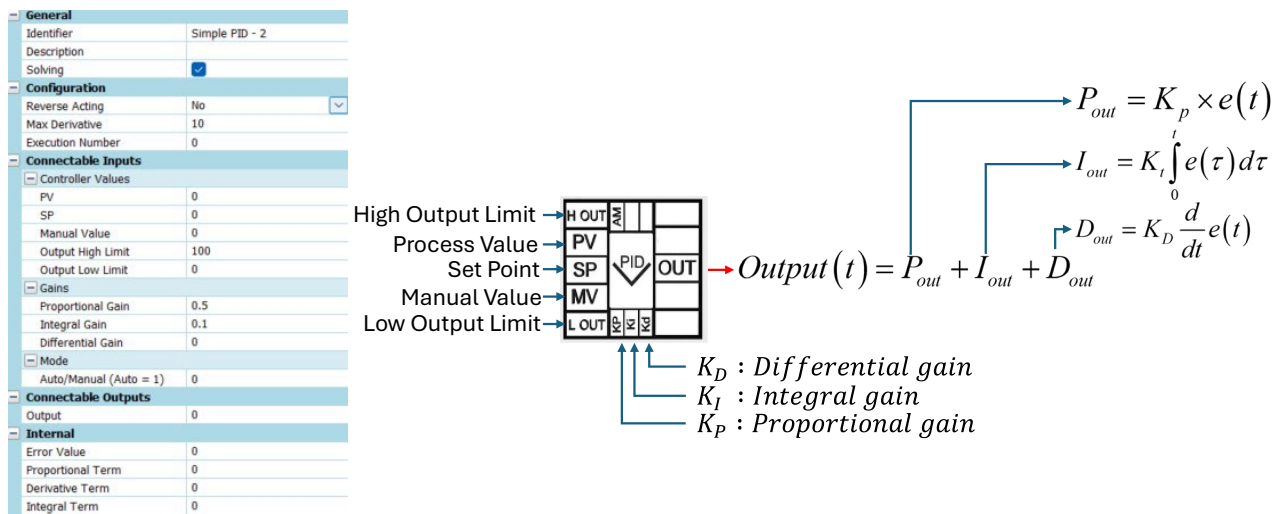


Figure 26: Simple PID input & output parameters

The proportional, integral and differential terms in the output are governed by the user inputs for corresponding gains which can be tuned by the user during controller development and testing.

Note that $e(t)$ in the above figure represents the error between the set point and the process variable at time, t. Incorporating a controller in a flow network is as simple as connecting a parameter such as pressure to the process variable input via a data transfer link. Similarly, the output of the controller can be connected to the relevant parameter that we want controlled using

a data transfer link. This will be illustrated in section 4.5.2.1 where the fuel flow is controlled using the *Two-Phase Tank* pressure set point.

3.3.8 Flow Components

This section briefly summarises the common flow components used to model the fluid flow and heat transfer in the present work.

3.3.8.1 Custom Losses: *Flow Resistance*

The *Flow Resistance* element can be used to model fluid flow for which there is no work done on/by the fluid except flow work. It is therefore used to model components with limited geometrical information and its simplicity lends itself to reducing computational expense in complex networks. It can be used to model simple pressure drops through components such as pipes, valves, and heat exchangers as explained in the Flownex Library manual [18]. This component is valid under the assumption that the flow is one-dimensional at its inlets and outlets and that no work other than flow work is done on/by the fluid which could be gas, liquid, and/or two-phase fluids. The *Flow Resistance* component can be seen in Figure 27 below.

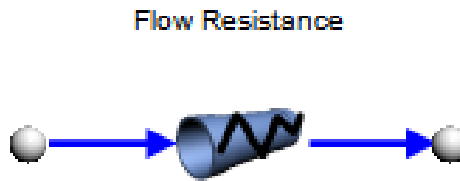


Figure 27: *Flow Resistance* component connecting two nodes in Flownex

The pressure drop through this component is modelled as a pure primary loss expressed in terms of its mass flowrate and flow area in equation (3.31).

$$\Delta P_0 = \frac{1}{2} \left(\frac{fL}{D} \right) \frac{|\dot{m}| \dot{m}}{\rho A^2} + \rho g \Delta z \quad (3.31)$$

where :

- f = Friction factor,
- L = Length[m] of the pipe,
- D = Inside diameter[m] of the pipe,
- ρ = Fluid density $[kg/m^3]$,
- $\dot{m}$ = Mass flowrate $[kg/s]$,
- A = Flow area $[m^2]$,
- g = Gravitational acceleration constant and,
- $\Delta z = z_e - z_i$ = Height difference between the inlet and outlet.

The user can choose between three different resistance behaviours, namely quadratic, linear, and square root resistance behaviour. Only the quadratic model was implemented in the present work.

The quadratic model should be applied for highly turbulent flows, which implies that friction factor fluctuations are insignificant for changes in fluid mass flowrate. The quadratic behaviour can be further classified into flows for which the density remains relatively constant vs fluid density which changes over the length of the flow resistance component.

As per the Flownex library manual [18], for highly turbulent flows with relatively constant fluid density, equation (3.31) becomes:

$$\Delta P_0 = \frac{|\dot{m}| \dot{m}}{A_f \cdot A_{sf} \cdot A_0} + \rho g \Delta z \quad (3.32)$$

With:

$$\frac{1}{A_f \cdot A_{sf} \cdot A_0} = \left(\frac{fL}{D} \right) \frac{1}{2\rho A^2} \quad (3.33)$$

Where:

- A_f = Flow admittance,
- A_{sf} = Admittance scaling factor,
- A_0 = Opening.

While for the case of highly turbulent flow with changing fluid density, (3.31) becomes:

$$\Delta P_0 = \frac{|\dot{m}| \dot{m}}{\rho \cdot A_f \cdot A_{sf} \cdot A_0} + \rho g \Delta z \quad (3.34)$$

With

$$\frac{1}{A_f \cdot A_{sf} \cdot A_0} = \left(\frac{fL}{D} \right) \frac{1}{2A^2} \quad (3.35)$$

3.3.8.2 Nodes and Boundaries

Boundary Condition components are used to specify the boundary conditions of a network and can only be connected to *Node* components as explained in the Flownex Library Manual [18]. By simply connecting a *Boundary Condition* to a *Node*, the user can specify a combination of pressure, temperature, quality, enthalpy, mass source and volume flow to that specific node. Not all combinations are possible, e.g. one cannot specify both mass source and pressure at the same node as this would over-constrain the problem.

Figure 28 below shows the inlet cooling air *Node* in the boiler modelled in the present work with the *Boundary Condition* connected.

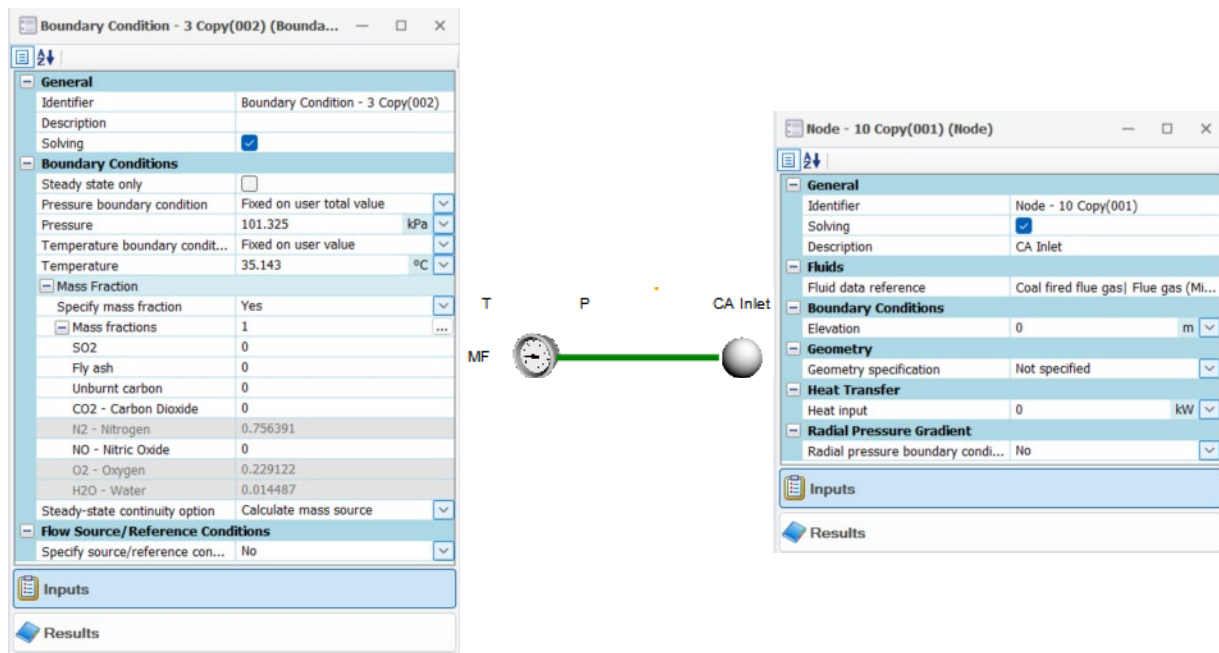


Figure 28: Boundary Condition & Flow Node visualisation in Flownex with input properties shown

Note that the *Boundary Condition* specifies the inlet pressure and temperature for the *Node* as well as the mass fraction compositions of the individual constituents for accurate fluid property evaluation.

Nodes are simply the endpoints of elements within a network [18]. For example, a *Pipe* element will have both an inlet and outlet *Node*. Nodes can also be used to model a reservoir (see 3.3.8.3 for *Two-Phase Tank*).

A *Node* can be broadly classified into two categories, namely solid *Node* and fluid (flow) *Node*. The former is a *Node* which could be specified to be a solid material, such as steel with a specified, mass and other geometrical factors. A solid *Node* can be useful for capturing the thermal mass of the material of a Flownex component which does not inherently contain thermal mass. For example, modelling a boiler drum volume can be accomplished with a *Two-Phase Tank*, but the thermal mass of the steel material of the drum can be captured in a solid *Node* by specifying the solid material (steel for example) as well as the material volume. A heat transfer component such as the *Composite Heat Transfer (CHT)* component can be used to connect the *Two-Phase Tank* to the solid *Node*. The connecting heat transfer component can specify a convective heat transfer coefficient which will allow convection from the fluid in the *Two-Phase Tank* to the solid *Node* representing the drum material in the example described above.

A fluid *Node* on the other hand which is used more often is a *Node* connected to a flow component and can either have zero volume or be specified to have a certain volume and vessel shape. The velocity in the node is assumed to be zero. The vessel shape can be specified as a polygon, vertical cylinder, or horizontal cylinder. The vertical and horizontal cylinders are straight forward in that the

user can specify the diameter and vessel volume or the diameter and the vessel length/height. Specification of endcaps are also possible for these cylinders.

The polygon vessel shape on the other hand allows the user to specify any arbitrary shape, through the combination of a horizontal area coupled with the vessel height. For shapes with non-constant cross-sectional area along the height, the polygon can be further sub-divided using the table option. This option provides a dimension table input where the user can specify the horizontal cross-sectional area coupled with a height for as many increments as necessary. This option was used to accurately track the level of the fire-tube evaporator in the present work as will be explained in 4.2.3.

For this method, the volume of the *Node* will be calculated as the sum of the volumes between the cross sections. This assumes that the centres of all the areas are directly above each other and the area changes gradually between two planes. The heights in the dimensions table are defined as the vertical distance from the horizontal cross section to the reference height, with upwards as positive. The areas in the dimensions table are defined as the area of the horizontal cross-sections.

3.3.8.3 Containers (*Two-Phase Tank*)

The *Two-Phase Tank* component can be used to model liquid-vapour separation and is therefore only valid for two-phase fluids. This component is essentially a node with restrictions on its inputs which aids the user in creating a more intuitive network as explained in the Flownex Library Manual [18]. Level tracking can consequently be performed during dynamic simulations to determine the height of the liquid level and whether a liquid, gas, or mixture is extracted from the outlets of the tank.

Level tracking inside the *Two-Phase Tank* is accomplished by either specifying the tank pressure or quality of the two-phase fluid inside the tank with a *Boundary Condition* to determine the volume occupied by both phases (liquid, and gas). Naturally, the liquid is separated to the bottom of the tank, while the gas occupies the volume above the liquid level [18].

For each component that is connected to the *Two-Phase Tank*, a branch height, height fraction, or volume fraction can be specified. This specification therefore assigns the positions of the inlets and outlets in terms of the total node volume.

Figure 29 illustrates an example of a *Two-Phase Tank* implementation using the height fraction specification option for a horizontal cylindrical shell volume with no endcaps. The connection details can be seen on the left-hand side of the figure, where height fractions of 0 are at the bottom of the shell, 1 the top of the shell and 0.5 mid-way along the height (i.e. 150 mm in the case of this 300 mm diameter tank).

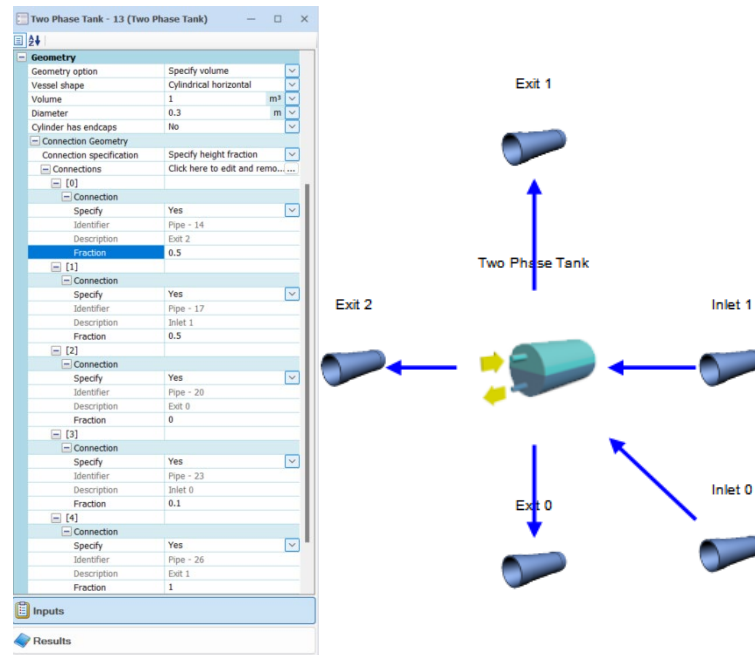


Figure 29: Example of Two-Phase Tank implementation for a horizontal cylindrical volume using height fraction specifications for branch connections

Consider the example taken from Flownex Library Manual [18] in Figure 30 below which uses the volume fraction specification for branch connections. Steam is selected as the two-phase fluid and the tank has two inlet pipes (1 and 3) and two outlet pipes (2 and 4). Pipes 1 and 2 are specified by the user as having a volume fraction of 0.1, while pipes 3 and 4 are specified as 0.9. The liquid volume fraction is determined to be 0.3 for instance which results in pipe 2 extracting liquid water (i.e. steam with quality of zero) from the tank, and pipe 4 extracting gas (i.e. steam with quality of 1).

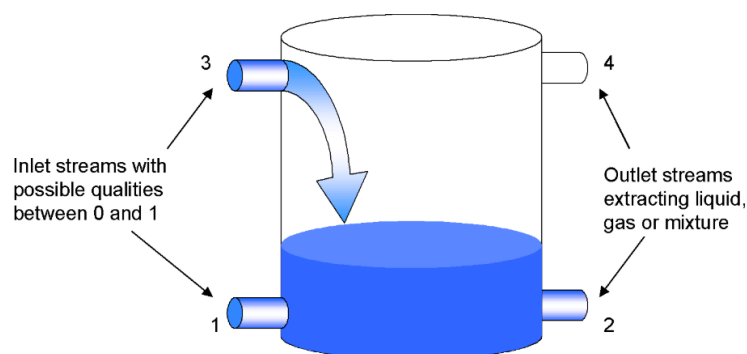


Figure 30: Flownex Level Tracking capability illustration copied from Flownex Library Manual [18]

The scenario above illustrates a typical *Two-Phase Tank* operation during steady state. For steady state implementations, the user could specify a quality *Boundary Condition* on the tank, after which Flownex would calculate the level at the steady state condition. Note that fixing the quality via *Boundary Condition* will also fix the level inside the tank. If the inlet and outlet flowrates are also fixed during steady state, it means that Flownex will compute a tank energy source which is accumulated to satisfy the energy balance. The *Two-Phase Tank* will have sensible energy entering and exiting the tank by virtue of the incoming and outgoing streams, but it can also gain or lose

energy through convective heat transfer from flue gas flowing through tubes of the evaporator in the present work as will be explained in 4.2.3. If the total energy from all these mechanisms balance for the specified quality/level constraint, we should expect the tank energy source to be nullified.

More important for the present work is Flownex's ability to track the level during a transient analysis. During a transient analysis where there is no boundary condition imposed on the tank's quality (or level), one of two scenarios can occur as per the Flownex Library Manual [18]:

1. Tank level drops: If the outlet streams are extracting more liquid than the inlet streams can supply, then the liquid volume will be depleted, after which the outlet streams will extract a mixture with the same quality as the mixing quality.
2. Tank level rises: If the outlet streams are extracting more gas than the inlet streams can supply, the liquid level rises until the entire tank (node) is filled with liquid. Consequently, the outlets will extract only liquid once the tank has been filled with liquid.

3.3.8.4 Piping

The *Pipe* component as shown in Figure 31 is used to model fluid flow inside of pipes with consideration of both primary and secondary losses if specified by the user [18].

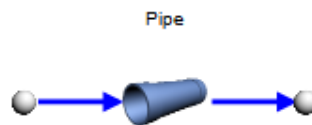


Figure 31: Pipe component connecting two nodes in Flownex

Pipe components are used in the present work to model the steam side fluid flow since this component, unlike the flow resistance component contains an internal volume needed for transient analyses.

All instances of *Pipe* components used in the present work specify the Darcy-Weisbach friction factor model as the primary loss type. A roughness value of 100 microns was specified as the roughness by selecting the maximum value in the “Welded Steel tubes” dropdown of the Flownex database as illustrated in Figure 32 below.

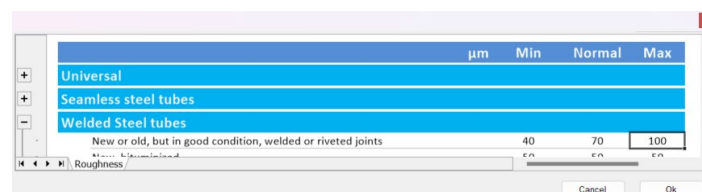


Figure 32: Tube roughness selection in Flownex for Pipe component

Since the workshop drawings are in hand from John Thompson, one could accurately specify the pipes in Flownex by selecting from a list of standard pipes from the ASME B36.10M data base for welded and seamless wrought steel pipe catalogue built-in to Flownex as illustrated in Figure 33.

ASME B36.10M
Welded and Seamless Wrought Steel Pipe

Click on blue cells to select schedule and press OK to confirm

Table 1 Dimensions and Weights of Welded and Seamless Wrought Steel Pipe

NPS	Customary Units			Identification [Standard (STD), Extra Strong (XS) or Double Extra Strong (XXS)]	Schedule No.	DN	SI Units		
	Outside Diameter	Wall Thickness	Plain End Weight				Outside Diameter	Wall Thickness	Plain End Mass
[Note 1]	in.	in.	lb/ft			[Note 2]	mm	mm	kg/m
1/8	0.405	0.049	0.19	...	10	6	10.3	1.24	0.28
1/8	0.405	0.057	0.21	...	30	6	10.3	1.45	0.32
1/8	0.405	0.068	0.24	STD	40	6	10.3	1.73	0.37
1/8	0.405	0.095	0.31	XS	80	6	10.3	2.41	0.47

ASME B36.10M / ASME B36.19M / ISO & SANS 4427-2 / ISO 4200 / SANS 62-1 / SANS 719 / SANS 96

Cancel Ok

Figure 33: Pipe database in Flownex containing industry standard pipe specifications

Other notable input options for a pipe component in Flownex allow the user to specify a fixed heat input and fixed mass flowrate through the pipe. Lastly a single *Pipe* component as placed on the drawing canvas can represent a user specified number of parallel pipes with further user specified discretisation increments along its length which becomes important when modelling heat transfer into a *Pipe* using specific heat transfer components such as *Convection* or *CHT* components.

3.3.8.5 Heat Exchangers (*Finned-Tube Heat Exchanger*)

The *Finned-Tube Heat Exchanger* component can be used to model both conventional and cylindrical type finned-tube heat exchangers [18]. This component is especially useful in capturing the thermal inertia of the heat exchanger material during transient simulations of heat transfer between two fluid streams. Furthermore, the inertia of the fluid itself is also included when calculating pressure drops through the hot and cold flow paths during transient simulations [18].

The *Finned-Tube Heat Exchanger* is a two-part component consisting of a primary side (fin-side) and a secondary side (tube-side) as illustrated in Figure 34. Each side of this component has its own unique set of inputs. This component can only handle two-phase flow on the tube side.

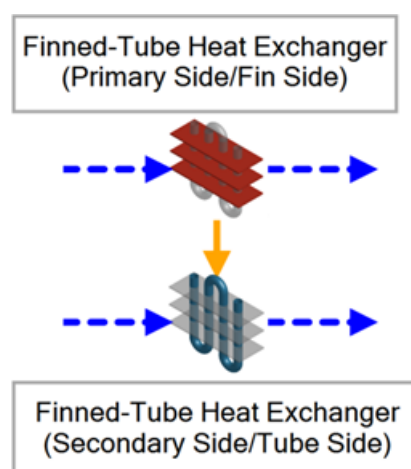


Figure 34: Finned-Tube Heat Exchanger component in Flownex

According to the Flownex Library Manual [18], the industry custom for circuiting conventional finned-tube heat exchangers is to count the tubes rows as the number of tubes high with the number of passes counted as the number of rows deep as illustrated in Figure 35 below.

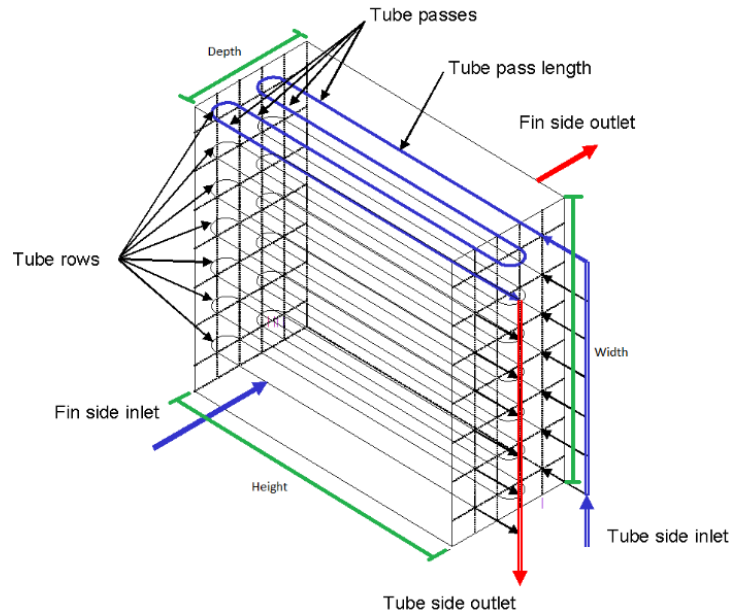


Figure 35: Flownex Finned-Tube Heat Exchanger circuiting convention taken from Flownex Library Manual [18]

This heat exchanger component is a discretised model which divides the overall component into a two-dimensional array of smaller cross-flow heat exchanger elements which each employ known correlations for cross-flow heat transfer [18]. The discretisation as illustrated in Figure 36 is accomplished by dividing the gas (fin) side flow into several parallel paths. A heat exchanger element is then formed at the intersection of a gas flow stream and a tube [18].

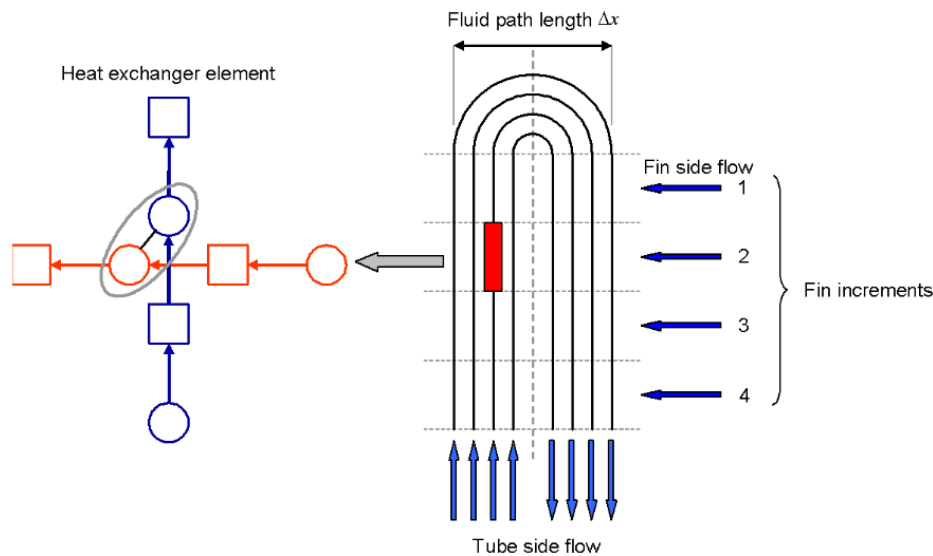


Figure 36: Flownex discretisation scheme for Finned-Tube Heat Exchanger taken from Flownex Library Manual [18]

The *Finned-Tube Heat Exchanger* component accurately accounts for the true geometry and arrangement through specifying the number of parallel circuits, number of passes and number of

tube rows as the inputs to the tube-side of the component as per the Flownex tube arrangement convention illustrated in Figure 37. A notable input on the fin-side is to specify the overall cross flow regime as being either parallel or counter flow according to the convention illustrated in Figure 38.

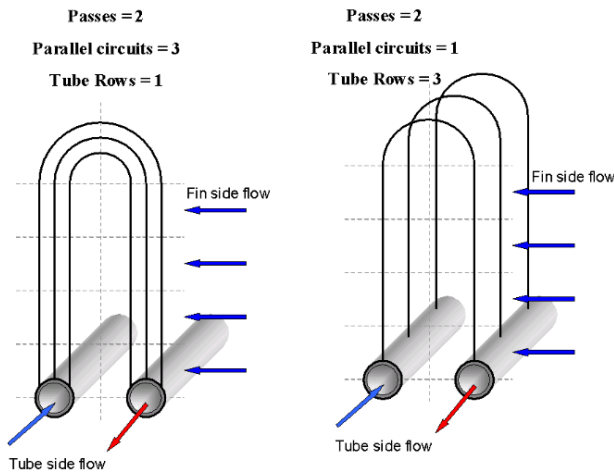


Figure 37: Finned-Tube Heat Exchanger tube arrangement convention, Schematic taken from Flownex Library Manual [18]

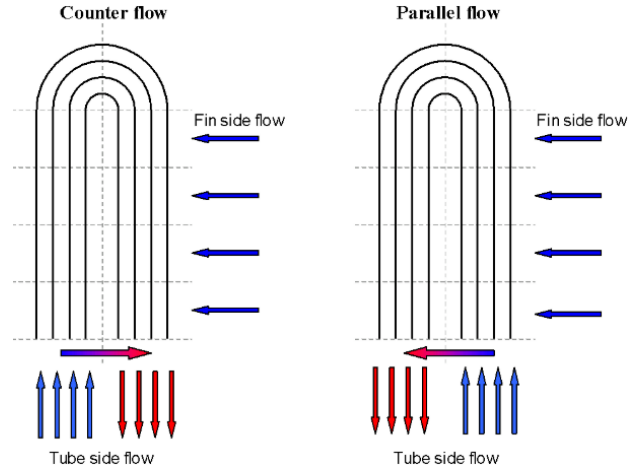


Figure 38: Finned-Tube Heat Exchanger overall Flow regime convention, Schematic taken from Flownex Library Manual [18]

3.3.8.6 Heat Transfer (Convection & Composite Heat Transfer)

The three main heat transfer components used in the present work from the Flownex library were the *Convection*, *Composite Heat Transfer (CHT)* and *Fluid Radiation* component.

As explained in the Flownex Library Manual [18] the convection component is able to model convective heat transfer at a surface by linking it to/from either a solid node or a flow element such as a *Pipe* as illustrated in Figure 39.

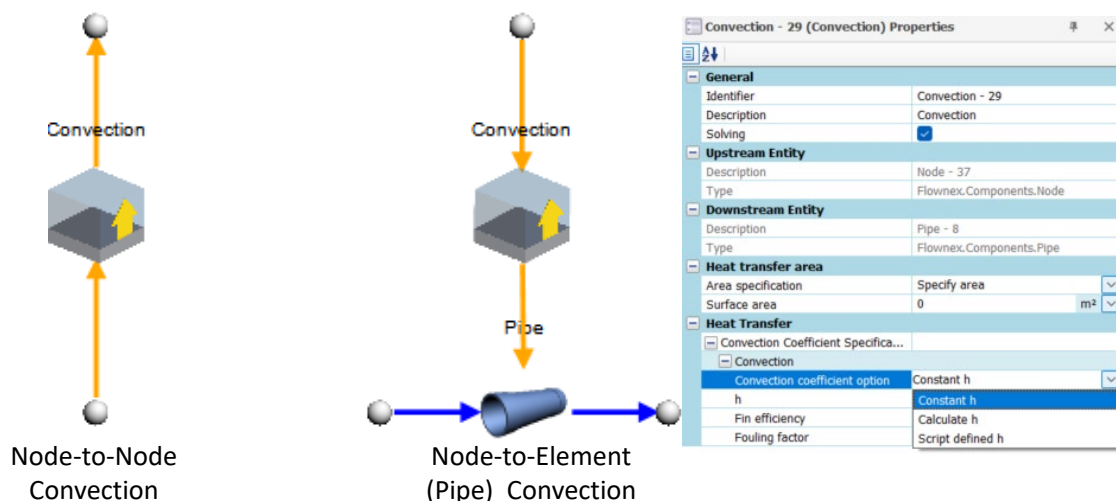


Figure 39: Convection connection configurations and input property options

The convective heat transfer coefficient can be user defined as a constant value, calculated or defined by a script with user specifications for the fin efficiencies and fouling factors as shown in Figure 39.

The *Composite Heat Transfer (CHT)* component like the *Convection* component can contain thermal resistance and inertia with the ability to be connected to flow nodes or elements [18]. It is however more robust, allowing variable heat transfer area in the component direction as well as cross component direction with flexibility to include combined convective, conductive and radiative heat transfer [18].

The thermal inertia is captured in the *CHT* component through a material specification coupled with a wall thickness and specification of conduction layers where the user can specify the number of nodes for the discretisation of a pipe wall as illustrated in the *CHT* implementation for the fire tube evaporator in the present work shown in Figure 40.

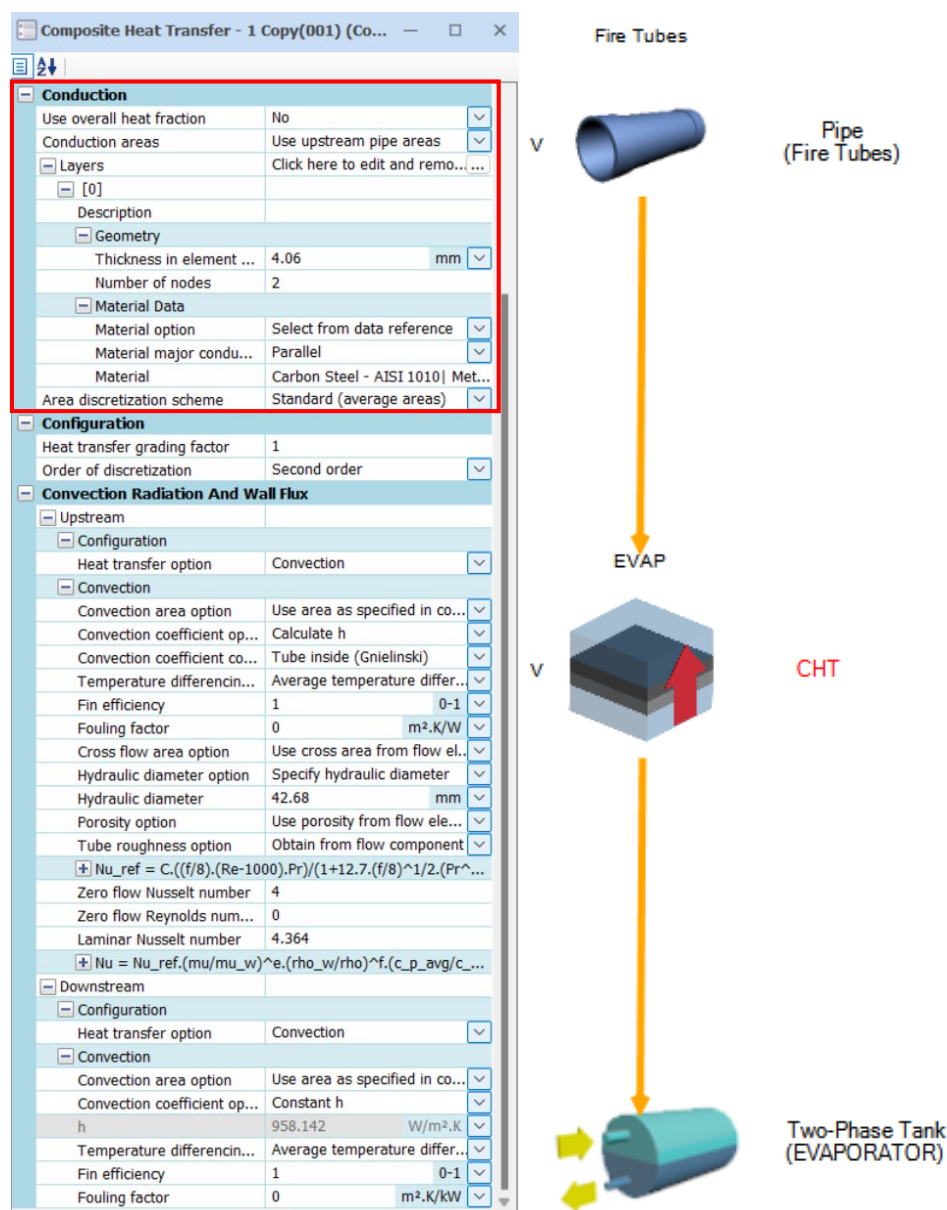


Figure 40: Example of CHT implementation in Flownex showing conduction layers for convective heat transfer in fire-tube evaporator

Note that convection was chosen for the upstream (flue gas to inner tube wall) specification and downstream (outer tube wall to water inside the two phase tank) specification, where the former makes use of the built-in Gnielinski pipe flow correlation and the latter specifies a constant heat transfer coefficient which is calculated in a *Script* and then linked to the input of the *CHT* component via a *Data Transfer Link* as will be explained in section 4.2.3.

3.3.8.7 Heat Transfer (*Fluid Radiation*)

The *Fluid Radiation* component is a heat transfer component which models radiative heat transfer between a solid surface such as a tube wall and a participating medium such as constituents within a flue gas mixture volume. This component forms part of the same heat transfer library as the *Convection* and *CHT* component, but it is presented here in its own dedicated section (3.3.8.7) since it is a big part of the calibration process in the present work.

This component can be used either to link a solid *Node* to a flow *Node* or to link a solid *Node* and a flow element (e.g. pipe). In both cases the solid node represents the metal surface, and the flow node/element represents the fluid with the participating medium as explained in the Flownex Library Manual [18].

This component assumes a constant surface emissivity, fluid emissivity and fluid absorptivity throughout the flow node/element containing the participating medium. The bulk fluid properties are used in the calculation of the net radiant heat transfer between the surrounded black body surface and the participating gasses governed by equation (3.36) [18].

$$\dot{Q}_{rad,net} = \sigma A (\varepsilon_f T_f^4 - \alpha_f T_w^4) \quad (3.36)$$

where,

A = surface area [m^2]

ε_f = fluid emissivity [-] - calculated at the existing gas temperature

T_f = fluid temperature [K]

α_f = fluid absorptivity [K] - calculated at the wall temperature

T_w = wall temperature [K]

The *Fluid Radiation* component offers the Lefebvre & Ballal and Leckner correlations for the calculation of fluid emissivity and absorptivity as 2 possible specification which are built-in to the component. These correlations involve considerations of view factors, mean beam lengths, partial pressures and mixture pressures in the methodology to calculate the emissivity and absorptivity.

Since the present research focusses on dynamic modelling of an integrated whole boiler system, the complexities involved with these factors are not within the project scope. As such, the fluid emissivity is not calculated in the present work. Instead, the fluid emissivity of the *Fluid Radiation*

component is specified with a user defined value as a constant based on typical values encountered in industry.

With the user specified emissivity configuration chosen, the net radiant heat transfer is simply written in equation (3.37) below as the product of the surface area (A) of the body and the heat flux (q'') which now includes a radiation correction/scaling factor (C) as seen in equation (3.38).

$$\dot{Q}_{rad,net} = Aq'' \quad (3.37)$$

where,

$$q'' = C\sigma \frac{\varepsilon_w}{[1 - (1 - \varepsilon_w) \cdot (1 - \alpha_f)]} \cdot [\varepsilon_f T_f^4 - \alpha_f T_w^4] \quad (3.38)$$

Note that in this configuration for a user specified fluid emissivity, the fluid absorptivity is calculated as per equation (3.39) with the user being required to specify the exponent “g”.

$$\alpha_f = \varepsilon_f \left(\frac{T_f}{T_w} \right)^g \quad (3.39)$$

As per the Flownex Library Manual [18], 4 is the maximum allowable value for “g”, ensuring a positive value for the radiant heat transfer. It can be noted here that Flownex caps the absorptivity value at 1 if the expression on the right-hand side of equation (3.39) evaluates to any value greater than 1. Additionally, the minimum allowable absorptivity is equal to the emissivity. This equality occurs for increasingly small values of “g”, i.e. approaching 0. One can therefore say with confidence that the absorptivity value is between the emissivity value at the lower extreme and unity at the upper extreme as per the inequality in equation (3.40) below.

$$\varepsilon_f \leq \alpha_f \leq 1 \quad (3.40)$$

The radiation correction factor which will be hereafter referred to as “C” or “C-factor” acts as an effective view factor accounting for uncertainties with fluid emissivity/absorptivity inputs, complex phenomena such as radiation back-scatter and even the surface area and uncertain radiation view angles which occur inside of the furnace in the context of the present work.

The *Fluid Radiation* component as implemented in the present work to model radiation from the hot flue gas inside the furnace to the water-walls of the furnace is shown in Figure 41 below along with the input properties window.

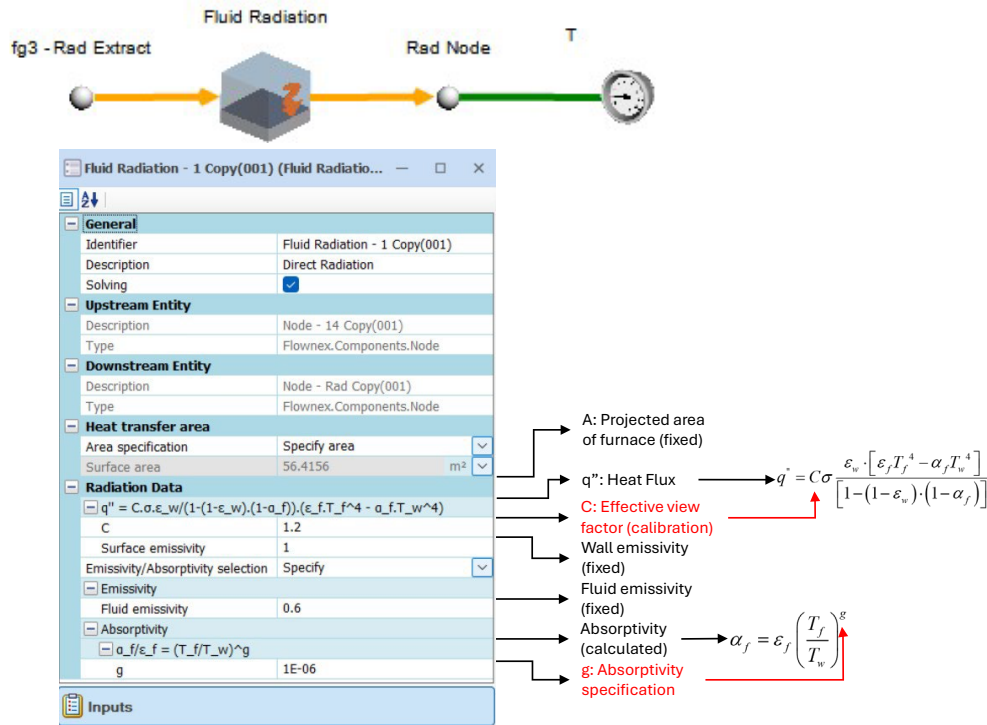


Figure 41: User input properties for Fluid Radiation component

Tuning the so-called C-factor will be carried out as part of the model calibration later. The inputs as per the red text in Figure 41 are the parameters that the user has some freedom over in the calibration, with the absorptivity having only a small effect on the radiative heat transfer for a fixed fluid emissivity. This absorptivity effect was tested parametrically in the present work.

It's important at this point to recall the direct radiation model as initially presented in equation (3.26) of section 3.2.2 to witness its parallels with the Flownex *Fluid Radiation* implementation. Both configurations of the equation include the same terms. The key difference is the inclusion of the C-factor in the Flownex implementation and the calculation technique used to determine the fluid absorptivity. The literature detailed the standard, low, and high particle load methods as techniques to calculate the gas-particle absorptivity, whereas the Flownex implementation requires the specification of the "g" factor. It is therefore clearly qualified here the reason for adopting a calibration technique through parametrically varying the C and g factors.

In the present work, the surface emissivity of the furnace walls is set to unity, while P. Rousseau et al. [7] indicate values between 0.8-0.85 when using the direct method. The unity surface emissivity over-estimates the radiative heat transfer by assuming the walls act like a black body. It is then intuitive that the eventual calibrated C-factor be some value between 0 and 1 to compensate for the over-estimation of the wall emissivity. Fixing all the other inputs therefore makes the calibration a very strong function of a single parameter (i.e. the C-factor) bearing in mind that the absorptivity is defined within clear limits as per equation (3.40) with effects of its value being tested also through parametric studies in the present work.

4. Model Development & Verification

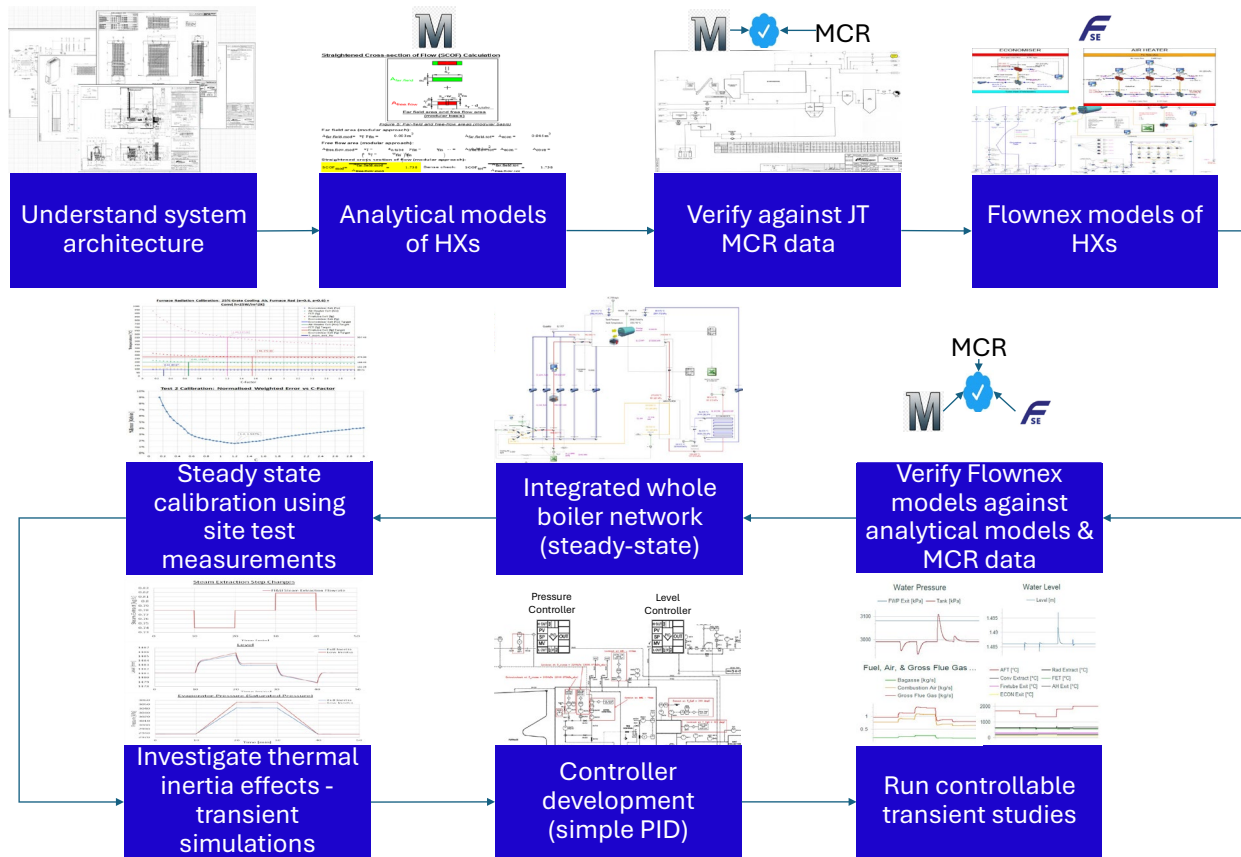


Figure 42: High level methodology outline

The flow chart in Figure 42 illustrates the process involved for the methodology of the present work at a high level. First, a detailed understanding of the system architecture was obtained by reviewing the workshop drawings of each of the heat exchangers (economiser, air heater, fire-tube evaporator, and water-cooled furnace) to learn how the various components are inter-connected.

Simple analytical models for each of the heat exchangers were then developed using Mathcad software. The analytical models were then verified using Maximum Continuous Rating (MCR) design data obtained from John Thompson which included data for mass flowrates, temperatures and pressures at various locations in the boiler system for all fluid streams; air, fuel, flue gas and water/steam.

Each of the heat exchangers were then modelled in Flownex and verified independently by comparing the heat transfer rates with both the MCR design data as well as the individual Mathcad

analytical models. The complete integrated whole boiler steady state model was completed by connecting each of the heat exchangers together.

The integrated model was calibrated using data from a site test conducted in 2021 at the rum distillery in Grenada, where the boiler was commissioned.

Once the model had been calibrated for a certain operating point, the thermal inertia effects were investigated through transient simulations. A simplified control system was then implemented by developing two simple control loops. The first loop controlled the boiler steam pressure with feedback to the fuel mass flow and the second controlled the boiler water level with feedback to the feed water mass flow. This then allowed simulation of controllable dynamic operational scenarios such as changes in fuel quality and steam extraction.

4.1 Analytical Model Development (Mathcad)

The detailed geometry from the workshop drawings and the inlet conditions to each of the heat exchangers from the MCR data were used as inputs to each of the analytical Mathcad heat exchanger models, while the outlet conditions and consequent heat transfer rates for each heat exchanger from the MCR data were used as targets to verify the analytical models. Note that this verification is simply to show that the implementation of the thermodynamics and correlations in the analytical model produces sensible results.

Since the geometries and inlet and outlet conditions are known, the development of analytical models for each of the heat exchangers in the boiler system is therefore accomplished efficiently through the effectiveness number of transfer units (E-NTU) methodology as opposed to the LMTD method which would be more applicable for designing a heat exchanger from scratch, where the heat transfer surface areas are the key unknown. The full Mathcad analytical models have been included in the appendices, while this chapter provides a brief overview of the methods used in terms of geometries and heat transfer correlations, applying the theory presented in section 3.1.

4.1.1 Economiser

The economiser is designed as an in-line finned-tube heat exchanger, where subcooled feedwater enters from a header at the top of the assembly and is split into six tube rows, each of which makes a total of fourteen passes inside the casing before collecting in the outlet header at the bottom of the assembly. The flue gas on the other hand makes only a single pass from the bottom of the assembly to the top as its flow traverses between the finned tubes as illustrated in Figure 43.

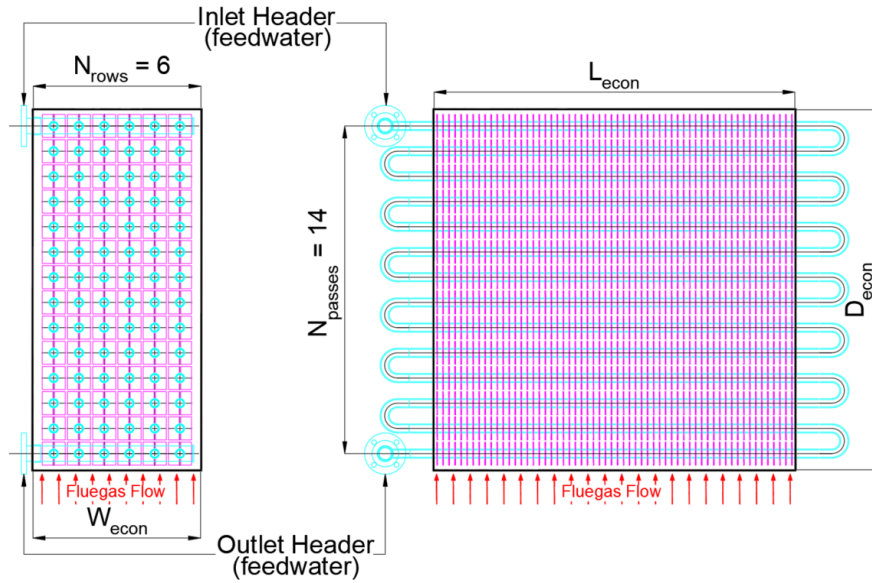


Figure 43: Economiser general arrangement

In addition to the general arrangement of the economiser shown in Figure 43, the detailed geometry required for the analytical model in terms of heat transfer surface areas and flow areas is illustrated in Figure 44. The tube diameter and thickness are self-explanatory, but the geometry of the “H-type” fins is fully defined by the fin thickness t_{fin} , fin height h_{fin} , fin width w_{fin} and fin slit s_{fin} . Note that the fin height and width are equal for this economiser which resembles a square-like fin with the fin slit creating the “H” geometry. The fin pitch P_{fin} is the spacing between successive fins on a single tube, while the longitudinal pitch s_L and transverse pitch s_T refers to the tube arrangement relative to the cross-flow fluid direction of the flue gas.

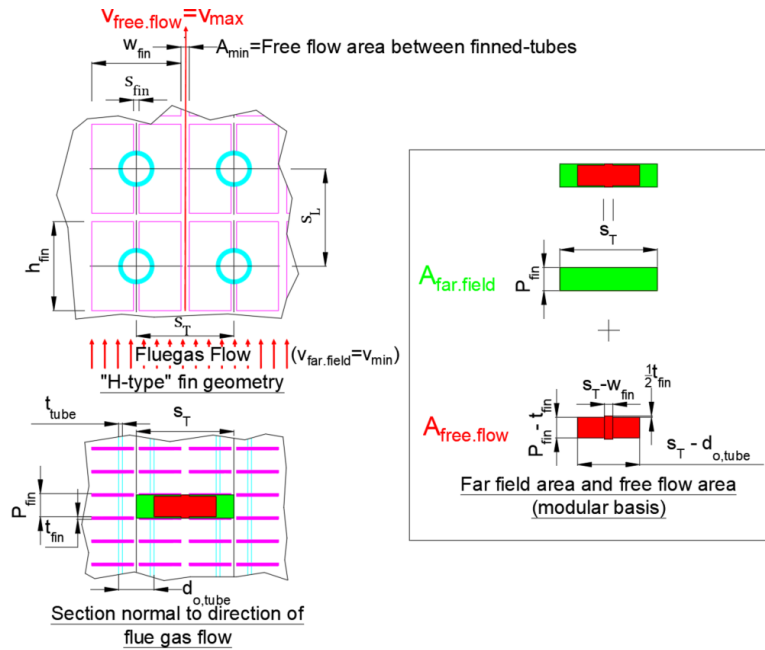


Figure 44: “H-type” fin geometry and far-field and free-flow areas (modular basis)

The maximum velocity on the fin side occurs at the smallest area, i.e. the free flow area. If one wants to determine the Straightened Cross Section of Flow (SCOF) as per equation (3.14) in section 0, then one needs to determine the ratio of the far field area (i.e. total area before the flow reaches the finned-tubes) to the free flow area (unobstructed area). This can be done on a modular basis by considering a single area unit which includes all the necessary geometries without any redundancy as illustrated in Figure 44 above.

Following the theory for heat transfer in banks of finned tubes in section 0, the above geometry is used to determine the Straightened Cross Section of Flow (SCOF) as per equation (3.14) which is subsequently used in equation (3.16) to calculate the bundle velocity. The bundle velocity is used to determine the Reynolds number of the flue gas through the bundle, using the tube outer diameter $d_{o,tube}$ as the characteristic length.

Similarly the Ratio of Surfaces (ROS) as per equation (3.12) is calculated as the ratio of total external surface area of the finned-tubes to the total external surface area of the bare tubes. The ROS is then used in the Nusselt number correlation for an inline arrangement as per equation (3.10).

The Gnielinski correlation for tubular flow as per equation (3.17) is used to determine the local heat transfer coefficient on the water-side by using a relative roughness based on a steel material and the inside pipe diameter together with the Reynolds number as per the Swamee and Jain friction factor correlation given in equation (3.19).

The water flows inside of smooth tubes, while the flue gas undergoes turbulence due to the fin and tube obstructed flow path. This, together with the fact that the minimum heat capacity rate between the flue gas and water was that of the flue gas meant that the ϵ -NTU correlation defined in equation (3.5) for cross flow with the minimum heat capacity fluid (flue gas) mixed and the maximum heat capacity fluid (water) unmixed was chosen.

For the fluid properties, the average conditions between the inlet and outlet is used. But, since the outlet is initially unknown, a manual successive approximation was implemented in the Mathcad model by using inlet conditions from the MCR data as global inputs, while the exit conditions of both streams were guessed (using the MCR data as a first approximation). After the ϵ -NTU method was employed, new exit temperatures of both streams were calculated and compared to the guessed values. If the exit temperatures were significantly different, then the calculated exit temperatures were updated to take the place of the “guessed” exit temperatures. It takes less than 4 iterations to converge as the initial outlet guesses are reasonably good. This simple successive approximation algorithm is illustrated at a high level in Figure 45. Refer to Appendix A for further details where the full analytical calculation is presented.

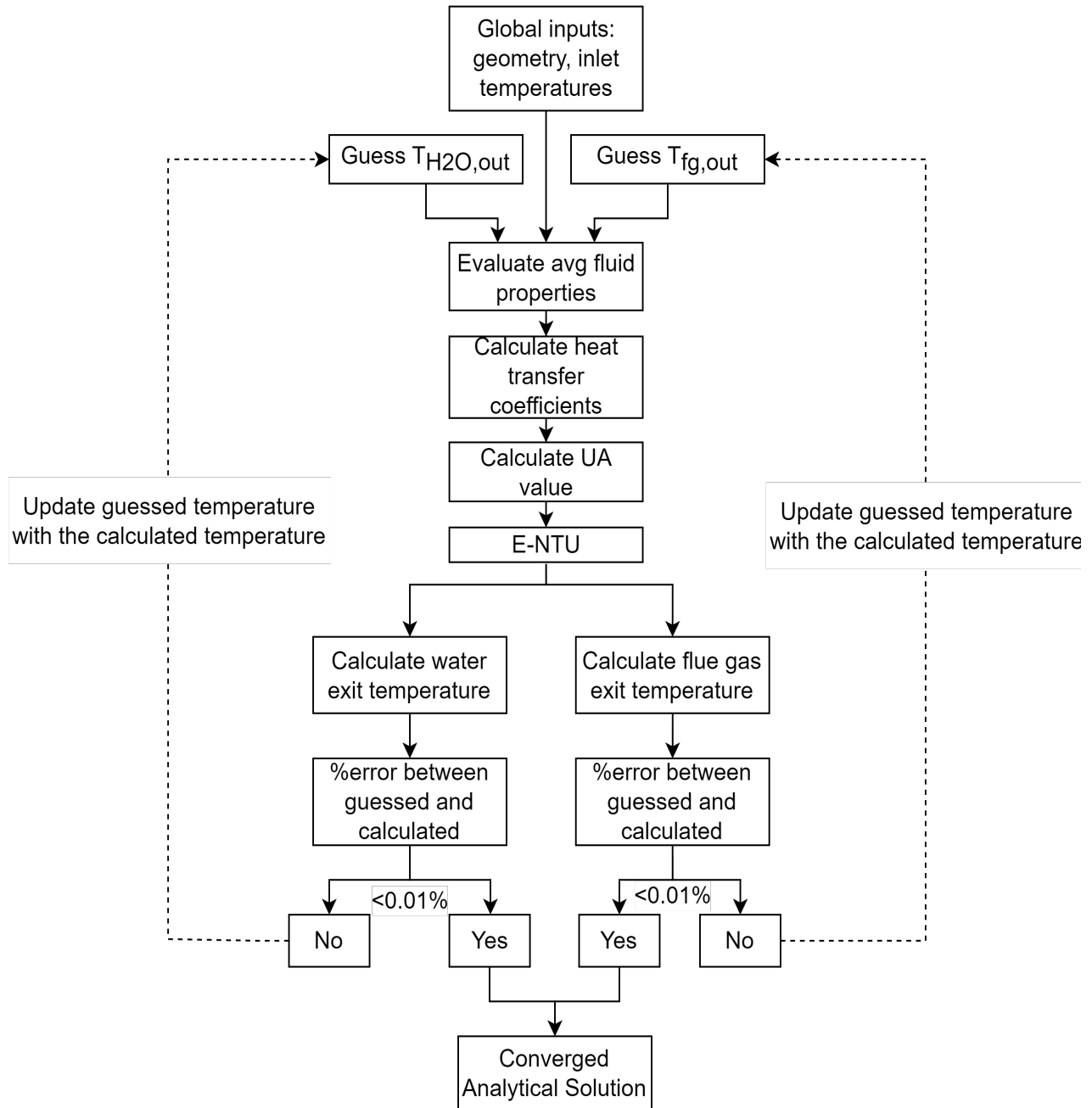


Figure 45: Economiser successive approximation algorithm used in Mathcad analytical model

The heat transfer rate computed using only the MCR data was 177.69 kW, while the analytical Mathcad computed heat transfer rate was 168.19 kW. There is therefore a 5.35% difference between the two. This is a reasonable result since the analytical model used air properties to approximate the flue gas properties.

A full comparison of the verification results between the MCR data, Mathcad analytical model and Flownex numerical model for the economiser is presented in the tables at the end of section 4.2.1.

4.1.2 Air Heater

The air heater design is similar to the economiser in that it is a tubular cross-flow heat exchanger. The difference is that now it is the flue gas which makes only a single tube pass from the top of the assembly to the bottom. The air is the fin-side fluid which makes a total of three passes between the tubes in a cross-flow regime as its flow zig-zags from the bottom left to the top right of the assembly. This heat exchanger contains bare tubes (i.e. no fins) and its general arrangement can be seen in Figure 46 below.

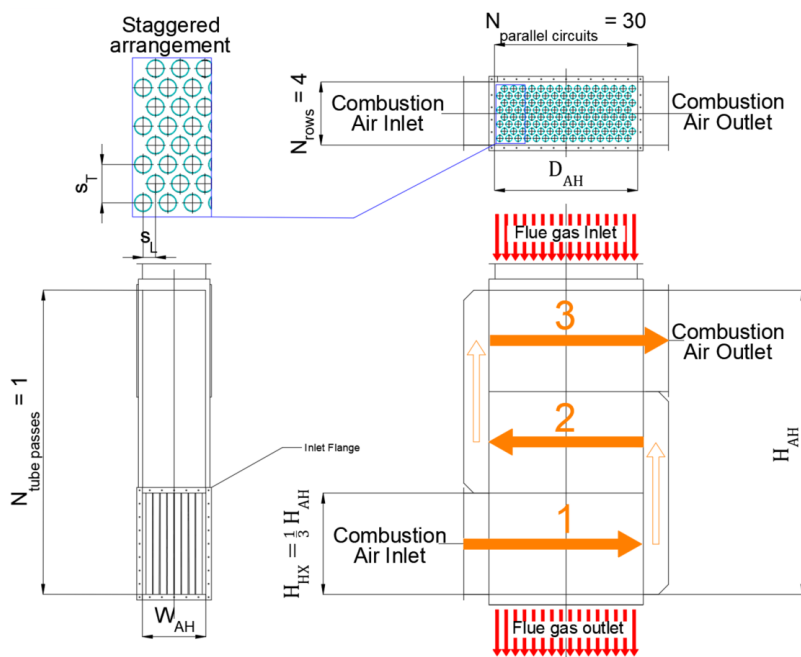


Figure 46: Air heater general arrangement

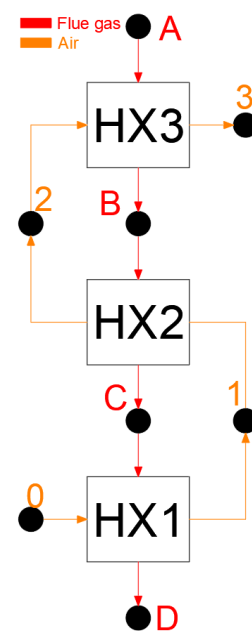


Figure 47: Discretised air heater arrangement – air flow (0-3) and flue gas flow (A-D)

H_{AH} in Figure 46 above refers to the total height of the air heater which is equivalent to the length of a single tube pass length of which the air heater has only one. Since the fin side fluid (air) has three passes, the overall air heater will be discretised into three sub heat exchangers as illustrated in Figure 47, that is one sub heat exchanger per air side pass. Consequently, there are also 3 sets of inlet and outlet pairs on the flue gas side (A-D). For this reason, the H_{HX} variable as seen in Figure 46 has been defined as one third of the total air heater height and consequently one third of a single total tube pass length.

Since this is also a cross-flow heat exchanger, the same methodology as employed for the economiser is used here. The staggered tube arrangement means that the calculation of the free-flow area (minimum area at which maximum fluid velocity occurs) is now considered to be normal to the diagonal tube pitch (see Figure 48 below) as opposed to the transverse pitch which was used for the economiser's inline tube arrangement.

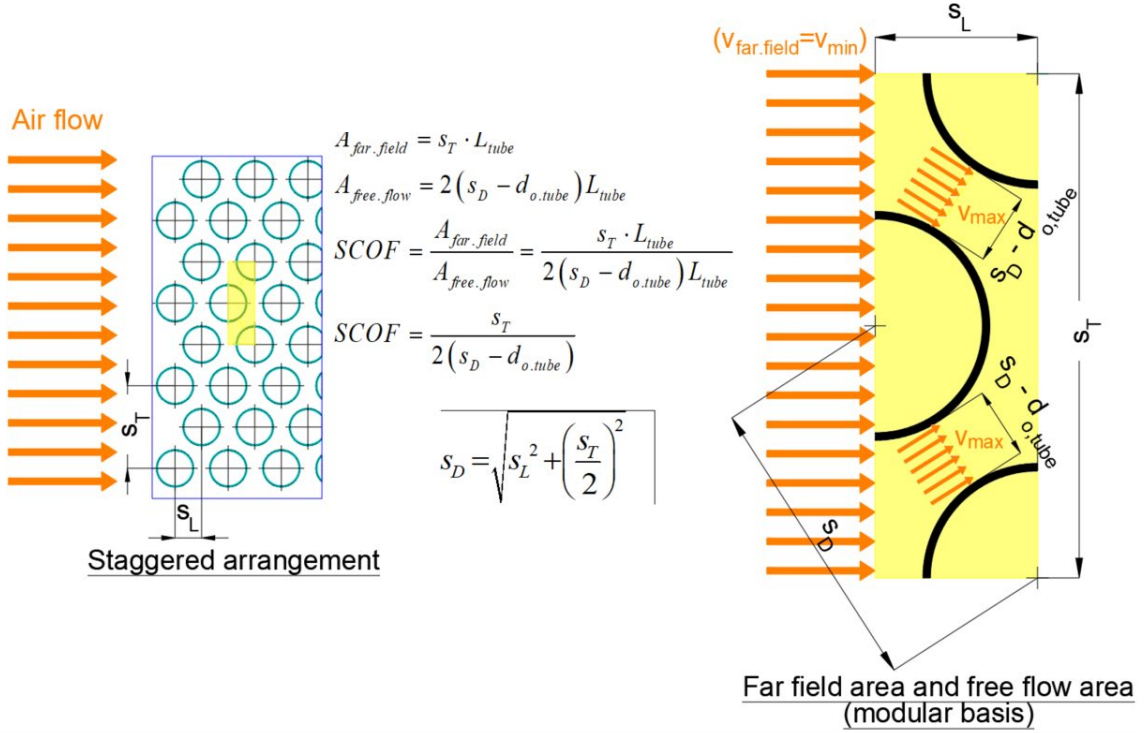


Figure 48: Air heater far-field and free-flow areas

In short, the free flow area shown above on a modular basis is two times the projected area at the smallest distance between tubes. This smallest distance is found by connecting the tube outer diameters perpendicularly along the transverse pitch plane.

The SCOF is determined as shown in Figure 48 and since the tubes are finless, the ROS as defined in equation (3.12) is unity. With all the geometric parameters defined, the Nusselt number is calculated using equation (3.11) for a staggered tube-bank arrangement. As with the economiser analytical model, the air heater model also uses the Gnielinski correlation as per equation (3.17) on the tube side fluid (flue gas side) as well as the same ϵ -NTU correlation as per equation (3.5) noting that the fluid with the minimum heat capacity that also undergoes turbulence via flow through the tube obstruction is now the combustion air.

The same successive approximation algorithm as used for the economiser is used for the air heater, but now there are 3 sets of exit temperatures to update due to the 3-element discretisation. The guessed outlet temperatures for the initialisation of the algorithm were accomplished by guessing the outlet temperatures of both streams from the overall air heater heat exchanger and then assuming a linear temperature profile to specify the initial 2 intermediate outlet temperatures. The high-level solution algorithm is illustrated in Figure 49 below and the detailed analytical analysis can be seen in Appendix B.

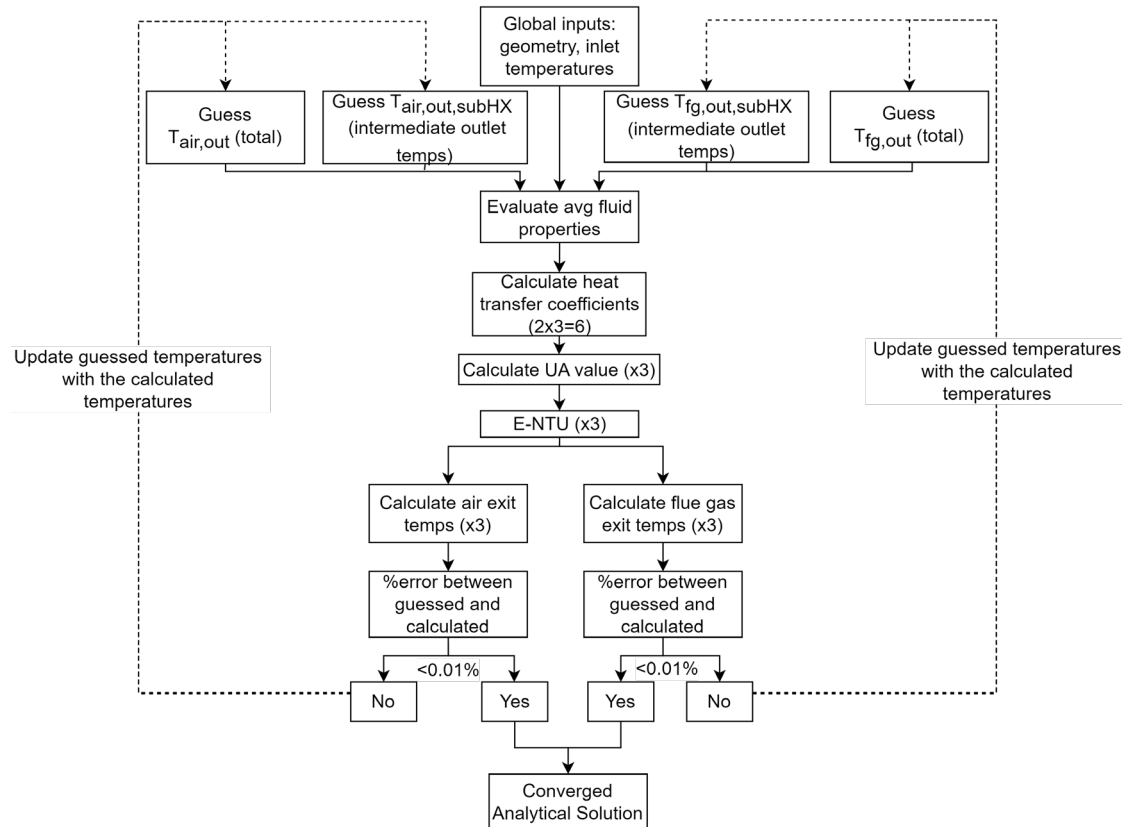


Figure 49: Air heater successive approximation algorithm used in Mathcad analytical model

The heat transfer rate computed using the inlet and outlet temperatures from the MCR data was 257.95 kW, while the analytical Mathcad model computed heat transfer rate was 235.52 kW. There is therefore verification agreement to within 8.70%. This is a reasonable result since the analytical model used air properties to approximate the flue gas properties. It must also be noted that the MCR data are predicted results from the Power Plant Simulator & Designer (PPSD) design software used by John Thompson. A good agreement between the MCR and analytical results therefore merely confirms that both models employ similar correlations and methods. John Thompson couples their inhouse boiler modelling methods with typical values such as unburnt carbon amounts from industry experience to tune certain efficiency factors in the PPSD environment. Assuming that the PPSD implementation has been validated to produce credible results, one can conclude that the analytical models also produce credible results.

A full comparison of the verification results between the MCR data, Mathcad analytical model and Flownex numerical model for the air heater is presented in the tables at the end of section 4.2.2.

4.1.3 Fire-Tube Evaporator

The firetube tube evaporator is like a shell and tube heat exchanger since there is a large volume of fluid (saturated water) inside the shell with a separate fluid (flue gas) which makes a single pass

inside of the 116 firetubes as illustrated in Figure 50. Since the water is saturated inside of the shell, there is a water level with saturated liquid at the bottom and saturated steam separated to the top. The average water-level set-point is designed to operate at 560mm above the horizontal centre of the shell.

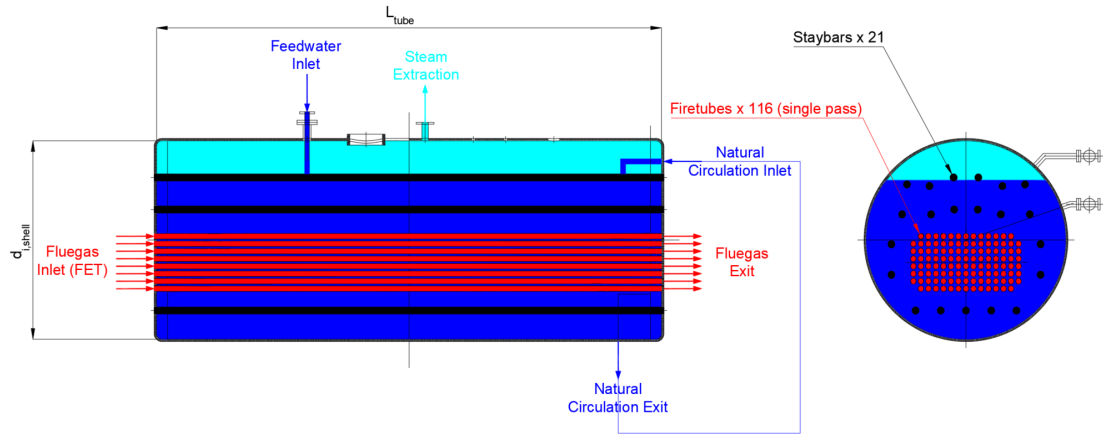


Figure 50: Fire-tube evaporator general arrangement

The Gnielinski equation for tubular flow as per equation (3.17) was used to model the forced convection of the flue gas flowing inside the tubes, while the Churchill and Chu correlation for natural convection as per equation (3.22) was used to model the heat transfer from the tubes to the surrounding water inside the evaporator shell.

Note that the Gorenflo correlation as per equation (2.5) with boiling buoyancy correction as per equation (2.6) implemented by P. Klein et al [1] in their Flownex model of a pure fire-tube boiler was not implemented in the present work. This is mainly due to the flue gas side limiting the overall heat transfer as its local heat transfer coefficient is $59.881 \text{ W/m}^2\text{K}$, compared to $1509.048 \text{ W/m}^2\text{K}$ for the water side. Including small buoyancy effects would therefore have an insignificant contribution to the overall heat transfer coefficient (U) in equation (3.8).

Any heat added to the saturated water inside of the evaporator would only go towards a phase change. This means that the water essentially has an infinite heat capacity at saturated conditions. As such the ϵ -NTU correlation for a single stream (i.e. flue gas) as per equation (3.6) was used to calculate the conductance (UA) value.

The MCR data available for the evaporator included the inlet feedwater mass flowrate, temperature and pressure as well as the main steam pressure (i.e. saturated drum pressure, $P_{\text{sat}} = 3101.325 \text{ kPa}$). However, only the exit flue gas side condition ($6984 \text{ kg/h @ } 338^\circ\text{C}$) is known. This means that the successive approximation algorithm this time needs to guess and iteratively update both the inlet flue gas temperature (i.e. furnace exit temperature FET) as well as the external tube wall temperature which drives the natural convection on the waterside as illustrated in Figure 51.

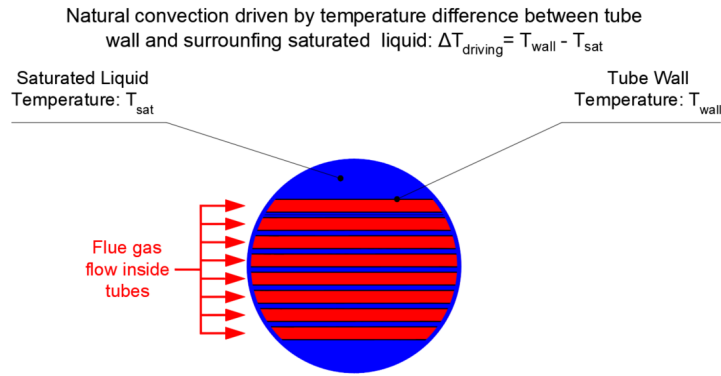


Figure 51: Close-up view of saturated liquid surrounding the fire-tubes

Since the water is saturated inside of the evaporator, one expects the entire drum to be at thermal equilibrium. As such a wall temperature of 0.5°C more than the saturated water temperature was chosen as a first approximation. The algorithm is summarised in Figure 52 below and the reader can refer to Appendix C for the full analytical calculation and partial verification.

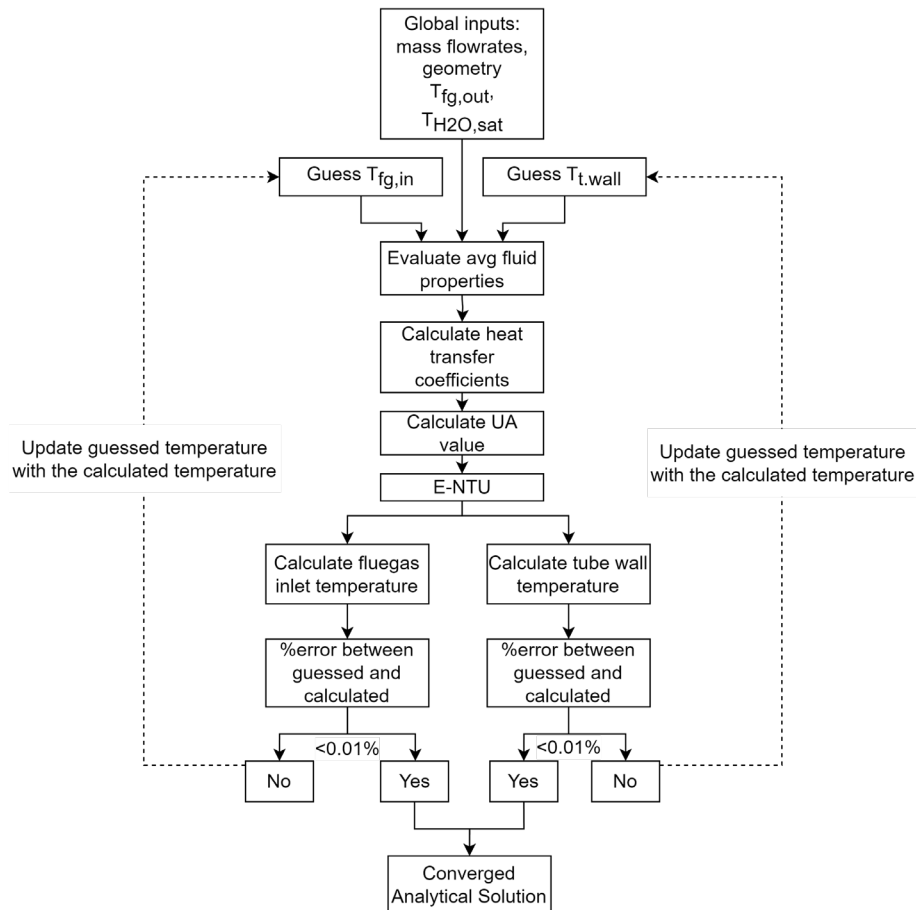


Figure 52: Fire-tube evaporator successive approximation algorithm used in Mathcad analytical model

The converged analytical evaporator heat transfer rate was 1327.055 kW, and the calculated analytical inlet flue gas temperature and tube wall temperature were 946.089°C and 245.663°C respectively. This implies a natural circulation driving temperature difference between the tube wall and the surrounding saturated liquid of 9.96°C.

Since the flue gas inlet condition was not reported in the available MCR data, the MCR evaporator heat transfer rate cannot be computed. Consequently, direct verification between the analytical result and the MCR data is not possible. In fact, through consultation with the John Thompson design engineers, there seems to be quite a large uncertainty of what the FET would be. Given that the correlations and method used are well established in heat transfer technology, there is reasonable confidence in the model. The analytical model can however still be used to verify the Flownex evaporator implementation. The results of this partial verification between the evaporator analytical and Flownex model is summarised the tables at the end of section 4.2.3.

4.1.4 Water-Cooled Furnace

The water-cooled furnace is the central (and most complex) heat exchanger of the whole boiler since the fuel, air, water and flue gas streams are all present. For the TORRIPAC architecture, the fuel (bagasse) is injected via a box ram through an opening in the front furnace wall. The fuel slides down the lower front wall onto grate. The primary combustion air is injected into the furnace through nozzles in the grate. The combustion flue gas rises up in the furnace and exits through a membrane shield (consisting of a number of water-filled pipes spaced some distance apart) at the furnace exit. The flow path is illustrated in Figure 53 below.

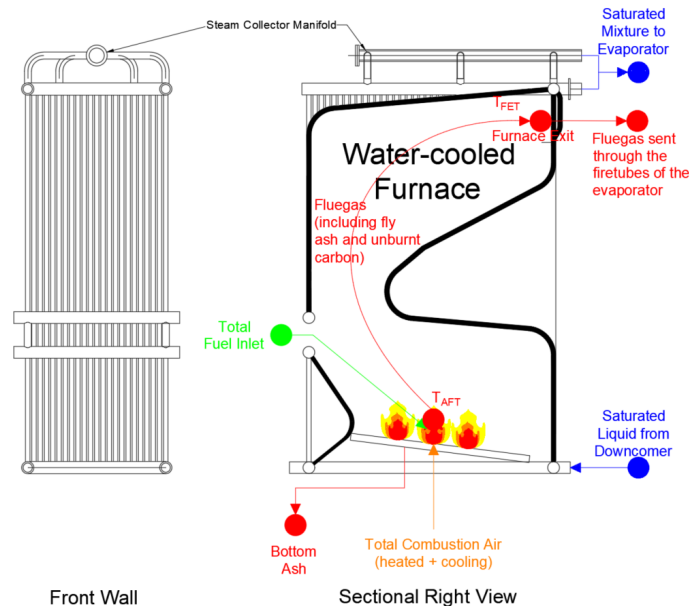


Figure 53: Water-cooled furnace general arrangement and fluid flow paths

An analytical model for the furnace was not developed in detail since the transfer of methodology into the Flownex model would not be a direct one. The main reason for this is because there are a few unknowns from the MCR data such as the fuel composition which can heavily affect the combustion heat, flue gas composition and flue gas fluid properties. Additionally, the furnace exit temperature was unknown.

It has already been qualified in sections 3.2 and 3.3.8.7 that the furnace model methodology in the present work leans heavily on calibration of the *Fluid Radiation* component used in the Flownex model. This section therefore only presents the combustion stoichiometric methodology used to determine the adiabatic flame temperature.

Adiabatic flame temperature calculation:

From the MCR data there are separate mass flowrates for the grate cooling air and the heated air. The split ratio is 10% of the total combustion air supplied as cooling air.

For the purposes of the TORRIPAC adiabatic flame temperature control volume, one should only be concerned with sensible heat coming into the furnace. Therefore, the assumption of instantaneous combustion need not separate the primary and secondary air.

The total combustion air is simply modelled as the mixture of the total grate cooling air and heated air as shown in Figure 54. The energy inputs include the sensible fuel energy and total combustion air sensible energy. After the combustion, the combustion zone loses heat via the hot flue gas exiting at the flame region. As discussed in section 3.2, the fly ash and unburnt carbon get carried out through the back pass with the flue gas while the bottom ash is propelled out at the base of the furnace by the grate illustrated in Figure 54.

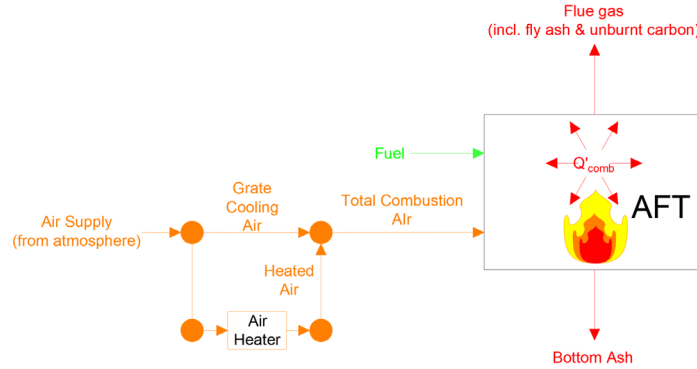


Figure 54: Adiabatic Flame Temperature (AFT) energy balance in combustion zone control volume

The energy balance equation in the combustion zone has been broken down into its lowest level in equation (4.1).

$$\begin{aligned} & \dot{m}_{fuel} h_{fuel}(T_{fuel}) + \dot{m}_{comb,air} h_{comb,air}(T_{comb,air}) + \dot{m}_{fuel} CV_{fuel} \\ & = \dot{m}_{fg,net} h_{fg}(T_{AFT}) + \dot{m}_{FA} h_{FA}(T_{AFT}) + \dot{m}_{BA} h_{BA}(T_{AFT}) + \dot{m}_{UC} HHV_C \end{aligned} \quad (4.1)$$

The terms on the left-hand side show sensible fuel and air energies coming into the control volume with enthalpies evaluated at their respective inlet temperatures. The last term on the left-hand side is the combustion heat calculated with a higher calorific value (CV or HHV) based on the fuel composition on a mass basis. The terms on the right hand side of equation (4.1) represent the energy leaving the control volume as hot flue gas, fly ash and bottom ash. The last term on the right-

hand side ($\dot{m}_{uc}HHV_c$) accounts for the reduced combustion heat due to unburnt carbon where the mass fraction of unburnt Carbon is assumed to be 0.61%.

The Mathcad model is presented in Appendix D to show a full implementation of the whole boiler energy balance and combustion analysis.

The Mathcad sheet made use of a flue gas property reference sheet for the evaluation of the fuel and flue gas properties. This reference sheet was developed in-house (ATProM) and includes fluid property polynomials for viscosity, thermal conductivity, enthalpy, and constant pressure specific heat capacity extracted from the NIST data base for CO₂, O₂, H₂O and SO₂. Note that the enthalpy polynomials were for a 25°C reference temperature. This is common practice in boiler process models to be consistent when employing the higher heating value (HHV) in the combustion calculations.

Fluid property mixture functions were defined in the reference sheet. The density mixing function assumes an ideal gas approximation requiring the minimum inputs of temperature, pressure and gas composition. The enthalpy and constant pressure specific heat capacity mixing function requires only temperature and gas composition in their mass fractions. Lastly, the thermal conductivity and viscosity mixing function uses the Wilke mixing method, requiring minimum inputs of temperature and gas composition.

Importantly the reference sheet can calculate solid phase properties as is necessary for the solid fuel (bagasse in the present work). The reference sheet was set-up to calculate the solid phase properties of coal making use of common values in literature for the specific heat capacity of coal at 1.38 kJ/kgK and fly ash at 0.73 kJ/kgK. These values are also valid for other carbon-based solid-fuels such as bagasse in the present work. This enables the calculation of the enthalpy of the fuel and the fly ash as a function of their temperatures as shown in equations (4.2) and (4.3).

$$h_{fuel}(T_{fuel}) = c_{fuel}(T_{fuel} - T_{ref,25^\circ C}) \quad (4.2)$$

$$h_{FA}(T_{FA}) = c_{FA}(T_{FA} - T_{ref,25^\circ C}) \quad (4.3)$$

The results of the analytical furnace model could not be validated to MCR data, due to a lack of furnace results. Also, the method employed in Flownex uses the ATProM combustion component, and solving the energy balance is performed by the Flownex solver. As such, the analytical model merely served as a “sanity check” when looking at the Flownex model results and completed the picture of the whole boiler analytical model. Insights gained in developing the model were valuable in “debugging” the Flownex model.

4.2 Steady State Numerical Model Development (Flownex)

4.2.1 Economiser

The *Finned-Tube Heat Exchanger* component in Flownex was the best fit for the economiser since it can accurately account for the true geometry and arrangement of the heat exchanger and more importantly account for the thermal mass of all the components (fins and tubes).

As explained in 3.3.8.5, the *Finned-Tube Heat Exchanger* in Flownex is a two-part component consisting of a fin-side and a tube-side, each with their own unique inputs. Each side requires a minimum number of inputs which include aspects of geometry, material data, losses, and heat transfer. These are higher-level inputs which require some preliminary calculations. These preliminary calculations are therefore done using a *Quick Script* (coded by the user in C#) in Flownex following the same methodology as the analytical Mathcad model.

Relevant results from the *Quick Script* are then transferred to the inputs of each side of the *Finned-Tube Heat Exchanger* component via *Data-Transfer Links* (dashed lines in Figure 55 below). Note that the implementation in Figure 55 uses air as the fluid to represent the flue gas on the fin side and that the pressure, temperature, and mass flowrate boundary conditions are that of the MCR design data. This was done to verify the Mathcad analytical model which also assumed air properties on the flue gas side.

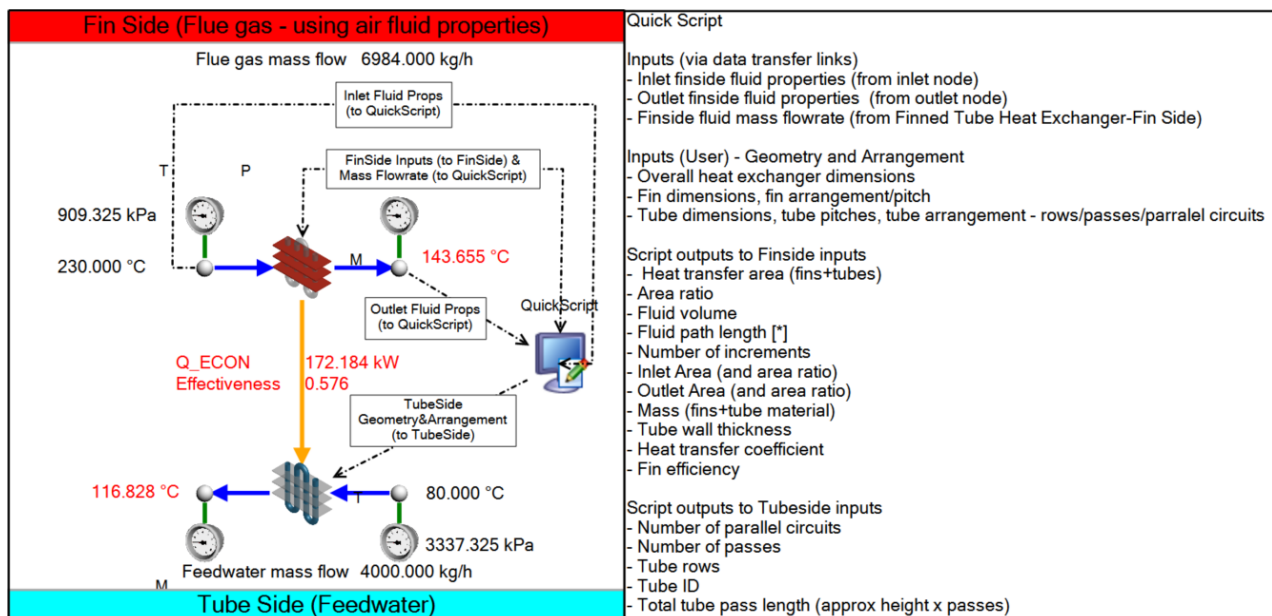


Figure 55: Economiser Flownex model (left), Quick Script inputs & outputs (right)

Figure 56 shows the input values for the fin-side and tube-side respectively, where the grey highlights indicate inputs which are fixed from the *Quick Script* via the *Data-Transfer Links*.

Figure 56: Fined-Tube Heat Exchanger inputs for fin-side (left) & tube-side (right)

All inputs on the fin-side barring the heat transfer coefficient and fin efficiency are analytical fixed values. The fin-side heat transfer coefficient and fin efficiency are both functions of the average fluid properties which are calculated in the *Quick Script*. The friction factor of 0.04 on the fin side is somewhat arbitrary since Flownex requires at least a primary or secondary loss to be specified. Other friction factor values have insignificant effects. Calculation of the tube-side heat transfer coefficient is accomplished by specifying Flownex's built-in Gnielinski correlation.

The tube arrangement is: 6 tube rows, 14 passes, and 1 parallel circuit as shown in Figure 57 which conforms to the Flownex convention for tube arrangement as illustrated in Figure 58.

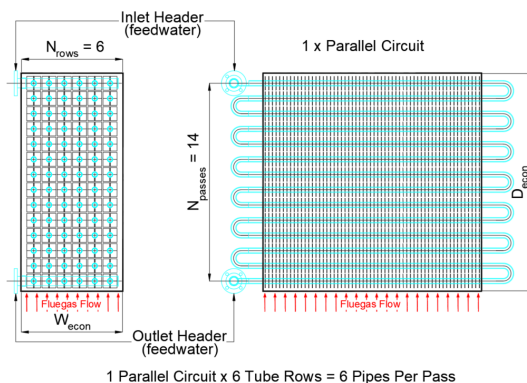


Figure 57: True TORRIPAC economiser tube arrangement

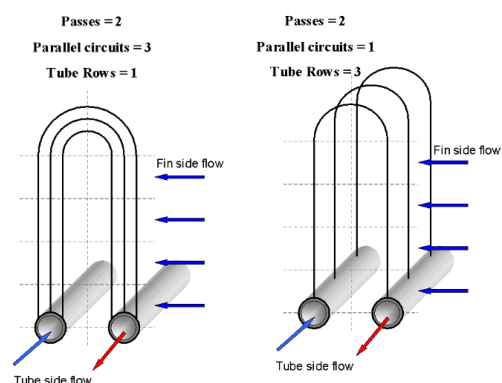


Figure 58: Flownex tube arrangement convention

In the true TORRIPAC economiser arrangement (Figure 59), one sees that the tube side fluid (water) heats up in the opposite direction to the flue gas flow and the flue gas cools down from inlet to outlet. This is an overall counter flow regime as per the Flownex convention illustrated (Figure 60).

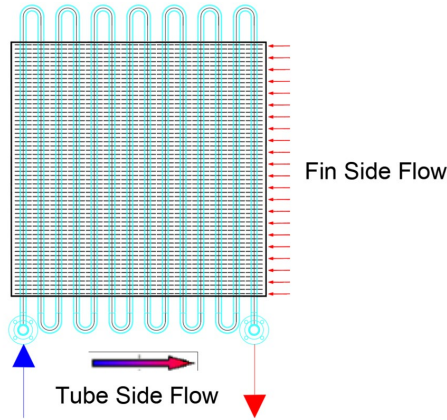


Figure 59: TORRIPAC economiser assembly flipped 90 degrees to show counter-flow regime

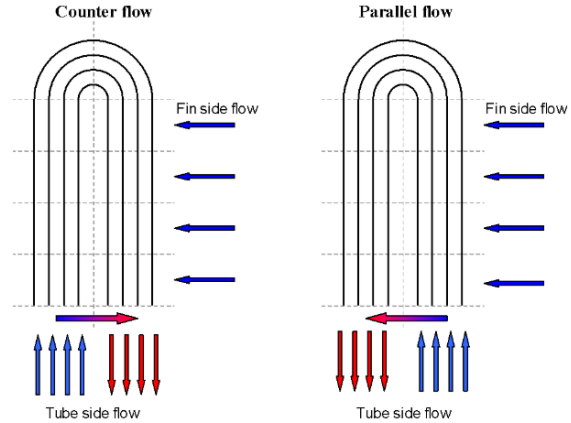


Figure 60: Flownex convention for overall flow regime for cross-flow over tube banks

The fouling factors on both sides were tested with values as per the TEMA guidelines mentioned in section 3.1.1. Fouling in these ranges proved to have negligible effects on the overall results. Thus fouling factors of zero were used for continuity between the analytical and numerical models.

The full economiser verification is detailed in Appendix A with key results presented in Table 1.

Table 1: Economiser verification results - fluid exit temperatures and heat transfer rates

Flue gas					Water				
	MCR	Analytical (air props)	Flownex 1 (air props)	Flownex 2 (flue gas props)		MCR	Analytical	Flownex 1	Flownex 2
Parameter	Unit	Value	Value	Value	Parameter	Unit	Value	Value	Value
Mass flow*	kg/s	1.94	1.94	1.94	Mass flow*	kg/s	1.11	1.11	1.11
Inlet temperature*	K	503.15	503.15	503.15	Inlet temperature*	K	353.15	353.15	353.15
Inlet pressure*	kPa(a)	909.33	909.33	909.33	Inlet pressure*	kPa(a)	3337.33	3337.33	3337.33
Exit temperature	K	427.15	418.78	416.81	Exit temperature	K	391.15	389.14	389.98
Exit pressure	kPa(a)	1008.33	909.32	909.32	Exit pressure	kPa(a)	3284.33	3336.54	3336.54
Heat Transfer Rate	kW	177.69	168.19	172.18	Heat Transfer Rate	kW	177.64	168.19	172.18

Note the starred items (*) in Table 1. These indicate the input boundary conditions that were taken from the MCR data and applied to each of the models.

Recall that the analytical model uses air properties to approximate the flue gas properties. As such, two configurations of the Flownex model were run – Flownex 1, using air properties for flue gas and Flownex 2 using flue gas properties by specifying the flue gas composition as per Figure 61. The composition in Figure 61 was the result of a stoichiometric combustion calculation performed in the ATProM combustion component. It was based on an assumed fuel composition since no data on the true bagasse composition used to generate the MCR data was available.

Mass Fraction	
Specify mass fraction	Yes
Mass fractions	1
SO2	2.1739E-19
Fly ash	0.000982234
Unburnt carbon	0.00112597
CO2 - Carbon Dioxide	0.111231
N2 - Nitrogen	0.60349
NO - Nitric Oxide	0
O2 - Oxygen	0.0437381
H2O - Water	0.239433

Figure 61: Flue gas composition employed in Flownex 2 model based on an assumed fuel composition

The reason for running two Flownex configurations was to allow a more direct verification between the analytical model and Flownex 1 model. If there is good agreement between the analytical model and Flownex 1 model, then one can have confidence that the Flownex implementation is accurate, allowing a more direct verification between the Flownex 2 model with real flue gas properties to the MCR data which would have also used real flue gas fluid properties. It should be noted that the Flownex 1 and Flownex 2 models are identical apart from the fluid specification on the flue gas side.

The employed flue gas composition is of course still a result of an assumed fuel composition and air-fuel-ratio but will still produce more accurate results truer to the MCR heat transfer rates. It is mainly the presence of significantly higher moisture (H₂O) that affects the fluid properties and ultimately the heat transfer rate results especially at elevated flue gas temperatures.

For completeness, the verification of all the models can be seen more clearly in the percentage error pairwise comparisons of the fluid exit temperatures and heat transfer rates as summarised in Table 2, Table 3, and Table 4 respectively.

Table 2: Economiser flue gas exit temperature [K] %error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		0.47%	-2.45%	-1.96%
Flownex 1	-0.47%		-2.91%	-2.42%
Flownex 2	2.52%	3.00%		0.51%
MCR	2.00%	2.48%	-0.50%	

Table 3: Economiser water exit temperature [K] %error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		-0.22%	-0.88%	-0.51%
Flownex 1	0.22%		-0.67%	-0.30%
Flownex 2	0.89%	0.67%		0.37%
MCR	0.52%	0.30%	-0.37%	

Table 4: Economiser heat transfer rate [kW] %error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		-2.32%	-8.82%	-5.35%
Flownex 1	2.37%		-6.65%	-3.10%
Flownex 2	9.67%	7.13%		3.81%
MCR	5.65%	3.20%	-3.67%	

Green table cells indicate errors below 5%, orange between 5% and 10% and red greater than 10%. The tables should be read, row to column. For example, the heat transfer rate as per the analytical Mathcad model agrees to within -5.35% of the MCR heat transfer rate as per Table 4. Similarly, the Flownex 2 model agrees with the MCR data to within 3.81%. This shows an improvement in the verification with the MCR data by virtue of using real flue gas properties in the Flownex 2 model. The same improvement is observed in the fluid exit temperatures in Table 2 and Table 3. It should be emphasised once more that if the exact flue gas composition used to generate the MCR data was

known and therefore specified in the Flownex 2 model that the Flownex 2 vs MCR error could be improved (i.e. minimised further) since the Flownex model implements best practice heat transfer correlations. This verification provides confidence that the stand-alone Flownex 2 model is ready to be integrated into the whole-boiler Flownex network of the TORRIPAC.

4.2.2 Air Heater

Although the air heater does not have fins, Flownex's *Finned-Tube Heat Exchanger* component was still used due to its material thermal storage capabilities. The finless tubes were simply considered by specifying a unity fin efficiency.

The 3 fin-side (air-side) passes were employed by connecting 3 heat exchangers in series (one *Finned-Tube Heat Exchanger* per air side pass). The overall air heater is therefore discretised into 3 sub heat exchangers. The geometry for all 3 heat sub heat exchangers is identical and care was taken to ensure conformance to the Flownex tube arrangement conventions regarding the number of parallel circuits and number of tube rows as illustrated for a single air side pass control volume (i.e. sub-heat exchanger) in Figure 62.

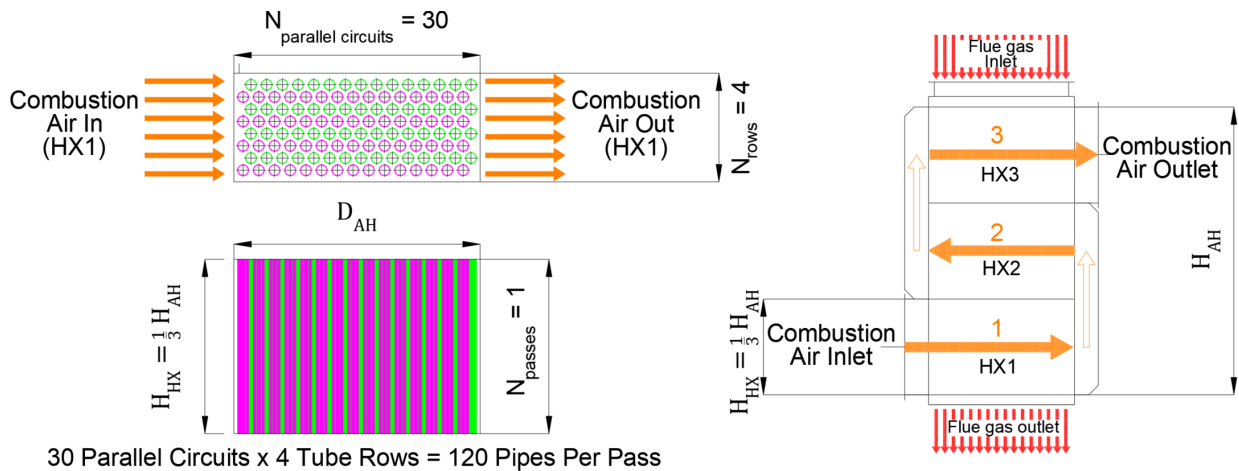


Figure 62: Sub-heat exchanger (HX1) illustrating conformance to the Flownex tube arrangement convention

Note that the sub-heat exchanger represented in Figure 62 is for the first air side pass (HX1) with flow from left to right. The arrangement for the remaining air side passes (HX2 & HX3) would be identical except for the air flow direction; right to left for HX2 and then returning left to right for HX3. Each sub-heat exchanger undergoes cross flow over a single tube pass, so the overall flow regime, whether specified as counter or parallel flow made no difference to the results. This makes sense since the 3-part overall discretisation already accounts for each air side pass independently.

Since the air heater contains 3 *Finned-Tube Heat Exchanger* components in series, it was necessary to employ one *Quick Script* per *Finned-Tube Heat Exchanger* component ("QuickScript1", "QuickScript2", and "QuickScript3" in Figure 63 below). "QuickScript0" is a global *Quick Script* which

takes in the overall air heater geometry and arrangement and outputs the geometric parameters for each of the 3 sub-heat exchangers which are transferred via *Data Transfer Links* to the 3 *Quick Scripts* downstream. *Quick Scripts* 1, 2, and 3 also take in the fluid properties from the relevant inlet and outlet nodes such that the heat transfer coefficient for both streams of each of the 3 heat exchangers can be calculated, ultimately being transferred to the inputs to the fin-side of each heat exchanger. The calculations done in these 3 *Quick Scripts* are the same as the analytical procedure explained in 4.1.2, whose full details can be seen in Appendix B.

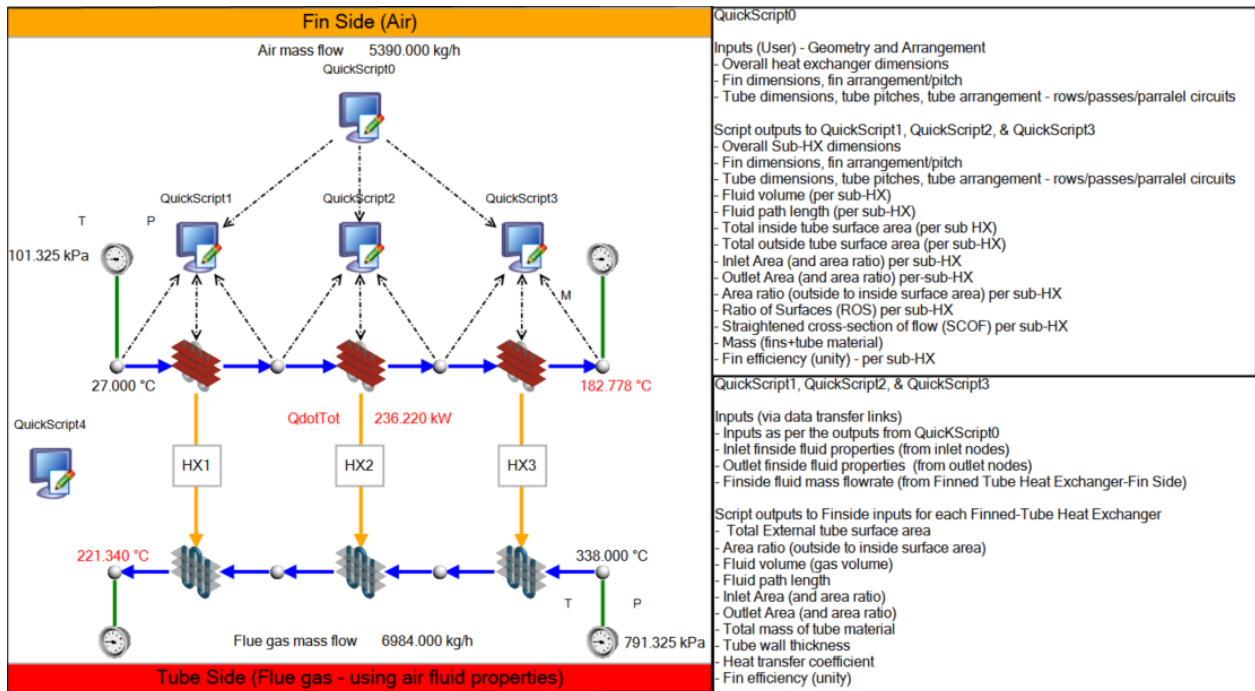


Figure 63: Air heater Flownex model (left), Quick Script inputs & outputs (right)

Lastly, “QuickScript4” is a post processing script which calculates the total air heater heat transfer rate as the sum of the 3 heat exchanger results. The *Data Transfer Links* to “Quickscript4” have been hidden to not clutter the canvas.

Note that the implementation in Figure 63 uses air as the fluid to represent the flue gas on the tube side and that the pressure, temperature and mass flow boundary conditions are that of the MCR design data. This was done to verify the Mathcad analytical model which also assumed air properties on the flue gas side.

Figure 64 shows the inputs for the fin-side and tube-side of HX1 which is the left most *Finned-Tube Heat Exchanger* with respect to Figure 63. Again, the grey highlights indicate inputs which have been transferred from the *Quick Scripts*. The inputs for HX2, and HX3 are identical to HX1 except for the heat transfer coefficient which would be a function of the average fluid properties relevant to each sub-heat exchanger. The built-in Flownex Gnielinski pipe flow correlation is used on the tube side.

Finned-Tube Heat Exchanger - Fin Side - 0 C...

- General**
 - Identifier: Finned-Tube Heat Exchanger - ...
 - Description:
 - Solving: ☒
- Connected Nodes**
 - Upstream node: Node - 4 Copy(003)
 - Downstream node: Node - 18 Copy(003)
- Fluids**
 - Fluid data reference: Air | Gases (Pure Fluids) (ATPr...
- Specification**
 - Specification option: General Specification
- Fin (Primary) Side Geometry**
 - Flow Direction: Counter
 - Heat transfer area: 23.9389 m²
 - Area ratio: 1.06723
 - Fluid volume: 0.403969 m³
 - Fluid path length: 1.4 m
 - Number of increments: 1
- Inlet**
 - Area: 0.56 m²
 - Area ratio: 0.281665
- Outlet**
 - Area: 0.56 m²
 - Area ratio: 0.281665
- Material Data**
 - Material option: Select from data reference
 - Mass: 363.169 kg
 - Wall thickness: 0.002 m
 - Material selection: Carbon Steel - AISI 1010 | Met...
- Losses**
 - Primary Losses**
 - Primary loss type: Specify friction factor
 - Friction factor: 0.04
 - Secondary Losses**
 - Using advanced model: No
- Heat Transfer**
 - Convection Coefficient Spe...
 - Convection**
 - Convection coefficient op...: Constant h
 - h: 70.4578 W/m².K
 - Fin efficiency: 1
 - Fouling factor: 0 m².K/kW
- Fixed Options**
- Inputs**
- Results**

Finned-Tube Heat Exchanger - Tube Side - 1...

- General**
 - Identifier: Finned-Tube Heat Exchanger - ...
 - Description:
 - Solving: ☒
- Connected Nodes**
 - Upstream node: Node - 31 Copy(003)
 - Downstream node: Node - 10 Copy(003)
- Fluids**
 - Fluid data reference: Air | Gases (Pure Fluids) (ATPr...
- Tube (Secondary) Side Geometry**
 - Number of parallel circuits: 30
 - Number of passes: 1
 - Tube rows: 4
 - Tube inside diameter: 59.5 mm
 - Total tube pass length (~Heig...): 1 m
- Losses**
 - Primary Losses**
 - Primary loss type: Darcy Weisbach
 - Roughness option: Select from database
 - Roughness selection: Seamless steel tubes - Ne...
 - Roughness: 100 μm
 - Secondary Losses**
 - K actual inlet: 0
 - K actual outlet: 0
- Heat Transfer**
 - Convection Coefficient Spe...
 - Convection**
 - Convection coefficient op...: Calculate h
 - Convection coefficient co...: Tube inside (Gnielinski)
 - Fin efficiency: 1
 - Fouling factor: 0 m².K/kW
 - Tube roughness option: Obtain from flow component
 - Nu_ref = C*((f/8).(Re-1000).Pr)/(1+12.7.(f/8)^1/2.(Pr^...**
 - C: 1
 - Zero flow Nusselt number: 4
 - Zero flow Reynolds num...: 0
 - Laminar Nusselt number: 4.364
 - Nu = Nu_ref.(mu/mu_w)^e.(rho_w/rho)^f.(c_p_avg/c_...**
 - e: 0
 - f: 0
 - g: 0
 - i: 0
 - j: 0
- Inputs**
- Results**

Figure 64: Finned-Tube Heat Exchanger inputs for HX1 fin-side (left) and tube-side (right)

The full air heater verification details can be found in Appendix B with a summary of the results given in Table 5 below.

Table 5: Air heater verification results - fluid exit temperatures and heat transfer rates

Air						Flue gas					
		MCR	Analytical	Flownex 1	Flownex 2			MCR	Analytical (air props)	Flownex 1 (air props)	Flownex 2 (flue gas props)
Parameter	Unit	Value	Value	Value	Value	Parameter	Unit	Value	Value	Value	Value
Mass flow*	kg/s	1.497	1.497	1.497	1.497	Mass flow*	kg/s	1.94	1.94	1.94	1.94
Inlet temperature*	K	300.15	300.15	300.15	300.15	Inlet temperature*	K	611.15	611.15	611.15	611.15
Inlet pressure*	kPa(a)	101.33	101.33	101.33	101.33	Inlet pressure*	kPa(a)	791.33	791.33	791.33	791.33
Exit temperature	K	470.15	455.52	455.93	471.29	Exit temperature	K	503.15	494.81	494.49	507.70
Exit pressure	kPa(a)	TBD	101.08	101.08	101.08	Exit pressure	kPa(a)	909.33	791.32	791.32	791.32
Heat Transfer Rate	kW	257.95	235.52	236.22	259.76	Heat Transfer Rate	kW	257.67	235.52	236.22	259.76

Note the starred items (*) in Table 5. These indicate the input boundary conditions that were taken from the MCR data and applied to each of the models.

Like the economiser verification, the air heater verification also has two Flownex configurations; Flownex 1 using air properties to approximate flue gas properties, and Flownex 2 using real flue gas properties for the same composition given in Figure 61.

The verification of all the models can be seen more clearly in the percentage error pairwise comparisons of the fluid exit temperatures and heat transfer rates as summarised in Table 6, Table 7, and Table 8 respectively.

Table 6: Air heater air temperature [K] %error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		-0.09%	-3.35%	-3.11%
Flownex 1	0.09%		-3.26%	-3.02%
Flownex 2	3.46%	3.37%		0.24%
MCR	3.21%	3.12%	-0.24%	

Table 7: Air heater flue gas exit temperature [K] %error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		0.06%	-2.54%	-1.66%
Flownex 1	-0.06%		-2.60%	-1.72%
Flownex 2	2.61%	2.67%		0.90%
MCR	1.69%	1.75%	-0.90%	

Table 8: Air heater heat transfer rate [kW] %error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		-0.30%	-9.33%	-8.70%
Flownex 1	0.30%		-9.06%	-8.42%
Flownex 2	10.29%	9.97%		0.70%
MCR	9.52%	9.20%	-0.70%	

There is good agreement (0.30%) between the heat transfer rates of the analytical model and Flownex 1 model since they both approximate the flue gas properties using air.

The effect of using accurate flue gas properties is even more pronounced in the air heater verification. This can be seen in Table 8 where the Flownex 2 model which uses flue gas properties has large errors of 10.29% and 9.97% when compared to the analytical and Flownex 1 models respectively which use air properties. Note the small error of 0.70% when verifying the Flownex 2 model with the MCR data. This indicates that the methodology of sequentially gaining confidence in verification between the analytical and Flownex 1 model such that verification between Flownex 2 and MCR is valid. The successful verification of Flownex 2 with the MCR data provides assurance that the stand-alone air heater Flownex model is ready to be integrated into the whole-boiler Flownex network of the TORRIPAC.

4.2.3 Fire-Tube Evaporator

The firetube evaporator model centres around the *Two-Phase Tank* component representing the full fluid volume (saturated water mixture). *Pipe* components are used for the various inlet and outlet water stream connections to the *Two-Phase Tank*. A *Pipe* component is also used for the fire tubes (i.e. flue gas side) with the heat transfer between the fire tubes and water inside the *Two-Phase Tank* being modelled with a *CHT* component employing natural convection on the water side,

forced pipe flow convection on the tube side as well as including conduction layers for the tube wall which will become important for the transient analysis.

Also important for the dynamic analysis is the inclusion of the thermal inertia of the evaporator shell and solid steel stay bars. The combined thermal mass of these two items have been lumped into a solid node with a volume and material specified with further explanation detailed in section 4.3.2. The preliminary standalone fire-tube evaporator Flownex Network is illustrated in Figure 65.

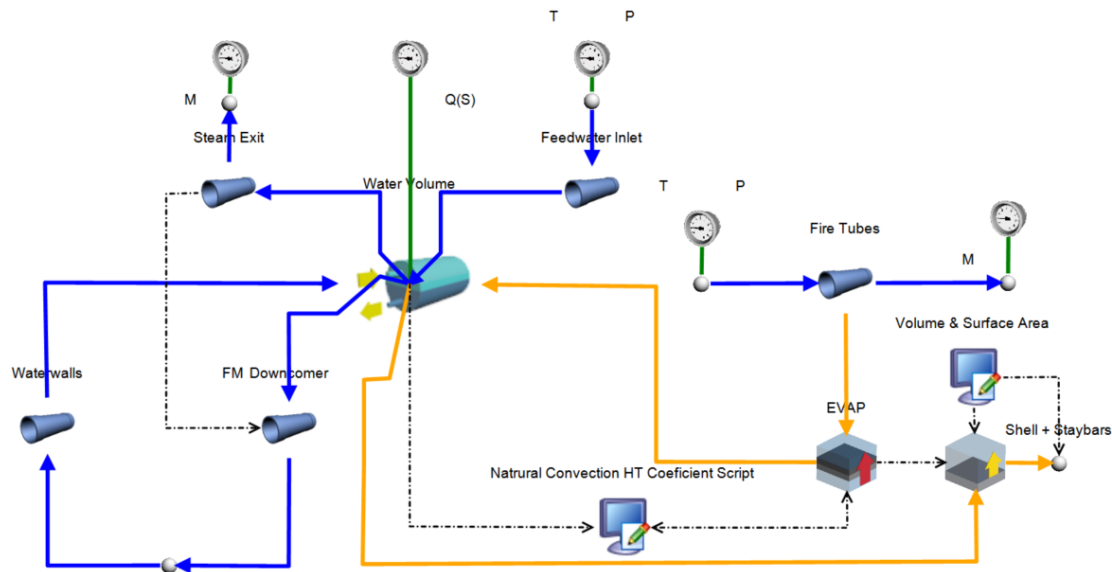


Figure 65: Fire-tube evaporator Flownex model

Two Phase Tank

To employ an accurate tank volume with the level tracking in mind, it was important to realise that the tank is not simply a horizontal cylindrical shell, since the firetubes and staybars also occupy some volume within the shell. To overcome this, the *Two-Phase Tank* volume was approximated using a polygon table as discussed in section 3.3.8.3 by slicing up the true fluid volume geometry into 1mm heights using CAD software.

Flownex requires that the polygon table be specified with height and area pairs. That is, for each strictly increasing height input starting at 0 m, a corresponding unique area be specified. In the case of 1mm slices, this is 1856 table entries from 0 m at the bottom of the tank to the total height 1.856 m (i.e. ID of the shell).

The true volume shape including curvature of end-caps was modelled in SolidWorks with extruded cuts at the locations of the firetubes and staybars as illustrated in Figure 66. The unique areas were extracted using SolidWorks at each height by evaluating the surface area of the top surface, after “extrude-cutting” the top side volume at each incremental height. The first three entries for the polygon table are shown in Figure 67 with a visual representation of the discretised continuous shape shown in Figure 68.

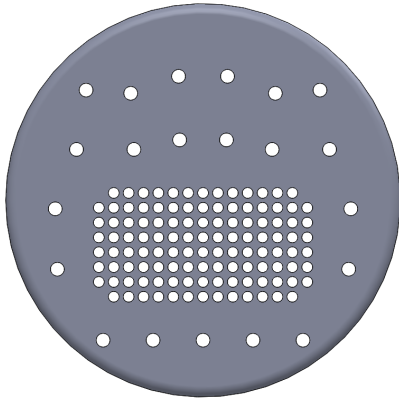


Figure 66: Right view of true volume shape (modelled in SolidWorks)

Two Phase Tank - 2B Copy(010) (Two Phase Tank...)		
Geometry		
Geometry option	Specify geometry	
Vessel shape	Polygon	
Table option	Use table	
Dimensions table	Click here to edit and remov...	
[0]		
Geometry		
Height	0	m
Area	0.563634	m²
[1]		
Geometry		
Height	0.001	m
Area	0.690845	m²
[2]		
Geometry		
Height	0.002	m
Area	0.798199	m²
[3]		
Inputs		
Results		

Figure 67: First three entries for Polygon Table of Two-Phase Tank in Flownex

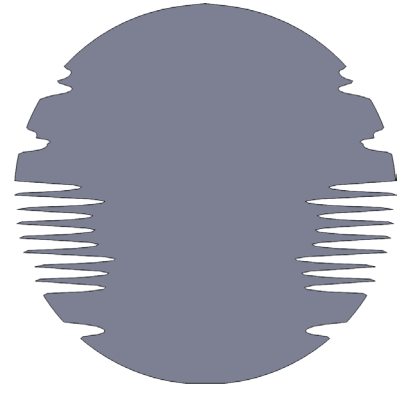


Figure 68: Right view visual representation of Flownex Polygon implementation

The total volume evaluated by SolidWorks of the true shape as per Figure 66 was 11.450 m³ and the volume computed by Flownex for the polygon table implementation agreed to within 0.0025%.

Knowing the saturated conditions as per the MCR data; $P_{\text{sat}} = 3101.325$ kPa and $T_{\text{sat}} = 235.703^\circ\text{C}$ it was possible to evaluate the saturated liquid density and saturated vapour density inside the evaporator. The volume of the liquid phase was evaluated in SolidWorks by extrude-cutting the vapour portion away at the average water-level set-point (1.488m above the bottom of the tank). The vapour volume is then simply the difference between the mixture volume and the liquid volume. The mixture quality inside the evaporator shell was then calculated to be 0.003448 as the ratio of vapour mass to mixture mass as shown in equation (4.4).

$$\text{Quality} = \frac{\rho_{\text{sat.vap}} V_{\text{sat.vap.Sworks}}}{\rho_{\text{sat.vap}} V_{\text{sat.vap.Sworks}} + \rho_{\text{sat.liq}} V_{\text{sat.liq.Sworks}}} \quad (4.4)$$

If the polygon table approximation is accurate then one expects the Flownex computed tank level to agree with the 1.488m level set-point provided that the 0.003448 quality constraint is imposed as a boundary condition on the *Two-Phase Tank*. Flownex underpredicted the average water level by 1.98mm (i.e. level of 1.486m). A 2 mm discrepancy is acceptable, but a level sensitivity analysis was then conducted to verify the level tracking at various heights inside the tank (i.e. various qualities).

The TORRIPAC control system includes a level controller to keep the average water level within a 2mm band of its set-point. At a water level 50mm below the AWL set-point, an alarm signal is sent to the Human Machine Interface (HMI). Audible alarms are tripped if the water level deviates from the AWL by 75mm or more with the relevant fans and grate speeds being stopped or reduced as illustrated in Figure 69.

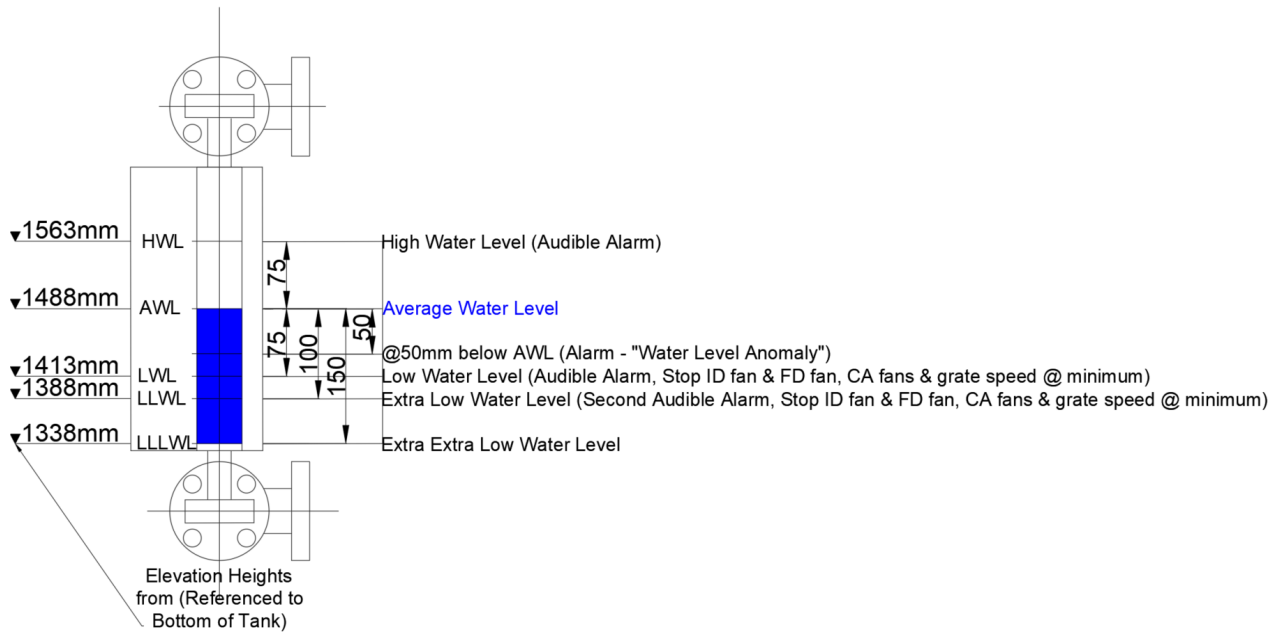


Figure 69: Water level meter installed onto evaporator of TORRIPAC

With the above picture in mind, the Flownex level sensitivity analysis was conducted for levels ranging from 1388mm (LLWL) to 1563mm (HWL) since the boiler should never operate at a water-level outside of this range. The average error between the Flownex computed level and the true level is 1.972mm with a minimum and maximum error of 1.964mm and 1.979mm respectively. The results for the level calibration over the above height range is shown in Figure 70.

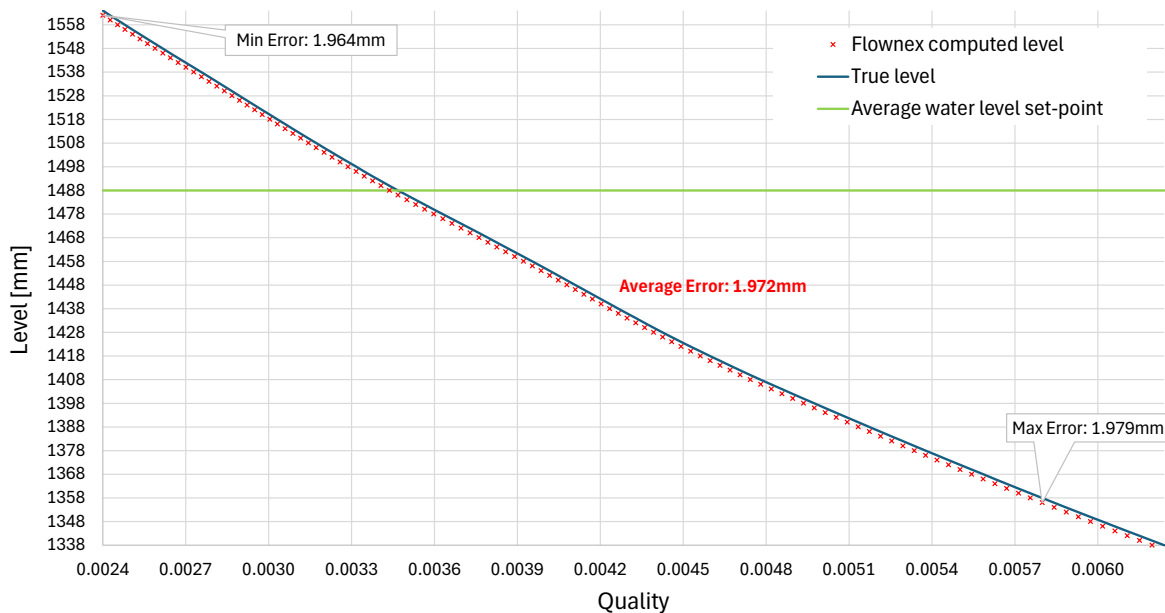


Figure 70: Flownex Two-Phase Tank level sensitivity results

This sensitivity analysis validates that the volume discretisation implemented through SolidWorks and Flownex is accurate within the operational level set-points of the TORRIPAC. The reader can refer to the level sensitivity results for the full evaporator height in Appendix C to see that the 2mm

offset holds true for the full evaporator height. Bear in mind that the discretisation methodology through transferring cross-sectional areas from SolidWorks to Flownex is a numerical discretisation. Flownex connects the areas at two successive heights with straight lines and therefore may not capture the true curvature of the volume, leading to the minor 2mm discrepancy possibly at the bottom of the vessel.

Pipes:

All pipes are specified with Darcy Weisbach as the primary loss type. “Welded Steel tubes – New or old, but in good condition, Max” was the roughness selection from the database in Flownex, and results in a roughness value of 100 microns.

An example of the input parameters is given for the downcomer pipes of the evaporator model in Figure 71 below. The full input list with motivations for each *Pipe* component can be reviewed in Appendix C.

Pipe - 4 Copy(001) (Pipe) Properties	
General	
Identifier	Pipe - 4 Copy(001)
Description	Downcomer
Solving	☒
Connected Nodes	
Upstream node	Node - 85 Copy(001)
Downstream node	Node - 10 Copy(002)
Fluids	
Fluid data reference	H2O - Water General (Two Ph...
Geometry	
Geometry option	Specify schedule
Database	DN: 150 Schedule: 40 STD ...
Wall thickness (from schedule)	0.00711 m
Profile option	Specify length
Length	3927 mm
Inlet	
Diameter (from schedule)	0.15408 m
Losses	
Primary Losses	
Primary loss type	Darcy Weisbach
Roughness option	Select from database
Roughness selection	Welded Steel tubes - New or old, but in good condition, Max
Roughness	100 µm
Secondary Losses	
Discretisation	
Number of increments	1
Number in parallel	2
Orifice	
Orifice diameter ratio	1
Heat Transfer	
Heat option	Fixed heat transfer
Heat input	0 kW
Heat Distribution Curve	
Use heat distribution curve	No
Transient and Water Hammer Related Inputs	
Wall option	Elastic
Young's Modulus option	Specify Young's Modulus
Young's Modulus	0 kPa
Restraint coefficient option	Calculate
Restraint type	Rigid pipe
Poisson's ratio	0

Figure 71: Inputs for downcomer Pipe component in Flownex evaporator model

Pipes: Two-Phase Tank Connections

There are four sets of pipes connected to the *Two-Phase Tank*; feedwater inlet to the top of the shell from the exit of the economiser, steam extraction from the top of the shell to factory, downcomer pipes (x2) from the bottom of the shell to the bottom manifolds of the furnace waterwalls and the natural circulation inlet to the shell (half of total evaporator height) from the steam collector header.

The TORRIPAC has inlets to the evaporator at two heights; the first is at exactly half of the evaporator height (below the water level), and the second is higher up (above the water level). For simplification, only the lower height connection has been modelled. The second connection is apparently meant to only carry steam, but since the water level is inside the tubes, it is likely that it simply carries a mixture of steam and water. From a modelling perspective, the multiple inlet pipes have no effect.

The “Specify height” connection option was chosen, such that the branch heights could be inputted accurately to the actual connection as per the workshop drawings. Importantly, the steam extraction connection was set to 1.856m (i.e. top of evaporator) such that saturated steam (quality of 1) is extracted from the *Two-Phase Tank*. Similarly, the downcomer connection was set to 0m (i.e. bottom of evaporator) such that saturated liquid is extracted from the bottom of the drum to enter the natural circulation loop.

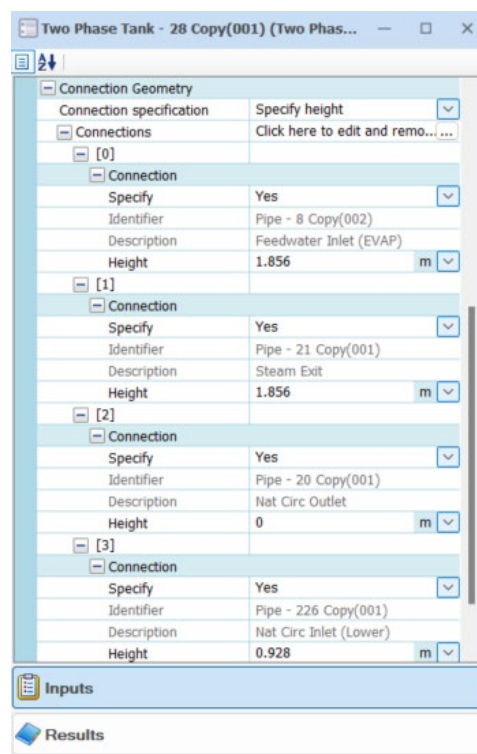


Figure 72: Two-Phase Tank branch height specification

For the waterwall *Pipe* component, the number of tubes in parallel is the sum of the front, rear, left, and right waterwall tubes. Note that for the stand-alone evaporator model, the lengths of the waterwalls were taken as the distance between the bottom header and the roof header. This is a simplification since the true geometry has the rear wall manipulated with a pronounced nose curving toward the front wall just above the grate to force the combustion air to flow over the incoming fuel. The front wall also curves to form the roof of the furnace to direct the flue gas into the duct connected to the firetubes of the evaporator.

More accurate waterwall lengths are included in the integrated model, but it should be noted that this is only done to model the thermal mass of the steel membrane waterwall tubes for dynamic investigations.

To get an accurate value for the natural circulation mass flowrate, the true geometries including all manifolds coupled with suitable k-forward factors and flow path geometries and elevations would need to be included. This is outside the scope of this research and no site data exists to validate the natural circulation mass flow.

The natural circulation rate does not have an impact on the heat transfer as long as the water does not reach a dry-out condition where all the liquid has evaporated into steam inside the water wall. This is because the water temperature will remain at the saturated condition, resulting in a uniform wall temperature on the inside of the furnace. As such, for the standalone evaporator model, the natural circulation mass flowrate was set equal to the steam extraction flowrate by setting a “fixed-mass flow” option on the downcomer *Pipe* component.

Note that by fixing the circulation rate, all pressure drop characteristics within the circulation loop are overwritten. I.e. the pressure drops in the circulation loop are “forced” to values that ensure the “fixed-mass flow” natural circulation mass flowrate.

Pipes: Firetubes

The *Pipe* component represents the firetubes in the drum in a parallel arrangement of 116 tubes of length equal to the inner length of the shell and tube inner diameter of 42.68mm.

Composite Heat Transfer & Natural Convection HT Coefficient QuickScript

The *CHT* component is used to model the heat transfer from the flue gas inside the firetubes to the tube walls and finally to the surrounding saturated liquid inside of the *Two-Phase Tank*. The convective heat transfer coefficient input properties can be seen in Figure 73.

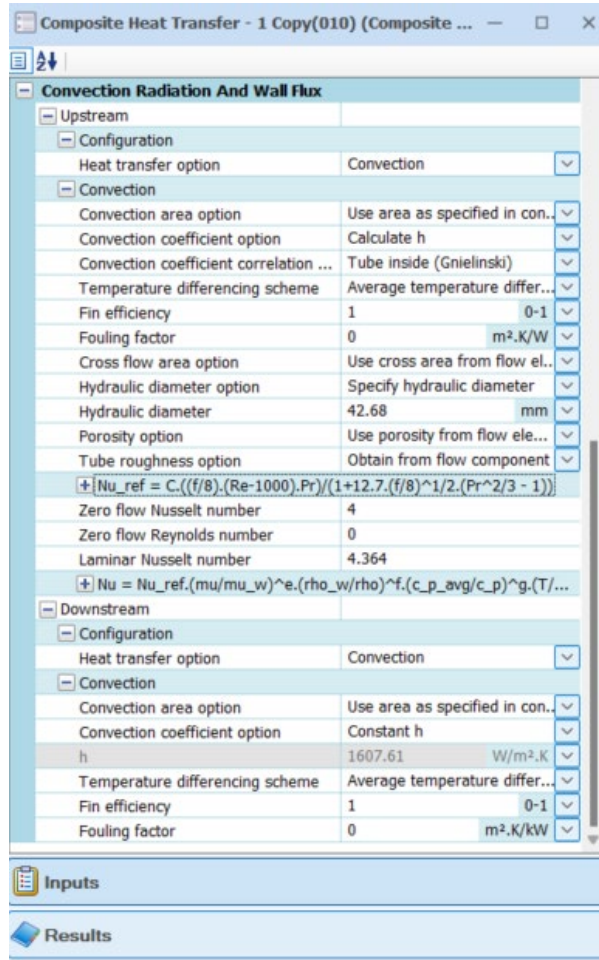


Figure 73: Composite Heat Transfer – convective property inputs

The upstream convective heat transfer (flue gas inside of fire-tubes) is specified using Flownex's built-in Gnielinski correlation, while the downstream natural convective heat transfer coefficient (to the surrounding water) is specified as constant, with its value being provided via a *Data-Transfer Link* from the *Quick Script* employing the Churchill and Chu correlation as per equation (3.22).

Boundary Conditions

Recall that the MCR data on the flue gas side only included the mass flowrate (6984 kg/h) and exit temperature (338°C) with no available data for the inlet temperature. As such the analytical model applied the exit temperature as a boundary value in the calculation of the inlet temperature which ended up being 946.089°C using air properties to approximate the flue gas properties. This again, implies that direct verification with MCR data is not possible. One can however compare the analytical and Flownex models in a partial verification.

The Flownex evaporator model requires that the temperature and pressure be specified for the flue gas at the inlet to the evaporator. The Flownex model verification against the analytical model was therefore accomplished by specifying 946.089°C as the inlet flue gas temperature to the fire tubes and resulted in an exit temperature of 338.343°C at the exit after a steady state solve. This

temperature agrees with the 338°C from the analytical model implementation. Other key boundary conditions are the steam mass flowrate and inlet feedwater temperature and pressure as obtained from the MCR data.

Recall from 3.3.8.3 that the steady state quality boundary condition imposed on the *Two-Phase Tank* will let Flownex calculate a non-zero energy source inside the tank if the energy balance is not satisfied by the existing incoming and exiting energies. For this reason, a fixed heat transfer was given to the *Pipe* component representing water-walls to account for the water-wall heat uptake which will eventually come from the radiation in the furnace model. This fixed heat transfer therefore acts like a boundary condition.

Figure 74 below shows the full stand-alone firetube evaporator Flownex model with the key inputs (tank connection heights, node elevations, waterwall heat input, temperature, pressure and mass source boundary conditions) as well as some verification results. Refer to Appendix C for the full detailed inputs for each component.

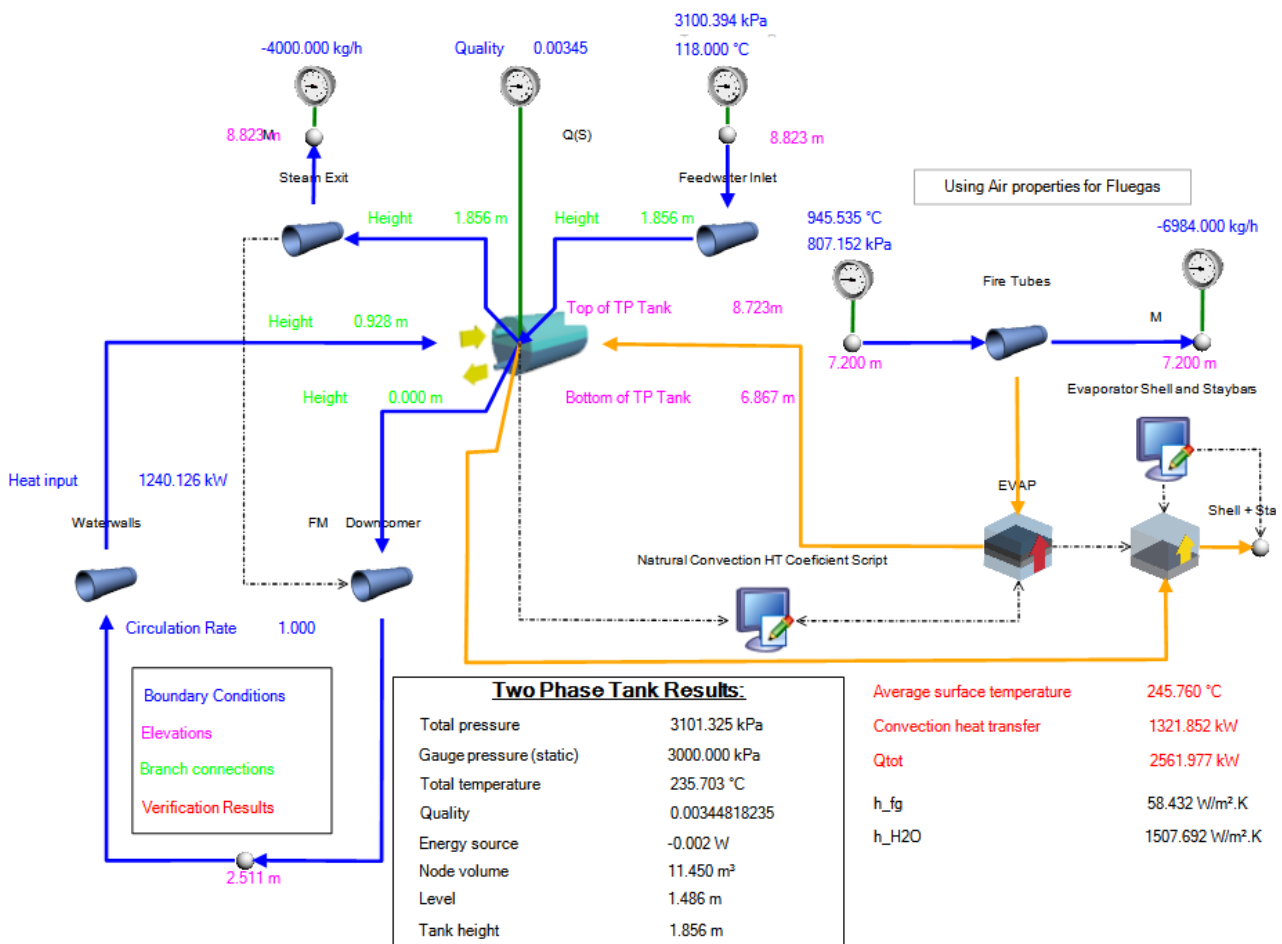


Figure 74: Complete fire-tube evaporator Flownex model showing key inputs & results

The full verification between the evaporator analytical and Flownex models can be found in Appendix C with a summary of the results given in Table 9.

Table 9: Evaporator verification results – flue gas temperature, tube wall temperature, and evaporator heat transfer

Flue gas						Water + wall + heat transfer rates					
	MCR		Analytical (air props)	Flownex 1 (air props)	Flownex 2 (flue gas props)		MCR		Analytical	Flownex 1	Flownex 2
Parameter	Unit	Value	Value	Value	Value	Parameter	Unit	Value	Value	Value	Value
Mass flow*	kg/s	1.94	1.94	1.94	1.94	Mass flow*	kg/s	1.11	1.11	1.11	1.11
Exit temperature**	K	611.15	611.15	611.49	622.27	Wall temeperature	K	/	518.81	518.79	520.53
Exit pressure*	kPa(a)	791.33	791.33	807.09	807.09	Evaporator heat transfer rate	kW	/	1327.06	1323.13	1636.34
Inlet temperature**	K	/	1219.24	1219.24	1219.24	Waterwall heat transfer rate	kW	/	1234.79	1238.85	925.64
Inlet pressure**(2% drop)	kPa(a)	/	807.15	807.15	807.15	Total heat transfer rate (waterwall+evap)	kW	2561.84	2561.84	2561.98	2561.98

Bearing in mind the previous discussions on a lack of MCR data for the inlet conditions it can be noted that there are discrepancies between flue gas exit temperatures between the various models. This stems from the discussions above regarding how each model was set-up. The exit temperature from the MCR data (338°C=611.15K) was used as a boundary condition to the analytical model. If it is assumed that the analytical model is valid, then the analytically computed inlet flue gas temperature (946.089°C=1219.24K) was needed to be applied as a boundary condition in the Flownex models. The Flownex models then solves for the exit temperature.

The Flownex 1 model calculates an exit flue gas temperature 611K which agrees with the analytical model. This again provides assurance that the Flownex 1 model is accurate and therefore the validity of the Flownex 2 model which incorporates real flue gas properties should follow suit. This is the best verification that can be done in the absence of an inlet flue gas condition to the evaporator. The 10K discrepancy between the exit flue gas temperature between the Flownex 1 and Flownex 2 model is attributed to the Flownex 2 model using real flue gas properties as was the case for the air heater and economiser verifications.

The percentage error pairwise comparisons of the flue gas exit temperature, fire-tube wall temperature and heat transfer rates as summarised in Table 10, Table 11, and Table 12 respectively.

These tables serve as a partial verification where good agreement between the results of the analytical and Flownex 1 model is observed. The heat transfer rates on the Flownex 2 model deviate significantly from the analytical and Flownex 1 model. Again, this is because we are not comparing “apples” with “apples”.

Table 10: Evaporator flue gas exit temperature [K]
%error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		-0.06%	-1.79%	/
Flownex 1	0.06%		-1.73%	/
Flownex 2	1.82%	1.76%		/
MCR	/	/	/	

Table 11: Evaporator fire-tube wall temperature [K]
%error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		0.01%	-0.33%	/
Flownex 1	-0.01%		-0.33%	/
Flownex 2	0.33%	0.34%		/
MCR	/	/	/	

Table 12: Evaporator heat transfer rate [kW] %error pairwise comparison

	Analytical	Flownex 1	Flownex 2	MCR
Analytical		0.30%	-18.90%	/
Flownex 1	-0.30%		-19.14%	/
Flownex 2	23.31%	23.67%		/
MCR	/	/	/	

Unfortunately, the Flownex 2 model cannot be verified against the MCR case due to lack of data but it can be inferred that it should agree with the MCR data since the only difference between the Flownex 1 and 2 models are the fluid property specification on the flue gas side. If this methodology is indeed valid, then it can further be inferred that real flue gas properties have an even larger effect at elevated temperatures as seen in the large discrepancy between the heat transfer rates of the Flownex 2 and Flownex 1 model of the evaporator.

In the absence of an explicit MCR verification, Table 13 below illustrates the progressive increase in errors on the heat transfer rate between the Flownex 1 (F1) and Flownex 2 (F2) models as the flue gas temperatures get higher from the economiser to the air heater and finally the evaporator.

Table 13: Heat transfer rate [kW] %errors between Flownex 2 and Flownex 1 models

	MCR flue gas exit temperature [K]	Heat transfer rate [kW] %error F2 vs F1
Economiser	427.15	7.13%
Air heater	503.15	9.97%
Evaporator	611.15	23.67%

Recall that although there are relatively high heat transfer rate errors between the F1 and F2 models of the economiser and air heater, the errors between the F2 and MCR for the economiser and air heater were small (3.81% and 0.70% respectively). Table 13 therefore does provide a level of confidence that the evaporator F2 model would be verifiable with MCR data if it existed.

4.2.4 Water-Cooled Furnace

The furnace exit temperatures were not included in the MCR data so the Furnace model cannot be directly verified against the MCR data. Instead, the furnace was calibrated using site data. The calibration was conducted on the whole boiler integrated model. This was necessary as the furnace energy balance is depended on the inlet air temperature, which in turn depends on the furnace exit temperature and heat uptake along the fire tube evaporator, economiser and air heater.

The water-cooled furnace Flownex model employed in the present work is an adaptation of a furnace model that was developed by ATProM. It is the most complex as it includes the combustion of the fuel and air streams, together with direct radiation to the waterwalls.

The Flownex model centres around a combustion excel workbook component (*Combust*) which performs all the combustion calculations based on the user defined fuel and air mass flowrates and compositions. The excel workbook performs the combustion calculations to output values such as the calculated calorific value, the combustion heat input, and the flue gas composition to name a few. These values are then given to the relevant flow *elements*, *Nodes* and *Boundary Conditions* via *Data-Transfer Links* as shown in Figure 76 on the landscape page following Figure 75.

All fluid streams, air, fuel, flue gas and water are modelled using *Flow Resistance* components. The *Fluid Radiation* component as introduced in section 3.3.8.7 employing the direct radiation model is used to model the radiative heat transfer to the water inside the waterwalls.

Excel Workbook: “Combust”

The excel workbook takes in 4 main user inputs: fuel mass flow, air mass flow, fuel Calorific value (CV), and a Boolean which allows the user to instruct the workbook to either make use of the user defined calorific value or to calculate a calorific value based on the fuel composition.

The workbook also has built-in values needed for the combustion analysis which include:

- Air moisture content ($\text{kg}_{\text{H}_2\text{O}}/\text{kg}_{\text{dry-air}}$)
- Molar masses of reactants (C, H, O, N, S)
- Molar masses of products (CO_2 , SO_2 , NO, H_2O , O_2 , N_2)
- Air composition (dry basis) & Air molar mass
- Fuel reactant formation & evaporation enthalpies.
- Stoichiometric coefficients

Lastly, the workbook requires the user to input the fuel composition on a wet basis (Ultimate fuel analysis), as well as to indicate the unburnt carbon as a percentage of the total carbon, the bottom ash as a percentage of the total ash, and an unaccounted loss factor. All these values were obtained from John Thompson. The bottom ash is specified as 80%, typical for solid grate fired biomass boilers. A sample combustion calculation following the theory in section 3.2.1 can be seen in Appendix D. The unaccounted loss is taken as 0.5% of the combustion heat and is a typical factor used in industry to account for radiation losses through the furnace. Similarly, a typical unburnt carbon fraction of 0.61% was used.

Figure 75 shows a screenshot of the combust excel workbook indicating a few links between the user inputs and the calculation outputs.

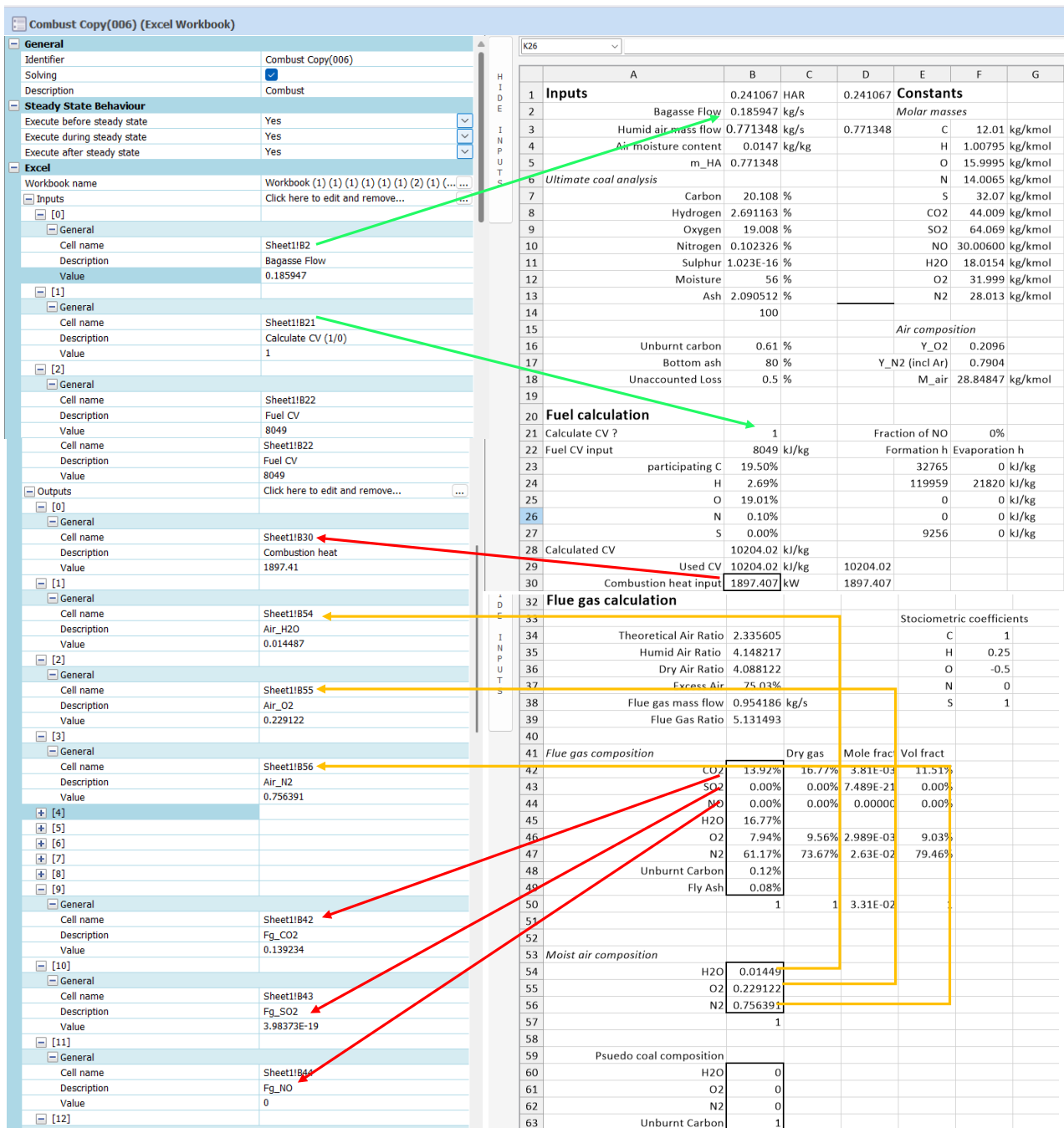


Figure 75: ATProM "Combust" excel workbook in Flownex showing some links between the input and output cells

The green arrows show cell references for the user inputs of fuel flow and CV Boolean, while the orange and red arrows illustrate the air and flue gas composition links from the cells to the output parameters of the workbook. Also shown is the combustion heat output cell link whose value is ultimately applied as a heat input to the flue gas flow resistance component.

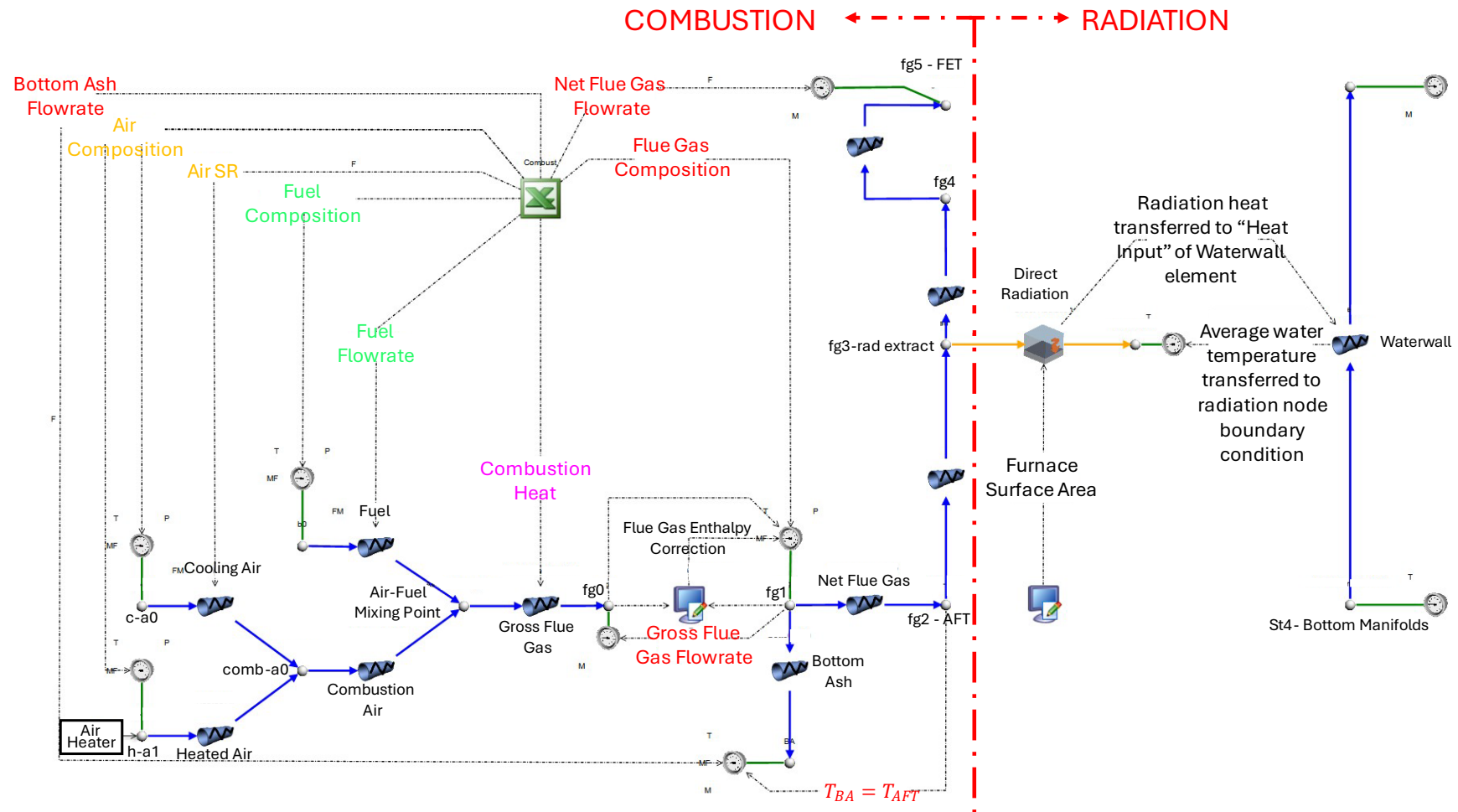


Figure 76: Water-cooled furnace Flownex model

Fluids & Compositions

Since the present work deals with a solid fuel which mixes with air to produce flue gas all in a common fluid network, Flownex requires that the same fluid be specified for each stream. Therefore, the ATProM library “Coal-fired flue gas” fluid was used which contains SO₂, Fly Ash, Unburnt Carbon, CO₂, N₂, NO, O₂, and H₂O as explained in section 3.3.2.

The fuel composition was initially taken as the ultimate analysis in *Table 14*, provided by John Thompson and then later renormalised using an updated 44% moisture content from a fuel analysis ctest conducted on the fuel at the Grenada site in 2021.

Constituent	Composition by mass [%]
Carbon	19.65
Hydrogen	2.630
Oxygen	18.58
Nitrogen	0.1
Sulphur	0.0
Moisture	57
Ash	2.043
Total	100
Calculated CV [MJ/kg]	10.204

Constituent	Composition by mass [%]
Carbon	25.60
Hydrogen	3.43
Oxygen	24.20
Nitrogen	0.13
Sulphur	0.0
Moisture	44
Ash	2.66
Total	100
Calculated CV [MJ/kg]	13.042

As expected, the lower fuel moisture gives rise to a higher carbon content and therefore a higher, calorific value of 13.04 MJ/kg in the 44% moisture case as opposed to 10.20 MJ/kg for the 56% moisture case. For the purposes of investigating dynamic operation, it was then important to include a feature in the Flownex combustion model which had the flexibility of changing the fuel moisture content such that the fuel composition could be automatically renormalised. This could emulate real life operation, where fuel quality can be highly variable (e.g. wet fuel).

The fuel, air, and flue gas compositions applied as boundary conditions to the relevant flow *Nodes* in the network are summarised in *Table 16* below for the MCR load case.

Species	Fuel Inlet Boundary Condition	Cooling Air Inlet Boundary Condition	Heated Air Inlet Boundary Condition	Gross Flue Gas Boundary Condition	Bottom Ash Boundary Condition
SO ₂	0	0	0	5.45E-19	0
Fly ash	0	0	0	0.001115662	1
UC	1	0	0	0.001278924	0
CO ₂	0	0	0	0.191928958	0
N ₂	0	0.756390688	0.756390688	0.60145465	0
NO	0	0	0	0	0
O ₂	0	0.229122271	0.229122271	0.036282348	0
H ₂ O	0	0.014487041	0.014487041	0.167939457	0

The sensible heat lost from the bottom ash is captured by assigning a 100% ash composition to the bottom ash outlet *Node* coupled with the bottom ash mass flowrate as calculated in the combustion excel workbook. Similarly humid air and flue gas compositions as calculated in the combustion excel workbook are given to the boundary conditions at the relevant nodes via *Data-Transfer Links*.

The fuel composition contains 100% carbon. This was done to mitigate Flownex errors regarding the sensible energy of the H₂O and other condensing gasses. Note that the 100% carbon composition was simply imposed for the purposes of capturing the sensible energy of the fuel entering the furnace (of which carbon is dominant), and that the actual fuel composition as per *Table 15* is already specified inside of the combustion excel workbook. This meant that the correct fuel, air, and consequent flue gas compositions are accurate inside of the excel workbook.

Iterative Script: Flue gas enthalpy correction

The change in gas composition for the different streams from air + fuel → flue gas is implemented by explicitly assigning the composition to each stream inlet node. To employ this, it is necessary to disconnect the flue gas flow from the air and fuel as seen in Figure 77. The combustion heat is added to the “Gross Flue Gas Flow” element, which is simply the mixture of fuel and air. This results in a fictitious flame temperature at “fg0” in Figure 77 since the gas composition is incorrect. The purpose of the script is therefore to adjust the flue gas node inlet temperature such that the enthalpy is the same as the air-fuel mixture, ultimately resulting in the accurate adiabatic flame temperature of the flue gas at node “fg1” in Figure 77.

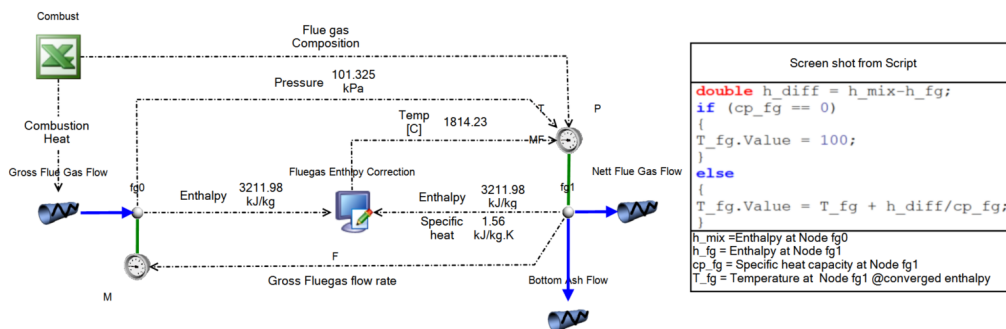


Figure 77: Flue gas enthalpy correction - Iterative Script

Flow Resistance components

Since this model is a simplified model with the aim of dynamic investigation, the air, fuel and flue gas streams are simply modelled using flow resistance elements using quadratic resistance behaviour with a specified flow admittance. The detailed ducting and complex flue gas flow path inside the furnace cavity are therefore not modelled explicitly. Figure 78 shows the inputs for the heated air stream *Flow Resistance* component. The inputs for the other *Flow Resistance* components are identical with the exception that a fixed mass flow is applied to the fuel stream and the cooling air stream.

To satisfy the mass balance with no redundancy, the net flue gas mass flowrate (i.e. excluding bottom ash) as calculated in the excel workbook is applied as a boundary condition at the exit of the furnace, with the mass flow fixed on the fuel *Flow Resistance* component, and the mass flow fixed on the cooling air *Flow Resistance* component. The heated air and combustion air *Flow Resistance* components (see pink box in Figure 79) do not have fixed mass flows specified, since Flownex will calculate this based on the mass balance.

The fuel flow rate and total combustion air flowrate are user defined inputs for the combustion excel workbook. The cooling air mass flow rate is taken as some percentage of the total user-specified air flow rate (approximately 10% for the MCR case as per John Thompson's MCR data). Flow elements containing the letters "FM" in Figure 79 indicate elements that have been specified a fixed mass flow.

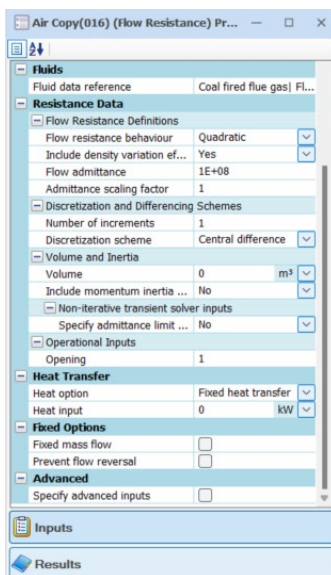


Figure 78: Flow Resistance inputs for heated air stream

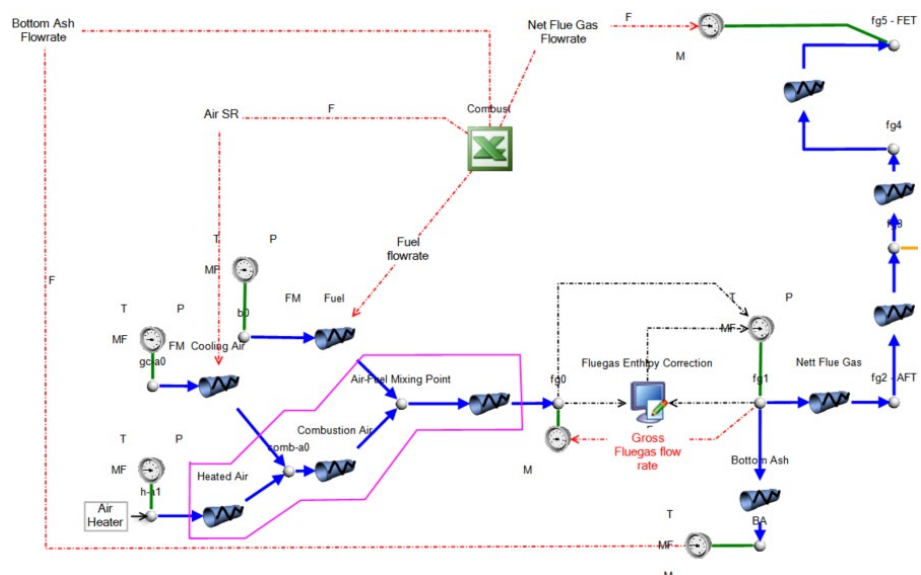


Figure 79: Furnace Flownex Model (combustion side) indicating mass source boundary conditions and fixed mass flow elements

Fluid Radiation Component

In the stand-alone furnace model, the fluid flow inside the waterwall has simply been modelled using a *Flow Resistance* component with the inlet node (i.e. bottom manifold) having a temperature and pressure boundary condition. The temperature is specified as the saturation temperature of the drum (235.7°C) as per the MCR data. The pressure (3150 kPa) is taken to be some value slightly higher than the drum pressure (3101.325 kPa) such that the fluid is sub-cooled with a quality less than 0. This is a valid approximation since the saturated liquid (quality 0) extracted from the drum would have an increase in pressure as it moves vertically down to the bottom manifolds of the waterwall. The mass source boundary condition imposed on the node at the top of the waterwall (i.e. top manifolds) is 4000kg/hr as per the MCR data.

Radiation heat transfer is employed by connecting the flue gas node, “fg3” to a solid node, “Rad Node” representing the steel material of the waterwalls using the *Fluid Radiation* component as illustrated in Figure 80.

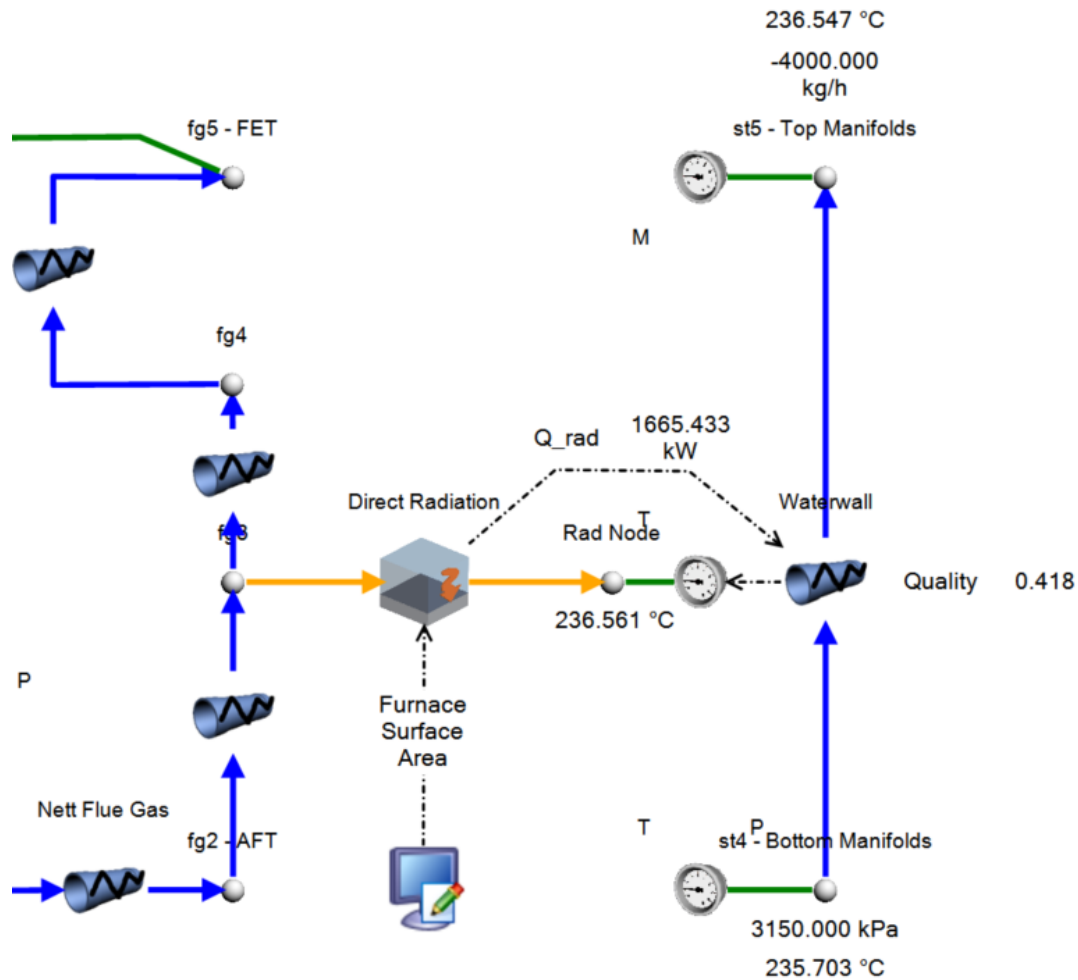


Figure 80: Furnace Flownex model (radiation side)

The “fg2” node is at the adiabatic flame temperature, and the “Rad Node” is set equal to the fluid temperature inside the waterwall by transferring the temperature from the “Waterwall” *Flow Resistance* to the temperature boundary condition imposed on “Rad Node”. With the radiation heat transfer surface area, emissivities and absorptivities specified, the heat transfer calculated inside the *Fluid Radiation* component is data transferred as heat input to the *Flow Resistance* component representing the waterwall.

The implementation here approximates the wall temperature to be equal to the average fluid temperature inside the waterwall. In practice the wall temperature will be slightly higher than the water inside. This detail will be included in the integrated model by replacing the *Flow Resistance* with *Pipe* components which reflect the true geometries of the membrane waterwall coupled with a *CHT* component with conduction layers specified to capture the thermal inertia of the steel material in the waterwalls.

Verification of the standalone Flownex furnace model can only be done for the combustion side by comparing the adiabatic flame temperature from an analytical calculation (see Appendix D) to the Flownex computed flame temperature. This will verify that the combustion calculations in the excel workbook coupled with custom flue gas fluid and flue gas enthalpy correction is consistent with the analytical calculation.

Verifying the furnace radiation contribution is not directly possible, since an energy balance on the whole boiler control volume (including air heater, economiser, evaporator and furnace) needs to be performed as explained in 3.2. Recall, in the partial verification of the fire-tube evaporator that an assumption was made on the furnace exit temperature (i.e. flue gas inlet temperature to the fire-tubes) based on the analytical model of the fire-tube evaporator. Consequently, the heat input into the waterwall was tuned to nullify the energy accumulation in the *Two-Phase Tank*. In theory the heat input to the waterwalls as specified in the evaporator stand-alone model should be equal to the radiation heat transfer inside the furnace. However, the mass flowrates used as boundary conditions for the fluid streams in the other heat exchanger Flownex models were taken directly from the MCR data and the flue gas compositions used there were based on an unverified fuel composition. The fuel, air, and consequently flue gas compositions which were used to develop the MCR design data is not known. Too many assumptions have been made in verifying the individual models which makes direct verification of the radiation in the furnace stand-alone model non-trivial especially since it is the central component.

In any case, once all models are integrated into a single network, a steady state calibration based on site data will be done. Section 4.4 deals with this overall model calibration with a large emphasis on calibrating the furnace radiation contribution.

4.3 Inclusion of Thermal Inertia

Substantial effort was made to include the thermal inertia of all the materials into the Flownex Model. Since this boiler is a unique coupling of fire-tube and water-tube designs, all thermal inertia should be captured to evaluate the transient performance of the system as it may be different to the performance of isolated fire-tube and water-tube boilers. The effects of the thermal inertia on transient response are a detail which steady state models lack.

4.3.1 Fluid Thermal Inertia

Flownex already accounts for the thermal inertia of the fluids themselves, but it was important to capture the volume of the water inside the system to observe the true fluid thermal inertia effects. The true water volume inside the evaporator shell has already been captured in the *Two-Phase*

Tank, but the water-volume inside the tubes of the water-cooled furnace (membrane walls) has not yet been captured accurately.

The furnace model was therefore updated to include accurate pipe lengths for the four water-walls with care taken to include the lengths of the pronounced bends in the rear wall and the roof. Figure 81 below shows the update from the initial stand-alone furnace model using a *Flow Resistance* to represent the waterwall to the current accurate pipe components employed in the integrated model to account for the physical water volume in the system.

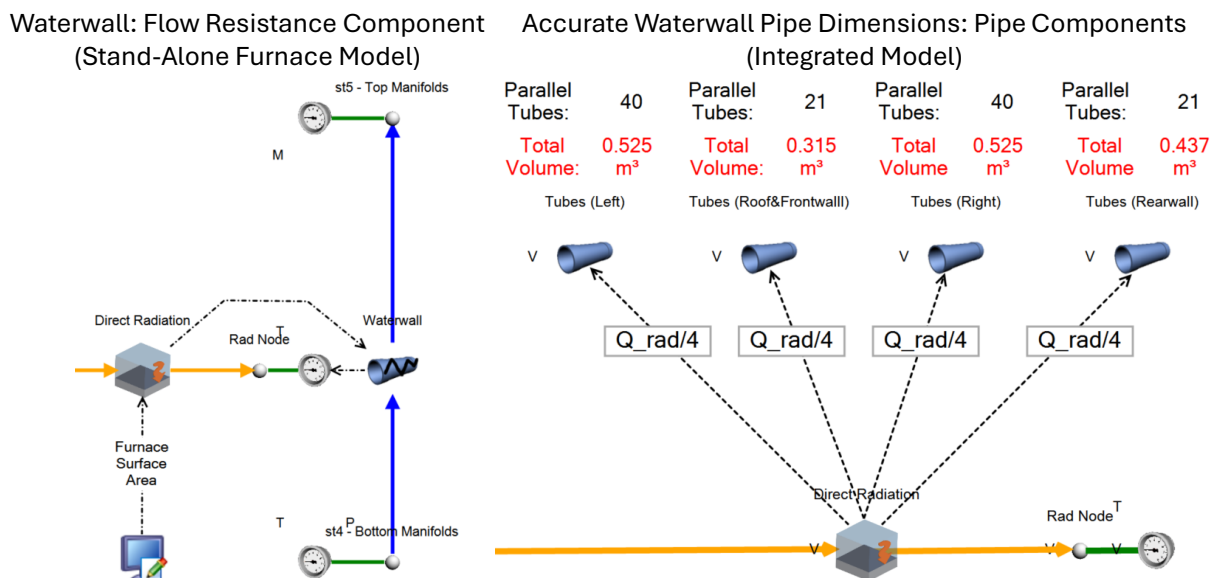


Figure 81: Model update from Flow Resistance in stand-alone furnace model (left) to detailed Pipe geometries in integrated model (right)

The heat input to the 4 waterwalls are now provided by *Data Transfer Links* from the *Fluid Radiation* component assuming an equal heat transfer split between each wall as illustrated in Figure 81. One could also have split the heat based on the relative area per wall, but that would still not account for the different view factors. As long as none of the wall experiences dry-out, the exact heat distribution is insignificant. A high enough circulation rate in the water wall tubes will ensure that this condition is met.

Although the flue gas flow passages are modelled with simple flow resistance components, its inertia is negligible in comparison to that of the water in the system as will be shown in Table 18.

4.3.2 Solid Steel Thermal Inertia

The thermal mass of the heat exchanger solid materials (i.e. steel tubes, fins, evaporator shell and solid steel stay bars) is significant and was included by carefully selecting suitable components which account for thermal inertia of solid materials via a combination of material specification, material

mass/volume, and wall thickness. The foregoing discussion briefly highlights how the steel thermal inertia was included.

The material specification for the various heat exchangers was selected as “Carbon Steel – AISI1010 | Metal (Solids)” from the Flownex charts and look-up tables. This material already has all the properties built-in, including a density of 7832 kg/m³, with its conductivity and specific heat capacity being governed by accurate polynomial scripts as discussed in section 3.3.2.

The thermal inertia is included for the economiser and air heater materials inside of the fin-side inputs to the *Finned-Tube Heat Exchanger* components. It is included for the tubes of the furnace and evaporator through coupling the pipe components with *CHT* components. Finally, the thermal mass for the evaporator shell and solid steel staybars are included in a solid *Node* with a volume specification encompassing the total volume of the shell material and the 21 solid steel staybars.

Finned-Tube Heat Exchanger



Carbon Steel - AISI 1010 | Metal (Solids) {Project}

Material Data	
Material option	Select from data reference
Mass	1282.55 kg
Wall thickness	0.00406 m
Material selection	Carbon Steel - AISI 1010 Metal (Solids) {Project}

Composite Heat Transfer (CHT)



Carbon Steel - AISI 1010 | Metal (Solids) {Project}

Conduction	
Use overall heat fraction	No
Conduction areas	Use upstream pipe areas
Layers	Click here to edit and remove...
Description	
Geometry	
Thickness in element d...	4.06 mm
Number of nodes	2
Material Data	
Material option	Select from data reference
Material major conduct...	Parallel
Material	Carbon Steel - AISI 1010 Metal (Solids) {Project}
Area discretization scheme	Standard (average areas)

Solid Node

Shell + Staybars



Carbon Steel - AISI 1010 | Metal (Solids) {Project}

Solid Mass	
Specify node solid volume	Yes
Solid data reference	Carbon Steel - AISI 1010 Metal (Solids) {Project}
User solid volume	1.04323 m ³

- Economiser (steel finned-tubes)
- Air heater (steel tubes)
- Water-cooled furnace (steel tubes)
- Firetube evaporator (steel tubes)
- Evaporator (steel shell)
- Evaporator (steel staybars)

Figure 82: Components used to capture solid material thermal inertia

The thermal inertia of the steel waterwall tubes was included through implementing four *CHT* components to transfer the heat from the radiation node to each of the four respective waterwalls. Each of the *CHT* components were specified with the pipe wall thickness (4.06 mm) and discretised with two nodes along the wall thickness to capture the conduction heat transfer from the outside of the pipe through the wall to the water on the inside of the pipes which will be important in the dynamic analysis. The upstream heat transfer (i.e. from the solid node) is specified as adiabatic, while the downstream heat transfer (i.e. to the water) is specified as convection using the Gnielinski correlation. Figure 83 below illustrates this update to the model.

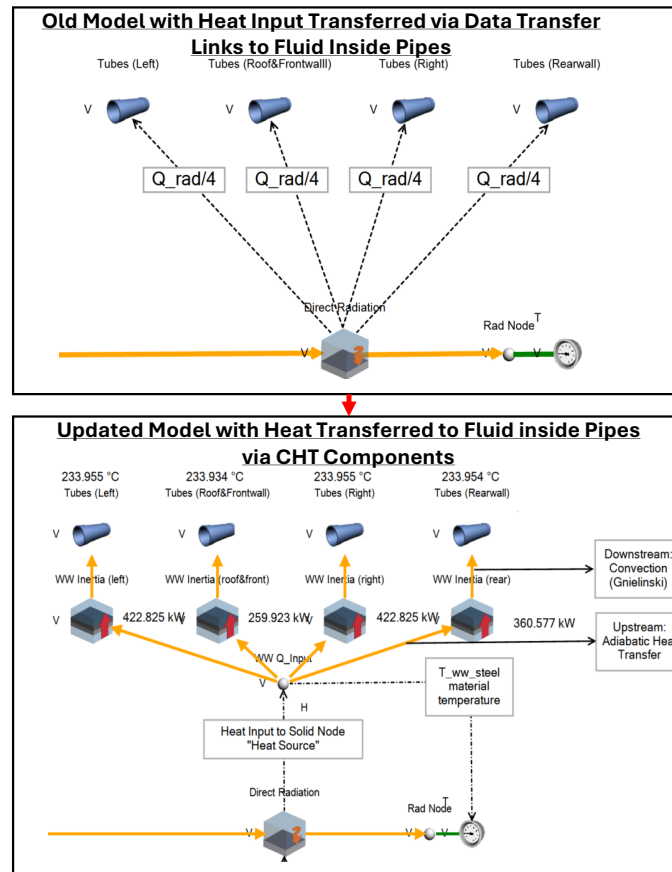


Figure 83: Furnace model update: waterwall steel thermal inertia

The total water mass in the system was just over 9 tonnes with the itemised breakdown shown in Table 18. The total air and flue gas mass included in the model only includes that captured by the finned-tube heat exchanger components. The furnace cavity and other gas passes were modelled with flow resistance components which do not capture the volume and mass of the fluid. This is however a non-issue since the water mass is orders of magnitude greater than the mass of the flue gas and air due to the gases having much smaller densities.

The total steel mass comes to just over 17 tonnes and is itemised in Table 17. These masses are in line with the nett calculated mass as per the workshop drawings of the individual heat exchangers. Note that the steel mass implemented in Flownex only includes the steel material involved in the heat transfer and therefore excludes heat exchanger casings and support structures.

Table 17: Steel inertia (mass)		
Component	Component capturing the inertia	Mass [kg]
Economiser finned-tubes	Finned-tube heat exchanger	1282.55
Air heater tubes	Finned-tube heat exchanger	1089.51
Evaporator (shell + staybars)	Solid node + Convection	8170.59
Evaporator firetubes	Pipes + CHT	2548.87
Waterwall tubes	Pipes + CHT	4443.09
Total Steel		17537.61

Table 18: Fluid inertia (mass)		
Fluid	Flow Component	Mass [kg]
Air	Finned-tube heat exchanger (AH): fin-side	1.08
Total air		1.08
Flue gas	Finned-tube heat exchanger (ECON): fin-side	0.95
Flue gas	Finned-tube heat exchanger (AH): tube-side	0.66
Flue gas	Pipes: Evaporator (firtubes)	0.41
Total flue gas		2.02
Subcooled water	Pipes: feedwater line	18.10
Subcooled water	Finned-tube heat exchanger (ECON): tube-side	87.24
Subcooled water	Pipes: downcomer + furnace manifolds (bottom)	453.45
Saturated water mixture	Pipes: furnace waterwalls + steam collector	512.87
Saturated water (liquid)	Two-Phase Tank: liquid phase	6750.51
Saturated water (vapour)	Two-Phase Tank: vapour phase	1234.02
Total water		9056.19

4.4 Steady State Model Calibration and Fine-tuning using Site Data

With all the stand-alone Flownex models that have been developed and verified as far as possible with analytical and available MCR data, they were integrated into a steady state integrated boiler model sequentially by connecting the relevant flow streams from one heat exchanger to the next.

Flow Resistance components with quadratic behaviour were used to connect the flue gas stream at the furnace exit to the evaporator inlet as well as to connect the flue gas stream at the exit of the evaporator to the air heater inlet. The TORRIPAC air heater and economiser are integrated into a single unit, so the outlet flue gas node from the air heater is simply linked to the inlet of the fin-side of the economiser *Finned-Tube Heat Exchanger* model. Lastly on the feedwater side, a *Pipe* component with length of 7m is linked from the feedwater pump exit node to the inlet node of the economiser and a 14m *Pipe* component is used to connect the feedwater exit from the economiser to the inlet of the *Two-Phase Tank*. The lengths of 7m and 14 m were approximated from the relative distances between the components as seen on the TORRIPAC site commissioning layout.

To provide a more elegant visual integration, a process flow diagram was created inside of Flownex by copying the “views” (see section 3.3.3) of the relevant flow nodes and elements onto a clean canvas and then using simple drawing annotations to show the links between various nodes and flow elements. The “view” nodes and elements still contain all the associated properties (inputs and results as described in section 3.3.3). The process flow diagram of the integrated model can be seen overleaf in Figure 84.

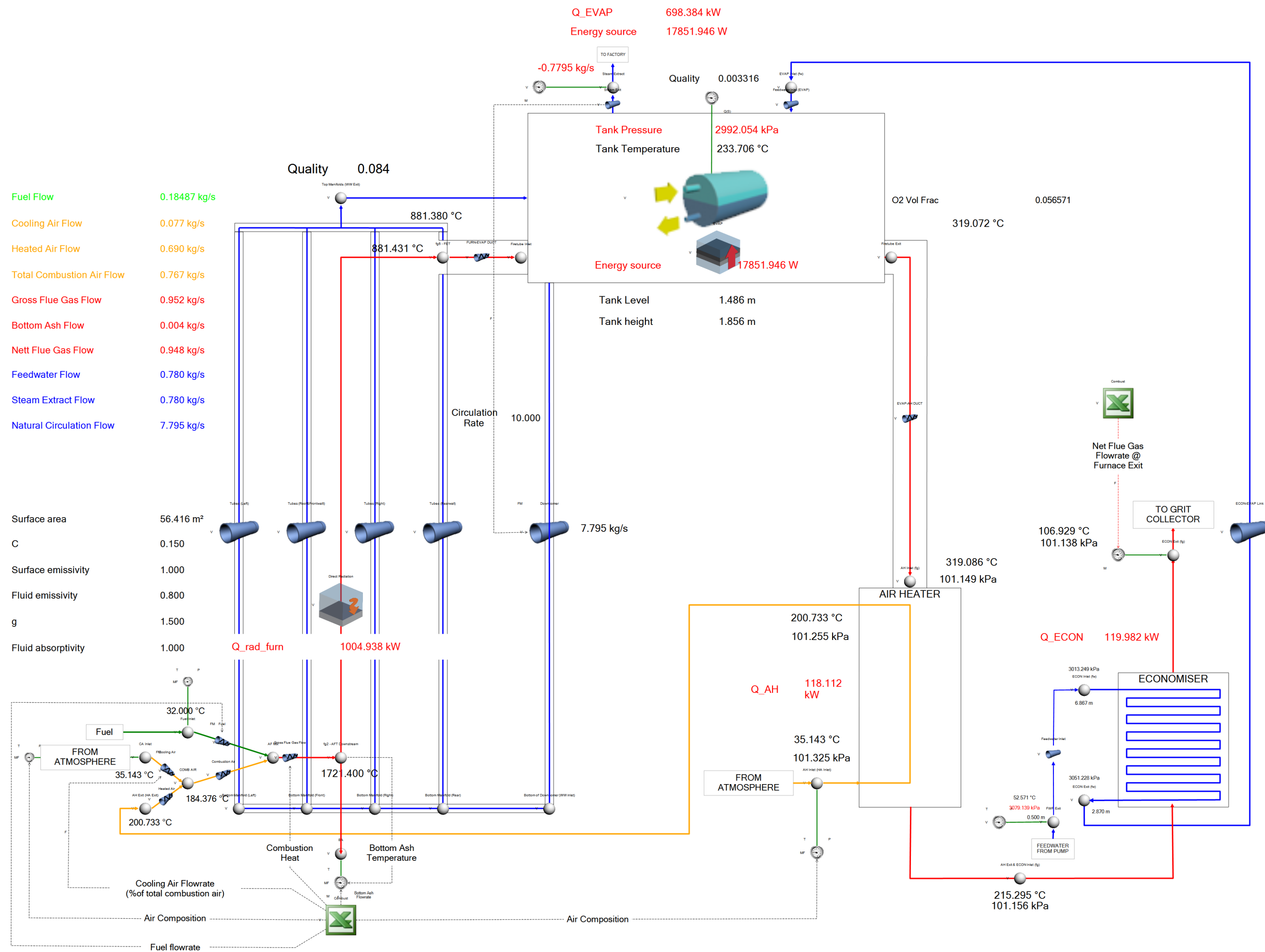


Figure 84: Integrated whole boiler model for TORRIPAC (uncalibrated)

Site test data was obtained from a steady load test conducted in 2021 at the site where the 4ton/hr TORRIPAC unit was commissioned. The test was conducted for a 2.8 tonne/hr load case (70% of MCR).

The raw data included ambient temperature and pressure as well as temperature and pressures of all fluid streams at key points in the boiler system. Measurements were logged every 10 minutes over a 1-hour testing period. A turbine flowmeter logged the feedwater volume at each 10-minute interval. Knowing the inlet feedwater temperature and pressure, it was possible to calculate the feedwater mass flowrate. Also included is the O₂ volume fraction on a dry-basis as measured by the flue gas analyser at the exit of the evaporator. No flowrates were measured other than the feedwater flow.

The raw data from the site test is shown in Table 19 below with the average values calculate over the hour test period in the final row.

Table 19: Raw Data from 2.8 tonne/hr Test Case on TORRIPAC

		Water Side				Air Side		Fluegas Side				Fuel Side?
Time [mins]	Time [s]	Volume [m³]	Main Steam Pressure [kPa-abs]	Economiser Inlet Temperature [°C]	Economiser Exit Temperature [°C]	Air Heater Inlet Temperature [°C]	Air Heater Exit Temperature [°C]	Furnace Exit Temperature [°C]	Evaporator Exit Temperature [°C]	Evaporator O2 Volume Fraction [%]	Economiser Exit Temperature [°C]	Dry-Bulb Temperature [°C]
0	0	175.310	3001.325	58	95	36	196	587	275	5.1	130	32
10	600	175.850	2991.325	56	94	35	194	548	273	6.7	130	32
20	1200	176.390	3001.325	54	92	34	210	537	272	4.8	132	32
30	1800	176.810	2981.325	54	89	36	195	524	270	5.9	136	32
40	2400	177.510	3011.325	52	88	36	198	580	274	5.9	132	32
50	3000	177.950	3011.325	48	88	35	198	578	276	4.9	133	32
60	3600	178.150	2951.325	46	82	34	198	548	271	6.3	133	32
Average			2992.754	52.571	89.714	35.143	198.429	557.429	273.000	5.657	132.286	32.000

The yellow highlighted values above are boundary conditions which will be imposed on the Flownex model and blue highlights are targeted conditions which should result from a successful calibration.

The pre-requisite settings before calibration includes fixing the circulation rate to be 10 times the steam extraction mass flow while also fixing the flow split between each of the four furnace walls to be equal, i.e. 25% of the natural circulation flowrate. This simply ensured that no dry-out occurs in any water wall.

The quality of the saturated mixture inside the *Two-Phase Tank* is also fixed such that the correct operating average water-level is achieved as explained in 4.2.3. Lastly the furnace radiation scaling factor (C) is set to a lower extreme of 0.15 since it will be tuned in the last step of the calibration.

The feedwater mass flow was determined using the average volume flow from the measurement data, inlet temperature and tank pressure to be 0.7759 kg/s. This was applied as the steam extraction boundary condition on the *Two-Phase Tank*. Only the tank pressure is known from the measurements, but the model requires an inlet pressure to the economiser (pump exit). This pressure is higher than the tank pressure because of pressure losses along the pipes. Initially a

pressure drop of 100 kPa between the pump and the inlet to the tank was assumed. This pressure becomes the parametric variable used to target the main steam pressure in calibration step 1.

Since the boiler is operating at approximately 70% load as per the MCR data, a crude assumption was made that the fuel and air mass flows could be scaled proportionally to initialise the calibration. Another notable assumption used for initialisation was to assume that the cooling air mass flow is 10% of the total combustion air as was the case for the MCR data. All relevant preliminary values and initial values for the calibration are summarised in Table 20.

Table 20: Global inputs and assumptions for whole boiler calibration initialisation

	Component	Parameter	Value	Unit	Rationale
Water Side	Feedpump outlet boundary condition	Temperature	52.571	°C	Average economiser inlet temperature as per site data
	Feedpump outlet boundary condition	Pressure	3092.754	kPa	Assumed 100 kPa pressure drop between feedpump outlet and evaporator saturation pressure
	Steam extraction boundary condition	Mass source	-0.77954	kg/s	Based on average volume flow as per site data at a density evaluated at feedpump outlet condition (see above)
	Pipe: Downcomer	Fixed mass flow	Yes		This fixes the downcomer mass flowrate as per the circulation rate assumption below
	Data Transfer: steam extract pipe to downcomer pipe	Multiplication factor for mass flowrate	10		Circulation rate assumed to be 10 based on similar boilers
	Pipe: Waterwalls (x3)	Fixed mass flow	Yes		This fixes the mass flowrate in 3 of the waterwalls to be equal as per assumption below that all 4 walls receive equal mass flowrate. The 4th wall is not fixed, it is solved by continuity in Flownex.
	Data Transfer : downcomer pipe to waterwall pipes (x3)	Multiplication factor for mass flowrate	0.25		Assumption of equal mass flowrate to each of the 4 walls (25%)
	Two Phase Tank boundary condition	Quality	0.003316		Determined using Solidworks for saturation pressure as per site data and average water level as per JT design (1.488m)
Fuel Side	Excel: "Bagasse Flow" input	Fuel flow	0.30467	kg/s	Scaled proportionally to steam flow using mcr vs site data (site data indicates approx 70% steam load case)
	Fuel inlet boundary condition	Temperature	32	°C	As per site data
	Fuel inlet boundary condition	Pressure	101.325	kPa	Ambient Site pressure
	Excel: Fuel Composition	Fuel composition			As per Table 20 based on 44% moisture
Air Side	Excel: "Air Flow" input	Air flow (total)	1.15499	kg/s	Scaled proportionally to steam flow using mcr vs site data (site data indicates approx 70% steam load case)
	Data Transfer: excel "Air Flow" to cooling air flow resistance	Multiplication factor for mass flowrate	0.1		Cooling air is 10% of total air as per MCR.
	Cooling air inlet + heated air inlet boundary condition	Temperature	35.143	°C	As per site data
	Cooling air inlet + heated air inlet boundary condition	Pressure	101.325	kPa	Ambient site pressure
Flue Gas Side	Fluid Radiation	Surface area	56.416	m ²	Calculated as projected area of all 4 walls and roof inside of furnace based on workshop drawings.
	Fluid Radiation	Fluid emissivity	0.6		Fluid emissivity of flue gas is usually between 0.6 and 0.8 for bagasse fired (biomass) boilers
	Fluid Radiation	g (fluid absorptivity to emissivity ratio)	1.5		Initially 1.5 as per example in Flownex manual. To be investigated parametrically
	Fluid Radiation	Surface emissivity	1		Set to upper extreme (assuming black body) such that radiation calibration is dominated by C factor
	Fluid Radiation	C	0.15		Initially chosen at a lower extreme to be tuned later. (should lie between 0 and 1)
	Economiser flue gas exit boundary condition	Mass source	-0.94782	kg/s	Value as calculated in excel workbook for nett flue gas flowrate (excluding bottom ash)
	Bottom ash Exit boundary condition	Mass source	-0.003935	kg/s	Value as calculated in excel workbook for bottom ash mass flow
	Bottom ash Exit boundary condition	Temperature	1721.4		Value as transferred from gross flue gas adiabatic flame temperature

The calibration is broken up into four main steps as detailed below.

Calibration Step 1: Feedwater mass flow and inlet pressure calibration through main steam pressure target

The calibration algorithm as illustrated on the right-hand side of Figure 85 entails guessing the feedwater inlet pressure and applying this guess along with the feedwater inlet temperature as per the site data to the *Boundary Condition* in the economiser model. The resulting feedwater density is evaluated at this temperature and pressure and then multiplied with the average volume flowrate from the turbine flow meter data to yield the feedwater mass flowrate. This mass flowrate is then applied to the steam extraction *Boundary Condition* in the evaporator model.

With the steam extraction and natural circulation mass flows now fixed, the model can be solved in steady state. The resulting pressure in the *Two-Phase Tank* is then checked against the expected main steam pressure as per the site data. If there is agreement with the main steam pressure, then the feedwater mass flowrate and economiser inlet pressure has been successfully calibrated. If not, the feedwater inlet pressure guess is updated iteratively until convergence.

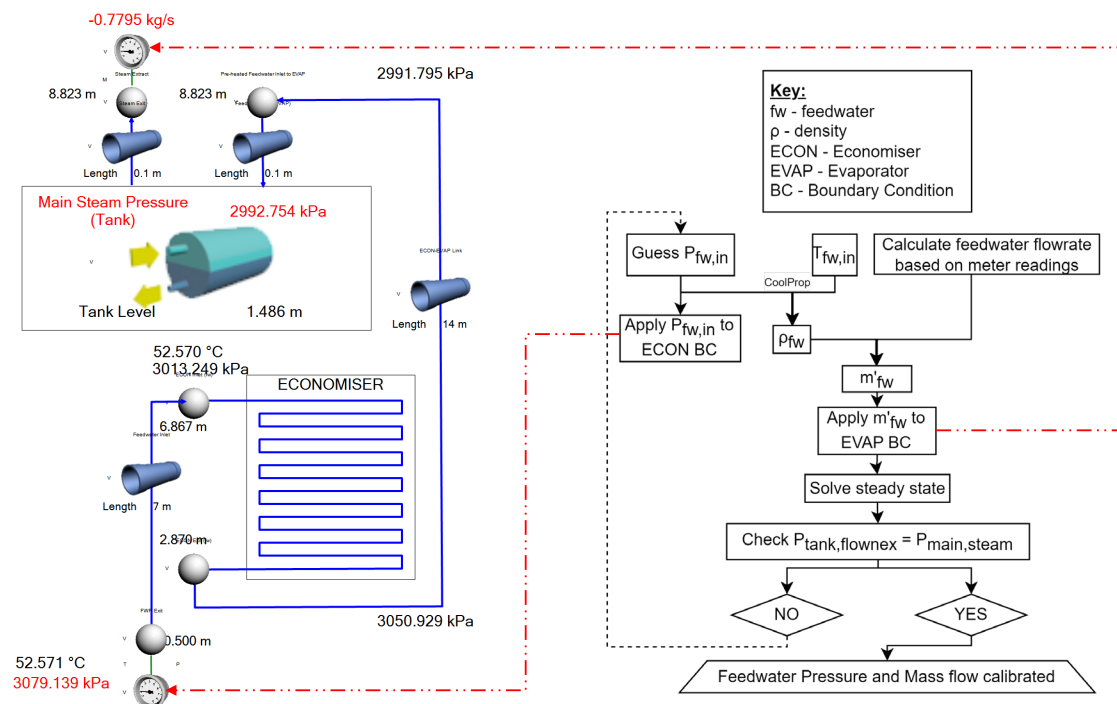


Figure 85: Calibration step 1 - results (left) and algorithm (right)

The left side of Figure 85 shows the calibrated feedwater pressure at 3079.139 kPa ensuring a *Two-Phase Tank* pressure of 2992.754 kPa (i.e. targeted main steam pressure from site data) with a corresponding mass flowrate of 0.7795 kg/s . Note the node elevations indicated in Figure 85 are accurate to the workshop drawings. The *Pipe* components linking the nodes are specified to use the Darcy Weisbach pressure characteristics. Including the elevation node details was done to more accurately capture elevation head pressure drops which seems to be consistent since in Figure 85 a

pressure rise over the economiser is observed due to a drop in elevation between the inlet and outlet nodes.

Calibration Step 2: Air-Fuel Ratio (AFR) calibration through oxygen volume fraction target

With the inlet feedwater pressure to the economiser now fixed after calibration step 1, the AFR can now be calibrated by fixing the fuel flow at 70% of MCR as per Table 20 and successively updating the “Air Flow” input to the combustion excel workbook. After solving steady state, the oxygen volume fraction computed by the excel workbook is compared to the expected value from the site data (5.657%). If the oxygen volume fraction is too high, then a smaller air mass flow is guessed for the same fixed fuel mass flow and vice versa. Once the oxygen volume fraction target is hit, the AFR has been calibrated. The process is illustrated Figure 86.

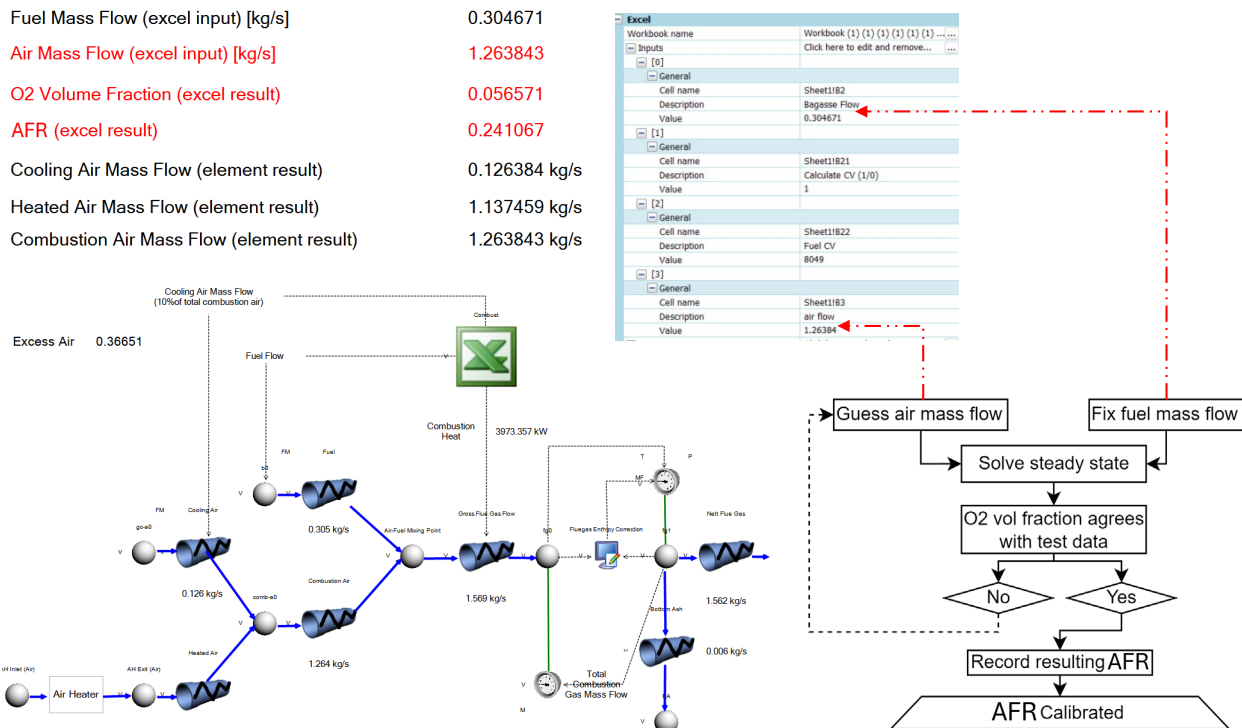


Figure 86: Calibration step 2 - AFR calibration targeting O₂ volume fraction

The actual calibrated fuel flowrate and air flowrate are not yet known, only the ratio between them is known. The fuel flow requirement will be calibrated in step 3.

Calibration Step 3: Fuel mass flow calibration through reduction of tank energy accumulation

Recall that a steady state quality boundary condition was imposed on the *Two-Phase Tank* to ensure that the level for the specific saturation condition (i.e. main steam pressure) agrees with the average water-level set-point of 1.488m referenced to the bottom of the tank. As discussed in section 3.3.8.3 for the uncalibrated model, this quality constraint gives rise to an energy source inside the tank. A

non-zero energy source for a tank with a specified quality, means that the overall energy balance of the tank's inlet and outlet streams and consequently the heat input to the tank is not satisfied.

The current process involves a reduction of the tank energy source by tuning the fuel flowrate. For a positive tank energy source, the boiler requires more heat input which in the integrated model is ultimately driven by an increase in fuel flowrate leading to higher combustion heat.

The iterative solution process is to guess a fuel mass flow with the AFR fixed at 0.241 as per calibration step 2 and then solve steady state to determine the tank energy source. To do this, the excel workbook was updated to include the fixed AFR in one of its cells. The combustion air mass flow is then calculated as the fuel flow divided by the AFR and therefore becomes an output of the excel workbook. Consequently, the data transfer link specifying the cooling air flow ratio is updated to point to the excel workbook output for the combustion air mass flow.

At this point the model still has a radiation scaling factor at a lower extreme of $C=0.15$. Increasing C increases the radiative heat transfer contribution, which increases the heat uptake in the waterwalls which has a knock-on effect to the tank energy source downstream. As such to calibrate the fuel flow rate, one does not require a complete nullification of the tank energy source, since fine tuning “ C ” in step 4 will do this. Instead, the aim with calibrating the fuel flowrate is to reduce the tank energy source to within 1% of the combustion heat input as illustrated in Figure 87 below.

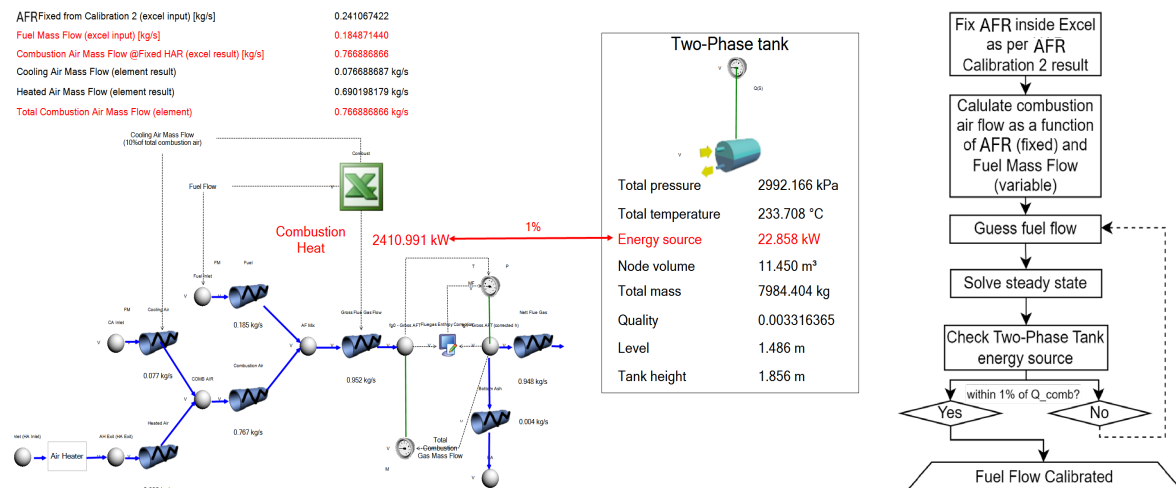


Figure 87: Calibration step 3 – Fuel flow calibration to reduce tank energy source to within 1% of combustion heat

The calibration up until the end of step 3 has been implemented based on certain assumptions due to lack of complete data sets for the site data. The main assumptions which the user can still control through the inputs to the model are the following:

1. Cooling air ratio (assumed to be 10% as per MCR)
2. Fluid emissivity (assumed to be 0.6 based on similar furnace models performed by John Thompson engineers)

3. Fluid emissivity vs. absorptivity ratio – g (assumed to be 1.5 as per Flownex documentation example)
4. Radiation view factor – C (set at 0.15, i.e. lower extreme)

Calibration Step 4: Furnace radiation calibration through C-factor parametric studies

At first it was assumed that points 1-3 above are valid inputs. This means that the C-factor can now be increased parametrically after each steady state solve while recording all the temperatures in the system. The recorded temperatures will be compared to the expected temperatures as per the site data (i.e. blue cells in Table 19). Ideally, there should exist a C-factor between 0 (no radiation) and 1 (maximum radiation) for which all the site data targets are reached. Recall the reasoning for the expectation of C being less than 1:

1. Confidence in fluid emissivity being 0.6
2. High confidence in furnace surface area calculation due to availability of detailed furnace layouts from John Thompson.
3. Fluid absorptivity investigated parametrically within its clearly defined limits as per (3.40) for the specified fluid emissivity of 0.6.
4. Over-estimation of the wall emissivity ($\epsilon_w=1$)

With the site measurement data that is available, points 1-4 above cover all bases in the argument for the expectation of a C-factor less than one.

The parametric study was set-up in Flownex by means of the “Sensitivity Analysis Setup”, using the “ C ” value from the *Fluid Radiation* component as the input variable as illustrated in Figure 88. The low value is set to 0.15 and the high value to 3, for a total of 58 runs (i.e. increments of 0.05 on the “ C ” scale).

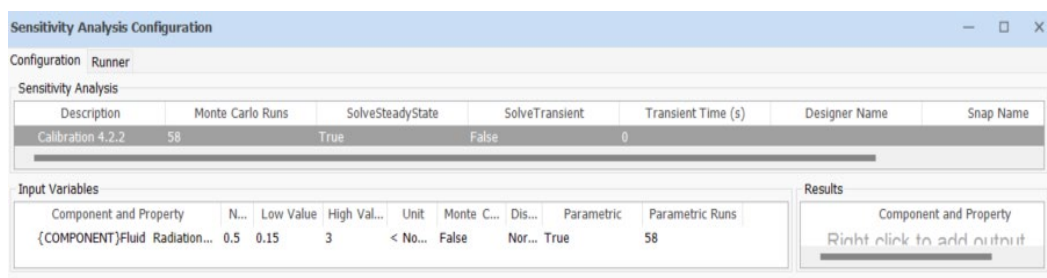


Figure 88: Sensitivity Analysis set-up for C-factor parametric study

An excel parameter table was set-up in Flownex such that all nodal temperatures as well as elemental heat transfer rates are recorded, thereby automating the data logging.

Figure 89 shows the results of the initial parametric study, where the solid horizontal lines are the target temperatures as per the site data, while the discrete dots represent the model temperatures for each C-value.

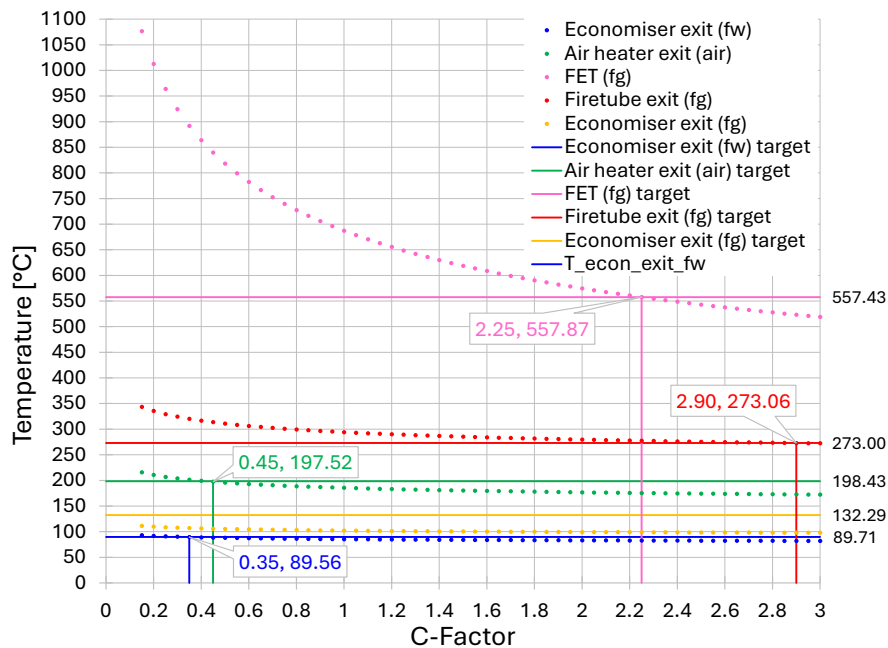


Figure 89: Furnace radiation calibration @10% cooling air and radiation with fluid emissivity =0.6 & absorptivity=1.0

As expected, the furnace exit temperature (FET) is the most sensitive to changes in the C-factor since it is the temperature immediately downstream of the radiation.

The second major observation is that the model underpredicts the flue gas temperature at the exit of the economiser for the full range of C-values. This is likely an inaccurate measurement since this temperature is not critical for boiler safety interlocks and control. The influence of the C-factor on the heated air and heated feedwater temperatures is much less pronounced, and the targets for the air and feedwater are reached at C-factors of 0.45 and 0.35 respectively.

The fact that the other flue gas temperature targets (furnace exit and evaporator exit) are only reached at very high C-values; 2.25 and 2.9 respectively could mean that there is in fact a significantly greater heat uptake in the waterwalls than is accounted for by radiation only. This implies that there is non-negligible convective heat transfer inside of the furnace which seems intuitive if one takes a closer look at the furnace geometry.

The rear wall of the furnace has a pronounced nose (see Figure 90) which protrudes over the grate to force the air over the incoming fuel. It seems likely that this pronounced nose, which is unique to this boiler will have localised convection due to the narrow region around the nose. Additionally, there would be convection effects as the flue gas exits the furnace at the top. The exit geometry is like a single row of a tube bank, i.e. the flue gas flows in-between gaps between successive tubes as it enters the duct connecting to the firetubes of the evaporator. Figure 90 illustrates the areas which would undergo forced convection.

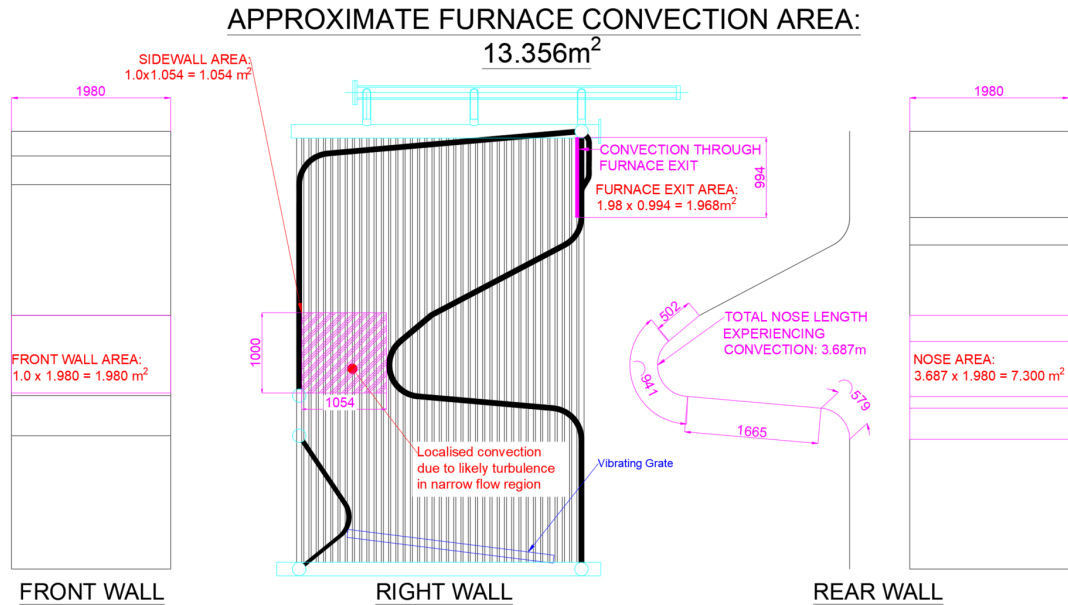


Figure 90: Effective furnace convection area approximation

The model was now updated to include this convection through 5 *CHT* components, each of which connects to the relevant waterwall with areas specified as per Figure 90. The fifth *CHT* component representing the furnace exit is also connected to the rear wall as illustrated in Figure 91 below.

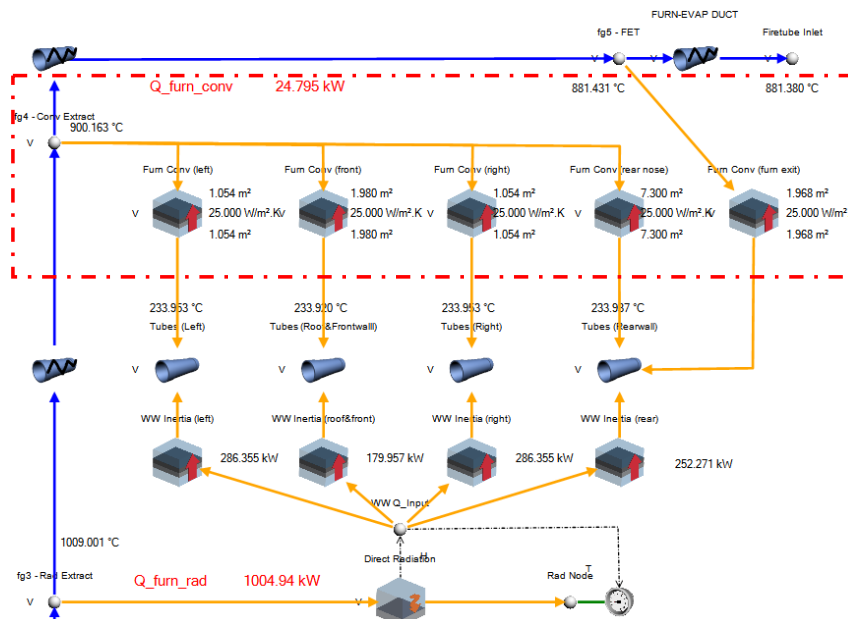


Figure 91: Furnace convection addition to Flownex model with CHT components

Since the focus of this research is on the dynamic analysis, a constant heat transfer coefficient was specified for the *CHT* components and parametric studies were run for values of 10, 15, 20 and 25 W/m²K which is indicative for forced convection over a flat plate. The results of all these parametric studies can be found in Appendix E.

The higher the convection heat transfer, the more heat is given off to the waterwalls and consequently the flue gas temperature targets will be reached at lower C-factors. $25\text{W/m}^2\text{K}$ is at the

upper limit of our assumption and is therefore used to drive down the C-factor value without enforcing unrealistic convection effects.

The inclusion of convection alone did shift the flue gas model temperatures to agree with the site targets at lower C-factors but there was still not a single C-factor for which all model temperatures agreed with all site measurement temperatures. The remaining variables that one has freedom to vary are the fluid absorptivity which can be varied with the “g” factor input for the *Fluid Radiation* component, as well as the cooling air split ratio which is unknown from the site data.

Recall from section 3.3.8.7 that the g-factor can be specified as any value, but Flownex caps the maximum calculated absorptivity at unity and the minimum absorptivity at a value equal to the emissivity. In this case the absorptivity can therefore be any value between 0.6 and 1. Setting the absorptivity to the lower extreme of 0.6 is accomplished by specifying a negligibly small “g” value of $1e-6$. Doing this further shifts the flue gas temperature targets to the left, i.e. closer to 1. The cooling air split ratio was also varied parametrically in search of the best calibration point turning out to be 25% cooling air.

The effect of these variable parameters of fluid absorptivity, furnace convection and cooling air ratio can be observed through the progression from Figure 92 to Figure 95. Figure 92 shows the C-factor parametric results for the scenario of pure furnace radiation at 10% grate cooling air, fluid emissivity of 0.6 and fluid absorptivity of 1. Figure 93 shows the results for an absorptivity of 0.6, Figure 94 includes the furnace convection at a convective heat transfer coefficient of $25 \text{ W/m}^2\text{K}$, and Figure 95 shows parametric results at a cooling air split ratio of 25%. An exhaustive set of graphs for the intermediate parametric studies for varying split ratio and convective heat transfer coefficients can be found in Appendix E.

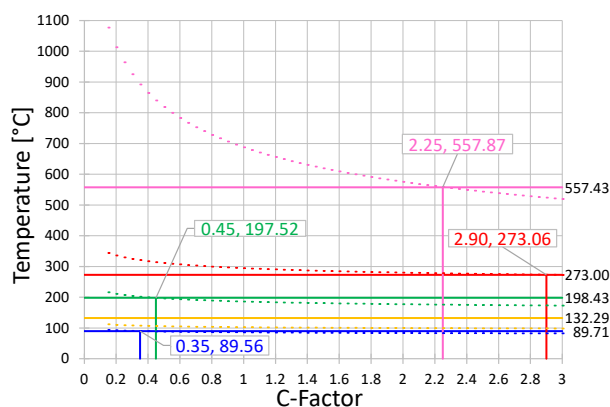


Figure 92: Furnace radiation calibration @10% cooling air and radiation with fluid emissivity =0.6 & absorptivity=1.0

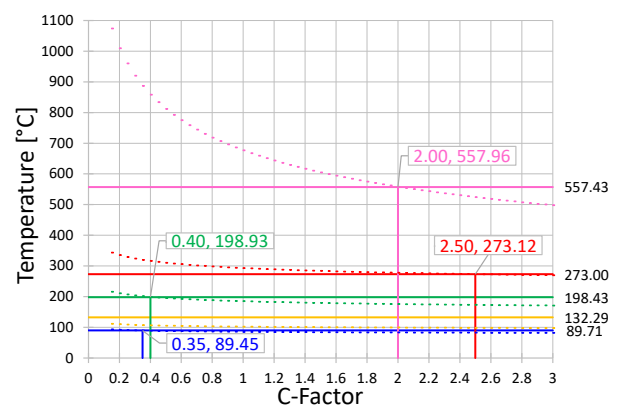


Figure 93: Furnace radiation calibration @10% Cooling air and radiation with fluid emissivity =0.6 & absorptivity=0.6

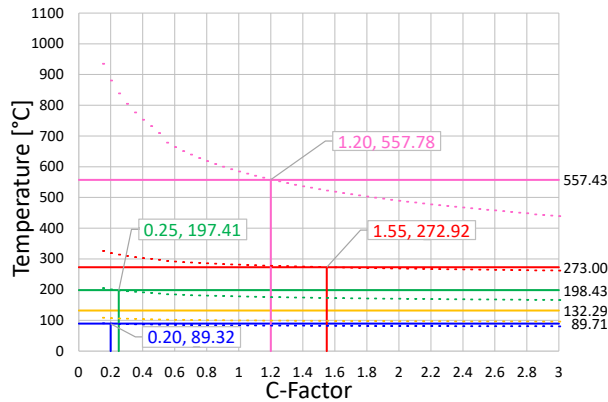


Figure 94: Furnace radiation calibration @10% cooling air and radiation with fluid emissivity =0.6 & absorptivity=0.6 Plus convection @ $h=25 \text{ W/m}^2\text{K}$

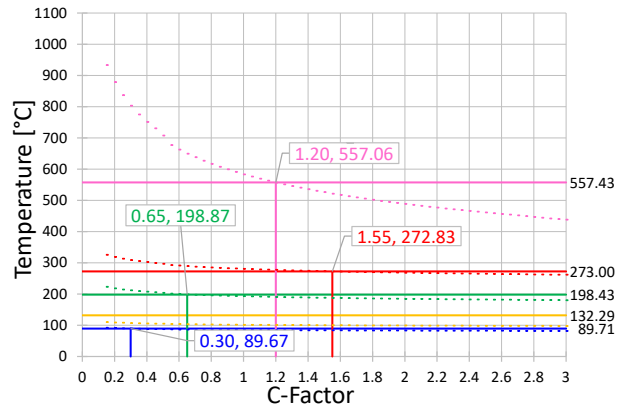


Figure 95: Furnace radiation calibration @25% cooling air and radiation with fluid emissivity =0.6 & absorptivity=0.6 Plus convection @ $h=25 \text{ W/m}^2\text{K}$

With the absorptivity at the lower extreme of 0.6, the flue gas temperatures at the furnace exit (pink dots) and evaporator exit (red dots) reach their targets at significantly lower C-factors of 2.0 and 2.5 respectively as seen Figure 93 compared to 2.25 and 2.9 for the case of unity absorptivity in Figure 92.

The results of Figure 94 are also for an absorptivity of 0.6 but this time with the inclusion of furnace convection effects which shows an even more dramatic shift in the flue gas temperatures at the furnace exit and evaporator exit, now reaching their targets at 1.20 and 1.55.

Note that the reduced absorptivity of 0.6 and the inclusion of furnace convection has also shifted the feedwater and heated air temperatures further to the left, i.e. diverging away from the C-factor at which the flue gas temperatures reach their targets. This can be seen in Figure 94 where the heated feedwater and air reach their site data targets at C-factors of 0.2 and 0.25 respectively. This is combatted by adjusting the cooling air split ratio which shifts the C-factor at which the air and water temperatures reach their targets closer to the C-factor at which the flue gas temperatures reach theirs. With the cooling air split ratio at 25% one can see the heated air temperature being reached at a C-factor of 0.65 as per Figure 95.

Measurement uncertainty

Recall that if all inputs are accurate then we expect all temperature targets to be met at the same C-factor. Since this is not the case, it is necessary to reconcile all the information by considering the uncertainty in the site data measurements.

The first thing to note is that all the calibration graphs presented thus far indicate the targets being met at a single data point from the model which is closest to the target temperature. This does not consider an allowable temperature band for calibration. The recorded temperatures from the site data could have up to a +5K error depending on the method of measurement and the apparatus used.

Secondly, the data logged from site already shows some fluctuations at each 10-minute interval, with a 60K difference between the minimum and maximum logged FET measurements being the most notable. The target temperatures as well as the boundary conditions imposed on the model were taken as the average of all data points over the 1-hour period as per Table 19. It is therefore necessary to conduct the calibration with the standard deviations of the recorded data in mind.

A more informative illustration of the model results referenced to the expected site data targets is illustrated in Figure 96 below which shows the absolute %error based on the Kelvin scale temperatures between the Flownex model and site measurements as a function of the C-Factor.

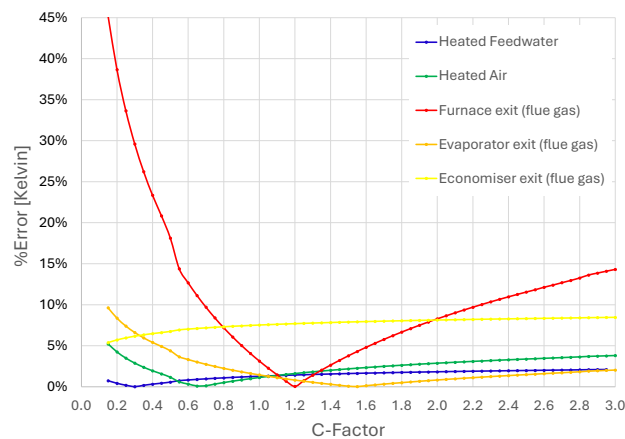


Figure 96: Absolute %error vs. C-factor for 25% cooling air, furnace radiation (emissivity=absorptivity=0.6) & furnace convection @ $h=25 \text{ W/m}^2\text{K}$

Finally, the data can be reconciled by providing an informative weighting to each site data target by considering the expected uncertainty in each of the five measurements using insights from John Thompson engineers. A pairwise comparison was conducted, comparing each measurement with one another to formulate a qualitative rank for the confidence expected in each of the five measurements.

Table 21: Pairwise comparison for confidence rankings of site measurements

	Heated feedwater	Heated air	FET	Flue gas (evaporator exit)	Flue gas (economiser exit)	Row total (rank)	Weighting [%]
Heated feedwater	-	0.60	0.90	0.50	0.95	2.95	29.50%
Heated air	0.40	-	0.85	0.40	0.90	2.55	25.50%
FET	0.10	0.15	-	0.10	0.60	0.95	9.50%
Flue gas (evaporator exit)	0.50	0.60	0.9	-	0.95	2.95	29.50%
Flue gas (economiser exit)	0.05	0.10	0.4	0.05	-	0.60	6.00%

The economiser flue gas exit temperature is not a critical parameter and therefore scores a low weighting. The FET measurement has a 60K difference between its minimum and maximum recorded temperature. The FET is measured with a suction pyrometer inserted into the duct

connecting the furnace exit to the fire-tube flue gas inlet. The suction pyrometer measures temperature only at a single point. There is complex flow phenomena at the furnace exit as the flow undergoes turbulence in that region. Best practice would be to record temperatures at multiple locations of the furnace exit plane such that a mass flow/area weighted temperature could be calculated. The FET therefore also scores a low weighting of 9.5%.

The flue gas temperature at the exit of the evaporator on the other hand scored a 29.5% weighting as it is an important parameter which is utilised in the TORRIPAC control system for safety interlocks. The heated air and feedwater temperatures also get a higher rating since they are temperatures that are easily measured with thermocouples in the ducts where the flow profile is not too erratic.

The weightings in the final column of Table 21 are applied to the absolute error for each data point of the relevant temperature for each value of the C-factor. All data can now be normalised to being a function of a single value (i.e. C-factor) by adding up the five weighted errors at each C-value. The results are visually represented in Figure 97, where the best calibration point is taken as the C-value with the lowest weighted error, i.e. $C=1.2$ at a weighted error of 1.537%.

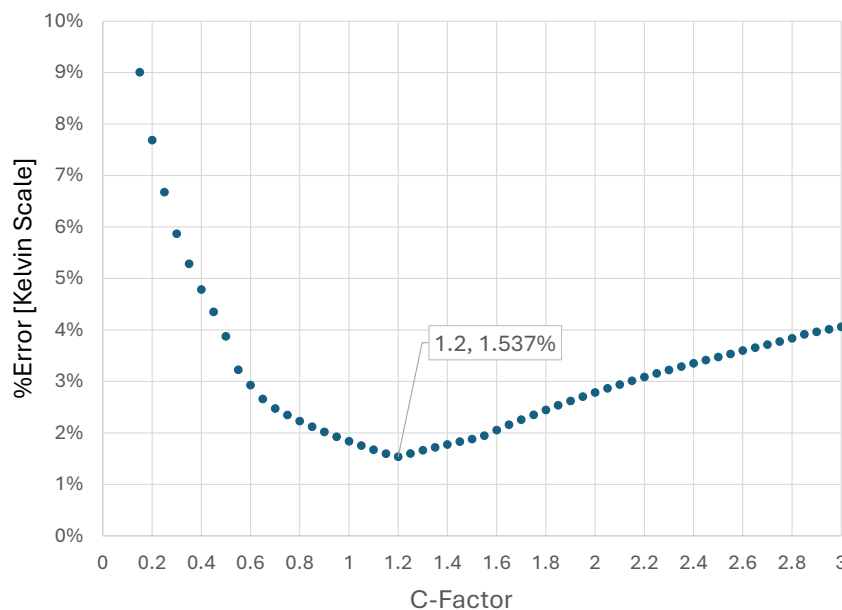


Figure 97: Final calibration results – normalised weighted error vs. C-factor

A C-factor of 1.2 is still higher than expected considering that the model already over-estimates the wall emissivity to be unity and crudely assumed the minimum possible fluid absorptivity being equal to the emissivity at a value of 0.6. However, this is the best calibration point one can get to without having to tune other parameters for which there is little confidence in tuning due to lack of more detailed information from site. Results of a high fidelity CFD simulation of the furnace heat transfer could also better characterise the complex heat transfer phenomena occurring inside the furnace. A CFD model of the TORRIPAC is being performed in a separate study by John Thompson, the results of which are still pending.

Table 22 below summarises the final calibrated results at the optimum C-factor of 1.2 which will be used as the steady state point for initialisation of the dynamic simulations. Since this factor is essentially describing the radiation view factor, it is assumed to remain constant for all other loads. There is no further information that could be used to make the factor a function of load as well.

Table 22: Site measurements vs. model for 70% (2.8 tonne/hr) load case

Parameter	Units	Site Measurement Data			Model
		Mean	Min	Max	
Water side					
ECON inlet temperature	K	325.72	319.15	331.15	325.72
ECON exit temperature	K	362.86	355.15	368.15	357.86
Flue gas side					
Furnace exit temperature (FET)	K	830.58	797.15	860.15	831.44
EVAP exit temperature	K	546.15	543.15	549.15	550.84
ECON exit temperature	K	405.44	403.15	409.15	374.49
Air side					
AH inlet temperature	K	308.29	307.15	309.15	308.29
AH exit temperature	K	471.58	467.15	483.15	464.08

Figure 98 shows a visual representation of where the model temperatures fall within the standard deviation of the measured site data through a box and whisker plot. Given the limited information, it is believed that the model does quite well in predicting the boiler performance.

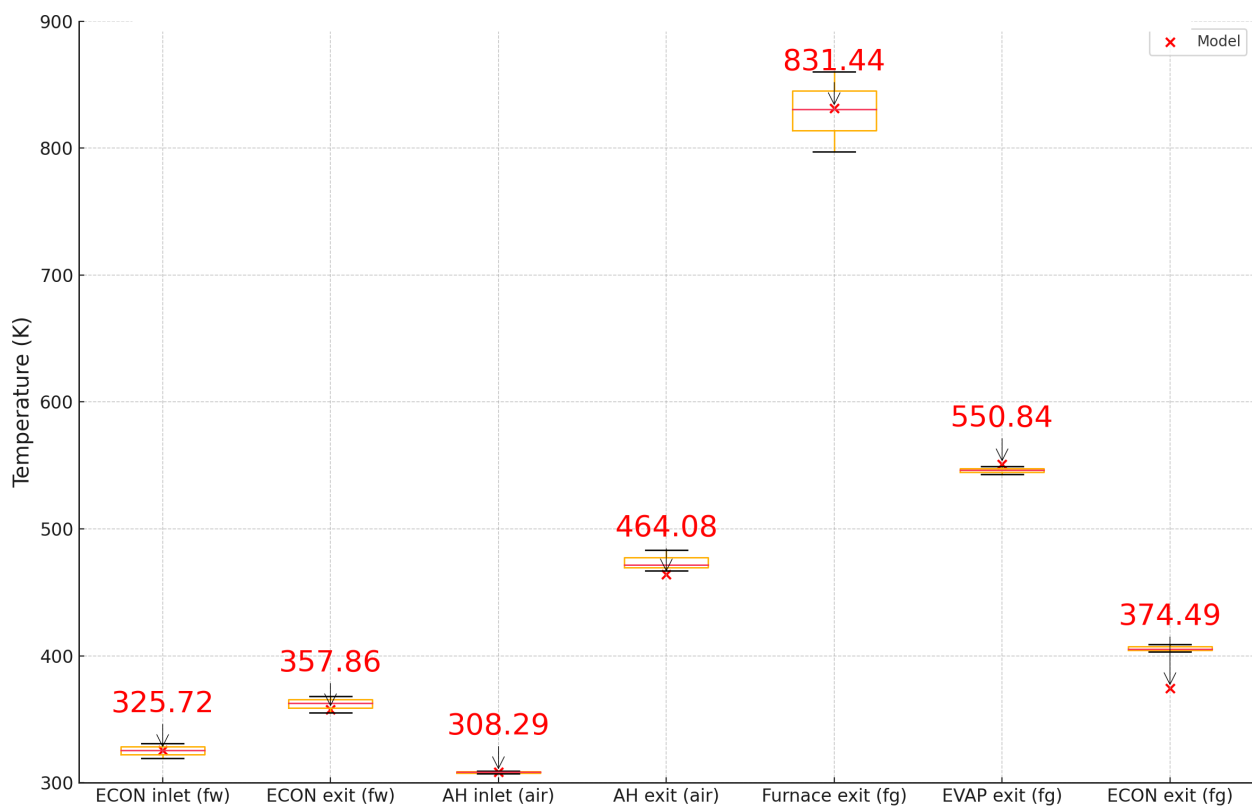


Figure 98: Box & Whisker Plot illustrating where model temperatures fall within the standard deviation of the site measurements taken over the 1-hour testing period

4.5 Controller Development

4.5.1 John Thompson TORRIPAC Control Philosophy

The TORRIPAC has a complex control system with numerous sensors, transmitters, controllers, switches as well as audible alarms and safety interlocks. Most of the control system is automated and connected to a human-machine interface (HMI) where an operator can monitor the boiler operation. Certain controls can also be switched into manual mode, allowing the operator to make necessary adjustments such as changing controller set-points such as fuel trim, oxygen trim, and water-level. The main control loops involved in the TORRIPAC control philosophy are briefly summarised in this section.

Vibrating grate and box ram integration:

The vibrating grate is controlled via variable speed drives (VSD) and has two main modes of operation: propel and settle. The grate receives its control signal from the pressure transmitter fitted to the evaporator as explained in John Thompsons internal control philosophy documentation [19] for the TORRIPAC. For a high steam demand, the settling time will decrease while the propel time will increase accordingly. Figure 99 illustrates the link between the box ram timing and the vibrating grate timing.

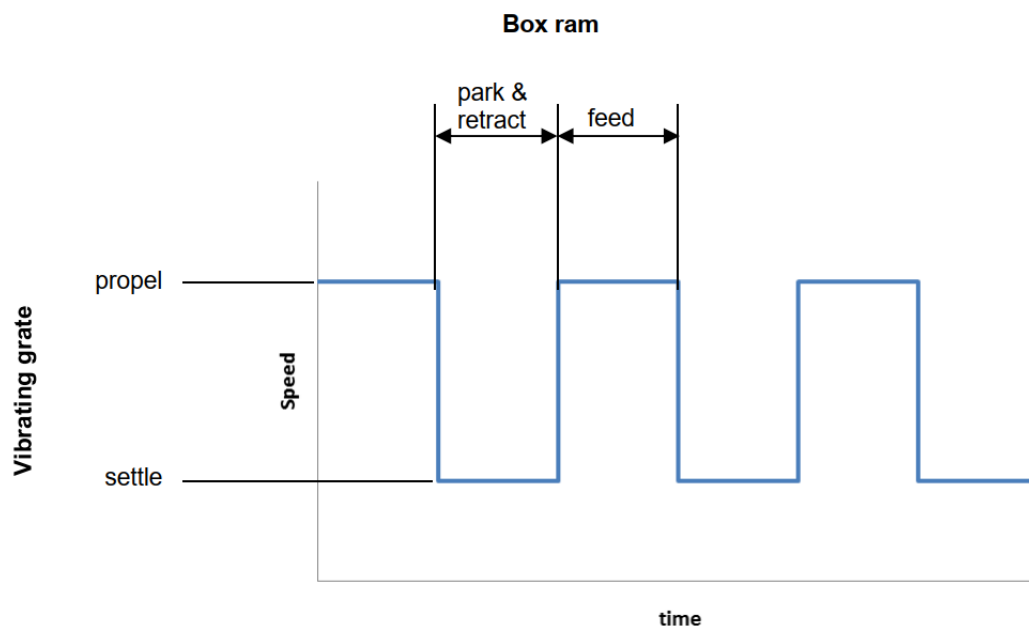


Figure 99: Vibrating grate and box ram operation, [19]

The box ram is in the park & retract mode while the vibrating grate is in settle mode. The box ram switches to feed mode as the vibrating grate propels the fuel. This ensures an evenly distributed fuel bed to aid uniform and complete combustion such that localised high fuel spots on the grate are minimised.

Boiler Steam Pressure control:

This loop contains a proportional-integral (PI) controller whose output varies the vibrating grate cycle and propel times which in turn varies the combustion air and fuel feed controllers which link to VSDs on the forced draft (FD) fans and the fuel conveyers respectively [19]. This controller essentially senses the change in energy (steam) extraction and responds by adjusting the fuel flow.

Fuel Feed Control:

The fuel feed system consists of a fuel hopper, hydraulically actuated walking floor, conveyors (horizontal and inclined), and a box ram to inject the solid fuel into the furnace. The walking floor and conveyors are all on VSDs whose speeds are a function of the hopper level [19].

Combustion Air Control:

The cooling air fans are linked to the vibrating grate propel time with an inverse relationship such that if the propel time is at a minimum, the cooling air fans will be at a maximum controlled via VSD ensuring constant cooling. The forced draft fans are also adjusted via VSD, and the speed is controlled to maintain a fixed wind box pressure downstream of the air heater [19]. This controller has the effect of maintaining the air-fuel ratio to the boiler, but it is supplemented with other controllers.

Oxygen Control:

This loop involves a controller and oxygen transmitter. The oxygen trim is PI controlled with an allowable set-point range of 2% to 6% [19]. The oxygen controller output signals necessary adjustments to the secondary air dampers.

Furnace Pressure Control:

The induced draft (ID) fan with PI control via VSD controls the furnace pressure with a set-point band between -50Pa and -100Pa [19]. The ID fan control should be set-up in tandem with the forced draft (FD) fan to ensure that the furnace is never pressurised.

Water Level Control:

A capacitance level transmitter fitted to the gauge-glass standpipe of the evaporator signals the feedwater pumps to adjust accordingly if a change in level is detected. The feedwater pumps are on VSDs and the control logic is single element PI [19].

4.5.2 Simplified Flownex Control Loops

A detailed control system is not in the project scope. Instead, the intention of this section is to demonstrate how the present model could be used for dynamic simulations. This is shown through the implementation of two simplified control loops as detailed below:

1. Boiler steam pressure control with feedback to the fuel flowrate.
2. Water-level control with feedback to the feedwater mass flowrate.

Figure 100 shows a comparison between the real control system and the simplified Flownex implementation.

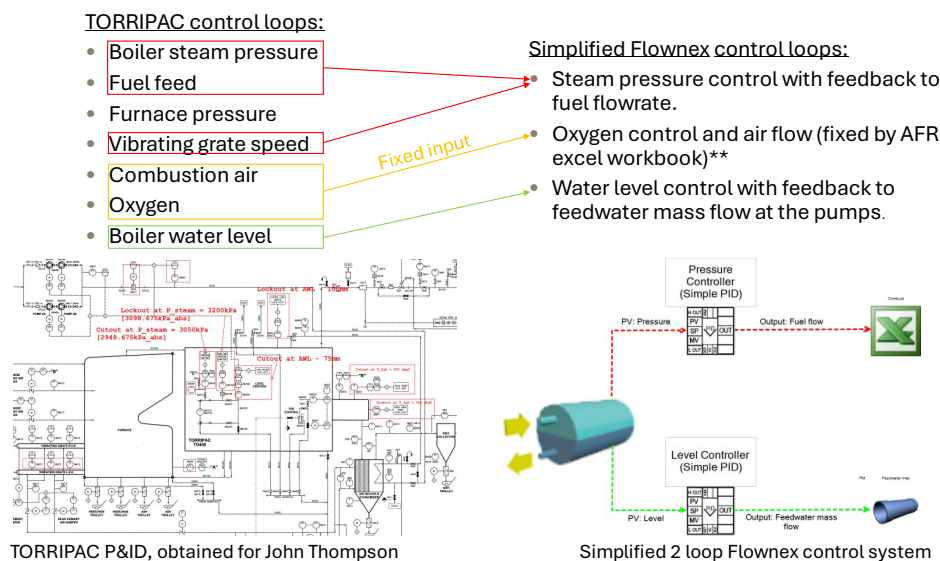


Figure 100: Comparison between real TORRIPAC control system (left) and Flownex control system (right)

The boiler steam pressure, fuel feed and vibrating speed all get reduced to a single steam pressure controller in the Flownex model. The furnace draught pressure control is not included in the Flownex model. Note also that the Flownex model has not implemented oxygen and combustion air control, instead the air-fuel-ratio (AFR) is specified as a fixed input as per the calibrated AFR of 0.241 determined in calibrated step 2. Additionally, the cooling air split ratio has been fixed at 0.25 via a *Data Transfer Link* to the cooling air *Flow Resistance* component. Future work can easily incorporate the combustion air and oxygen control by de-coupling the air-fuel-ratio and including a separate combustion air and oxygen controller.

4.5.2.1 Boiler Steam Pressure Controller Development

The pressure controller was first developed independently of the level controller which therefore required the feedwater mass flow to be set as fixed (equal to the steam extraction). It should be noted that the boiler steam pressure and water level are both parameters of the *Two-Phase Tank* and are somewhat interconnected. This means that for a fixed feedwater flow equal to the fixed

steam extraction, that a constant pressure should also result in a constant level, but this constant level is not necessarily at the design set-point of 1.488m/1.486m because the level controller has not yet been implemented.

Before testing the automatic controller behaviour during a transient run, the controller was set to manual mode with the manual value (i.e. controller output) equal to the fuel flowrate at steady state and pressure set-point equal to the *Two-Phase Tank* pressure at steady state. The steady state-initialisation was taken as the calibrated point as per section 4.4, i.e. main steam pressure of 2992.76 kPa and fuel flowrate of 0.186 kg/s. The controller bounds were chosen to be 0.1 kg/s above and below the steady state fuel flowrate of 0.186 kg/s. The proportional and integral gains on the controllers were then tuned in real time during a transient run, in search of gains which would produce a stable response within the specified controller band using a time step size of 1 second. The results from the manual user tuning of the gains are shown in Figure 101 to Figure 104 below.

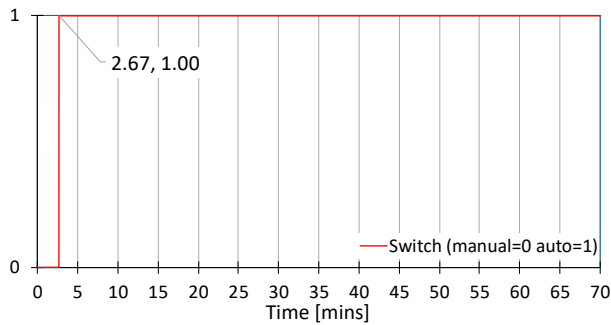


Figure 101: Pressure controller tuning @dt=1s - Pressure Switch

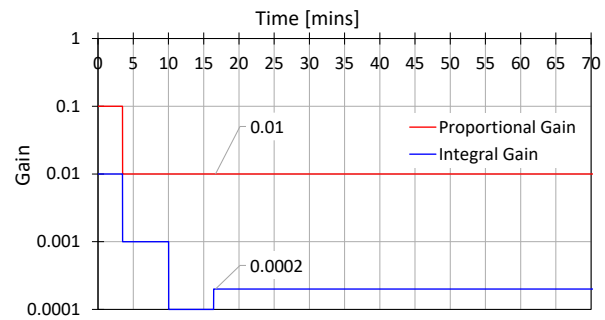


Figure 102: Pressure controller tuning @dt=1s - Gains

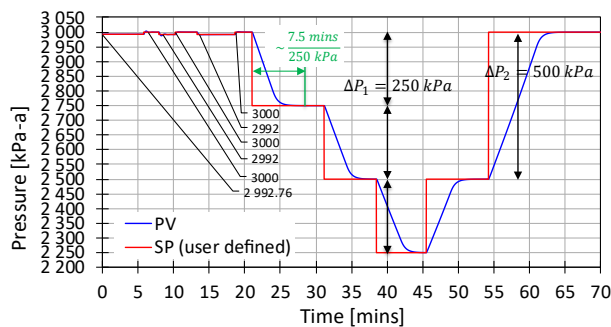


Figure 103: Pressure controller tuning @dt=1s - Pressure Tracking

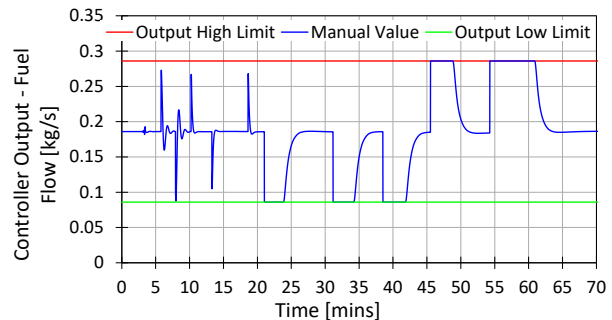


Figure 104: Pressure controller tuning @dt=1s - Controller Output (fuel flow)

The transient event was started with the controllers in manual mode and then switched into auto mode at 2.67 minutes as seen in Figure 101. The proportional and integral gains were tuned in real time starting with values of 0.1 and 0.01 respectively with the final tuned values being 0.01 and 0.0002 as indicated in Figure 102.

In practice tuning of controllers is important to obtain the correct stable behaviour and appropriate control response. This exercise demonstrates that it's possible to tune the controllers using the software as opposed to tuning it on the plant.

The gains were tuned after each real time user defined set-point change. Initially the set-point was toggled between the steady state-set-point of 2992 kPa and 3000 kPa for the first 20 minutes as illustrated in Figure 103. The controller output to the fuel flowrate is illustrated in Figure 104 where pronounced overshoots are observed between 0 and 20 minutes where the pressure set-point changes are only 8 kPa. There are no significant overshoots after 20 minutes where the pressure set-point changes are only 8 kPa. There are no significant overshoots after 20 minutes where the pressure set-point changes are changed in increments of 250 kPa up until 55 minutes.

The pressure raising curve as per John Thompson's TORRIPAC O&M manual indicates 250 kPa per 10-minute interval for an accelerated boiler start-up scenario. This was loosely used as a benchmark for the response time when tuning the pressure controller. It can be observed in Figure 103 that it takes approximately 7.5 minutes for the boiler to move between 250 kPa set-point changes.

The *Two-Phase Tank* pressure and level are interconnected as mentioned previously. If the pressure controller successfully maintains the pressure at the specified set-point, then we expect the level to also remain constant. However, since the level control has not yet been implemented, the level will find a steady point based on the pressure inside the tank because the feedwater mass flowrate has been fixed equal to the steam extraction. The level response is plotted along with the pressure response in Figure 105.

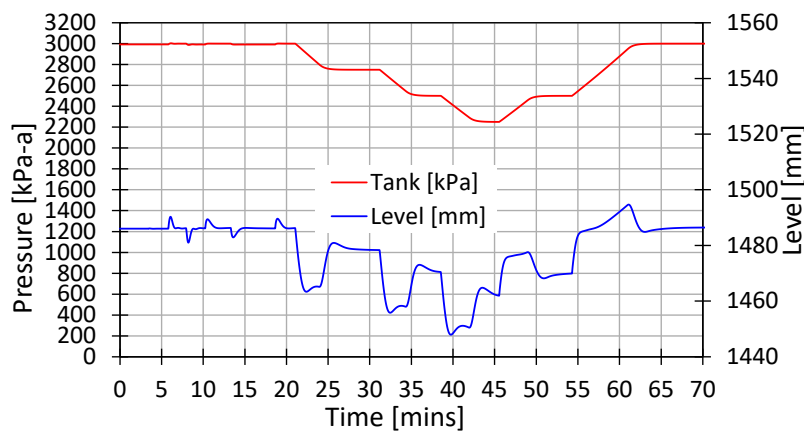


Figure 105: Pressure controller tuning @dt=1s - Two-Phase Tank pressure & level response

As expected, the level response is slower since it occurs secondary to the pressure set-point changes. This is the reason one does not see completely flat plateaus after 20 minutes in the blue-curve corresponding to the plateaus in the red curve because not enough time was allowed for the water level to settle before the next pressure set-point change was imposed.

The controller was heuristically verified by setting up a transient scenario through the “Actions Set-up” dialogue as explained in 3.3.6. The first action in the actions list was to fix the feedwater mass flow on the feedwater inlet *Pipe* using a Boolean command at time zero. This is followed by an action to switch the controller into auto mode triggered at 300 seconds. The remaining actions were to apply step changes to the controller set-point in increments of 250 kPa as shown.

The controller parameters after tuning are summarised in Table 23 with the pressure control through fuel flow adjustment verification illustrated for 250 kPa set-point changes shown in Figure 106.

Table 23: Pressure controller tuned values ($dt=1s$)

Connectable Inputs	
Controller Values	
Process Variable (PV): Two-Phase Tank Pressure	2992.755
Set-Point (SP): Two-Phase Tank Pressure	2992.755
Manual Value: Fuel Flowrate (equal to controller output)*	0.185947
Output High Limit: Fuel Flowrate	0.286000
Output Low Limit: Fuel Flowrate	0.086000
Gains	
Proportional Gain	0.01
Integral Gain	0.0002
Differential Gain	0

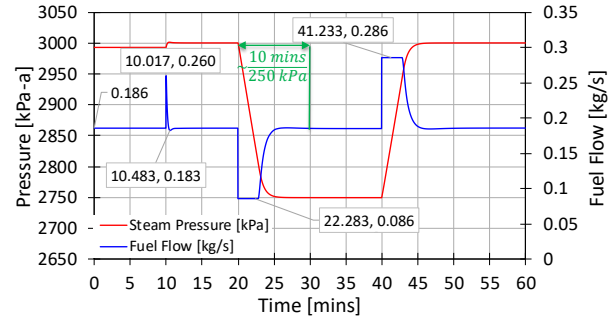


Figure 106: Pressure control verification for 250 kPa set-point changes ($dt=1s$)

The transient response shown in Figure 106 corresponds to the transient event specification as per . When the pressure set-point is changed from 2992.76 kPa to 3000 kPa at 10 minutes one notices a sharp spike in the fuel flowrate from the steady state fuel flowrate of 0.186 kg/s to 0.260 kg/s and then overshooting to 0.183 kg/s before settling again at 0.186 kg/s. The next set-point change occurs at a value of 2750 kPa at 20 minutes where the fuel flow now drops rapidly hitting the lower controller limit of 0.086 kg/s. The controller holds the fuel flowrate at this lower extreme as the boiler pressure drops to the new lower set-point but starts returning to the 0.186 kg/s steady point as the pressure nears the new set-point at 27 minutes. We see the same behaviour in reverse between 40 and 45 minutes as the pressure set-point is shifted back up to 3000 kPa.

With shorter simulation times in mind for transient investigations the same transient scenario was used to test the controller using a time step size of 5 seconds. Figure 107 compares the pressure and fuel flow response of the 1 second and 5 second time step size configurations.

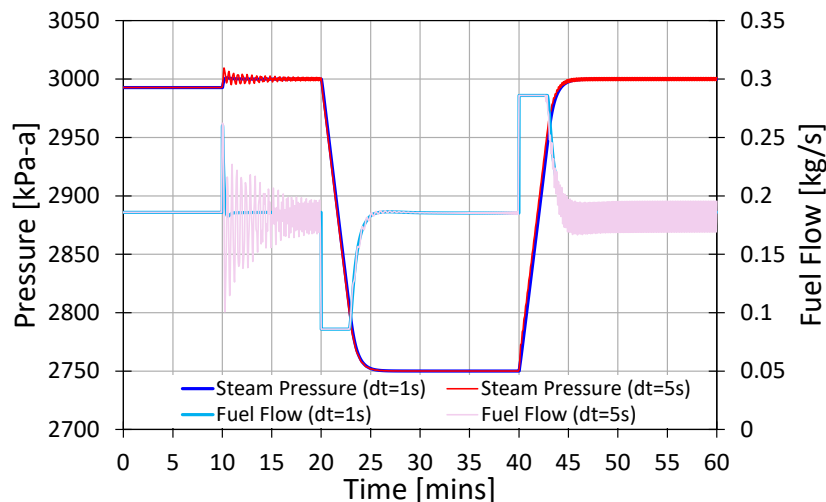


Figure 107: Time step sensitivity for pressure controller

It is clear from the figure above that a time step size of 5 seconds leads to some numerical instability in the simulation indicating that the controller limits and tuned gains for a time step size of 1 second is not adequate for a larger time step of 5 seconds.

A time step size of 1 second leads to unnecessarily long solver time for such a complex network. As such the time step size was set to 5 seconds in Flownex and the pressure controller was redesigned using the same methodology as presented previously. The final redesigned controller values are given in Table 24 below and the verification graph presented in Figure 108.

Table 24: Pressure controller tuned values ($dt=5s$)

Connectable Inputs	
Controller Values	
Process Variable (PV): Two-Phase Tank Pressure	2992.755
Set-Point (SP): Two-Phase Tank Pressure	2992.755
Manual Value: Fuel Flowrate (equal to controller output)*	0.185947
Output High Limit: Fuel Flowrate	0.294000
Output Low Limit: Fuel Flowrate	0.078000
Gains	
Proportional Gain	0.0006
Integral Gain	0.000001
Differential Gain	0

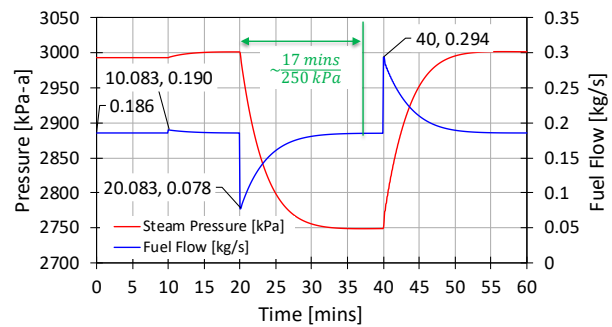


Figure 108: Pressure control verification for 250 kPa set-point changes ($dt=5s$)

A comparison between the responses in Figure 106 and Figure 108 indicates a much slower response for the 5 second time step tuned controller where it takes approximately 17 minutes to move between 250 kPa set-point changes as opposed to approximately 10 minutes for the 1 second time step tuned controller. Nevertheless, the pressure controller successfully adjusts the fuel flowrate accordingly to manage the boiler steam pressure at the specified set-point.

4.5.2.2 Water Level Controller Development

The level controller monitors the *Two-Phase Tank* level and adjusts the feedwater flowrate accordingly with its output connected to the feedwater *Pipe* component upstream of the economiser. Recall in the pressure controller development that this feedwater flow was fixed and set equal to the steam extraction flowrate using a *Data Transfer Link* from the steam extraction *Pipe* component to the feedwater inlet *Pipe* component. For the development of the level controller, the feedwater flowrate is again fixed but this time it is linked to the output of the level controller.

Like the pressure controller development, the level controller was set to manual mode at steady state with the manual value (i.e. controller output) equal to the steam flowrate at steady state (0.7795 kg/s). The level set-point was also set equal to the operational design average water level of 1.486m. The lower and upper controller limits on the feedwater flowrate output were set to 0.01 kg/s and 2.0 kg/s respectively. The proportional and integral gains were then tuned in real time. The results of the manual tuning are shown in Figure 109 to Figure 112.

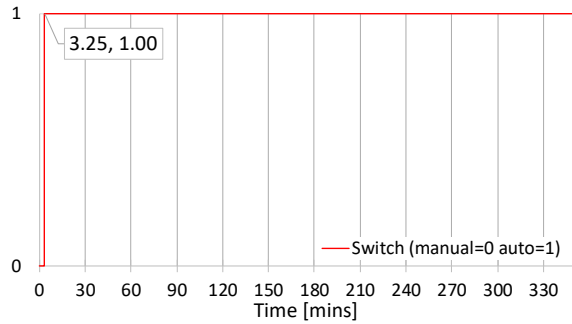


Figure 109: Level controller tuning - Level Switch

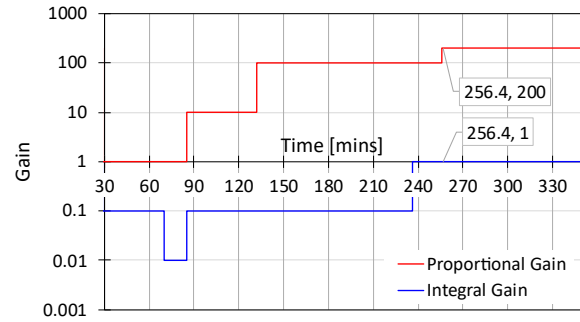


Figure 110: Level controller tuning - Gains

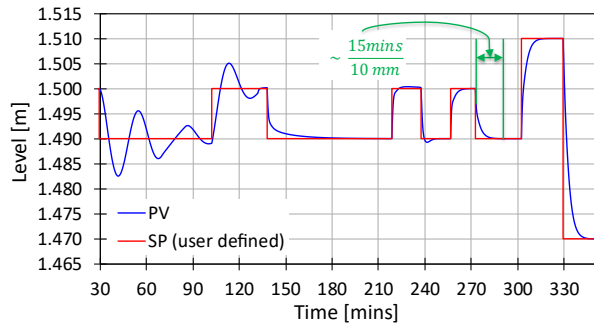


Figure 111: Level controller tuning - Level Tracking

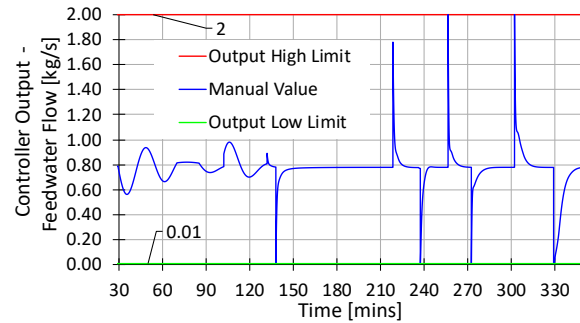


Figure 112: Level controller tuning - controller output (feedwater flow)

Although the level here is being controlled independently, one can still observe the relationship between the level and the pressure inside the *Two-Phase Tank* illustrated in Figure 113 below.

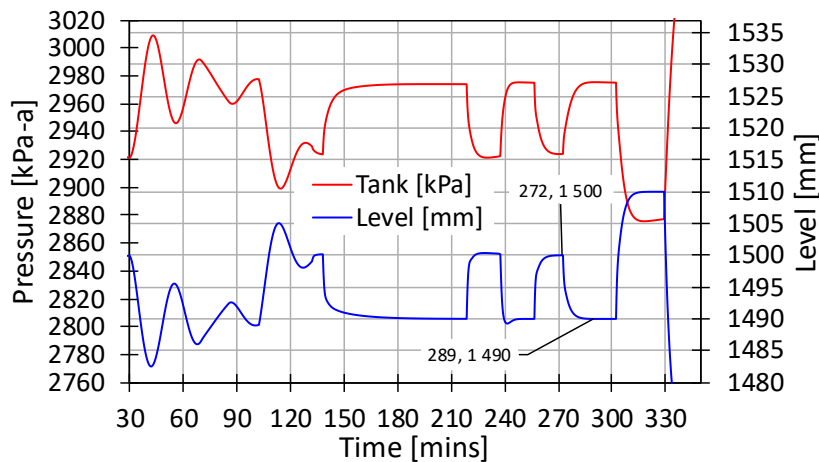


Figure 113: Level controller tuning - Two-Phase Tank pressure & level response

This time the fuel flowrate and steam extraction remain unchanged. Instead, the level set-point is toggled between 1490 mm and 1500 mm. The level controller sends a signal to adjust feedwater flowrate to target the new level set-point. We observe mirrored behaviour in the pressure and level in Figure 113. For example, at 272 minutes when the level set-point is changed from 1500 mm to 1490 mm, the level follows an exponential curve downward to reach the new level set-point at 289 minutes while the pressure increases accordingly following an upward exponential trajectory.

The level controller was heuristically verified by setting up a transient scenario through the “Actions Set-up” dialogue. The first action in the actions list was to fix the feedwater mass flow on the feedwater inlet pipe using a Boolean command at time zero. This is followed by an action to switch the controller into auto mode triggered at 300 seconds. The remaining actions were to apply step changes to the controller set-point to 1.5 m at 600 seconds and then in increments of 0.01 m (10 mm).

The controller parameters after tuning are summarised in Table 25 with the level control through feedwater flow verification illustrated alongside in Figure 129.

Table 25: Level controller tuned values

Controller Values	
Process Variable (PV): Two-Phase Tank Level	1.486
Set-Point (SP): Two-Phase Tank Level	1.486
Manual Value: Feedwater Flowrate (equal to controller output)*	0.77954
Output High Limit: Feedwater Flowrate	2.00
Output Low Limit: Feedwater Flowrate	0.01
Gains	
Proportional Gain	200
Integral Gain	1
Differential Gain	0

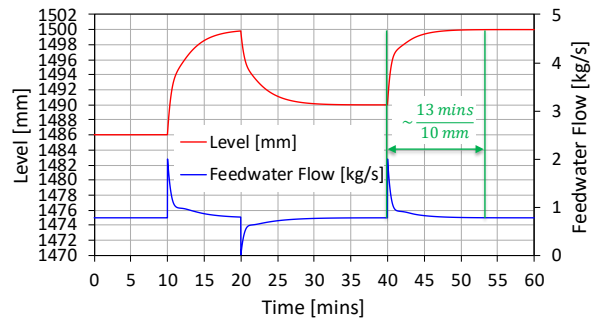


Figure 114: Level control verification for 10 mm set-point changes

The response from the controller values in Table 25 was the same regardless of if 1 second or 5 seconds was used as the time step size. It takes approximately 13 minutes for the system to settle at a new level set-point 10 mm above or below the preceding set-point. In the absence of transient site data for validating the actual transient response time, 13 minutes per 10 mm level shift seems reasonable considering the large volume/mass of water in the shell. The response time between 10 mm set-points may vary slightly depending on the cross-section at which the water-level is currently at. Recall that the tank volume is essentially a horizontal cylinder with some voids, so the volume between successive 10 mm increments will vary as a function of the height above the bottom of the horizontal cylinder.

4.5.2.3 Dual Pressure-Level Control System Development

Now that each of the two controllers have been developed and tested independently, the entire boiler system with both controllers active can be tested. This is done by enabling both controllers with the tuned controller values set as per Table 24 and Table 25 and being initialised from a known calibration point. To aid initialisation of the transient scenario, both controllers are set-to manual and then switched into auto mode some short time after initialisation of the transient.

Once the dynamic simulation has started and both controllers are in auto mode various disturbances can be imposed on the system such as changing the fuel quality and/or steam extraction. One can then verify if both controllers are effective in controlling their respective parameters.

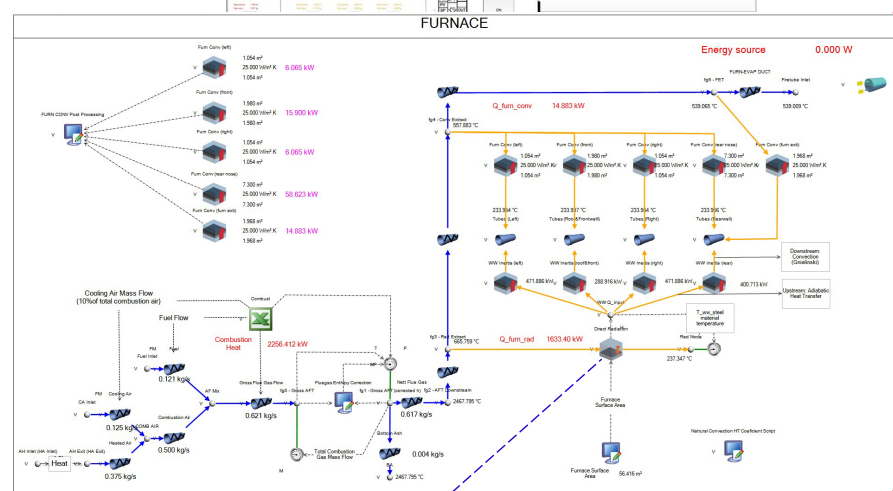
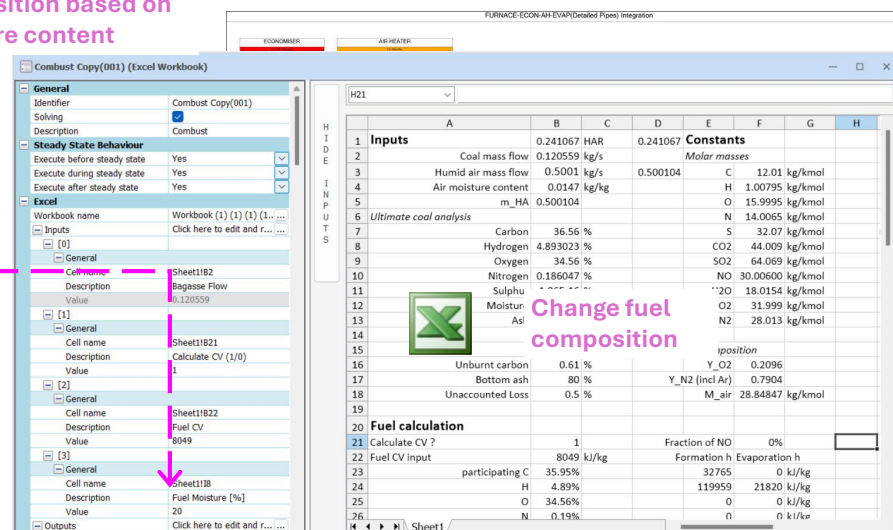
The set-point of each controller will remain unchanged. The disturbances will be imposed elsewhere to emulate real life operational transient scenarios. For example, the quality of solid fuels, especially biomass fuels like bagasse can be highly variable. The ability to change the fuel moisture content in a dynamic simulation can therefore provide valuable insights on the boiler's response to fuel quality variability. Additionally, changing the steam extraction during a transient run emulates steam demand variability which is common for various industries and informed by the rate of production processes in the relevant factory. The results from these two scenarios are presented in section 5.4.

It is important to qualify that the controller tuning methodology as illustrated in section 4.5.2.1 and 4.5.2.2 employs a trial and error approach in search of a stable response. Future studies could focus on methodologies to determine required gains and controller limits for optimised response for specific boiler operational scenarios.

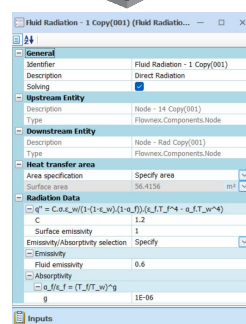
4.6 Overview of Integrated Whole-Boiler Model

Figure 115 overleaf presents the full model from a step back. At the centre of the figure is the main canvas page in Flownex which contains the relevant “views” of the important nodes and elements. The individual models for each heat exchanger can be seen radiating out from the centre via the red dashed lines. The blue dashed arrows point to a deeper level of detail within each of the models where key parameters can be changed as necessary. Finally, the magenta dashed arrows and text identify the locations at which these key parameters can be adjusted; for calibration purposes, off-design condition testing and or transient response investigations.

Renormalise fuel composition based on moisture content



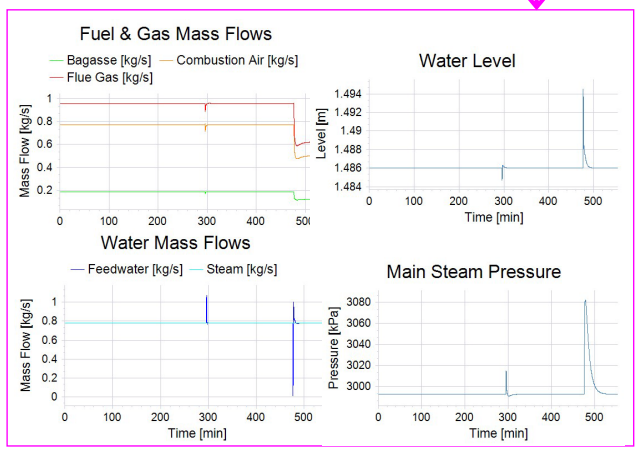
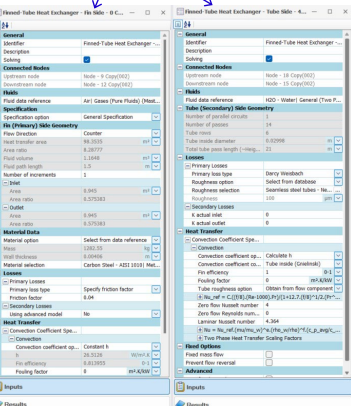
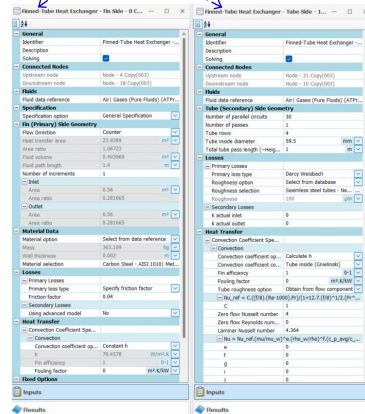
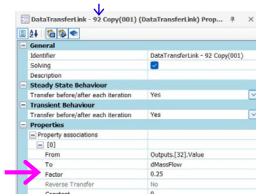
Tune radiation contribution @Fluid Radiation component via fluid emissivity, absorptivity, and C-factor



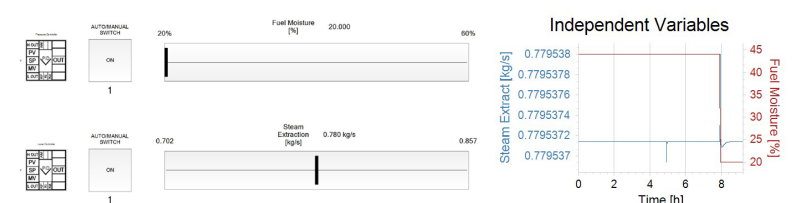
$$q'' = C \sigma \epsilon_f (T_f^4 - T_w^4) \left(\frac{A_f}{A_w} \right) \left(\frac{1}{1 - \epsilon_f} + \frac{1}{\epsilon_f} \right)$$

A: Projected area of furnace (fixed)
C: Effective view factor (calculated)
Emissivity/Absorptivity selection: Specify
Emissivity: 0.6
Absorptivity: 0.6
Inputs: g

Adjust cooling air mass flow split @Data Transfer Link

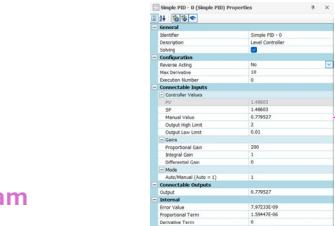


Implement system disturbances through slider controls



Change steam extraction @boundary condition

Calibrate feedwater pumps exit condition @boundary condition



Monitor system response through line graphs

Tune controller gains

Figure 115: Complete integrated controllable TORRIPAC boiler FlowNex model

5. Model Application

5.1 Steady State Operating Conditions

The model has been developed using the design base information, and all heat exchangers except the furnace and evaporator results compared favourably with the MCR design data. The integrated model, and primarily the furnace, has been calibrated using 70% MCR site data. Given the care that was taken to implement reputable correlations that change with operating conditions, and geometry that accurately represents the boiler, it is expected that the model will produce credible results for all normal operating conditions.

At the very low load conditions however, combustion instability may result in different heat uptake in the furnace, hence the model results may be inaccurate there. Unfortunately, no low load steady operating data was available to test the model's performance. Nevertheless, start-up transients could be performed using the model, possibly supplemented with low load calibration if data becomes available.

The focus of this project was to develop a dynamic model of the boiler. Though steady state prediction is important to achieve credible transient results, most of the modelling effort was placed in calibrating the model for a known condition and then using this calibration point to initialise the transient system to investigate dynamic characteristics. As such, results for other operating conditions will not be discussed here.

However, the transient scenarios presented in section 5.4 did solve the boiler for different load set-points such that the potential steady state conditions could be extracted. These results are presented for interest's sake in Appendix F.

5.2 Grid and Time Step Sensitivity

This section discusses the importance of grid and time step sensitivity in the context of transient simulations. Time step size refers to the period between successive temporal solutions, while grid size refers to the spatial discretisation of the flow network.

The effects of time step size and grid size are illustrated here through a case study that uses a small piece of the furnace model. This is the combined radiative and convective heat transfer from the flue gas to a single waterwall (left wall) as illustrated in Figure 116.

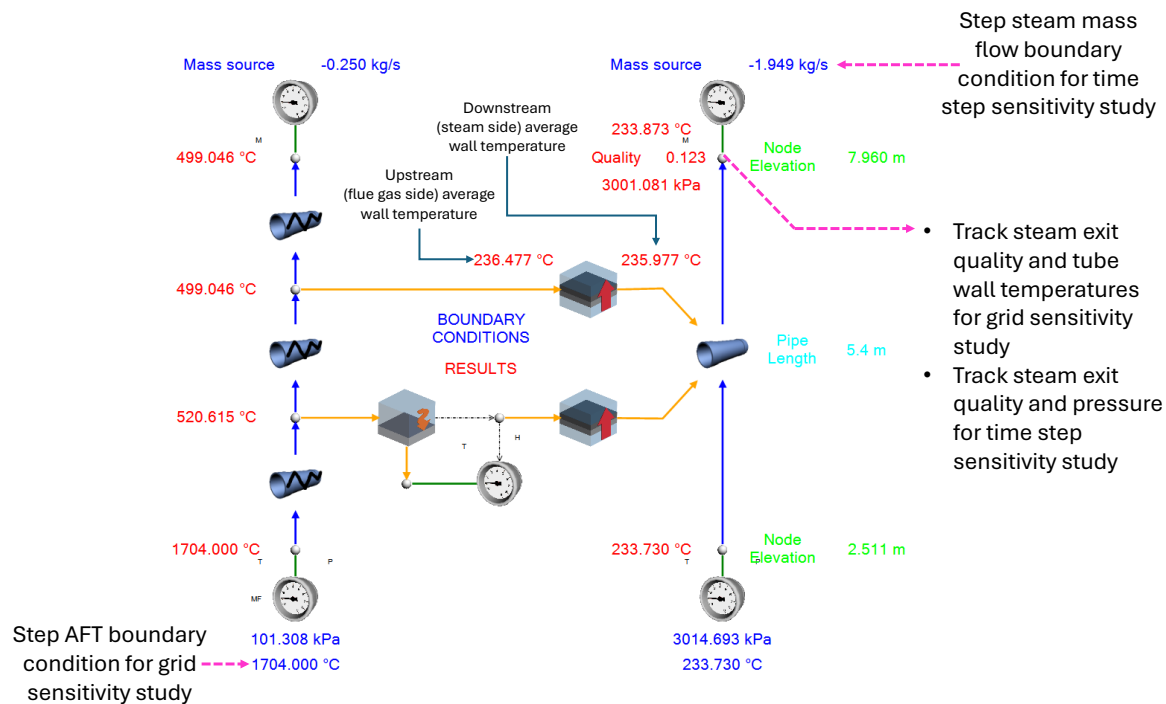


Figure 116: Grid & Time Step Sensitivity Case Study: Combined radiative and convective heat transfer to the left furnace waterwall

5.2.1 Grid Sensitivity

Grid size in the forgoing context of heat transfer to a single furnace waterwall is limited to the discretisation of the pipe wall thickness. The 5.45m length of the water column has been discretised into 10 increments in the present work but this has little effect in the overall energy balance solution since the water properties do not have any meaningful change along the length of the pipe.

To illustrate the effects of grid sensitivity, the left waterwall in Figure 116 was discretised across the pipe thickness by specifying the number of nodes in the element direction as an input to the connected *CHT* components. The wall thickness is 4.06 mm. To assess grid independence, one must determine the minimum number of nodes for which the solution does not change significantly.

The transient scenario implemented involved initialising the system from a steady point at time zero. Then a 46°C step up in the adiabatic flame temperature boundary condition (1704-1750°C) was imposed at time ($t=10s$). A time step size of 0.001 seconds was used for a total simulation time of 120 seconds for 3 cases of increasingly fine grid resolutions - 2, 5, and 10 nodes along the pipe wall thickness. The quality of the steam exit node was tracked as an indicator of the grid sensitivity.

Figure 117 below shows the results for the full 120 second simulation, while Figure 118 shows the same results plotted at a narrower time band (9.5-15s).

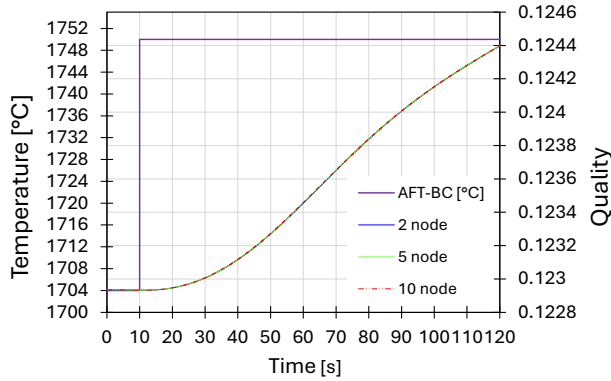


Figure 117: Wall thickness grid sensitivity effect on exit steam quality (0-120s)

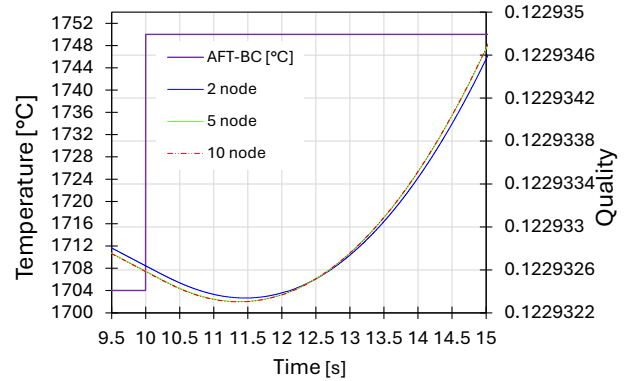


Figure 118: Wall thickness grid sensitivity effects on exit steam quality (9.5-15s)

Inspection of Figure 117 seems to indicate that the solution is grid independent even at a very course discretisation of only 2 nodes. However, upon zooming in to a 5.5 second time scale, Figure 118 demonstrates deviation between the 2 node solution with the 5 and 10 node solution. The 5 and 10 node solution are indistinguishable at this time scale. It can therefore be said that the solution is grid independent at 5 nodes. Figure 119 and Figure 120 below show the outer and inner tube wall temperatures responses respectively at the same 5.5 second time scale as used in Figure 118.

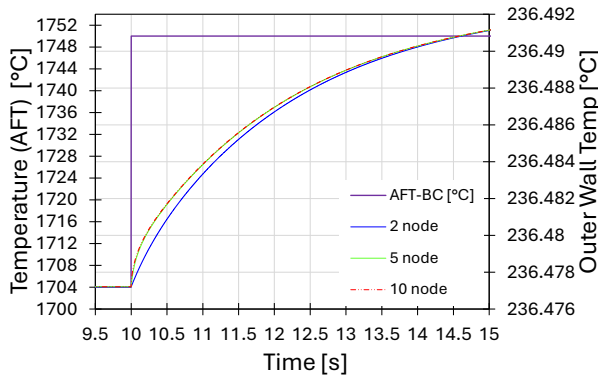


Figure 119: Wall thickness grid sensitivity effect on average outer tube wall temperature (9.5-15s)

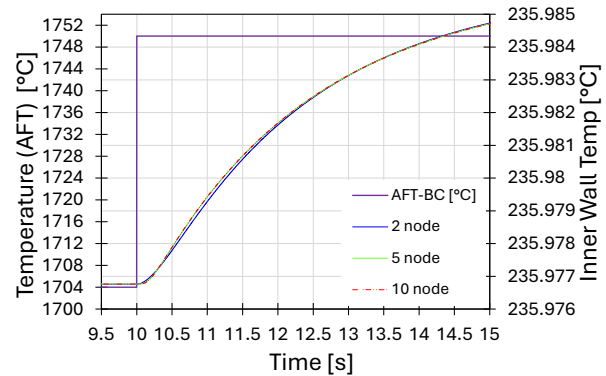


Figure 120: Wall thickness grid sensitivity effect on average inner tube wall temperature (9.5-15s)

The tube wall temperatures show a better indicator of the deviation of the 2-node solution. It is important to perform grid independence studies, and the required investigative questions should inform whether a finer grid resolution is necessary. For the purposes of the present work which performs simulations in the order of minutes-hours and is more concerned with the changes in total energies in the system - even a course resolution of 2 nodes along a 4.06 mm thick pipe would produce results indistinguishable from finer resolutions (as shown in Figure 117).

5.2.2 Time Step Sensitivity

Time step sensitivity studies are very important in dynamic analyses since the chosen time step can influence the accuracy and stability of the transient solution. Numerical instability can occur at

excessively large time steps. The specific transient scenario investigated influences one's choice for a time step size. If it is intended to observe fast transient phenomena such as water-hammer and pressure waves, then one needs to choose a suitably small-time step size. If one is more interested in system level inertia and total energy changes over longer durations, then larger time step sizes are more apt and will save significantly on computational expense.

The Alpha Crank-Nicolson factor (α_{CN}) also influences the stability of a transient solution. It is a temporal relaxation factor between 0 and 1 which aids stability of the solution. An Alpha Crank-Nicolson factor of 0 corresponds to a fully explicit scheme, while a value of 1 is used for a fully implicit implementation. In general, a good rule of thumb for small to moderate time steps is 0.7. A fully implicit scheme ($\alpha_{CN}=1$) could be used to stabilise a simulation with large time-steps in the range of 1-5 seconds where the details of fast wall conduction and pressure waves is not important to be captured.

To illustrate the effects of time step sensitivity, the same case study model as used for the grid sensitivity study is used here – i.e. heat transfer to the left water wall as per Figure 116. The difference now is that it is the steam extraction mass flow boundary condition that is stepped down by 5% at time ($t=10s$) – that is a step from 1.95 kg/s to 1.825 kg/s. The simulation end time was set to 400 seconds.

The variable that is tracked dynamically is the water wall outlet quality. Since the wall contains a saturated mixture, dropping the steam flow rate should naturally result in an increase in the quality of the saturated mixture at the top (exit). The time step sizes investigated ranged from small (0.0001-0.01s) to moderate (0.01-0.5s). Large timesteps of (1-5s) were also included.

Figure 121 shows the exit quality response at increasing time step sizes and Figure 122 illustrates how changing the Alpha Crank-Nicolson factor can stabilise the solution when using large time steps (i.e. 5 seconds). Note that all responses in Figure 121 used 0.7 as the Alpha Crank-Nicolson factor.

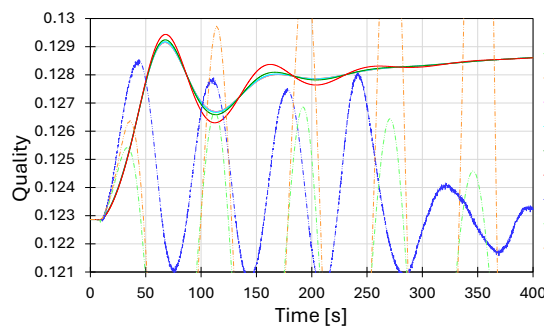


Figure 121: Time step sensitivity effects on waterwall steam exit quality (0-400s)

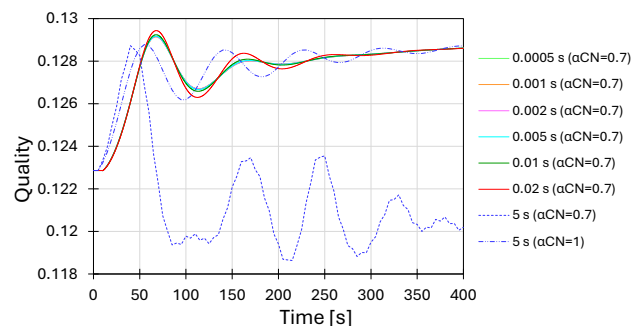


Figure 122: Effect of the Alpha Crank-Nicolson factor at large time steps

The quality response in Figure 121 shows little deviation for time step sizes from 0.0005 seconds and 0.01 seconds, with slightly more pronounced peak and trough amplitudes at a time step size of

0.02 seconds. Time step sizes from 0.05 seconds to 1 second show significant deviation with a much larger amplitude response which fluctuates around a lower mean quality value. It is observed in Figure 122 that employing a fully implicit temporal scheme (i.e. $\alpha_{CN}=1$) at a large time step of 5 seconds can produce a response which tracks the higher resolution time responses when viewed over a larger time band (i.e. 400s).

A possible reason for moderately sized time steps (0.05-1.0s) being inaccurate and unstable for the specific transient scenario shown here could be due to pressure wave phenomena that occur due to a very sudden step change in the steam mass flow.

The speed of sound in the waterwall is approximately 504.162 m/s as evaluated at the average steam temperature and pressure of 233.96°C and 3000 kPa respectively. This means that a pressure wave could take approximately 10.9 milliseconds (0.0109s) to reach the bottom of the 5.45 meter column and a total of 21.8 milliseconds (0.0218s) to pass to the bottom and return to the top of the waterwall. If it is required that pressure wave phenomena be observed, then it is important to choose a suitably small-time step size. A factor of 10 smaller than 0.0218 seconds is a safe assumption to capture possible pressure wave phenomena in this case.

The outlet pressure for the same transient scenario has also been tracked. It is plotted in Figure 123 below from 9.9 seconds (i.e. just before the mass flow disturbance) to 10.3 seconds to observe the fast transient effects.

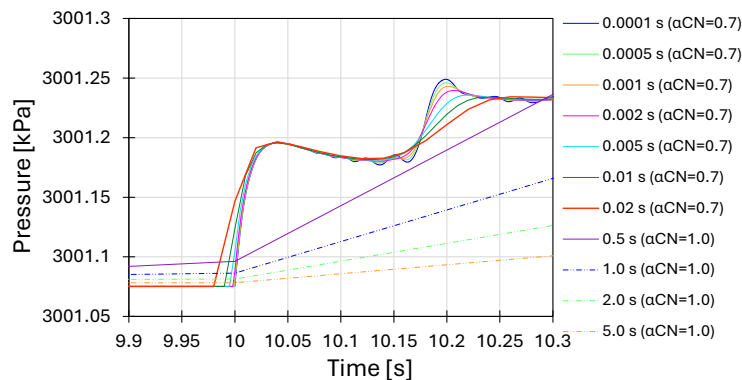


Figure 123: Time step sensitivity effects on waterwall steam exit pressure (9.9-10.3s)

The pressure peaks of the lower frequency wave can be observed at around 10.025 seconds and again at 10.2 seconds. All time step sizes up to 0.02 seconds at least follow the same trend even though their peaks are slightly out of phase. The time step size of 0.0001 second (blue curve) is able to capture some higher frequency behaviour within the lower frequency wave time band. It can also be noted that the lower frequency pressure waves die out around the 12 second mark (not plotted in the above figure).

The exact physical phenomena and/or numerical accuracy from these results are not claimed here. This analysis simply shows the importance of investigating time step dependence and how the

choice of time step size can affect the accuracy and/or stability of different transient solution responses. The time step sensitivity case study presented here further shows how smaller time steps can capture fast transient phenomena, while large time steps coupled with suitable solver stability settings will miss the details of fast transient phenomena but still be reliable in capturing system level inertia responses.

5.3 Thermal Inertia Studies

To investigate the effect (and importance) of thermal inertia on the transient response, the fully integrated model was run for two uncontrolled transient scenarios:

1. Fuel flow step changes.
2. Steam extraction step changes.

A separate low inertia configuration of the model was set-up by simply changing all the AISI-1010 carbon steel materials to the custom low inertia steel with the negligibly small specific heat capacity of $1\text{e-}6$ W/kgK as explained in 3.3.2. The thermal inertia of the fluids is still fully accounted for in both the full inertia model and low inertia model, so the transient response results in this section only illustrate the effect of the solid steel thermal inertia.

The transient scenarios in this section have been set-up with no active controllers. The feedwater mass flow entering the evaporator has been fixed equal to the steam extraction such that the transient could be run without a quality boundary condition on the two-phase tank during the transient simulation. Without this fixed flow, even small changes in the main steam pressure results in massive transient mass flows entering the evaporator, causing the level to overshoot and fill the evaporator completely. Liquid then gets extracted at the steam extraction point which immediately results in solver instabilities. The cause of this instability is linked to the absence of an actual controlled pump driving the flow, as well as feedwater pressure drops that may be too low as compared to the actual plant.

Since pressure drops between the pump exit and evaporator inlet do not have any noticeable effect in the energy balance, no effort was made to model this accurately. There is also no data to properly verify the pressure drop along the feedwater line. Future studies could include a suitable pump and/or orifice to emulate the pressure drop along this feedwater line if design or site data can be obtained to verify the true pressure drop characteristics.

5.3.1 Fuel Flow Step Changes

The transient scenario for the fuel flow profile illustrated in Figure 124 was set-up using the “actions set-up” dialogue where the transient is initiated from the steady state calibration point (fuel flow of

0.186 kg/s). The fuel flow was then toggled 10% up and down in 10-minute intervals. Figure 125 and Figure 126 show the pressure and level response respectively of the *Two-Phase Tank* for both inertia configurations.

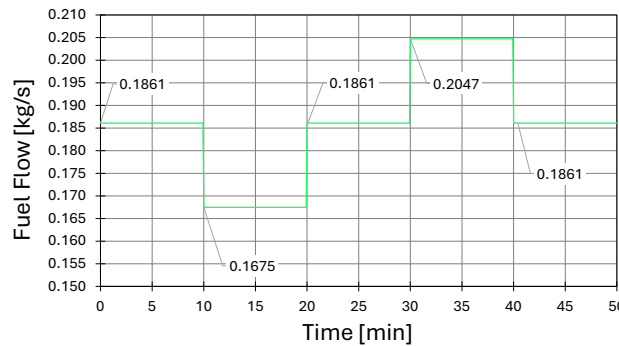


Figure 124: Fuel flow input profile

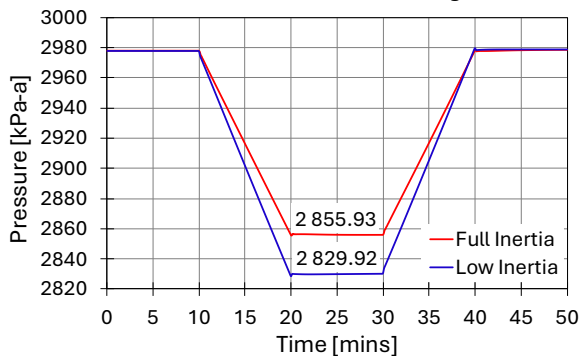


Figure 125: Two-Phase Tank pressure response for fuel step changes (Full Inertia & Low Inertia configuration)

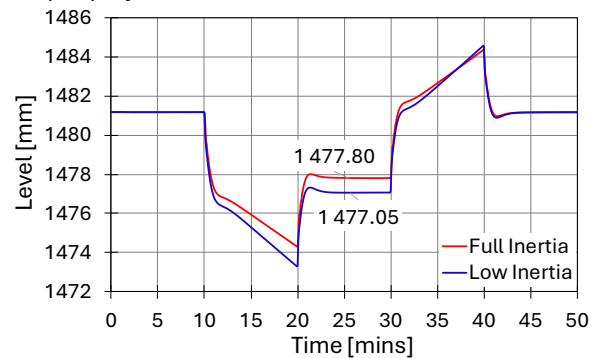


Figure 126: Two-Phase Tank level response for fuel step changes (Full Inertia & Low Inertia configuration)

One can see in the figures above that a decrease in fuel flow results in the expected drop in the boiler steam pressure. As expected, the low inertia case responds faster than the full inertia case. As the boiler pressure drops off for a constant steam extraction, less fuel flow causes less steam generation and consequently the level drops off as the water buffer in the evaporator gets used up.

Since the system is uncontrolled, the full inertia and low inertia cases would likely keep diverging as seen for example from 10 minutes to 20 minutes. The reason that there are plateaus at 0-10, 20-30, and 40-50 minutes is because the fuel flow is at its original calibrated point (i.e. 0.186 kg/s) at these intervals. If only one disturbance (i.e. fuel flow change) was implemented the boiler pressure would decrease continuously and the *Two-Phase Tank* would be completely emptied of its water.

The fact that one sees the full inertia case lagging the low inertia in an uncontrolled transient scenario already indicates the importance of including the inertia since this lag is what would happen in practice. This information is important for controller designers as it can be used to inform time delays between triggers in the control system as well as inform the controller gains.

Controllers developed using a dynamic simulation which does not include thermal inertia would therefore not necessarily be optimum for a real-life plant.

5.3.2 Steam Extraction Step Changes

The transient scenario for the steam extraction profile illustrated in Figure 127 was initialised at the steady-state calibration point (0.7795 kg/s steam flow) and step changes equal to 5% of the steady-state steam flow were imposed at 10-minute intervals. Figure 128 and Figure 129 show the pressure and level response respectively of the *Two-Phase Tank* for both inertia configurations.

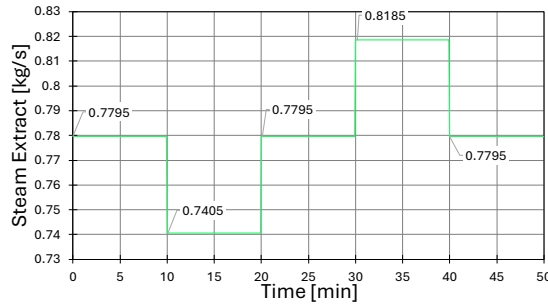


Figure 127: Steam extract input profile

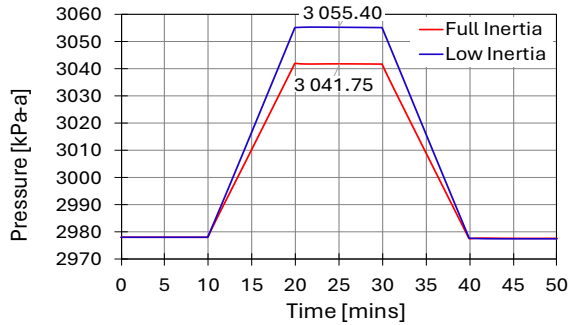


Figure 128: Two-Phase Tank pressure response for steam step changes (Full Inertia & Low Inertia configuration)

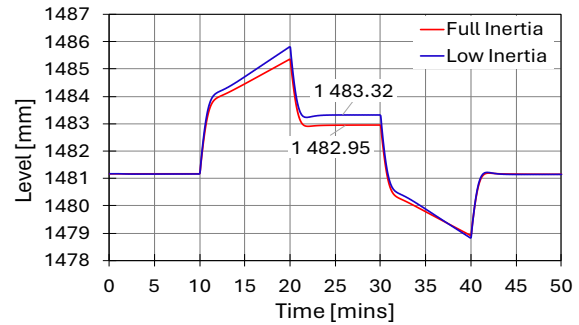


Figure 129: Two-Phase Tank level response for steam step changes (Full Inertia & Low Inertia configuration)

The analysis here is inverse to the fuel flow step change analysis. The reduced steam extraction for the same rate of steam generation results in a rise in the water level as the buffer gets filled.

5.4 Transient Studies with Active Control

Two transient simulations are presented here: the first is for a fuel moisture profile and the second, a steam demand profile.

5.4.1 Fuel Moisture Profile

A fuel moisture profile as illustrated by the red lines in Figure 130 and Figure 131 was implemented in the action set-up for the transient scenario. The simulation is initialised at the steady state calibration point as per section 4.4 where the fuel moisture content was 44%. Thereafter step changes in the fuel moisture content are applied up to 50% one hour into the simulation and to 56% after 1.5 hours. The remaining step changes are also implemented in 1.5-hour increments, this time decreasing to 40% at 4 hours and finally to 35% at 5.5 hours for a total simulation time of 7 hours.

The boiler steam pressure response is plotted with the fuel moisture profile in Figure 130 and the water level plotted in Figure 131.

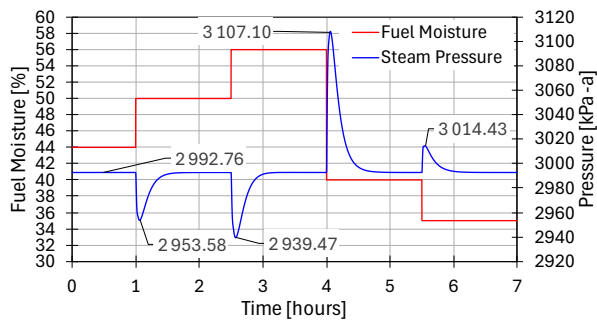


Figure 130: Boiler steam pressure response to fuel moisture changes

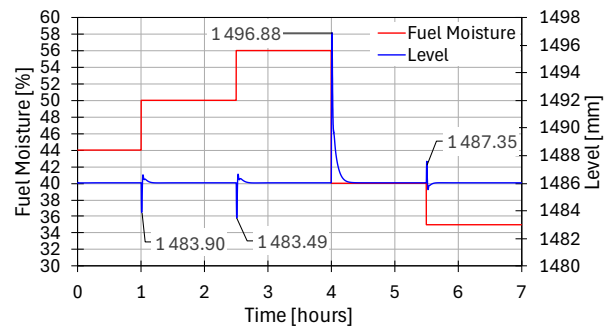


Figure 131: Boiler level response to fuel moisture changes

As the fuel moisture is increased from 44% to 50% at the one-hour mark, the fuel quality decreases due to a lower carbon content. This means that less energy is available for combustion for the same fuel flowrate and consequently there is a drop in boiler steam pressure to 2953.58 kPa as shown in Figure 130. The drop in pressure then signals the pressure controller to increase the fuel flowrate accordingly to maintain the evaporator at its pressure set-point of 2992.76 kPa. The pressure controller successfully brings the pressure back to its set-point within 45 minutes. One can see the same behaviour in reverse when the fuel moisture is decreased at the 4- and 5.5-hour marks. The level controller responds to the intermediate change in water level by adjusting the feedwater flow rate until the level is back to the desired setpoint.

Figure 132 and Figure 133 show the controller response by plotting the controller process variables on the same graph as the consequent controller outputs. That is the pressure response to the fuel flow from its controller output in Figure 132 and the level response to the feedwater flow from its controller output illustrated in Figure 133.

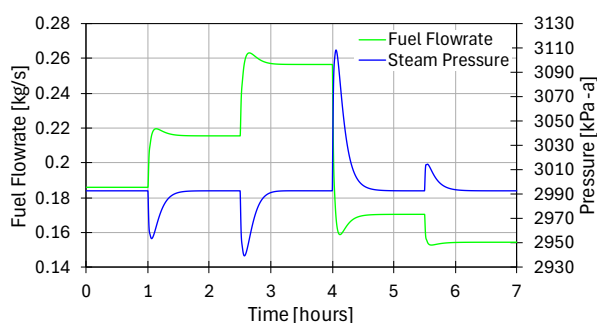


Figure 132: Boiler steam pressure response to fuel mass flow changes

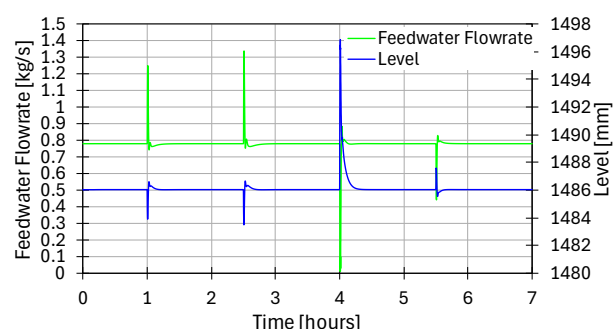


Figure 133: Boiler feedwater response to level changes

The level responds and settles more quickly than the pressure. This can be seen in Figure 131 at the first fuel moisture step change at the one-hour mark, where the level shows a sharp spike 3 mm below the set-point before overshooting and finally re-settling all in the space of 12 minutes. Note

that the settling time for the larger disturbance at 4 hours is slightly longer at approximately 25 minutes but there is no overshoot. These results are a function of the specific tuned controller values. Future studies could easily tune the steam pressure controller as the primary controller to increase its response time relative to the level controller response.

Figure 134 and Figure 135 illustrate the consequent changes in the flue gas temperatures.

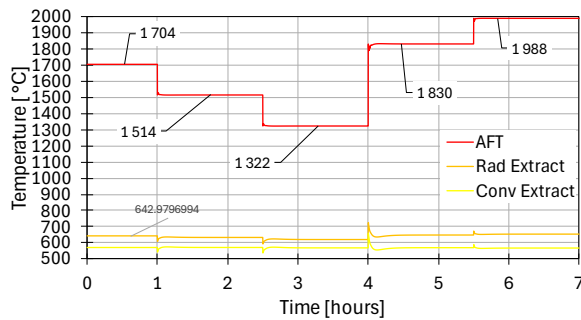


Figure 134: Flue gas temperatures upstream of furnace exit

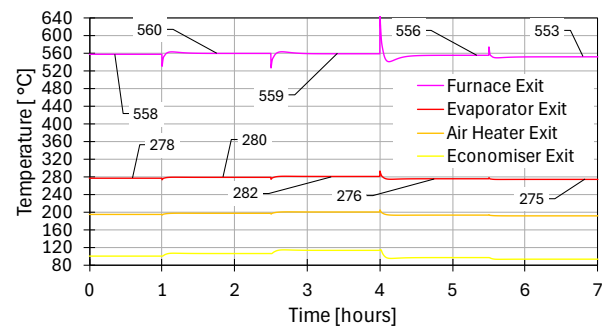


Figure 135: Flue gas temperatures downstream of furnace exit

It is clear in Figure 134 that the adiabatic flame temperature (AFT) is the most sensitive to the fuel moisture changes which makes sense due to the influence of the fuel composition on the combustion calculations. The AFT has an indirect relationship with the fuel moisture content due to a drop in calorific value for a fuel with higher moisture and vice versa. The additional moisture also captures more heat in the form of phase change rather than sensible heat, resulting in lower temperatures.

The flue gas temperatures downstream as plotted in Figure 135 show an opposite trend, that is, the temperatures increase with an increase in fuel moisture. This is due to increased heat absorption in the furnace since more energy is now required to evaporate the additional moisture in the fuel. The latent heat required to evaporate the moisture therefore increases the overall heat demand in the furnace as illustrated by the increase in combustion heat in Figure 136.

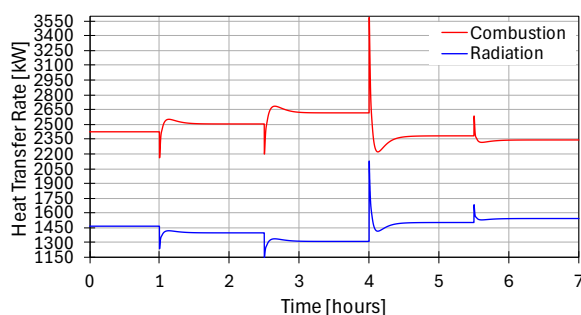


Figure 136: Combustion & radiation heat transfer rates

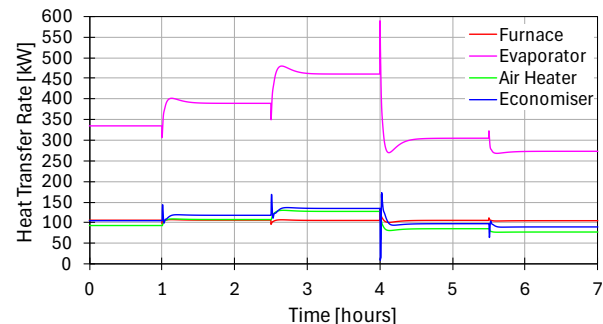


Figure 137: Convection heat transfer rates

The combustion heat increases with the increase in fuel moisture because the pressure controller has increased the fuel flowrate accordingly. Similarly, the combustion heat decreases when the fuel moisture decreases since the controller now signals for less fuel to be supplied. The sensitivity in

the combustion energy therefore seems to be dominated by the mass flowrate more than the newly calculated calorific value (CV) as calculated by the excel combustion workbook for the new fuel moisture content. On the other hand, one observes a drop in radiation heat transfer for increased fuel moisture and a rise in radiation heat transfer for a drop in fuel moisture content. The flue gas specific heat capacity increases when there is a higher moisture content. This results in lower flue gas temperatures for the same enthalpy.

5.4.2 Steam Demand Profile

The steam demand profile employed through the action set-up is displayed in red in Figure 138 and Figure 139 below. The demand profile starts at 70% MCR as per the steady state calibration in section 4.4 and then is changed to 85% after 1 hour, up to the 100% MCR condition (4 ton/hr) at 2.5 hours and then down to 60% and 50% (i.e. half load) at 4 hours and 5.5 hours respectively. The resulting steam pressure response and level response are plotted in Figure 138 and Figure 139 respectively.

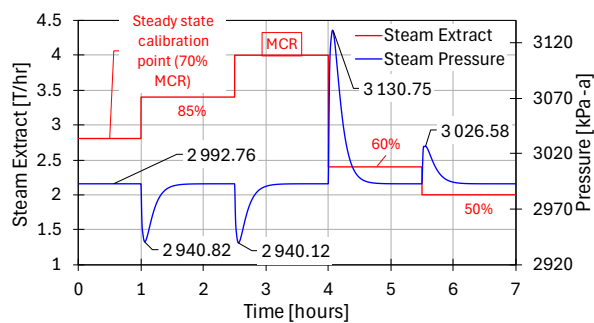


Figure 138: Boiler steam pressure response to steam demand profile

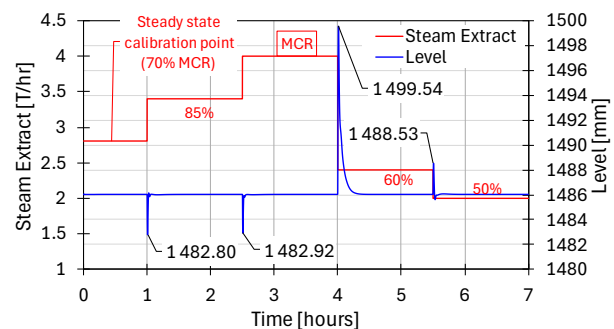


Figure 139: Boiler level response to steam demand profile

A similar trend as the fuel moisture change scenario is observed where the direction of the pressure and level response are identical. However, the magnitudes of the fluctuations are different and occur for a different reason. An increase in steam extraction for the same fuel flow again means that the boiler pressure will drop, thereby signalling the pressure controller to increase the fuel flowrate. Additionally, the resulting drop in the level signals the level controller to increase the feedwater flowrate such that it balances with the new steam extraction to maintain the water level at its set-point of 1486 mm.

Figure 140 shows the fuel, air, and flue gas mass flow responses. The fuel is increased or decreased by the pressure controller to maintain the correct evaporator pressure. Recall that the combustion air mass flow has been fixed inside of the combustion excel workbook to ensure the constant calibrated air-fuel-ratio (AFR) of 0.241. In reality, there would be a slight delay between the responses of the fuel and air flowrate governed by a separate combustion air and/or oxygen

controller. Figure 141 illustrates the controller's ability to output the correct feedwater mass flow to track the steam extraction flowrate. This demonstrates success of the steam mass balance.

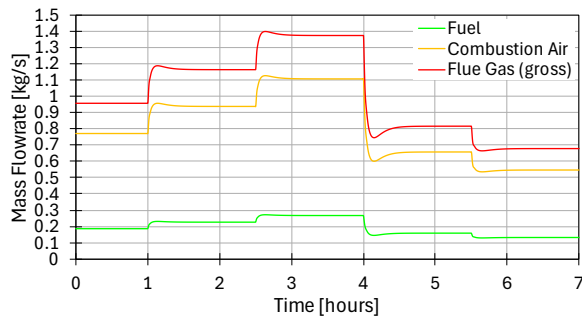


Figure 140: Fuel & gas mass flowrates

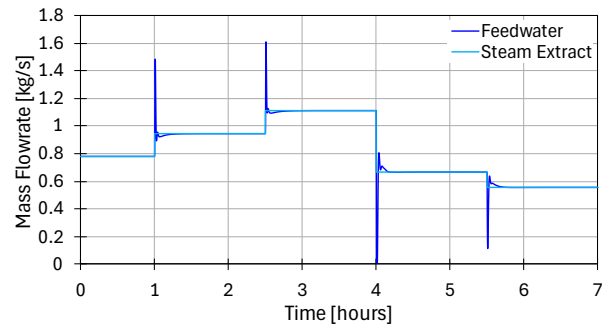


Figure 141: Feedwater flowrate referenced to steam extraction flowrate

The responses of all other process values will follow accordingly. The coupled fuel and air mass flowrate increase results in increased flue gas flowrate, ultimately increasing the heat transfer rate in the air heater. The air is therefore heated to a higher temperature as seen in Figure 142. Similarly, the heated feedwater temperature at the economiser exit increases and decreases accordingly albeit by very small margins as seen in Figure 143.

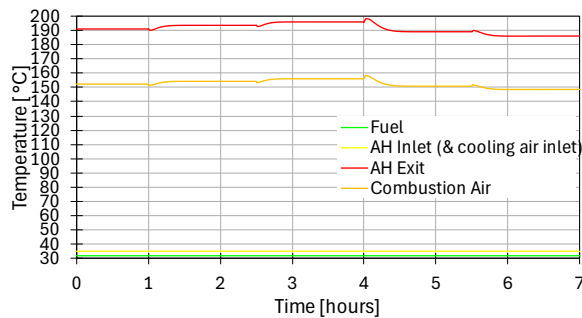


Figure 142: Fuel & air temperatures

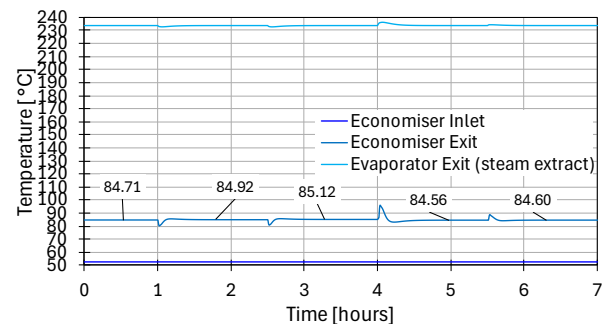


Figure 143: Water temperatures

The flue gas temperatures upstream of the furnace exit are illustrated in Figure 144 while the flue gas temperatures at the exit of each of the heat exchangers can be seen in Figure 145. Note that the “Rad extract” and “Conv extract” curves in Figure 144 are the temperatures at the flue gas nodes where the radiation is extracted via the *Fluid Radiation* component and where the furnace convection is extracted from the *CHT* component respectively.

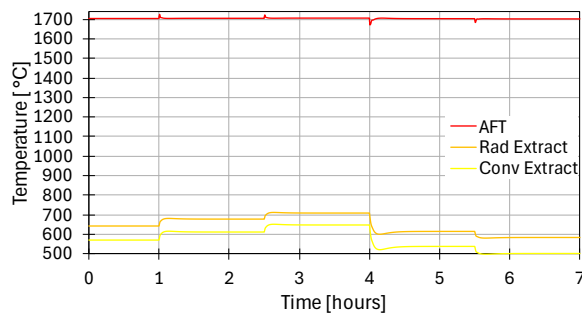


Figure 144: Flue gas temperatures upstream of furnace exit

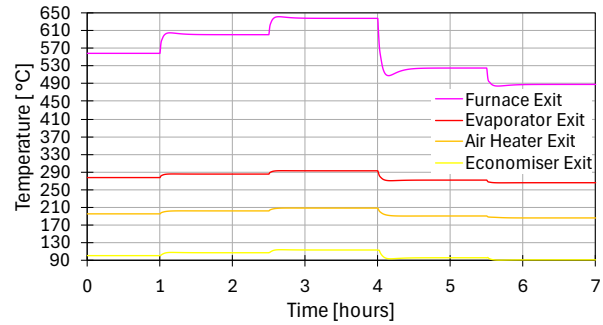


Figure 145: Flue gas temperatures

The AFT shows some small spikes before settling again. This is because the AFT is a function of the combustion calculations based on the fuel and air compositions. So the increased fuel flowrate to satisfy an increased steam extraction will naturally increase the combustion and radiation energy as can be seen in Figure 146 but does not influence the adiabatic flame temperature as seen by the constant line in Figure 144. This contrasts with Figure 134 where the change in fuel moisture (and composition) shows a very pronounced effect on the AFT. This illustrates the large influence of fuel quality on the boiler's performance.

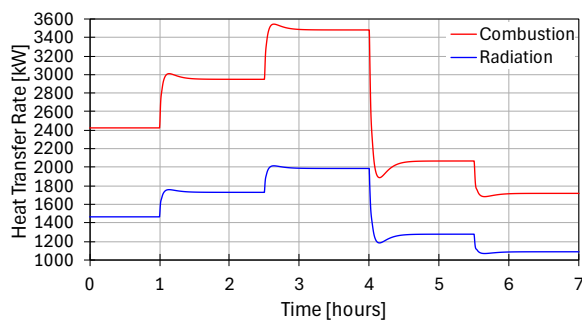


Figure 146: Combustion & radiation heat transfer rates

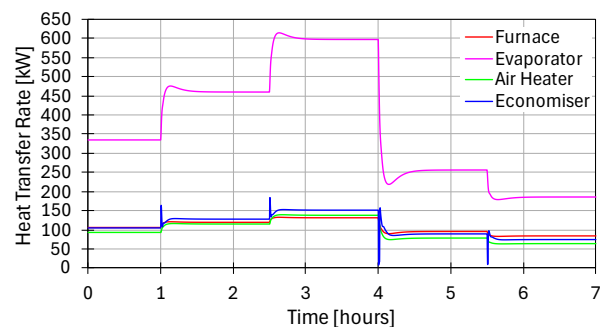


Figure 147: Convection heat transfer rates

The effect of the thermal inertia of the solid steel materials can be observed by a comparison between the wall temperatures at the inner diameter (ID) and outer diameter (OD) for the waterwall tubes in Figure 148 and of the fire-tubes of the evaporator plotted in Figure 149.

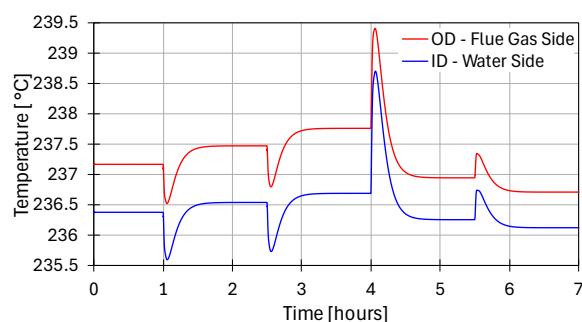


Figure 148: Furnace tube wall temperatures

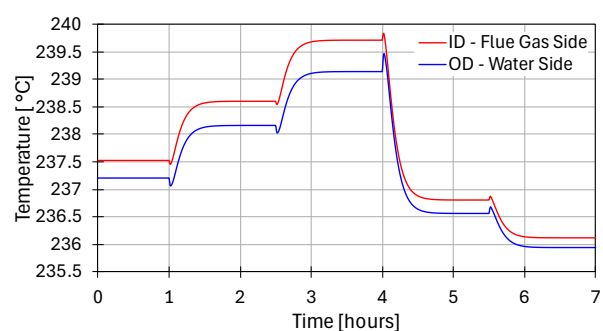


Figure 149: Fire-tube wall temperatures

The temperature difference between the outer and inner wall, although small, demonstrates the dynamic behaviour more accurately as a result of including the thermal inertia of the steel tubes.

6. Conclusions and Future Work

6.1 Summary & Conclusions

The literature review identified gaps in published work on the topic of dynamic modelling of boilers and highlighted the industry relevance for the development of dynamic boiler models to address the ever-changing energy landscape and requirement for flexibility in dispatchable power. The description of the TORRIPAC boiler was presented aided with a process flow diagram. The heat transfer mechanisms of both fire-tube and water-tube boilers were presented to provide insights into the hybrid water-tube—fire-tube boiler under investigation in the present research. Various boiler modelling methodologies employed in previous studies were also presented including a dynamic Flownex model of a three-pass firetube boiler.

The heat transfer theory chapter presented the thermodynamic principles necessary for modelling various heat exchangers in the context of the present work. This theory included Nusselt Number correlations for forced convection through finned-tube heat exchangers as needed for the air heater and economiser Flownex models of the TORRIPAC, natural convection over horizontal cylinders needed for the fire-tube evaporator model as well as the relevant ϵ -NTU correlations. Brief theory on combustion and radiative heat transfer applicable to furnace modelling was also presented.

The Flownex theory section briefly introduced the various software features and the application of common tools such as embedded excel workbooks, scripts, etc. that would be necessary in the context of the present work. Thereafter, brief summaries of the theory behind the common flow components used in the present work was presented. These included *Nodes*, *Boundary Conditions*, *Two-Phase Tanks*, and *Composite Heat Transfer* components to name a few.

The model development and verification chapter (Chapter 4) showed the natural progression of the models from analytical Mathcad standalone models to the numerical Flownex models with verification and increased confidence at each step.

The analytical Mathcad implementation employed the relevant heat transfer correlations and ϵ -NTU methods outlined in the theory chapter, followed by verification with the MCR data in section 4.1. Section 4.2 demonstrated the numerical Flownex implementation and verification with both the analytical and MCR data with summary pairwise comparison tables presented at the end of the section for each standalone heat exchanger model. The pairwise comparison tables showed the progression and natural increase in confidence with each step.

Key results showed agreement between the standalone Flownex 2 models with MCR data for the economiser and air heater of 3.81% and 0.70% respectively. These errors are acceptable with the

context of the unknown fuel composition used to generate the MCR data. Consequently, the assumed flue gas composition employed in the verification of the standalone models are not necessarily accurate.

Although the Flownex evaporator model could not be verified against MCR data due to the lack of a furnace exit temperature, there was agreement between the Flownex 1 and analytical Mathcad model to within 0.30% on the heat transfer rate with Table 13 demonstrating the implications of increased flue gas temperatures making the approximation of flue gas properties with air less accurate.

The methods for including the solid steel thermal inertia were presented for the economiser, air heater, furnace water-walls as well as the firetube evaporator shell, staybars and firetubes.

The steady state model calibration with fine tuning using site data in section 4.4 presented a thorough methodology for calibrating the integrated Flownex model with the limited site measurement data that was available. Calibration step one demonstrated the process of tuning the feedwater pump exit condition to target the site data main steam pressure in the *Two-Phase Tank*. Step 2 tuned the air mass flow with a fixed fuel flow, targeting the oxygen volume fraction (5.67%) at the evaporator exit as per the site data leading to a calibrated air-fuel-ratio of 0.241. Step 3 then tuned the fuel flowrate down to 0.185 kg/s to drive down the *Two-Phase Tank* energy source to within 1% of the combustion heat.

Results from select cases of the fine-tuning applied through parametric studies in step 4 were presented. The effects of fluid absorptivity, cooling air split ratios and localised furnace convection effects were illustrated. The best calibration point resulted from fined tune parameters as follows: inclusion of localised furnace convection at the furnace nose and exit plane using a convective heat transfer coefficient of 25 W/m²K; absorptivity value at the lower extreme of 0.6; and a cooling air split ratio of 25%.

A pairwise comparison giving ranks to the confidence of each of the site temperature measurements was employed to linearise the errors to a single error value per C-factor. It was then determined that the best calibration point with the lowest normalised error of 1.537% occurred for a C-factor of 1.2. The model temperatures at the calibrated C-factor of 1.2 were then plotted on a Box and Whisker plot to illustrate where they fall within the standard deviation of the measurements taken over the one-hour testing period. The best fine-tuned calibration point was used as the basis to initialise transient studies.

The controller development chapter introduced the existing TORRIPAC control philosophy followed by the presentation of the methodology for developing a simplified two-loop control system using simple analogue PID controller components from the Flownex DCS library. The pressure controller

with feedback to the fuel flow was heuristically verified with a response time of 17 minutes between 250 kPa set-point changes. The level controller was heuristically verified with a response time of 13 minutes between 10 mm set-point changes. The time step sensitivity between a 1 second and 5 second case showed numerical instability at the larger time step (5s) for the same controller values on the pressure controller. This required the pressure controller to be redesigned using a 5 second time step size to increase the computational efficiency of the transient solver. The time step sensitivity on the level controller did not show any variations between the 1 second and 5 second time step test cases.

The controller tuning methodology in the present work used a trial-and-error approach in search of stable responses, but future studies should focus on methodologies to determine required gains and controller limits for optimised response for specific boiler operational scenarios.

The thermal inertia studies investigated through transient runs of the low inertia and full inertia model configurations demonstrated the inherent lag in response by virtue of including accurate thermal masses for the solid steel materials. This can serve as the starting point for future studies where the lag times can be quantified and form as valuable insights for controller designers for optimising time delays for controller triggers and inform the selection of optimum measurement devices and other control equipment.

The transient studies with active control presented the results for two common transient boiler operational scenarios: fuel moisture variability and steam demand fluctuations. Although the responses of these transient runs cannot be validated due to lack of transient measurement data, the results are in line with what is expected. For example, the boiler steam pressure dropping for an increase in fuel moisture followed by the active pressure controller successfully increasing the fuel flow to restore the boiler pressure back to its set-point is exactly what is expected to happen in a real-life operational scenario and is also in line with the findings of P. Klein et al. [1].

Conclusions on the actual response times however can only be discussed in reference to the specific tuned controller values used in this work. The actual expected response time would need to come from a controlled transient site test which can be investigated as an extension of the present research in future studies.

6.2 Project Success

The overall success of the present research is summarised through the attainment of the lower-level goals as follows.

The analytical models successfully integrated the information provided through workshop drawings and applicable heat transfer correlations, yielding results that were verified with MCR data generated by John Thompson through their PPSD implementation.

Further verification was realised through the numerical Flownex implementation of the analytical models yielding good agreement with both the analytical model results as well as the MCR data.

Although there were limitations on available site data, a diligent calibration and fine-tuning methodology through parametric investigations and a qualitative ranking system to normalise the associated measurement uncertainties proved to effectively reconcile conflicting data to fall in line with an overall calibration point. This methodology saw the Flownex model temperatures agreeing with the site measurement temperatures to within 1.537% at the eventual tuned C-factor of 1.2. The box and whisker plot further shows that 3 out of 5 temperatures agreed within the standard deviation of the measurements taken over the 1-hour period. The two model outlier temperatures were the FET and economiser flue gas exit temperature. The reasons for these two outliers have been outlined in section 4.4.

The results of the uncontrolled transient simulations showed the realistic thermal inertia configuration's pressure and level responses lagging the low inertia configuration thereby validating the importance of including all solid steel thermal masses in transient simulations.

The goal of proving the ability of Flownex to emulate active control was realised through implementing the tuned dual pressure-level PID control system. The pressure controller successfully adjusted the fuel flow based on sensed changes in the evaporator pressure due to fuel quality and steam demand disturbances. The level-controller successfully sensed level fluctuations due to steam demand changes and its output adjusted the feedwater flow accordingly. Although there is no transient data to validate the active control, the TORRIPAC's response shows similar behaviour to the transient fire-tube Flownex model with active control results as presented in Figure 12 of section 2.6.4 from the study of P. Klein et al [1].

With the sequential success of each of the above lower-level goals, the main objective of developing a robust, fully integrated dynamic whole boiler model for the 4 T/hr TORRIPAC unit complete with active control loops has been achieved.

The thorough methodology employed has yielded a working model which is ready for use. Even in the absence of transient site data, the model can be used as is by John Thompson to investigate other transient operational scenarios and fine tune the model with new data sets and additional control loops to optimise the existing control philosophy. With further refinement and validation, John Thompson can structure the model to act as a TORRIPAC simulator complete with robust user controls and a human machine interface (HMI) within the Flownex simulation environment.

6.3 Recommendations for Future Work

Future work could include the following:

- Different thermodynamic correlations can be investigated to characterise the heat transfer in the evaporator by testing nucleate boiling correlations.
- Detailed uncertainty analysis on site data measurements.
- Validating convection effects and other complex phenomena inside the furnace through high fidelity 3-dimensional CFD modelling. This can better inform the furnace calibration of the Flownex model.
- Conduct controlled transient testing on the TORRIPAC boiler to record data which can validate the dynamic Flownex model. This could include transient scenarios such as, boiler start-up, shut down, load changes and varying the fuel quality (i.e. moisture content).
- Inclusion of boiler ancillary equipment such as pumps, fans, and valves coupled with models for the existing TORRIPAC control equipment including variable speed drives for the turbomachinery and the relevant sensors, triggers and alarms. Accurately capturing all equipment and the existing TORRIPAC control philosophy can then form the basis for control optimisation studies with the added benefit of creating a robust integrated simulator which can be used to train John Thompson boiler operators.
- Inclusion of applicable secondary loss factors to the furnace water-wall pipe components to characterise the natural circulation flowrate. Validation of this natural circulation flowrate with CFD could follow.

7. List of References

- [1] P. Klein, M. Potgieter, and B. A. Ferreira, "Systems based CFD modelling of package steam boilers." 2021. [Online]. Available: <http://hdl.handle.net/10204/12411>
- [2] F. J. Gutiérrez Ortiz, "Modeling of fire-tube boilers," *Appl. Therm. Eng.*, vol. 31, no. 16, pp. 3463–3478, 2011, doi: 10.1016/j.applthermaleng.2011.07.001.
- [3] "Horizontal Return Tubular Boiler – HVAC Troubleshooting." [Online]. Available: <http://www.hvacspecialists.info/boilers/horizontal-return-tubular-boiler.html>
- [4] D. Aguilar Vizcarra, D. Esenarro, and C. Rodriguez, "Three steps mixed (Fire tube–water tube) vertical boiler to optimize thermal performance," *Fluids*, vol. 6, no. 3, 2021, doi: 10.3390/fluids6030093.
- [5] R. Laubscher and P. Rousseau, "CFD study of pulverized coal-fired boiler evaporator and radiant superheaters at varying loads," *Appl. Therm. Eng.*, vol. 160, no. April, p. 114057, 2019, doi: 10.1016/j.applthermaleng.2019.114057.
- [6] R. Laubscher and S. van der Merwe, "Heat transfer modelling of semi-suspension biomass fired industrial watertube boiler at full- and part-load using CFD," *Therm. Sci. Eng. Prog.*, vol. 25, no. March, p. 100969, 2021, doi: 10.1016/j.tsep.2021.100969.
- [7] P. Rousseau, R. Laubscher, and B. T. Rawlins, "Heat Transfer Analysis Using Thermofluid Network Models for Industrial Biomass and Utility Scale Coal-Fired Boilers," *Energies*, vol. 16, no. 4. 2023. doi: 10.3390/en16041741.
- [8] J. Kuronen, M. Hotti, and S. Tuuri, "Modelling and Dynamic Simulation of Cyclically Operated Pulverized Coal-Fired Power Plant," *Proc. 9th EUROSIM Congr. Model. Simulation, EUROSIM 2016, 57th SIMS Conf. Simul. Model. SIMS 2016*, vol. 142, pp. 122–128, 2018, doi: 10.3384/ecp17142122.
- [9] F. Alobaid, N. Mertens, R. Starkloff, T. Lanz, C. Heinze, and B. Eppel, "Progress in dynamic simulation of thermal power plants," *Prog. Energy Combust. Sci.*, vol. 59, pp. 79–162, 2017, doi: 10.1016/j.pecs.2016.11.001.
- [10] M. Tognoli, B. Najafi, and F. Rinaldi, "Dynamic modelling and optimal sizing of industrial fire-tube boilers for various demand profiles," *Appl. Therm. Eng.*, vol. 132, pp. 341–351, 2018, doi: 10.1016/j.applthermaleng.2017.12.082.
- [11] V. D. I. H. Atlas, *VDI Heat Atlas*. 2010. doi: 10.1007/978-3-540-77877-6.
- [12] Y. A. Cengel and A. J. Ghajar, *Heat and Mass Transfer, Fundamentals & Applications*, 5th ed. New York, USA: McGraw-Hill, 2015.
- [13] L. Zeghadnia, J. Loup, and B. Achour, "Explicit solutions for turbulent flow friction factor : A review , assessment and approaches classification," *Ain Shams Eng. J.*, vol. 10, no. 1, pp. 243–252, 2019, doi: 10.1016/j.asej.2018.10.007.
- [14] Gregory Nellis and Sanford Klein, *Heat Transfer*, 1st ed. Cambridge, U.K.: Cambridge University Press, 2019.
- [15] Flownex, "Flownex Theory Manual," no. December, p. 165, 2022, [Online]. Available: www.flownex.com

- [16] G. U. Manual, "General User Manual," no. December, 2012.
- [17] Flownex SE, "Control Library Manual," no. December, 222AD.
- [18] Flownex SE, "Flownex Library Manual," no. August, 2022.
- [19] ACTOM, "CONTROL PHILOSOPHY TORRIPAC (John Thompson internal report)." 2019.

Appendix A. Economiser Verification Details

☞ Reference: C:\Users\pmb87\Desktop\Coolprop Fluid Properties rev 1.xmcd(R)

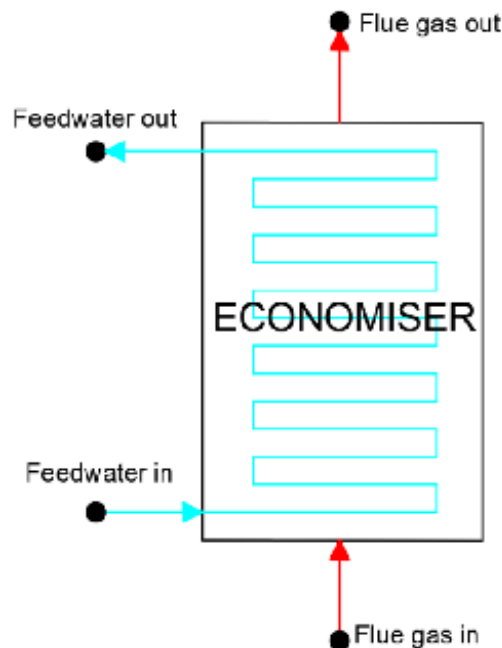
ECONOMISER VERIFICATION (Analytical vs. MCR vs. Flownex)

E-NTU Cross Flow (C_{min} Mixed - C_{max} Unmixed)

Water runs through the tubes and fluegas flows between the finned tubes in a crossflow regime. There are 6 tube rows (6 water streams) linking the inlet and outlet water headers. Each tube line makes 14 passes from the inlet header (top) through the economiser before entering the outlet header (bottom).

Since the fluegas will undergo much turbulence (i.e. mixing) due to the obstructions (finned-tubes) in its flow path and the fact that the minimum heat capacity rate is that of the fluegas, the E-NTU correlation for cross flow with C_{min} mixed and C_{max} unmixed is used in this model as shown below.

$$\epsilon_{C_{min}.mixed.C_{max}.unmixed} = 1 - e^{-\left(\frac{1}{C}\right) \left[1 - e^{(-C \cdot NTU)}\right]}$$



Reference List:

- [1] TO400 TORRIPAC BOILER PROCESS FLOW DIAGRAM, 68756-03, rev 0.
- [2] THO400-40 ECONOMISER ECONOMISER ASSEMBLY, 68759-05, rev B.
- [3] VDI Heat Atlas, (pg 1273-1277), M1 Heat Transfer to Finned Tubes
- [4] Heat and Mass Transfer, Fundamentals & Applications, Yunus A. Cengel, Afshin J. Ghajar.
- [5] Zeghadnia, J. Loup, and B. Achour, "Explicit solutions for turbulent flow friction factor 'a0: A review , assessment and approaches classification," *Ain Shams Eng. J.*, vol. 10, no. 1, pp. 243252, 2019, doi: 10.1016/j.asej.2018.10.007
- [6] Heat Transfer in Single and Multiphase Systems, Greg F. Naterer, pg 491

P_{ATM} := 101.325kPa

[Atmospheric pressure.]

John Thompson MCR Design Data [1]

This section summarises the Maximum Capacity Rating design data as per John Thompsons designs [1].

It includes mass flowrates as well as inlet and outlet temperatures and pressures for both streams (water and flue gas).

A simple heat transfer rate calculation $\dot{Q}_{dot} = \dot{m}_{dot} \cdot c_p \cdot \Delta T$ is applied to determine the heat transfer rate as per the MCR data. This value will then be compared to the heat transfer rate of the analytical model and Flownex model as a verification.

MCR Design Data

Flue gas Side

Mass flowrate:	$\dot{m}_{dot,fg,JT} := 6984 \frac{\text{kg}}{\text{hr}}$	[1]
Inlet temperature:	$T_{fg,in,JT} := 230^\circ\text{C}$	[1]
Outlet temperature:	$T_{fg,out,JT} := 154^\circ\text{C}$	[1]
Inlet pressure:	$P_{fg,in,JT} := 808\text{kPa} + P_{ATM} = 909.325\text{ kPa}$	[1]
Outlet pressure:	$P_{fg,out,JT} := 907\text{kPa} + P_{ATM} = 1008.325\text{ kPa}$	[1]
	$P_{fg,out,JT} := 909.324\text{kPa}$	[Flownex result]

Water Side

Mass flowrate:	$\dot{m}_{dot,H2O,JT} := 4000 \frac{\text{kg}}{\text{hr}}$	[1]
Inlet temperature:	$T_{H2O,in,JT} := 80^\circ\text{C}$	[1]
Outlet temperature:	$T_{H2O,out,JT} := 118^\circ\text{C}$	[1]
Inlet pressure:	$P_{H2O,in,JT} := 3236\text{kPa} + P_{ATM} = 3337.325\text{ kPa}$	[1]
Outlet pressure:	$P_{H2O,out,JT} := 3183\text{kPa} + P_{ATM} = 3284.325\text{ kPa}$	[1]
	$P_{H2O,out,JT} := 3336.540\text{kPa}$	[Flownex result]

Note that for consistency in the verification, that the flownex computed outlet pressures were used for the MCR heat transfer calculation. In reality the fluegas actually as a pressure rise from inlet to outlet through the economiser due to the ID fan and stack effect. This pressure difference has insignificant effects on the heat transfer.

MCR Heat Transfer Rate (using water side conditions)

$$\text{Average water temperature: } T_{H2O,avg,JT} := \frac{T_{H2O,in,JT} + T_{H2O,out,JT}}{2} = 99.^\circ\text{C}$$

$$\text{Average water pressure: } P_{H2O,avg,JT} := \frac{P_{H2O,in,JT} + P_{H2O,out,JT}}{2} = 3336.932\text{ kPa}$$

Water specific heat capacity evaluated at average P&T:

$$c_{PH2O,JT} := C_{pfluid}(\text{"water"}, P_{H2O,avg,JT}, T_{H2O,avg,JT}, "", "", "", "") = 4.207 \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

$$\text{Heat Transfer Rate: } \dot{Q}_{dot,MCR} := \dot{m}_{dot,H2O,JT} \cdot c_{PH2O,JT} \cdot (T_{H2O,out,JT} - T_{H2O,in,JT}) = 177.636\text{ kW}$$

ECONOMISER - Analytical Model

- Finned tube heat exchanger
- External fins "H-type"
- Water inlet header at the top of the assembly
- Water outlet header at the bottom of the assembly
- Flue gas inlet at the bottom of the assembly
- Flue gas exit at the top of the assembly
- Tubeside fluid - subcooled water
- Finside fluid - flue gas
- 6 tube rows (parallel streams)
- 14 tube passes
- Single finside (flue gas) pass

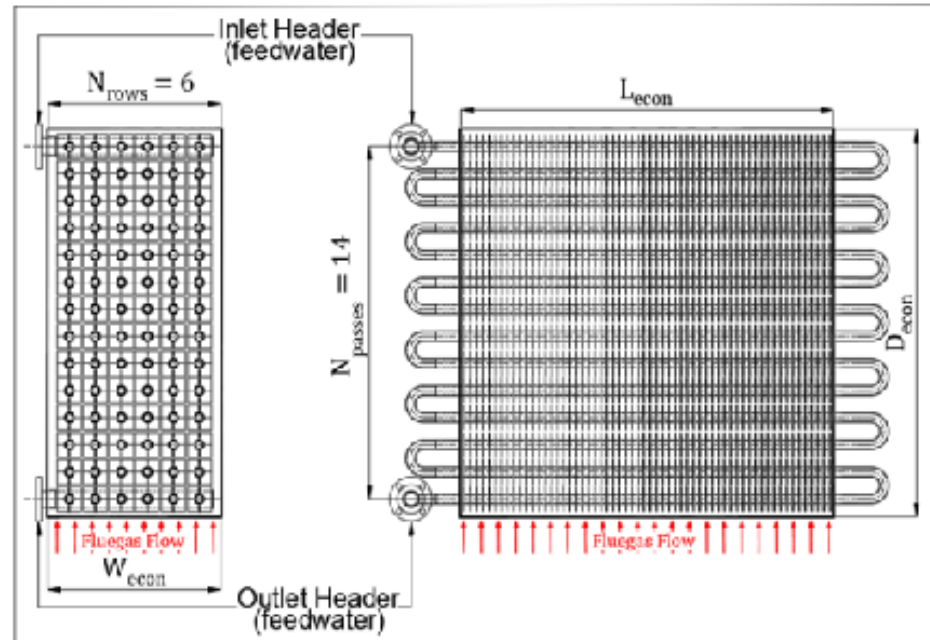


Figure 1: Economiser General Arrangement

GEOMETRIC INPUTS

Tube Arrangement

Number of rows (branches teeing off from headers): $N_{\text{rows}} := 6$ [2]

Number of passes (this is the same for each tube row): $N_{\text{passes}} := 14$ [2]

Number of parallel circuits: $N_{\text{par.circ}} := 1$ [2]

The total number of fluegas wetted tubes is therefore: $N_{\text{tubes}} := N_{\text{rows}} \cdot N_{\text{passes}} = 84$

Overall Economiser Dimensions

Economiser depth (flue gas pass length): $D_{\text{econ}} := 1.5\text{m}$

Width of heat exchanger (inside width of economiser casing): $W_{\text{econ}} := 630\text{mm}$ $w_{\text{HX}} := 697\text{mm}$ [2]

Height of heat exchanger (equal to single tube pass length): $H_{\text{econ}} := 1.5\text{m}$ [2]

Cross-sectional area of economiser (perp to fg flow direction): $A_{\text{econ}} := W_{\text{econ}} \cdot H_{\text{econ}} = 0.945\text{m}^2$

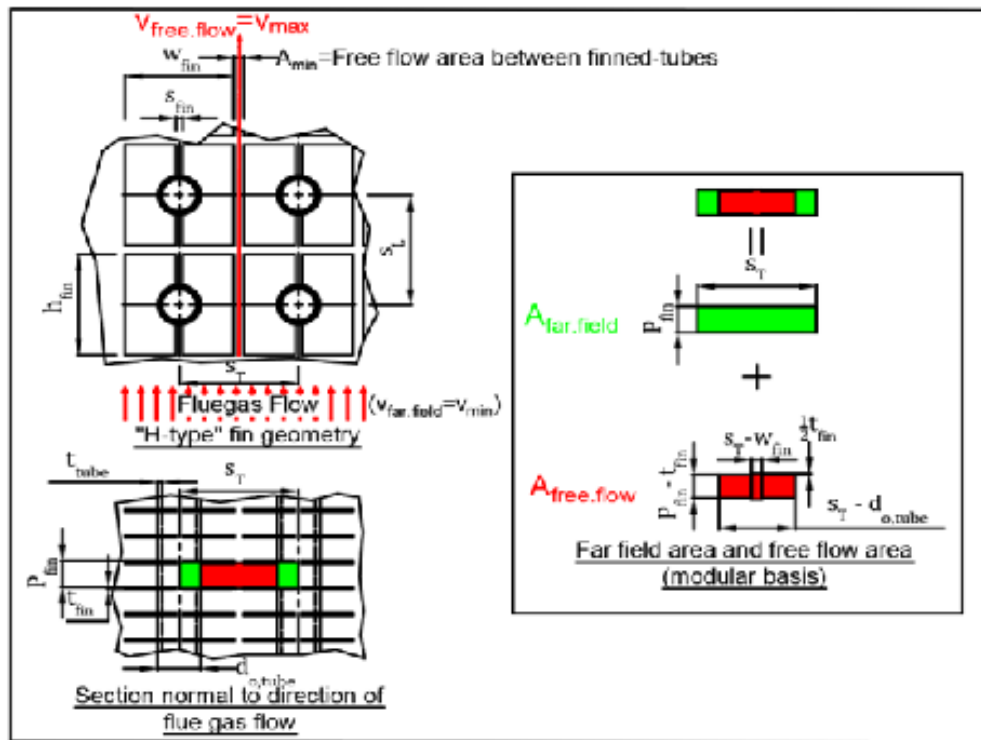


Figure 2: "H-type" fin geometry, far-field and free-flow areas and velocities (modular basis)

Fin Geometry:

Fin Height (parallel to fluegas flow direction):

$$h_{fin} := 96\text{mm} \quad [2]$$

Fin Width (perpendicular to fluegas flow direction):

$$w_{fin} := 96\text{mm} \quad [2]$$

Fin Slit (perpendicular to flow direction):

$$s_{fin} := 6\text{mm} \quad [2]$$

Fin Thickness:

$$t_{fin} := 2.8\text{mm} \quad [2]$$

Fin Arrangement (per tube):

Fin Pitch (distance between successive fins along a single tube):

$$p_{fin} := 25\text{mm} \quad [2]$$

Number of fins (for a single tube):

$$N_{fpt} := 60 \quad [2]$$

Tube Geometry:

Tube OD:

$$d_{o,tube} := 38.1\text{mm} \quad [2]$$

Tube Thickness:

$$t_{tube} := 4.06\text{mm} \quad [2]$$

Tube ID:

$$d_{i,tube} := d_{o,tube} - 2 \cdot t_{tube} = 29.98\text{mm} \quad [2]$$

Tube Length (gas pass width):

$$L_{tube} := 1500\text{mm} \quad [2]$$

Tube Arrangement:

Transverse Pitch (tube spacing perpendicular to fluegas flow direction):

$$s_T := 105\text{mm} \quad [2]$$

Longitudinal Pitch (tube spacing parallel to fluegas flow direction):

$$s_L := 105\text{mm} \quad [2]$$

H-Type Fin Geometry - Detailed:

Equation of circle (outer diameter of tube):

$$x^2 + y^2 = \left(\frac{d_{o.tube}}{2}\right)^2 \quad x_{fin} := \frac{s_{fin}}{2} = 3 \cdot \text{mm} \quad y_{fin} := \sqrt{\left(\frac{d_{o.tube}}{2}\right)^2 - x_{fin}^2} = 18.812 \cdot \text{mm}$$

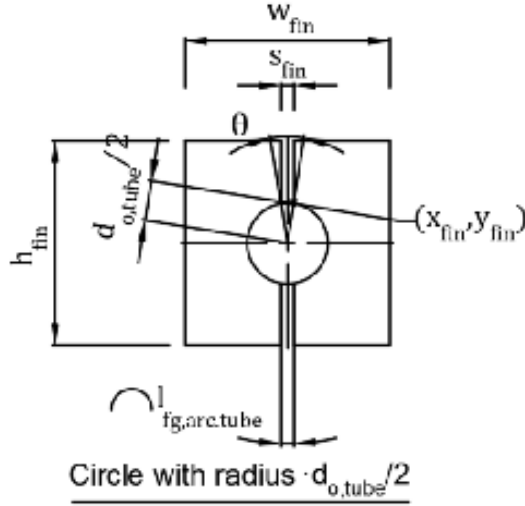


Figure 3: "H-type" fin geometry - detailed, including arc-length

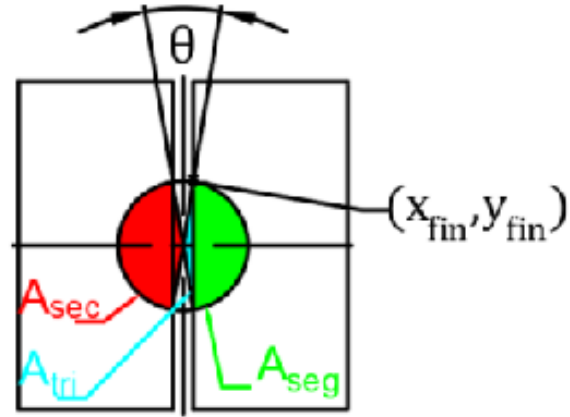


Figure 4: "H-type" area of circle segment needed for fin surface area calculation

Angle between adjacent intersections between fin-tube interface of "H-Type" fin subtended from the centre of the tube:

$$\theta := 2 \cdot \text{atan}\left(\frac{x_{fin}}{y_{fin}}\right) = 0.316 \cdot \text{rad} \quad \theta = 18.121 \cdot \text{deg}$$

Arc length of tube in contact with the fluegas on one side of the tube (inbetween the fin space, s_fin):

$$l_{fg.arc.tube} := \left(\theta \cdot \frac{d_{o.tube}}{2}\right) = 0.006 \cdot \text{m}$$

Area of red circle sector:

$$A_{sec} := \pi \cdot \left(\frac{d_{o.tube}}{2}\right)^2 \cdot \left(\frac{\pi - \theta}{2\pi}\right) = 0.001 \cdot \text{m}^2$$

Area of blue triangle piece of the circle sector: $A_{tri} := \frac{1}{2} \cdot x_{fin} \cdot (2 \cdot y_{fin}) = 0 \cdot \text{m}^2$

Area of green circle segment:

$$A_{seg} := A_{sec} - A_{tri} = 456.22 \cdot \text{mm}^2$$

Cross sectional flow area obstructed by fins and tubes

$$A_{fin.obstr} := (w_{fin} \cdot t_{fin}) \cdot N_{fpt} \cdot N_{rows} = 0.097 \cdot \text{m}^2$$

$$A_{tube.obstr} := [d_{o.tube} \cdot (H_{econ} - N_{fpt} \cdot t_{fin})] \cdot N_{rows} = 0.304 \cdot \text{m}^2$$

$$A_{obstr} := A_{fin.obstr} + A_{tube.obstr} = 0.401 \cdot \text{m}^2$$

HEAT TRANSFER SURFACE AREAS:**Inside surface area of tubes (full water-wetted surface area of tubes):**

$$SA_{i.tube} := \pi \cdot d_{i.tube} \cdot L_{tube} = 0.141 \text{ m}^2$$

$$SA_{i.tube.tot} := N_{tubes} \cdot SA_{i.tube} = 11.867 \text{ m}^2$$

Outside surface area of tubes (full fluegas-wetted surface area of tube material only):

$$SA_{o.tube} := \pi \cdot d_{o.tube} \cdot (L_{tube} - N_{fpt} \cdot t_{fin}) + (2 \cdot l_{fg.arc.tube} \cdot t_{fin}) \cdot N_{fpt} = 0.161 \text{ m}^2$$

$$SA_{o.tube.tot} := N_{tubes} \cdot SA_{o.tube} = 13.562 \text{ m}^2$$

Outside surface area of fins (full fluegas-wetted surface area of fin material only):

$$SA_{fin} := (2 \cdot h_{fin} \cdot w_{fin} + 2 \cdot h_{fin} \cdot t_{fin} + 2 \cdot w_{fin} \cdot t_{fin}) - [2 \cdot (s_{fin} \cdot h_{fin} + 2 \cdot A_{seg}) + 2 \cdot t_{fin} \cdot s_{fin}] \dots = 0.017 \text{ m}^2$$

$$+ 2 \cdot (h_{fin} - 2 \cdot y_{fin}) \cdot t_{fin}$$

$$SA_{o.fin} := N_{fpt} \cdot SA_{fin} = 1.009 \text{ m}^2$$

$$SA_{o.fin.tot} := N_{tubes} \cdot SA_{o.fin} = 84.791 \text{ m}^2$$

Outside surface area of bare tubes (i.e. if there were no fins):

$$SA_{o.tube.bare} := \pi \cdot d_{o.tube} \cdot L_{tube} = 0.18 \text{ m}^2$$

$$SA_{o.tube.bare.tot} := N_{tubes} \cdot SA_{o.tube.bare} = 15.082 \text{ m}^2$$

Total Internal Flow (water) heat transfer area per tube:

$$SA_{i.tube} = 0.141 \text{ m}^2$$

Total Internal Flow (water) heat transfer area for all tubes:

$$SA_{i.tube.tot} = 11.867 \text{ m}^2$$

Total External Cross Flow (fluegas) heat transfer area per finned-tube:

$$SA_{o.fin.tube} := SA_{o.fin} + SA_{o.tube} = 1.171 \text{ m}^2$$

Total External Cross Flow (fluegas) heat transfer area for all finned-tubes:

$$SA_{o.fin.tube.tot} := SA_{o.fin.tot} + SA_{o.tube.tot} = 98.354 \text{ m}^2$$

$$SA_{o.fin.tube} \cdot N_{tubes} = 98.354 \text{ m}^2$$

Ratio of finned-tube heat transfer surface area to bare tube heat transfer surface area:

$$ROS := \frac{SA_{o.fin.tube}}{SA_{o.tube.bare}} = 6.521$$

$$\frac{SA_{o.fin.tube.tot}}{SA_{o.tube.bare.tot}} = 6.521$$

Ratio of external to internal surface area:

$$Area_{Ratio} := \frac{SA_{o.fin.tube.tot}}{SA_{i.tube.tot}} = 8.288$$

Straightened Cross-section of Flow (SCOF) Calculation

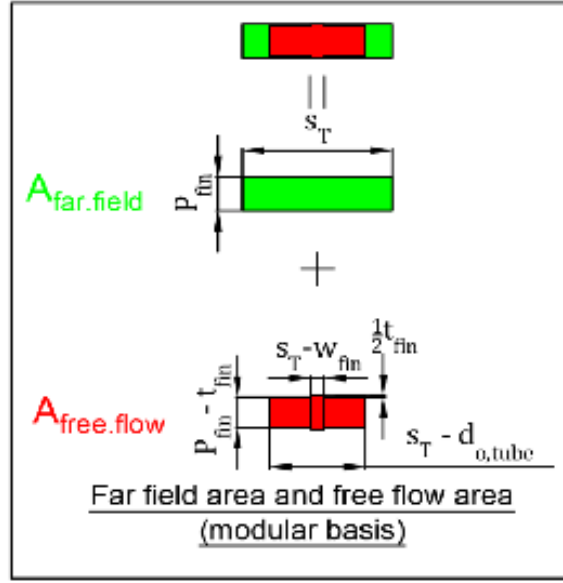


Figure 5: Far-field and free-flow areas (modular basis)

Far field area (modular approach):

$$A_{far.field.mod} := s_T \cdot P_{fin} = 0.003 \text{ m}^2$$

$$A_{far.field.tot} := A_{econ} = 0.945 \text{ m}^2$$

Free flow area (modular approach):

$$A_{free.flow.mod} := (s_T - d_{o,tube}) \cdot (P_{fin} - t_{fin}) + (s_T - w_{fin}) \cdot t_{fin} = 0.002 \text{ m}^2$$

$$A_{free.flow.tot} := A_{econ} - A_{obstr} = 0.544 \text{ m}^2$$

Straightened cross section of flow (modular approach):

$$SCOF_{mod} := \frac{A_{far.field.mod}}{A_{free.flow.mod}} = 1.738$$

$$\text{Sense check: } SCOF_{tot} := \frac{A_{far.field.tot}}{A_{free.flow.tot}} = 1.738$$

Total fluid volume (fluegas inside of economiser):

$$V_{econ} := A_{econ} \cdot D_{econ} = 1.417 \cdot \text{m}^3$$

$$V_{tubes} := \frac{\pi}{4} \cdot d_{o,tube}^2 \cdot H_{econ} \cdot N_{tubes} = 0.144 \cdot \text{m}^3$$

$$V_{fins} := N_{fpt} \cdot N_{tubes} \cdot [(h_{fin} \cdot w_{fin} \cdot t_{fin}) - (s_{fin} \cdot h_{fin} \cdot t_{fin}) - (2 \cdot A_{seg} \cdot t_{fin})] = 0.109 \cdot \text{m}^3$$

$$V_{fluid} := V_{econ} - V_{fins} - V_{tubes} = 1.165 \cdot \text{m}^3$$

Total finned-tube material mass:

$$V_{tube.mat} := \frac{\pi}{4} \cdot (d_{o,tube}^2 - d_{i,tube}^2) \cdot L_{tube} \cdot N_{tubes} = 0.055 \cdot \text{m}^3$$

$$V_{fin.mat} := V_{fins} = 0.109 \cdot \text{m}^3$$

$$V_{fin.tube.mat} := V_{tube.mat} + V_{fin.mat} = 0.164 \cdot \text{m}^3$$

$$\rho_{steel} := 7832 \frac{\text{kg}}{\text{m}^3}$$

$$m_{fin.tube.mat} := V_{fin.tube.mat} \cdot \rho_{steel} = 1282.548 \text{ kg}$$

Flue gas Side Inputs:

All inputs below, barring the outlet temperature have been explained in the beginning of this report.

Flue gas mass flowrate: $m_{\text{dot.fg.in}} := m_{\text{dot.fg.JT}} = 1.94 \frac{\text{kg}}{\text{s}}$

Inlet flue gas pressure (absolute): $P_{\text{fg.in}} := P_{\text{fg.in.JT}} = 909.325 \cdot \text{kPa}$

Outlet fluegas pressure (absolute): $P_{\text{fg.out}} := P_{\text{fg.out.JT}} = 909.324 \cdot \text{kPa}$

Inlet flue gas temperature: $T_{\text{fg.in}} := T_{\text{fg.in.JT}} = 230 \cdot ^\circ\text{C}$

Outlet fluegas temperature: $T_{\text{fg.out}} := 145.625 \cdot ^\circ\text{C}$ [Successive approximation with first guess as 154 degC (i.e. as per MCR data)]

Note: Red highlight indicates variables that are updated iteratively.

Note: Expressions with borders are inputs that can be changed when verifying/validating different operating points based on a different boiler load.

Fluegas Fluid Properties:

[Assuming properties similar to air.]

Fluegas Density evaluated at T_{fg.in} and P_{fg.in}:

$$\rho_{fg.in} := \rho_{fluid}("air", P_{fg.in}, T_{fg.in}, "", "", "") = 6.277 \frac{kg}{m^3}$$

Average fluegas-side Temperature and Pressure:

$$T_{fg.avg} := \frac{T_{fg.in} + T_{fg.out}}{2} = 187.813 \cdot ^\circ C \quad P_{fg.avg} := \frac{P_{fg.in} + P_{fg.out}}{2} = 909.324 \cdot kPa$$

Specific Heat Capacity:

$$c_{p,fg} := Cp_{fluid}("air", P_{fg.avg}, T_{fg.avg}, "", "", "", "") = 1027.525 \cdot \frac{J}{kg \cdot K}$$

Density:

$$\rho_{fg} := \rho_{fluid}("air", P_{fg.avg}, T_{fg.avg}, "", "", "") = 6.854 \frac{kg}{m^3}$$

Thermal Conductivity:

$$\lambda_{fg} := \lambda_{fluid}("air", P_{fg.avg}, T_{fg.avg}, "") = 0.038 \cdot \frac{W}{m \cdot K}$$

Dynamic Viscosity:

$$\mu_{fg} := \mu_{fluid}("air", P_{fg.avg}, T_{fg.avg}, "") = 0 \cdot \frac{kg}{m \cdot s}$$

Kinematic Viscosity:

$$\nu_{fg} := \frac{\mu_{fg}}{\rho_{fg}} = 0 \frac{m^2}{s}$$

Prandtl Number:

$$Pr_{fg} := Pr_{fluid}("air", P_{fg.avg}, T_{fg.avg}, "") = 0.7$$

Fluegas Heat Transfer Rate:

$$Q_{dot,fg} := m_{dot,fg.in} \cdot c_{p,fg} \cdot (T_{fg.in} - T_{fg.out}) = 168.193 \cdot kW$$

DETERMINING THE FLUEGAS HEAT TRANSFER COEFFICIENT FOR FINNED TUBES:

Methodology as per [3].

$$Nu = 0.22 \cdot Re_{d.o.tube}^{0.6} \cdot ROS^{-0.15} \cdot Pr_{fg}^{\frac{1}{3}}$$
, in which the Reynolds Number uses the tube OD as the characteristic length and uses the maximum velocity at the smallest cross-sectional area of flow, $Re_{d.o.tube} = \frac{v_{max} \cdot d_{o.tube}}{\nu} = \frac{v_{max} \cdot d_{o.tube} \cdot \rho}{\mu}$, where:

$v_{max} = SCOF \cdot v_{far.field} = \text{velocity, in, the, smallest, cross, section}$

Finally the flue gas heat transfer coefficient may be calculated as:

$$h_{fg} = \frac{Nu \cdot \lambda_{fg}}{d_{o.tube}}$$

Inlet velocity of fluegas into the economiser (i.e. far field velocity):

$$v_{far.field} := \frac{\dot{m}_{dot.fg.in}}{\rho_{fg.in} \cdot A_{econ}} = 0.3270584 \frac{m}{s}$$

Max velocity in the pipe bundle:

$$v_{max} := SCOF_{mod} \cdot v_{far.field} = 0.568 \frac{m}{s}$$

Reynolds Number:

$$Re_{d.o.tube} := \frac{v_{max} \cdot d_{o.tube}}{\nu_{fg}} = 5786.91$$

Fluegas Nusselt Number:

$$Nu := 0.22 \cdot Re_{d.o.tube}^{0.6} \cdot ROS^{-0.15} \cdot Pr_{fg}^{\frac{1}{3}} = 26.68$$

Fluegas Heat Transfer Coefficient:

$$h_{fin.side} := \frac{Nu \cdot \lambda_{fg}}{d_{o.tube}} = 26.351 \cdot \frac{W}{m^2 \cdot K}$$

Water Side Inputs:

All inputs below, barring the outlet temperature have been explained in the beginning of this report.

Inlet water mass flowrate: $m_{\text{dot.H2O.in}} := m_{\text{dot.H2O.JT}} = 1.111 \frac{\text{kg}}{\text{s}}$

Inlet water pressure (absolute): $P_{\text{H2O.in}} := P_{\text{H2O.in.JT}} = 3337.325 \cdot \text{kPa}$

Outlet water pressure (absolute): $P_{\text{H2O.out}} := P_{\text{H2O.out.JT}} = 3336.54 \cdot \text{kPa}$

Inlet water temperature: $T_{\text{H2O.in}} := T_{\text{H2O.in.JT}} = 80 \cdot ^\circ\text{C}$

Outlet water temperature: $T_{\text{H2O.out}} := 115.989 \cdot ^\circ\text{C}$ [Successive approximation with first guess as 118 degC (i.e. as per MCR data)]

Note: Blue highlight indicates variables that are updated iteratively.

Note: Expressions with borders are inputs that can be changed when verifying/validating different operating points based on a different boiler load.

Water Fluid Properties:**Average water-side Temperature and Pressure:**

$$T_{H2O.avg} := \frac{T_{H2O.in} + T_{H2O.out}}{2} = 97.995.^{\circ}\text{C} \quad P_{H2O.avg} := \frac{P_{H2O.in} + P_{H2O.out}}{2} = 3336.932.\text{kPa}$$

Water Specific Heat Capacity evaluated at T.avg and P.avg:

$$c_{p,H2O} := C_{p,fluid}(\text{"water"}, P_{H2O.avg}, T_{H2O.avg}, \text{"", ""}, \text{"", ""}) = 4206.061. \frac{\text{J}}{\text{kg} \cdot \text{K}}$$

Water density evaluated at T.avg and P.avg:

$$\rho_{H2O} := \rho_{fluid}(\text{"water"}, P_{H2O.avg}, T_{H2O.avg}, \text{"", ""}, \text{"", ""}) = 961.287 \frac{\text{kg}}{\text{m}^3}$$

Inlet water density:

$$\rho_{H2O.in} := \rho_{fluid}(\text{"water"}, P_{H2O.in}, T_{H2O.in}, \text{"", ""}, \text{"", ""}) = 973.236 \frac{\text{kg}}{\text{m}^3}$$

Thermal Conductivity:

$$\lambda_{H2O} := \lambda_{fluid}(\text{"water"}, P_{H2O.avg}, T_{H2O.avg}, \text{"", ""}) = 0.678 \text{ m} \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Dynamic Viscosity:

$$\mu_{H2O} := \mu_{fluid}(\text{"water"}, P_{H2O.avg}, T_{H2O.avg}, \text{"", ""}) = 0. \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Kinematic Viscosity:

$$\nu_{H2O} := \frac{\mu_{H2O}}{\rho_{H2O}} = 3.001 \times 10^{-7} \cdot \frac{\text{m}^2}{\text{s}}$$

Prandtl Number:

$$Pr_{H2O} := Pr_{fluid}(\text{"water"}, P_{H2O.avg}, T_{H2O.avg}, \text{"", ""}) = 1.789$$

Total water-side heat transfer rate:

$$\dot{Q}_{dot,H2O} := \dot{m}_{dot,H2O.in} \cdot c_{p,H2O} \cdot (T_{H2O.out} - T_{H2O.in}) = 168.191.\text{kW}$$

DETERMINING THE WATER HEAT TRANSFER COEFFICIENT FOR FLOW INSIDE TUBES (Gnielinski correlation [4]):

Total water mass flowrate:

$$\dot{m}_{\text{dot.H2O.in}} = 1.111 \frac{\text{kg}}{\text{s}}$$

Water mass flowrate per tube row:

$$\dot{m}_{\text{dot.row}} := \frac{\dot{m}_{\text{dot.H2O.in}}}{N_{\text{rows}}} = 0.185 \frac{\text{kg}}{\text{s}}$$

Water velocity per tube line:

$$v_{\text{row}} := \frac{\dot{m}_{\text{dot.row}}}{\rho_{\text{H2O.in}} \cdot \left(\frac{\pi}{4} \cdot d_{\text{i.tube}}^2 \right)} = 0.27 \frac{\text{m}}{\text{s}}$$

Water Reynolds Number for water flow inside a tube:

$$Re_{\text{d.i.tube}} := \frac{\rho_{\text{H2O}} \cdot v_{\text{row}} \cdot d_{\text{i.tube}}}{\mu_{\text{H2O}}} = 26926.294$$

Surface Roughness (sheet metal, new) as per [4]:

$$\varepsilon_{\text{steel}} := 100 \mu\text{m}$$

For sheet metal, new (max)

Smooth to rough correlation
Swamee and Jain, (1976)

[5]

Friction factor:

$$f := \left(\frac{1}{-2 \cdot \log \left(\frac{\frac{\varepsilon_{\text{steel}}}{d_{\text{i.tube}}}}{3.7} + \frac{5.74}{Re_{\text{d.i.tube}}^{0.9}} \right)} \right)^2 = 0.0313033$$

$$5 \cdot 10^3 < Re_{\text{d.i.tube}} = 26926.294 < 5 \cdot 10^8$$

$$10^{-6} = 0.000001 < \frac{\varepsilon_{\text{steel}}}{d_{\text{i.tube}}} = 0.003 < 10^{-2} = 0.01$$

Water Nusselt Number:

$$Nu_{\text{G.pipe.flow}} := \frac{\left(\frac{f}{8} \right) \cdot (Re_{\text{d.i.tube}} - 1000) \cdot Pr_{\text{H2O}}}{1 + 12.7 \cdot \left(\frac{f}{8} \right)^{0.5} \cdot \left(Pr_{\text{H2O}}^{\frac{2}{3}} - 1 \right)} = 131.868 \quad [4]$$

Water Heat Transfer Coefficient:

$$h_{\text{tube.side}} := \frac{Nu_{\text{G.pipe.flow}} \cdot \lambda_{\text{H2O}}}{d_{\text{i.tube}}} = 2983.329 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Properties of Fin and Tube Material:

Thermal conductivity, density and specific heat capacity of AISI 1010 at 300 K:

$$\lambda_{\text{steel}} := 63.9 \frac{\text{W}}{\text{m} \cdot \text{K}} \quad \rho_{\text{steel}} := 7832 \frac{\text{kg}}{\text{m}^3} \quad c_{\text{p,steel}} := 434 \frac{\text{J}}{\text{kg} \cdot \text{K}} \quad [4]$$

$$\lambda_{\text{tube}} := \lambda_{\text{steel}} = 63.9 \frac{\text{W}}{\text{m} \cdot \text{K}} \quad [4]$$

DETERMINING THE UA VALUE:

Thermal resistance due to the water boundary layer on the inner tube wall:

$$R_i := h_{\text{tube.side}} = 2983.329 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Thermal resistance due to fouling on the inside of the tubes (water-side):

$$R_{f,i} := 0 \cdot \frac{\text{m}^2 \cdot \text{K}}{\text{W}}$$

Thermal resistance due to the tube wall thermal conductivity:

$$R_w := \frac{\ln\left(\frac{d_{o,\text{tube}}}{d_{i,\text{tube}}}\right)}{2\pi \cdot \lambda_{\text{tube}} \cdot L_{\text{tube}} \cdot N_{\text{tubes}}} = 4.738 \times 10^{-6} \cdot \frac{\text{K}}{\text{W}}$$

Thermal resistance due to fouling on the outside of the tubes (fluegas-side):

$$R_{f,o} := 0 \cdot \frac{\text{m}^2 \cdot \text{K}}{\text{W}}$$

Thermal resistance due to fluegas boundary layer on the outer tube wall:

$$R_o := h_{\text{fin.side}} = 26.351 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Inside tube surface efficiency (water-side): $\eta_{\text{H}_2\text{O}} := 1$

[Unity for no fins.]

Outside tube surface efficiency:

Approximating H-type fins as rectangular fins to determine the fin efficiency:

$$\varphi_1 := 1.28 \cdot \frac{h_{\text{fin}}}{d_{o,\text{tube}}} \cdot \sqrt{\left(\frac{w_{\text{fin}}}{h_{\text{fin}}} - 0.2\right)} = 2.8847037593 \quad \varphi := (\varphi_1 - 1) \cdot (1 + 0.35 \cdot \ln(\varphi_1)) = 2.583547718$$

$$X := \varphi \cdot \frac{d_{o,\text{tube}}}{2} \cdot \sqrt{\frac{2 \cdot h_{\text{fin.side}}}{\lambda_{\text{tube}} \cdot t_{\text{fin}}}} = 0.8446861627$$

$$\eta_{\text{fin}} := \frac{\tanh(X)}{X} = 0.815$$

Outside tube surface efficiency:

$$\eta_{\text{fg}} := 1 - \frac{SA_{o,\text{fin}}}{SA_{o,\text{fin.tube}}} \cdot (1 - \eta_{\text{fin}}) = 0.840372$$

Therefore we can calculate the UA value as:

$$UA := \left(\frac{1}{\eta_{\text{H}_2\text{O}} \cdot R_i \cdot SA_{i,\text{tube.tot}}} + \frac{R_{f,i}}{\eta_{\text{H}_2\text{O}} \cdot SA_{i,\text{tube.tot}}} \dots \right)^{-1} = 2032.025 \cdot \frac{\text{W}}{\text{K}}$$

$$+ R_w + \frac{R_{f,o}}{\eta_{\text{fg}} \cdot SA_{o,\text{fin.tube.tot}}} + \frac{1}{\eta_{\text{fg}} \cdot R_o \cdot SA_{o,\text{fin.tube.tot}}}$$

E-NTU For Cross Flow over a Tube Bank:

The hot fluid is the fluegas and is denoted by the subscript H for the foregoing analysis.
The cold fluid is the air and is denoted by the subscript C for the foregoing analysis.

Heat capacity of the hot stream:

$$C_H := \dot{m}_{\text{dot.fg.in}} \cdot c_{p.\text{fg}} = 1.993 \cdot \frac{\text{kW}}{\text{K}}$$

Heat capacity of the cold stream:

$$C_C := \dot{m}_{\text{dot.H2O.in}} \cdot c_{p.\text{H2O}} = 4.673 \cdot \frac{\text{kW}}{\text{K}}$$

Minimum heat capacity:

$$C_{\min} := \min(C_H, C_C) = 1.993 \cdot \frac{\text{kW}}{\text{K}}$$

Maximum heat capacity:

$$C_{\max} := \max(C_H, C_C) = 4.673 \cdot \frac{\text{kW}}{\text{K}}$$

Number of Transfer Units:

$$\text{NTU} := \frac{UA}{C_{\min}} = 1.019$$

Heat Capacity Ratio:

$$C := \frac{C_{\min}}{C_{\max}} = 0.427$$

E-NTU correlation for cross-flow (single-pass), c_min mixed and c_max unmixed:

$$\varepsilon := 1 - e^{-\left[-\left(\frac{1}{C} \right) \cdot \left[1 - e^{(-C \cdot \text{NTU})} \right] \right]} = 0.562 \quad [6]$$

Hot-side Temperatures:

$$T_{H.\text{in}} := T_{\text{fg.in}} = 230.^\circ\text{C}$$

Cold-side Temperatures:

$$T_{C.\text{in}} := T_{\text{H2O.in}} = 80.^\circ\text{C}$$

$$Q_{\text{dot.E.NTU}} := \varepsilon \cdot C_{\min} \cdot (T_{H.\text{in}} - T_{C.\text{in}}) = 168.192 \cdot \text{kW}$$

Outlet temperatures, %error based on E-NTU method for single pass cross flow (c_min mixed, c_max unmixed) and heat transfer rate %error:

Fluegas:

$$T_{H.out} := \frac{\dot{m}_{fg.in} \cdot c_{p,fg} \cdot T_{H.in} - \dot{Q}_{dot.E.NTU}}{\dot{m}_{fg.in} \cdot c_{p,fg}} = 145.625 \cdot ^\circ\text{C}$$

$$\text{Error}_{T_{fg.out}} := \frac{T_{H.out} - T_{fg.out}}{T_{fg.out}} = 0. \%$$

$$\dot{Q}_{dot,fg} = 168.193 \cdot \text{kW}$$

$$\text{Error}_{\dot{Q}_{dot,fg}} := \frac{\dot{Q}_{dot.E.NTU} - \dot{Q}_{dot,fg}}{\dot{Q}_{dot,fg}} = -0. \%$$

Water:

$$T_{C.out} := \frac{\dot{Q}_{dot.E.NTU} + \dot{m}_{H2O.in} \cdot c_{p,H2O} \cdot T_{C.in}}{\dot{m}_{H2O.in} \cdot c_{p,H2O}} = 115.989 \cdot ^\circ\text{C}$$

$$\text{Error}_{T_{H2O.out}} := \frac{T_{C.out} - T_{H2O.out}}{T_{H2O.out}} = 0. \%$$

$$\dot{Q}_{dot,H2O} = 168.191 \cdot \text{kW}$$

$$\text{Error}_{\dot{Q}_{dot,H2O}} := \frac{\dot{Q}_{dot.E.NTU} - \dot{Q}_{dot,H2O}}{\dot{Q}_{dot,H2O}} = 0.001. \%$$

Check Convergence:

$$T_{fg.out} = 145.625 \cdot ^\circ\text{C} \quad T_{H2O.out} = 115.989 \cdot ^\circ\text{C}$$

$$T_{H.out} = 145.625 \cdot ^\circ\text{C} \quad T_{C.out} = 115.989 \cdot ^\circ\text{C}$$

$$\text{Error}_{T_{fg.out}} = 0. \% \quad \text{Error}_{T_{H2O.out}} = 0. \%$$

$$\text{Error}_{\dot{Q}_{dot,fg}} = -0. \% \quad \text{Error}_{\dot{Q}_{dot,H2O}} = 0.001. \%$$

$$\dot{Q}_{dot.ECON} := \dot{Q}_{dot.E.NTU} = 168.192 \cdot \text{kW}$$

$$UA_{ECON} := UA = 2.032 \cdot \frac{\text{kW}}{\text{K}}$$

$$\varepsilon_{ECON} := \varepsilon = 0.562$$

$$h_{H2O} := h_{tube.side} = 2983.329 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

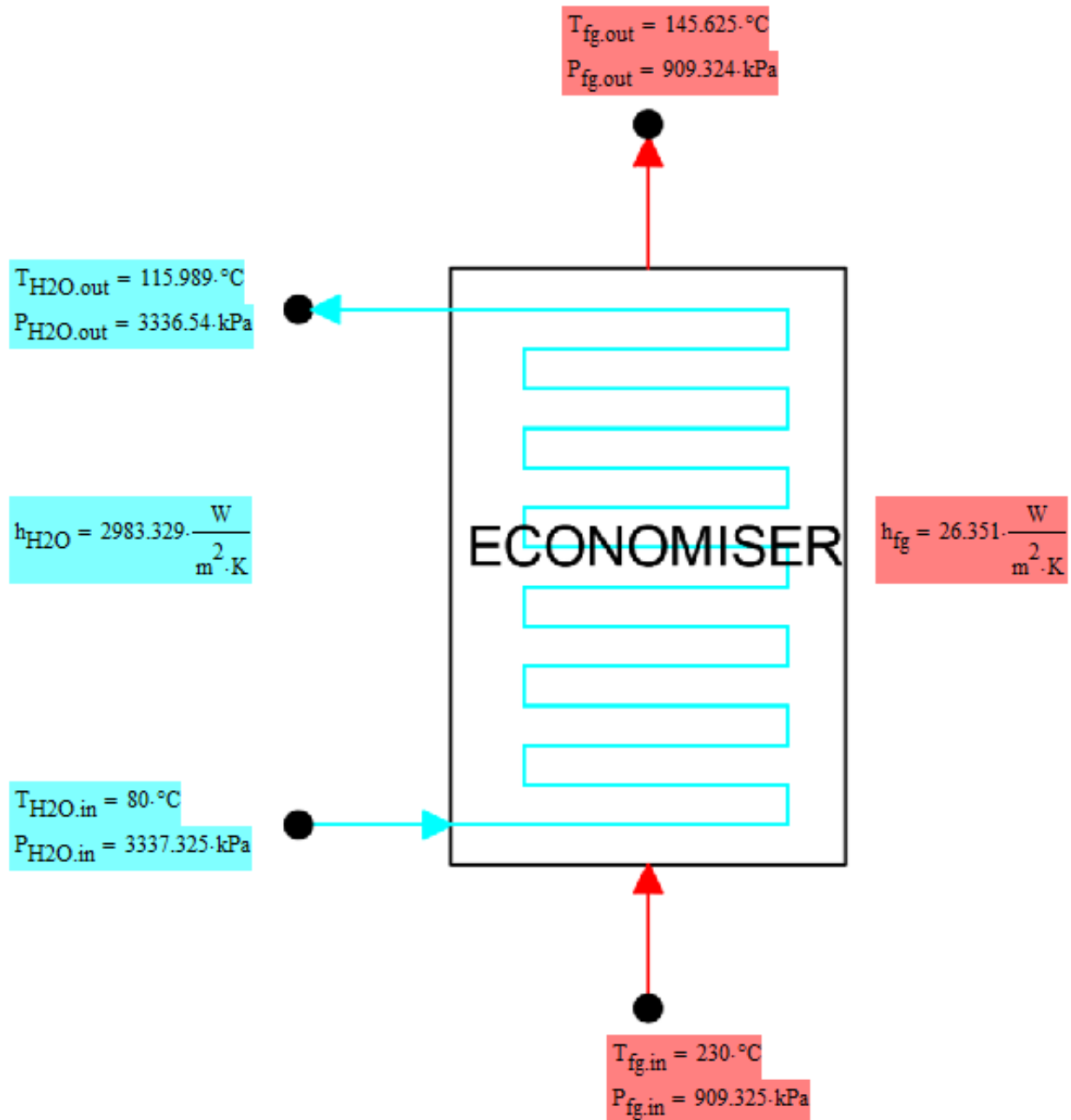
$$h_{fg} := h_{fm.side} = 26.351 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

SUMMARY OF CONVERGED RESULTS:

$$\dot{Q}_{\text{dot.ECON}} = 168.192 \cdot \text{kW}$$

$$UA_{\text{ECON}} = 2.032 \cdot \frac{\text{kW}}{\text{K}}$$

$$\varepsilon_{\text{ECON}} = 0.562$$



ECONOMISER - Flownex Model

The inputs used for the flownex model follow from the analytical model. The "quick script" employed in flownex to calculate the fin-side heat transfer coefficient follow the same methodology as the analytical model.

The difference is that the "quick-script" in flownex receive information such as fluid properties and mass flowrates from flow elements in the network with the use of data transfer links, whereas the analytical model requires the user to evaluate the fluid properties and manually update in a successive approximation.

The *Finned-Tube Heat Exchanger* component was used to model the heat transfer in the air heater. This was coupled with a "quick-script" which employed the same heat transfer calculation methodology as presented in the analytical model.

The inputs to each side of the *Finned-Tube Heat Exchanger* component are given below.

Flownex Inputs (Fin-side)

Fluids

Fluid data reference:

[For verification of analytical model]

"Air| Gases (Pure Fluids) {ATProM Database}"

[For intergrated model with the fluid speicified mass fractions equal to that of the combustion products.]

"Coal fired flue gas| Flue gas (Mixed Fluids) {ATProM Database}"

Note that the fluegas composition used were as a result of the combustion stoichiometric calculations and are shwon below:

☐ Mass Fraction	
Specify mass fraction	Yes ▼
☐ Mass fractions	1 ...
SO2	2.1739E-19
Fly ash	0.000982234
Unburnt carbon	0.00112597
CO2 - Carbon Dioxide	0.111231
N2 - Nitrogen	0.60349
NO - Nitric Oxide	0
O2 - Oxygen	0.0437381
H2O - Water	0.239433
Steady-state continuity option	Calculate mass source ▼

Specification

Specification option: "General Specification" [To ensure compatibility with the analytical methdology]

Fin (Primary) Side Geometry

Flow Direction: "Counter" [Makes no difference since the flownex model uses 3 sub HX in series]

Heat transfer area: $SA_{o,fin,tube,tot} = 98.354 \text{ m}^2$ [External tube surface area for all tubes]

Area ratio: $Area_{Ratio} = 8.288$ [Outside SA to inside SA ratio]

Fluid volume: $V_{fluid} = 1.165 \text{ m}^3$ [Volume occupied by the air]

Fluid path length: $L_{fg,pass} := D_{econ} = 1.5 \text{ m}$ [Length of single air side pass]

Number of increments: $N_{inc} = 1$ [Insignificant difference if discretised further]

Inlet Area: $A_{inlet} := A_{econ} = 0.945 \text{ m}^2$ [Far-field air inlet area]

Inlet Area Ratio: $A_{in,ratio} := SCOF_{mod}^{-1} = 0.575$ [Ratio of free-flow area to far-field area at inlet]

Outlet Area: $A_{outlet} := A_{econ} = 0.945 \text{ m}^2$ [Far-field air outlet area]

Outlet Area Ratio: $A_{out,ratio} := SCOF_{mod}^{-1} = 0.575$ [Ratio of free-flow area to far-field area at outlet]

Material Data

Material option: "Select from data reference"

Mass: $m_{fin,tube,mat} = 1282.548 \text{ kg}$ [Mass of all tube material]

Wall thickness: $t_{tube} = 4.06 \text{ mm}$

Material selection: "Carbon Steel - AISI 1010| Metals (Solids) {Master}"

Losses

Primary loss type: "Specify friction factor"

Friction factor: $f_{fin,side} := 0.04$ [Insignificant but flownex requires a loss input to solve]

Heat Transfer

Convection Coefficient option: "Constant h"

Fin efficiency: $\eta_{fin} = 0.815$ [Calculated inside "quick-script" using the analytical methodology]

Fouling factor: $R_{f,o} = 0 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$ [Fouling not included - has small effects]

h: As calculated in script following the analytical methodology

Fixed Options

Fixed mass flow: No

Prevent flow reversal: No

Advanced

Specify advanced inputs: No

Flownex Inputs (Tube-side)**Fluids**

Fluid data reference: H2O - Water| General (Two Phase Fluids) {Master}

Tube (Secondary) Side Geometry

Number of parallel circuits: $N_{\text{par.circ}} = 1$

Number of passes: $N_{\text{passes}} = 14$ [In accordance with flownex convetions and general arrangement]

Tube rows: $N_{\text{rows}} = 6$

Tube inside diameter: $d_{\text{i.tube}} = 29.98 \text{ mm}$

Total tube pass length: $L_{\text{tube.pass}} := N_{\text{passes}} \cdot L_{\text{tube}} = 21 \text{ m}$

Losses

Primary loss type: "Darcy Weisbach"

Roughness option: "Select from database"

Roughness selection: "Seamless steel tubes - New, unused - Max"

Roughness: $100 \mu\text{m}$ [As per above selection]

K actual inlet: 0 [Secondary losses not included]

K actual outlet: 0 [Secondary losses not included]

Heat Transfer

Convection coefficient option: "Calculate h"

Convection coefficient correlation selection: "Tube inside (Gnielinski)"

Fin efficiency: $\eta_{\text{fin.tube.side}} := 1$ [Finless tubes on inside]

Fouling factor: $R_{\text{fi}} = 0 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$ [Fouling not included - has small effects]

Tube roughness option: "Obtain from flow component"

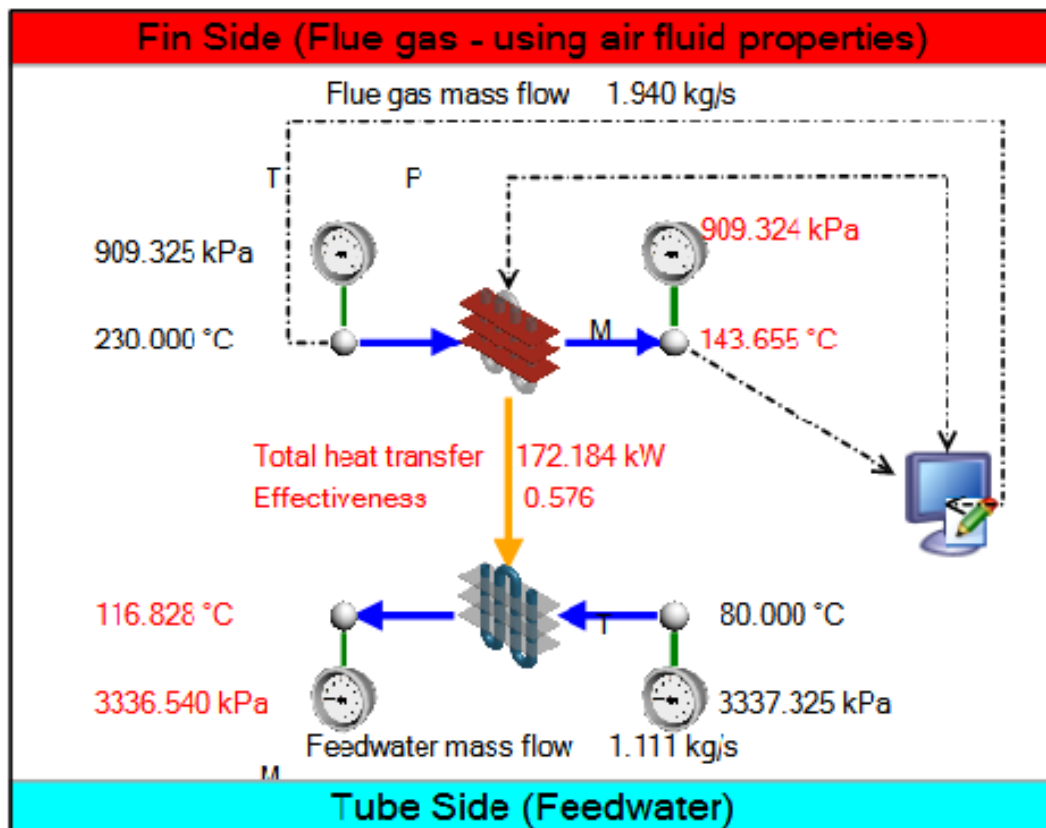
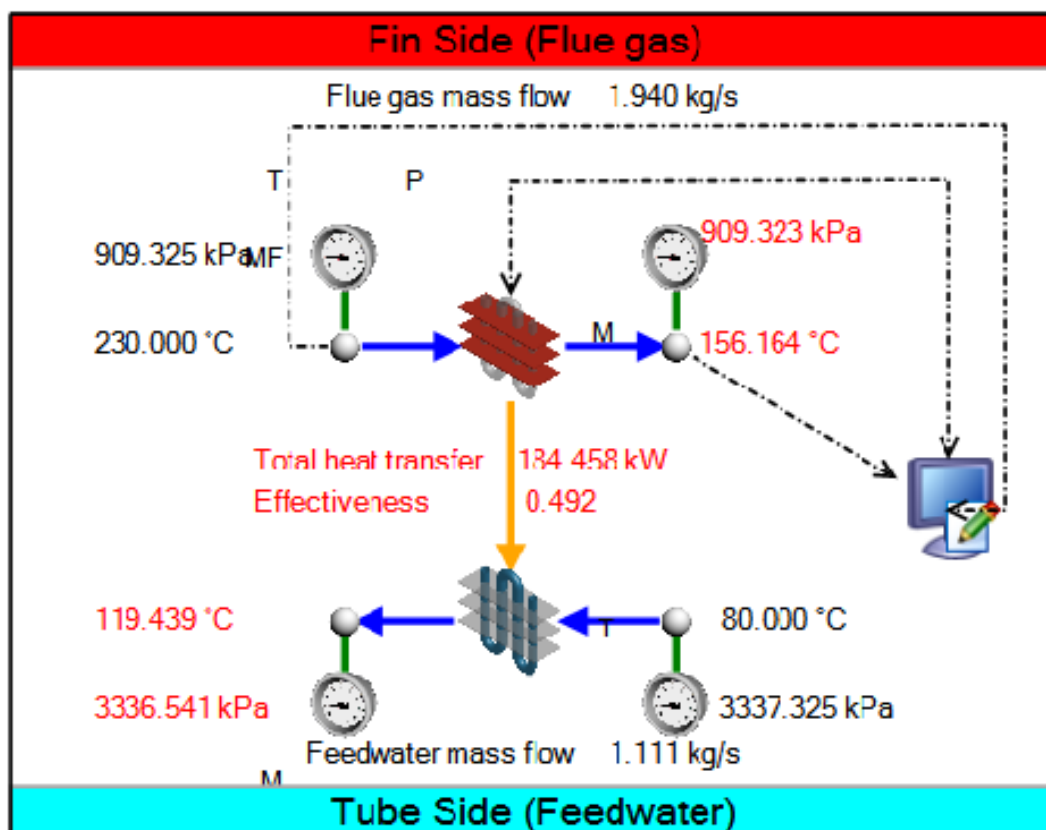
Fixed Options

Fixed mass flow: No

Prevent flow reversal: No

Advanced

Specify advanced inputs: No

FLOWNEX RESULTS (air props on flue gas side)**FLOWNEX RESULTS (real flue gas properties)**

Comparison of MCR, Analytical, and Flownex Results

Heat Transfer Rate (MCR Data):

$$Q_{\text{dot.MCR}} = 177.636 \text{ kW}$$

Heat Transfer Rate (Analytical - air props for flue gas):

$$Q_{\text{dot.analytical.air}} := Q_{\text{dot.ECON}} = 168.192 \text{ kW}$$

Heat Transfer Rate (Flownex - air props for flue gas):

$$Q_{\text{dot.flownex.air}} := 172.184 \text{ kW}$$

Heat Transfer Rate (Flownex - flue gas props):

$$Q_{\text{dot.flownex.fg}} := 184.458 \text{ kW}$$

$$\text{Error}_{\text{analytical.air vs. MCR}} := \frac{Q_{\text{dot.analytical.air}} - Q_{\text{dot.MCR}}}{Q_{\text{dot.MCR}}} = -5.316\%$$

$$\text{Error}_{\text{flownex.air vs. MCR}} := \frac{Q_{\text{dot.flownex.air}} - Q_{\text{dot.MCR}}}{Q_{\text{dot.MCR}}} = -3.069\%$$

$$\text{Error}_{\text{flownex.fg vs. MCR}} := \frac{Q_{\text{dot.flownex.fg}} - Q_{\text{dot.MCR}}}{Q_{\text{dot.MCR}}} = 3.841\%$$

$$\text{Error}_{\text{flownex.air vs. analytical.air}} := \frac{Q_{\text{dot.flownex.air}} - Q_{\text{dot.analytical.air}}}{Q_{\text{dot.analytical.air}}} = 2.373\%$$

From the above error analysis between the various models and MCR data we can say the following:

- The Flownex model has been verified against the analytical model (2.373% error)
- The Flownex model has been verified against the MCR data (3.841% error)

Note that a direct verification between the analytical model and the MCR data was not explicitly accomplished due to limitations on including the real fluegas properties in the mathcad model.

However, since we have agreement between the flownex model and analytical model for the case where both models approximate fluegas properties using air as the fluid with an error of 2.373%, we can confidently conclude that the analytical model has been implicitly verified against the MCR data. This is because the flownex model results for the case where the real fluegas properties are used agrees with the MCR data to within 3.841%.

These errors are all small enough to have confidence in the steady state verification of the standalone analytical and Flownex model.

Appendix B. Air Heater Verification Details

☞ Reference: C:\Users\pmb87\Desktop\Coolprop Fluid Properties rev 1.xmcd(R)

AIR HEATER VERIFICATION (Analytical vs. MCR vs. Flownex)

E-NTU Cross Flow (C_{min} Mixed - C_{max} Unmixed) - 3 HX in series

Multipass Tubular Heat Exchanger.

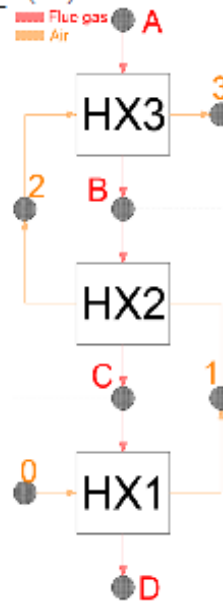
Fluegas travels through the air heater (top-down) in one pass inside the tubes.

Combustion air is blown via a FD fan into the bottom level of the air heater.

The air enters the air heater on the bottom left side, makes 3 passes through the air heater before exiting the air heater at the top right side. This model utilises 3 individual heat exchangers (HX1, HX2 and HX3) which are connected as per the diagram below.

Since the air will undergo much turbulence (i.e. mixing) due to obstructions (tubes) in its flow path and the fact that the minimum heat capacity rate is that of the air, the E-NTU correlation for cross flow with C_{min} mixed and C_{max} unmixed is used in this model as shown below:

$$\epsilon_{C_{min}.mixed.C_{max}.unmixed} = 1 - e^{-\left[-\left(\frac{1}{C} \right) \cdot \left[1 - e^{(-C \cdot NTU)} \right] \right]}$$



Reference List:

- [1] TO400 TORRIPAC BOILER PROCESS FLOW DIAGRAM, 68756-03, rev 0
- [2] TH0400-40 ECONOMISER AIRHEATER GENERAL ARRANGEMENT, 68659-03, rev 0
- [3] TH0400-40 ECONOMISER AIR HEATER TUBE PLATES, 68759-13, rev 0
- [4] VDI Heat Atlas, (pg 1273-1277), M1 Hat Transfer to Finned Tubes
- [5] Heat and Mass Transfer, Fundamentals & Applications, Yunus A. Cengel, Afshin J. Ghajar.
- [6] Zeghadnia, J. Loup, and B. Achour, "Explicit solutions for turbulent flow friction factor 'a0: A review , assessment and approaches classification," *Ain Shams Eng. J.*, vol. 10, no. 1, pp. 243252, 2019, doi: 10.1016/j.asej.2018.10.007
- [7] Heat Transfer in Single and Multiphase Systems, Greg F. Naterer, pg 491

Atmospheric Pressure: $P_{atm} := 101.325 \text{ kPa}$

John Thompson MCR Design Data [1]

This section summarises the Maximum Capacity Rating design data as per John Thompsons designs [1].

It includes mass flowrates as well as inlet and outlet temperatures and pressures for both streams (air and fluegas).

A simple heat transfer rate calculation $\dot{Q} = \dot{m} \cdot c_p \cdot \Delta T$ is applied to determine the heat transfer rate as per the MCR data. This value will then be compared to the heat transfer rate of the analytical model and Flownex model as a verification.

A note on the MCR pressures and model pressures:

MCR air pressures were not specified in [1].

The inlet air pressure is therefore assumed to be atmosphere. In reality the inlet air pressure to the air heater is downstream of a forced draft fan but this slight pressure increase is not significant for our analysis on the heat transfer.

Initially the outlet air pressure was assumed to just be 2% lower than inlet pressure in the analytical model. That being said, the pressures used in the analytical mathcad model presented here have been updated to be the pressures computed by the Flownex model. This was done to ensure consistency between models since the analytical model does not explicitly employ a momentum balance pressure solve.

We are not concerned with modelling complex pressure drop characteristics in the analytical model since the actual pressure drops are very small and have insignificant impacts on the fluid properties and consequently the heat transfer.

Since we are discretising the air heater into 3 sub-heat exchangers (one per air side pass), we could simply use the inlet and outlet pressures, and assume a linear pressure drop distribution to determine the intermediate pressure at the exit of each sub-heat exchanger. This was the methodology used in the first iteration of the analytical model. But, again the flownex computed outlet pressures were used for the heat transfer calculations for both MCR and analytical model for consistency and continuity when verifying the models against the MCR data.

Note that MCR data indicates a fluegas pressure rise from inlet to outlet through the air heater. This is likely due to the ID fan downstream. Since the flownex model does not include the ID fan, the outlet fg pressure for the analytical model is taken to be that calculated by flownex (see below).

The flownex computed pressures are: $P_{\text{flownex.air.out}} := 101.083\text{kPa}$

$P_{\text{flownex.fg.out}} := 791.320\text{kPa}$

MCR Design Data

Air Side

Mass flowrate: $\dot{m}_{\text{air.in.JT}} := 5390 \frac{\text{kg}}{\text{hr}}$ [1]

Inlet temperature: $T_{\text{air.in.JT}} := 27^\circ\text{C}$ [1]

Outlet temperature: $T_{\text{air.out.JT}} := 197^\circ\text{C}$ [1]

Inlet pressure: $P_{\text{air.in.JT}} := P_{\text{atm}} = 101.325\text{-kPa}$ [assumption as explained above]

Outlet pressure: $P_{\text{air.out.JT}} := P_{\text{flownex.air.out}} = 101.083\text{-kPa}$ [flownex result as explained above]

Flue gas Side

Mass flowrate: $\dot{m}_{\text{fg.JT}} := 6984 \frac{\text{kg}}{\text{hr}}$ [1]

Inlet temperature: $T_{\text{fg.in.JT}} := 338^\circ\text{C}$ [1]

Outlet temperature: $T_{\text{fg.out.JT}} := 230^\circ\text{C}$ [1]

Inlet pressure: $P_{\text{fg.in.JT}} := 690\text{kPa} + P_{\text{atm}} = 791.325\text{-kPa}$ [1]

Outlet pressure: $P_{\text{fg.out.JT}} := 808\text{kPa} + P_{\text{atm}} = 909.325\text{-kPa}$ [1]

$P_{\text{fg.out.JT}} := P_{\text{flownex.fg.out}} = 791.32\text{-kPa}$ [flownex result as explained above]

MCR Heat Transfer Rate (using air side conditions)

Average air temperature: $T_{\text{air.avg.JT}} := \frac{T_{\text{air.in.JT}} + T_{\text{air.out.JT}}}{2} = 112^\circ\text{C}$

Average air pressure: $P_{\text{air.avg.JT}} := \frac{P_{\text{air.in.JT}} + P_{\text{air.out.JT}}}{2} = 101.204\text{-kPa}$

Air specific heat capacity evaluated at average T&P:

$c_{p,\text{air.JT}} := C_{p,\text{fluid}}(\text{"air"}, P_{\text{air.avg.JT}}, T_{\text{air.avg.JT}}, \text{"", ""}, \text{"", ""}) = 1.012 \frac{\text{kJ}}{\text{kg}\cdot\text{K}}$

Heat Transfer Rate: $\dot{Q}_{\text{dot.MCR}} := \dot{m}_{\text{dot.air.JT}} \cdot c_{p,\text{air.JT}} (T_{\text{air.out.JT}} - T_{\text{air.in.JT}}) = 257.699\text{-kW}$

AIR HEATER - Analytical Model

- Multipass tubular heat exchanger
- Finless (bare tube)
- Finside fluid - air
- Tubeside fluid - flue gas
- Flue gas inlet duct at the top of the assembly
- Flue gas outlet duct as bottom of assembly
- Flue gas makes single pass inside the tubes (from top to bottom)
- Air inlet duct at bottom left of assembly
- Air outlet duct at top right of assembly
- Air makes 3 passes between the tubes
- 4 tube rows
- Single tube pass (flue gas)
- 30 parallel circuits
- Staggered tube arrangement

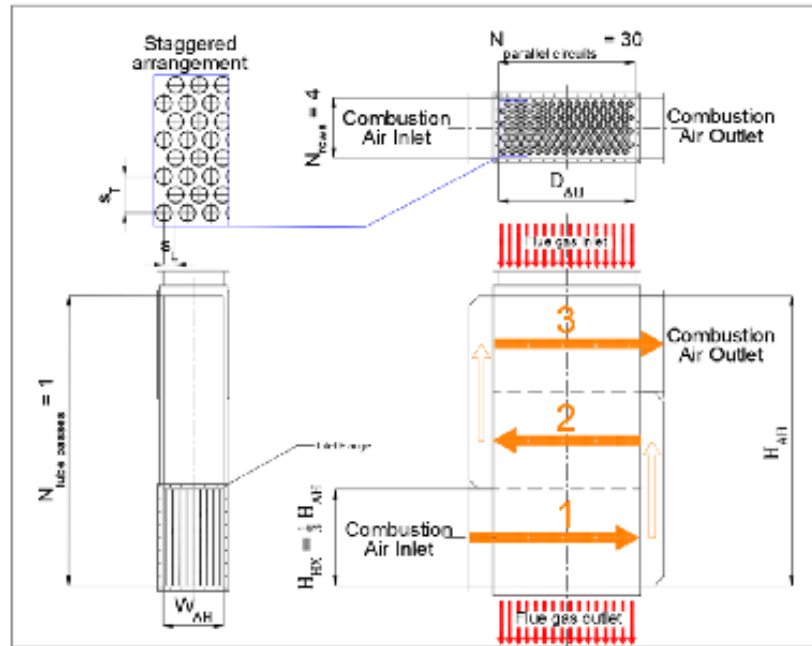


Figure 1: Air Heater General Arrangement

GEOMETRIC INPUTS

Tube Arrangement

Number of rows (perpendicular to flue gas flow direction):

$$N_{\text{rows}} := 4$$

Number of passes (this is the same for each tube row):

$$N_{\text{passes}} := 1$$

Number of parallel circuits (tubes parallel to flue gas flow):

$$N_{\text{par.circ}} := 30$$

The total number of wetted tubes in cross-flow is:

$$N_{\text{tubes}} := N_{\text{rows}} \cdot N_{\text{par.circ}} = 120$$

Overall Air Heater Dimensions

Air heater depth:

$$D_{\text{AH}} := 1400\text{mm} \quad [2]$$

Air heater height:

$$H_{\text{AH}} := 3000\text{mm} \quad [2]$$

Air heater width:

$$W_{\text{AH}} := 560\text{mm} \quad [2]$$

Sub-Heat Exchanger Dimensions (HX)

Depth (i.e. single finside pass length):

$$D_{\text{HX}} := D_{\text{AH}} = 1400\text{mm}$$

Height (perpendicular to cross flow direction):

$$H_{\text{HX}} := \frac{1}{3} \cdot H_{\text{AH}} = 1000\text{mm}$$

Width (perpendicular to cross flow direction):

$$W_{\text{HX}} := W_{\text{AH}} = 560\text{mm}$$

Cross-sectional area of air heater (perp to air flow crossflow direction): $A_{\text{HX}} := H_{\text{HX}} \cdot W_{\text{HX}} = 0.56\text{m}^2$

Tube Geometry:

Tube OD:	$d_{o,tube} := 63.5\text{mm}$	[2]
Tube thickness:	$t_{tube} := 2\text{mm}$	[2]
Tube ID:	$d_{i,tube} := d_{o,tube} - 2 \cdot t_{tube} = 59.5\text{-mm}$	
Tube Length:	$L_{tube} := 3\text{m}$	[2]

Tube Arrangement:

Staggered arrangement

 Transverse Pitch (spacing perpendicular to flue gas flow direction): $s_T := 140\text{mm}$ [3]

 Longitudinal Pitch (spacing parallel to flue gas flow direction): $s_L := 45\text{mm}$ [3]

Diagonal Pitch:

$$s_D := \sqrt{s_L^2 + \left(\frac{s_T}{2}\right)^2} = 83.217\text{-mm}$$

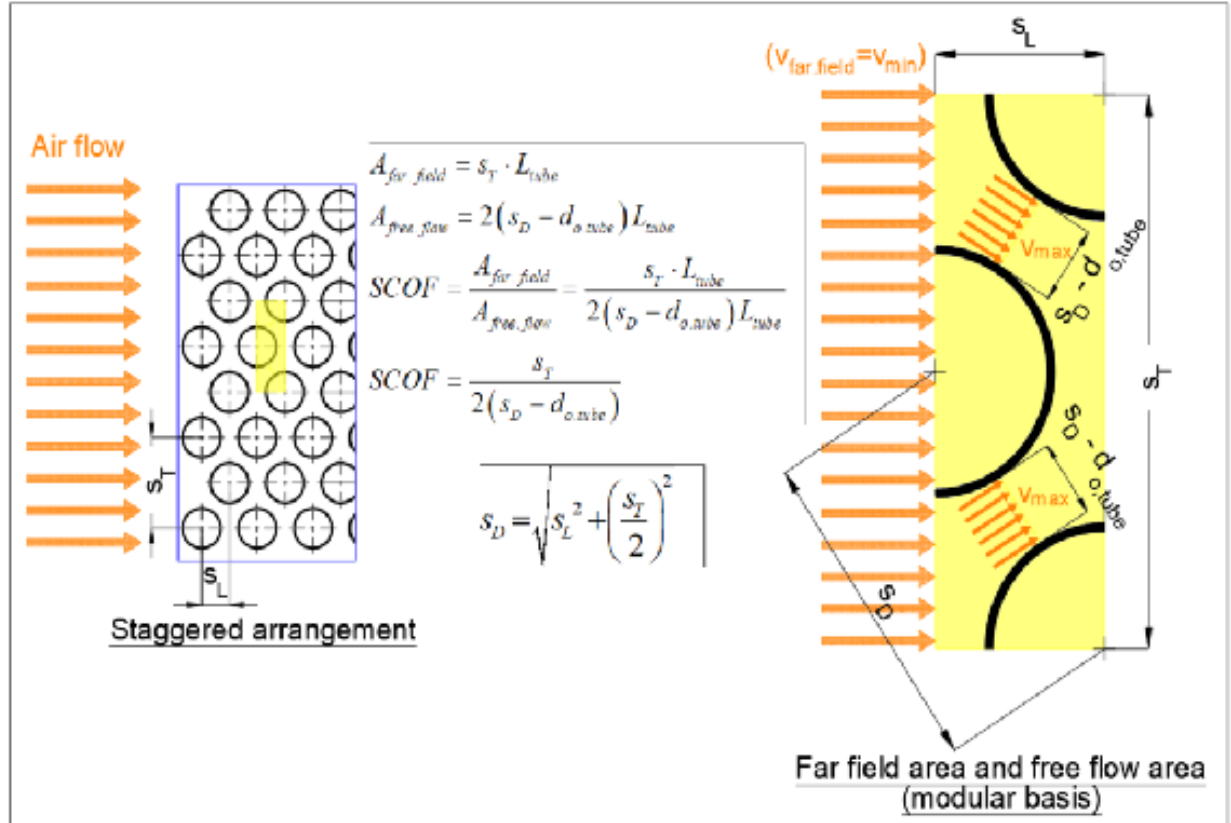
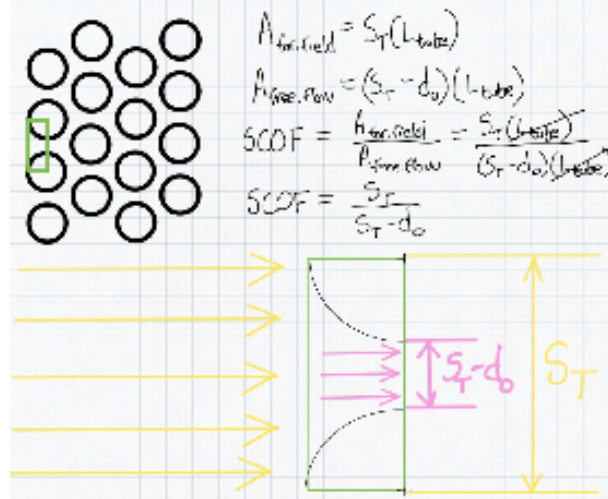


Figure 2: Far-field and free-flow areas and velocities (modular basis)

Straightened Cross-section of Flow (SCOF) Calculation

1a. Free Flow Area based on Modular unit (transverse plane)



Far field area

$$A_{far.field.1} := s_T \cdot H_{HX} = 0.14 \text{ m}^2$$

Free flow width

$$w_{free.flow.1a} := s_T - d_{o.tube} = 76.5 \text{ mm}$$

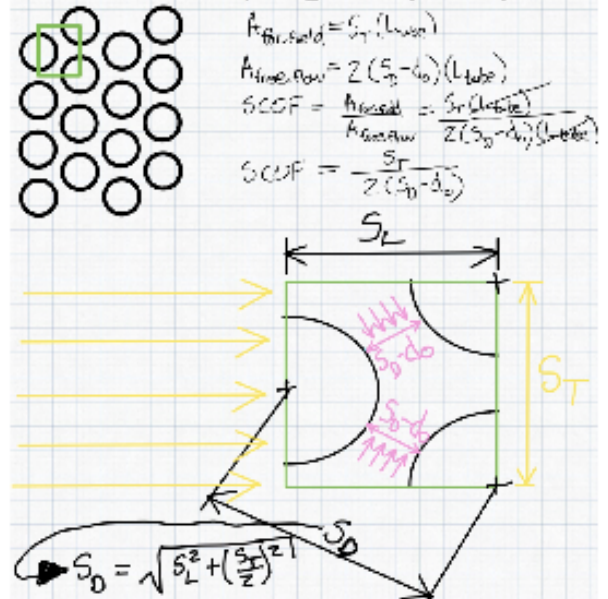
Free flow area

$$A_{free.flow.1a} := w_{free.flow.1a} \cdot H_{HX} = 0.077 \text{ m}^2$$

Straightened cross-section of flow

$$SCOF_{1a} := \frac{A_{far.field.1}}{A_{free.flow.1a}} = 1.83$$

1b. Free Flow Area based on Modular unit (diagonal plane)



Far field area

$$A_{far.field.1} = 0.14 \text{ m}^2$$

Free flow width

$$w_{free.flow.1b} := 2 \cdot (s_D - d_{o.tube}) = 39.433 \text{ mm}$$

Free flow area

$$A_{free.flow.1b} := w_{free.flow.1b} \cdot H_{HX} = 0.039 \text{ m}^2$$

Straightened cross-section of flow

$$SCOF_{1b} := \frac{A_{far.field.1}}{A_{free.flow.1b}} = 3.55$$

$$SCOF := \begin{cases} SCOF_{1a} & \text{if } A_{free.flow.1a} < A_{free.flow.1b} \\ SCOF_{1b} & \text{if } A_{free.flow.1b} < A_{free.flow.1a} \end{cases} = 3.55$$

Free flow area should be taken as the smallest area. In our case the transverse plane area is the smallest.

Check that overall frontal inlet area, overall free flow area (diagonal) and overall inlet area ratio agrees with the modular approach:

$$A_{HX,overall} := A_{HX} = 0.56 \text{ m}^2$$

$$A_{free.flow,overall} := 2 \cdot N_{rows} \cdot (s_D - d_{o.tube}) \cdot H_{HX} = 0.158 \text{ m}^2$$

$$\frac{A_{HX,overall}}{A_{free.flow,overall}} = 3.55$$

Heat Transfer Surface Areas:**Fluegas-side Heat Transfer Area:**

$$SA_{i.tube} := \pi \cdot d_{i.tube} \cdot L_{tube} = 0.561 \text{ m}^2$$

$$SA_{i.tube.tot} := N_{tubes} \cdot SA_{i.tube} = 67.293 \text{ m}^2$$

Fluegas-side Heat Transfer Area (per HX):

$$SA_{i.tube.tot.HX} := \frac{SA_{i.tube.tot}}{3} = 22.431 \text{ m}^2$$

Air-side Heat Transfer Area:

$$SA_{o.tube} := (\pi \cdot d_{o.tube} \cdot L_{tube}) = 0.598 \text{ m}^2$$

$$SA_{o.tube.tot} := N_{tubes} \cdot SA_{o.tube} = 71.817 \text{ m}^2$$

Air-side Heat Transfer Area (per HX):

$$SA_{o.tube.tot.HX} := \frac{SA_{o.tube.tot}}{3} = 23.939 \text{ m}^2$$

Ratio of finned-tube heat transfer surface area to bare tube heat transfer surface area:

$$ROS := 1$$

[Finless tubes]

Ratio of external to internal surface area:

$$Area_{Ratio} := \frac{SA_{o.tube.tot}}{SA_{i.tube.tot}} = 1.067$$

Total fluid volume (air inside of air heater):

$$V_{AH} := W_{AH} \cdot H_{AH} \cdot D_{AH} = 2.352 \cdot \text{m}^3$$

$$V_{tubes} := \frac{\pi}{4} \cdot d_{o.tube}^2 \cdot H_{AH} \cdot N_{tubes} = 1.14 \cdot \text{m}^3$$

$$V_{fluid} := V_{AH} - V_{tubes} = 1.212 \cdot \text{m}^3$$

Total fluid volume (air inside of sub-HX):

$$V_{fluid.HX} := \frac{V_{fluid}}{3} = 0.404 \cdot \text{m}^3$$

Tube material mass:

$$V_{tube.mat} := \frac{\pi}{4} \cdot (d_{o.tube}^2 - d_{i.tube}^2) \cdot L_{tube} \cdot N_{tubes} = 0.139 \cdot \text{m}^3$$

$$\rho_{steel} := 7832 \frac{\text{kg}}{\text{m}^3}$$

$$m_{tube.mat} := V_{tube.mat} \cdot \rho_{steel} = 1089.507 \text{ kg}$$

Tube material mass (per sub-HX):

$$m_{tube.mat.HX} := \frac{m_{tube.mat}}{3} = 363.169 \text{ kg}$$

Air-side Inputs:

All inputs below, barring the outlet temperatures have been explained in the beginning of this report.

Air mass flowrate: $\dot{m}_{\text{air.in}} := \dot{m}_{\text{air.in.JT}} = 1.497 \frac{\text{kg}}{\text{s}}$

Inlet air pressure (absolute): $P_{\text{air.in}} := P_{\text{air.in.JT}} = 101.325 \cdot \text{kPa}$

Outlet air pressure (absolute): $P_{\text{air.out}} := P_{\text{air.out.JT}} = 101.083 \cdot \text{kPa}$

Nodal Pressures: $P_{\text{air.0}} := P_{\text{air.in}} = 101.325 \cdot \text{kPa}$

$P_{\text{air.1}} := 101.254 \text{ kPa}$ [from flownex result]

Intermediate pressures: $P_{\text{air.2}} := 101.173 \text{ kPa}$ [from flownex result]

$P_{\text{air.3}} := P_{\text{air.out}} = 101.083 \cdot \text{kPa}$

Note that the intermediate pressures (1&2) could have just been used by assuming a linear pressure drop distribution. Instead the flownex results have just been used since it will give better verification since we are not interested in solving for pressure in the analytical model.

Inlet air temperature: $T_{\text{air.in}} := 27^\circ\text{C}$

Outlet air temperature: $T_{\text{air.out}} := 182.372^\circ\text{C}$ [Successive approximation with first guess as 197 degC (i.e. as per MCR data)]

Nodal Temperatures: $T_{\text{air.0}} := T_{\text{air.in}} = 27^\circ\text{C}$

$T_{\text{air.1}} := 81.633^\circ\text{C}$

$SR := \frac{808}{5390} = 14.991\%$

Intermediate temperatures: $T_{\text{air.2}} := 133.509^\circ\text{C}$

$T_{\text{air.3}} := T_{\text{air.out}} = 182.372^\circ\text{C}$

Note: Green Highlights indicate variables that are updated iteratively.

Note: Expressions with borders are inputs that can be changed when verifying/validating different operating points based on a different boiler load

HX1 Air Fluid Properties:

Air Density evaluated at T.air.in and P.air.in: This should give a more accurate result for the inlet velocity.

$$\rho_{\text{air.in.HX1}} := \rho_{\text{fluid}}(\text{"air"}, P_{\text{air.0}}, T_{\text{air.0}}, \text{"", ""}, \text{"", ""}) = 1.176 \frac{\text{kg}}{\text{m}^3}$$

Average air-side Temperature and Pressure:

$$T_{\text{air.avg.HX1}} := \frac{T_{\text{air.0}} + T_{\text{air.1}}}{2} = 54.317 \cdot ^\circ\text{C} \quad P_{\text{air.avg.HX1}} := \frac{P_{\text{air.0}} + P_{\text{air.1}}}{2} = 101.29 \cdot \text{kPa}$$

Air Specific Heat Capacity evaluated at T.air.avg and P.air.avg:

$$c_{p,\text{air.HX1}} := C_{p,\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX1}}, T_{\text{air.avg.HX1}}, \text{"", ""}, \text{"", ""}) = 1.008 \cdot \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

Air Density evaluated at T.air.avg and P.air.avg:

$$\rho_{\text{air.HX1}} := \rho_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX1}}, T_{\text{air.avg.HX1}}, \text{"", ""}, \text{"", ""}) = 1.078 \frac{\text{kg}}{\text{m}^3}$$

Air Thermal Conductivity:

$$\lambda_{\text{air.HX1}} := \lambda_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX1}}, T_{\text{air.avg.HX1}}, \text{"", ""}) = 0.028 \text{ m} \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Air Dynamic Viscosity:

$$\mu_{\text{air.HX1}} := \mu_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX1}}, T_{\text{air.avg.HX1}}, \text{"", ""}) = 0 \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Air Kinematic Viscosity:

$$\nu_{\text{air.HX1}} := \frac{\mu_{\text{air.HX1}}}{\rho_{\text{air.HX1}}} = 0 \frac{\text{m}^2}{\text{s}}$$

Air Prandtl Number:

$$Pr_{\text{air.HX1}} := Pr_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX1}}, T_{\text{air.avg.HX1}}, \text{"", ""}) = 0.704$$

Total air-side heat transfer rate:

$$\dot{Q}_{\text{dot.air.HX1}} := \dot{m}_{\text{dot.air.in}} \cdot c_{p,\text{air.HX1}} (T_{\text{air.1}} - T_{\text{air.0}}) = 82.426 \cdot \text{kW}$$

HX2 Air Fluid Properties:

Air Density evaluated at T.air.in and P.air.in: This should give a more accurate result for the inlet velocity.

$$\rho_{\text{air.in.HX2}} := \rho_{\text{fluid}}(\text{"air"}, P_{\text{air.1}}, T_{\text{air.1}}, \text{"", ""}, \text{"", ""}) = 0.994 \frac{\text{kg}}{\text{m}^3}$$

Average air-side Temperature and Pressure:

$$T_{\text{air.avg.HX2}} := \frac{T_{\text{air.1}} + T_{\text{air.2}}}{2} = 107.571 \cdot ^\circ\text{C} \quad P_{\text{air.avg.HX2}} := \frac{P_{\text{air.1}} + P_{\text{air.2}}}{2} = 101.213 \cdot \text{kPa}$$

Air Specific Heat Capacity evaluated at T.air.avg and P.air.avg:

$$c_{p,\text{air.HX2}} := C_{p,\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX2}}, T_{\text{air.avg.HX2}}, \text{"", ""}, \text{"", ""}, \text{"", ""}) = 1.012 \cdot \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

Air Density evaluated at T.air.avg and P.air.avg:

$$\rho_{\text{air.HX2}} := \rho_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX2}}, T_{\text{air.avg.HX2}}, \text{"", ""}, \text{"", ""}, \text{"", ""}) = 0.926 \frac{\text{kg}}{\text{m}^3}$$

Air Thermal Conductivity:

$$\lambda_{\text{air.HX2}} := \lambda_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX2}}, T_{\text{air.avg.HX2}}, \text{"", ""}) = 0.032 \text{ m} \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Air Dynamic Viscosity:

$$\mu_{\text{air.HX2}} := \mu_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX2}}, T_{\text{air.avg.HX2}}, \text{"", ""}) = 0 \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Air Kinematic Viscosity:

$$\nu_{\text{air.HX2}} := \frac{\mu_{\text{air.HX2}}}{\rho_{\text{air.HX2}}} = 0 \frac{\text{m}^2}{\text{s}}$$

Air Prandtl Number:

$$Pr_{\text{air.HX2}} := Pr_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX2}}, T_{\text{air.avg.HX2}}, \text{"", ""}) = 0.7$$

Total air-side heat transfer rate:

$$\dot{Q}_{\text{dot.air.HX2}} := \dot{m}_{\text{dot.air.in}} \cdot c_{p,\text{air.HX2}} (T_{\text{air.2}} - T_{\text{air.1}}) = 78.601 \cdot \text{kW}$$

HX3 Air Fluid Properties:

Air Density evaluated at T.air.in and P.air.in: This should give a more accurate result for the inlet velocity.

$$\rho_{\text{air.in.HX3}} := \rho_{\text{fluid}}(\text{"air"}, P_{\text{air.2}}, T_{\text{air.2}}, \text{"", ""}, \text{"", ""}) = 0.867 \frac{\text{kg}}{\text{m}^3}$$

Average air-side Temperature and Pressure:

$$T_{\text{air.avg.HX3}} := \frac{T_{\text{air.2}} + T_{\text{air.3}}}{2} = 157.941 \cdot ^\circ\text{C} \quad P_{\text{air.avg.HX3}} := \frac{P_{\text{air.2}} + P_{\text{air.3}}}{2} = 101.128 \cdot \text{kPa}$$

Air Specific Heat Capacity evaluated at T.air.avg and P.air.avg:

$$c_{p,\text{air.HX3}} := C_{p,\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX3}}, T_{\text{air.avg.HX3}}, \text{"", ""}, \text{"", ""}) = 1.018 \cdot \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

Air Density evaluated at T.air.avg and P.air.avg:

$$\rho_{\text{air.HX3}} := \rho_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX3}}, T_{\text{air.avg.HX3}}, \text{"", ""}, \text{"", ""}) = 0.817 \frac{\text{kg}}{\text{m}^3}$$

Air Thermal Conductivity:

$$\lambda_{\text{air.HX3}} := \lambda_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX3}}, T_{\text{air.avg.HX3}}, \text{"", ""}) = 0.036 \text{ m} \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Air Dynamic Viscosity:

$$\mu_{\text{air.HX3}} := \mu_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX3}}, T_{\text{air.avg.HX3}}, \text{"", ""}) = 0 \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Air Kinematic Viscosity:

$$\nu_{\text{air.HX3}} := \frac{\mu_{\text{air.HX3}}}{\rho_{\text{air.HX3}}} = 0 \frac{\text{m}^2}{\text{s}}$$

Air Prandtl Number:

$$Pr_{\text{air.HX3}} := Pr_{\text{fluid}}(\text{"air"}, P_{\text{air.avg.HX3}}, T_{\text{air.avg.HX3}}, \text{"", ""}) = 0.698$$

Total air-side heat transfer rate:

$$\dot{Q}_{\text{dot.air.HX3}} := \dot{m}_{\text{dot.air.in}} \cdot c_{p,\text{air.HX3}} \cdot (T_{\text{air.3}} - T_{\text{air.2}}) = 74.494 \cdot \text{kW}$$

DETERMINING THE AIR HEAT TRANSFER COEFFICIENT FOR FINNED TUBES:

Methodology as per [4].

$$Nu = 0.38 \cdot Re_{d.o.tube}^{0.6} \cdot ROS^{-0.15} \cdot Pr^{\frac{1}{3}}$$
, in which the Reynolds Number uses the tube OD as the characteristic length and uses the maximum velocity at the smallest cross-sectional area of flow, $Re_{d.o.tube} = \frac{v_{max} \cdot d_{o.tube}}{\nu} = \frac{v_{max} \cdot d_{o.tube} \cdot \rho}{\mu}$, where:

$v_{max} = SCOF \cdot v_{far.field} = \text{velocity, in, the, smallest, cross, section,}$

Finally the air heat transfer coefficient may be calculated as:

$$h_{air} = \frac{Nu \cdot \lambda_{air}}{d_{o.tube}}$$

HX1 Air Heat Transfer Coefficient

Inlet velocity of air into the air heater (i.e. far field velocity):

$$v_{far.field.HX1} := \frac{\dot{m}_{dot.air.in}}{\rho_{air.in.HX1} \cdot A_{HX}} = 2.273 \frac{m}{s}$$

Note that the inlet air density is used rather than the average density of the inlet and outlet.

Max velocity in the pipe bundle:

$$v_{max.HX1} := SCOF \cdot v_{far.field.HX1} = 8.069 \frac{m}{s}$$

Reynolds Number for air flow across pipe bundle:

$$Re_{d.o.tube.HX1} := \frac{v_{max.HX1} \cdot d_{o.tube}}{\nu_{air.HX1}} = 27836.273$$

Air Nusselt Number:

$$Nu_{HX1} := 0.38 \cdot Re_{d.o.tube.HX1}^{0.6} \cdot ROS^{-0.15} \cdot Pr_{air.HX1}^{\frac{1}{3}} = 156.938$$

Air Heat Transfer Coefficient:

$$h_{air.HX1} := \frac{Nu_{HX1} \cdot \lambda_{air.HX1}}{d_{o.tube}} = 70.178 \frac{W}{m^2 \cdot K}$$

HX2 Air Heat Transfer Coefficient

Inlet velocity of air into the air heater (i.e. far field velocity):

$$v_{\text{far.field.HX2}} := \frac{\dot{m}_{\text{air.in}}}{\rho_{\text{air.in.HX2}} \cdot A_{\text{HX}}} = 2.689 \frac{\text{m}}{\text{s}} \quad \text{Note that the inlet air density is used rather than the average density of the inlet and outlet.}$$

Max velocity in the pipe bundle:

$$v_{\text{max.HX2}} := \text{SCOF} \cdot v_{\text{far.field.HX2}} = 9.547 \frac{\text{m}}{\text{s}}$$

Reynolds Number for air flow across pipe bundle:

$$\text{Re}_{\text{d.o.tube.HX2}} := \frac{v_{\text{max.HX2}} \cdot d_{\text{o.tube}}}{\nu_{\text{air.HX2}}} = 25258.019$$

Air Nusselt Number:

$$\text{Nu}_{\text{HX2}} := 0.38 \cdot \text{Re}_{\text{d.o.tube.HX2}}^{0.6} \cdot \text{ROS}^{-0.15} \cdot \text{Pr}_{\text{air.HX2}}^{\frac{1}{3}} = 147.759$$

Air Heat Transfer Coefficient:

$$h_{\text{air.HX2}} := \frac{\text{Nu}_{\text{HX2}} \cdot \lambda_{\text{air.HX2}}}{d_{\text{o.tube}}} = 74.79 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

HX3 Air Heat Transfer Coefficient

Inlet velocity of air into the air heater (i.e. far field velocity):

$$v_{\text{far.field.HX3}} := \frac{\dot{m}_{\text{air.in}}}{\rho_{\text{air.in.HX3}} \cdot A_{\text{HX}}} = 3.085 \frac{\text{m}}{\text{s}} \quad \text{Note that the inlet air density is used rather than the average density of the inlet and outlet.}$$

Max velocity in the pipe bundle:

$$v_{\text{max.HX3}} := \text{SCOF} \cdot v_{\text{far.field.HX3}} = 10.954 \frac{\text{m}}{\text{s}}$$

Reynolds Number for air flow across pipe bundle:

$$\text{Re}_{\text{d.o.tube.HX3}} := \frac{v_{\text{max.HX3}} \cdot d_{\text{o.tube}}}{\nu_{\text{air.HX3}}} = 23334.899$$

Air Nusselt Number:

$$\text{Nu}_{\text{HX3}} := 0.38 \cdot \text{Re}_{\text{d.o.tube.HX3}}^{0.6} \cdot \text{ROS}^{-0.15} \cdot \text{Pr}_{\text{air.HX3}}^{\frac{1}{3}} = 140.784$$

Air Heat Transfer Coefficient:

$$h_{\text{air.HX3}} := \frac{\text{Nu}_{\text{HX3}} \cdot \lambda_{\text{air.HX3}}}{d_{\text{o.tube}}} = 78.761 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Fluegas-side Fluid properties:

[Assuming properties similar to air.]

All inputs below, barring the outlet temperatures have been explained in the beginning of this report.

Flue gas mass flowrate: $\dot{m}_{fg.in} := \dot{m}_{fg.JT} = 1.94 \frac{\text{kg}}{\text{s}}$

Inlet flue gas pressure (absolute): $P_{fg.in} := 690 \cdot \text{kPa} + P_{atm} = 791.325 \cdot \text{kPa}$

Outlet flue gas pressure (absolute): $P_{fg.out} := 791.320 \text{kPa}$

Nodal Pressures:

$$P_{fg.A} := P_{fg.in} = 791.325 \cdot \text{kPa}$$

$$P_{fg.B} := 791.323 \text{kPa}$$

[from flownex result]

Intermediate pressures:

$$P_{fg.C} := 791.322 \text{kPa}$$

[from flownex result]

$$P_{fg.D} := P_{fg.out} = 791.32 \cdot \text{kPa}$$

Note that the intermediate pressures (B&C) could have just been used by assuming a linear pressure drop distribution. Instead the flownex results have just been used since it will give better verification since we are not interested in solving for pressure in the analytical model.

Inlet fluegas temperature: $T_{fg.in} := 338 \text{ } ^\circ\text{C}$

Outlet fluegas temperature: $T_{fg.out} := 221.659 \text{ } ^\circ\text{C}$ [Successive approximation with first guess as 230 degC (i.e. as per MCR data)]

Nodal Temperatures:

$$T_{fg.A} := T_{fg.in} = 338 \cdot ^\circ\text{C}$$

$$T_{fg.B} := 301.492 \text{ } ^\circ\text{C}$$

Intermediate temperatures:

$$T_{fg.C} := 262.674 \text{ } ^\circ\text{C}$$

$$T_{fg.D} := T_{fg.out} = 221.659 \cdot ^\circ\text{C}$$

Note: Red Highlights indicate variables that are updated iteratively.

Note: Expressions with borders are inputs that can be changed when verifying/validating different operating points based on a different boiler load

HX1 Fluegas Fluid Properties

Average fluegas-side Temperature and Pressure:

$$T_{fg,avg,HX1} := \frac{T_{fg,C} + T_{fg,D}}{2} = 242.166 \cdot ^\circ\text{C} \quad P_{fg,avg,HX1} := \frac{P_{fg,C} + P_{fg,D}}{2} = 791.321 \cdot \text{kPa}$$

Fluegas Specific Heat Capacity evaluated at $T_{air,avg}$ and $P_{air,avg}$:

$$c_{p,fg,HX1} := C_{p,fluid}("air", P_{fg,avg,HX1}, T_{fg,avg,HX1}, "", "", "", "") = 1.036 \cdot \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

Fluegas Density evaluated at $T_{air,avg}$ and $P_{air,avg}$:

$$\rho_{fg,HX1} := \rho_{fluid}("air", P_{fg,avg,HX1}, T_{fg,avg,HX1}, "", "", "", "") = 5.335 \cdot \frac{\text{kg}}{\text{m}^3}$$

Fluegas Thermal Conductivity:

$$\lambda_{fg,HX1} := \lambda_{fluid}("air", P_{fg,avg,HX1}, T_{fg,avg,HX1}, "") = 0.041 \cdot \frac{\text{W}}{\text{m} \cdot \text{K}}$$

Fluegas Dynamic Viscosity:

$$\mu_{fg,HX1} := \mu_{fluid}("air", P_{fg,avg,HX1}, T_{fg,avg,HX1}, "") = 0 \cdot \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Fluegas Kinematic Viscosity:

$$\nu_{fg,HX1} := \frac{\mu_{fg,HX1}}{\rho_{fg,HX1}} = 0 \cdot \frac{\text{m}^2}{\text{s}}$$

Fluegas Prandtl Number:

$$Pr_{fg,HX1} := Pr_{fluid}("air", P_{fg,avg,HX1}, T_{fg,avg,HX1}, "") = 0.701$$

Total fluegas-side heat transfer rate:

$$\dot{Q}_{dot,fg,HX1} := \dot{m}_{dot,fg,in} \cdot c_{p,fg,HX1} \cdot (T_{fg,C} - T_{fg,D}) = 82.427 \cdot \text{kW}$$

HX2 Fluegas Fluid Properties

Average fluegas-side Temperature and Pressure:

$$T_{fg,avg,HX2} := \frac{T_{fg,B} + T_{fg,C}}{2} = 282.083 \cdot ^\circ\text{C} \quad P_{fg,avg,HX2} := \frac{P_{fg,B} + P_{fg,C}}{2} = 791.322 \cdot \text{kPa}$$

Fluegas Specific Heat Capacity evaluated at $T_{air,avg}$ and $P_{air,avg}$:

$$c_{p,fg,HX2} := C_{p,fluid}("air", P_{fg,avg,HX2}, T_{fg,avg,HX2}, "", "", "", "") = 1.044 \cdot \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

Fluegas Density evaluated at $T_{air,avg}$ and $P_{air,avg}$:

$$\rho_{fg,HX2} := \rho_{fluid}("air", P_{fg,avg,HX2}, T_{fg,avg,HX2}, "", "", "", "") = 4.951 \cdot \frac{\text{kg}}{\text{m}^3}$$

Fluegas Thermal Conductivity:

$$\lambda_{fg,HX2} := \lambda_{fluid}("air", P_{fg,avg,HX2}, T_{fg,avg,HX2}, "") = 0.043 \cdot \frac{\text{W}}{\text{m} \cdot \text{K}}$$

Fluegas Dynamic Viscosity:

$$\mu_{fg,HX2} := \mu_{fluid}("air", P_{fg,avg,HX2}, T_{fg,avg,HX2}, "") = 0 \cdot \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Fluegas Kinematic Viscosity:

$$\nu_{fg,HX2} := \frac{\mu_{fg,HX2}}{\rho_{fg,HX2}} = 0 \cdot \frac{\text{m}^2}{\text{s}}$$

Fluegas Prandtl Number:

$$Pr_{fg,HX2} := Pr_{fluid}("air", P_{fg,avg,HX2}, T_{fg,avg,HX2}, "") = 0.702$$

Total fluegas-side heat transfer rate:

$$\dot{Q}_{dot,fg,HX2} := \dot{m}_{dot,fg,in} \cdot c_{p,fg,HX2} \cdot (T_{fg,B} - T_{fg,C}) = 78.601 \cdot \text{kW}$$

HX3 Fluegas Fluid Properties

Average fluegas-side Temperature and Pressure:

$$T_{fg,avg,HX3} := \frac{T_{fg,A} + T_{fg,B}}{2} = 319.746 \cdot ^\circ\text{C} \quad P_{fg,avg,HX3} := \frac{P_{fg,A} + P_{fg,B}}{2} = 791.324 \cdot \text{kPa}$$

Fluegas Specific Heat Capacity evaluated at $T_{air,avg}$ and $P_{air,avg}$:

$$c_{p,fg,HX3} := C_{p,fluid}("air", P_{fg,avg,HX3}, T_{fg,avg,HX3}, "", "", "", "") = 1.052 \cdot \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

Fluegas Density evaluated at $T_{air,avg}$ and $P_{air,avg}$:

$$\rho_{fg,HX3} := \rho_{fluid}("air", P_{fg,avg,HX3}, T_{fg,avg,HX3}, "", "", "", "") = 4.636 \cdot \frac{\text{kg}}{\text{m}^3}$$

Fluegas Thermal Conductivity:

$$\lambda_{fg,HX3} := \lambda_{fluid}("air", P_{fg,avg,HX3}, T_{fg,avg,HX3}, "") = 0.046 \cdot \frac{\text{W}}{\text{m} \cdot \text{K}}$$

Fluegas Dynamic Viscosity:

$$\mu_{fg,HX3} := \mu_{fluid}("air", P_{fg,avg,HX3}, T_{fg,avg,HX3}, "") = 0 \cdot \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Fluegas Kinematic Viscosity:

$$\nu_{fg,HX3} := \frac{\mu_{fg,HX3}}{\rho_{fg,HX3}} = 0 \cdot \frac{\text{m}^2}{\text{s}}$$

Fluegas Prandtl Number:

$$Pr_{fg,HX3} := Pr_{fluid}("air", P_{fg,avg,HX3}, T_{fg,avg,HX3}, "") = 0.704$$

Total fluegas-side heat transfer rate:

$$\dot{Q}_{dot,fg,HX3} := \dot{m}_{dot,fg,in} \cdot c_{p,fg,HX3} \cdot (T_{fg,A} - T_{fg,B}) = 74.493 \cdot \text{kW}$$

DETERMINING THE FLUEGAS HEAT TRANSFER COEFFICIENT FOR FLOW INSIDE TUBES (Gnielinski correlation [5]):

HX1 Fluegas Heat Transfer Coefficient

Total fluegas mass flowrate:

$$\dot{m}_{\text{dot.fg.in}} = 1.94 \frac{\text{kg}}{\text{s}}$$

Fluegas mass flowrate per tube:

$$\dot{m}_{\text{dot.fg.pt}} := \frac{\dot{m}_{\text{dot.fg.in}}}{N_{\text{tubes}}} = 0.016 \frac{\text{kg}}{\text{s}}$$

Fluegas velocity per tube:

$$v_{\text{fg.pt.HX1}} := \frac{\dot{m}_{\text{dot.fg.pt}}}{\rho_{\text{fg.HX1}} \cdot \left(\frac{\pi}{4} \cdot d_{\text{i.tube}}^2 \right)} = 1.09 \frac{\text{m}}{\text{s}}$$

Fluegas Reynolds Number for fluegas flow inside a tube:

$$Re_{\text{d.i.tube.HX1}} := \frac{\rho_{\text{fg.HX1}} \cdot v_{\text{fg.pt.HX1}} \cdot d_{\text{i.tube}}}{\mu_{\text{fg.HX1}}} \quad Re_{\text{fg.HX1}} := Re_{\text{d.i.tube.HX1}} = 12470.934$$

Surface Roughness (sheet metal, new) as per [5]

$$\varepsilon_{\text{steel}} := 100 \mu\text{m} \quad \text{For sheet metal, new (max)} \quad [5]$$

Friction factor:

$$f_{\text{HX1}} := \left(\frac{1}{-2 \cdot \log \left(\frac{\frac{\varepsilon_{\text{steel}}}{d_{\text{i.tube}}}}{3.7} + \frac{5.74}{Re_{\text{d.i.tube.HX1}}^{0.9}} \right)} \right)^2 = 0.032 \quad \text{Smooth to rough correlation Swamee and Jain, (1976)} \quad [6]$$

Fluegas Nusselt Number:

$$Nu_{\text{fg.HX1}} := \frac{\left(\frac{f_{\text{HX1}}}{8} \right) \cdot (Re_{\text{fg.HX1}} - 1000) \cdot Pr_{\text{fg.HX1}}}{1 + 12.7 \cdot \left(\frac{f_{\text{HX1}}}{8} \right)^{0.5} \cdot \left(Pr_{\text{fg.HX1}}^{\frac{2}{3}} - 1 \right)} = 38.985 \quad [5]$$

Fluegas Heat Transfer Coefficient:

$$h_{\text{fg.HX1}} := \frac{Nu_{\text{fg.HX1}} \cdot \lambda_{\text{fg.HX1}}}{d_{\text{i.tube}}} = 26.877 \cdot \frac{\text{W}}{\text{m}^2 \text{K}}$$

HX2 Fluegas Heat Transfer Coefficient

Total fluegas mass flowrate:

$$\dot{m}_{\text{fg.in}} = 1.94 \frac{\text{kg}}{\text{s}}$$

Fluegas mass flowrate per tube:

$$\dot{m}_{\text{fg.pt}} = 0.016 \frac{\text{kg}}{\text{s}}$$

Fluegas velocity per tube:

$$v_{\text{fg.pt.HX2}} := \frac{\dot{m}_{\text{fg.pt}}}{\rho_{\text{fg.HX2}} \cdot \left(\frac{\pi}{4} \cdot d_{\text{i.tube}}^2 \right)} = 1.174 \frac{\text{m}}{\text{s}}$$

Fluegas Reynolds Number for fluegas flow inside a tube:

$$Re_{\text{d.i.tube.HX2}} := \frac{\rho_{\text{fg.HX2}} \cdot v_{\text{fg.pt.HX2}} \cdot d_{\text{i.tube}}}{\mu_{\text{fg.HX2}}} \quad Re_{\text{fg.HX2}} := Re_{\text{d.i.tube.HX2}} = 11838.698$$

Friction factor:

$$f_{\text{HX2}} := \left(\frac{1}{-2 \cdot \log \left(\frac{\frac{\epsilon_{\text{steel}}}{d_{\text{i.tube}}}}{3.7} + \frac{5.74}{Re_{\text{d.i.tube.HX2}}^{0.9}} \right)} \right)^2 = 0.033$$

Smooth to rough correlation
Swamee and Jain, (1976) [6]

Fluegas Nusselt Number:

$$Nu_{\text{fg.HX2}} := \frac{\left(\frac{f_{\text{HX2}}}{8} \right) \cdot (Re_{\text{fg.HX2}} - 1000) \cdot Pr_{\text{fg.HX2}}}{1 + 12.7 \cdot \left(\frac{f_{\text{HX2}}}{8} \right)^{0.5} \cdot \left(Pr_{\text{fg.HX2}}^{\frac{2}{3}} - 1 \right)} = 37.307$$

[5]

Fluegas Heat Transfer Coefficient:

$$h_{\text{fg.HX2}} := \frac{Nu_{\text{fg.HX2}} \cdot \lambda_{\text{fg.HX2}}}{d_{\text{i.tube}}} = 27.245 \cdot \frac{\text{W}}{\text{m}^2 \text{K}}$$

HX3 Fluegas Heat Transfer Coefficient

Total fluegas mass flowrate:

$$\dot{m}_{\text{dot.fg.in}} = 1.94 \frac{\text{kg}}{\text{s}}$$

Fluegas mass flowrate per tube:

$$\dot{m}_{\text{dot.fg.pt}} = 0.016 \frac{\text{kg}}{\text{s}}$$

Fluegas velocity per tube:

$$v_{\text{fg.pt.HX3}} := \frac{\dot{m}_{\text{dot.fg.pt}}}{\rho_{\text{fg.HX3}} \cdot \left(\frac{\pi}{4} \cdot d_{\text{i.tube}}^2 \right)} = 1.254 \frac{\text{m}}{\text{s}}$$

Fluegas Reynolds Number for fluegas flow inside a tube:

$$Re_{\text{d.i.tube.HX3}} := \frac{\rho_{\text{fg.HX3}} \cdot v_{\text{fg.pt.HX3}} \cdot d_{\text{i.tube}}}{\mu_{\text{fg.HX3}}} \quad Re_{\text{fg.HX3}} := Re_{\text{d.i.tube.HX3}} = 11314.498$$

Friction factor:

$$f_{\text{HX3}} := \left(\frac{1}{-2 \cdot \log \left(\frac{\frac{\epsilon_{\text{steel}}}{d_{\text{i.tube}}}}{3.7} + \frac{5.74}{Re_{\text{d.i.tube.HX3}}^{0.9}} \right)} \right)^2 = 0.033$$

Smooth to rough correlation
Swamee and Jain, (1976) [6]

Fluegas Nusselt Number:

$$Nu_{\text{fg.HX3}} := \frac{\left(\frac{f_{\text{HX3}}}{8} \right) \cdot (Re_{\text{fg.HX3}} - 1000) \cdot Pr_{\text{fg.HX3}}}{1 + 12.7 \cdot \left(\frac{f_{\text{HX3}}}{8} \right)^{0.5} \cdot \left(Pr_{\text{fg.HX3}}^{\frac{2}{3}} - 1 \right)} = 35.919$$

[5]

Fluegas Heat Transfer Coefficient:

$$h_{\text{fg.HX3}} := \frac{Nu_{\text{fg.HX3}} \cdot \lambda_{\text{fg.HX3}}}{d_{\text{i.tube}}} = 27.585 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Properties of Tube Material:

Thermal conductivity of AISI 1010 at 300K: $\lambda_{\text{steel}} := 63.9 \frac{\text{W}}{\text{m} \cdot \text{K}}$ [5]

Thermal Conductivity of Tubes: $\lambda_{\text{tube}} := \lambda_{\text{steel}} = 63.9 \frac{\text{W}}{\text{m} \cdot \text{K}}$

Steel density: $\rho_{\text{steel}} = 7832 \frac{\text{kg}}{\text{m}^3}$ [5]

Steel specific heat capacity: $c_{\text{p.steel}} := 434 \frac{\text{J}}{\text{kg} \cdot \text{K}}$ [5]

HX1 UA Value

Thermal Reistances:

Thermal resistance due to the fluegas boundary layer on the inner tube wall:

$$R_{i.HX1} := h_{fg.HX1} = 26.877 \cdot \frac{W}{m^2 \cdot K}$$

Thermal resistance due to fouling on the inside of the tubes (fluegas-side):

$$R_{f.i.HX1} := 0 \frac{m^2 \cdot K}{W} \quad [\text{Assuming no fouling for now.}]$$

Thermal resistance due to wall thermal conductivity:

$$R_{w.HX1} := \frac{\ln\left(\frac{d_{o.tube}}{d_{i.tube}}\right)}{2\pi \cdot \lambda_{tube} \cdot \frac{L_{tube}}{3} \cdot N_{tubes}} = 0 \cdot \frac{K}{W}$$

Thermal resistance due to fouling on the outside of the tubes (air-side):

$$R_{f.o.HX1} := 0 \frac{m^2 \cdot K}{W} \quad [\text{Assuming no fouling for now.}]$$

Thermal resistance due to air boundary layer on the outer tube wall:

$$R_{o.HX1} := h_{air.HX1} = 70.178 \cdot \frac{W}{m^2 \cdot K}$$

Inside tube surface efficiency (fluegas-side): $\eta_{fg.HX1} := 1$ [Unity for no fins.]

Outside tube surface efficiency (air-side): $\eta_{air.HX1} := 1$ [Unity for no fins.]

Therefore we can calculate the UA value as:

$$UA_{HX1} := \left(\frac{1}{\eta_{fg.HX1} \cdot R_{i.HX1} \cdot SA_{i.tube.tot.HX}} + \frac{R_{f.i.HX1}}{\eta_{fg.HX1} \cdot SA_{i.tube.tot.HX}} + R_{w.HX1} + \frac{R_{f.o.HX1}}{\eta_{air.HX1} \cdot SA_{o.tube.tot.HX}} + \frac{1}{\eta_{air.HX1} \cdot R_{o.HX1} \cdot SA_{o.tube.tot.HX}} \right)^{-1}$$

$$UA_{HX1} = 0.443 \cdot \frac{kW}{K}$$

HX2 UA Value

Thermal Reistances:

Thermal resistance due to the fluegas boundary layer on the inner tube wall:

$$R_{i.HX2} := h_{fg.HX2} = 27.245 \cdot \frac{W}{m^2 \cdot K}$$

Thermal resistance due to fouling on the inside of the tubes (fluegas-side):

$$R_{f.i.HX2} := 0 \frac{m^2 \cdot K}{W} \quad [\text{Assuming no fouling for now.}]$$

Thermal resistance due to wall thermal conductivity:

$$R_{w.HX2} := \frac{\ln\left(\frac{d_{o.tube}}{d_{i.tube}}\right)}{2\pi \cdot \lambda_{tube} \cdot \frac{L_{tube}}{3} \cdot N_{tubes}} = 0 \cdot \frac{K}{W}$$

Thermal resistance due to fouling on the outside of the tubes (air-side):

$$R_{f.o.HX2} := 0 \frac{m^2 \cdot K}{W} \quad [\text{Assuming no fouling for now.}]$$

Thermal resistance due to air boundary layer on the outer tube wall:

$$R_{o.HX2} := h_{air.HX2} = 74.79 \cdot \frac{W}{m^2 \cdot K}$$

Inside tube surface efficiency (fluegas-side): $\eta_{fg.HX2} := 1$ [Unity for no fins.]

Outside tube surface efficiency (air-side): $\eta_{air.HX2} := 1$ [Unity for no fins.]

Therefore we can calculate the UA value as:

$$UA_{HX2} := \left(\frac{1}{\eta_{fg.HX2} \cdot R_{i.HX2} \cdot SA_{i.tube.tot.HX}} + \frac{R_{f.i.HX2}}{\eta_{fg.HX2} \cdot SA_{i.tube.tot.HX}} + R_{w.HX2} + \frac{R_{f.o.HX2}}{\eta_{air.HX2} \cdot SA_{o.tube.tot.HX}} + \frac{1}{\eta_{air.HX2} \cdot R_{o.HX2} \cdot SA_{o.tube.tot.HX}} \right)^{-1}$$

$$UA_{HX2} = 0.455 \cdot \frac{kW}{K}$$

HX3 UA Value

Thermal Reistances:

Thermal resistance due to the fluegas boundary layer on the inner tube wall:

$$R_{i.HX3} := h_{fg.HX3} = 27.585 \cdot \frac{W}{m^2 \cdot K}$$

Thermal resitance due to fouling on the inside of the tubes (fluegas-side):

$$R_{f.i.HX3} := 0 \frac{m^2 \cdot K}{W} \quad [\text{Assuming no fouling for now.}]$$

Thermal resistance due to wall thermal conductivity:

$$R_{w.HX3} := \frac{\ln\left(\frac{d_{o.tube}}{d_{i.tube}}\right)}{2\pi \cdot \lambda_{tube} \cdot \frac{L_{tube}}{3} \cdot N_{tubes}} = 0 \cdot \frac{K}{W}$$

Thermal resitance due to fouling on the outside of the tubes (air-side):

$$R_{f.o.HX3} := 0 \frac{m^2 \cdot K}{W} \quad [\text{Assuming no fouling for now.}]$$

Thermal resistance due to air boundary layer on the outer tube wall:

$$R_{o.HX3} := h_{air.HX3} = 78.761 \cdot \frac{W}{m^2 \cdot K}$$

Inside tube surface efficiency (fluegas-side): $\eta_{fg.HX3} := 1$ [Unity for no fins.]

Outside tube surface efficiency (air-side): $\eta_{air.HX3} := 1$ [Unity for no fins.]

Therefore we can calculate the UA value as:

$$UA_{HX3} := \left(\frac{1}{\eta_{fg.HX3} \cdot R_{i.HX3} \cdot SA_{i.tube.tot.HX}} + \frac{R_{f.i.HX3}}{\eta_{fg.HX3} \cdot SA_{i.tube.tot.HX}} + R_{w.HX3} + \frac{R_{f.o.HX3}}{\eta_{air.HX3} \cdot SA_{o.tube.tot.HX}} + \frac{1}{\eta_{air.HX3} \cdot R_{o.HX3} \cdot SA_{o.tube.tot.HX}} \right)^{-1}$$

$$UA_{HX3} = 0.466 \cdot \frac{kW}{K}$$

HX1 E-NTU For Cross Flow over a Tube Bank:

The hot fluid is the fluegas and is denoted by the subscript H for the foregoing analysis.
The cold fluid is the air and is denoted by the subscripts C for the foregoing analysis.

Heat capacity of the Hot stream:

$$C_{H.HX1} := \dot{m}_{\text{dot.fg.in}} \cdot c_{p.\text{fg.HX1}} = 2.01 \cdot \frac{\text{kW}}{\text{K}}$$

Heat Capacity of the Cold stream:

$$C_{C.HX1} := \dot{m}_{\text{dot.air.in}} \cdot c_{p.\text{air.HX1}} = 1.509 \cdot \frac{\text{kW}}{\text{K}}$$

Minimum heat capacity:

$$C_{\min.HX1} := \min(C_{H.HX1}, C_{C.HX1}) = 1.509 \cdot \frac{\text{kW}}{\text{K}}$$

Maximum heat capacity:

$$C_{\max.HX1} := \max(C_{H.HX1}, C_{C.HX1}) = 2.01 \cdot \frac{\text{kW}}{\text{K}}$$

Number of Transfer Units:

$$NTU_{HX1} := \frac{UA_{HX1}}{C_{\min.HX1}} = 0.294$$

Heat capacity ratio:

$$C_{HX1} := \frac{C_{\min.HX1}}{C_{\max.HX1}} = 0.751$$

E-NTU correlation for cross-flow (single-pass), both fluids unmixed:

$$\epsilon_{HX1} := 1 - e^{-\left[-\left(\frac{1}{C_{HX1}} \right) \cdot \left[1 - e^{(-C_{HX1} \cdot NTU_{HX1})} \right] \right]} \quad [7]$$

$$\epsilon_{HX1} = 0.232$$

HX2 E-NTU For Cross Flow over a Tube Bank:

The hot fluid is the fluegas and is denoted by the subscript H for the foregoing analysis.
The cold fluid is the air and is denoted by the subscripts C for the foregoing analysis.

Heat capacity of the Hot stream:

$$C_{H.HX2} := \dot{m}_{\text{dot.fg.in}} \cdot c_{p.\text{fg.HX2}} = 2.025 \cdot \frac{\text{kW}}{\text{K}}$$

Heat Capacity of the Cold stream:

$$C_{C.HX2} := \dot{m}_{\text{dot.air.in}} \cdot c_{p.\text{air.HX2}} = 1.515 \cdot \frac{\text{kW}}{\text{K}}$$

Minimum heat capacity:

$$C_{\min.HX2} := \min(C_{H.HX2}, C_{C.HX2}) = 1.515 \cdot \frac{\text{kW}}{\text{K}}$$

Maximum heat capacity:

$$C_{\max.HX2} := \max(C_{H.HX2}, C_{C.HX2}) = 2.025 \cdot \frac{\text{kW}}{\text{K}}$$

Number of Transfer Units:

$$NTU_{HX2} := \frac{UA_{HX2}}{C_{\min.HX2}} = 0.301$$

Heat capacity ratio:

$$C_{HX2} := \frac{C_{\min.HX2}}{C_{\max.HX2}} = 0.748$$

E-NTU correlation for cross-flow (single-pass), both fluids unmixed:

$$\varepsilon_{HX2} := 1 - e^{-\left[-\left(\frac{1}{C_{HX2}} \right) \cdot \left[1 - e^{(-C_{HX2} \cdot NTU_{HX2})} \right] \right]} \quad [7]$$

$$\varepsilon_{HX2} = 0.236$$

HX3 E-NTU For Cross Flow over a Tube Bank:

The hot fluid is the fluegas and is denoted by the subscript H for the foregoing analysis.
The cold fluid is the air and is denoted by the subscripts C for the foregoing analysis.

Heat capacity of the Hot stream:

$$C_{H.HX3} := \dot{m}_{\text{dot.fg.in}} \cdot c_{p.\text{fg.HX3}} = 2.04 \cdot \frac{\text{kW}}{\text{K}}$$

Heat Capacity of the Cold stream:

$$C_{C.HX3} := \dot{m}_{\text{dot.air.in}} \cdot c_{p.\text{air.HX3}} = 1.525 \cdot \frac{\text{kW}}{\text{K}}$$

Minimum heat capacity:

$$C_{\min.HX3} := \min(C_{H.HX3}, C_{C.HX3}) = 1.525 \cdot \frac{\text{kW}}{\text{K}}$$

Maximum heat capacity:

$$C_{\max.HX3} := \max(C_{H.HX3}, C_{C.HX3}) = 2.04 \cdot \frac{\text{kW}}{\text{K}}$$

Number of Transfer Units:

$$NTU_{HX3} := \frac{UA_{HX3}}{C_{\min.HX3}} = 0.305$$

Heat capacity ratio:

$$C_{HX3} := \frac{C_{\min.HX3}}{C_{\max.HX3}} = 0.747$$

E-NTU correlation for cross-flow (single-pass), both fluids unmixed:

$$\varepsilon_{HX3} := 1 - e^{-\left[-\left(\frac{1}{C_{HX3}} \right) \cdot \left[1 - e^{(-C_{HX3} \cdot NTU_{HX3})} \right] \right]} \quad [7]$$

$$\varepsilon_{HX3} = 0.239$$

HX1 Results

Hot-side Temperatures:

$$T_{H.in.HX1} := T_{fg.C} = 262.674^{\circ}\text{C}$$

Cold-side Temperatures:

$$T_{C.in.HX1} := T_{air.0} = 27^{\circ}\text{C}$$

$$Q_{dot.E.NTU.HX1} := \epsilon_{HX1} \cdot C_{min.HX1} \cdot (T_{H.in.HX1} - T_{C.in.HX1}) = 82.426 \cdot \text{kW}$$

$$Q_{dot.fg.HX1} = 82.427 \cdot \text{kW}$$

$$Q_{dot.air.HX1} = 82.426 \cdot \text{kW}$$

Outlet temperatures, %error based on E-NTU method for single pass cross flow (c_min mixed and c_max unmixed) and heat transfer rate %error:

Fluegas:

$$T_{H.out.HX1} := \frac{m_{dot.fg.in} \cdot c_{p.fg.HX1} \cdot T_{H.in.HX1} - Q_{dot.E.NTU.HX1}}{m_{dot.fg.in} \cdot c_{p.fg.HX1}} = 221.659^{\circ}\text{C}$$

$$\text{Error}_{T_{fg.out.HX1}} := \frac{T_{H.out.HX1} - T_{fg.D}}{T_{fg.D}} = 0\%$$

$$Q_{dot.fg.HX1} = 82.427 \cdot \text{kW}$$

$$\text{Error}_{Q_{dot.fg.HX1}} := \frac{Q_{dot.E.NTU.HX1} - Q_{dot.fg.HX1}}{Q_{dot.fg.HX1}} = -0.001\%$$

Air:

$$T_{C.out.HX1} := \frac{Q_{dot.E.NTU.HX1} + m_{dot.air.in} \cdot c_{p.air.HX1} \cdot T_{C.in.HX1}}{m_{dot.air.in} \cdot c_{p.air.HX1}} = 81.633^{\circ}\text{C}$$

$$\text{Error}_{T_{air.out.HX1}} := \frac{T_{C.out.HX1} - T_{air.1}}{T_{air.1}} = 0\%$$

$$Q_{dot.air.HX1} = 82.426 \cdot \text{kW}$$

$$\text{Error}_{Q_{dot.air.HX1}} := \frac{Q_{dot.E.NTU.HX1} - Q_{dot.air.HX1}}{Q_{dot.air.HX1}} = 0.001\%$$

Check Convergence:

$$T_{fg.D} = 221.659^{\circ}\text{C}$$

$$T_{air.1} = 81.633^{\circ}\text{C}$$

$$T_{H.out.HX1} = 221.659^{\circ}\text{C}$$

$$T_{C.out.HX1} = 81.633^{\circ}\text{C}$$

$$\text{Error}_{T_{fg.out.HX1}} = 0\%$$

$$\text{Error}_{T_{air.out.HX1}} = 0\%$$

$$\text{Error}_{Q_{dot.fg.HX1}} = -0.001\%$$

$$\text{Error}_{Q_{dot.air.HX1}} = 0.001\%$$

HX2 Results

Hot-side Temperatures:

$$T_{H.in.HX2} := T_{fg.B} = 301.492^{\circ}\text{C}$$

$$Q_{dot.E.NTU.HX2} := \epsilon_{HX2} \cdot C_{min.HX2} \cdot (T_{H.in.HX2} - T_{C.in.HX2}) = 78.601\text{ kW}$$

$$Q_{dot.fg.HX2} = 78.601\text{ kW}$$

$$Q_{dot.air.HX2} = 78.601\text{ kW}$$

Cold-side Temperatures:

$$T_{C.in.HX2} := T_{air.1} = 81.633^{\circ}\text{C}$$

Outlet temperatures, %error based on E-NTU method for single pass cross flow (c_min mixed and c_max unmixed) and heat transfer rate %error:

Fluegas:

$$T_{H.out.HX2} := \frac{m_{dot.fg.in} \cdot c_{p.fg.HX2} \cdot T_{H.in.HX2} - Q_{dot.E.NTU.HX2}}{m_{dot.fg.in} \cdot c_{p.fg.HX2}} = 262.674^{\circ}\text{C}$$

$$\text{Error}_{T_{fg.out.HX2}} := \frac{T_{H.out.HX2} - T_{fg.C}}{T_{fg.C}} = 0\%$$

$$Q_{dot.fg.HX2} = 78.601\text{ kW}$$

$$\text{Error}_{Q_{dot.fg.HX2}} := \frac{Q_{dot.E.NTU.HX2} - Q_{dot.fg.HX2}}{Q_{dot.fg.HX2}} = -0.001\%$$

Air:

$$T_{C.out.HX2} := \frac{Q_{dot.E.NTU.HX2} + m_{dot.air.in} \cdot c_{p.air.HX2} \cdot T_{C.in.HX2}}{m_{dot.air.in} \cdot c_{p.air.HX2}} = 133.509^{\circ}\text{C}$$

$$\text{Error}_{T_{air.out.HX2}} := \frac{T_{C.out.HX2} - T_{air.2}}{T_{air.2}} = -0\%$$

$$Q_{dot.air.HX2} = 78.601\text{ kW}$$

$$\text{Error}_{Q_{dot.air.HX2}} := \frac{Q_{dot.E.NTU.HX2} - Q_{dot.air.HX2}}{Q_{dot.air.HX2}} = -0.001\%$$

Check Convergence:

$$T_{fg.C} = 262.674^{\circ}\text{C}$$

$$T_{air.2} = 133.509^{\circ}\text{C}$$

$$T_{H.out.HX2} = 262.674^{\circ}\text{C}$$

$$T_{C.out.HX2} = 133.509^{\circ}\text{C}$$

$$\text{Error}_{T_{fg.out.HX2}} = 0\%$$

$$\text{Error}_{T_{air.out.HX2}} = -0\%$$

$$\text{Error}_{Q_{dot.fg.HX2}} = -0.001\%$$

$$\text{Error}_{Q_{dot.air.HX2}} = -0.001\%$$

HX3 Results

Hot-side Temperatures:

$$T_{H.in.HX3} := T_{fg.A} = 338.^{\circ}\text{C}$$

Cold-side Temperatures:

$$T_{C.in.HX3} := T_{air.2} = 133.509.^{\circ}\text{C}$$

$$Q_{dot.E.NTU.HX3} := \epsilon_{HX3} \cdot C_{min.HX3} \cdot (T_{H.in.HX3} - T_{C.in.HX3}) = 74.494 \cdot \text{kW}$$

$$Q_{dot.fg.HX3} = 74.493 \cdot \text{kW}$$

$$Q_{dot.air.HX3} = 74.494 \cdot \text{kW}$$

Outlet temperatures, %error based on E-NTU method for single pass cross flow (c_min mixed and c_max unmixed) and heat transfer rate %error:

Fluegas:

$$T_{H.out.HX3} := \frac{\dot{m}_{dot.fg.in} \cdot c_{p.fg.HX3} \cdot T_{H.in.HX3} - Q_{dot.E.NTU.HX3}}{\dot{m}_{dot.fg.in} \cdot c_{p.fg.HX3}} = 301.492.^{\circ}\text{C}$$

$$\text{Error}_{T_{fg.out.HX3}} := \frac{T_{H.out.HX3} - T_{fg.B}}{T_{fg.B}} = -0.0\%$$

$$Q_{dot.fg.HX3} = 74.493 \cdot \text{kW}$$

$$\text{Error}_{Q_{dot.fg.HX3}} := \frac{Q_{dot.E.NTU.HX3} - Q_{dot.fg.HX3}}{Q_{dot.fg.HX3}} = 0.001\%$$

Air:

$$T_{C.out.HX3} := \frac{Q_{dot.E.NTU.HX3} + \dot{m}_{dot.air.in} \cdot c_{p.air.HX3} \cdot T_{C.in.HX3}}{\dot{m}_{dot.air.in} \cdot c_{p.air.HX3}} = 182.372.^{\circ}\text{C}$$

$$\text{Error}_{T_{air.out.HX3}} := \frac{T_{C.out.HX3} - T_{air.3}}{T_{air.3}} = 0.0\%$$

$$Q_{dot.air.HX3} = 74.494 \cdot \text{kW}$$

$$\text{Error}_{Q_{dot.air.HX3}} := \frac{Q_{dot.E.NTU.HX3} - Q_{dot.air.HX3}}{Q_{dot.air.HX3}} = 0.0\%$$

Check Convergence:

$$T_{fg.B} = 301.492.^{\circ}\text{C}$$

$$T_{air.3} = 182.372.^{\circ}\text{C}$$

$$T_{H.out.HX3} = 301.492.^{\circ}\text{C}$$

$$T_{C.out.HX3} = 182.372.^{\circ}\text{C}$$

$$\text{Error}_{T_{fg.out.HX3}} = -0.0\%$$

$$\text{Error}_{T_{air.out.HX3}} = 0.0\%$$

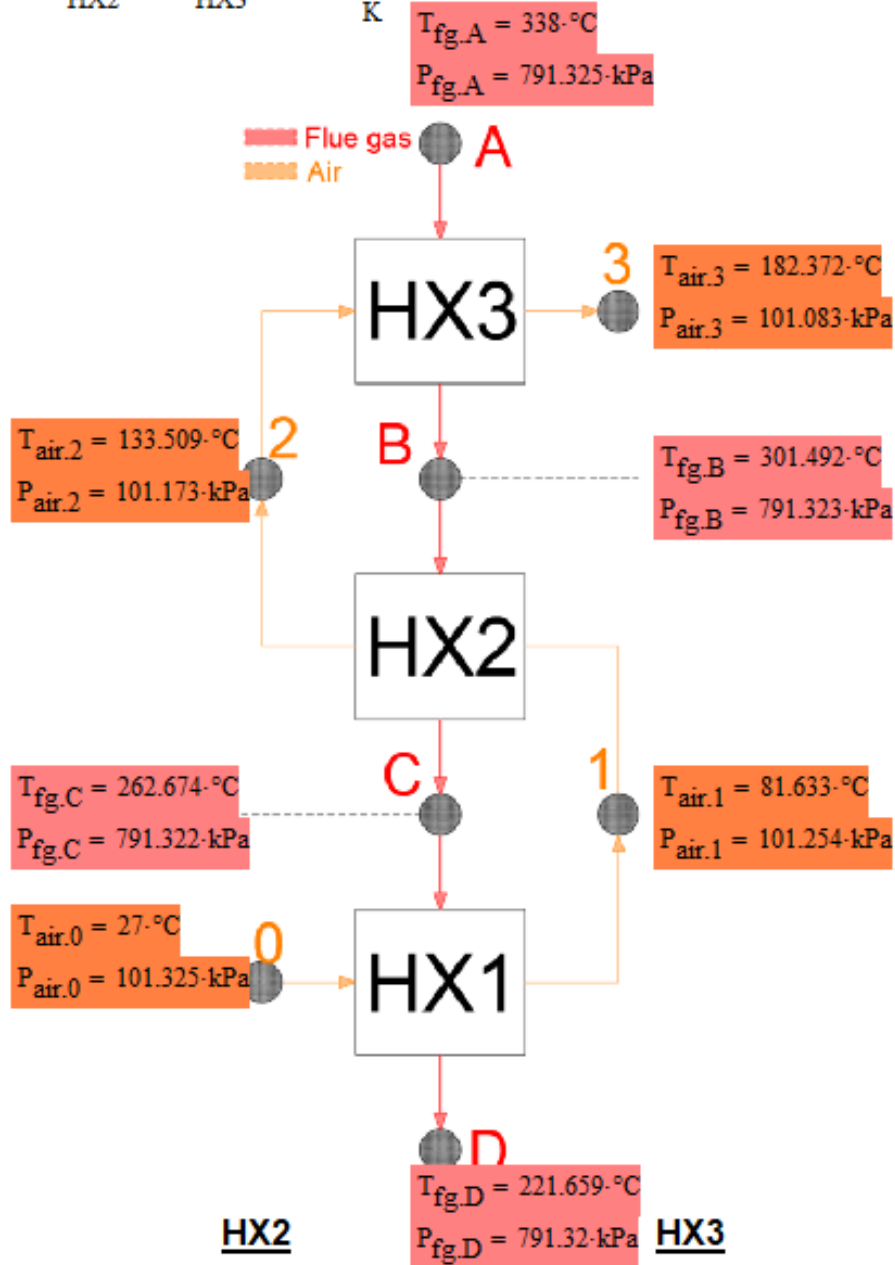
$$\text{Error}_{Q_{dot.fg.HX3}} = 0.001\%$$

$$\text{Error}_{Q_{dot.air.HX3}} = 0.0\%$$

SUMMARY OF CONVERGED RESULTS:

$$Q_{\dot{\text{AH}}} := Q_{\dot{\text{ENTU.HX1}}} + Q_{\dot{\text{ENTU.HX2}}} + Q_{\dot{\text{ENTU.HX3}}} = 235.521\text{-kW}$$

$$UA_{\text{AH}} := UA_{\text{HX1}} + UA_{\text{HX2}} + UA_{\text{HX3}} = 1.364 \cdot \frac{\text{kW}}{\text{K}}$$

**HX1**

$$h_{\text{air.HX1}} = 70.178 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

$$h_{\text{fg.HX1}} = 26.877 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

$$\epsilon_{\text{HX1}} = 0.232$$

$$Q_{\dot{\text{ENTU.HX1}}} = 82.426\text{-kW}$$

HX2

$$h_{\text{air.HX2}} = 74.79 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

$$h_{\text{fg.HX2}} = 27.245 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

$$\epsilon_{\text{HX2}} = 0.236$$

$$Q_{\dot{\text{ENTU.HX2}}} = 78.601\text{-kW}$$

HX3

$$h_{\text{air.HX3}} = 78.761 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

$$h_{\text{fg.HX3}} = 27.585 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

$$\epsilon_{\text{HX3}} = 0.239$$

$$Q_{\dot{\text{ENTU.HX3}}} = 74.494\text{-kW}$$

AIR HEATER - Flownex Model

The inputs used for the flownex model follow from the analytical model. The "quick scripts" employed in flownex to calculate the fin-side heat transfer coefficient follow the same methodology as the analytical model.

The difference is that the "quick-scripts" in flownex receive information such as fluid properties and mass flowrates from flow elements in the network with the use of data transfer links, whereas the analytical model requires the user evaluate the fluid properties and manually update in a successive approximation.

3 x *Finned-Tube Heat Exchanger* components connected in series were used to model the heat transfer in the air heater. These heat exchanger components were coupled with "quick-scripts" which employed the same heat transfer calculation methodology as presented in the analytical model.

The inputs to each of the three *Finned-Tube Heat Exchanger* components were identical, barring the heat transfer coefficient which would be calculated by the "quick-script" and then given to the fin-side of the *Finned-Tube Heat Exchanger* component via data transfer links from the script.

Flownex Inputs (Fin-side)

Fluids

Fluid data reference:

[For verification of analytical model]

"Air| Gases (Pure Fluids) {ATProM Database}"

[For integrated model with the fluid specified mass fractions equal to that of air, i.e. only O₂, N₂, and N₂ specified]

"Coal fired flue gas| Flue gas (Mixed Fluids) {ATProM Database}"

☐ Mass Fraction	
Specify mass fraction	Yes v
☐ Mass fractions	1 ...
SO ₂	0
Fly ash	0
Unburnt carbon	0
CO ₂ - Carbon Dioxide	0
N ₂ - Nitrogen	0.772219
NO - Nitric Oxide	0
O ₂ - Oxygen	0.204782
H ₂ O - Water	0.0229989
Steady-state continuity option	Calculate mass source v

Specification

Specification option: "General Specification" [To ensure compatibility with the analytical methodology]

Fin (Primary) Side Geometry

Flow Direction: "Counter" [Makes no difference since the flownex model uses 3 sub HX in series]

Heat transfer area: $SA_{o.tube.tot.HX} = 23.939 \text{ m}^2$ [External tube surface area for all tubes]

Area ratio: $Area_{Ratio} = 1.067$ [Outside SA to inside SA ratio]

Fluid volume: $V_{fluid.HX} = 0.404 \text{ m}^3$ [Volume occupied by the air]

Fluid path length: $L_{air.pass} := D_{AH} = 1.4 \text{ m}$ [Length of single air side pass]

Number of increments: $N_{inc} := 1$ [Insignificant difference if discretised further]

Inlet Area: $A_{inlet} := A_{HX} = 0.56 \text{ m}^2$ [Far-field air inlet area]

Inlet Area Ratio: $A_{in.ratio} := SCOF^{-1} = 0.282$ [Ratio of free-flow area to far-field area at inlet]

Outlet Area: $A_{outlet} := A_{HX} = 0.56 \text{ m}^2$ [Far-field air outlet area]

Outlet Area Ratio: $A_{out.ratio} := SCOF^{-1} = 0.282$
[Ratio of free-flow area to far-field area at outlet]

Material Data

Material option: "Select from data reference"

Mass: $m_{tube.mat.HX} = 363.169 \text{ kg}$ [Mass of all tube material]

Wall thickness: $t_{tube} = 2 \text{ mm}$

Material selection: "Carbon Steel - AISI 1010| Metals (Solids) {Master}"

Losses

Primary loss type: "Specify friction factor"

Friction factor: $f := 0.04$ [Insignificant but flownex requires a loss input to solve]

Heat Transfer

Convection Coefficient option: "Constant h"

Fin efficiency: $\eta_{fin} := 1$ [Finless tubes]

Fouling factor: $R_{f.o} := 0 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$ [Fouling not included - has small effects]

h: As calculated in script following the analytical methodology

Fixed Options

Fixed mass flow: No

Prevent flow reversal: No

Advanced

Specify advanced inputs: No

Flownex Inputs (Tube-side)

Fluids

Fluid data reference:

[For verification of analytical model]

"Air| Gases (Pure Fluids) {ATProM Database}"

[For intergrated model with the fluid speicified mass fractions equal to that of the combustion products.]

"Coal fired flue gas| Flue gas (Mixed Fluids) {ATProM Database}"

Note that the fluegas composition used were as a result of the combustion stoichiometric calculations and are shwon below:

☐ Mass Fraction	
Specify mass fraction	Yes
☐ Mass fractions	1
SO2	2.1739E-19
Fly ash	0.000982234
Unburnt carbon	0.00112597
CO2 - Carbon Dioxide	0.111231
N2 - Nitrogen	0.60349
NO - Nitric Oxide	0
O2 - Oxygen	0.0437381
H2O - Water	0.239433
Steady-state continuity option	Calculate mass source

Tube (Secondary) Side Geometry

Number of parallel circuits: $N_{\text{par.circ}} = 30$

Number of passes: $N_{\text{passes}} = 1$ [In accordance with flownex
conventions and general
arrangement]

Tube rows: $N_{\text{rows}} = 4$

Tube inside diameter: $d_{\text{i.tube}} = 59.5 \text{ mm}$

Total tube pass length: $L_{\text{tube.pass}} = \frac{L_{\text{tube}}}{3} = 1 \text{ m}$

Losses

Primary loss type: "Darcy Weisbach"

Roughness option: "Select from database"

Roughness selection: "Seamless steel tubes - New, unused - Max"

Roughness: $100 \mu\text{m}$ [As per above selection]

K actual inlet: 0 [Secondary losses not included]

K actual outlet: 0 [Secondary losses not included]

Heat Transfer

Convection coefficient option: "Calculate h"

Convection coefficient correlation selection: "Tube inside (Gnielinski)"

Fin efficiency: $\eta_{fin} = 1$ [Finless tubes on inside]Fouling factor: $R_{f,i} := 0 \frac{W}{m^2 \cdot K}$ [Fouling not included - has small effects]

Tube roughness option: "Obtain from flow component"

Fixed Options

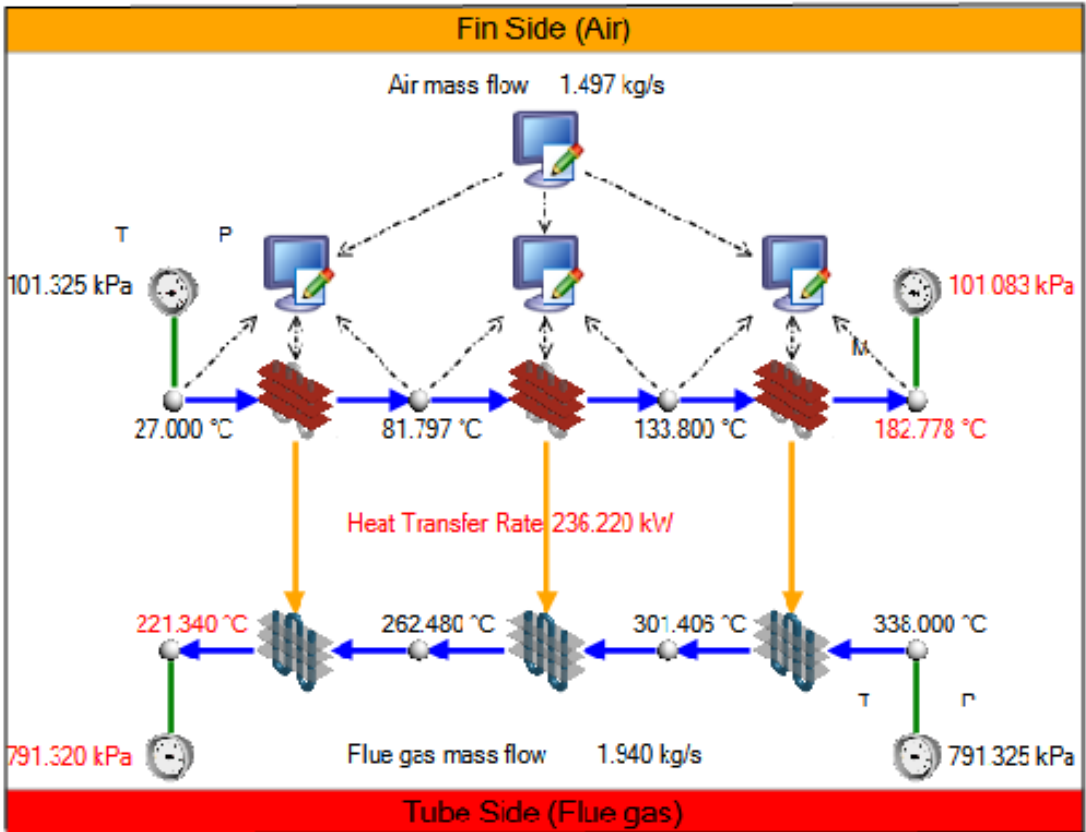
Fixed mass flow: No

Prevent flow reversal: No

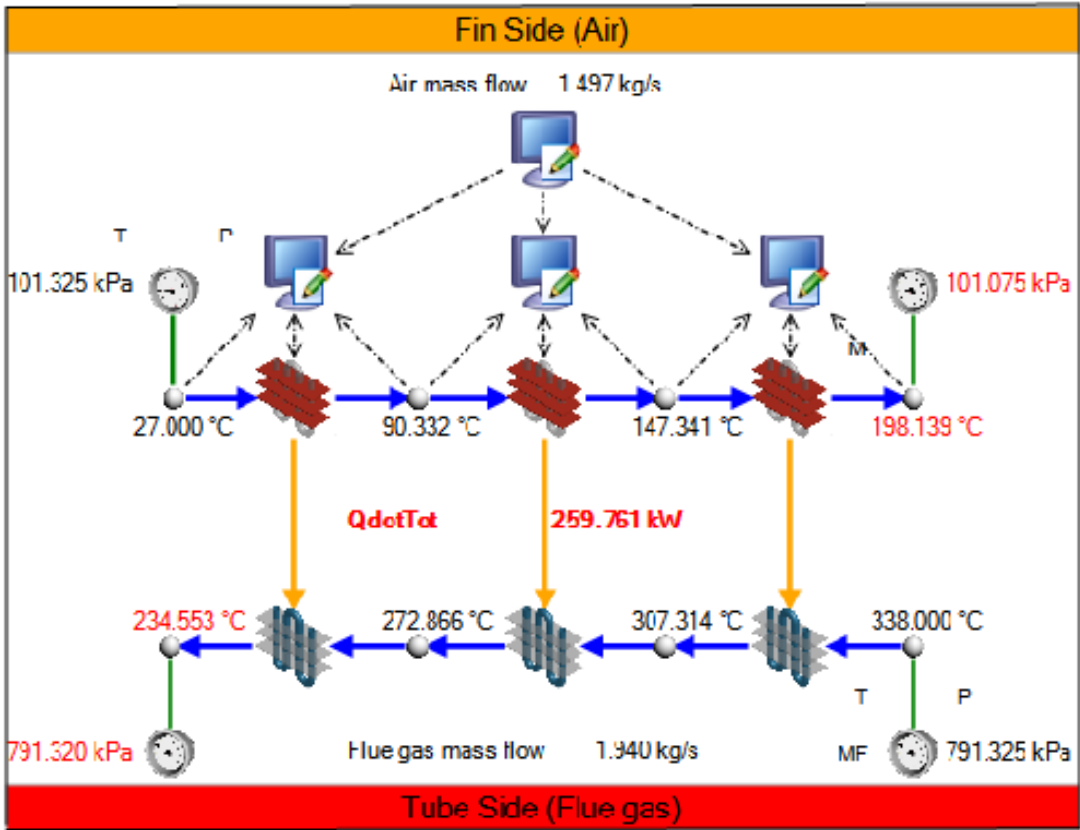
Advanced

Specify advanced inputs: No

FLOWNEX RESULTS (air props on air and flue gas side)



FLOWNEX RESULTS (air props on air side and flue gas props on flue gas side)



Comparison of MCR, Analytical, and Flownex Results

Heat Transfer Rate (MCR Data): $Q_{\text{dot.MCR}} = 257.699 \text{ kW}$

Heat Transfer Rate (Analytical - air props for flue gas): $Q_{\text{dot.analytical.air}} := Q_{\text{dot.AH}} = 235.521 \text{ kW}$

Heat Transfer Rate (Flownex - air props for flue gas): $Q_{\text{dot.flownex.air}} := 236.220 \text{ kW}$

Heat Transfer Rate (Flownex - flue gas props): $Q_{\text{dot.flownex.fg}} := 259.761 \text{ kW}$

$$\text{Error}_{\text{analytical.air.vs.MCR}} := \frac{Q_{\text{dot.analytical.air}} - Q_{\text{dot.MCR}}}{Q_{\text{dot.MCR}}} = -8.606\%$$

$$\text{Error}_{\text{flownex.air.vs.MCR}} := \frac{Q_{\text{dot.flownex.air}} - Q_{\text{dot.MCR}}}{Q_{\text{dot.MCR}}} = -8.335\%$$

$$\text{Error}_{\text{flownex.fg.vs.MCR}} := \frac{Q_{\text{dot.flownex.fg}} - Q_{\text{dot.MCR}}}{Q_{\text{dot.MCR}}} = 0.8\%$$

$$\text{Error}_{\text{flownex.air.vs.analytical.air}} := \frac{Q_{\text{dot.flownex.air}} - Q_{\text{dot.analytical.air}}}{Q_{\text{dot.analytical.air}}} = 0.297\%$$

From the above error analysis between the various models and MCR data we can say the following:

- The Flownex model has been verified against the analytical model (0.297% error)
- The Flownex model has been verified against the MCR data (0.8% error)

Note that a direct verification between the analytical model and the MCR data was not explicitly accomplished due to limitations on including the real fluegas properties in the mathcad model.

However, since we have agreement between the flownex model and analytical model for the case where both models approximate fluegas properties using air as the fluid with an error of 0.297%, we can confidently conclude that the analytical model has been implicitly verified against the MCR data. This is because the flownex model results for the case where the real fluegas properties are used agrees with the MCR data to within 0.8%.

Appendix C. Fire-Tube Evaporator Verification Details

Reference: C:\Users\lmb87\Desktop\Cooliron Fluid Properties rev 1.xmcd(R)

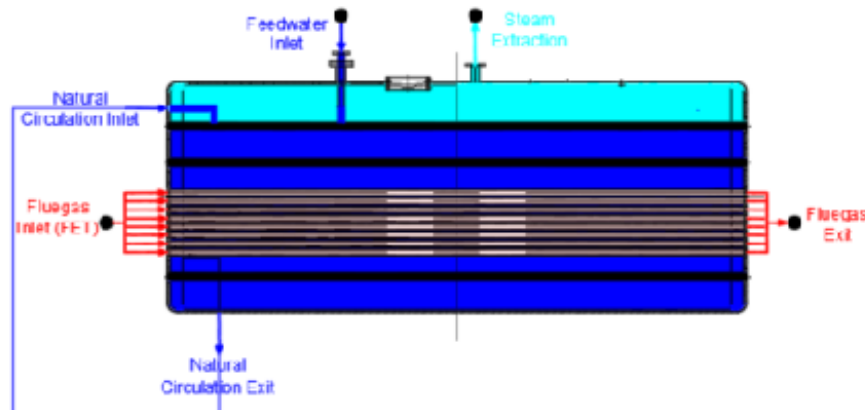
FIRE-TUBE EVAPORATOR VERIFICATION (Analytical vs. MCR vs. Flownex)

E-NTU - Single Stream with Tube Wall Temperature Calculated

E-NTU method for a single stream heat exchanger was used since the water inside the shell is a saturated liquid-vapour mix which exists at the saturation temperature and pressure as given by [3].

The Churchill and Chu correlation for natural convection around a cylinder was used to calculate the water Nusselt number and ultimately the water heat transfer coefficient. It should be noted that the thermal expansion coefficient was evaluated at the saturated temperature inside the evaporator. **The tube wall temperature as well as the fluegas inlet temperature are guessed for the first iteration. After the full calculation is completed, new values for the tube wall temperature and the fluegas inlet temperature are calculated and then fed back into the next iteration. The reference temperature at which the water fluid properties were evaluated at was the saturation temperature. The saturated liquid properties were used in the Churchill and Chu correlation since the fire tubes (containing the fluegas) are submerged in the saturated liquid (not the saturated vapour).**

As per VDI Heat Atlas, correlations which take into account the buoyancy effects are not always reliable and often produce differences of up to 20% as compared with experimental data. As such, buoyancy effects are not included in this simple analytical model. More importantly it is the fluegas side heat transfer coefficient that dominates. The fluegas side heat transfer coefficient was calculated using the Gnielinski pipe flow correlation.



Reference List:

- [1] TO400 TORRIPAC BOILER PROCESS FLOW DIAGRAM, 68756-03, rev 0
- [2] MODEL TO400 (3200 kPa) BOILER SHELL, 68758-06, rev 0
- [3] MODEL TO400 (3200 kPa) BOILER STANDPIPES, 68758-07, rev 1
- [4] Heat and Mass Transfer, Fundamentals & Applications, Yunus A. Cengel, Afshin J. Ghajar.
- [5] Zeghadnia, J. Loup, and B. Achour, "Explicit solutions for turbulent flow friction factor 'a0: A review , assessment and approaches classification," *Ain Shams Eng. J.*, vol. 10, no. 1, pp. 243252, 2019, doi: 10.1016/j.asej.2018.10.007
- [6] Gregory Nellis and Sanford Klein, *Heat Transfer*, 1st ed. Cambridge, U.K.: Cambridge University Press, 2019.
- [7] Heat Transfer in Single and Multiphase Systems, Greg F. Naterer, pg 491

$P_{ATM} = 101.325 \text{ kPa}$

[Atmospheric Pressure].

John Thompson MCR Design Data [1]

This section summarises the Maximum Capacity Rating (MCR) design data as per John Thompsons designs [1].

It includes:

- Feedwater and steam extract mass flowrates.
- Inlet feedwater pressure and temperature
- Steam extract temperature and pressure (i.e. saturated conditions)
- Fluegas mass flowrate
- Fluegas outlet temperature and pressure

A simple heat transfer rate calculation $Q_{\dot{\text{dot}}} = m_{\dot{\text{dot}}} \cdot c_p \cdot \Delta T$ is applied to determine

the heat transfer rate as per MCR data. This value will then be compared to the heat transfer rate of the analytical model and Flownex as a verification.

Water Side

$$m_{\dot{\text{dot}}.\text{H2O.JT}} := 4000 \frac{\text{kg}}{\text{hr}} \quad [1]$$

$$P_{\text{H2O.in.JT}} := 3183\text{kPa} + P_{\text{ATM}} = 3284.325\text{-kPa} \quad [1]$$

$$T_{\text{H2O.in.JT}} := 118^\circ\text{C} \quad [1]$$

$$P_{\text{H2O.out.JT}} := 3000\text{kPa} + P_{\text{ATM}} = 3101.325\text{-kPa} \quad [1]$$

$$T_{\text{H2O.out.JT}} := 236^\circ\text{C} \quad [1]$$

Fluegas Side

$$m_{\dot{\text{dot}}.\text{fg.JT}} := 6984 \frac{\text{kg}}{\text{hr}} \quad [1]$$

$$P_{\text{fg.out.JT}} := 690\text{kPa} + P_{\text{ATM}} = 791.325\text{-kPa} \quad [1]$$

$$T_{\text{fg.out.JT}} := 338^\circ\text{C} \quad [1]$$

MCR Heat Transfer Rate (using water side conditions)

$$\text{Feedwater enthalpy: } h_{\text{fw.JT}} := h_{\text{fluid}}(\text{"water"}, P_{\text{H2O.in.JT}}, T_{\text{H2O.in.JT}}, "", "", "") = 497.515 \frac{\text{kJ}}{\text{kg}}$$

$$\text{Saturated pressure: } P_{\text{sat}} := P_{\text{H2O.out.JT}} = 3101.325\text{-kPa}$$

$$\text{Saturated temperature: } T_{\text{sat}} := T_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, "", "", "") = 235.703^\circ\text{C}$$

$$\text{Vapour enthalpy: } h_{\text{sat.vap}} := h_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, "", "", 1, "") = 2803.174 \frac{\text{kJ}}{\text{kg}}$$

$$\text{Liquid enthalpy: } h_{\text{sat.liq}} := h_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, "", "", 0, "") = 1017.1 \frac{\text{kJ}}{\text{kg}}$$

$$\text{Heat Transfer Rate: } Q_{\dot{\text{dot}}.\text{MCR}} := m_{\dot{\text{dot}}.\text{H2O.JT}} \cdot (h_{\text{sat.vap}} - h_{\text{fw.JT}}) = 2561.843\text{-kW}$$

Note that this heat transfer rate is actually the sum of the radiative heat transfer in the furnace and the convective heat transfer of the Firetube evaporator.

FIRETUBE EVAPORATOR - Analytical Model

Feedwater inlet at top of drum

- Large cylindrical shell filled with a saturated liquid-vapour mixture (water)
- Average water level (AWL) 560mm above horizontal centre of drum
- Steam extraction at top of drum
- Natural circulation facilitated via downcomer linking to water-wall and back into drum
- 116 Firetubes (single flue gas pass)
- Flue gas inlet to tubes on the left (from the furnace exit upstream)
- Flue gas exit from tubes on the right (to air heater downstream)
- 21 solid steel staybars (rods for structural support)

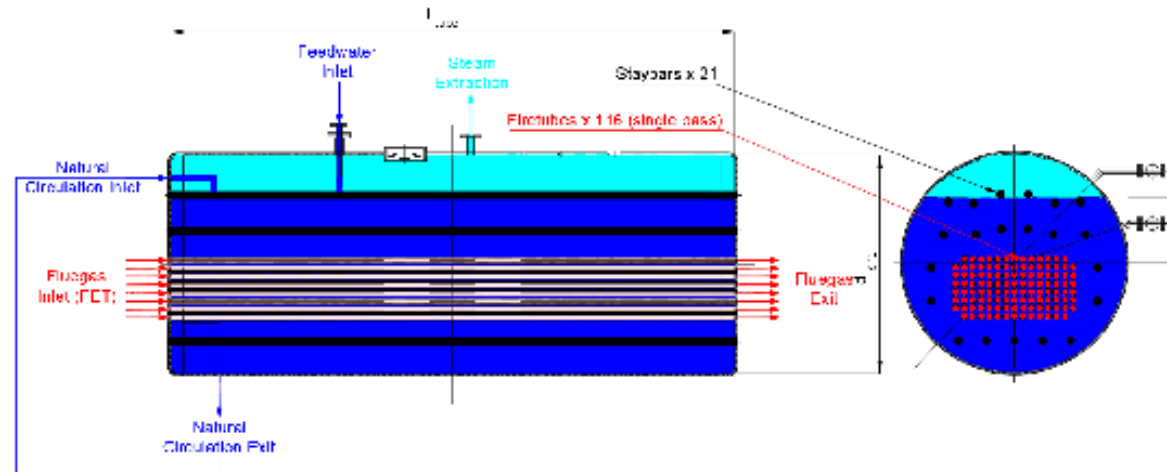


Figure 1: Firetube Evaporator General Arrangement

GEOMETRIC INPUTS

Shell Geometry:

Shell OD: $d_{o.shell} := 1900\text{mm}$ [2]

Shell thickness: $t_{shell} := 22\text{mm}$ [2]

Shell ID: $d_{i.shell} := d_{o.shell} - 2 \cdot t_{shell} = 1856\text{-mm}$

Shell end plate thickness: $t_{e.p.shell} := 22\text{mm}$ [2]

Shell length (inner): $L_{i.shell} := 4706\text{mm}$ [2]

Shell length (outer): $L_{o.shell} := L_{i.shell} + 2 \cdot t_{e.p.shell} = 4750\text{-mm}$

Tube Geometry:

Tube OD: $d_{o.tube} := 50.8\text{mm}$ [2]

Tube wall thickness: $t_{tube} := 4.06\text{mm}$ [2]

Tube ID: $d_{i.tube} := d_{o.tube} - 2 \cdot t_{tube} = 42.68\text{-mm}$

Tube length: $L_{tube} := L_{i.shell} = 4706\text{-mm}$ [2]

Tube Arrangement:

Bulk Arrangement (104 tubes):

- 13 vertical rows (columns) at equal 70mm pitches.
- Each column contains 8 tubes at equal 70mm pitches.
- 1 vertical row (column) on the far left spaced 70mm from the adjacent vertical row (column).

Vertical tube row (column) on the far left (6 tubes):

- 6 tubes spaced at equal 70mm pitches in the vertical row (column) with the top most tube parallel with the tubes in the second row of the bulk arrangement and the bottom most tube parallel to the second last row of the bulk arrangement.

Vertical tube row (column) on the far right (6 tubes):

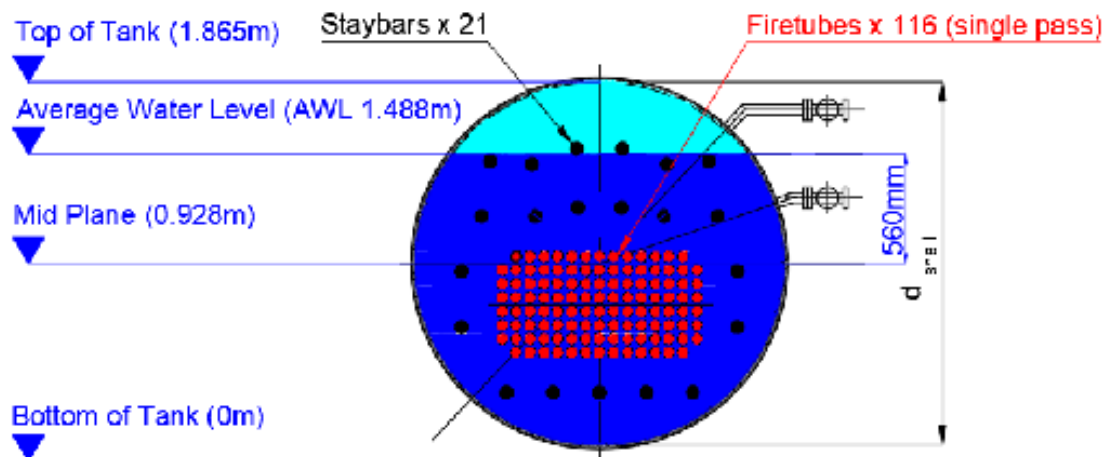
- Mirror image of the far left vertical row/column.

$$N_{\text{tubes}} := 116$$

[2]

Average water level:A.W.L reference to horizontal centre line of shell: $AWL := 560\text{mm}$ **Water Volume:**

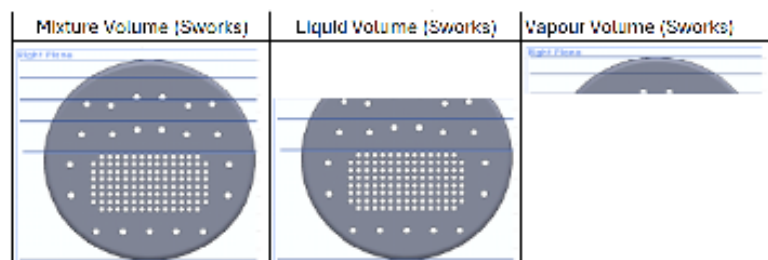
Circle geometry defined as per figure below:

**Figure 2: Firetube Evaporator General Arrangement (AWL)****Fluid Volumes**

$$V_{\text{sat.mix.Sworks}} := 11.45025108\text{m}^3$$

$$V_{\text{sat.liq.Sworks}} := 9.680658824\text{m}^3$$

$$V_{\text{sat.vap.Sworks}} := 1.769592258\text{m}^3$$

**Figure 3: Solidworks Fluid Volumes**Sense
check:

$$V_{\text{sat.mix.Sworks}} - V_{\text{sat.liq.Sworks}} - V_{\text{sat.vap.Sworks}} = -0.000000002\text{m}^3$$

Heat Transfer Surface Areas:**Fluegas Heat Transfer Surface Area:**

$$SA_{fg} := (\pi \cdot d_{i,tube} \cdot L_{i,shell}) \cdot N_{tubes} = 73.195 \text{ m}^2$$

Water heat Transfer Surface Area:

$$SA_{H2O} := (\pi \cdot d_{o,tube} \cdot L_{i,shell}) \cdot N_{tubes} = 87.121 \text{ m}^2$$

Feedwater Inputs:

$$\text{Inlet mass flowrate:} \quad m_{\dot{\text{fw}}} := m_{\dot{\text{H2O,JT}}} = 4000 \cdot \frac{\text{kg}}{\text{hr}} \quad m_{\dot{\text{fw}}} = 1.111 \frac{\text{kg}}{\text{s}} \quad [1]$$

$$\text{Inlet temperature:} \quad T_{\text{fw}} := 118 \text{ }^\circ\text{C} \quad [1]$$

$$\text{Inlet Pressure:} \quad P_{\text{fw}} := 3183 \text{ kPa} + P_{\text{ATM}} = 3284.325 \text{ kPa} \quad [1]$$

Specific Heat Capacity:

$$c_{p,\text{fw}} := C_{p,\text{fluid}}(\text{"water"}, P_{\text{fw}}, T_{\text{fw}}, \text{"", ""}, \text{"", ""}) = 4232.498 \cdot \frac{\text{J}}{\text{kg} \cdot \text{K}}$$

Density:

$$\rho_{\text{fw}} := \rho_{\text{fluid}}(\text{"water"}, P_{\text{fw}}, T_{\text{fw}}, \text{"", ""}, \text{"", ""}) = 946.243 \frac{\text{kg}}{\text{m}^3}$$

Thermal Conductivity:

$$\lambda_{\text{fw}} := \lambda_{\text{fluid}}(\text{"water"}, P_{\text{fw}}, T_{\text{fw}}, \text{"", ""}) = 0.684 \cdot \frac{\text{W}}{\text{m} \cdot \text{K}}$$

Dynamic Viscosity:

$$\mu_{\text{fw}} := \mu_{\text{fluid}}(\text{"water"}, P_{\text{fw}}, T_{\text{fw}}, \text{"", ""}) = 2.371 \times 10^{-4} \cdot \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Kinematic Viscosity:

$$\nu_{\text{fw}} := \frac{\mu_{\text{fw}}}{\rho_{\text{fw}}} = 2.505 \times 10^{-7} \cdot \frac{\text{m}^2}{\text{s}}$$

Prandtl Number:

$$Pr_{\text{fw}} := Pr_{\text{fluid}}(\text{"water"}, P_{\text{fw}}, T_{\text{fw}}, \text{"", ""}) = 1.467$$

Saturation State Checks:

$$\text{Outlet mass flowrate:} \quad m_{\dot{\text{steam}}} := m_{\dot{\text{fw}}} = 4000 \cdot \frac{\text{kg}}{\text{hr}} \quad m_{\dot{\text{steam}}} = 1.111 \frac{\text{kg}}{\text{s}} \quad [1]$$

$$\text{Outlet steam temperature:} \quad T_{\text{steam}} := T_{\text{H2O,out,JT}} = 236 \cdot ^\circ\text{C} \quad [1]$$

$$\text{Outlet steam pressure:} \quad P_{\text{steam}} := P_{\text{H2O,out,JT}} = 3101.325 \text{ kPa} \quad [1]$$

Check the saturation temperature using coolprop:

$$T_{\text{sat,check}} := T_{\text{fluid}}(\text{"water"}, P_{\text{steam}}, \text{"", ""}, \text{"", ""}) = 235.703 \cdot ^\circ\text{C}$$

$$\frac{T_{\text{steam}} - T_{\text{sat,check}}}{T_{\text{sat,check}}} = 0.058\% \quad \text{ACCEPTABLE.} \quad \text{Check saturation condition with CoolProp}$$

Therefore to be consistent, the saturation temperature used in the rest of the analysis is the temperature evaluated by CoolProp at the saturation pressure:

$$P_{\text{sat}} := P_{\text{steam}} = 3101.325 \text{ kPa}$$

$$T_{\text{sat}} := T_{\text{sat,check}} = 235.703 \cdot ^\circ\text{C}$$

Saturated Liquid Fluid Properties:**Specific Heat Capacity:**

$$c_{p,\text{sat.l}} := C_{p,\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", "", 0, "", ""}) = 4734.119 \cdot \frac{\text{J}}{\text{kg} \cdot \text{K}}$$

Density:

$$\rho_{\text{sat.l}} := \rho_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", 0, "", ""}) = 819.36 \frac{\text{kg}}{\text{m}^3} \quad v_{\text{sat.l}} := \frac{1}{\rho_{\text{sat.l}}} = 0.0012205 \frac{\text{m}^3}{\text{kg}}$$

Thermal Conductivity:

$$\lambda_{\text{sat.l}} := \lambda_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", 0}) = 0.631 \cdot \frac{\text{W}}{\text{m} \cdot \text{K}}$$

Dynamic Viscosity:

$$\mu_{\text{sat.l}} := \mu_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", 0}) = 1.132 \times 10^{-4} \frac{\text{kg}}{\text{s} \cdot \text{m}}$$

Kinematic Viscosity:

$$\nu_{\text{sat.l}} := \frac{\mu_{\text{sat.l}}}{\rho_{\text{sat.l}}} = 1.382 \times 10^{-7} \frac{\text{m}^2}{\text{s}}$$

Prandtl Number:

$$Pr_{\text{sat.l}} := Pr_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", 0}) = 0.849$$

Saturated Vapour Fluid Properties:**Specific Heat Capacity:**

$$c_{p,\text{sat.v}} := C_{p,\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", "", 1, "", ""}) = 3653.399 \cdot \frac{\text{J}}{\text{kg} \cdot \text{K}}$$

Density:

$$\rho_{\text{sat.v}} := \rho_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", 1, "", ""}) = 15.509 \frac{\text{kg}}{\text{m}^3} \quad v_{\text{sat.v}} := \frac{1}{\rho_{\text{sat.v}}} = 0.064 \frac{\text{m}^3}{\text{kg}}$$

Thermal Conductivity:

$$\lambda_{\text{sat.v}} := \lambda_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", 1}) = 0.046 \cdot \frac{\text{W}}{\text{m} \cdot \text{K}}$$

Dynamic Viscosity:

$$\mu_{\text{sat.v}} := \mu_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", 1}) = 1.691 \times 10^{-5} \frac{\text{kg}}{\text{s} \cdot \text{m}}$$

Kinematic Viscosity:

$$\nu_{\text{sat.v}} := \frac{\mu_{\text{sat.v}}}{\rho_{\text{sat.v}}} = 1.09 \times 10^{-6} \frac{\text{m}^2}{\text{s}}$$

Prandtl Number:

$$Pr_{\text{sat.v}} := Pr_{\text{fluid}}(\text{"water"}, P_{\text{steam}}, \text{"", 1}) = 1.331$$

Saturated Liquid Enthalpy:

$$h_{\text{sat.l}} := h_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", "", 0, ""}) = 1017.1 \cdot \frac{\text{kJ}}{\text{kg}}$$

Saturated Vapour Enthalpy:

$$h_{\text{sat.v}} := h_{\text{fluid}}(\text{"water"}, P_{\text{sat}}, \text{"", "", 1, ""}) = 2803.174 \cdot \frac{\text{kJ}}{\text{kg}}$$

Latent heat of vapourization:

$$h_{\text{vapourisation}} := h_{\text{sat.v}} - h_{\text{sat.l}} = 1786.074 \cdot \frac{\text{kJ}}{\text{kg}}$$

Saturated vapour mass:

$$m_{\text{sat.v}} := \rho_{\text{sat.v}} \cdot V_{\text{sat.vap.Sworks}} = 27.445 \text{ kg}$$

Saturated liquid mass:

$$m_{\text{sat.l}} := \rho_{\text{sat.l}} \cdot V_{\text{sat.liq.Sworks}} = 7931.945 \text{ kg}$$

Total mass of sat liq-vap mixture:

$$m_{\text{sat.mix}} := m_{\text{sat.l}} + m_{\text{sat.v}} = 7959.391 \text{ kg}$$

Steam quality:

$$x_{\text{steam}} := \frac{m_{\text{sat.v}}}{m_{\text{sat.mix}}} = 0.0034481823$$

Flue gas Side Inputs:

All inputs below, barring the flue gas inlet emperature have been explained in the beginning of this report.

$$\text{Flue gas mass flowrate: } m_{\text{dot},\text{fg}} := m_{\text{dot},\text{fg},\text{JT}} = 6984 \cdot \frac{\text{kg}}{\text{hr}} \quad m_{\text{dot},\text{fg}} = 1.94 \frac{\text{kg}}{\text{s}} \quad [1]$$

$$\text{Outlet flue gas pressure (abs): } P_{\text{fg},\text{out}} := P_{\text{fg},\text{out},\text{JT}} = 791.325 \cdot \text{kPa} \quad [1]$$

$$\text{Inlet flue gas pressure (abs): } P_{\text{fg},\text{in}} := 1.02 \cdot P_{\text{fg},\text{out}} = 807.152 \cdot \text{kPa} \quad [\text{Assuming 2\% pressure drop.}]$$

$$\text{Outlet flue gas temperature: } T_{\text{fg},\text{out}} := T_{\text{fg},\text{out},\text{JT}} = 338 \cdot ^\circ\text{C} \quad [1]$$

$$\text{Inlet flue gas temperature: } T_{\text{fg},\text{in}} := 946.0893563 \cdot ^\circ\text{C} \quad [\text{Successive approximation with first guess as } 800 \text{ degC}]$$

Fluegas Fluid Properties:

Fluegas Density evaluated at $T_{\text{fg},\text{in}}$ and $P_{\text{fg},\text{in}}$:

$$\rho_{\text{fg},\text{in}} := \rho_{\text{fluid}}(\text{"air"}, P_{\text{fg},\text{in}}, T_{\text{fg},\text{in}}, \text{"", ""}, \text{"", ""}) = 2.301 \frac{\text{kg}}{\text{m}^3}$$

Average fluegas-side temperature and pressure:

$$T_{\text{fg},\text{avg}} := \frac{T_{\text{fg},\text{in}} + T_{\text{fg},\text{out}}}{2} = 642.045 \cdot ^\circ\text{C} \quad P_{\text{fg},\text{avg}} := \frac{P_{\text{fg},\text{in}} + P_{\text{fg},\text{out}}}{2} = 799.238 \cdot \text{kPa}$$

Note: Fluegas properties taken to that of air at the same T and P .

Fluegas specific heat capacity evaluated at $T_{\text{fg},\text{avg}}$ and $P_{\text{fg},\text{avg}}$:

$$c_{p,\text{fg}} := C_{p,\text{fluid}}(\text{"air"}, P_{\text{fg},\text{avg}}, T_{\text{fg},\text{avg}}, \text{"", ""}, \text{"", ""}) = 1.125 \cdot \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

Fluegas density evaluated at $T_{\text{fg},\text{avg}}$ and $P_{\text{fg},\text{avg}}$:

$$\rho_{\text{fg}} := \rho_{\text{fluid}}(\text{"air"}, P_{\text{fg},\text{avg}}, T_{\text{fg},\text{avg}}, \text{"", ""}, \text{"", ""}) = 3.034 \frac{\text{kg}}{\text{m}^3}$$

Fluegas Thermal Conductivity:

$$\lambda_{\text{fg}} := \lambda_{\text{fluid}}(\text{"air"}, P_{\text{fg},\text{avg}}, T_{\text{fg},\text{avg}}, \text{"", ""}) = 0.063 \cdot \frac{\text{W}}{\text{m} \cdot \text{K}}$$

Fluegas Dynamic Viscosity:

$$\mu_{\text{fg}} := \mu_{\text{fluid}}(\text{"air"}, P_{\text{fg},\text{avg}}, T_{\text{fg},\text{avg}}, \text{"", ""}) = 4.088 \times 10^{-5} \cdot \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

Fluegas Kinematic Viscosity:

$$\nu_{\text{fg}} := \frac{\mu_{\text{fg}}}{\rho_{\text{fg}}} = 1.347 \times 10^{-5} \cdot \frac{\text{m}^2}{\text{s}}$$

Fluegas Prandtl Number:

$$Pr_{\text{fg}} := Pr_{\text{fluid}}(\text{"air"}, P_{\text{fg},\text{avg}}, T_{\text{fg},\text{avg}}, \text{"", ""}) = 0.725$$

Total fluegas-side heat transfer rate:

$$Q_{\text{dot},\text{fg}} := m_{\text{dot},\text{fg}} \cdot c_{p,\text{fg}} \cdot (T_{\text{fg},\text{in}} - T_{\text{fg},\text{out}}) = \text{■} \cdot \text{kW}$$

$$m_{\text{dot},\text{fg}} \cdot \left(h_{\text{fluid}}(\text{"air"}, P_{\text{fg},\text{in}}, T_{\text{fg},\text{in}}, \text{"", ""}, \text{"", ""}) \dots \right. \\ \left. + h_{\text{fluid}}(\text{"air"}, P_{\text{fg},\text{out}}, T_{\text{fg},\text{out}}, \text{"", ""}, \text{"", ""}) \right) = \text{■} \cdot \text{kW}$$

DETERMINING THE FLUEGAS HEAT TRANSFER COEFFICIENT FOR FLOW INSIDE TUBES (Gnielinski correlation [4]):

Total fluegas mass flowrate:

$$\dot{m}_{\text{dot.fg}} = 1.94 \frac{\text{kg}}{\text{s}}$$

Fluegas mass flowrate per tube:

$$\dot{m}_{\text{dot.fg.pt}} := \frac{\dot{m}_{\text{dot.fg}}}{N_{\text{tubes}}} = 0.017 \frac{\text{kg}}{\text{s}}$$

Fluegas velocity per tube:

$$v_{\text{fg.pt}} := \frac{\dot{m}_{\text{dot.fg.pt}}}{\rho_{\text{fg}} \cdot \left(\frac{\pi \cdot d_{\text{i.tube}}^2}{4} \right)} = 3.852 \frac{\text{m}}{\text{s}}$$

Fluegas Reynolds Number for fluegas flow inside a tube:

$$Re_{\text{d.i.tube}} := \frac{\rho_{\text{fg}} \cdot v_{\text{fg.pt}} \cdot d_{\text{i.tube}}}{\mu_{\text{fg}}} = 12204.322 \quad Re_{\text{fg}} := Re_{\text{d.i.tube}} = 12204.322$$

Surface Roughness (sheet metal, new) as per [4].

$$\varepsilon_{\text{steel}} := 100 \mu\text{m} \quad \text{For sheet metal, new (max).} \quad [4]$$

Friction factor:

$$5 \cdot 10^3 = 5000 < Re_{\text{d.i.tube}} = 12204.322 < 5 \cdot 10^8 = 500000000 \quad \text{Smooth to rough correlation Swamee and Jain, (1976)} \quad [5]$$

$$10^{-6} = 0.000001 < \frac{\varepsilon_{\text{steel}}}{d_{\text{i.tube}}} = 0.002 < 10^{-2} = 0.01$$

$$f := \left(\frac{1}{-2 \cdot \log \left(\frac{\frac{\varepsilon_{\text{steel}}}{d_{\text{i.tube}}}}{3.7} + \frac{5.74}{Re_{\text{d.i.tube}}^{0.9}} \right)} \right)^2 = 0.033$$

Fluegas Nusselt Number:

$$Nu_{\text{fg}} := \frac{\left(\frac{f}{8} \right) \cdot (Re_{\text{d.i.tube}} - 1000) \cdot Pr_{\text{fg}}}{1 + 12.7 \cdot \left(\frac{f}{8} \right)^{0.5} \cdot \left(\frac{2}{3} Pr_{\text{fg}} - 1 \right)} = 40.315 \quad [4]$$

Fluegas Heat Transfer Coefficient:

$$h_{\text{fg}} := \frac{Nu_{\text{fg}} \cdot \lambda_{\text{fg}}}{d_{\text{i.tube}}} = 59.884 \frac{\text{W}}{\text{m}^2 \text{K}}$$

DETERMINING THE WATER HEAT TRANSFER COEFFICIENT:

Since the fire tubes are submerged in the water they are likely to be in thermal equilibrium with the bulk fluid (i.e. sat mix of water). Therefore the average tube wall temperature in contact with the fluid should be very close to the saturation temperature. The tube wall temperature will be slightly above the saturation temperature since this temperature difference is the driving force for heat transfer. The initial guess is $T_{\text{wall}} = T_{\text{sat}} + 1\text{K}$.

Tube wall temperature: $T_{\text{t.wall}} := 245.6631897^\circ\text{C}$

[Successive approximation with the first guess assuming a ΔT between fluid and wall of 1K.]

$$T_{\text{sat}} = 235.703^\circ\text{C}$$

Ignoring buoyancy effects for now and assuming natural convection driven purely by temperature difference, the Churchill and Chu Nusselt Number correlation for natural convection around a long horizontal cylinder as per [6] is:

$$\text{Nu} = \left[0.60 + \frac{0.387 \cdot \text{Ra}^{\frac{1}{4}}}{\left[1 + \left(\frac{0.559}{\text{Pr}} \right)^{\frac{4}{9}} \right]^{\frac{1}{4}} \left[1 + \left(\frac{\text{Ra}}{16} \right)^{\frac{1}{4}} \right]} \right]^{\frac{4}{3}} \quad \text{where}$$

$$\text{Ra} = \text{Gr} \cdot \text{Pr}$$

$$\text{Gr} = \frac{g \cdot L^3}{\nu^2} \cdot \frac{\rho_\infty - \rho_w}{\rho_w}$$

or

$$\text{For convection driven solely by a temperature difference: } \text{Gr} = \frac{g \cdot L^3}{\nu^2} \cdot \beta \cdot (T_w - T_\infty), \quad \text{where}$$

$$(\beta = \text{volume expansion coefficient}) = \frac{1}{T_w} = \frac{1}{T_{\text{t.wall}}} \quad \text{holds for ideal gases.}$$

The Nusselt Number correlation above holds for $0 < \text{Pr} < \infty$ and $10^{-5} < \text{Ra} < 10^{12}$, and the characteristic length is the diameter, $L = d_{\text{o.tube}}$.

Note that for the fire-tube evaporator at hand:

$$T_w = T_{\text{t.wall}}$$

$$T_\infty = T_{\text{sat}}$$

Note also that the fluid properties should be evaluated at the reference temperature taken as the average temperature of the bulk fluid temperature and the tube wall temperature:

$$T_{\text{ref}} = \frac{T_w + T_\infty}{2} = \frac{T_{\text{t.wall}} + T_{\text{sat}}}{2}$$

Reference Temperature:

$$T_{\text{ref}} := \frac{T_{\text{t.wall}} + T_{\text{sat}}}{2} = 240.683 \cdot ^\circ\text{C}$$

NOTE: This temperature was initially used to evaluate the fluid properties of the water. However this temperature means that the water in contact with the tubes is a superheated vapour (which is NOT the case!)

Coefficient of Volume Expansion:

$$\beta := \frac{1}{T_{\text{sat}}} = 0.002 \cdot \text{K}^{-1}$$

EVALUATE FLUID PROPERTIES USING SATURATED LIQUID WATER:

These properties have already been evaluated previously. Below is a summary of the values:

$$\mu_{\text{sat},1} = 1.132 \times 10^{-4} \frac{\text{kg}}{\text{s} \cdot \text{m}} \quad \rho_{\text{sat},1} = 819.36 \frac{\text{kg}}{\text{m}^3} \quad \nu_{\text{sat},1} = 1.382 \times 10^{-7} \frac{\text{m}^2}{\text{s}} \quad \text{Pr}_{\text{sat},1} = 0.849$$

Water thermal conductivity:

$$\lambda_{\text{H}_2\text{O}} := \lambda_{\text{sat},1} = 0.631 \frac{\text{W}}{\text{m} \cdot \text{K}}$$

Gravitational Acceleration:

$$g = 9.807 \frac{\text{m}}{\text{s}^2}$$

Characteristic Length for Grashof Number:

$$L := d_{\text{o.tube}} = 0.051 \text{ m}$$

Grashof Number:

$$\text{Gr} := \frac{g \cdot L^3}{\nu_{\text{sat},1}^2} \cdot \beta \cdot (T_{\text{t.wall}} - T_{\text{sat}}) = 1.318 \times 10^9$$

Rayleigh Number:

$$\text{Ra} := \text{Gr} \cdot \text{Pr}_{\text{sat},1} = 1.119 \times 10^9$$

Water Nusselt Number for Natural Convection around a horizontal Cylinder:

$$\text{Nu}_{\text{H}_2\text{O}} := \left[0.6 + \frac{0.387 \cdot \text{Ra}^{\frac{1}{4}}}{\left[1 + \left(\frac{0.559}{\text{Pr}_{\text{sat},1}} \right)^{\frac{4}{9}} \right]^{\frac{1}{4}}} \right]^2 = 123.05$$

NOTE: The correct characteristic length for this correlation is indeed the OD of the tube as per [6].

Water Heat Transfer Coefficient:

$$h_{\text{H}_2\text{O}} := \frac{\text{Nu}_{\text{H}_2\text{O}} \cdot \lambda_{\text{H}_2\text{O}}}{L} = 1529.252 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Properties of Tube Material:

Assume tubes are made of AISI 1010:

Thermal conductivity of AISI 1010 at 300K:

$$\lambda_{\text{steel}} := 63.9 \frac{\text{W}}{\text{m} \cdot \text{K}} \quad [4]$$

$$\lambda_{\text{tube}} := \lambda_{\text{steel}} = 63.9 \frac{\text{W}}{\text{m} \cdot \text{K}}$$

DETERMINING THE UA VALUE:**Thermal Resistances:**

Thermal resistance due to the fluegas boundary layer on the inner tube wall:

$$R_i := h_{fg} = 59.884 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Thermal resistance due to fouling on the inside of the tubes (fluegas-side):

$$R_{f,i} := 0 \frac{\text{m}^2 \cdot \text{K}}{\text{W}}$$

Thermal resistance due to wall thermal conductivity:

$$R_w := \frac{\ln \left(\frac{d_{o,\text{tube}}}{d_{i,\text{tube}}} \right)}{2 \cdot \pi \cdot \lambda_{\text{tube}} \cdot L_{\text{tube}} \cdot N_{\text{tubes}}} = 7.946 \times 10^{-7} \frac{\text{K}}{\text{W}}$$

Thermal resistance due to fouling on the outside of the tubes (water-side):

$$R_{f,o} := 0 \frac{\text{m}^2 \cdot \text{K}}{\text{W}}$$

Thermal resistance due to water boundary layer on the outer tube wall:

$$R_o := h_{\text{H}_2\text{O}} = 1529.252 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

Inside tube surface efficiency (fluegas-side): $\eta_{fg} := 1$

[Unity for no fins.]

Outside tube surface efficiency (water-side): $\eta_{\text{H}_2\text{O}} := 1$

[Unity for no fins.]

Therefore we can calculate the UA value as:

$$UA := \left(\frac{1}{\eta_{fg} \cdot R_i \cdot SA_{fg}} + \frac{R_{f,i}}{\eta_{fg} \cdot SA_{fg}} + R_w + \frac{R_{f,o}}{\eta_{\text{H}_2\text{O}} \cdot SA_{\text{H}_2\text{O}}} + \frac{1}{\eta_{\text{H}_2\text{O}} \cdot R_o \cdot SA_{\text{H}_2\text{O}}} \right)^{-1}$$

$$UA = 4229.39 \frac{\text{W}}{\text{K}}$$

$$\frac{UA}{SA_{fg}} = 57.782 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

$$R_i = 59.884 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

$$R_o = 1529.252 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$

E-NTU For Single Stream Heat Exchanger:

The hot fluid is the fluegas and is denoted by the subscript, H for the foregoing analysis.

The cold fluid is the water and is denoted by the subscript, C for the foregoing analysis.

Since there is a saturated liquid-vapour mix existing at the same temperature and pressure, the heat transferred to the water contributes 100% to the evaporation of the water. This implies that the water in the evaporator has an infinite specific heat capacity:

Therefore:

$$c_{p,\text{sat.mix}} = \text{infinity} \quad c_{p,\text{sat.mix}} := \infty \cdot \frac{\text{J}}{\text{kg} \cdot \text{K}} \quad \text{Note that mathcad will likely view this as an extremely large truncated value.}$$

Heat capacity of the hot stream:

$$C_H := m_{\text{dot.fg}} \cdot c_{p,\text{fg}} = 2.182 \cdot \frac{\text{kW}}{\text{K}}$$

Heat capacity of the cold stream:

$$C_C := m_{\text{dot.steam}} \cdot c_{p,\text{sat.mix}}$$

Minimum heat capacity:

$$C_{\min} := C_H = 2.182 \cdot \frac{\text{kW}}{\text{K}}$$

Maximum heat capacity:

$$C_{\max} := C_C$$

Number of transfer units:

$$\text{NTU} := \frac{UA}{C_{\min}} = 1.938$$

Heat capacity ratio:

$$C := \frac{C_{\min}}{C_{\max}} = 0 \quad [\text{C.max in the denom is infinite and therefore drives the heat capacity ratio to zero.}]$$

For single stream ($C = 0$), the E-NTU correlation is calculated as $\varepsilon = 1 - e^{(-\text{NTU})}$:

$$\varepsilon := 1 - e^{(-\text{NTU})} = 0.856 \quad [7]$$

At this point we have the following:

$$T_{fg.out} = 338.^\circ\text{C} \quad [\text{Given as per [1].}]$$

$$T_{fg.in} = 946.089.^\circ\text{C} \quad [\text{Guessed value which is updated iteratively. Initial guess is } 800^\circ\text{C}.]$$

$$T_{sat} = 235.703.^\circ\text{C} \quad [\text{Saturation temperature evaluated by Coolprop at the pressure given as per [1].}]$$

$$T_{t.wall} = 245.663.^\circ\text{C} \quad [\text{Guessed value which is updated iteratively. Initial guess is } 0.5^\circ\text{C above } T_{sat}]$$

$$\varepsilon = 0.856 \quad [\text{Calculated above.}]$$

Iterative solve is now conducted by calculating the actual fluegas inlet and actual tube wall temperatures and then feeding them back into the analysis until convergence:

Calculate heat transfer rate using E-NTU from $T_{fg.in}$, T_{sat} and effectiveness:

$$Q_{dot.E.NTU} := \varepsilon \cdot C_{min} \cdot (T_{fg.in} - T_{sat}) = 1327.074 \cdot \text{kW}$$

Calculate heat transfer rate of the fluegas to the inner tube wall based on the assumed fluegas inlet temperature:

$$Q_{dot.fg} := m_{dot.fg} \cdot c_{p.fg} \cdot (T_{fg.in} - T_{fg.out}) = 1327.055 \cdot \text{kW}$$

Calculate the actual fluegas inlet temperature by equating $Q_{dot.E.NTU} = Q_{dot.fg.actual}$, where, $Q_{dot.fg.actual} = m_{dot.fg.in} \cdot c_{p.fg} \cdot (T_{fg.in.actual} - T_{fg.out})$:

$$T_{fg.in.actual} := \frac{Q_{dot.E.NTU}}{m_{dot.fg} \cdot c_{p.fg}} + T_{fg.out} = 946.098.^\circ\text{C}$$

Calculate heat transfer rate from the outer tube wall to the surrounding water based on the assumed tube wall temperature:

$$Q_{dot.t.wall} := h_{H2O} \cdot SA_{H2O} \cdot (T_{t.wall} - T_{sat}) = 1327.059 \cdot \text{kW}$$

Calculate the actual tube wall temperature by equating $Q_{dot.E.NTU} = Q_{dot.t.wall.actual}$, where, $Q_{dot.t.wall.actual} = h_{H2O} \cdot SA_{H2O} \cdot (T_{t.wall.actual} - T_{sat})$:

$$T_{t.wall.actual} := \frac{Q_{dot.E.NTU}}{h_{H2O} \cdot SA_{H2O}} + T_{sat} = 245.663.^\circ\text{C}$$

Calculate percentage errors between actual temperatures and guessed temperatures:**Flue gas inlet temperature:**

$$\text{Error}_{T_{fg.in}} := \frac{T_{fg.in.actual} - T_{fg.in}}{T_{fg.in}} = 0.001\%$$

Tube wall temperature:

$$\text{Error}_{T_{t.wall}} := \frac{T_{t.wall.actual} - T_{t.wall}}{T_{t.wall}} = 0\%$$

1. Calculate percentage errors between heat transfer rates between E-NTU and fluegas-inner tube wall:

$$\text{Error}_{Q_{dot.fg}} := \frac{Q_{dot.E.NTU} - Q_{dot.fg}}{Q_{dot.fg}} = 0.001\%$$

2. Calculate percentage errors between heat transfer rates between E-NTU and outer tube-water:

$$\text{Error}_{Q_{dot.t.wall}} := \frac{Q_{dot.E.NTU} - Q_{dot.t.wall}}{Q_{dot.t.wall}} = 0.001\%$$

Check for convergence:

$$T_{fg.in} = 946.0893563\text{ }^{\circ}\text{C} \quad Q_{dot.E.NTU} = 1327.07\text{ kW}$$

$$T_{fg.in.actual} = 946.0982034\text{ }^{\circ}\text{C} \quad Q_{dot.fg} = 1327.05\text{ kW}$$

$$\text{Error}_{T_{fg.in}} = 0.001\% \quad \text{Error}_{Q_{dot.fg}} = 0.001\%$$

$$T_{t.wall} = 245.6631897\text{ }^{\circ}\text{C} \quad Q_{dot.E.NTU} = 1327.07\text{ kW}$$

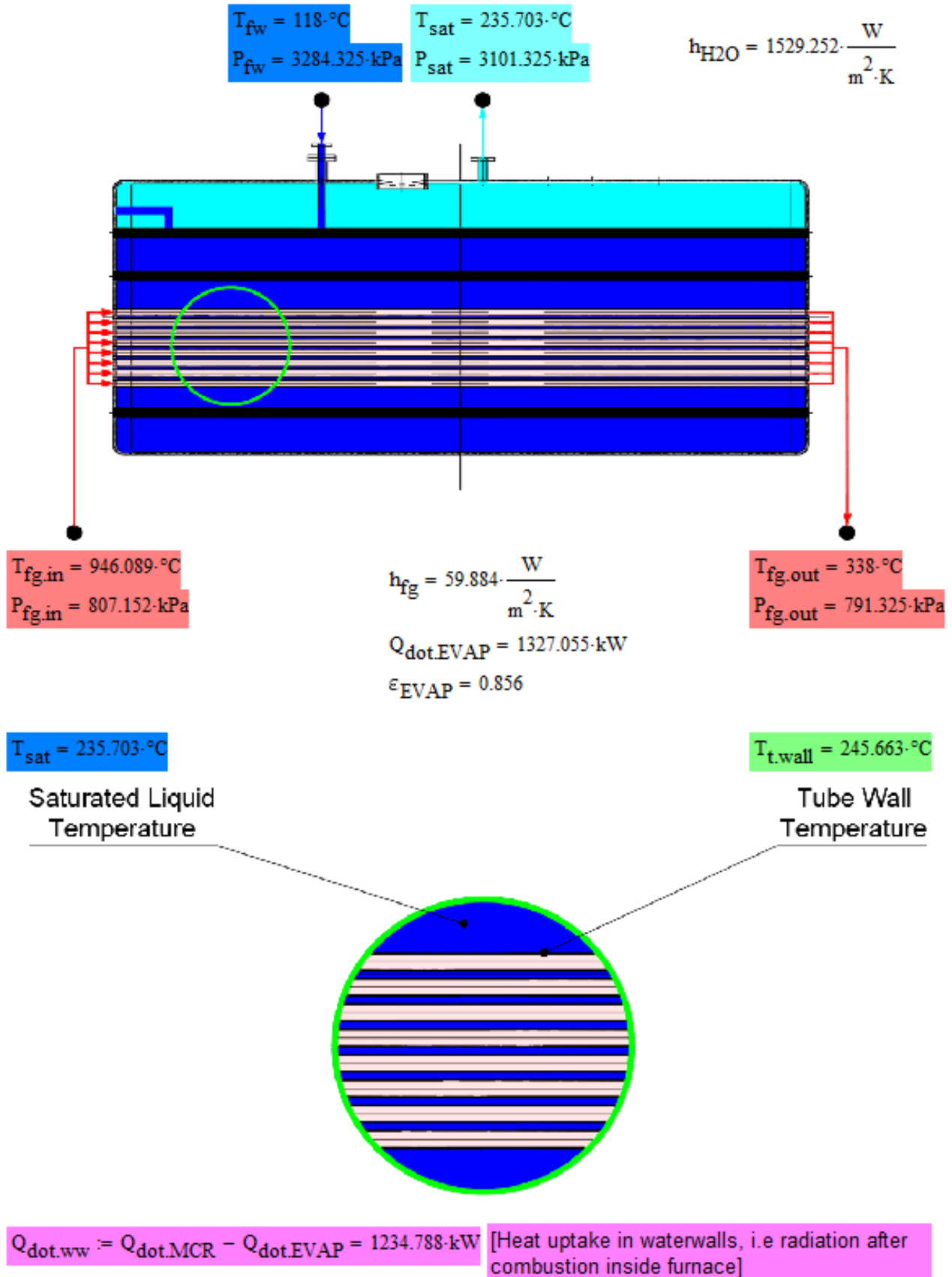
$$T_{t.wall.actual} = 245.6633026\text{ }^{\circ}\text{C} \quad Q_{dot.t.wall} = 1327.06\text{ kW}$$

$$\text{Error}_{T_{t.wall}} = 0\% \quad \text{Error}_{Q_{dot.t.wall}} = 0.001\%$$

$$Q_{dot.EVAP} := Q_{dot.fg}$$

$$\epsilon_{EVAP} := \epsilon$$

[A total of 69 iterations before convergence]

SUMMARY OF CONVERGED RESULTS

FIRETUBE EVAPORATOR - Flownex Model

The inputs described here are for the standalone evaporator model and include the following:

- Pipe components:
 - Fire Tubes
 - Feedwater Inlet
 - Downcomer
 - Waterwall
 - Steam extract
- Nodes
- Two Phase Tank
- Boundary Conditions
- Composite Heat Transfer Component - Gnielinski Forced convection of flue gas side & Churchill and Chu Natural Convection on water side
- Quick Script - Employing Churchill and Chu correlation to determine the water heat transfer coefficient
- Data Transfer Links:
 - Mass flowrate from steam extract pipe transferred to Downcomer pipe
 - Saturated liquid fluid properties from Two-phase tank to Quick Script

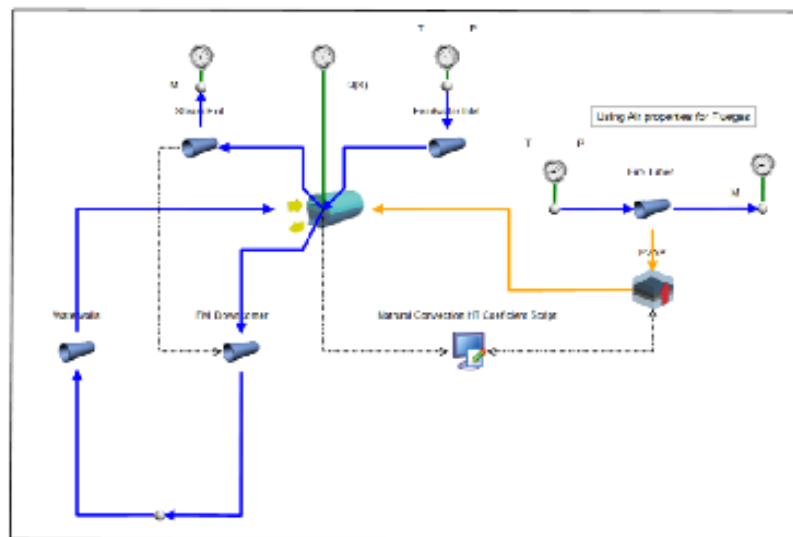


Figure A: Firetube Evaporator - Flownex Model

Note: Air was used on the fluegas side for verification with the analytical model which also uses air to approximate the flue gas properties. However, the real flue gas properties were also used in a separate run with mass fractions as seen below from a combustion analysis based on the fuel and air compositions.

Mass Fraction	
Specify mass fraction	Yes
Mass fractions	1
SO2	2.1739E-19
Fly ash	0.000962234
Unburnt carbon	0.00112597
CO2 - Carbon Dioxide	0.111731
N2 - Nitrogen	0.60349
NO - Nitric Oxide	0
O2 - Oxygen	0.0437381
H2O - Water	0.239433
Steady-state continuity option	Calculate mass source

Figure B: Fluegas composition based on fuel & air compositions from combustion analysis

Pipe Component Inputs



Geometry:

- The actual pipe specifications are indicated on the relevant John Thompson Workshop drawings.
- Wherever possible, the geometry of the pipe components in Flownex were extracted from the relevant data base which is built-in to Flownex.
- Most of the piping as per JTs drawings were found in the ASME B36.10M database for Welded and Seamless Wrought Steel Pipe

ASME B36.10M
Welded and Seamless Wrought Steel Pipe

Click on blue cells to select schedule and press OK to confirm

Table 1 Dimensions and Weights of Welded and Seamless Wrought Steel Pipe

NPS	Customary Units			Identification [Standard (STD), Extra Strong (XS) or Double Extra Strong (XXS)]	Schedule No.	DN	SI Units		
	Outside Diameter	Wall Thickness	Plain End Weight				Outside Diameter	Wall Thickness	Plain Ma
[Note 1]	in.	in.	lb/ft			[Note 2]	mm	mm	kg/m
1/8	0.405	0.049	0.19	...	10	6	10.3	1.24	0.2
1/8	0.405	0.057	0.21	...	30	6	10.3	1.45	0.3
1/8	0.405	0.068	0.24	STD	40	6	10.3	1.73	0.3
1/8	0.405	0.095	0.31	XS	80	6	10.3	2.41	0.4
1/4	0.54	0.065	0.33	...	10	8	13.7	1.65	0.4
1/4	0.54	0.073	0.36	...	30	8	13.7	1.85	0.5

Figure C: ASME B36.10M Database in Flownex

- By choosing the suitable pipe specification, Flownex automatically assigns the relevant nominal bore and wall thickness.
- The length of the pipes were specified manually to suit the true lengths as per the Workshop drawings (unless stated otherwise below).

Losses:

For each pipe component:

Primary loss type: "Darcy Weisbach"

Roughness option: "Select from Database"

Roughness selection: "Welded Steel tubes - New or old, but in good condition, welded or riveted joints - Max"

Roughness: 100 μ m

Discretisation:

The number of pipes in parallel are accurate as per the workshop drawings and the number of increments is 1 for non-critical flows and 10 for pipes which are involved in heat transfer via composite heat transfer components.

Summary of All Pipe Inputs

Fire Tubes: Pipe Inputs	
Carbon Steel - AISI 1010 Metal (Solids) {Project}	
Length	4.706 m
Wall thickness	4.060 mm
ID	42.680 mm
Number in parallel	116
Number of increments	10
Roughness	100.000 μm

Discretised into 10 increments for heat transfer accuracy.

Feedwater Inlet: Pipe Inputs	
DN: 32 Schedule: 40 STD	
Diameter (from schedule)	35.080 mm
Wall thickness (from schedule)	3.560 mm
Length	200.000 mm
Number of increments	1
Number in parallel	1
Roughness	100.000 μm

200mm length chosen for verification of standalone evaporator model.

The true piping network length can be included in the integrated boiler model.

Waterwall Pipe Lengths

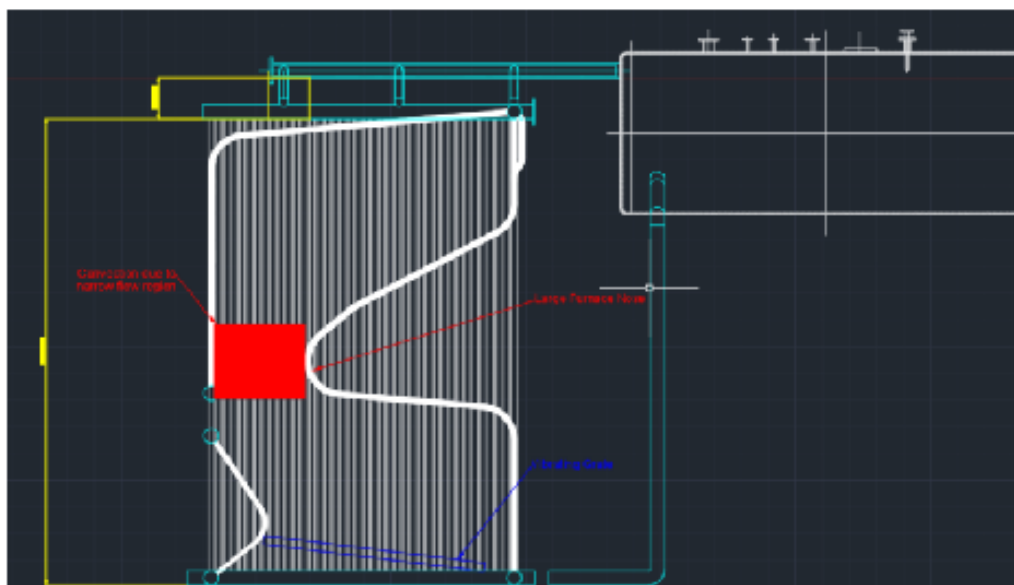


Figure D: Waterwall Pipe Lengths taken as Sidewall Height

Waterwalls: Pipe Inputs

Carbon Steel - AISI 1010| Metal (Solids) {Project}

Length	5.450 m
Wall thickness	4.060 mm
ID	55.380 mm
Number in parallel	122
Number of increments	10
Roughness	100.000 μm
Heat input	1240.129 kW

Length taken as vertical distance between bottom header and top header (i.e. length of side wall). The rear wall curves inside the furnace and the front wall curves into the roof but these lengths can be incorporated in the integrated model.

122 membrane tubes making up waterwall

Discretised into 10 increments for heat transfer accuracy (future proofing)
Heat input is designed such that the tank has a zero energy accumulation for the specified tank quality boundary condition.

Downcomer: Pipe Inputs

DN: 150 Schedule: 40 STD

Diameter (from schedule)	154.080 mm
Wall thickness (from schedule)	7.110 mm
Length	4.500 m
Number of increments	1
Number in parallel	2
Roughness	100.000 μm

Steam Extract: Pipe Inputs

DN: 50 Schedule: 80 XS

Diameter (from schedule)	49.220 mm
Wall thickness (from schedule)	5.540 mm
Length	200.000 mm
Number of increments	1
Number in parallel	1
Roughness	100.000 μm

200mm length chosen for verification of standalone evaporator model.

The true piping network length can be included in the integrated boiler model. This is not necessary since of boiler control volume stops at the steam extract from the drum.

Node Inputs



Elevations were given to the relevant flow nodes in an attempt to account for elevation heads. In some cases the effect of the elevation difference between nodes are overwritten if for example a fixed mass flow is specified on a pipe component linking to this node.

Node elevations taken as per JT site general arrangement drawings:

- Bottom of waterwall: 2.511m
- Bottom of tank: 6.867m
- Feedwater inlet node: 8.823m (100mm above top of tank)
- Steam extract outlet node: 8.823m (100mm above top of tank)
- Fluegas inlet and outlet nodes: 7.2m (middle of tank)

Branch connection heights

- Bottom of tank: 0m (REFERENCE DATUM)
- Top of tank: 1.856m
- Feedwater inlet branch: 1.856m (i.e. top of tank)
- Natural circulation inlet branch: 0.928m (i.e. at centre of tank)
- Steam extract branch: 1.856m (i.e. top of tank) - extract saturated liquid ($x=1$)
- Downcomer branch: 0m (i.e. bottom of tank) - to extract saturated liquid

Note: The branch heights are relative to the Two-Phase Tank (so the true elevations of the branches would include the elevation specified for the Two Phase Tank itself. The elevation specified for the Two Phase Tank is from the Ground Level to the bottom of the tank.

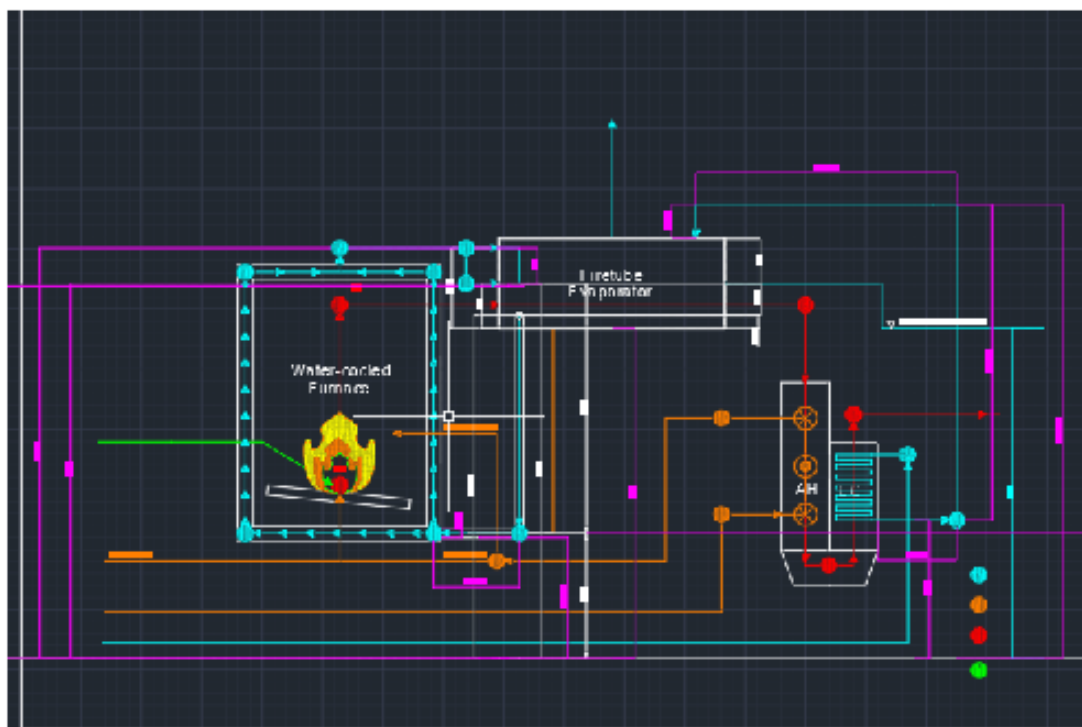


Figure E: Schematic of Boiler Architecture Indicating Elevation Heights

Composite Heat Transfer Inputs



Conduction

Conduction areas:	"Use upstream pipe areas"	[This would be the OD surface area of the tubes - in contact with the water]
Layers (Geometry)		
Thickness in element direction:	$t_{\text{tube}} = 4.06\text{-mm}$	[as per JT drawings]
Number of nodes:	2	[sufficient to capture conduction effects with limited computational expense]
Layers (Material Data)		
Material option:	"Select from data reference"	
Material major conductivity direction:	"Parallel"	
Material:	"Carbon Steel - AISI 1010 Metal (Solids) {Project}"	
Area discretisation scheme:	"Standard (average areas)"	

Configuration

Heat transfer grading factor:	1	[Default setting]
Order of discretisation:	"Second order"	[Default setting]

Convection Radiation And Wall Flux

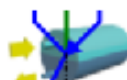
Upstream (Configuration)

Heat transfer option:	"Convection"	[mode of heat transfer from flue gas to pipe wall is convection]
Upstream (Convection)		
Convection area option:	"Use as specified in conduction"	
Convection coefficient option:	"Calculate h"	
Convection coefficient correlation specification:	"Tube inside (Gnielinski)"	
Temperature differencing scheme:	"Average temperature difference"	
Fin efficiency:	1	
Fouling factor:	$0 \frac{\text{m}^2 \cdot \text{K}}{\text{W}}$	[Fouling not included, has small effects]
Cross flow area option:	"Use cross area from flow element"	
Hydraulic diameter option:	"Specify hydraulic diameter"	
Hydraulic diameter:	$d_{\text{h,tube}} = 42.68\text{-mm}$	
Porosity option:	"Use porosity from flow element"	
Tube roughness option:	"Obtain from flow element"	

Downstream (Configuration)

Heat transfer option:	"Convection"	[mode of heat transfer from pipe wall to surrounding water]
Downstream (Convection)		
Convection area option:	"Use as specified in conduction"	
Convection coefficient option:	"Constant h"	
h:	$h_{\text{H}_2\text{O}} = 1529.252 \cdot \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$	[This is calculated in the Quick Script using the Churchill and Chu correlation]
Temperature differencing scheme:	"Average temperature difference"	
Fin efficiency:	1	
Fouling factor:	$0 \frac{\text{m}^2 \cdot \text{K}}{\text{W}}$	[Fouling not included, has small effects]

Two Phase Tank Inputs:



Flownex treats the *Two Phase Tank* as a special flow node with limited inputs. The nature of a flow node is that it should be completely filled with fluid but it allows for separation between the liquid and gas phases.

Fluids

Fluid data reference:

H2O - Water| General (Two Phase Fluids)
{Master}"

Solver guess values

Specify guess values:

"No"

Specify incondensable concentration guess value: "No"

Boundary Conditions

Elevation:

6.686m [This is the elevation to the bottom of
the tank as per JT drawings]

Geometry

Since the tank needs to be filled by fluid, it is necessary to discretise the evaporator to account for the true fluid volume. The evaporator is a horizontal shell but it contains both fire tubes and staybars which occupy some of the volume inside the shell.

The volume was discretised by modelling the true volume "shape" in Solidworks, slicing it into 1mm heights and extracting the top surface area at each slice. The combination of height and planar surface area will be used as inputs to a polygon table in Flwonex.

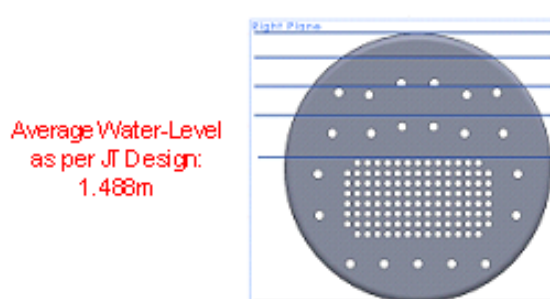


Figure F: True Fluid Volume
"Shape" (Solidworks)

Two-Phase Tank	
Tank height	1.856 m
Tank Level	1.486 m
Node volume	11.450 m ³
Total mass	7985.080 kg
Specific volume	0.001 m ³ /kg
Quality	0.003
P _{sat}	2999.998 kPa
T _{sat}	233.853 °C
Energy source	0.000 W

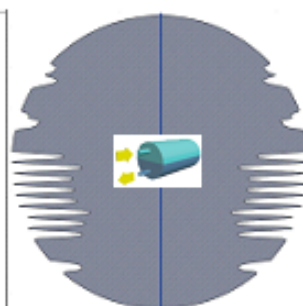


Figure G: Visual representation of flownex
polygon volume implementation

Geometry option: "Specify geometry"

Vessel shape: "Polygon"

Table option: "Use table"

Dimensions table:

Dimensions table		Click here to edit and remove...	...
[0]			
Geometry			
Height	0	m	▼
Area	0.563634	m ²	▼
[1]			
Geometry			
Height	0.001	m	▼
Area	0.690845	m ²	▼
[2]			
Geometry			
Height	0.002	m	▼
Area	0.798199	m ²	▼
[3]			
Geometry			
Height	0.003	m	▼
Area	0.892849	m ²	▼
[4]			

Figure D: First 4 entries of dimensions table (bottom of tank @0m and top of tank @1.865m)

Connection Geometry:

Connection specification: "Specify height"

Connections:

Connections		Click here to edit and remove...	...
[0]			
Connection			
Specify	Yes		▼
Identifier	Pipe - 8 Copy(002)		
Description	Feedwater Inlet		
Height	1.856	m	▼
[1]			
Connection			
Specify	Yes		▼
Identifier	Pipe - 21 Copy(002)		
Description	Steam Exit		
Height	1.856	m	▼
[2]			
Connection			
Specify	Yes		▼
Identifier	Pipe - 4 Copy(004)		
Description	Downcomer		
Height	0	m	▼
[3]			
Connection			
Specify	Yes		▼
Identifier	Pipe - 23 Copy(002)		
Description	Waterwalls		
Height	0.928	m	▼

Figure E: Branch connection heights

Boundary Conditions

Feedwater Inlet

Temperature: $T_{fw} = 118\text{ }^{\circ}\text{C}$ [as per MCR]
 Pressure: 3100.394 kPa [designed such that two phase tank pressure is as per MCR]

Steam Extract

Mass source: $-m_{\dot{\text{dot}}fw} = -4000 \cdot \frac{\text{kg}}{\text{hr}}$ [as per MCR. Mass sink at outlet]

Flue gas Inlet

Temperature: $T_{fg.in} = 946.089\text{ }^{\circ}\text{C}$ [as per analytical model]
 Pressure: $P_{fg.in} = 807.152\text{ kPa}$ [as per analytical model]

Flue gas outlet

Mass source: $-m_{\dot{\text{dot}}fg} = -6984 \cdot \frac{\text{kg}}{\text{hr}}$ [as per MCR. Mass sink at outlet]

Two Phase Tank

Quality: $\text{Quality} := 0.0034481823464$ [designed such that tank level is as per JT's designs]

Waterwall

Heat input: $Q_{\dot{\text{dot}}WW} := 1240.122\text{ kW}$ [designed such that tank energy accumulation is nullified]

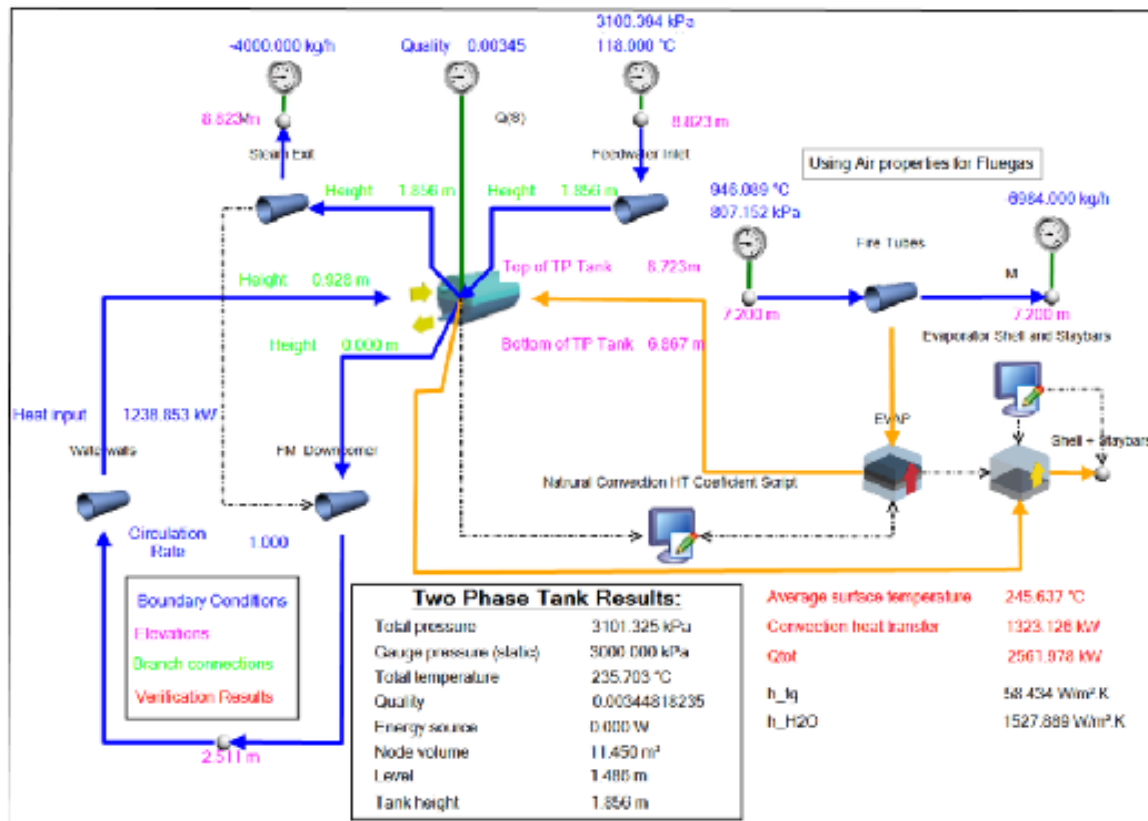
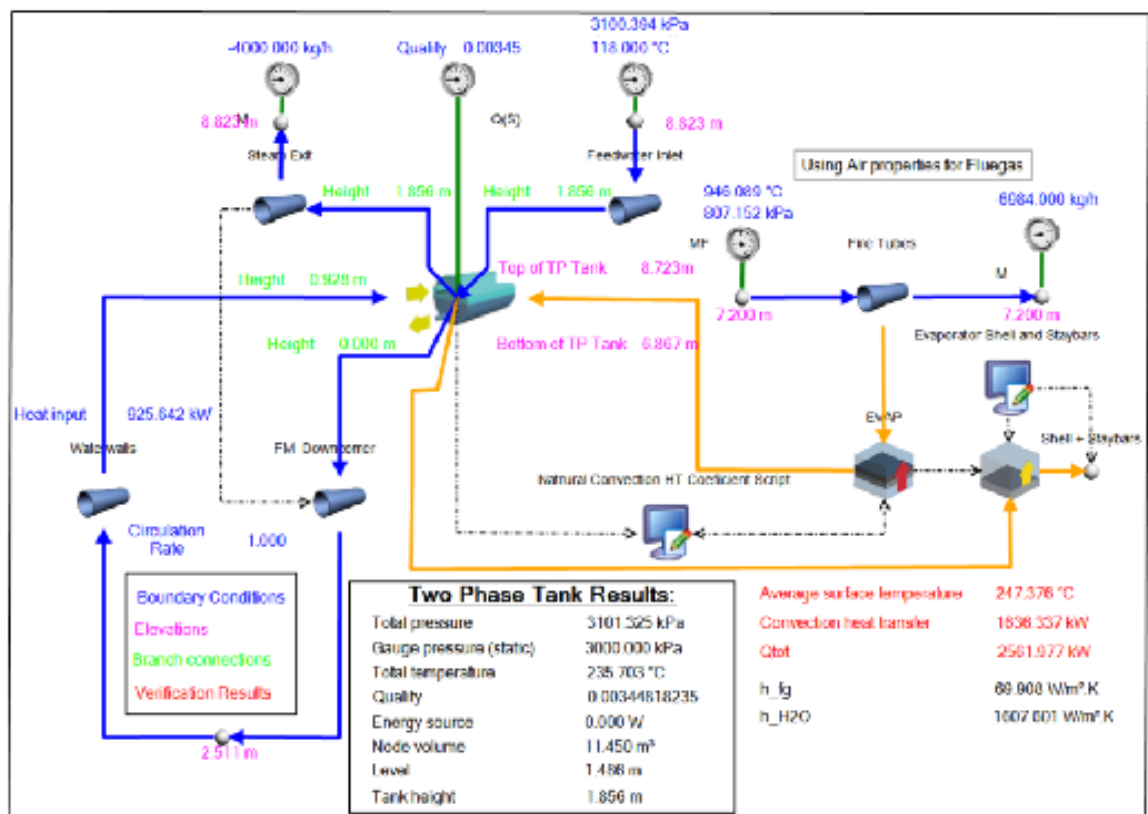
Note: At steady state operation, the quality constraint on the two-phase tank ensures the correct level. But the quality constraint requires that all energy inlets and outlets balance to produce this level. As such it calculates the energy accumulation in the tank. In the bigger picture of the entire boiler system, the water that enters the twophase tank from the waterwalls would be heated via radiation in the furnace. So for the purposes of verifying the standalone evaporator model, a heat input has been given to the waterwalls (emulating the radiative heat transfer).

The heat input that nullifies the tank energy source should then be close to the waterwall heat uptake from combustion in the furnace in the intergrated whole boiler model.

Downcomer

Circulation Rate: $CR := 1$

The natural circulation is a complex phenomena which requires accurate pressure drop characteristic of the complex flowpaths of the furnace walls which is outside the scope of this research. Therefore a fixed mass flow is applied to the downcomer pipe, thereby overriding any pressure drop characteristics in the circulation loop. The natural circulation mass flow has been set equal to the steam extraction for the case of the stand alone evaporator verification. The circulation rate can be much higher, but its contibution the heat transfer is not significant enough to warrant accurate pressure drop modelling. The focus is on the dynamic behavior. The circulation rate should atleast be high enough to avoid dry-out inside the waterwalls. The circulation rate is set to 10 in the integrated model.

FLOWNEX RESULTS (air props on fluegas side)**FLOWNEX RESULTS (fluegas props on fluegas side)**

Comparison of MCR, Analytical, and Flownex Results

This MCR data does not have data for the circulation loop nor does it have data for the inlet fluegas stream (i.e. at the furnace exit). So the heat transfer rate as per MCR is actually the combination of the furnace radiation and the convection in the evaporator. This is the heat required to take the subcooled feedwater to a saturated steam state at the exit of the evaporator.

Heat Transfer Rate (MCR Data):

$$Q_{\text{dot.totalMCR}} := Q_{\text{dot.MCR}} = 2561.843 \text{ kW}$$

Analytical Results (air props for flue gas props)

Heat Transfer Rate of Evaporator only (analytical using air props for flue gas):

$$Q_{\text{dot.EVAP.analytical.air}} := Q_{\text{dot.fg}} = 1327.055 \text{ kW}$$

Heat Transfer Rate for Waterwall heat uptake (analytical using air props for flue gas):

$$Q_{\text{dot.ww.analytical.air}} := Q_{\text{dot.MCR}} - Q_{\text{dot.EVAP.analytical.air}} = 1234.788 \text{ kW}$$

Tube wall temperature:

$$T_{\text{t.wall.analytical.air}} := T_{\text{t.wall}} = 245.663 \text{ }^{\circ}\text{C}$$

Flownex Results (air props for flue gas props)

Total Furnace+Evaporator Heat Transfer Rate (Flownex using air props):

$$Q_{\text{dot.totalFlownex.air}} := 2561.978 \text{ kW}$$

Heat Transfer Rate of Evaporator only (Flownex using air props for flue gas):

$$Q_{\text{dot.EVAP.Flownex.air}} := 1323.126 \text{ kW}$$

Heat Transfer Rate for Waterwall heat uptake (Flownex using air props for flue gas):

$$Q_{\text{dot.ww.Flownex.air}} := 1238.853 \text{ kW}$$

Sense check: $Q_{\text{dot.totalFlownex.air}} - Q_{\text{dot.EVAP.Flownex.air}} = 1238.852 \text{ kW}$

$$T_{\text{t.wall.Flownex.air}} := 245.637 \text{ }^{\circ}\text{C}$$

Flownex Results (flue gas props for flue gas props)

Total Furnace+Evaporator Heat Transfer Rate (Flownex using real flue gas props):

$$Q_{\text{dot.totalFlownex.fg}} := 2561.977 \text{ kW}$$

Heat Transfer Rate of Evaporator only (Flownex using real flue gas props):

$$Q_{\text{dot.EVAP.Flownex.fg}} := 1636.337 \text{ kW}$$

Heat Transfer Rate for Waterwall heat uptake (Flownex using real flue gas props):

$$Q_{\text{dot.ww.Flownex.fg}} := 925.642 \text{ kW}$$

Sense check: $Q_{\text{dot.totalFlownex.fg}} - Q_{\text{dot.EVAP.Flownex.fg}} = 925.64 \text{ kW}$

$$T_{\text{t.wall.Flownex.fg}} := 247.376 \text{ }^{\circ}\text{C}$$

$$\text{Error}_{Q_{\text{evap.Flownex.air.vs.analytical.air}}} := \frac{Q_{\text{dot.EVAP.Flownex.air}} - Q_{\text{dot.EVAP.analytical.air}}}{Q_{\text{dot.EVAP.analytical.air}}} = -0.296\%$$

$$\text{Error}_{T_{\text{wall.Flownex.air.vs.analytical.air}}} := \frac{T_{\text{t.wall.Flownex.air}} - T_{\text{t.wall.analytical.air}}}{T_{\text{t.wall.analytical.air}}} = -0.005\%$$

$$\text{Error}_{Q_{\text{tot.Flownex.air.vs.MCR}}} := \frac{Q_{\text{dot.total.Flownex.air}} - Q_{\text{dot.total.MCR}}}{Q_{\text{dot.total.MCR}}} = 0.005\%$$

$$\text{Error}_{Q_{\text{tot.Flownex.fg.vs.MCR}}} := \frac{Q_{\text{dot.total.Flownex.fg}} - Q_{\text{dot.total.MCR}}}{Q_{\text{dot.total.MCR}}} = 0.005\%$$

Appendix D. Water-Cooled Furnace “Example” Calculation

Water-Cooled Furnace Example Calculations

Fluid properties

Include << C:\Users\pmb87\OneDrive\Documents\MASTERS\009. MathCad Models\IAPWS-IF97 Water-Prop rev2.0 Prime5.0.mcdx

Include << C:\Users\pmb87\OneDrive\Documents\MASTERS\009. MathCad Models\Jacques' Furnace MathCad Model\WF\Fluegas-Prop rev 2.0 Prime5.0-ATProM-PG.mcdx

$$1 \text{ kJ} := 1000 \text{ J}$$

$$1 \text{ kmol} := 1000 \text{ mol}$$

$$P_{atm} := 101.325 \text{ kPa}$$

Important values from reference sheets:

4) SOLID PHASE PROPERTIES:

The solid phase components in the flue gas is modelled with constant specific heat values. For the enthalpy functions the reference temperature is set to 25 C.

Previous version had 273.15K (0 C) as the reference temperature but it is now 25C to align with the reference for HHV

$$c_{coal} := 1.38 \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

$$c_{UC} := 0.71 \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

$$c_{FA} := 0.73 \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

$$T_{h_ref} := 25 \text{ } ^\circ\text{C}$$

$$h_{coal}(Temp) := c_{coal} \cdot (Temp - T_{h_ref})$$

$$h_{UC}(Temp) := c_{UC} \cdot (Temp - T_{h_ref})$$

$$h_{FA}(Temp) := c_{FA} \cdot (Temp - T_{h_ref})$$

$$\rho_{coal} := 2150 \cdot \frac{\text{kg}}{\text{m}^3}$$

Figure 0: Screenshot from Fluegas Property Calculator Mathcad Reference Sheet

WATER-COOLED FURNACE - Combustion Example

The analytical furnace model has been developed to partially verify its results against the MCR data. Note that the MCR data does not contain a furnace exit condition. As such the furnace exit condition on the fluegas side will be taken as the condition specified as the flue gas input to the tubes in the firetube evaporator analytical model.

Additionally the fuel composition and combustion methodology used to produce the MCR data is unknown so the whole boiler energy balance is not possible to verify. The combustion heat and flue gas composition is very sensitive to the fuel composition so this section just serves to show the methodology based on an assumed fuel composition and other simplifying assumptions.

$$Q'_{rad} = C \cdot \sigma \cdot A \cdot (\epsilon_f \cdot T_f^4 - \alpha_f \cdot T_W^4)$$

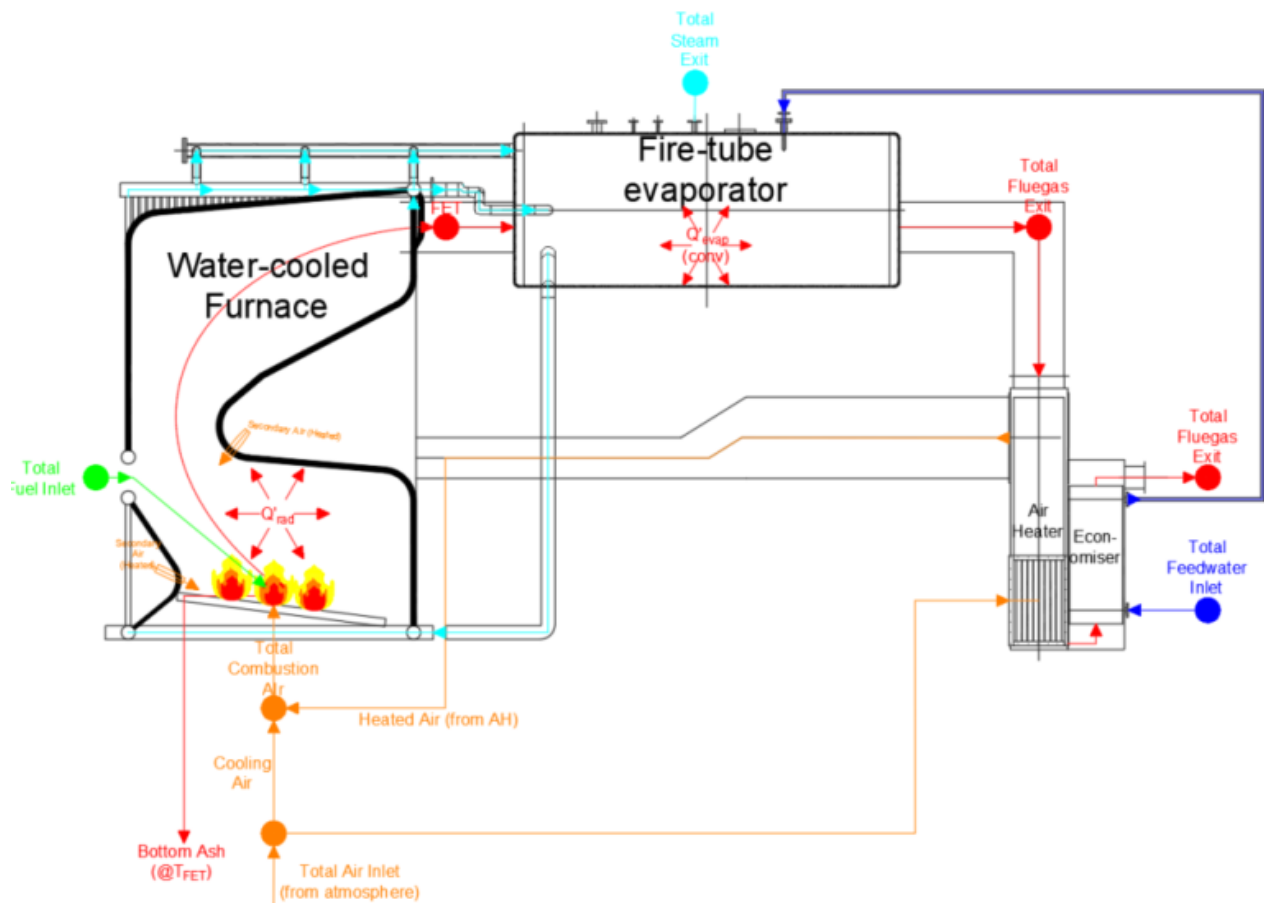


Figure 1: Overall Boiler Process Flow Diagram

References:

- [1] TO400 TORRIPAC BOILER PROCESS FLOW DIAGRAM 68756-03, rev 0
- [2] FIRE-TUBE EVAPORATOR VERIFICATION (Analytical vs. MCR vs. Flownex)

MCR Data [1]: Water Side Inputs:

$$m'_{steam,MCR} := 4000 \frac{kg}{hr} \quad [1]$$

Main steam mass flow rate: $m'_{main,steam} := m'_{steam,MCR} = 1.111 \frac{kg}{s}$

Main steam pressure: $P_{main,steam} := 3000 \text{ kPa} + P_{atm} = 3101.325 \text{ kPa} \quad [1]$

Main steam temperature: $T_{main,steam} := T_{steam}(P_{main,steam}, "", "", "") = 235.708 \text{ }^{\circ}\text{C}$

Main steam enthalpy: $h_{main,steam} := h_{steam}(P_{main,steam}, "", "", 1, "") = 2803.284 \frac{kJ}{kg}$

Feedwater mass flow rate: $m'_{fw} := m'_{main,steam} = 1.111 \frac{kg}{s} \quad [1]$

Feedwater pressure (EVAP in): $P_{fw,EVAP} := 3183 \text{ kPa} + P_{atm} = 3284.325 \text{ kPa} \quad [1]$

Feedwater temperature (EVAP in): $T_{fw,EVAP} := 118 \text{ }^{\circ}\text{C} \quad [1]$

Feedwater enthalpy (EVAP in): $h_{fw,EVAP} := h_{steam}(P_{fw,EVAP}, T_{fw,EVAP}, "", "", "") = 497.482 \frac{kJ}{kg}$

Sense check (sub-cooled state): $x_{fw,EVAP} := x_{steam}(P_{fw,EVAP}, T_{fw,EVAP}, "", "", "") = -0.302$

Feedwater pressure (ECON in): $P_{fw,ECON} := 3236 \text{ kPa} + P_{atm} = 3337.325 \text{ kPa} \quad [1]$

Feedwater temperature (ECON in): $T_{fw,ECON} := 80 \text{ }^{\circ}\text{C} \quad [1]$

Feedwater enthalpy (ECON in): $h_{fw,ECON} := h_{steam}(P_{fw,ECON}, T_{fw,ECON}, "", "", "") = 337.567 \frac{kJ}{kg}$

Sense check (sub-cooled state): $x_{fw,ECON} := x_{steam}(P_{fw,ECON}, T_{fw,ECON}, "", "", "") = -0.396$

Heat to Convert subcooled feedwater to saturated steam:

Drum inlet to drum outlet: $Q'_{steam,1} := m'_{main,steam} \cdot (h_{main,steam} - h_{fw,EVAP}) = 2562.002 \text{ kW}$

Economiser inlet to drum outlet: $Q'_{steam,2} := m'_{main,steam} \cdot (h_{main,steam} - h_{fw,ECON}) = 2739.685 \text{ kW}$

$$Q'_{steam} := Q'_{steam,1} = 2562.002 \text{ kW}$$

$$Q'_{steam} := Q'_{steam,2} = 2739.685 \text{ kW}$$

$$x_{drum} := 0.00344818235$$

[Quality evaluated at the average water level of 1.488m at the main steam pressure of the MCR saturated condition]

$$h_{nat,circ,out} := h_{steam}(P_{main,steam}, "", "", 0, "") = 1017.117 \frac{kJ}{kg}$$

$$h_{nat,circ,in} := h_{steam}(P_{main,steam}, "", "", x_{drum}, "") = 1023.276 \frac{kJ}{kg}$$

$$Q_{steam,ww} := m'_{main,steam} \cdot (h_{nat,circ,in} - h_{nat,circ,out}) = 6.843 \text{ kW}$$

Air Side Inputs:

As seen in Figure 2 below, there are two air supply streams:

1. Cooling air (for grate cooling)

- Supplied from ambient conditions
- Blowed by CA fan to the front zone of the grate

2. Heated air

- Supplied from ambient conditions
- Blowed by FD fan to air heater
- Heated in the air heater
- Splits into primary and secondary air streams
- Primary air - supplied to the middle and rear zones of the grate
- Secondary air - injected into the free-board (above the grate) through nozzles in the lower front wall and the rear wall

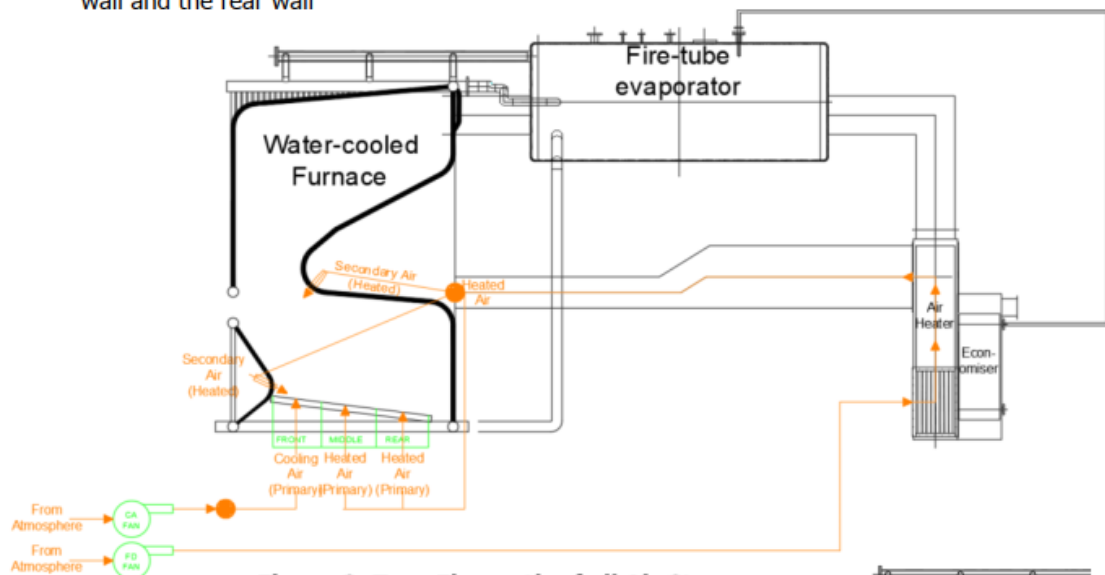


Figure 2: True Flowpath of all Air Streams

Note all air streams (including) the cooling are injected into the furnace and therefore all partake in the combustion with the fuel.

From a simplified combustion analysis point of view, where we assume instantaneous and complete combustion, we are only interested in a single air temperature to model the sensible heat of the air stream. As such the temperature of the air as seen by the furnace for combustion purposes will be taken as the mixing temperature of the heated air and the cooling air.

$$\begin{aligned} \text{Cooling air mass flow rate: } m'_{CA} &:= 539 \frac{\text{kg}}{\text{hr}} & [1] \\ \text{Cooling air pressure: } P_{CA} &:= P_{atm} = 101.325 \text{ kPa} & [\text{From atmosphere}] \\ \text{Cooling air temperature: } T_{CA} &:= 27 \text{ }^\circ\text{C} & [1] \end{aligned}$$

$$\begin{aligned} \text{Heated air mass flow rate: } m'_{HA} &:= 5390 \frac{\text{kg}}{\text{hr}} & [1] \\ \text{Heated air pressure: } P_{HA} &:= P_{atm} = 101.325 \text{ kPa} & [\text{Assume negligible } \Delta P] \\ \text{Heated air temperature: } T_{HA} &:= 213.57 \text{ }^\circ\text{C} & [1] \\ & \quad T_{HA} := 197 \text{ }^\circ\text{C} & [1] \\ \text{Air specific humidity: } \omega_{air} &:= 0.0147 & [\text{JT Mathcad}] \end{aligned}$$

Note: A heated air temperature of 213.57 degC ensures a mixing temperature of 197 deg C. But this was not used since the heated air temperature as per [1] is 197 degC.

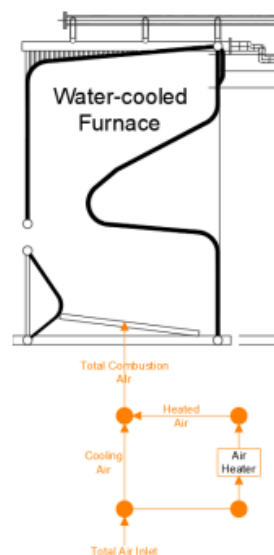


Figure 3: Simplified Air Flow Pat

Total combustion air mass flow rate: $m'_{COMBA} := m'_{CA} + m'_{HA} = 1.647 \frac{kg}{s}$ [Continuity]

Furnace excess air ratio: $air_{furn.excess} := 0.248807932934316$ [Excess air as calculated in
Theoretical Furnace Air Ratio: $\alpha_{air} := 100\% + air_{furn.excess} = 1.249$ flownex for the MCR air
Final Gas excess air ratio: $air_{final.gas.excess} := 0.248807932934316$ and fuel flowrates. NOTE:
Theoretical Final Gas Air Ratio: $\alpha_{exit} := 100\% + air_{final.gas.excess} = 1.249$ 25% excess air is typical
for similar furnaces.]

Calculate ambient air (moist air) composition:

$$m'_{air.MCR} := m'_{COMBA} = 1.647 \frac{kg}{s} \quad m'_{air.MCR} = 5929 \frac{kg}{hr}$$

Dry air composition: $Y_{O_2} := 0.2096$ [Reference??]
 $Y_{N_2} := 1 - Y_{O_2} = 0.7904$

Dry air molar mass: $M_{air} := Y_{O_2} \cdot M_{O_2} + Y_{N_2} \cdot M_{N_2} = 28.848803072 \frac{kg}{kmol}$ [Ambient air composition method
as per Flownex implementation]

[Assume that dry air only contains O2 and N2]

$$m'_{COMBA.MCR} := m'_{CA} + m'_{HA} = 1.647 \frac{kg}{s}$$

O2 in air: $Y_{O_2.air} := Y_{O_2} \cdot \frac{M_{O_2}}{M_{air}} = 0.232486195814121$

N2 in air: $Y_{N_2.air} := Y_{N_2} \cdot \frac{M_{N_2}}{M_{air}} = 0.767513804185879$

Sense check: $Y_{O_2.air} + Y_{N_2.air} = 1$

Moist air composition: $Y_{amb.air} := \begin{bmatrix} 0 \\ Y_{N_2.air} \\ \frac{1 + \omega_{air}}{Y_{O_2.air}} \\ \frac{1 + \omega_{air}}{\omega_{air}} \\ \frac{1 + \omega_{air}}{0} \end{bmatrix} = \begin{bmatrix} 0 \\ 0.75639 \\ 0.22912 \\ 0.01449 \\ 0 \end{bmatrix} \quad gases = \begin{bmatrix} "CO_2" \\ "N_2" \\ "O_2" \\ "H_2O" \\ "SO_2" \end{bmatrix}$

Sense check: $\sum Y_{amb.air} = 1$

Cooling air enthalpy: $h_{CA} := h_{fluegas}(T_{CA}, Y_{amb.air}) = 38.619 \frac{kJ}{kg}$

Heated air enthalpy: $h_{HA} := h_{fluegas}(T_{HA}, Y_{amb.air}) = 211.927 \frac{kJ}{kg}$

Combustion air enthalpy: $h_{COMBA} := \frac{m'_{CA} \cdot h_{CA} + m'_{HA} \cdot h_{HA}}{m'_{COMBA}} = 196.171 \frac{kJ}{kg}$

Combustion air temperature: $T_{comb.air}(T_{guess}) := \text{root}((h_{fluegas}(T_{guess}, Y_{amb.air}) - h_{COMBA}), T_{guess}) \quad T_{guess} := 170 \text{ } ^\circ\text{C}$

$$T_{COMBA} := T_{comb.air}(T_{guess}) = 181.875 \text{ } ^\circ\text{C}$$

Sense check: $h_{fluegas}(T_{COMBA}, Y_{amb.air}) = 196.171 \frac{kJ}{kg} \quad h_{COMBA} = 196.171 \frac{kJ}{kg}$

Ambient Conditions:

Ambient Pressure: $P_{amb} := P_{atm} = 101.325 \text{ kPa}$ [1]

Ambient Temperature: $T_{amb} := 27 \text{ }^{\circ}\text{C}$ [1]

Fuel Side Inputs:

Fuel temperature: $T_{fuel} := T_{amb} = 27 \text{ }^{\circ}\text{C}$ [Assumed equal to ambient temperature]

Fuel Pressure: $P_{fuel} := P_{amb} = 101.325 \text{ kPa}$ [Assumed equal to ambient pressure]

Fuel sensible heat: $h_{fuel} := h_{coal}(T_{fuel}) = 2.76 \frac{\text{kJ}}{\text{kg}}$

Fuel (Bagasse) Higher heating Value: $HHV_{fuel} := 8.049 \cdot 10^6 \frac{\text{J}}{\text{kg}} = 8049 \frac{\text{kJ}}{\text{kg}}$ [JT Mathcad]

Mass Fractions (Wet Basis):

Fly Ash: $Y_{FA} := 0.2$

Bottom Ash: $Y_{BA} := 1 - Y_{FA} = 0.8$

Carbon (Participating): $Y_C := 0.19041$

Unburned Carbon: $Y_{UC} := 0.0061$

Carbon (Total): $Y_{C,tot} := Y_C + Y_{UC} = 0.197$

Hydrogen: $Y_H := 0.0263$

Oxygen: $Y_O := 0.18576$

Sulphur: $Y_S := 10^{-16} = 1 \cdot 10^{-16}$

Water: $Y_{H2O} := 0.57$

Ash: $Y_{ASH} := 0.02043$

Nitrogen: $Y_N := 0.0010$

Mass Fraction Check: $Y_{C,tot} + Y_H + Y_O + Y_N + Y_S + Y_{H2O} + Y_{ASH} = 1$ $Y_{H2O,dry} := \frac{Y_{H2O}}{1 - Y_{H2O}} = 1.326$

Mass Fractions (Dry Basis):

$$Y_{C,tot,dry} := \frac{Y_{C,tot}}{1 - Y_{H2O}} = 0.457$$

$$Y_{H,dry} := \frac{Y_H}{1 - Y_{H2O}} = 0.061$$

$$Y_{O,dry} := \frac{Y_O}{1 - Y_{H2O}} = 0.432$$

$$Y_{N,dry} := \frac{Y_N}{1 - Y_{H2O}} = 0.002$$

$$Y_{S,dry} := \frac{Y_S}{1 - Y_{H2O}} = 0$$

$$Y_{ASH,dry} := \frac{Y_{ASH}}{1 - Y_{H2O}} = 0.048$$

Mass Fractions (Wet Basis) for a new moisture content:

New fuel moisture content: $Y_{H2O,new} := 44\%$ [Renegade moisture test results, 2021]

Unburned Carbon: $Y_{UC} := 0.0061$

Carbon (Total): $Y_{C,tot} := Y_{C,tot,dry} \cdot (1 - Y_{H2O,new}) = 0.256$

Carbon (Participating): $Y_C := Y_{C,tot} - Y_{UC}$

Hydrogen: $Y_H := Y_{H,dry} \cdot (1 - Y_{H2O,new}) = 0.034$

Oxygen: $Y_O := Y_{O,dry} \cdot (1 - Y_{H2O,new}) = 0.242$

Nitrogen: $Y_N := Y_{N,dry} \cdot (1 - Y_{H2O,new}) = 0.001$

Sulphur: $Y_S := Y_{S,dry} \cdot (1 - Y_{H2O,new}) = 1.30232558139535 \cdot 10^{-16}$

Water: $Y_{H2O} := Y_{H2O,new} = 0.44$

Ash: $Y_{ASH} := Y_{ASH,dry} \cdot (1 - Y_{H2O,new}) = 0.027$

Mass Fraction Check: $Y_{C,tot} + Y_H + Y_O + Y_N + Y_S + Y_{H2O} + Y_{ASH} = 1$

Formation Enthalpies Evaporation Enthalpies

$$\begin{bmatrix} \text{"C"} \\ \text{"H"} \\ \text{"O"} \\ \text{"N"} \\ \text{"S"} \end{bmatrix} h_{form} := \begin{bmatrix} 32765 \\ 119959 \\ 0 \\ 0 \\ 9256 \end{bmatrix} \cdot \frac{\text{kJ}}{\text{kg}} \quad h_{evap} := \begin{bmatrix} 0 \\ 21820 \\ 0 \\ 0 \\ 0 \end{bmatrix} \cdot \frac{\text{kJ}}{\text{kg}} \quad \text{Carbon energy content: } HHV_C := h_{form_0} = 32765 \frac{\text{kJ}}{\text{kg}}$$

Specific heat capacities of solid fuel constituents 9from reference Mathcad worksheet:

$$c_{coal} = 1.38 \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

$$c_{UC} = 0.71 \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

$$c_{FA} = 0.73 \frac{\text{kJ}}{\text{kg} \cdot \text{K}}$$

Calculated CV:

$$CV_{calc} := \left(Y_C \cdot h_{form_0} + Y_H \cdot h_{form_1} + Y_O \cdot h_{form_2} + Y_N \cdot h_{form_3} + Y_S \cdot h_{form_4} \right) \downarrow = 13041.448 \frac{\text{kJ}}{\text{kg}} \\ + \left(Y_C \cdot h_{evap_0} + Y_H \cdot h_{evap_1} + Y_O \cdot h_{evap_2} + Y_N \cdot h_{evap_3} + Y_S \cdot h_{evap_4} \right)$$

Combustion Heat Input:

Rather than using the fuel flow as per MCR, the fuel flow will be calculated in this analytical model with the MCR fuel flow as the target.

Combustion calculations

Convert all fuel mass fractions (except ash) to molar amounts per kg of fuel.

Number of Moles of each Fuel Constituent:

$$N_C := \frac{Y_C}{M_C} \cdot 1 \text{ kg} = 20.799 \text{ mol}$$

$$N_H := \frac{Y_H}{M_H} \cdot 1 \text{ kg} = 33.981 \text{ mol}$$

$$N_O := \frac{Y_O}{M_O} \cdot 1 \text{ kg} = 15.12 \text{ mol}$$

$$N_N := \frac{Y_N}{M_N} \cdot 1 \text{ kg} = 0.093 \text{ mol}$$

$$N_S := \frac{Y_S}{M_S} \cdot 1 \text{ kg} = 0 \text{ mol}$$

$$N_{H_2O} := \frac{Y_{H_2O}}{M_{H_2O}} \cdot 1 \text{ kg} = 24.424 \text{ mol}$$

Mixture Moles: Number of Moles of Fuel:

$$N_{fuel} := N_C + N_O + N_S + N_H + N_N = 69.994 \text{ mol}$$

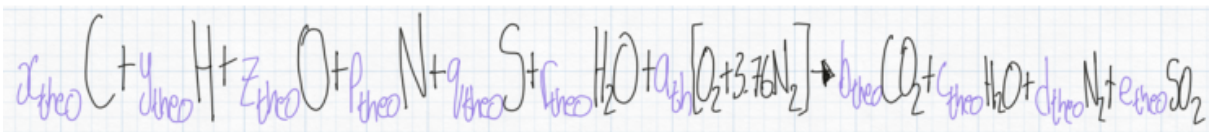
Mixture Molar Mass: Molar Mass of Fuel:

$$M_{fuel} := \frac{N_C}{N_{fuel}} \cdot M_C + \frac{N_O}{N_{fuel}} \cdot M_O + \frac{N_S}{N_{fuel}} \cdot M_S + \frac{N_H}{N_{fuel}} \cdot M_H + \frac{N_N}{N_{fuel}} \cdot M_N = 0.008 \frac{\text{kg}}{\text{mol}}$$

Calculate Theoretical Air-Fuel Ratio:

$$\text{kg}_{fuel} := \text{kg}$$

$$\text{kg}_{air} := \text{kg}$$



$$x_{theo} := N_C$$

$$y_{theo} := N_H$$

$$z_{theo} := N_O$$

$$p_{theo} := N_N$$

$$q_{theo} := N_S$$

$$r_{theo} := N_{H_2O}$$

$$b_{theo} := x_{theo} = 20.799 \text{ mol}$$

$$c_{theo} := \frac{y_{theo} + 2 \cdot r_{theo}}{2} = 41.414 \text{ mol}$$

$$e_{theo} := q_{theo} = 0 \text{ mol}$$

$$a_{theo} := \frac{2 \cdot b_{theo} + c_{theo} + 2 \cdot e_{theo} - z_{theo} - r_{theo}}{2} = 21.734 \text{ mol}$$

$$d_{theo} := \frac{a_{theo} \cdot 3.7619 \cdot 2 + p_{theo}}{2} = 81.809 \text{ mol}$$

$$AF_{theo} := \frac{a_{theo} \cdot (2 \cdot M_O + 3.7619 \cdot 2 \cdot M_N)}{1 \text{ kg}_{fuel}} = 2.986 \frac{\text{kg}_{air}}{\text{kg}_{fuel}}$$

Calculate Actual Air-Fuel Ratio:

Calculate furnace and boiler exit air fuel ratios and water in air mole amounts:

$$AF_{actual} := AF_{theo} \cdot \alpha_{air} = 3.729$$

$$AF_{actual,exit} := AF_{theo} \cdot \alpha_{exit} = 3.729$$

Calculate the amount of moisture entering the combustion control volume with the actual dry air:

$$m_{H_2O,air} := \omega_{air} \cdot AF_{actual} = 0.0548 \frac{kg}{kg}$$

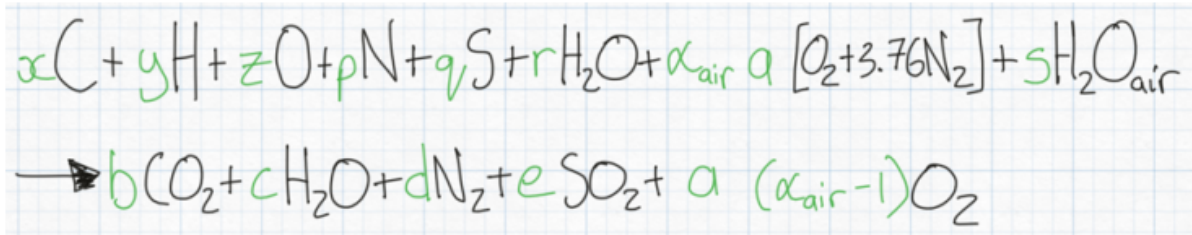
already per kg fuel

$$m_{H_2O,air,exit} := \omega_{air} \cdot AF_{actual,exit} = 0.0548 \frac{kg}{kg}$$

$$N_{H_2O,air} := \frac{m_{H_2O,air}}{M_{H_2O}} \cdot 1 \text{ kg} = 3.043 \text{ mol}$$

$$N_{H_2O,air,exit} := \frac{m_{H_2O,air,exit}}{M_{H_2O}} \cdot 1 \text{ kg} = 3.043 \text{ mol}$$

Using the moles of water entering with the combustion air the final form of the chemical reaction can be balanced:



Balance furnace chemical reaction to find gas composition on mass basis:

$$xC + yH + zO + pN + qS + rH_2O + \alpha_{air} a^{th} \cdot (O_2 + 3.76 N_2) + sH_2O = bCO_2 + cH_2O + dN_2 + eSO_2 + a^{th} \cdot (\alpha_{air} - 1) \cdot O_2$$

$$x := N_C \quad y := N_H \quad z := N_O \quad p := N_N \quad q := N_S \quad r := N_{H_2O} \quad s := N_{H_2O,air}$$

$$b := x \quad c := \frac{y + 2 \cdot r + 2 \cdot s}{2} \quad e := q$$

$$a := z + r + \alpha_{air} \cdot a \cdot 2 + s - 2 \cdot b - c - 2 \cdot e - a \cdot (\alpha_{air} - 1) \cdot 2 \xrightarrow{\text{solve, } a} 21.734284513773154062 \cdot \text{mol}$$

$$d := \frac{p + \alpha_{air} \cdot a \cdot 3.7619 \cdot 2}{2} \quad \frac{2 \cdot b + c + 2 \cdot e - z - r - s}{2} = 21.7342845137732 \text{ mol}$$

$$N_{fg} := a \cdot (\alpha_{air} - 1) + b + c + d + e = 172.816 \text{ mol}$$

Calculate Mole Fractions of Fluegas Constituents:

$$X_{fg} := \begin{bmatrix} \frac{b}{N_{fg}} \\ \frac{d}{N_{fg}} \\ \frac{(\alpha_{air} - 1) \cdot a}{N_{fg}} \\ \frac{c}{N_{fg}} \\ \frac{e}{N_{fg}} \end{bmatrix} = \begin{bmatrix} 0.1204 \\ 0.5911 \\ 0.0313 \\ 0.2573 \\ 2.3502 \cdot 10^{-17} \end{bmatrix}$$

$$gases = \begin{bmatrix} \text{"CO2"} \\ \text{"N2"} \\ \text{"O2"} \\ \text{"H2O"} \\ \text{"SO2"} \end{bmatrix}$$

$$\sum X_{fg} = 1$$

Calculate Mass Fractions of Fluegas Constituents:

$$M_{fg} := \begin{bmatrix} M_{CO_2} \\ M_{N_2} \\ M_{O_2} \\ M_{H_2O} \\ M_{SO_2} \end{bmatrix} = \begin{bmatrix} 0.04401 \\ 0.02801 \\ 0.032 \\ 0.01802 \\ 0.06406 \end{bmatrix} \frac{\text{kg}}{\text{mol}}$$

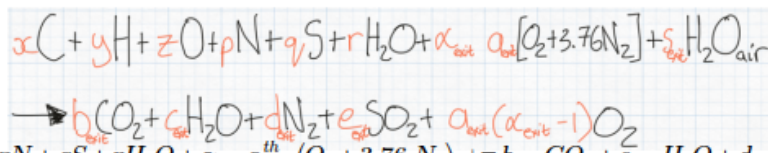
$$M_{fg,avg} := X_{fg} \cdot M_{fg} = 0.0274914 \frac{\text{kg}}{\text{mol}}$$

$$Y_{fg} := \frac{\overrightarrow{X_{fg} \cdot M_{fg}}}{M_{fg,avg}} = \begin{bmatrix} 0.1927 \\ 0.6023 \\ 0.0364 \\ 0.1686 \\ 5.4767 \cdot 10^{-17} \end{bmatrix} \frac{\text{kg}}{\text{kg}}$$

$$\sum Y_{fg} = 1$$

$$Y_{fg} = \begin{bmatrix} 0.192671725078338 \\ 0.602328698373747 \\ 0.036422002254334 \\ 0.168577574293582 \\ 0 \end{bmatrix} \quad \text{gases} = \begin{bmatrix} \text{"CO2"} \\ \text{"N2"} \\ \text{"O2"} \\ \text{"H2O"} \\ \text{"SO2"} \end{bmatrix}$$

Balance chemical reaction to find gas composition on mass basis at boiler exit:



$$xC + yH + zO + pN + qS + rH_2O + \alpha_{\text{exit}} a^{\text{th}} \cdot (\text{O}_2 + 3.76 \text{N}_2) + s_{\text{exit}} \text{H}_2\text{O} = b_{\text{exit}} \text{CO}_2 + c_{\text{exit}} \text{H}_2\text{O} + d_{\text{exit}} \text{N}_2 + e_{\text{exit}} \text{SO}_2 + a^{\text{th}} \cdot (\alpha_{\text{exit}} - 1) \cdot \text{O}_2$$

$$s_{\text{exit}} := N_{\text{H}_2\text{O},\text{air},\text{exit}} \quad b_{\text{exit}} := x \quad e_{\text{exit}} := q$$

$$c_{\text{exit}} := \frac{y + 2 \cdot r + 2 \cdot s_{\text{exit}}}{2}$$

$$a_{\text{exit}} := \frac{2 \cdot b_{\text{exit}} + c_{\text{exit}} + 2 \cdot e_{\text{exit}} - z - r - s_{\text{exit}}}{2} = 21.734 \text{ mol}$$

$$d_{\text{exit}} := \frac{p + \alpha_{\text{exit}} \cdot a_{\text{exit}} \cdot 3.76 \cdot 2}{2}$$

$$N_{fg,\text{exit}} := a_{\text{exit}} \cdot (\alpha_{\text{exit}} - 1) + b_{\text{exit}} + c_{\text{exit}} + d_{\text{exit}} + e_{\text{exit}} = 172.764 \text{ mol}$$

$$X_{fg,\text{exit}} := \begin{bmatrix} \frac{b_{\text{exit}}}{N_{fg,\text{exit}}} \\ \frac{c_{\text{exit}}}{N_{fg,\text{exit}}} \\ \frac{d_{\text{exit}}}{N_{fg,\text{exit}}} \\ \frac{e_{\text{exit}}}{N_{fg,\text{exit}}} \\ \frac{a_{\text{exit}} (\alpha_{\text{exit}} - 1)}{N_{fg,\text{exit}}} \end{bmatrix} = \begin{bmatrix} 0.12 \\ 0.591 \\ 0.031 \\ 0.257 \\ 0 \end{bmatrix}$$

$$M_{\text{avg},\text{exit}} := X_{fg,\text{exit}} \cdot M_{fg} = 0.027 \frac{\text{kg}}{\text{mol}}$$

$$\sum X_{fg,\text{exit}} = 1$$

$$Y_{fg,\text{exit}} := \frac{\overrightarrow{X_{fg,\text{exit}} \cdot M_{fg}}}{M_{\text{avg},\text{exit}}} = \begin{bmatrix} 0.1927 \\ 0.6022 \\ 0.0364 \\ 0.1686 \\ 5.4784 \cdot 10^{-17} \end{bmatrix} \frac{\text{kg}}{\text{kg}}$$

$$\sum Y_{fg,\text{exit}} = 1$$

Vectors for fuel constituent data:

$$Fuel := \begin{bmatrix} \text{"C(participating)"} \\ \text{"H"} \\ \text{"O"} \\ \text{"N"} \\ \text{"S"} \end{bmatrix}$$

$$Stoch_{coef} := \begin{bmatrix} 1 \\ 0.25 \\ -0.5 \\ 0 \\ 1 \end{bmatrix}$$

$$Y_{fuel} := \begin{bmatrix} Y_C \\ Y_H \\ Y_O \\ Y_N \\ Y_S \end{bmatrix}$$

$$Y_{fuel} = \begin{bmatrix} 0.24982 \\ 0.034251162790698 \\ 0.24192 \\ 0.001302325581395 \\ 1.30232558139535 \cdot 10^{-16} \end{bmatrix}$$

$$M_{fuel,c} := \begin{bmatrix} M_C \\ M_H \\ M_O \\ M_N \\ M_S \end{bmatrix}$$

$$M_{fuel,c} = \begin{bmatrix} 0.012 \\ 0.001 \\ 0.016 \\ 0.014 \\ 0.032 \end{bmatrix} \frac{kg}{mol}$$

$$N_{fuel,c} := \begin{bmatrix} N_C \\ N_H \\ N_O \\ N_N \\ N_S \end{bmatrix}$$

$$N_{fuel,c} = \begin{bmatrix} 20.799 \\ 33.981 \\ 15.12 \\ 0.093 \\ 0 \end{bmatrix} mol$$

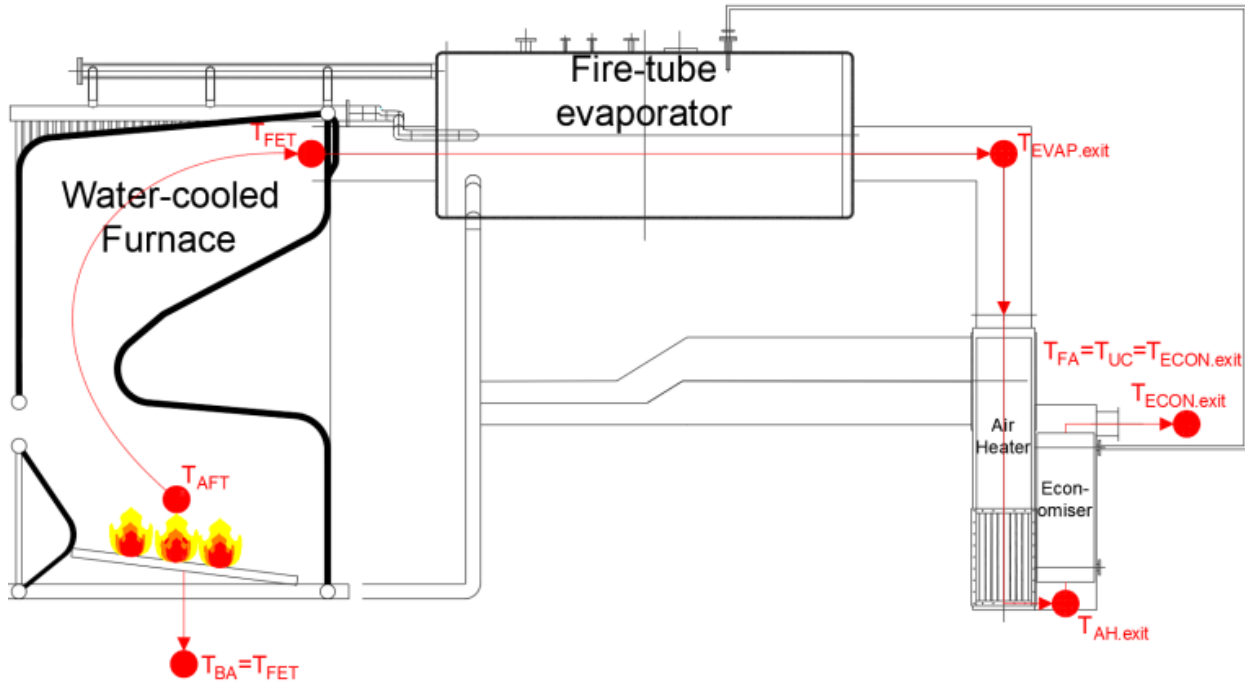
Loss Factors:

Radiation Loss Factor:

$$Rad_{loss} := 0.5\% = 0.005$$

Unaccounted Carbon Loss Factor:

$$C_{unacc.loss} := 0.5\% = 0.005$$

Flue Gas Side Inputs:**Figure 4: Fluegas Flow (Temperatures)**

Notes:

- Not clear if MCR flue gas mass flow rate includes fly ash and unburnt carbon
- No bottom ash mass flowrate data
- No furnace exit temperature (the inlet temperature to the firetube evaporator as per the fire tube evaporator flownex model will be used as the furnace exit temperature (FET)).
- The bottom ash is assumed to exit the bottom of the boiler at the FET.
- The fly ash and all the unburnt carbon is assumed to leave at the flue gas temperature at the exit of the economiser (i.e. exit of the whole boiler).

Flue gas mass flowrate:

Furnace exit temperature (FET): $T_{FET} := 945.535 \text{ }^{\circ}\text{C}$ Furnace exit pressure (FEP): $P_{FEP} := 807.152 \text{ kPa}$ Evaporator exit temperature: $T_{fg, EVAP} := 338 \text{ }^{\circ}\text{C}$ Evaporator exit pressure: $P_{fg, EVAP} := 690 \text{ kPa}$ Air heater exit temperature: $T_{fg, AH} := 230 \text{ }^{\circ}\text{C}$ Air heater exit pressure: $P_{fg, AH} := 808 \text{ kPa}$ Economiser exit temperature: $T_{fg, ECON} := 154 \text{ }^{\circ}\text{C}$ Economiser exit pressure: $P_{fg, ECON} := 907 \text{ kPa}$ Bottom ash temperature: $T_{BAsh} := T_{FET} = 945.535 \text{ }^{\circ}\text{C}$ Fly ash temperature: $T_{FAsh} := T_{FET} = 945.535 \text{ }^{\circ}\text{C}$ Unburnt carbon temperature: $T_{UC} := T_{FET} = 945.535 \text{ }^{\circ}\text{C}$ $T_{FAsh} := T_{fg, ECON} = 154 \text{ }^{\circ}\text{C}$ $T_{UC} := T_{fg, ECON} = 154 \text{ }^{\circ}\text{C}$

[1]

$$m'_{fg, MCR} := 6984 \frac{\text{kg}}{\text{hr}}$$

[Inlet boundary condition to partially verified firetube evaporator flownex model]

[Inlet boundary condition to partially verified firetube evaporator flownex model]

[1]

$$T_{fg, EVAP, flownex} := 349.126 \text{ }^{\circ}\text{C}$$

[1]

$$P_{fg, EVAP, flownex} := 807.086 \text{ kPa}$$

[1]

$$T_{fg, AH, flownex} := 234.553 \text{ }^{\circ}\text{C}$$

[1]

$$P_{fg, AH, flownex} := 791.320 \text{ kPa}$$

[1]

$$T_{fg, ECON, flownex} := 156.164 \text{ }^{\circ}\text{C}$$

[1]

$$P_{fg, ECON, flownex} := 909.323 \text{ kPa}$$

[Assumed equal to FET]

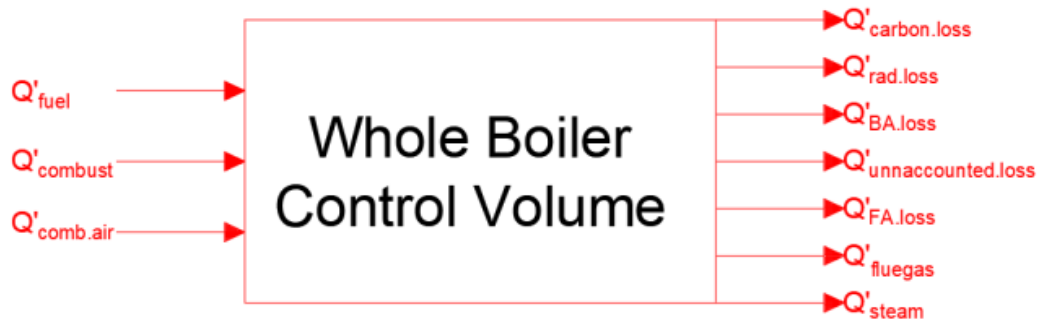
[Assumed equal to FET]

[Assumed equal to FET]

[Assumed equal to econ exit (i.e. total boiler exit)]

[Assumed equal to econ exit (i.e. total boiler exit)]

Furnace exit enthalpy:	$h_{fg.FURN} := h_{fluegas}(T_{FET}, Y_{fg.exit}) = 1611.818 \frac{\text{kJ}}{\text{kg}}$
Evaporator exit enthalpy:	$h_{fg.EVAP} := h_{fluegas}(T_{fg.EVAP}, Y_{fg.exit}) = 798.733 \frac{\text{kJ}}{\text{kg}}$
Air heater exit enthalpy:	$h_{fg.AH} := h_{fluegas}(T_{fg.AH}, Y_{fg.exit}) = 666.63 \frac{\text{kJ}}{\text{kg}}$
Economiser exit enthalpy:	$h_{fg.ECON} := h_{fluegas}(T_{fg.ECON}, Y_{fg.exit}) = 575.991 \frac{\text{kJ}}{\text{kg}}$
Bottom ash enthalpy:	$h_{BAsh} := h_{FA}(T_{BAsh}) = 671.991 \frac{\text{kJ}}{\text{kg}}$
Fly ash enthalpy:	$h_{FAsh} := h_{FA}(T_{FAsh}) = 94.17 \frac{\text{kJ}}{\text{kg}}$



Whole Boiler Energy Balance to Determine the Required Fuel Flowrate

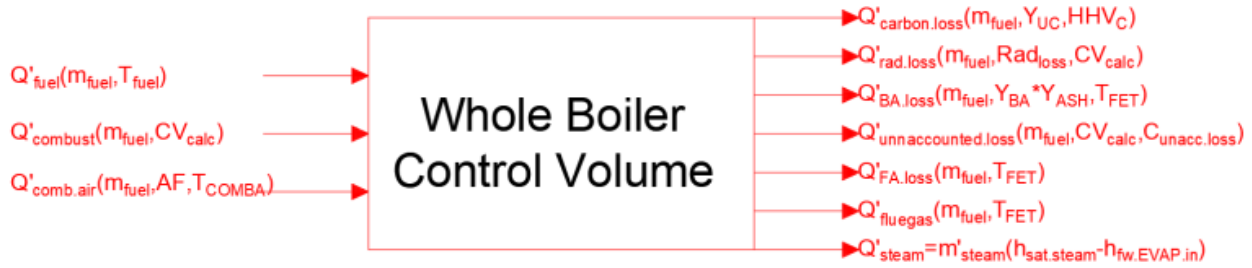


Figure 5: Boiler Energy Balance

Energy inlets:

$$Q'_{fuel}(m'_{fuel}) := m'_{fuel} \cdot h_{fuel}$$

$$Q'_{comb}(m'_{fuel}) := m'_{fuel} \cdot CV_{calc}$$

$$Q'_{comb.air}(m'_{fuel}) := m'_{fuel} \cdot AF_{actual.exit} \cdot (1 + \omega_{air}) \cdot h_{COMBA}$$

Energy outlets:

$$Q'_{carbon.loss}(m'_{fuel}) := m'_{fuel} \cdot Y_{UC} \cdot HHV_C$$

$$Q'_{rad.loss}(m'_{fuel}) := m'_{fuel} \cdot Rad_{loss} \cdot CV_{calc}$$

$$Q'_{bot.ash.loss}(m'_{fuel}) := m'_{fuel} \cdot (Y_{BA} \cdot Y_{ASH}) \cdot h_{BAsh}$$

$$Q'_{unacc.loss}(m'_{fuel}) := m'_{fuel} \cdot CV_{calc} \cdot C_{unacc.loss}$$

$$Q'_{sensible.flyash.exit}(m'_{fuel}) := m'_{fuel} \cdot (Y_{FA} \cdot Y_{ASH}) \cdot h_{FAsh}$$

$$Q'_{fg.out}(m'_{fuel}) := m'_{fuel} \cdot (1 - Y_{ASH} - Y_{UC} + AF_{actual.exit} \cdot (1 + \omega_{air})) \cdot h_{fg.FURN}$$

$$Q'_{steam} = 2562.002 \text{ kW}$$

$$Q'_{steam} = 2562.002 \text{ kW}$$

$$Q_{steam,ww} = 6.843 \text{ kW}$$

Balance energy equation to calculate flowrate (Boiler Energy Balance)

Guess Values	$m'_{fuel.guess} := 1 \frac{kg}{s}$
Constraints	$Q'_{fuel}(m'_{fuel.guess}) \downarrow + Q'_{comb}(m'_{fuel.guess}) \downarrow + Q'_{comb.air}(m'_{fuel.guess}) \downarrow = Q'_{carbon.loss}(m'_{fuel.guess}) \downarrow + Q'_{rad.loss}(m'_{fuel.guess}) \downarrow + Q'_{bot.ash.loss}(m'_{fuel.guess}) \downarrow + Q'_{unacc.loss}(m'_{fuel.guess}) \downarrow + Q'_{sensible.flyash.exit}(m'_{fuel.guess}) \downarrow + Q'_{fg.out}(m'_{fuel.guess}) + Q'_{steam}$
Solver	$m'_{fuel} := \text{find}(m'_{fuel.guess}) = 0.443 \frac{kg}{s}$

Fuel Mass Flowrate:

$$m'_{fuel} = 0.442963761107891 \frac{kg}{s}$$

$$m'_{fuel} = 1594.67 \frac{kg}{hr}$$

$$m'_{fuel.MCR} = 1564 \frac{kg}{hr}$$

$$Error_{fuel} := \frac{m'_{fuel.MCR} - m'_{fuel}}{m'_{fuel.MCR}} = -1.961\%$$

Humid Air Mass Flowrate:

$$m'_{air} := m'_{fuel} \cdot AF_{actual.exit} \cdot (1 + \omega_{air}) = 1.67599940248141 \frac{kg}{s}$$

$$m'_{air} = 6033.598 \frac{kg}{hr}$$

$$m'_{air.MCR} = 5929 \frac{kg}{hr}$$

$$Error_{air} := \frac{m'_{air.MCR} - m'_{air}}{m'_{fuel.MCR}} = -6.688\%$$

$$m'_{CA} = 539 \frac{kg}{hr}$$

$$m'_{HA} = 5390 \frac{kg}{hr}$$

Theoretical Air Ratio:

$$TAR := \frac{M_{air}}{Y_{O_2}} \cdot \frac{1}{kg} \cdot \sum Stoch_{coef} \cdot N_{fuel,c} = 2.99145082943064$$

Humid Air Ratio:

$$HAR := \frac{m'_{HA}}{m'_{fuel}} = 3.38001063219605$$

Dry Air Ratio:

$$DAR := \frac{HAR}{1 + \omega_{air}} = 3.33104428126151$$

Excess Air Ratio:

$$air_{excess} := \frac{DAR}{TAR} - 1 = 0.113521321657658$$

Fluegas Mass Flowrate:

$$m'_{fg} := m'_{air} + m'_{fuel} \cdot (1 - Y_{BA} \cdot Y_{ASH}) = 2.10953458722077 \frac{kg}{s}$$

$$m'_{fg} = 7594.325 \frac{kg}{hr}$$

$$m'_{fg.MCR} = 6984 \frac{kg}{hr}$$

$$Error_{fg} := \frac{m'_{fg.MCR} - m'_{fg}}{m'_{fg.MCR}} = -8.739\%$$

Fluegas Ratio:

$$FGR := \frac{m'_{fg}}{m'_{fuel}} = 4.76231866449897$$

Bottom Ash Flowrate:

$$m'_{BA} := m'_{fuel} \cdot Y_{BA} \cdot Y_{ASH} = 0.009428576368527 \frac{kg}{s}$$

Fly Ash Flowrate:

$$m'_{FA} := m'_{fuel} \cdot Y_{FA} \cdot Y_{ASH} = 0.002357144092132 \frac{kg}{s}$$

Energy inlets:

$$Q'_{sens.fuel} := Q'_{fuel} (m'_{fuel}) = 1.223 \text{ kW}$$

$$Q'_{combust} := Q'_{comb} (m'_{fuel}) = 5776.889 \text{ kW}$$

$$Q'_{comba} := Q'_{comb.air} (m'_{fuel}) = 328.783 \text{ kW}$$

Energy outlets:

$$Q'_{c.loss} := Q'_{carbon.loss} (m'_{fuel}) = 88.534 \text{ kW}$$

$$Q'_{steam} = 2562.002 \text{ kW}$$

$$Q'_{radiation.loss} := Q'_{rad.loss} (m'_{fuel}) = 28.884 \text{ kW}$$

$$Q'_{Botash} := Q'_{bot.ash.loss} (m'_{fuel}) = 6.336 \text{ kW}$$

$$Q'_{unaccount.loss} := Q'_{unacc.loss} (m'_{fuel}) = 28.884 \text{ kW}$$

$$Q'_{Flyash} := Q'_{sensible.flyash.exit} (m'_{fuel}) = 0.222 \text{ kW}$$

$$Q'_{fg} := Q'_{fg.out} (m'_{fuel}) = 3392.032 \text{ kW}$$

Boiler energy balance checks:

$$Q'_{in} := Q'_{sens.fuel} + Q'_{combust} + Q'_{comba} = 6106.894 \text{ kW}$$

$$Q'_{out} := Q'_{c.loss} + Q'_{steam} + Q'_{radiation.loss} + Q'_{Botash} + Q'_{unaccount.loss} + Q'_{Flyash} + Q'_{fg} = 6106.894 \text{ kW}$$

$$Q'_{in} - Q'_{out} = 0 \text{ W}$$

Calculate adiabatic flame temperature (AFT Energy Balance): $P_{fg} := P_{amb} = 101.325 \text{ kPa}$

$$\begin{aligned}
 m'_{fuel} &= 0.443 \frac{\text{kg}}{\text{s}} & m'_{fg} &= 2.11 \frac{\text{kg}}{\text{s}} & m'_{FA} &= 0.002 \frac{\text{kg}}{\text{s}} & m'_{BA} &= 0.009 \frac{\text{kg}}{\text{s}} & m'_{air} &= 1.676 \frac{\text{kg}}{\text{s}} \\
 h_{fuel} &= 2.76 \frac{\text{kJ}}{\text{kg}} & & & h_{BAsh} &= 671.991 \frac{\text{kJ}}{\text{kg}} & h_{COMBA} &= 196.171 \frac{\text{kJ}}{\text{kg}} \\
 Y_{UC} &= 0.006 & & & T_{BAsh} &= 945.535 \text{ }^\circ\text{C} \\
 HHV_C &= 32765 \frac{\text{kJ}}{\text{kg}} \\
 CV_{calc} &= 13041.448 \frac{\text{kJ}}{\text{kg}} & c_{UC} &= 710 \frac{\text{m}^2}{\text{s}^2 \cdot \text{K}}
 \end{aligned}$$

$$T_{fg} := 1000 \text{ K}$$

$$\begin{aligned}
 & (m'_{fg} - m'_{FA}) \cdot h_{fluegas}(T_{fg}, Y_{fg}) + m'_{FA} \cdot h_{FA}(T_{fg}) + m'_{BA} \cdot h_{BAsh} + m'_{fuel} \cdot Y_{UC} \cdot HHV_C \\
 & - m'_{fuel} \cdot h_{fuel} - m'_{air} \cdot h_{COMBA} - m'_{fuel} \cdot CV_{calc} \downarrow = 0
 \end{aligned}$$

$$T_{flame} := \text{find}(T_{fg}) = 2069.035 \text{ K}$$

$$T_{flame} = 1795.885 \text{ }^\circ\text{C} \quad m'_{fuel} \cdot CV_{calc} = 5776.889 \text{ kW} \quad h_{flame} := h_{fluegas}(T_{flame}, Y_{fg}) = 2851.671 \frac{\text{kJ}}{\text{kg}}$$

$$T_{fg} := 1000 \text{ K}$$

$$\begin{aligned}
 & m'_{fg} \cdot h_{fluegas}(T_{fg}, Y_{fg}) + m'_{FA} \cdot h_{FA}(T_{fg}) + m'_{BA} \cdot h_{BAsh} + m'_{fuel} \cdot Y_{UC} \cdot HHV_C \\
 & - m'_{fuel} \cdot h_{fuel} - m'_{air} \cdot h_{COMBA} - m'_{fuel} \cdot CV_{calc} \downarrow = 0
 \end{aligned}$$

$$T_{AFT} := \text{find}(T_{fg}) = 2066.783 \text{ K}$$

$$\begin{aligned}
 T_{AFT} &= 1793.633 \text{ }^\circ\text{C} & m'_{fuel} \cdot CV_{calc} &= 5776.889 \text{ kW} & h_{AFT} &:= h_{fluegas}(T_{AFT}, Y_{fg}) = 2848.487 \frac{\text{kJ}}{\text{kg}} \\
 HeatInput &:= m'_{fuel} \cdot HHV_{fuel} - m'_{fuel} \cdot Y_{UC} \cdot HHV_C = 3476.882 \text{ kW}
 \end{aligned}$$

$$m'_{main,steam} = 1.1111111111111111 \frac{\text{kg}}{\text{s}}$$

$$m'_{steam,MCR} = 1.111 \frac{\text{kg}}{\text{s}}$$

$$m'_{fuel} = 0.443 \frac{\text{kg}}{\text{s}}$$

$$m'_{fuel,MCR} = 0.434 \frac{\text{kg}}{\text{s}}$$

$$\frac{m'_{fuel,MCR} - m'_{fuel}}{m'_{fuel,MCR}} = -1.961\%$$

$$m'_{COMBA} = 1.6469 \frac{\text{kg}}{\text{s}}$$

$$m'_{COMBA,MCR} = 1.647 \frac{\text{kg}}{\text{s}}$$

$$m'_{fg} = 2.1095 \frac{\text{kg}}{\text{s}}$$

$$m'_{fg,MCR} = 1.94 \frac{\text{kg}}{\text{s}}$$

$$\frac{m'_{fg,MCR} - m'_{fg}}{m'_{fg,MCR}} = -8.739\%$$

$$C_{factor} := 1.2 \quad \varepsilon_W := 1.0 \quad \varepsilon_f := 0.6 \quad \alpha_f := \varepsilon_f = 0.6$$

$$T_{W,upstream} := 642.981 \text{ }^{\circ}\text{C} \quad T_{W,downstream} := 237.167 \text{ }^{\circ}\text{C}$$

$$T_W := \frac{T_{W,upstream} + T_{W,downstream}}{2} = 440.074 \text{ }^{\circ}\text{C}$$

$$q_{furn} := C_{factor} \cdot \sigma \cdot \frac{\varepsilon_W \cdot (\varepsilon_f \cdot T_{flame}^4 - \alpha_f \cdot T_W^4)}{(1 - (1 - \varepsilon_W) \cdot (1 - \alpha_f))} = 737.631 \frac{\text{kW}}{\text{m}^2}$$

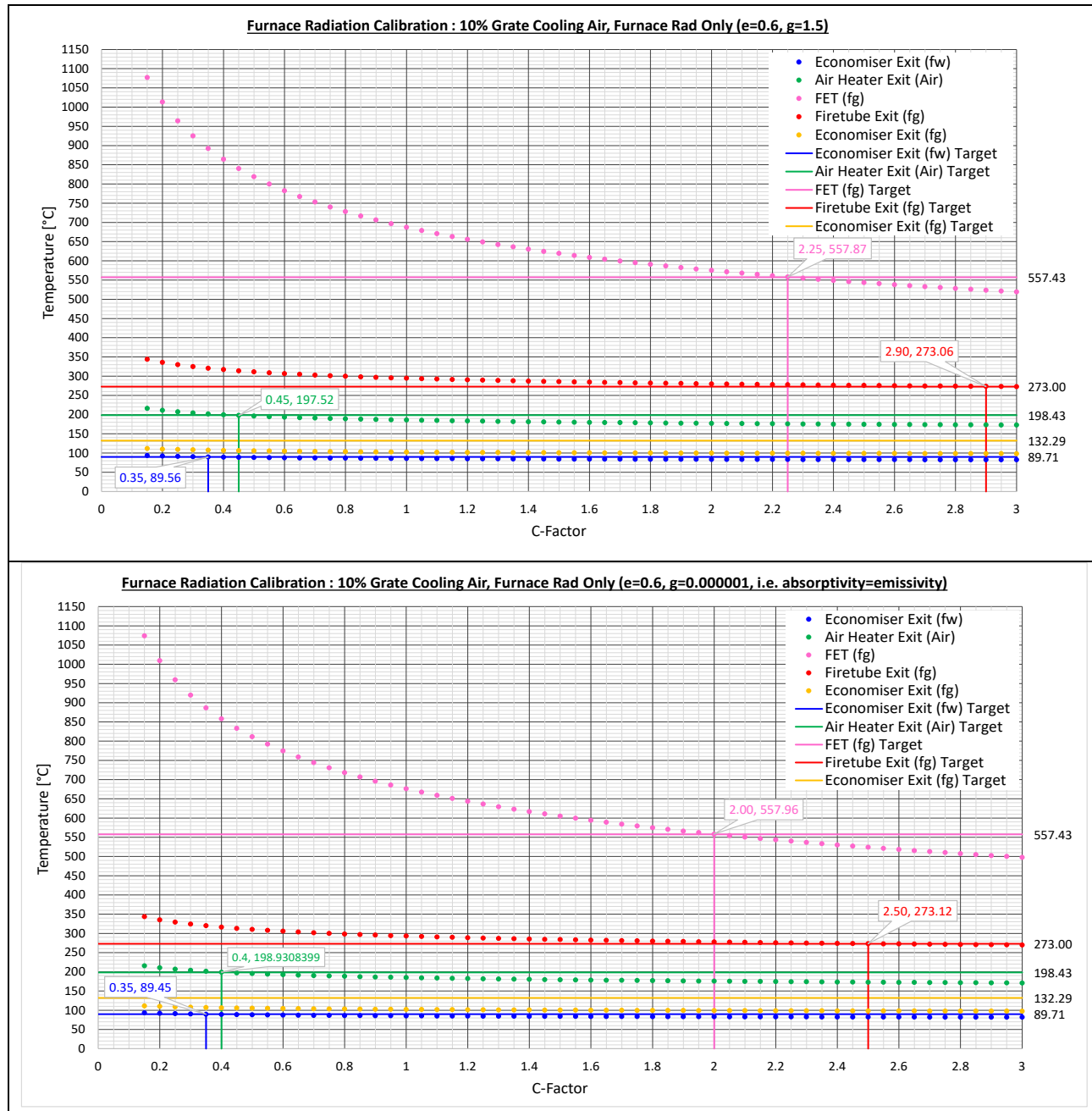
$$A_{furn} := 56.4156 \text{ m}^2$$

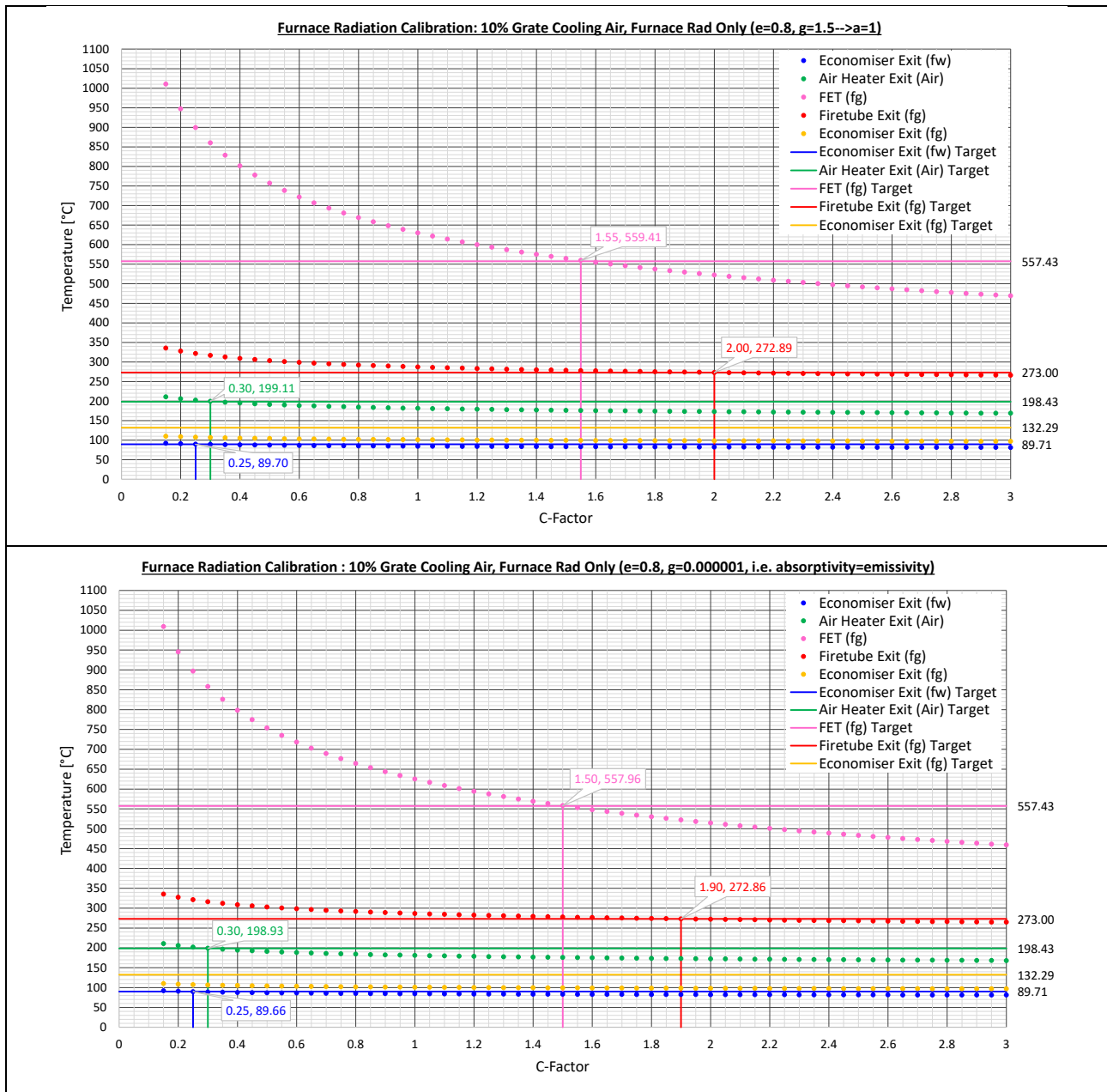
$$Q'_{rad} := q_{furn} \cdot A_{furn} = 41613.91 \text{ kW}$$

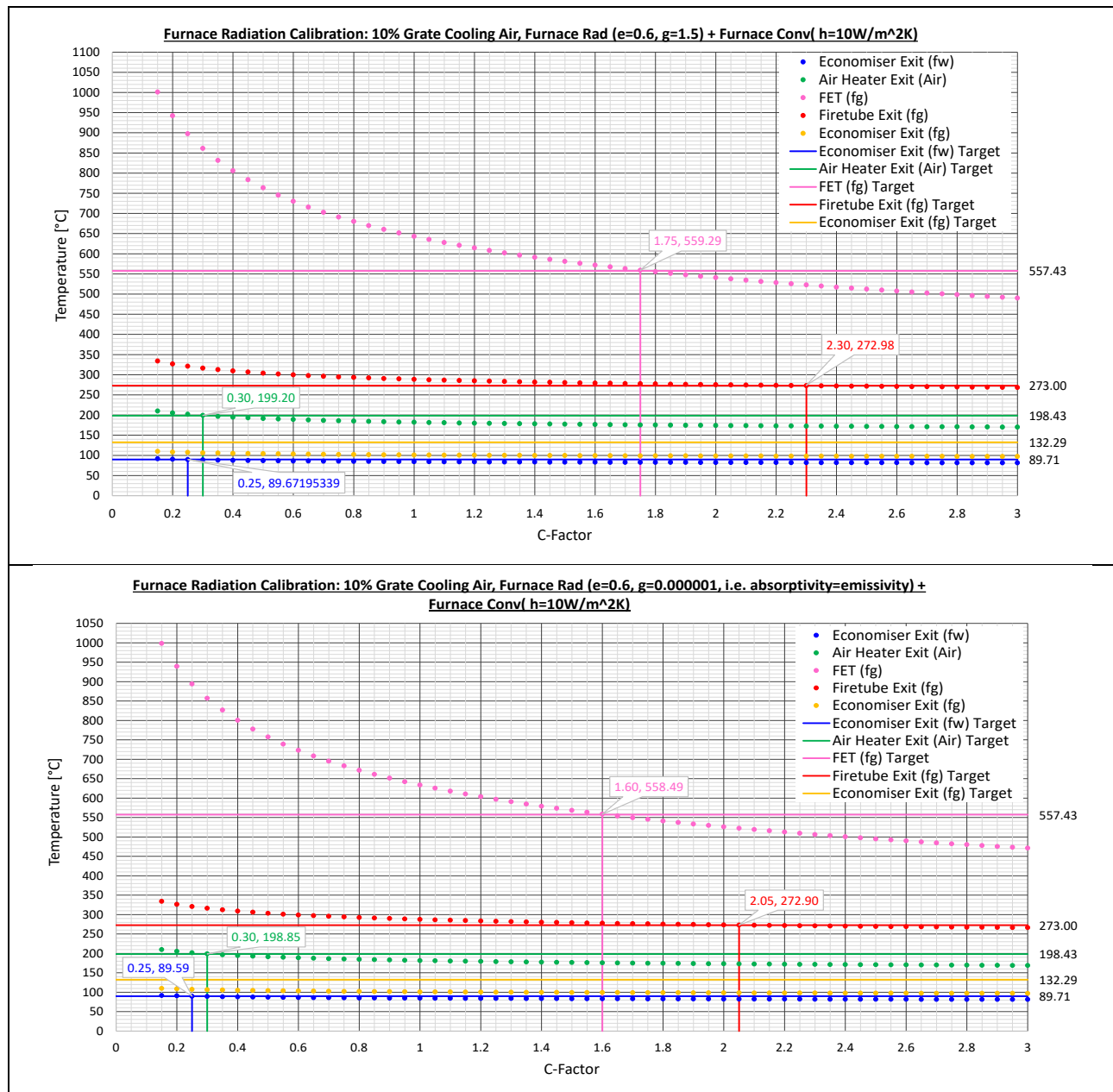
IGNORE THESE.
THESE VALUES ARE NOT USING
MCR FURNACE. It is using site
calibrated integrated model.

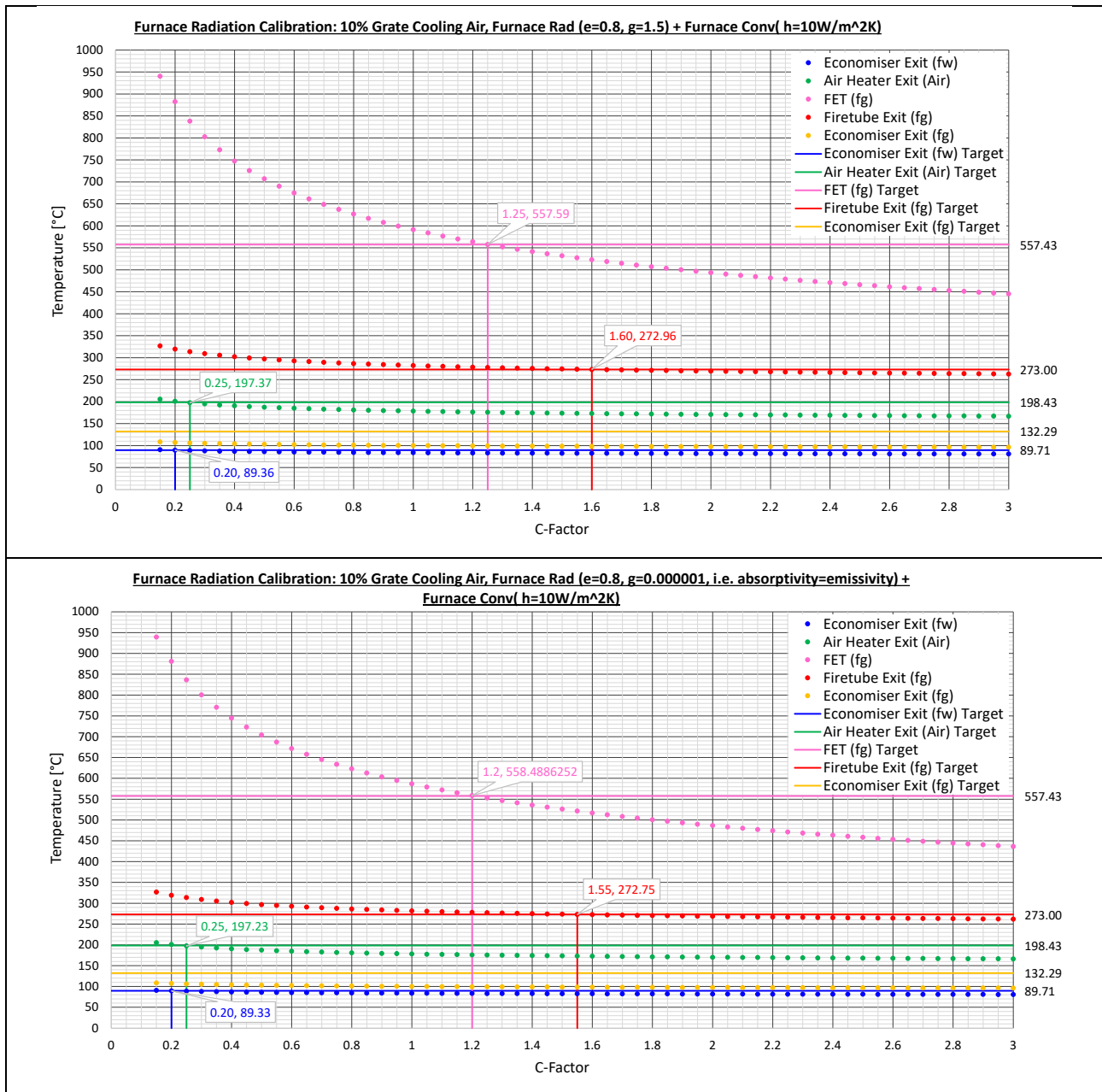
Appendix E. Steady State Calibration and Fine Tuning Details (Parametric Studies)

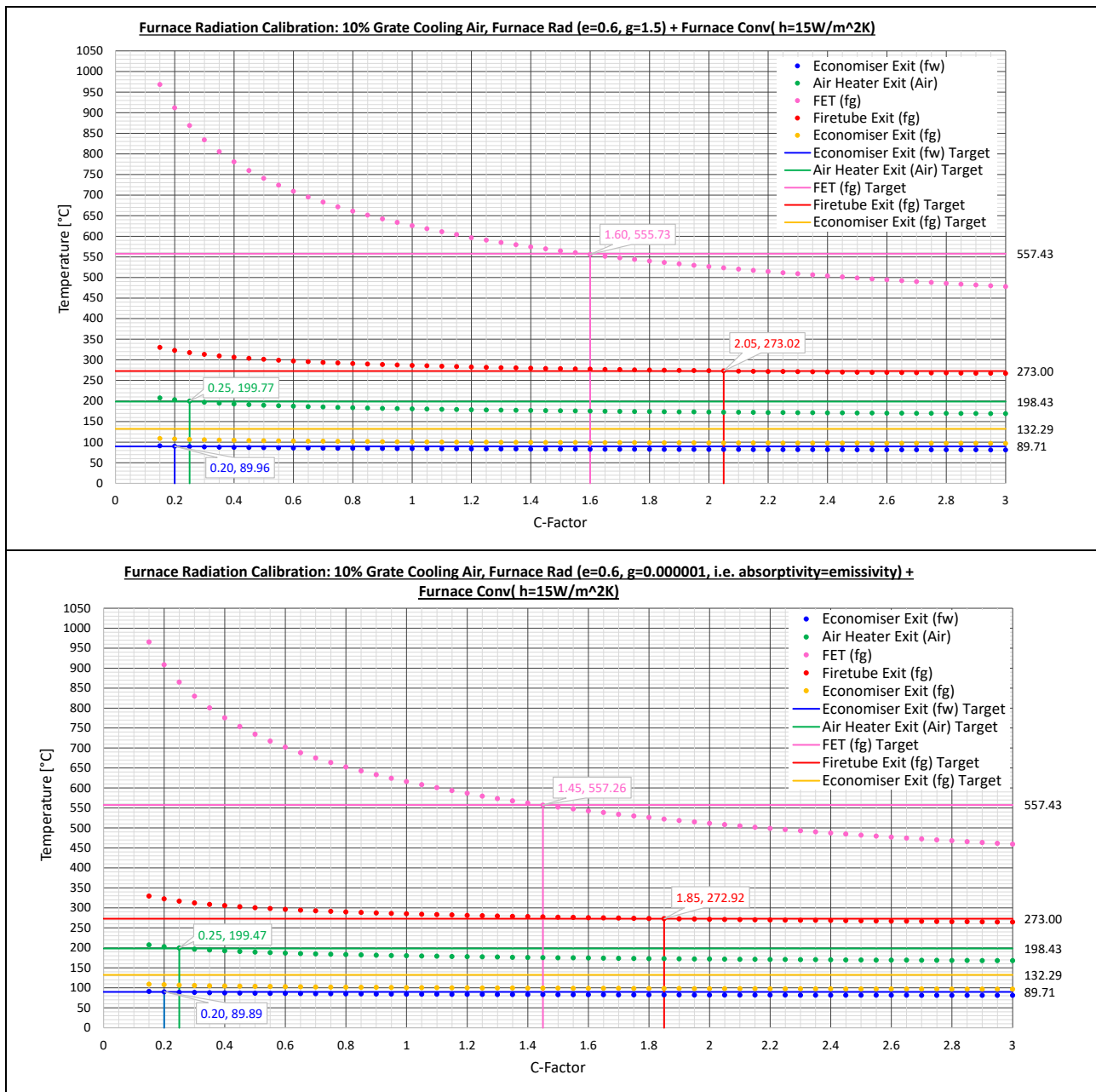
10% Grate Cooling Air – Radiation Only

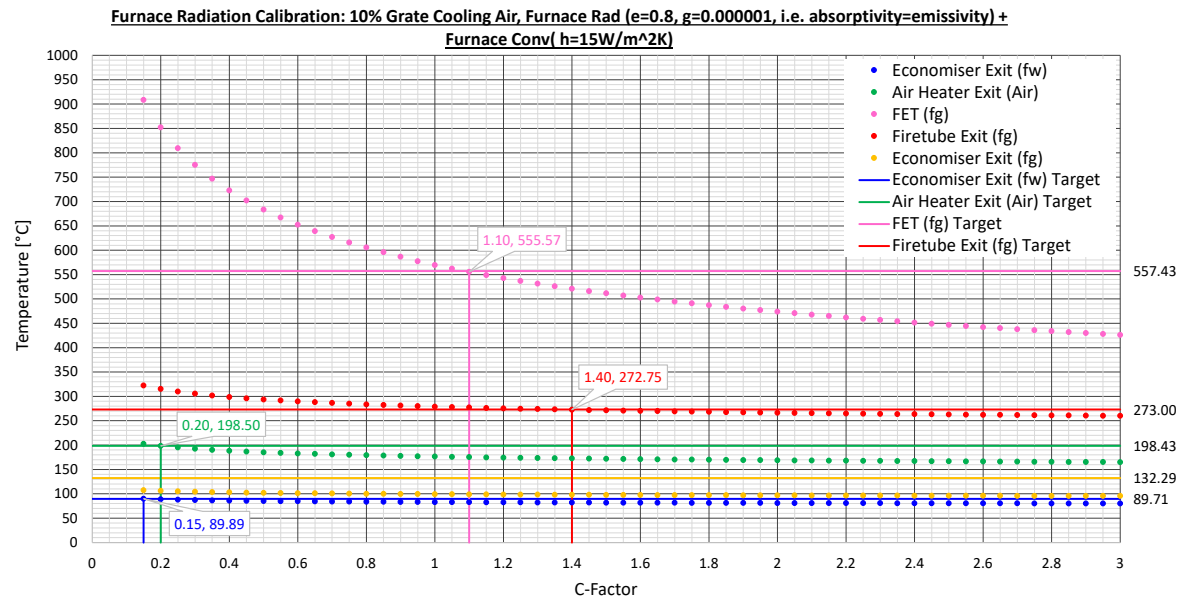
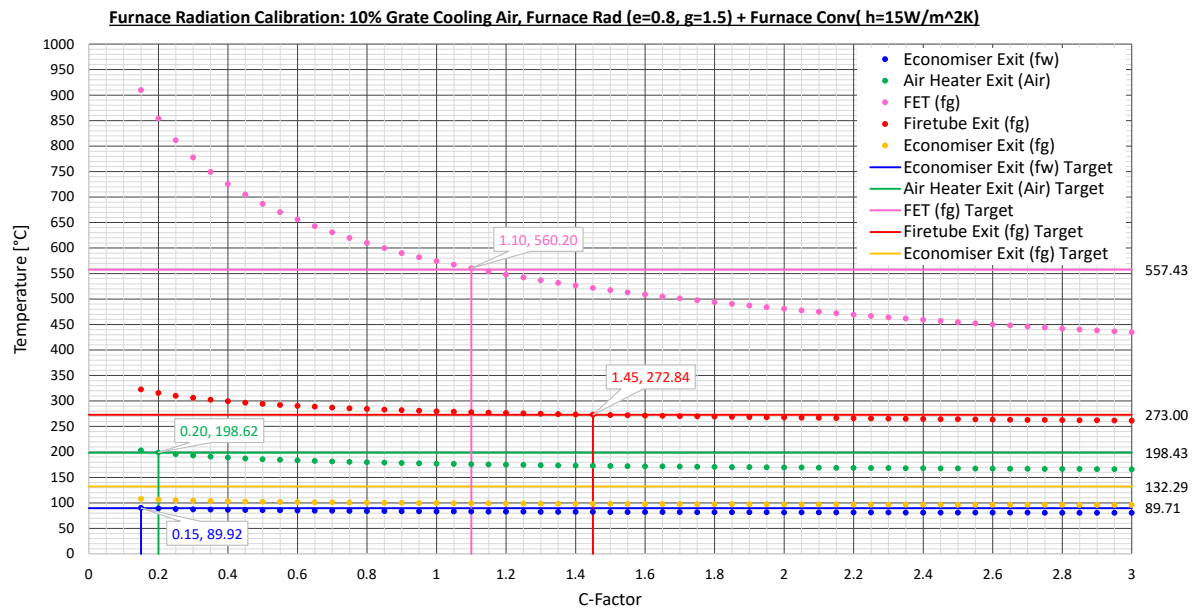


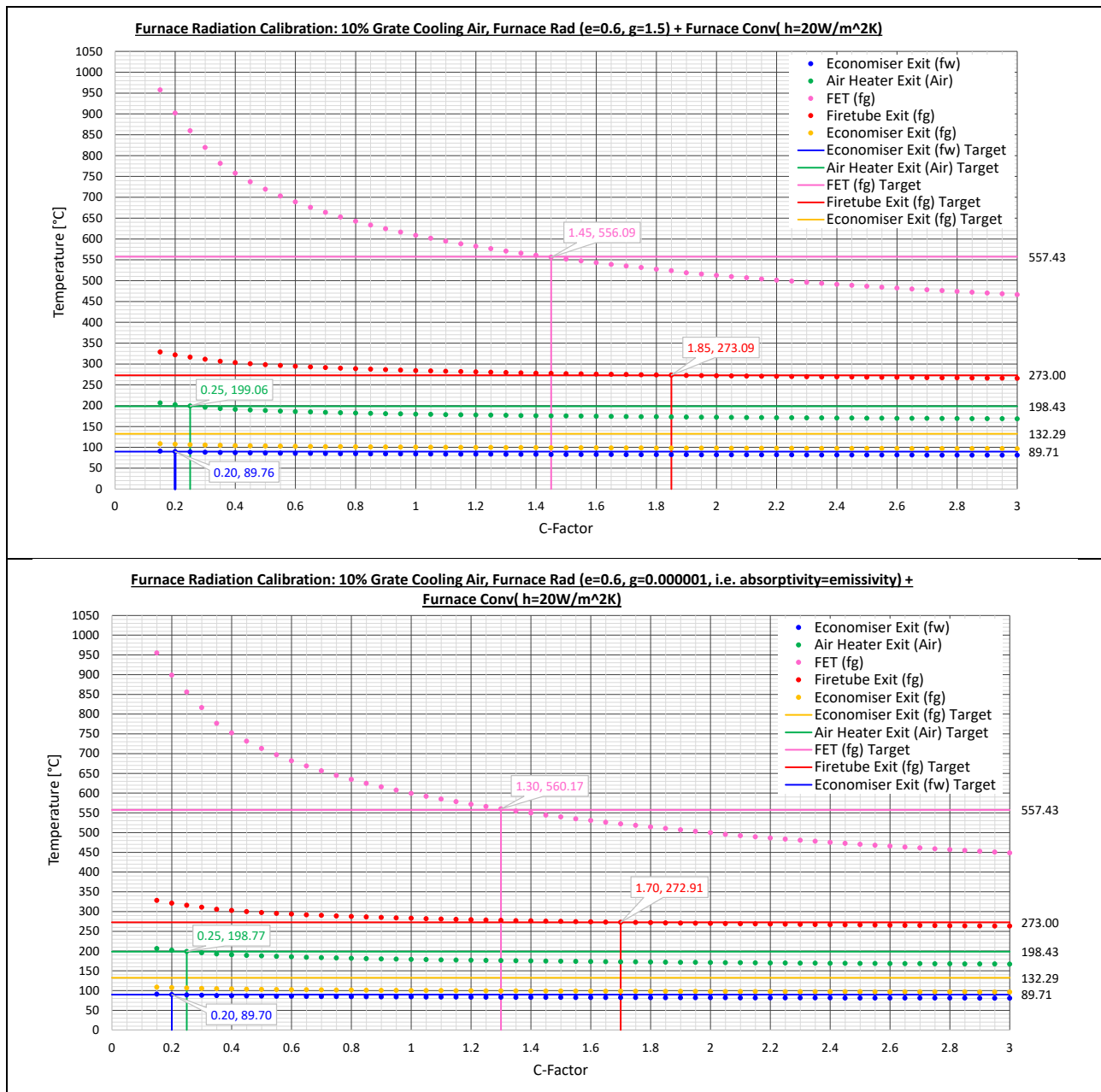


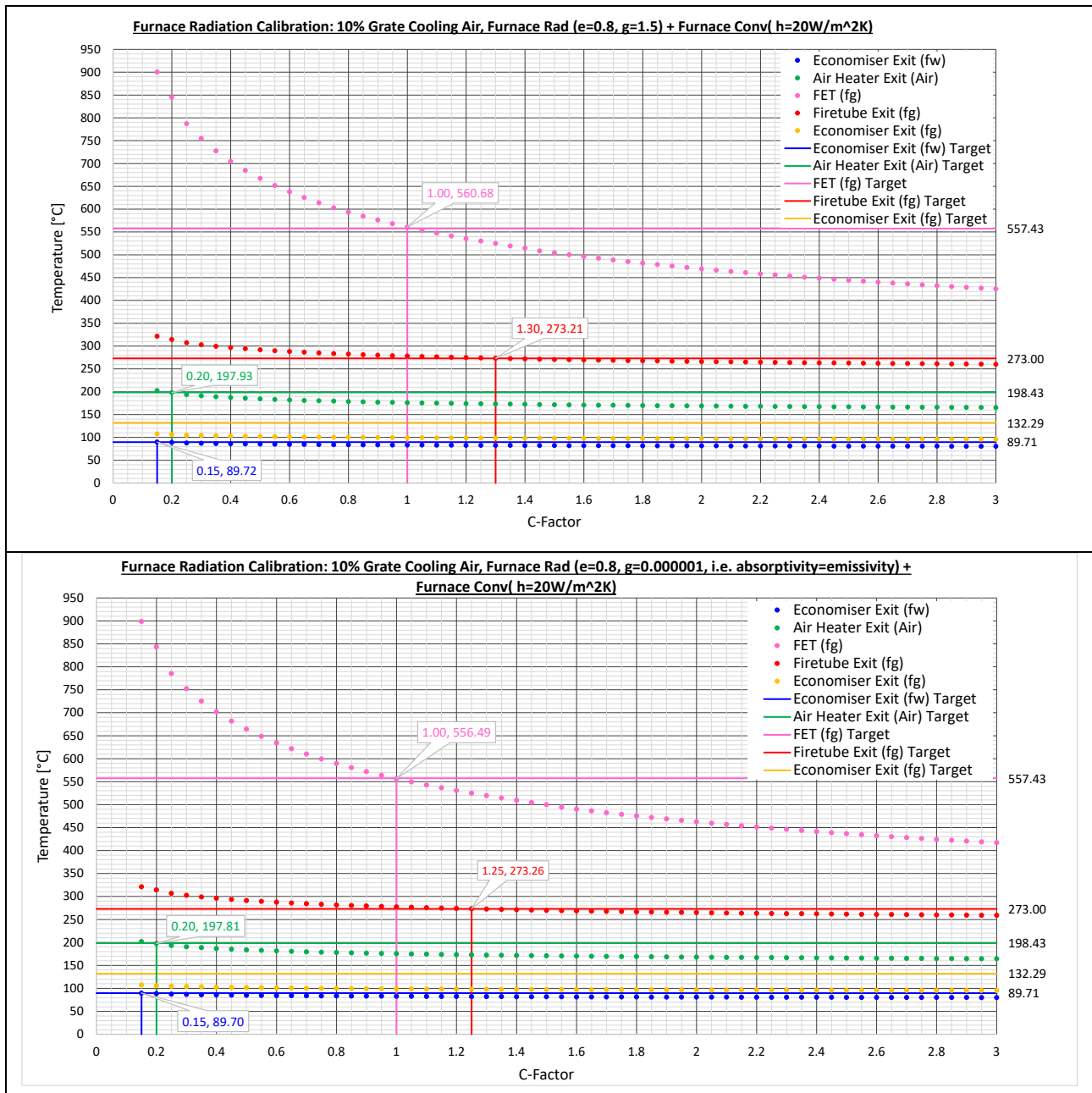
10% Grate Cooling Air – Radiation + Convection @10W/m²K

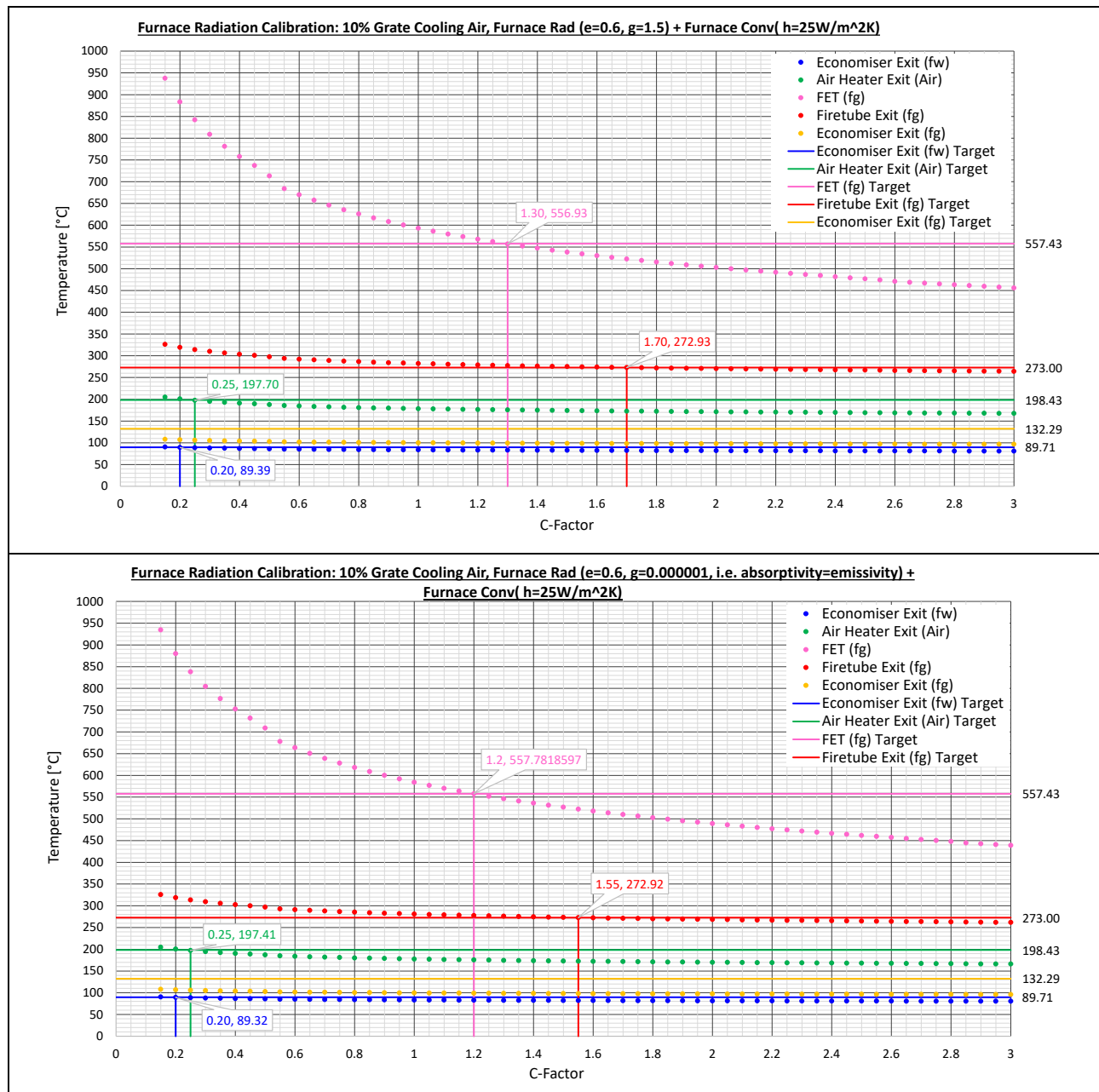


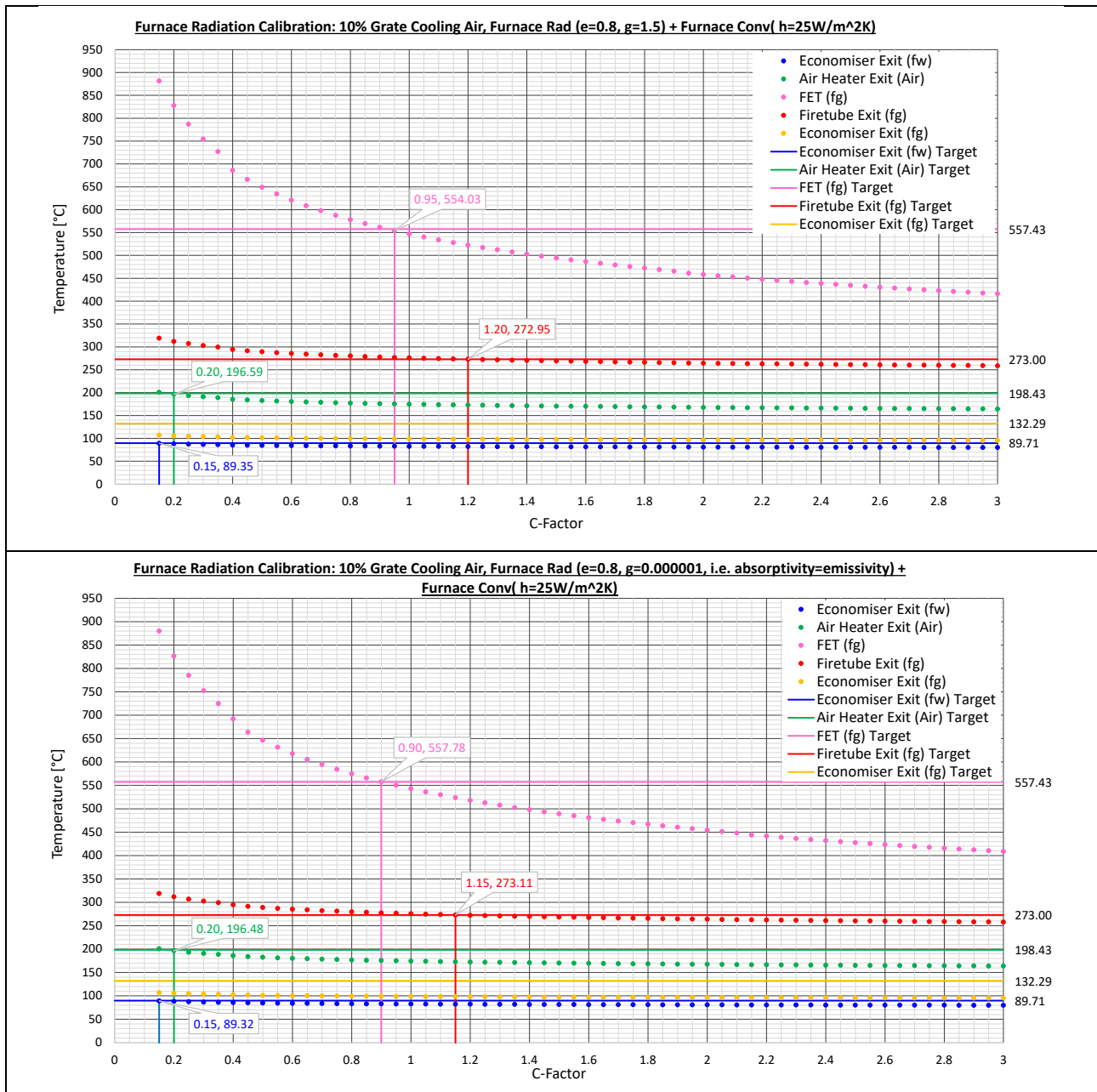
10% Grate Cooling Air – Radiation + Convection @15W/m²K



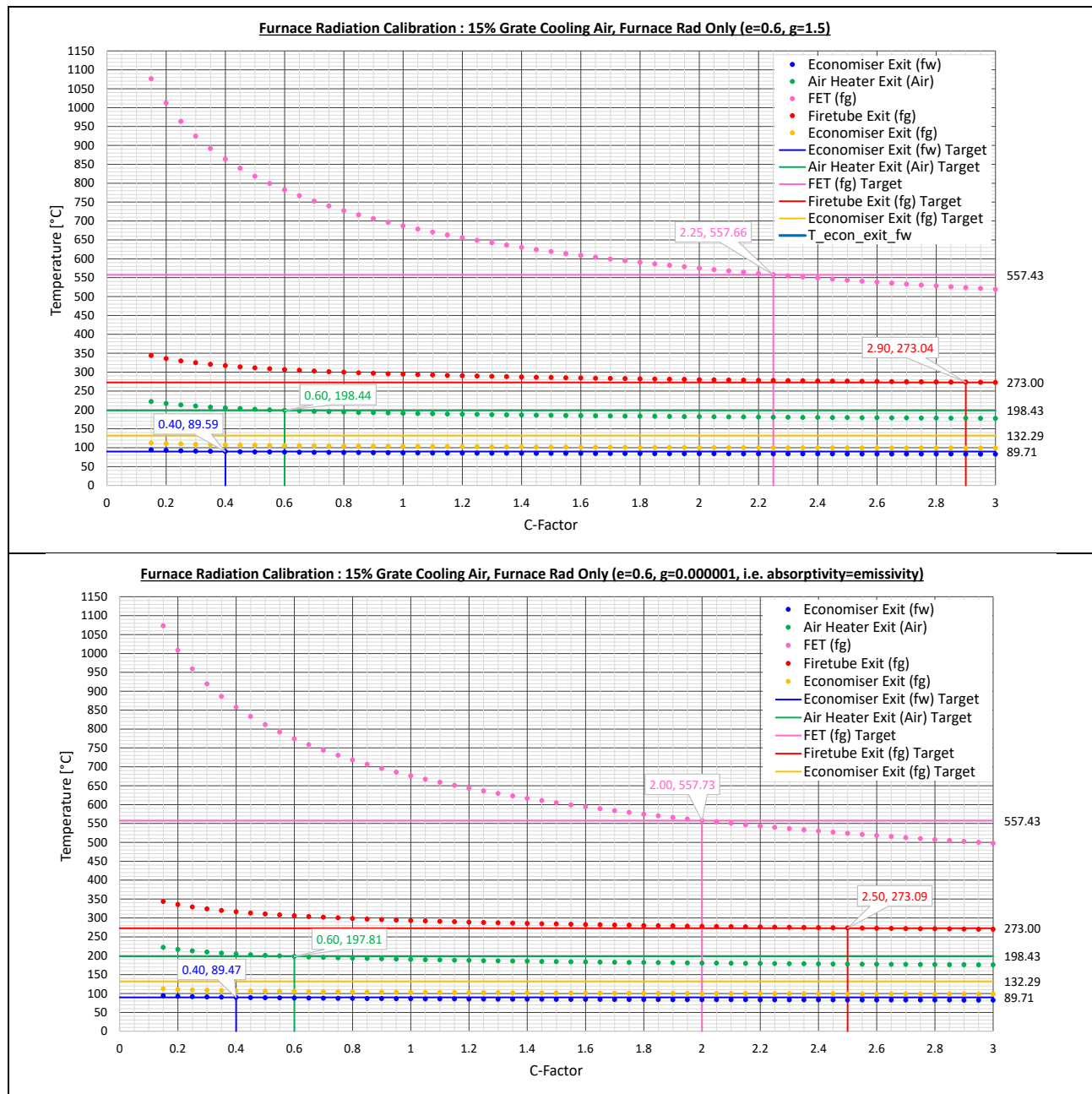
10% Grate Cooling Air – Radiation + Convection @20W/m²K

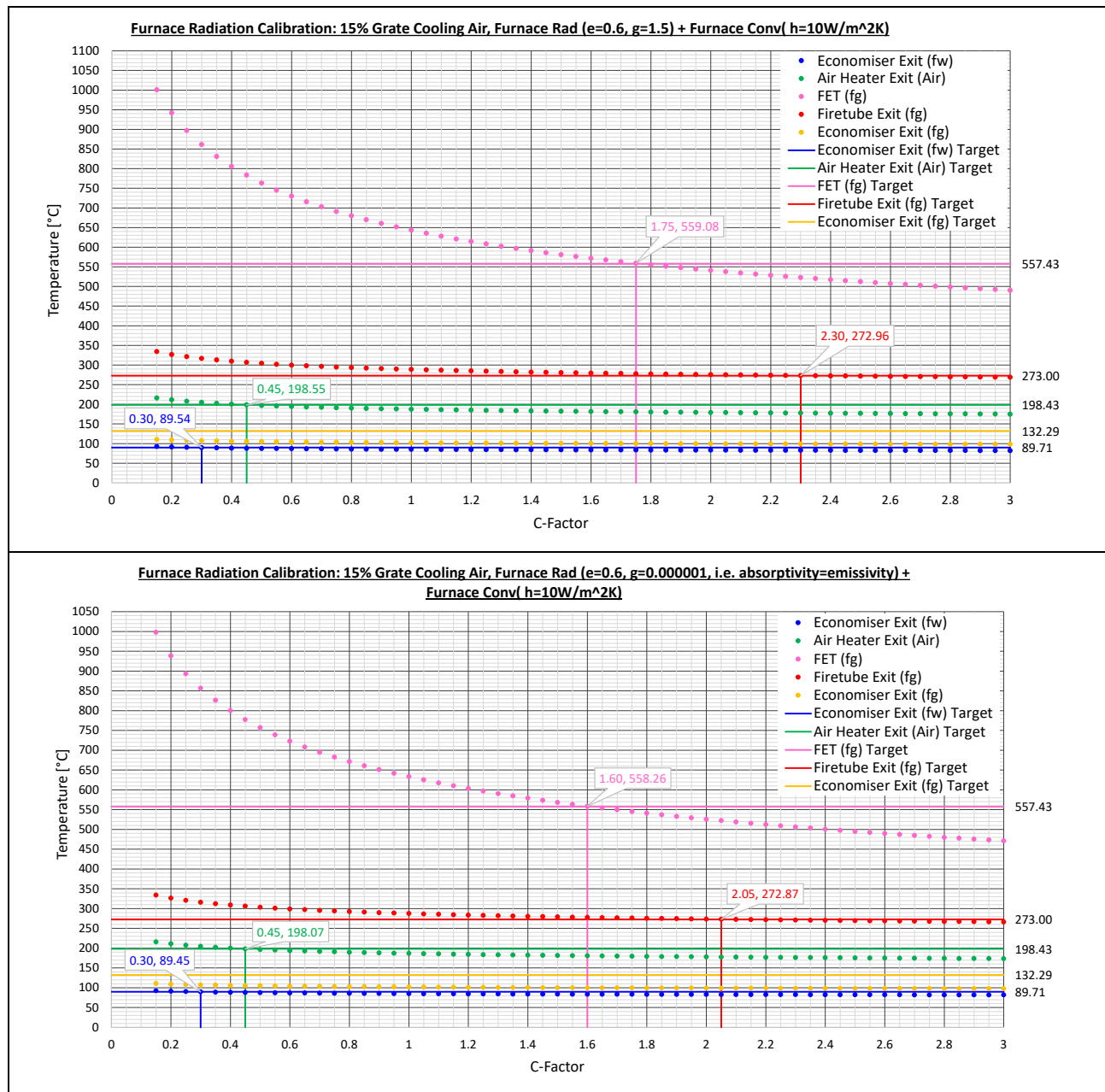


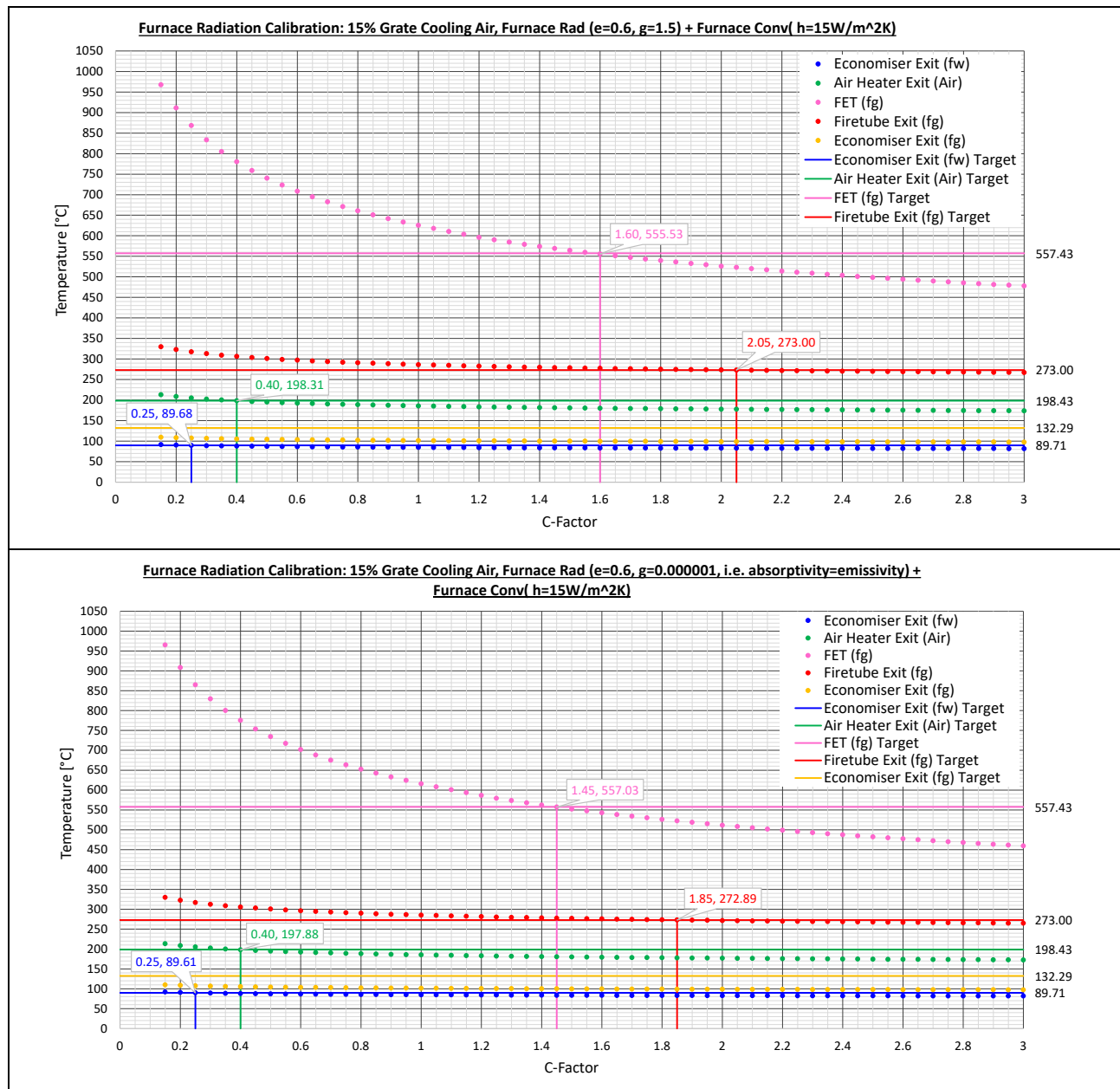
10% Grate Cooling Air – Radiation + Convection @25W/m²K

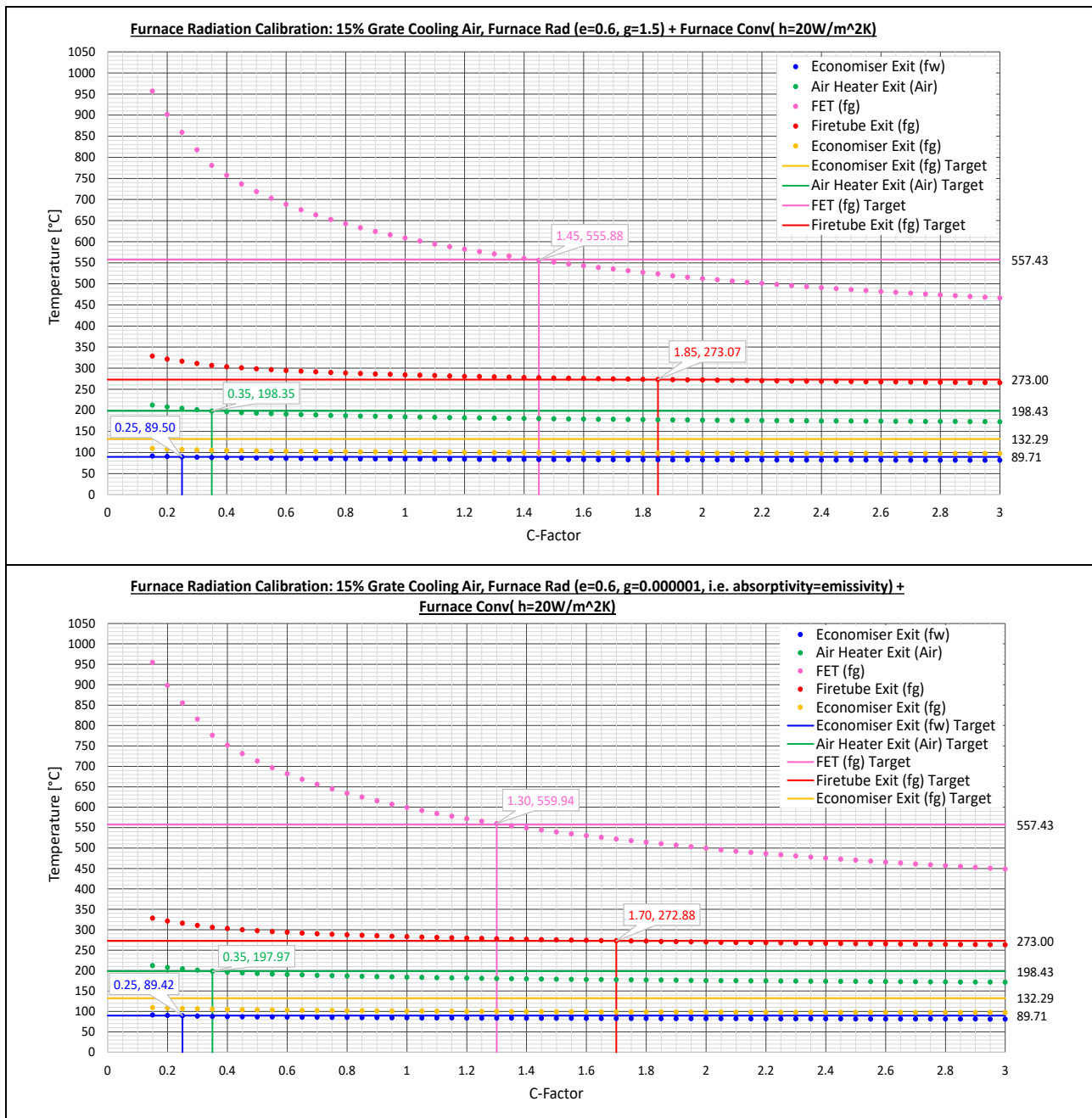


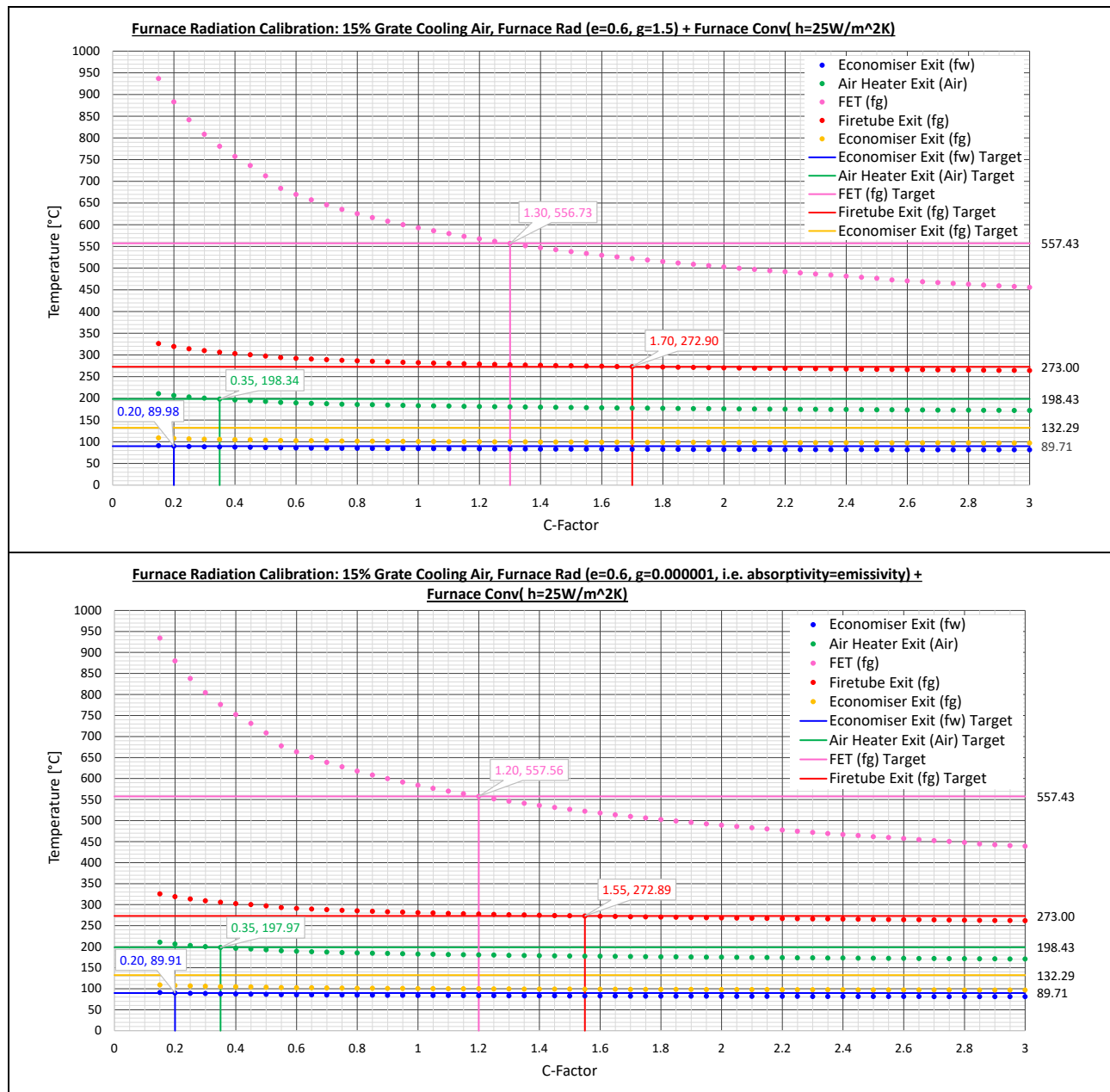
15% Grate Cooling Air – Radiation Only



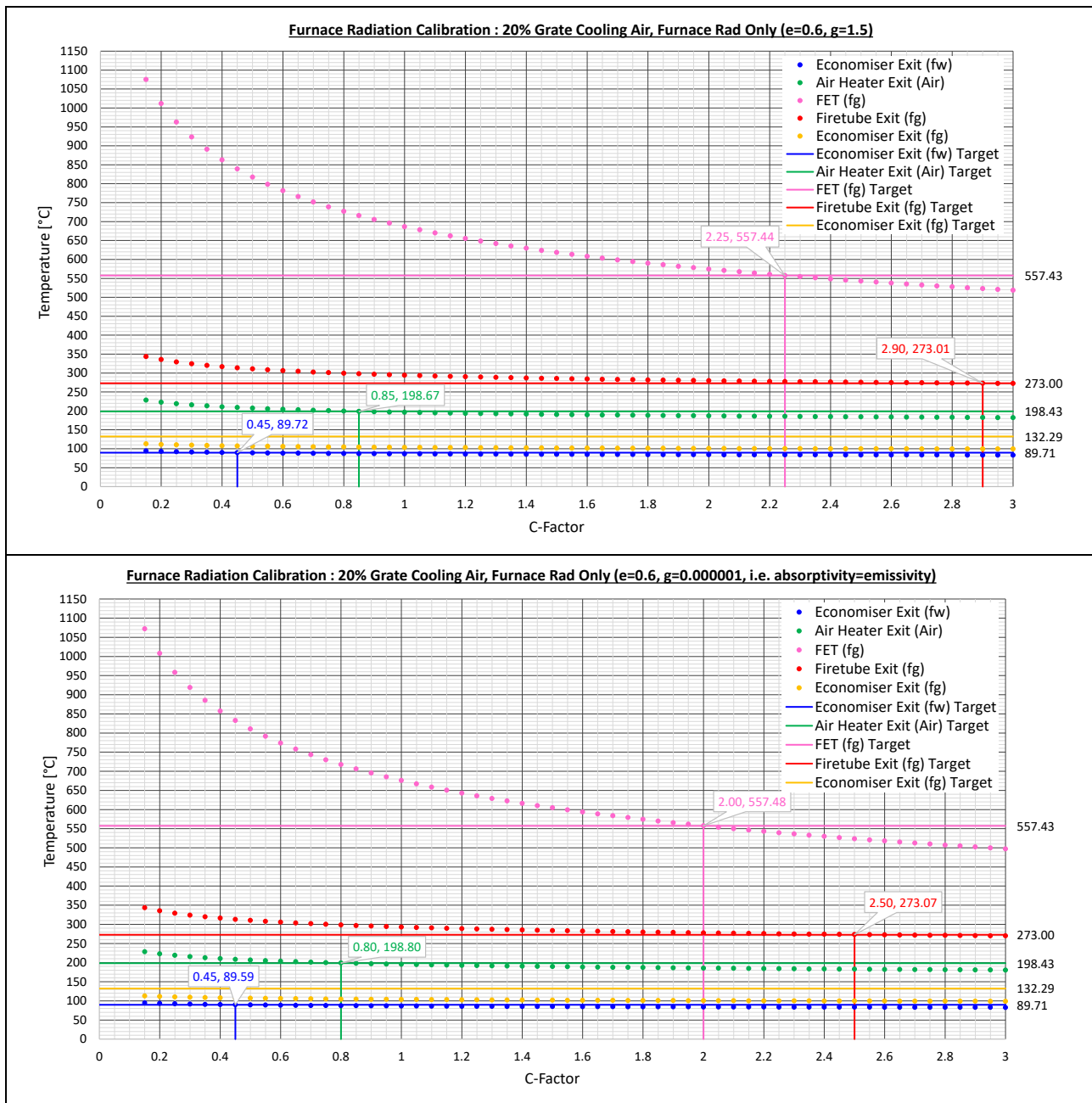
15% Grate Cooling Air – Radiation + Convection @10 W/m²K

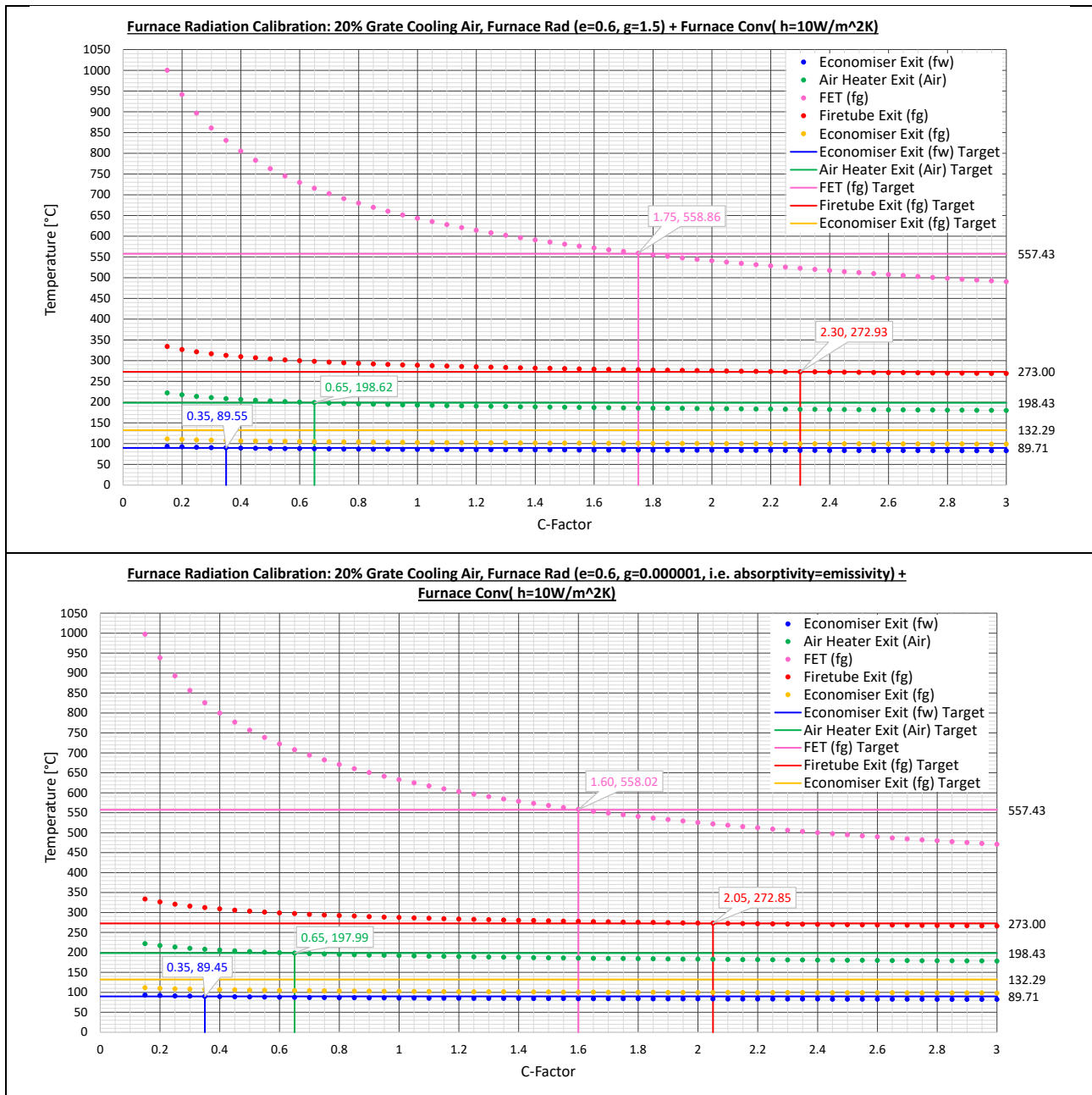
15% Grate Cooling Air – Radiation + Convection @15 W/m²K

15% Grate Cooling Air – Radiation + Convection @20 W/m²K

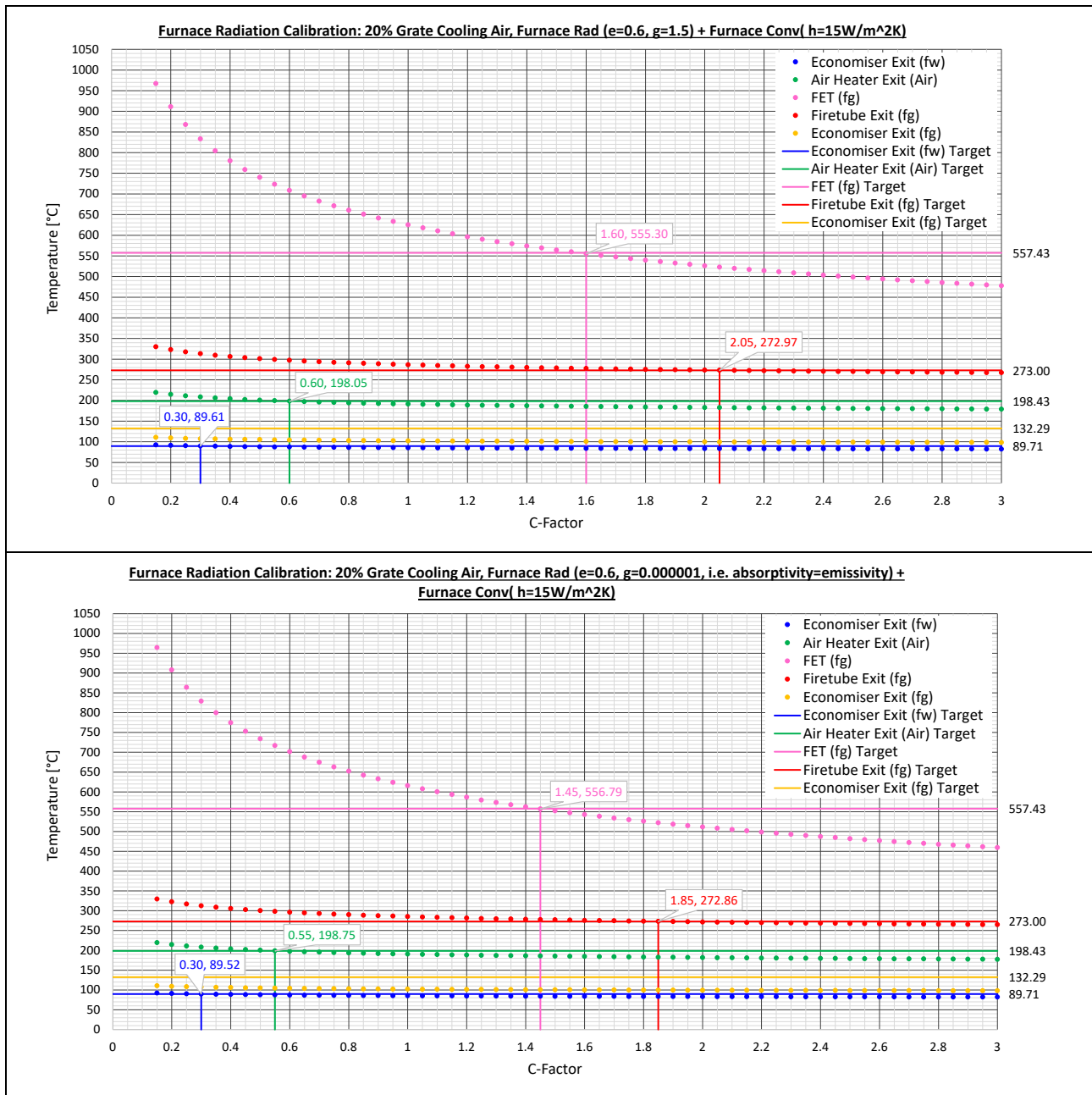
15% Grate Cooling Air – Radiation + Convection @25 W/m²K

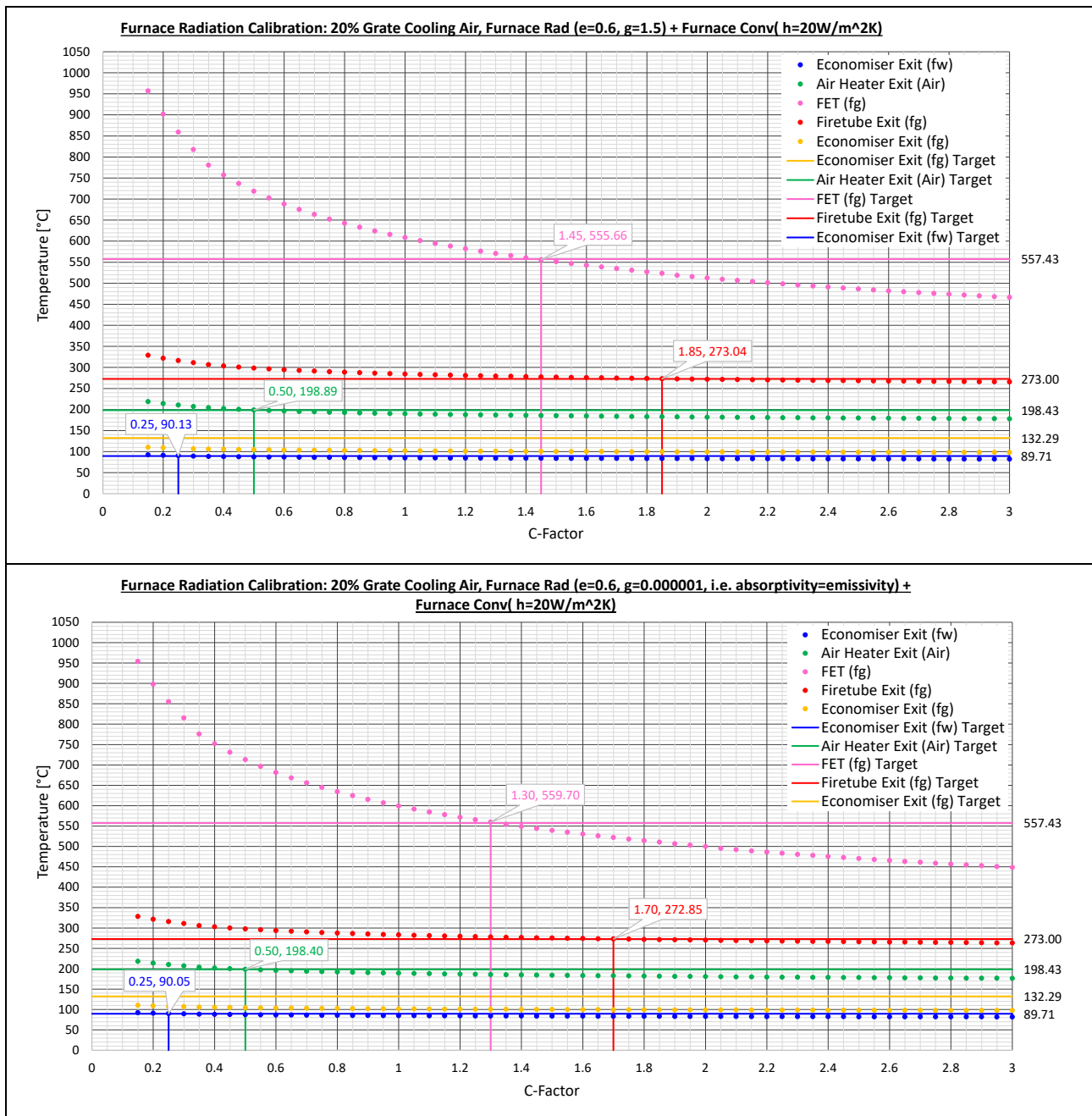
20% Grate Cooling Air – Radiation Only

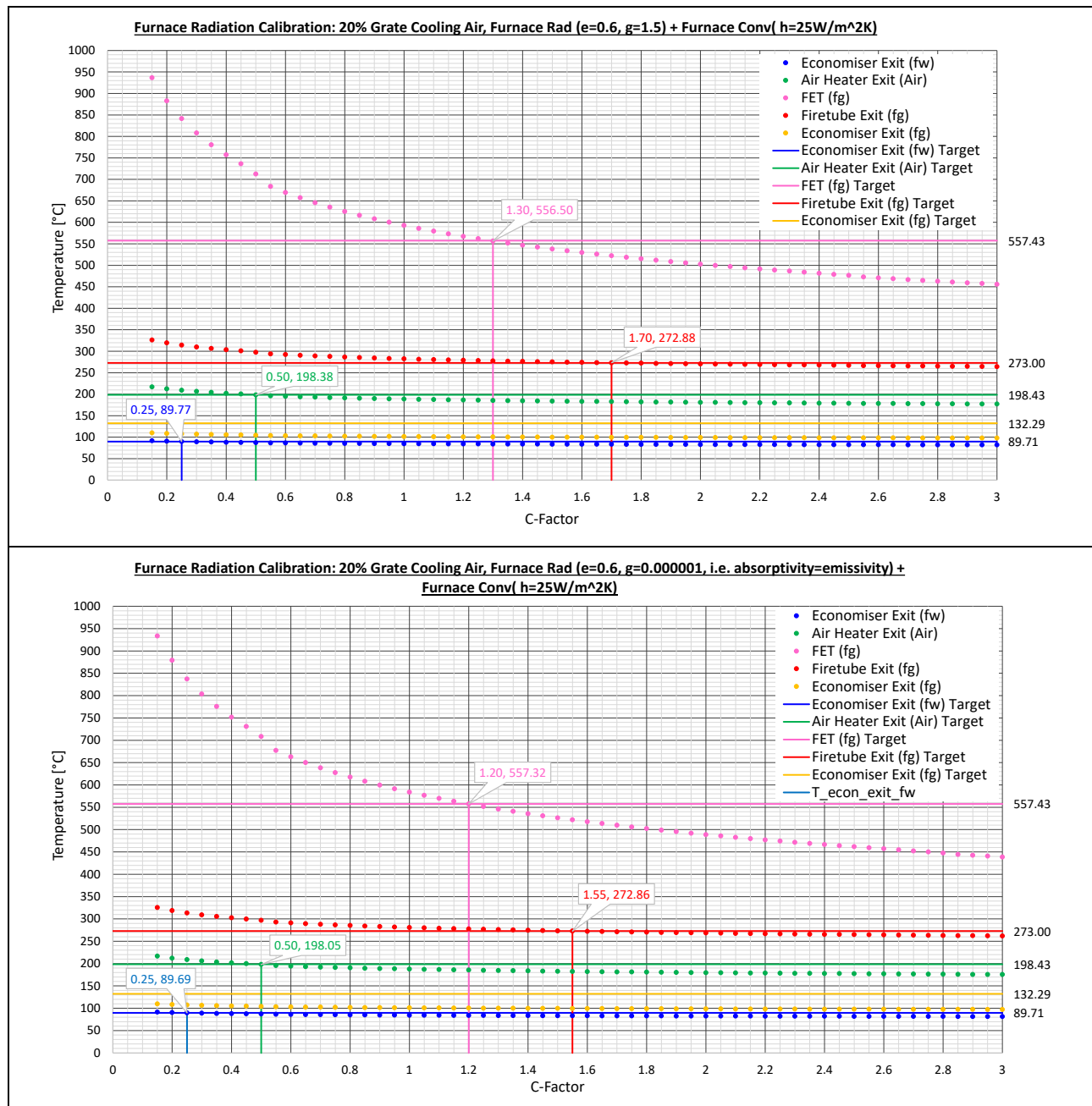
20% Grate Cooling Air – Radiation + Convection @10 W/m²K



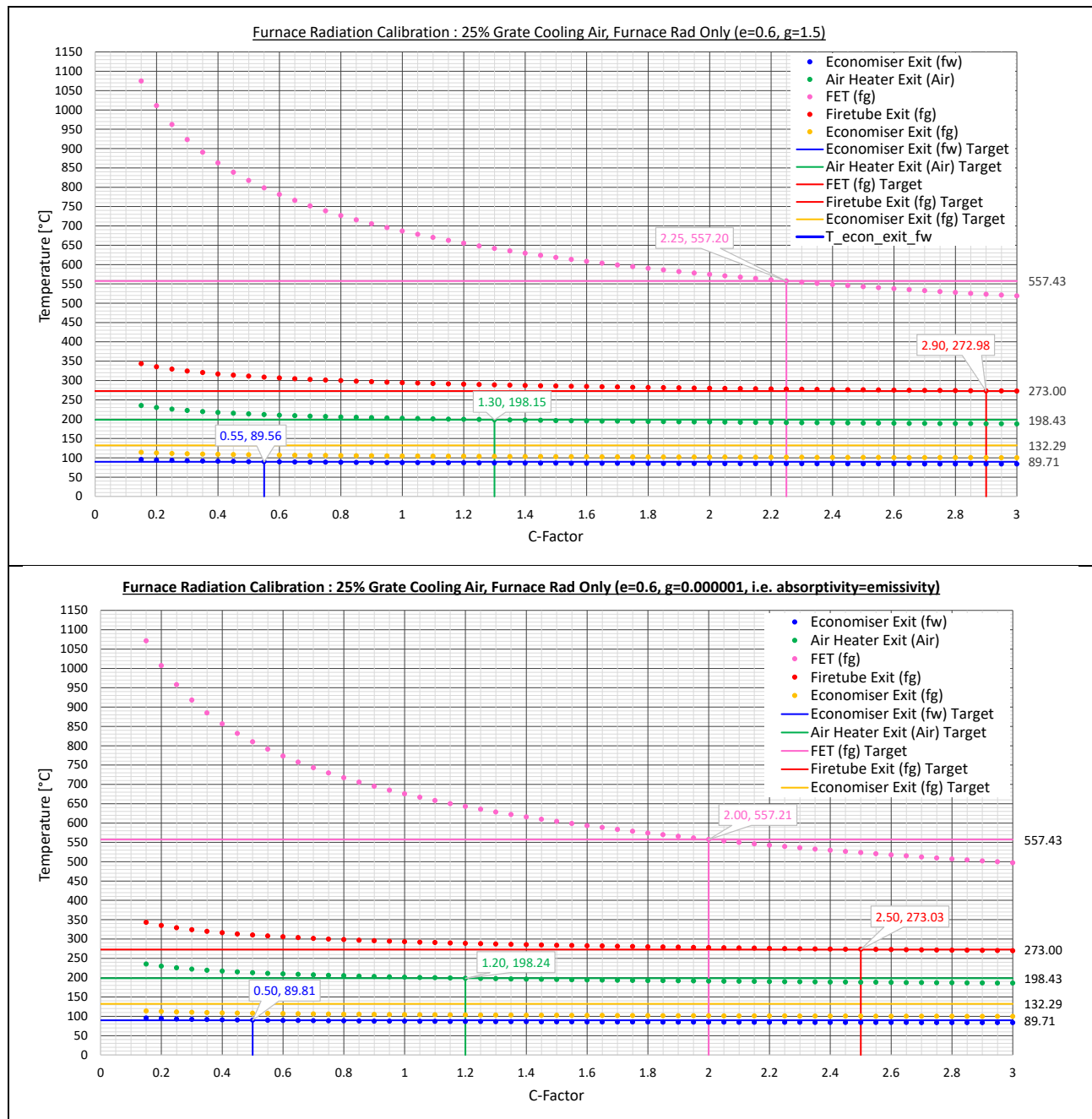
20% Grate Cooling Air – Radiation + Convection @15 W/m²K

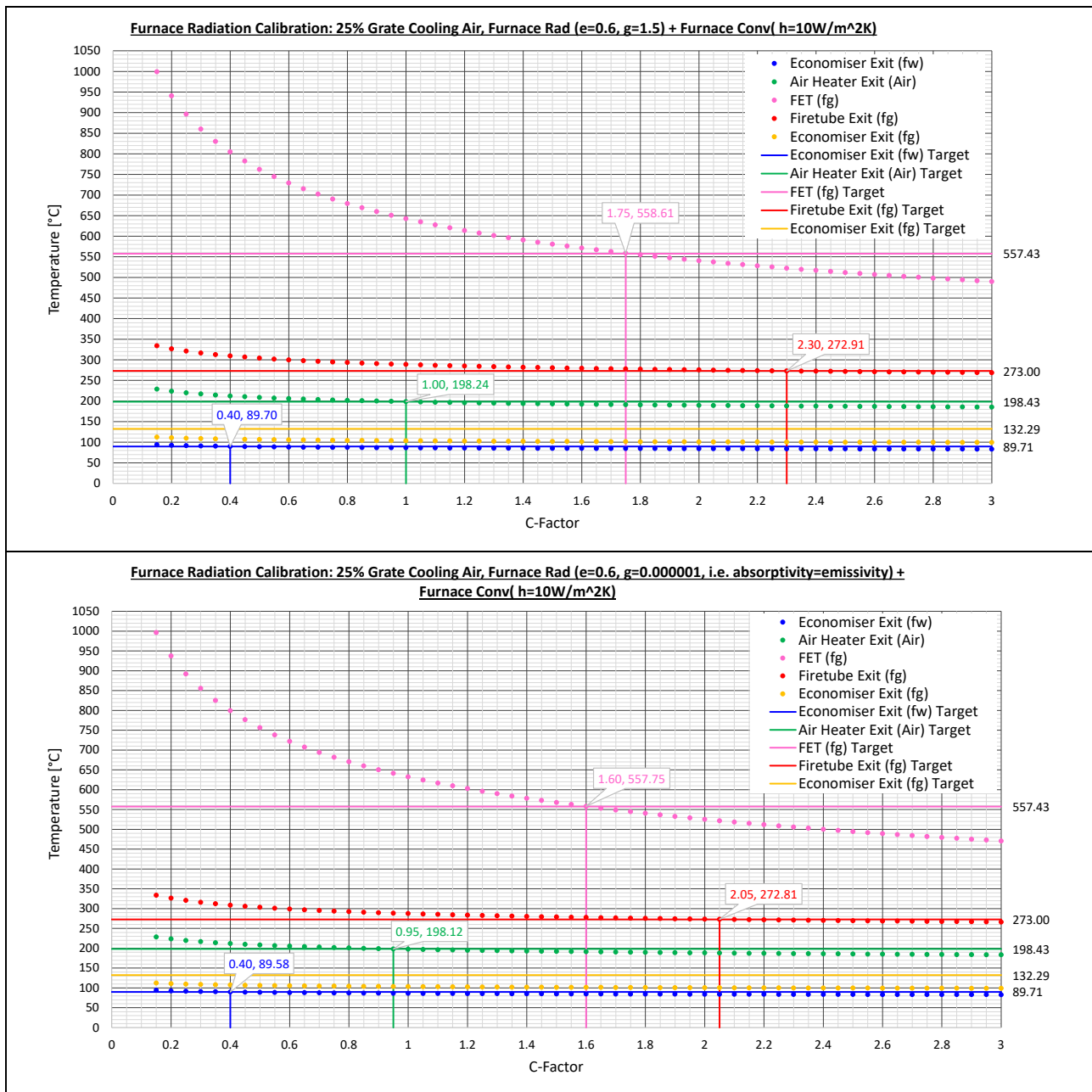


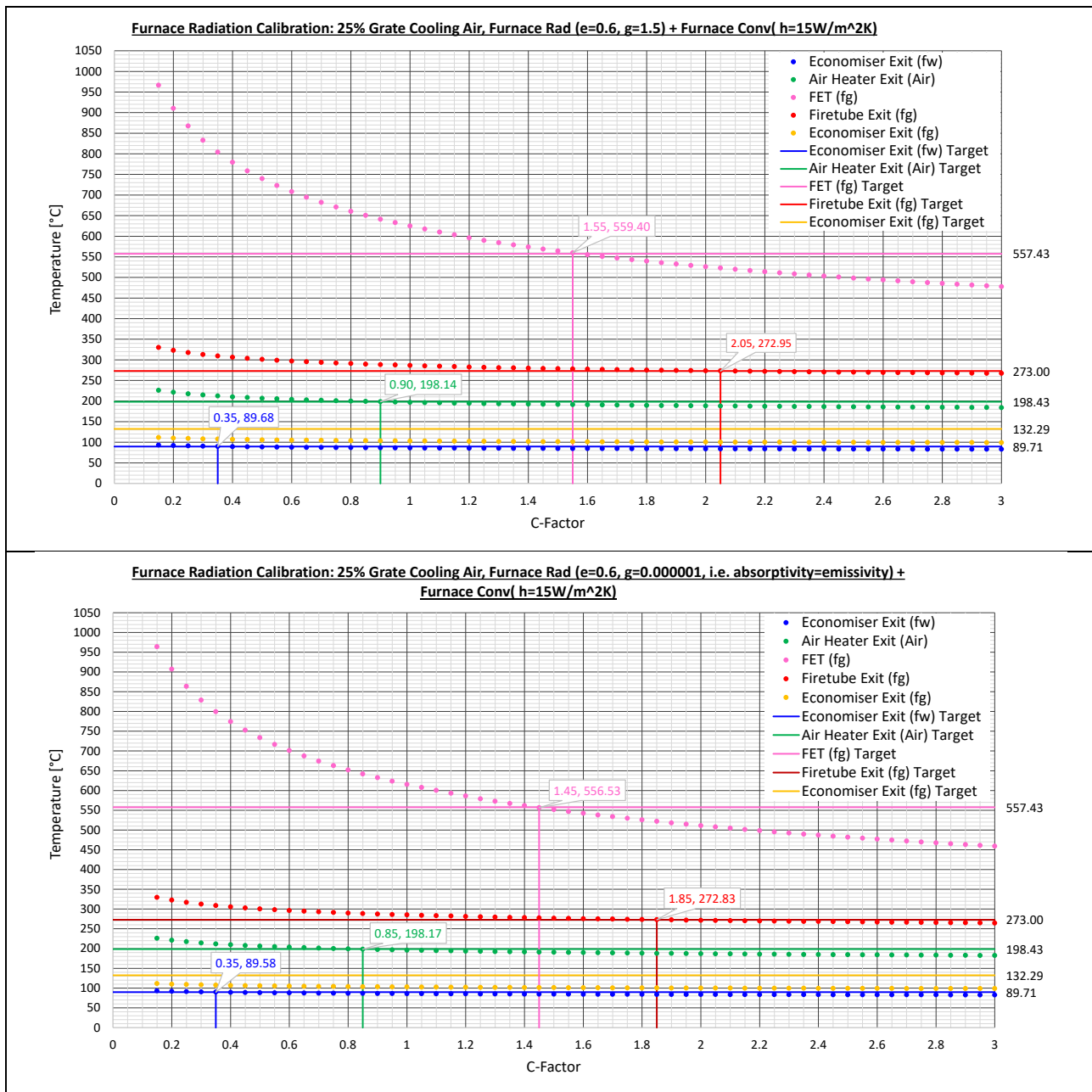
20% Grate Cooling Air – Radiation + Convection @20 W/m²K

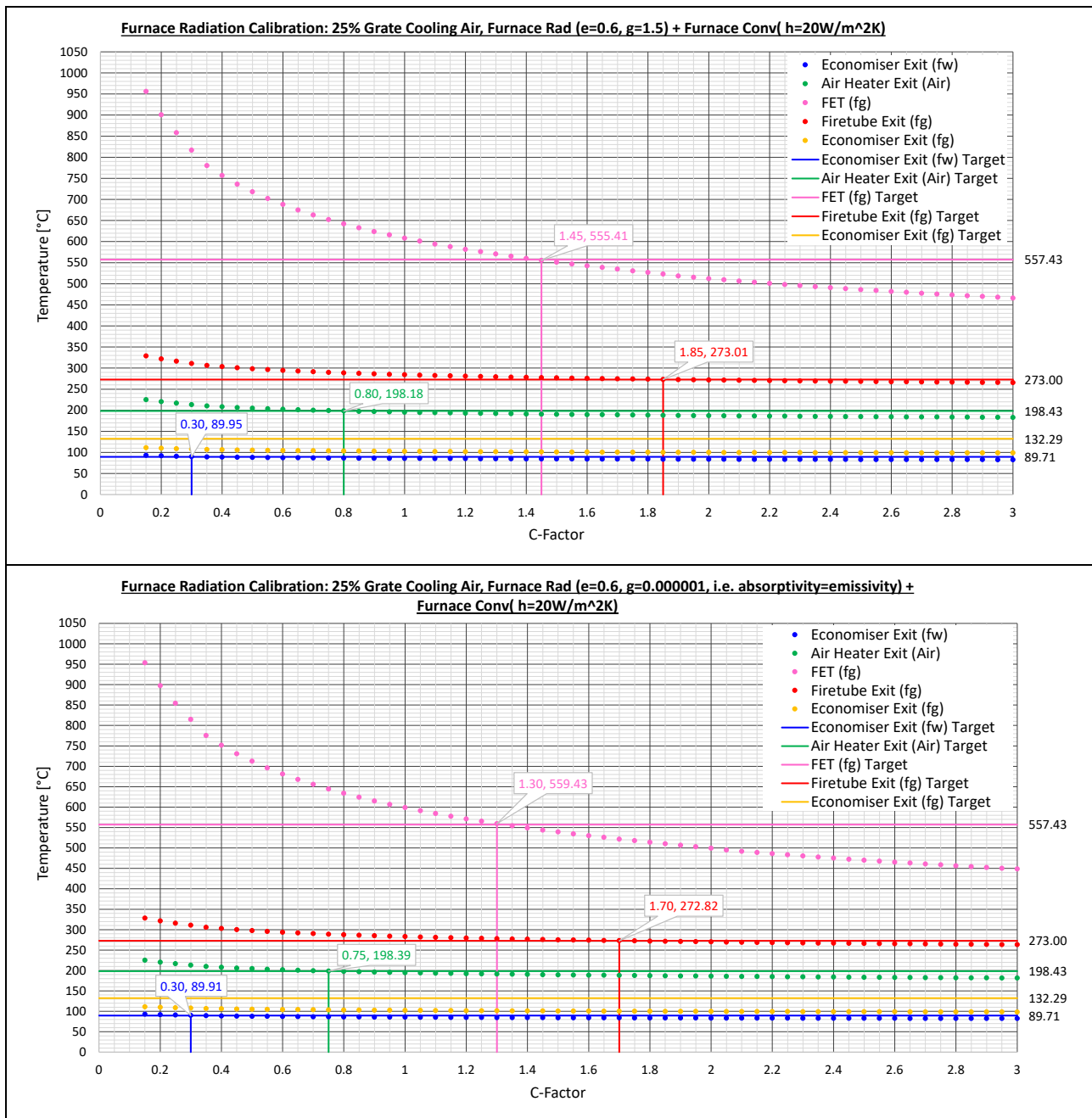
20% Grate Cooling Air – Radiation + Convection @25 W/m²K

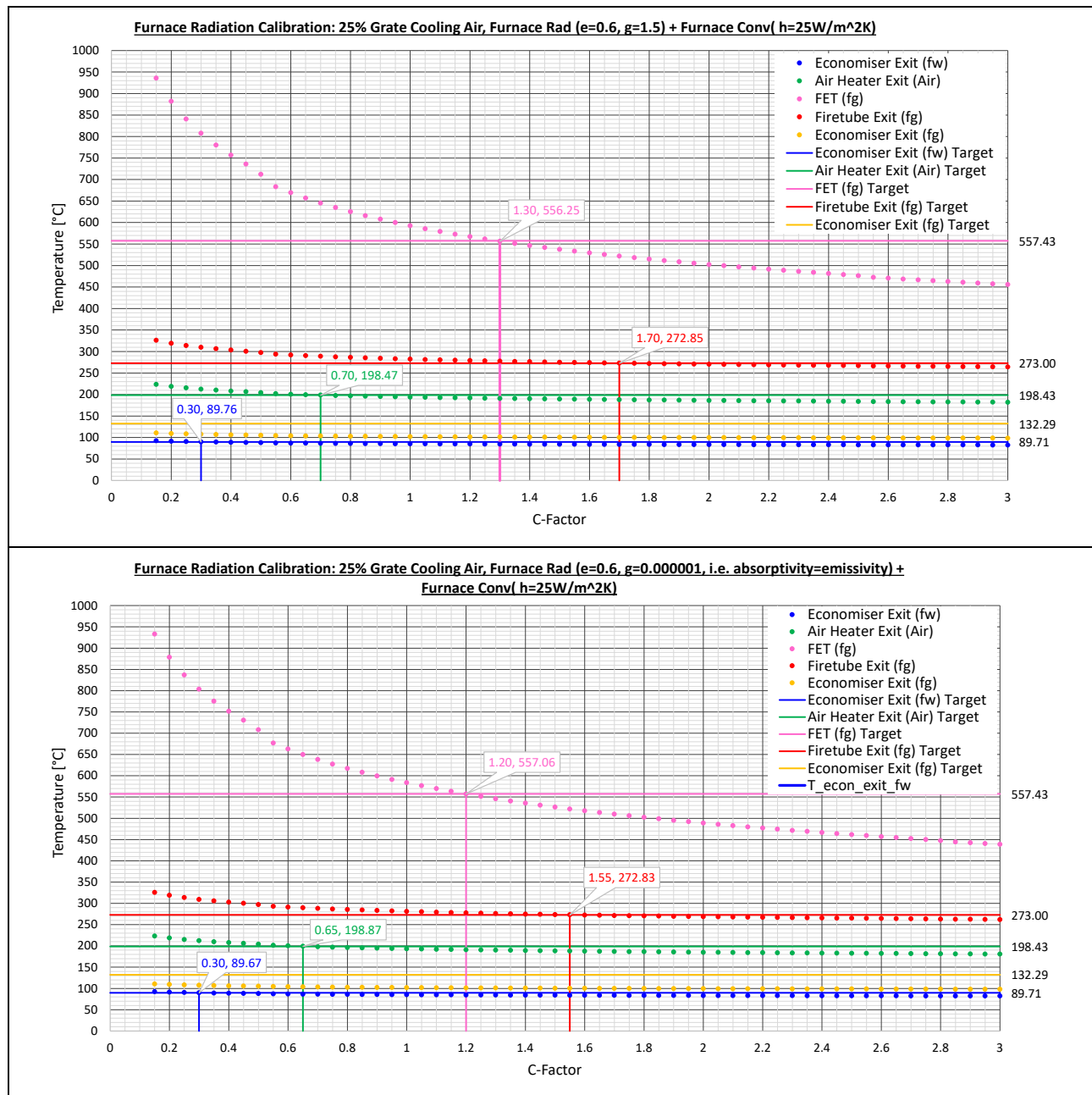
25% Grate Cooling Air – Radiation Only



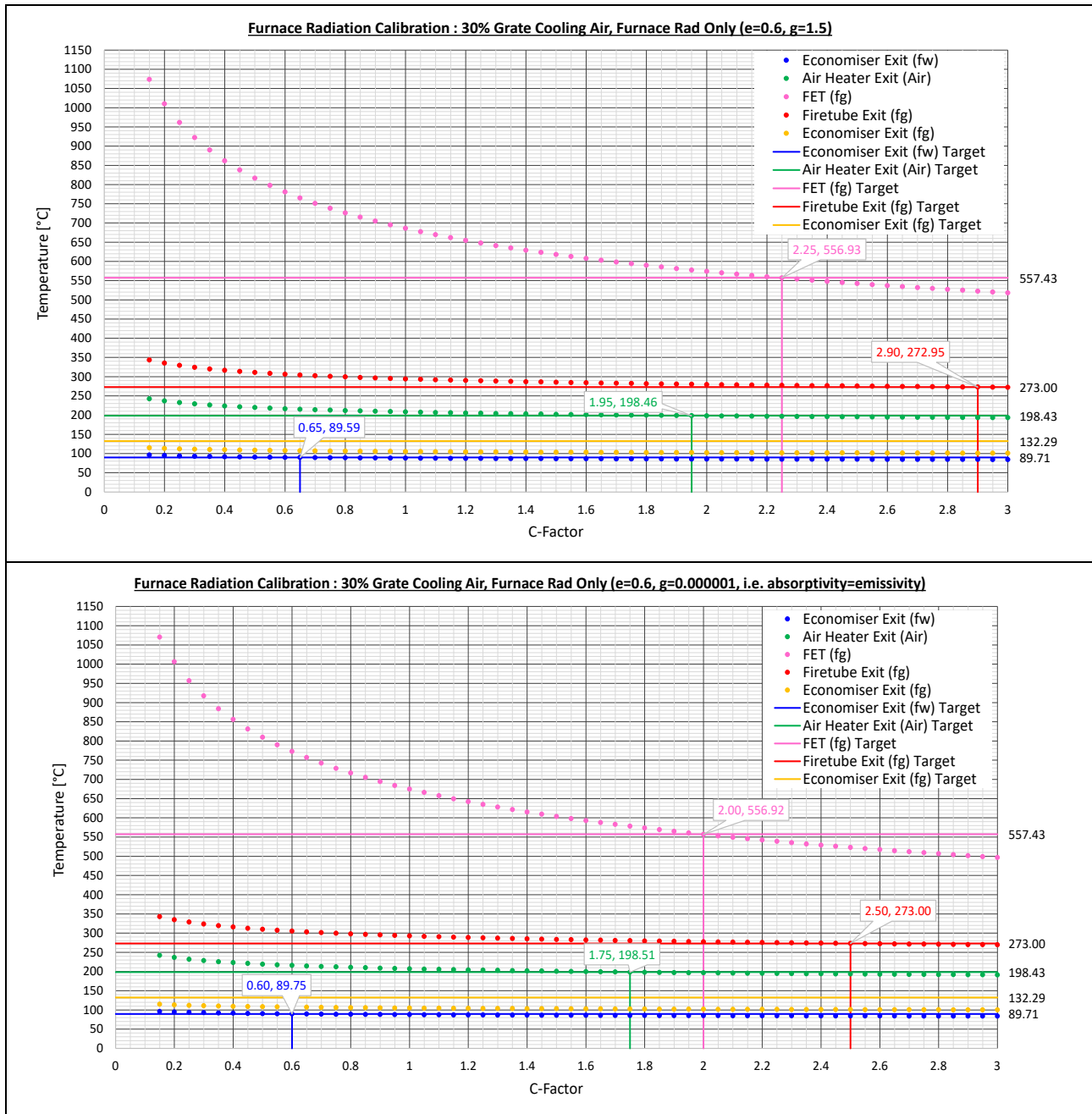
25% Grate Cooling Air – Radiation + Convection @10 W/m²K

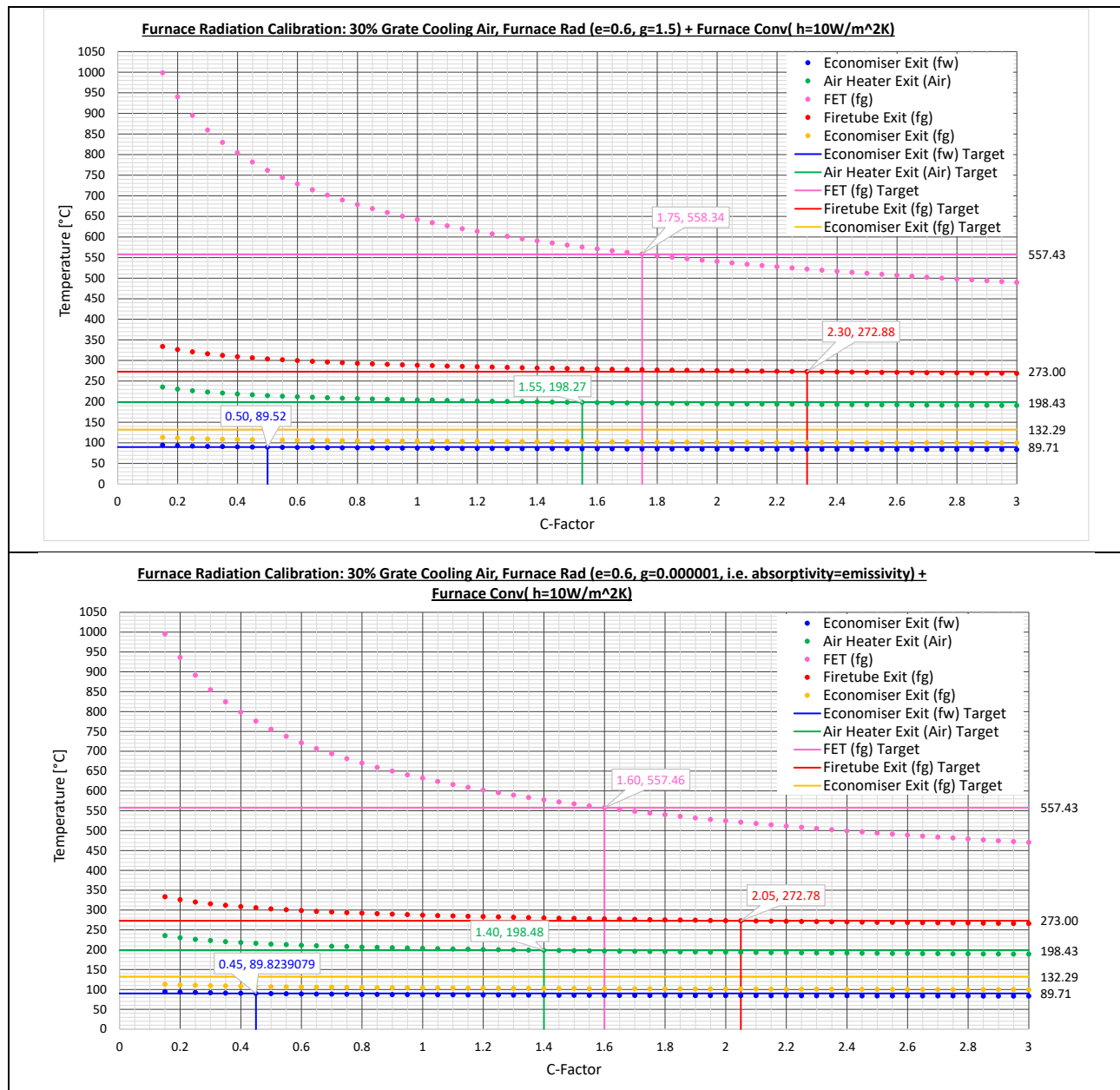
25% Grate Cooling Air – Radiation + Convection @15 W/m²K

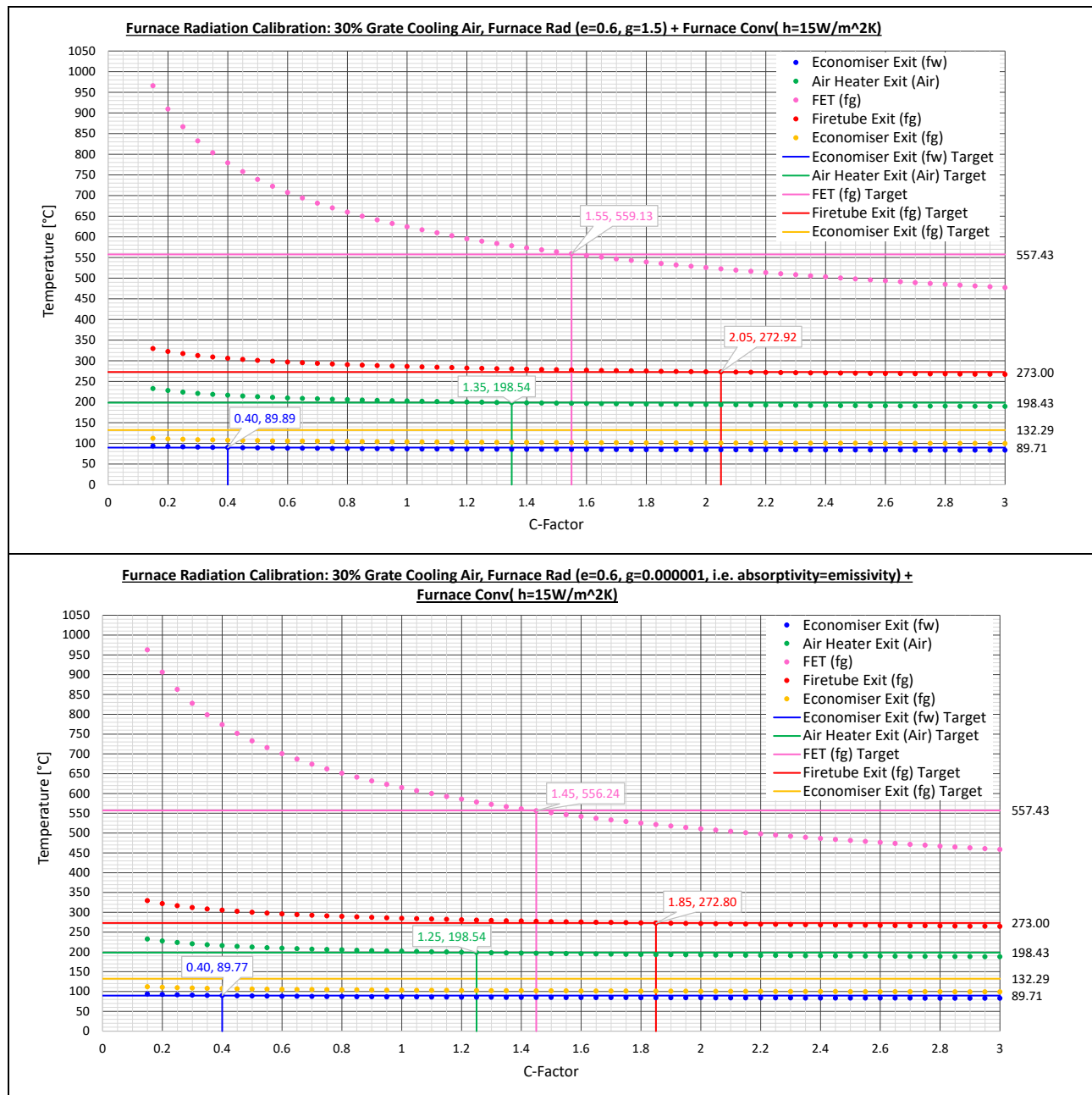
25% Grate Cooling Air – Radiation + Convection @20 W/m²K

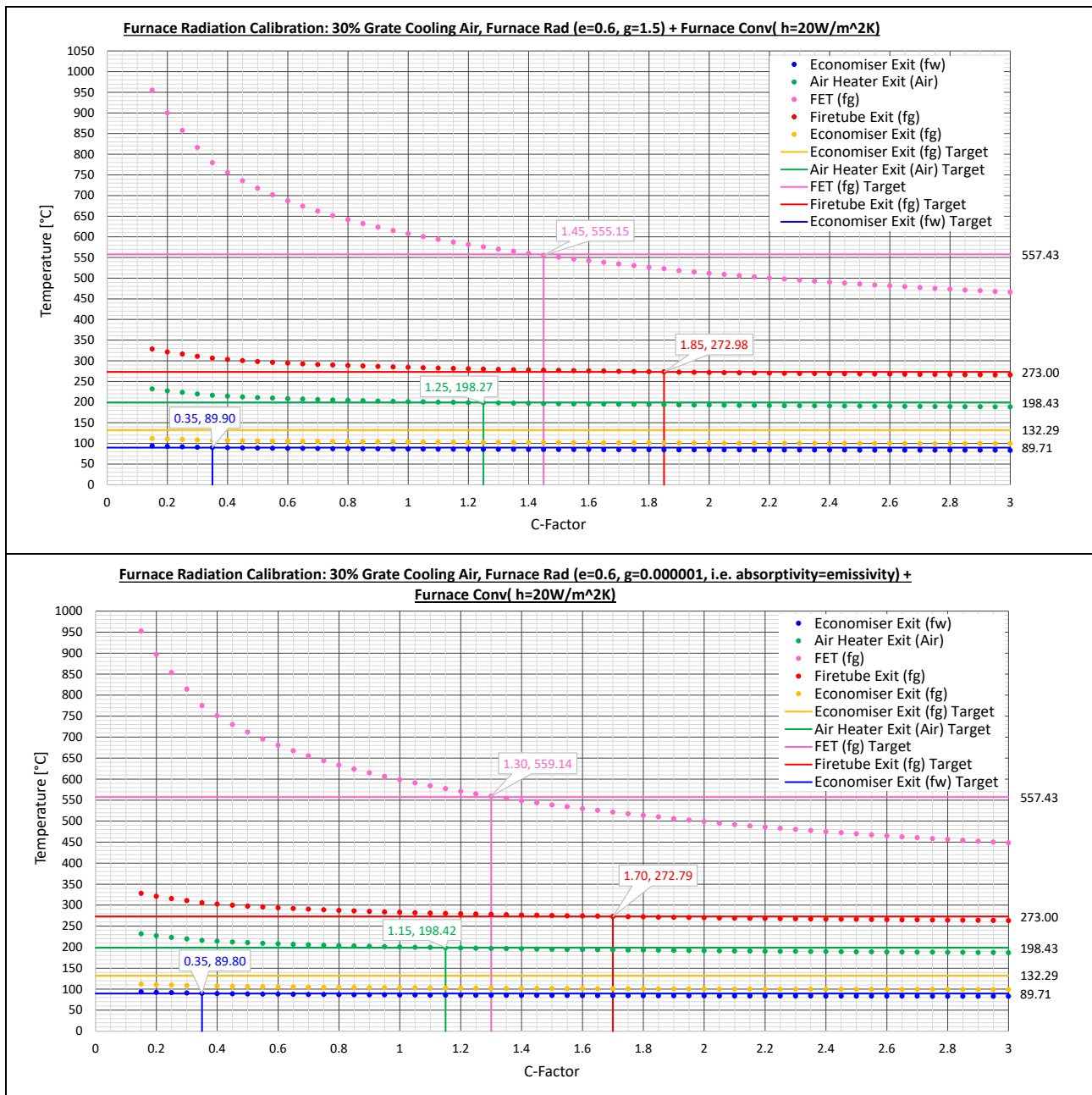
25% Grate Cooling Air – Radiation + Convection @25 W/m²K

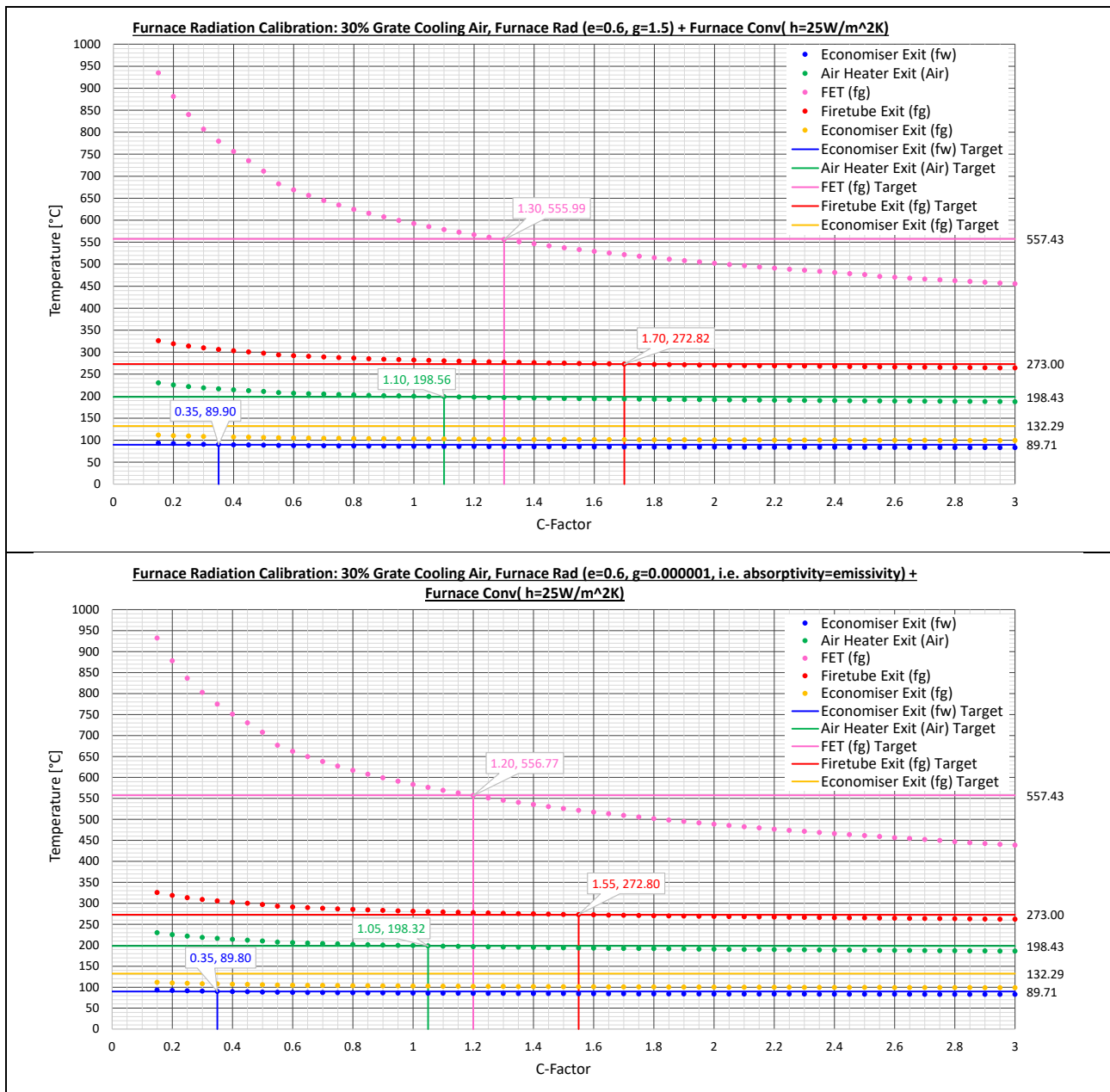
30% Grate Cooling Air – Radiation Only

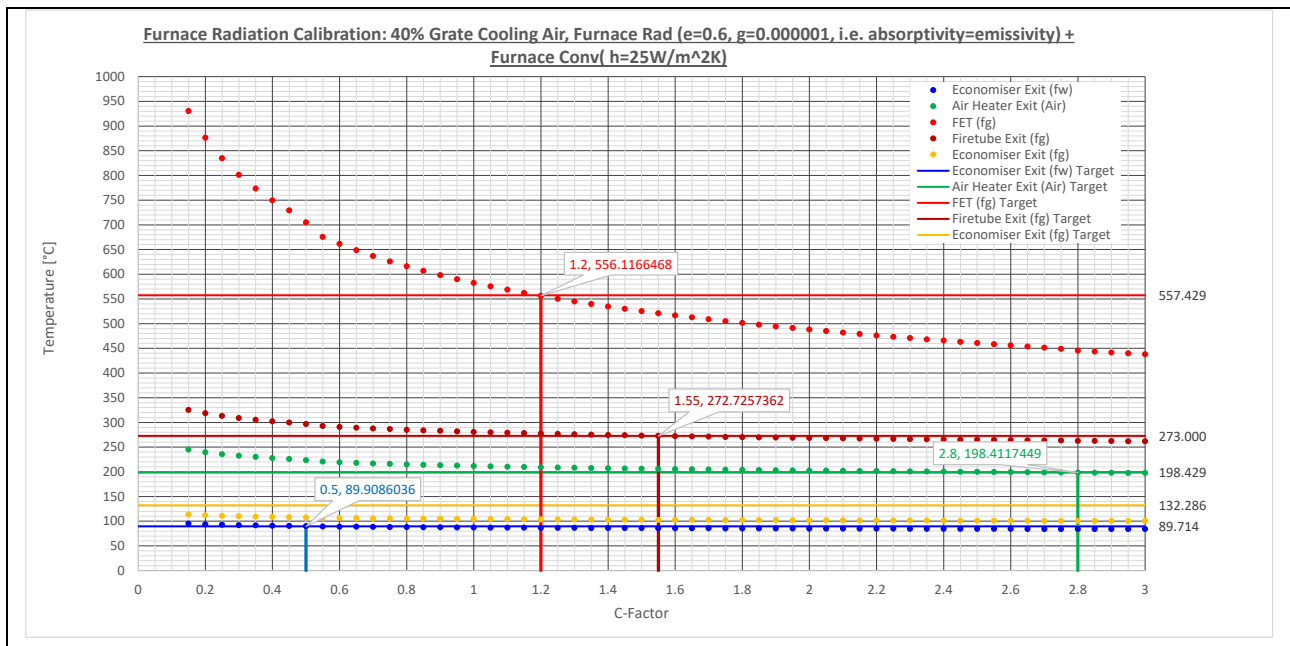


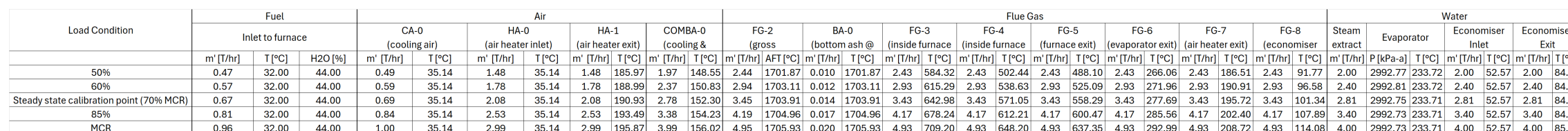
30% Grate Cooling Air – Radiation + Convection @10 W/m²K

30% Grate Cooling Air – Radiation + Convection @15 W/m²K

30% Grate Cooling Air – Radiation + Convection @20 W/m²K

30% Grate Cooling Air – Radiation + Convection @25 W/m²K

40% Grate Cooling Air – Radiation + Convection @25 W/m²K



Load Condition	Heat Transfer Rates						Efficiency [%]
	Combustion Heat [kW]	Radiation Heat [kW]	Furnace Convection [kW]	EVAP Convection [kW]	AH Convection [kW]	ECON Convection [kW]	
50%	1719.027	1089.414	84.001	185.611	63.858	74.441	87.10%
60%	2067.920	1279.017	95.550	256.081	78.373	89.232	86.96%
Steady state calibration point (70% MCR)	2425.018	1466.151	105.997	334.691	93.080	104.800	86.79%
85%	2948.897	1730.312	119.481	459.850	115.068	127.801	86.56%
MCR	3481.141	1987.812	131.340	567.131	137.895	151.287	86.34%