ASME Author review and rebuttal to reviewers’ comments and suggestions.

**Paper:** TSEA-21-1636

**Title:** Application of computational fluid dynamics and process modelling to investigate low-load operation of a subcritical utility-scale boiler.

**Reviewer - RECOM-TSEA-21-1636-0-304193**

The following is the responses to the above reviewers’ comments and suggestions. Where applicable the page number and implemented changes are given.

**RE: Technical Changes**

* Figure 1, Page 6 - Need to swap label for middle and bottom burners

Response: Changes have been made as highlighted on Page 6, Fig. 1

* Page 3, the first paragraph of the section “2.1.1 Fluid flow, turbulence and combustion modelling” – The sentence “The governing equations for the gas phase are written below in their respective Reynolds.” seem to be incomplete.

Response: The sentence was incomplete; it should have read: “The governing equations for the gas phase are written below in their respective Reynolds averaged forms”. This has been made on Page3, Section 2.1.1

* Page 3 – Equations (1) and (2) – the same notation *Sm* is used in both mass and momentum conservation equations.

Response: The mass source term has been changed to S, with the momentum source being kept at Sm, subsequently the source term definitions have been added to the nomenclature. Page 2 and 3 highlight these changes

* Page 10 – At the beginning it was written “The works of Dugum et al [9]...”. It is either Dugum et al [1] or Du et al [9].

Response: The reference should be Dugum et al [1], Page 10 highlights these changes.

**RE: Remarks and Suggestions**

1. Section “2.1.2 Particle modelling” ‐ In the last sentence the C programs are mentioned. Define what ‘C’ stands for. If those are the User Defined Functions that Ansys Fluent offers, it should be said so.

Response: The C refers to the C programming language, and as you pointed out the User-Defined Functions are indeed utilized. The changes to the wording are written as follows: "To solve the additional transport equations and their source terms user defined functions, utilized by *ANSYS Fluent*, were developed using the *C/C++* programming language." Highlighted on Page 4, Section 2.1.2

1. Section “2.3 Process simulation model” – A 1D discretized model of the furnace evaporator, platen SH, final SH, and subsequent downstream heat exchangers was developed using Flownex SE® 2021. This is an interesting feature of the work, but I am missing some details on how the two software packages (Fluent and Flownex) were coupled. Did both run in parallel with a continuous exchange of information, or in a decoupled manner. A more precise description would be very useful, especially since it is a novel work.

Response: The data transfer of the models is achieved using a one-way approach, where each CFD case represents a continuous operational steady state load, the solution data is transferred to the Flownex process model with no feedback loop. The reason for this is that the radiation heat transfer is the dominant form of heat transfer with temperature to the power of four, thus the effects of the surface temperature can be deemed negligible.

The following phrase has been included in the text, which is highlighted on Page 5, Section 2.3:

“The data transfer between the two models makes use of a one-way approach, where the CFD simulation data is transferred to the Flownex process model when convergence of each case is achieved, a further explanation of the coupling interface is given in Section 3.1”

1. Section “3.1 Geometry & process model set‐up”, paragraph 1 – Some details of the numerical grid of the boiler furnace should be given (i.e. number and distribution of cells), and preferably illustrated by a figure showing the grid and some important details (e.g. how were the burners and passages through superheaters discretized)

Response: The numerical grid consisted of 6 million cells, to ensure grid independence a further two numerical grids were made consisting of 3.5 and 10 million cells respectively. The aspect ratio was kept below 20 and orthogonal quality above 0.2.

To address this an additional phrase is incorporated into the paper, which is highlighted on page 8, section 3.3.

The phrase is written as such: “A numerical mesh consisting of approximately 6 million cells was used for the CFD simulations of this current study. To ensure that the results are grid size independent, simulations were also performed for mesh sizes of 3.5 million and 10 million cells during the validation study of Section 4.1. To ensure an accurate numerical simulation the aspect ratio and mesh orthogonality quality where kept below 20 and above 0.2 respectively”.

An additional figure has been added illustrating the burner mesh details and is provided on page 6, Fig. 2

1. Section “3.1 Geometry & process model set‐up”, paragraph 2 – I assume the burners are not of the swirling type, since it is not mentioned. What are the boundary conditions for the burner outlets? (Uniform velocity profiles and uniform coal particles concentration)? – opposing wall mounted swirl burners

In section 4.1. it is said that “The model inputs and thermal boundary conditions can be obtained from the study conducted by Laubscher and Rousseau [6], where using the same boiler of the present study”. That paper indeed offers some more details, however, neither grid details (apart from the number of cells) nor discrete phase boundary conditions are given.

Response: Addressing the first part of the remark, the burners are in fact swirling type, this has been clarified on Page 5 Section 3.1, with the following text being written:

“The boiler furnace is fed by six mills, each supplying a pulverised fuel and primary air (PA) mixture to a burner row consisting of six opposing wall mounted swirl burners. The swirl vanes were not modelled in this study, rather an axial and tangential velocity component were used for the secondary air (SA) inlets with these values coming from a detailed burner model, which includes the vanes, supplied by the burner manufacturer. The fuel/PA mixture is injected through the inner annulus of the burner while the SA air is fed through the outer annulus as seen in Fig. 2.”

The PA inlets consist of a uniform velocity inlet with a particle concentration based on the original mass scalar transport field, while the SA inlets are set to velocity inlets with an axial and tangential component. Note that for non-firing burners the PA inlet is set zero with no scalar concentration being set.

Secondly, the inputs provided from the study conducted by Laubscher and Rousseau [6] provide the inputs for the validation study for load cases of 100, 80 and 60 loads, which include the burner fuel, PA and SA flowrates and the various thermal boundary conditions.

The discrete phase boundary conditions are provided in the different paper that I forgot to add, subsequently I have added the reference to this paper in the text (Page 9 Section 4.1), for interest the paper is titled “Coupled simulation and validation of a utility-scale pulverized coal-fired boiler radiant final-stage superheater” by the same authors Laubscher and Rousseau.

1. Section “3.1 Geometry & process model set‐up” – what are the boundary conditions for the walls? Are some (e.g. the gas‐side wall temperature) provided from Flownex SE and used in Fluent CFD? These are important details and should be elaborated.

Response: The modelling of the walls in Fluent makes use of ANSYS Fluents convection thermal boundary condition type, which requires the free-stream temperatures, internal heat transfer coefficients, wall thicknesses and material conductivities. This is flux type boundary condition where the heat flux is calculated using the overall internal heat transfer coefficient and the change in temperature of the wall temperature (calculated by fluent) and the internal free stream temperature. The values of the internal heat transfer coefficient and the free stream temperature were calculated using process model for the low load case.

The following phrase and equations have been added to Section 3.1, page 6 elaborating on the wall boundary conditions.

“The CFD wall boundary conditions were modelled using ANSYS Fluent v19.5’s convection boundary condition type, which requires the internal free stream temperature, the internal heat transfer coefficient, wall thicknesses and material conductivities. Being a flux boundary condition the wall heat flux (q\_wall) is calculated using Eqn. ().

The overall heat transfer coefficient (U) is subsequently determined using the following expression.

A 1D process model was used to estimate the furnace, platen and final SH internal heat transfer coefficients and internal temperatures for the low load case.”

1. Section “3.2 Model inputs”, last paragraph – It would be informative if the authors could add the values of the excess air coefficient for the two different non‐firing burner SA flowrates.

Response: The excess air coefficients are added to Table 3, Page 7 as an additional row.

1. Figure 7 – The velocities are predominantly low, high velocities are encountered only close to the burner, resulting in predominantly blue color over the displayed sections. Consider modifying the color‐bar by reducing the maximum limit (50 m/s) to a lower value.

Response: Figure 7, Page 11 has been subsequently adjusted with a maximum limit 30 m/s.

**Reviewer - RECOM-TSEA-21-1636-0-304197**

The following is the responses to the above reviewers’ comments and suggestions. Where applicable the page number and implemented changes are given.

1. 2.1.1 Section. The authors state that “To correctly account for the particle inertial effects on the gas phase convection, the model makes use of an effective density which is defined…”. The reviewer is puzzled about this, as the particle phase is defined as an individual phase in Eulerian-Eulerian model in Section 2.1.2. Why to define the effective density?

Response: The use of an effective density is account for homogenous mixture being denser due to the presence of the pseudo particles, allows the model to account for the inertial effectives the particles would have on the gas in a very simple modelling addition. To clarify the developed model is similar to mixture model of ANSYS Fluent, where a single momentum equation is solved, however in order to adequately resolve the solid fuel combustion the resolution of the particle temperature is required. This takes our model to a thermal non-equilibrium representation whereby the gas and particle temperature cannot be assumed to be the same. Thus, our model makes use of a scalar field to resolve the energy exchange of the pseudo particle. In effect the produced model resolves two energy equations, one for the homogenous mixture and the pseudo particle. Not only is the resolution of the particle temperature paramount for combustion modeling but is essential for the resolution of the radiation field where particles have been shown to have a significant effect.

Additional text has been added as follows to page 3, Section 2.1 clarifying some of the differences of the developed model.

“The following modelling approach makes use of a Eulerian-Eulerian approach. This is similar to the Mixture Model of ANSYS Fluent v19.5, however the Mixture Model resolves a single energy equation which is used to determine the mixtures temperature. The model discussed in this section provides a user defined pseudo-particle energy equation that can resolve the pseudo-particles temperature. This is an important feature as it allows the adequate resolution of the solid fuel combustion phenomena and radiative heat transfer.”

1. 2.1.1 Section. As we know, the DPM model has been extensively used for the subcritical utility-scale boilers, except for the CFB boilers. In the subcritical utility-scale boiler, the particle load is very low (generally <5%), and the DPM model is adequate. If using the two-fluid model, how to track the evolution of solid phase at particle scale? How to descript the change of particle property, such as particle size, particle density, and particle component?

Response: The use of the scalar fields helps in resolving the particle properties based on the proximate analysis. The particles size is a fixed quantity set to the average volume-based diameter, taken from the Roslin Rammler distribution.

The study is not focused on the particle evolution but rather the macro effects that the average amount of particles, present in a cell, has on the radiative, temperature and combustion characteristics.

1. .2.1.2 Section. As indicated by the authors, the pseudo-particles scalar fields are used to define the fuel characteristics based on the proximate analysis composition. In my mind, in two-fluid model, the particle phase can be easily defined as an individual phase (mixture). Why using the scalar fields? Without the scalar fields, the Eulerian-Eulerian model can also predict the flow and combustion behavior in the boiler reasonably.

Response: The answers to remarks 1 and 2 we believe would resolve the above remark.

1. Page 6, Figure 1. Please check the order of the burner arrangement. The burner order seems to be wrong.

Response: Similar observation to previous reviewer, Changes have been made as highlighted on Page 6, Fig. 1

1. Section 3.2. Please clearly illustrate the differences of the Case 1 and 4, 2 and 5, and 3 and 6.

Response: Differences arise in the use of less SA air from the non-firing burners for cases 4, 5 and 6, whereas cases 1, 2 and 3 make use of the operational standard SA air flowrate (5 kg/s), these values are provided in table 3. Important to note that the firing arrangements stay the same. The reason being to investigate the effects of dry gas losses on the boiler efficiency.

Additional text is added to the paragraph of Section 3.2 to address these differences on page 8.

“A secondary air flow rate of 5 kg/s is typically fed through the non-firing burners, to ensure sufficient cooling of the burner and mixing of fuel and air in the combustion chamber, with these three cases numbered 1, 2 and 3. The result is a high air-fuel ratio in the furnace, which leads to higher dry gas losses. To lower this loss, this study will investigate the effect of reducing the air-fuel ratio by lowering the non-firing burner SA flowrate from 5 to 2.5 kg/s, with these three cases numbered 4, 5 and 6. Two permutations of the SA flowrate, at the non-firing burners, are used for each of the firing configurations mentioned above. Table 3 shows the input conditions for Cases 1 to 6. This data was obtained via conventional boiler mass and energy balance calculations.”

1. 3.3 Section, how to couple the 1D model with the CFD model in simulations. How to keep them in sync?

Response: Similar to previous reviewer remark number 2, an additional explanation to the coupling method is provided in that response. Changes are highlighted on Page 5, Section 2.3. and the coupling interface description is given in Section 3.1 Page 6.

1. 4.2 Section. CFD Model verification. Please supply more industrial data for comparisons, for example, furnace temperature.

Response: Unfortunately, data is not available for the extended operation at this low load, nor for the various burner firing configurations, only start up data is available which cannot be effectively used for the verification of steady state operation at low load. Purpose of study to inform the strategy of the firing configuration, not to validate existing configuration.

1. Page 9. “With this is mind”-🡪” “With this in mind”.

Response: Change has been made and highlighted on Page 9.

1. Conclusion Section. Please refine.

Response: Cannot think of anything to add.