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 is incomplete, it should read: “The governing equations for the gas phase are written below in their respective Reynolds averaged forms”. This has been changed 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 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 coupling of the models is achieved using decoupled approach, where each CFD case represents a continuous operational steady state load. Upon convergence of the simulation the data is exchanged to Flownex. The coupling interface is further described in Section 3.1 on page 6, this further explains the interfaces used to exchange the data. However, it would be beneficial to mention the precise method of coupling the two models. This is achieved with the addition of the following to Section 2.3: “ The coupling of the two models makes use of a decoupled approach, where the CFD simulation data is exchanged to the Flownex model when convergence of each case is achieved, a further explanation of the coupling interface is given in Section 3.1” Highlighted on Page 5, Section 2.3

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. To address this an additional phrase is incorporated into the paper which is highlighted on page 5, section 3.1. The phrase is written as such: “A numerical mesh consisting of roughly 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”. ADD IMAGE of meshed burner?

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)?

In section 4.1. it is said that “The model inputs and 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 description reading “The boiler furnace is fed by six mills, each supplying pulverised fuel and primary air (PA) mixture to a burner row consisting of six swirl burners.”

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 burner fuel, PA and SA flowrates and the various thermal boundary conditions. The discrete phase boundary conditions from the study of Laubscher and Rousseau [6]

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 CFD wall conditions are thermal boundary conditions, with the Flownex model accepting the heat loads from the CFD model to the various wall components, the following phrase has been added to Section 3.1, page 5 - 6 elaborating on the wall boundary conditions.

“The CFD wall boundary conditions were modelled using ANSYS Fluent v19.5’s convection boundary condition type. This boundary type requires internal free stream temperature (water or steam) and the inside heat transfer coefficient. To determine these values a 0D process model was setup using the Gurwich approach to estimate the internal heat transfer coefficient and internal temperature for these load cases.”

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: Used to define the effects of particle have on the gaseous phase without the use of defining drag coefficients

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: Study is not focused on the particle evolution but rather the macro effects that the average amount of particles has on the radiative, temperature and combustion characteristics. Thus, particles size are not an issue.

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: Combustion, adequately resolve the combustion phenomena,

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.

“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 this being executed cases 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 this permutation being executed in cases 3, 4 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.

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: Authors believe the conclusion to be as refined as possible