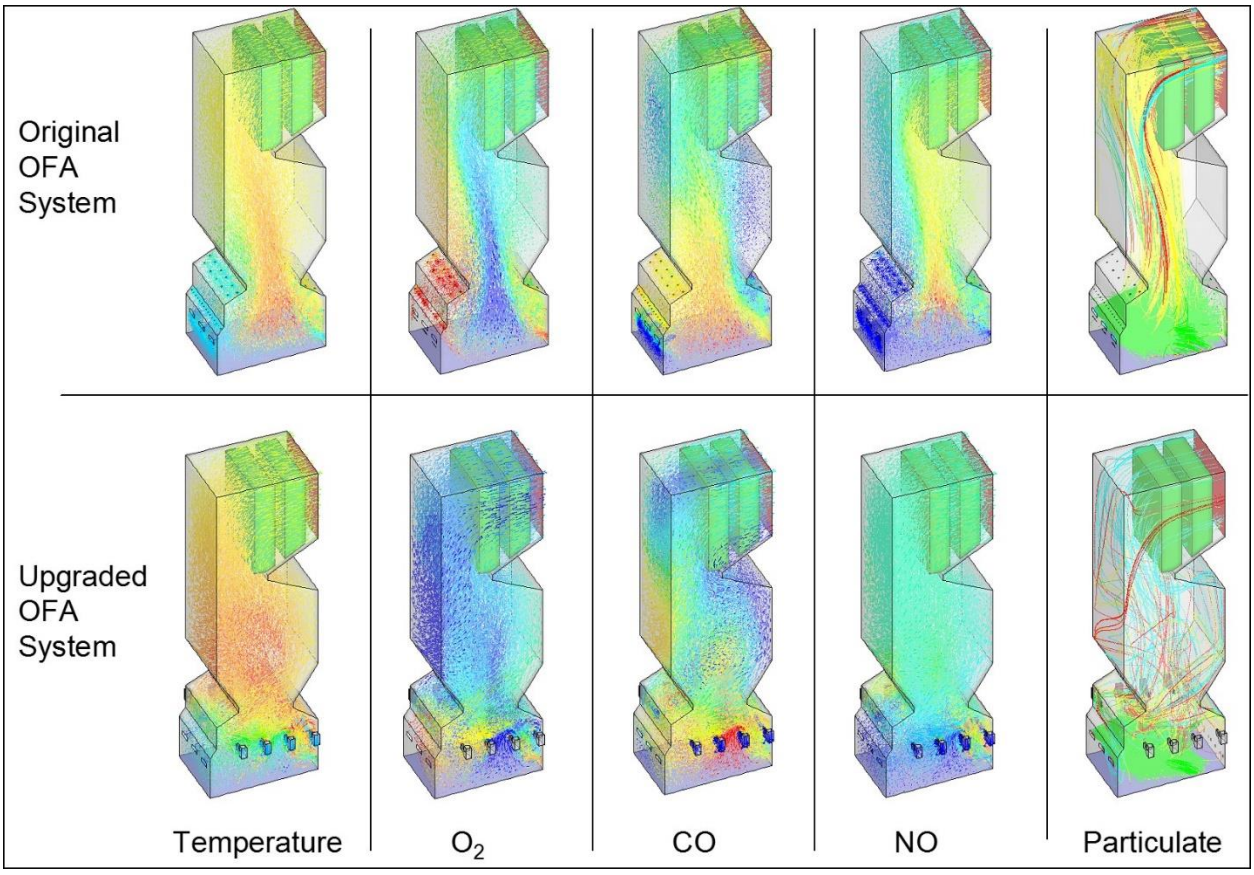
JANSEN
Combustion & Boiler Technologies, Inc.

Computational Fluid Dynamics (CFD) Modeling

Computational Fluid Dynamics (CFD) is an excellent tool for understanding current combustion air flow and heat transfer, and it becomes invaluable when considering options for improving boiler operation.

CFD can model the entire flue gas development process and flow from the bottom of the boiler to the stack inlet. It produces fluid flow patterns, fuel combustion processes, fuel capacity, emissions performance, and heat transfer profiles in the various boiler-related equipment.

The scope of a CFD project typically includes field data collection, set-up of a model to evaluate current flow characteristics, and the use of this model to evaluate conceptual improvements to the combustion system.



Actual Project Analysis

JANSEN
Combustion & Boiler Technologies, Inc.

Jansen has been conducting in-house CFD modeling for over 30 years. We've used it to modify operating parameters, and design and build upgraded components such as:

- Superheaters, Economizers, Air heaters
- Multiclone Dust Collectors
- Ducting and Precipitator Inlet
- Ducting and Stack Inlet
- Cyclones

CFD modeling is used to investigate, predict, and/or optimize performance of boiler equipment and can be used to develop a path forward to address:

- Overall boiler furnace combustion (see C-06).
- Heat transfer and erosion/corrosion factors.
- Poor collection efficiency or high pressure drop in cyclones.
- Heat transfer analysis in superheaters, generating banks, and economizers.
- Analysis of flow pattern and particulate distribution in inlet ducting to precipitators.
- Biasing of flow in ducting.
- Analysis of pressure losses and flow uniformity in flue gas ducting.
- Poor flow distribution in wet or dry ESPs.
- Spray cooler, scrubber, and cascade/cyclone evaporator performance predictions.