Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 143 (2017) 381–388

Energy Procedia

www.elsevier.com/locate/procedia

World Engineers Summit – Applied Energy Symposium & Forum: Low Carbon Cities & Urban Energy Joint Conference, WES-CUE 2017, 19–21 July 2017, Singapore

Computational Fluid Dynamics along the Energy Value Chain: From Natural Gas processing to Industrial End-Use

Mathieu Ourliac[a]*, Tang Yee Wah[b], Akim Rida[a]

[a]*ENGIE Lab CRIGEN, Saint Denis, France*
[b]*ENGIE Lab Singapore, Singapore*

Abstract

Systems involving fluid flows are commonly found along the energy value chain and are most of the time complex. Having good understanding and control are essential to warrant the system performance. Computational fluid dynamics (CFD) is a unique tool which predicts fluid flow phenomena using numerical algorithms. It gives access to non-measureable variables and allows the visualization of fluid flow patterns.

With the rapid advancement in computer sciences, it is now possible to handle most industrial processes involving fluid flows with CFD. The scope of application is large and issues that can be addressed are numerous, from process design validation and optimization to management of operational conditions changes to troubleshooting.

The application of CFD in troubleshooting will be highlighted through a few example of studies conducted by ENGIE Lab, on upstream natural gas treatment systems to downstream industrial end-use for combined heat and power plant and petrochemical furnace.

Peer-review under responsibility of the scientific committee of the World Engineers Summit – Applied Energy Symposium & Forum: Low Carbon Cities & Urban Energy Joint Conference.

Keywords: Computational fluid dynamics; combustion; industrial thermoprocessing equipment, burners, gas processing, combined heat and power; cracking furnace

* Corresponding author. Tel.: 0033-1-49-22-49-84.
E-mail address: mathieu.ourliac@engie.com

Peer-review under responsibility of the scientific committee of the World Engineers Summit – Applied Energy Symposium & Forum: Low Carbon Cities & Urban Energy Joint Conference.
10.1016/j.egypro.2017.12.700

1. Introduction

Computational Fluid Dynamic (CFD) enables the study on the dynamics of flow. Building a CFD model for a device, system or fluid process, allows the access to many information, not available on the real system. Changes in operating conditions or design, without any mechanical damage risk, can be studied. With a correct modeling, the software will generate a prediction of the fluid dynamics and the related physical phenomena. With long experience, even very complex physical phenomena can be handled: granular flow, highly turbulent flow, complex chemistry, moving bodies, fluid structure interaction and so on.

The main ‘risk’ of CFD is that a result is obtained, in any circumstances. The main responsibility of the engineer is therefore to perceive the flow dynamics, in order to select the most suitable approach and sub models. The engineer must also question, at every step, his own simulation results.

With the progress in modeling, software and computational infrastructure, it is now possible to handle very large simulations with complex physics such as: gas platform, industrial furnace, plant ventilation. There are three major reasons to use CFD:

- Insight. It is very useful to use CFD to design a system involving fluid flow prior to any prototype. There are many phenomena that can witnessed through CFD, which would not be visible through any other means. CFD gives a deeper insight into the design.
- Foresight. CFD is a tool for predicting what will happen to the fluids under a given set of circumstances by quickly answering many ‘*what if?*’ questions. It allows the prediction of design performance within a short time and the testing of many variations until an optimal result is achieved. All of these can be done before physical prototyping and testing.
- Troubleshooting. To our view, it is a very efficient and little known application of CFD. When a system involving fluid does not operate properly, CFD gives quick access to the invisibles in real operation. Root causes of malfunction can be identified and curative solutions can be validated with minimal operational risk.

2. CFD applications along the energy value chain

This powerful predictive tool can be applied on nearly the whole of the energy value chain in the oil and gas industry (Fig. 1), ranging from drilling to production and processing. A few examples are the pressure contours on stator and rotor in a downhole turbine, design and performances of a downhole steam generator, mud flow behavior for optimum cuttings removal in drill, contour gas volume fraction in a gas liquid separator etc.

Second generation bio-methane production involves fluidized bed reactors. Granular multiphase flows are complex, chemistry of pyrolysis is a key parameter and char content must be evaluated for various range of incoming biomass (size and humidity). ENGIE Lab used CFD to improve design of gasification reactor on steam injections location and the angle at the bottom of the reactor.

Finally, one main use of energy is heating by combustion. Progress in turbulent combustion modeling allows CFD to handle every combustion regime and heat transfer type (radiative and convective). This is presented in a petrochemical furnace and duct burner in a combined heat and power plant. Those two industrial thermoprocessing equipment faced operational difficulties. With CFD, these issues were resolved within a few weeks.