The role of CFD in the conceptual design of high rise buildings

Kermin CHOK
Structural Engineer
Meinhardt
Singapore
Kermin.chok@gmail.com

Kermin Chok received his civil engineering degree from NorthwesternUniverity, IL and MIT, MA. He has worked for SOM, Halvorson and Partners before becoming a technical director at Meinhardt. His research interests revolve around digital design and technology in structural engineering.

Yuya ANDO
General Manager
Cradle North America Inc.
Dayton, Ohio, USA
ando@cradle-cfd.com

Yuya Ando is General Manager at Cradle North America Inc., a subsidiary of Software Cradle Co., Ltd. He has been with Cradle for 10 years and has worked as a sales engineer and global marketing manager. He is currently managing Cradle North America pursuing strategic partnerships with several CAE companies and universities.

V V Sunil Kumar VYTLA
CFD Engineer
Cradle North America Inc.
Dayton, Ohio, USA
sunil@cradle-cfd.com

V V Sunil Kumar Vytla received his MS in Mechanical Engineering from University of Kentucky, KY and his Ph.D. in Engineering from Wright State University, OH. He is currently working as CFD Engineer at Cradle North America.

Summary

This paper discusses the authors' work of bringing high end computational fluid dynamics (CFD)analysis into the building design process. The authors have used two projects with clustered towers as reference for this effort. Full aerodynamic wind tunnel testing had already been completed and thus a data set was available for comparison. The paper will discuss the CFD methodologies used, successes and challenges. A comparison of wind loads from code calculated, wind tunnel testing and CFD will also be made. Finally, using the insights from the benchmarked CFD processes, the beneficial aerodynamic effects of venting through mid-height atria is examined in a proposed tower.

Keywords:CFD, Structural Design, Automation, High Rise Buildings.

1. Introduction

Wind engineering of high rise buildings is often be an iterative and linear process. In the design of high rise structures, wind tunnel testing is often initiated only after the concept and schematic design is complete. However, the structural system and key sizing of elements is usually confirmed after the completion of schematic design. Major re-designs of the structure and architecture can happen if building loads from wind tunnel testing are much larger than previously assumed code-based loading.

This paper explores how high-end computational fluid dynamics can be incorporated into the conceptual design of high rise structures. The goal is to supplement the traditional wind tunnel testing workflow to provide guidance on possible adverse wind effects on the proposed structure as early in the design stage as possible. These adverse wind effects result from individual tower vortex shedding or wake buffeting from adjacent towers.

With the incorporation of CFD, beneficial shapings of the architecture massing could be iteratively explored before the design concept is finalized. Such pro-active studies could have a major impact on structural costs and sustainability efforts.

This paper presents our work in comparing physical wind tunnel testing and CFD in two case studies of towers either clustered together or located in a dense urban environment. Based on these comparisons, a study is presented on the beneficial effects of incorporating large mid-height atria spaces on a proposed tower.

2. Modern Computational Fluid Dynamics (CFD)

2.1 Established Industrial Applications

The use of computational fluid dynamics is well established in the mechanical and aeronautical

https://doi.org/10.2749/222137912805110844