

Computational Fluid Dynamics Using ANSYS CFD

With the advancement of high performance computing (HPC), the capability to perform larger and more complex simulations has been put into the hands of more users.

The application of HPC has existed for many years, however the scalability has often been an issue, limiting the fidelity of analysis. ANSYS Inc. has spent significant effort in developing their software to obtain essentially linear scaling up to thousands of CPU's.

The ANSYS CFD Workshop will give the user the following:

- A brief introduction to CFD background.
- A background into the fundamentals of HPC as it relates to CFD
- Take the user through the generation of a complete CFD model.
- Demonstration of HPC as applied to the computational model.

The course will be taught by Mr. Ian Whenham, a senior CFD analyst from SimuTech Group's Toronto office. Ian has over 10 years of experience in performing consulting services and customer support for the ANSYS CFD product. This has included the development of CFD techniques for SimuTech Group's customers.

