# ME 566: CFD for Engineering Design

Fall 2012 Course Outline and Operation

**Instructor:** Gordon Stubley CPH 3677 x32875 stubley@uwaterloo.ca

NOTE: This version is the end-f-term. Changes made during the term are noted in red text and deletions are noted in blue text with ~~strikethrough.~~

## Course Goals and Learning Outcomes

1. Develop an enriched understanding of the physics of complex fluid flows:
   (a) **Analyze** a simulated or measured flow field to determine the most significant flow processes using quantitative analysis.
   (b) **Identify** how the significant flow processes impact flow design performance.
   (c) **Select** an optimal flow design from a set of flow design alternatives.
2. Introduce the major approximations implied in CFD analysis of complex turbulent flows and the methods for controlling the resulting errors:
   (a) **Describe** the basic computational methods used to obtain a solution and the relationship between CFD model specification and the CFD simulation.
   (b) **Generate** a CFD simulation.
   (c) **Evaluate** a CFD simulation to determine the sensitivity of the simulation to the model components and parameters.
   (d) **Evaluate** a CFD simulation to determine the simulation errors due to discretization schemes and iterative solution controls using quantitative analysis.
3. Develop a familiarity with the steps required to complete a flow analysis project:
   (a) **Describe** the required steps in a complete CFD analysis.
   (b) **Design** a CFD model.
   (c) **Communicate** the findings of a CFD analysis in a complete and concise report.

**Required References:**

[intr] ANSYS Inc., 2010, *ANSYS CFX Introduction, Release 13.0* , available on LEARN

[ref] ANSYS Inc., 2010, *ANSYS CFX Reference Guide, Release 13.0* , available on LEARN

[mod] ANSYS Inc., 2010, *ANSYS CFX-Solver Modeling Guide, Release 13.0* , available on LEARN

[thry] ANSYS Inc., 2010, *ANSYS CFX-Solver Theory Guide, Release 13.0* , available on LEARN

[W] White, F.M., 2008, *Fluid Mechanics, 7th Edition*, McGraw-Hill, Toronto.

**Recommended Reference:**

[V&M] Versteeg, H.K., and Malalasekera, W., 2007, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd Edition*, Addison Wesley Longman Ltd., Harlow England.

## Course Topics

### 1) Overview of CFD

| | |
|---|---|
| - design challenges in fluids engineering | |
| - components of a CFD project | [intr] 1. |
| - 1D fluid flow review (piping networks) | [W] 6.1-6.10 |

### 2) Operational Steps in CFD

| | |
|---|---|
| - problem definition | see ANSYS CFX Student User Manual |
| - CFD model specification | |
| - geometry modelling | |
| - mesh generation | |
| - physical modelling | |
| - solver operation | |
| - visualization and analysis of results | |

### 3) Quantification of Flow Processes

| | |
|---|---|
| - mass and momentum flows | [W] 3.2-3.4,4.2-4.3 [thry] 1.1 |
| - fluid stresses | [W] 2.1,2.2,4.3,6.5 [thry] 1.1 |
| - losses in fluid flow systems | [W] 3.5,3.7, 4.8-4.10 [thry] 2.1-2.2 |
| - scales of motion | [W] 5.4,7.1,7.4-7.5,6.6,8.1,8.4 |

### 4)Discretization Method

| | |
|---|---|
| - control volume topology | [thry] 10.1 |
| - discrete flows and control volume balance | [thry] 10.1 |
| - discrete equation transport properties | [thry] 10.3 |
| - discretization error | [ref] 6 [thry] 10.3 |
| - verification and validation | [ref] 6.1,6.4 |

### 5) Iterative Solution Convergence

| | |
|---|---|
| - discrete equation set | [thry] 10.2 |
| - iterative solution error and residual | [thry] 10.2 |
| - acceleration of convergence | [thry] 10.2 |
| - convergence error | [ref] 6.2 |

### 6) Physical Modelling

| | |
|---|---|
| - boundary conditions | [mod] 2. , 4. [thry] 2.8 |
| - fluid properties | [W] 1.8-1.9, Appendix A |
| - turbulence model | [W] 6.5 [thry] 2.1 - 2.3 |
| - model error | [ref] 6.3 |

### 7) Mesh Generation

| | |
|---|---|
| - structure meshes | see ANSYS Workbench Help (online) |
| - multi-block mesh topologies | |
| - unstructured meshes | |

### Course Requirements

**Class Participation - 15%:** There will be a variety of group and individual learning activities. Student engagement in the learning processes will be based on completion of learning activities and, to a lesser extent, on participation in class discussions, and contributions to maintaining a positive learning environment.

**Homework - 40%:** There will be three required ~~four~~ homework assignments in the first 2/3 of the course to prepare for the summative project. The fourth assignment will be optional.

**Project - 40%:** There will be one summative project. For the project you will be required to analyse a proposed fluid flow design change and to present your results in a comprehensive professional report.

**Oral Interview - 5%:** There will be a 15 minute interview at the end of the course to present your project findings and to confirm your understanding of key course concepts.

### Grading Policies

**Deadlines:** All homework assignments and the project will have firm submission deadlines. Work submitted after the deadline will receive zero credit.

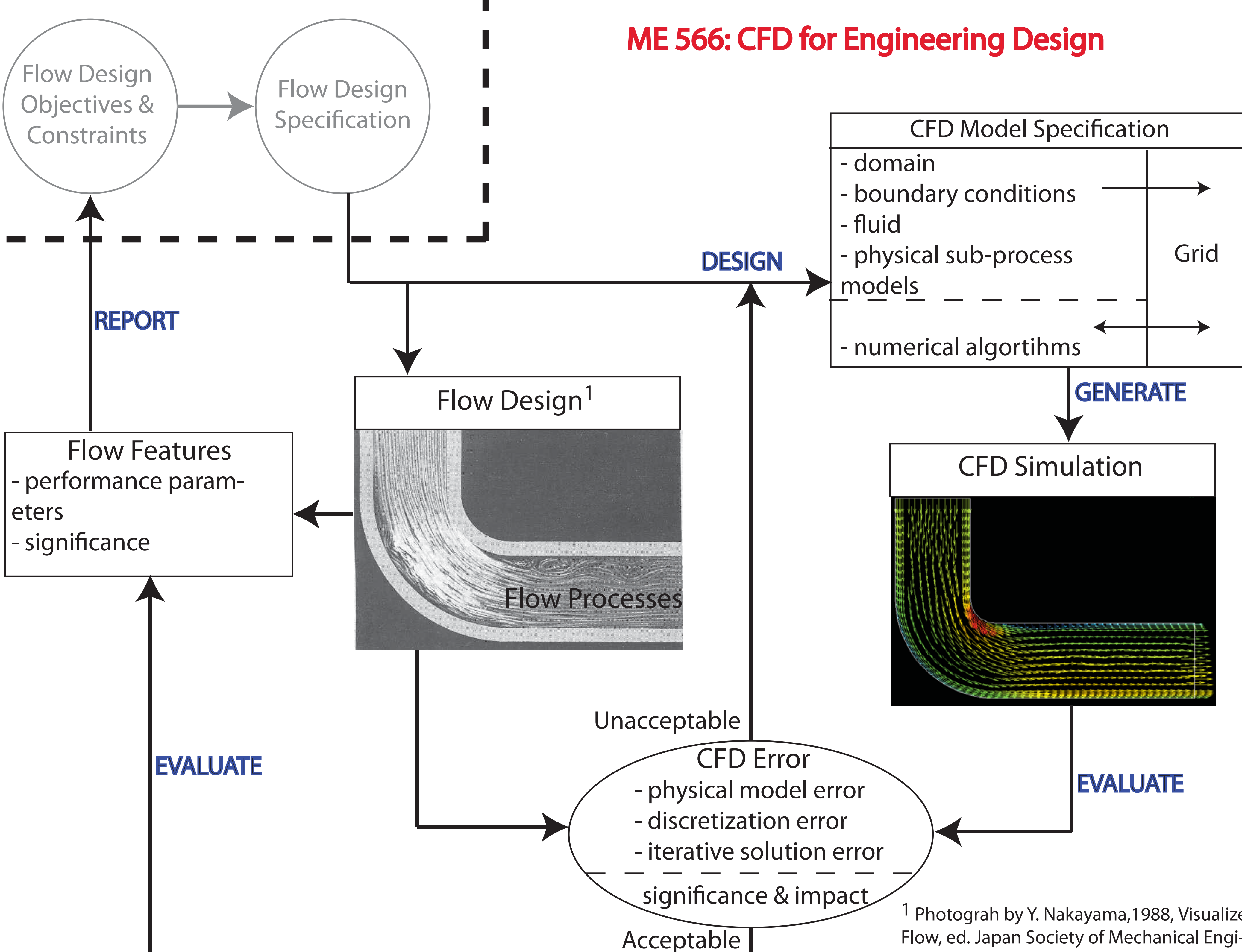
**Academic Integrity:** Integrity is a virtue in successful engineering careers. While students are encouraged to discuss concepts and challenges with each other, each student is to do all calculations, computer work, and written submissions on their own or in groups as specified. See Faculty of Engineering Course Responsibilities

### Other Course Information

**Software:** The course is based on ANSYS Workbench and CFX Version 13.0, codes written and marketed by ANSYS Inc. The software is available on the Nexus Windows system.

**Support Tutor:** Yusuke Koda, ERC-2018, ext. 38712, ykoda@uwaterloo.ca, will provide support during tutorials and will help with the homework grading. Questions about course concepts, homework expectations, etc. should be directed to the instructor.

# ME 566: CFD for Engineering Design




[1] Photograh by Y. Nakayama,1988, Visualized Flow, ed. Japan Society of Mechanical Engineers, Pergamon Press, Figure 108.