PIANC Workshop
13-14th September 2011

Part 3: CFD in lock design - progress and challenges

By Dr.ir. T. DE MULDER
BELGIUM

PIANC
Setting the course

Computational Fluid Dynamics (CFD) in lock design: Progress and challenges

Dr.ir. T. De Mulder

Flemish Authorities – Mobility and Public Works Department
Waterbouwkundig Laboratorium – Antwerp, Belgium

flanders
HYDRAULICS RESEARCH

- Introduction
- Terminology
- CFD: some important ingredients
- Use of CFD in design
- Concluding remarks

Introduction

PIANC Workshop “Innovations in Navigation Lock design”
Brussels, Belgium, 15-17 October 2009

*Paper 10 by **C. Thorenz** (BAW)*
“Computational Fluid Dynamics in lock design – State of the art”

RTFP !

Thank you for your kind attention !

This presentation:

- some difference in focus
- my personal (coloured) view
- some literature survey

Computational Fluid Dynamics

•Generic term for all kinds of numerical simulations of fluid flows (0D/1D/2D/3D)

vs.

➡ •3D models based on Navier-Stokes equations (**CFD**)

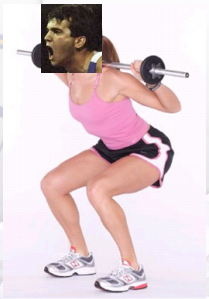
Fluent, CFX, Flow3D, Star-CD/Comet*,…(commercial)*
OpenFOAM*,… (open source, GNU)*
NaSt3DGP *(Univ. Bonn & BAW)*
ADH *(USACE)*
…

Numerical modelling in design

Huge computational effort !

0D/1D/2D models

3D CFD models
without/with free surface
without/with FSI



CFD models: some important ingredients

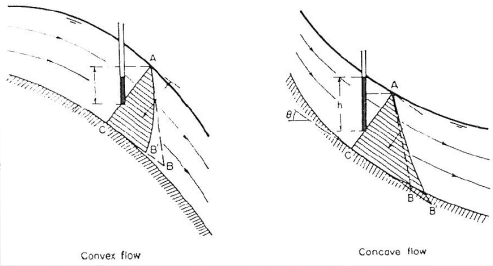
- Velocity and pressure formulation
- Free surface modelling
- Turbulence modelling
- Boundary conditions
- Mesh generation and optimization
- Fluid-structure interaction
- Numerical diffusion and dispersion
- Convergence
- Computer programme
- Computer platform
- CFD practitioner
- …

Strongly coupled !

Velocity-pressure formulation

Solvers based on **Shallow Water Equations** (1D/2D/"3D")

- gently curved streamlines
- vertical acceleration negligible
- hydrostatic pressure distribution
- unknowns: **(depth-averaged) velocity**
  water depth



CFD solvers (3D)

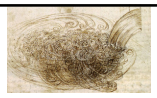
- unknowns: **(local) velocity** components and **pressure**
- any (i.e. non-hydrostatic) pressure distribution
- large computational effort

Free-surface modelling

CFD codes

- originally developed for pressurized flow
- **free-surface** flow is an "add-on"
- increases computational effort / burden
- experienced CFD practitioners often from outside
  "open channel hydraulics community"
- "rigid lid" approximation

Turbulence modelling



- Direct Numerical Simulation (DNS)

resolve all scales

$$N_x N_y N_z \propto (L/\eta)^3 \propto Re_L^{9/4}$$

- Large Eddy Simulation (LES)

resolve large (energetic) eddies
model subgrid scales

$$(N_x N_y N_z)_{\text{viscous sublayer near wall}} \propto Re_L^{1.8}$$

- Hybrid LES / RANS
- Reynolds-Averaged Navier-Stokes models (RANS)

resolve time average flow

$$\nu_{molecular} \approx 10^{-6}\ [m^2/s]$$

$$\nu_{eddy} \approx u\ [m/s] * \ell\ [m]$$

  - Eddy Viscosity Models
    - ✓ 1-equation-models
    - ✓ 2-equation-models
      - k = turbulent kinetic energy
      - ε = dissipation rate
      - ω = specific dissipation rate (~k/ε)
      - …
    - ✓ etc.
  - Reynolds Stress Models (6 eqs.)

model	turb.velocity scale u	turb.length scale ℓ
$k-\varepsilon$	$\sqrt{k}$	$k^{3/2}/\varepsilon$
$k-\omega$	$\sqrt{k}$	$k^{1/2}/\omega$

Turbulence modelling

k-ε models

- "standard" k-ε model (flat plate boundary layer)
- modified versions:
  - free jet
  - impinging jet
  - adverse pressure gradient
  - swirling flow,…

There are hundreds of RANS models…
Indicative of search for something better and/or something more "universal"…

k-ω models

- "standard" k-ω model
- modified versions

etc.

Turbulence modelling

Validation of turbulence models ?

Let's make abstraction of intrinsic accuracy of experimental data against which model predictions are compared !?

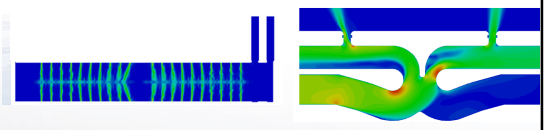
➔ benchmark cases in literature

➔ some relevant for lock design

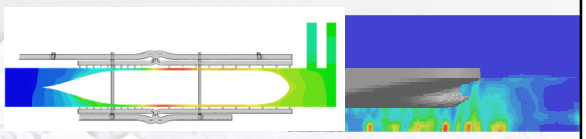
(head losses and hawser forces):

- Free jet
- 3D wall jet
- Impinging jet
- Separated flow in 2D diffuser
- Separated flow in 3D diffuser
- 90° bend
- …

(GUPC/CICP, 2010 ; S. Roux & N. Badano, 2011)

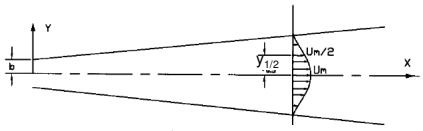


(CPP, 2007)

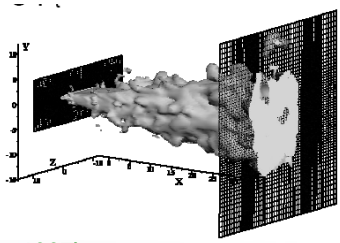


Turbulence modelling

Free jet



$$\text{spreading rate} = \frac{d}{dx}(\text{half width}) = \dot{y}_{1/2}$$



(Bardina et al., 1997)

Plane jet	Spreading rate
Experiment	0.100–0.110
Launder-Sharma k-ε model	0.108
Wilcox k-ω model	0.092–0.132
Menter *SST* model	0.112
Spalart-Allmaras model	0.143

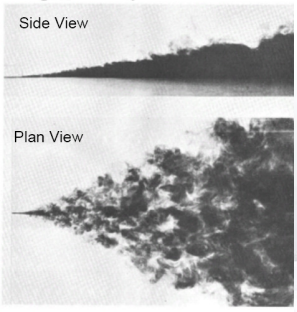
Round jet	Spreading rate
Experiment	0.086–0.095
Launder-Sharma k-ε model	0.120
Wilcox k-ω model	0.169–0.356
Menter *SST* model	0.127
Spalart-Allmaras model	0.253

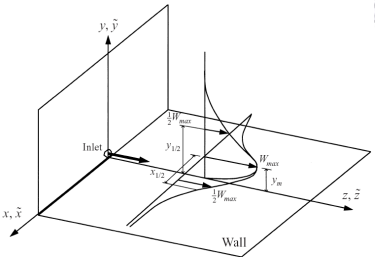
round jet / plane jet anomaly ➔ revised models (e.g. 1998 k-ω)

Turbulence modelling

3D Wall jet

(B. Launder & W. Rodi, 1983)





$$\frac{\dot{x}_{1/2}}{\dot{y}_{1/2}} = \frac{\frac{dx_{1/2}}{dz}}{\frac{dy_{1/2}}{dz}} = \frac{\text{lateral spreading rate}}{\text{spreading rate normal to wall}}$$

(T.J. Craft & B.E. Launder, 2001)

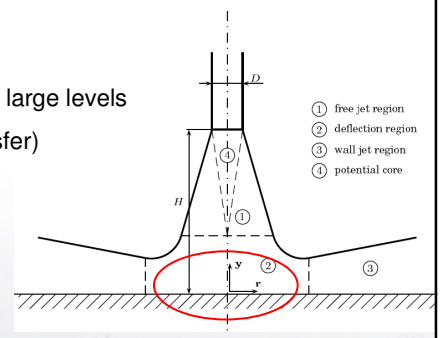
- **Experiments**: **5 to 9**
- **Eddy Viscosity Models** (k-ε) : **0.9**
- **Reynolds Stress Models**: **1 to 15**

Turbulence modelling

Impinging jet

(T.J. Craft et al., 1993)

Most Eddy Viscosity Models predict too large levels of turbulence (hence mixing / heat transfer) near **stagnation point**



(~streamline curvature)

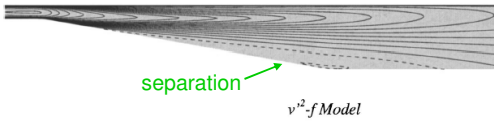
(~wall treatment)

➔ ad hoc correction to models

or change in turbulence model's transported variables (e.g. RSM)

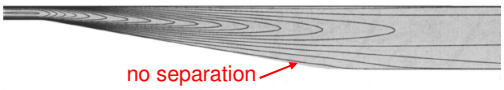
Turbulence modelling

Separated flow in 2D diffuser (G. Iaccarino, 2001)



$v^{'2}$-f Model

- Four equation $\overline{v'^2} - f$ model of Durbin
  - ▷ Modified form of the $k - \epsilon$ equations
  - ▷ $\overline{v'^2}$ equation – turbulence velocity scale
  - ▷ Production function f for $\overline{v'^2}$



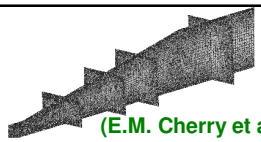
Low-Reynolds k-ε Model

- Low Reynolds number $k - \epsilon$ model of Launder and Sharma
  - ▷ No wall functions
  - ▷ Damping functions: algebraic relationships near the wall

Fig. 4 Mean streamwise velocity—CFX. Contour levels Min=−0.05; max=1.0, Δ=0.05 (dashed lines negative values).

Turbulence modelling

Separated flow in 3D diffuser



(E.M. Cherry et al., 2006)

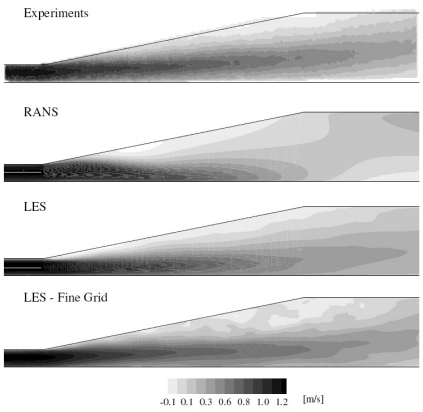


FIGURE 6. Streamwise velocity isolevels in a longitudinal section midspan in the diffuser 1

(all models: too thin boundary layer near bottom wall)

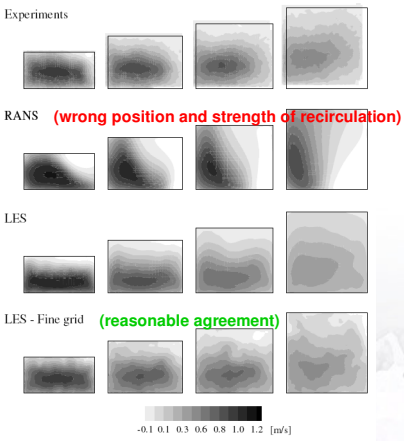


FIGURE 5. Streamwise velocity isolevels in four cross-sections along the diffuser 1.

Turbulence modelling

Separated flow in 3D diffuser (A. Karvinen & H. Ahlstedt, 2008)

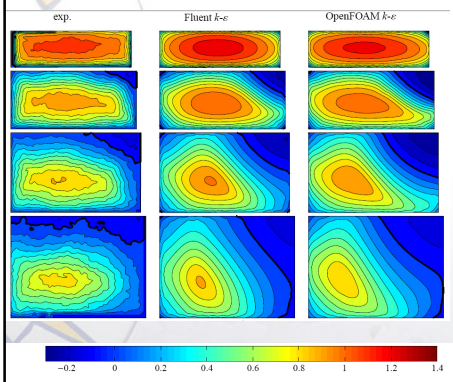


Figure 4: Mean streamwise velocities, u/u_{inlet}, at diffuser cross-sections various distances downstream from the origin using k-ε model. Contour lines are spaced 0.1 apart. The zero-streamwise-velocity contour line is thicker than others.

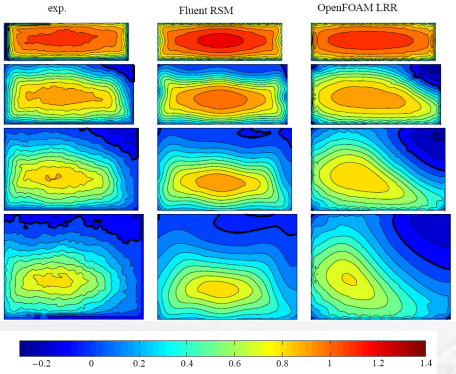


Figure 8: Mean streamwise velocities, u/u_{inlet}, at diffuser cross-sections various distances downstream from the origin using RSM and LRR. Contour lines are spaced 0.1 apart. The zero-streamwise-velocity contour line is thicker than others.

considerable differences between results of different turbulence models/codes !
(but: also differences in wall treatment, advection scheme, numerical parameters,…!)

No “Olympics” !

But burden to choose !

PIANC Setting the course

Turbulence modelling

Separated flow in 3D diffuser (A. Karvinen & H. Ahlstedt, 2008)

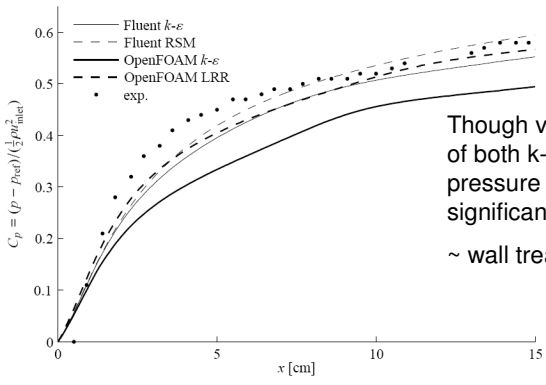


Figure 12: Pressure coefficient, C_p, along the bottom wall at the line $z = 1.665$ cm. p_{ref} is the pressure at $x = 0$ at the midpoint of the bottom flat wall (opposite the expanding wall).

Though velocity-contour figures of both k-ε results are similar, pressure coefficients differ significantly.

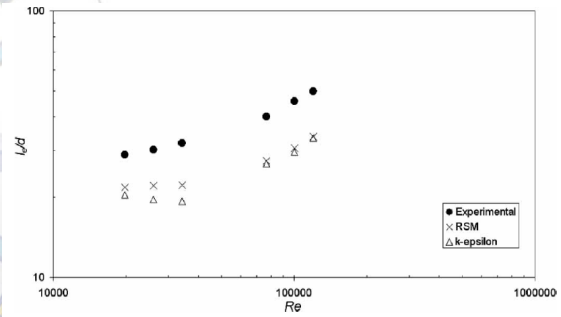
~ wall treatment (?)

Most validation efforts in literature: velocity instead of pressure!

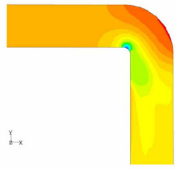
PIANC Setting the course

Turbulence modelling

90° elbow bends **(N. Crawford et al., 2009)**



Pressure drop of k–ε and RSM models expressed as an equivalent length (l_e/d), bend of $R/r = 1.3$



R = radius of curvature

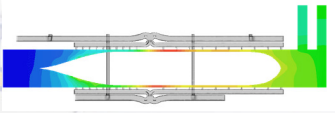
r = radius of circular cross section

discrepancy experimental vs. numerical:
- 24 to 33% for R/r = 1.3
- 7 to 9% for R/r = 5
- 1 to 5% for R/r = 2

(~ streamline curvature ; pressure gradients ; flow separation)

Turbulence modelling

The current body of knowledge arising from numerous **validation studies** does not provide unambiguous recommendations **(M.A. Leschziner, 2003)**

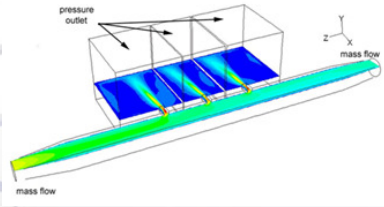


Even if it were the case…
one would need different models in
different parts of computational domain ?!

Many practical flows are highly 3D in which **inviscid pressure driven structures** occur and then turbulence stresses (hence turbulence modeling) become less important…

Boundary conditions

- where to limit computational domain ?



- which conditions to be imposed on boundaries ?

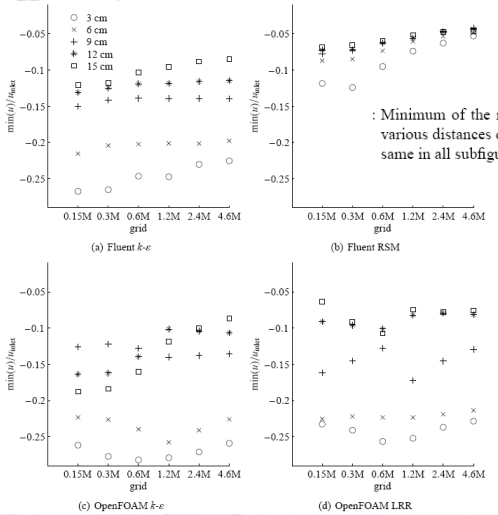
✓ wall treatment ?

✓ open boundaries ?

➔ burden for <u>both</u> RANS and LES !

Mesh resolution

Separated flow in 3D diffuser **(A. Karvinen & H. Ahlstedt, 2008)**



: Minimum of the mean streamwise velocity, $\min(u)/u_{\text{inlet}}$, at diffuser cross-sections various distances downstream of the origin using different grids. The legends are the same in all subfigures.

Grid dependency of results !?

Mesh resolution

Unstructured grids

e.g. **ADH** solver of USACE

incl. adaptive grid refinement (**R.L. Stockstill, 2009**)

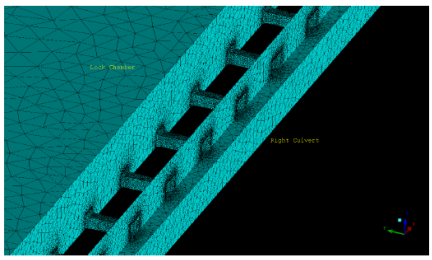


Figure 6. Surface mesh of right culvert and sidewall ports (culvert roof removed).

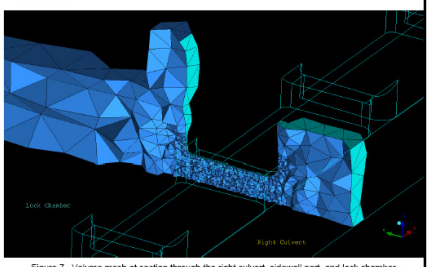


Figure 7. Volume mesh at section through the right culvert, sidewall port, and lock chamber.

Mesh quality

- Sudden transition from fine to large mesh cells (i.e. lack of smoothness) might degrade accuracy

- The *maximum angle condition* for two-dimensional triangulations states that

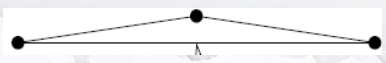
$$\exists \gamma_0 < \pi \quad \forall \mathcal{T}_h \in \mathcal{F} \quad \forall K \in \mathcal{T}_h : \quad \gamma_K \leq \gamma_0, \tag{4}$$

where γ_K is the maximal angle of K (compare with (2)).

The minimum angle condition (2) implies the maximum angle condition (4). To see this, denote the angles of a triangle $K \in \mathcal{T}_h$ by α, β, and γ and assume that $\alpha \leq \beta \leq \gamma$. If $\alpha \geq \alpha_0 > 0$ then

$$\gamma = \pi - \beta - \alpha \leq \pi - 2\alpha_0.$$

Too small and too large angles in 2D triangular Finite Element mesh might degrade accuracy

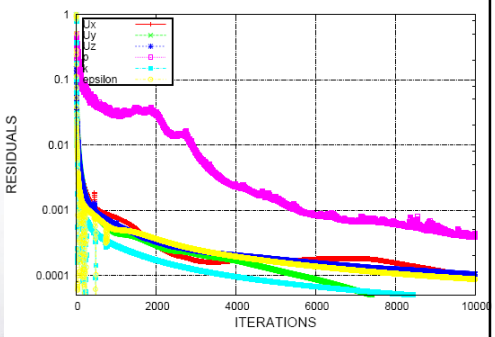


(3D ? ; Finite Volumes ?)

- **mesh optimization** (Laplacian smoothing, edge swapping,…) !?

Convergence

- Set of coupled, non-linear equations
- Iterative solution
- Convergence monitoring
  - ✓different rates for numerical and physical parameters
  - ✓slow convergence rate or convergence stall (~ turbulence models) (~ free surface models)



> "*One must watch the convergence of a numerical code as carefully as a father watching his four year old play near a busy road*"
>
> **J.P. Boyd** (Univ. Michigan)

Computer programme

- availability of source code ?
- willingness to modify code (e.g. turbulence model, boundary conditions,...) ?
- documentation (manual, validation,…) ?
- availability of options (input parameters) ?

CFD models: some important ingredients

- Velocity and pressure formulation
- Free surface modelling
- Turbulence modelling
- Boundary conditions
- Mesh generation and optimization
- Fluid-structure interaction
- Numerical diffusion and dispersion
- Convergence
- Computer programme
- Computer platform
- CFD practitioner
- …

Strongly coupled !

Quality of CFD results is function of all ingredients !

So is uncertainty !

CFD practitioner

Has an extremely important role:

- selection of **ingredients**
- **validation** of flow solver
- quantification of **uncertainty** (**UQ**)

Is a permanent (lifelong ?) task !

"Knowledge of the flow solver as well as the flow is the key to better prediction and successful innovation"

C.P. van Dam, paraphrasing A.B. Haines

"Understanding grows only logarithmically with the number of floating point operations."

J.P. Boyd

Use of CFD in design

Drag prediction in aeronautics

Critical ingredients for prediction up to 10^{-4} **(C.P. van Dam, 2003)**

- computer models that include all pertinent **geometry details**
- large meshes with high **grid resolution** in relevant areas
- well-developed numerical **solvers** (little **numerical viscosity**)
- fully-**converged** numerical solutions
- boundary-layer **transition** locations that match empirical ones
- **turbulence** models validated for flows encountered
- employing a drag evaluation methodology that is complete

Use of CFD in design

Aeronautical design

The emphasis on **simplicity**, **computational speed** and **robustness** in a design context militates against the adoption of more advanced models **(M.A. Leschziner, 2003)**

Before new methods and models can be accepted, questions concerning with **accuracy**, **reliability** and **efficiency** must be answered, in **absolute** terms and/or in **relative** terms, before they can be used with confidence **(K. Sermeus & H. Deconinck, 2003)**

Absolute truth cannot be provided…

…but relative truth is (!?)

Is this proven or is it merely a substitute for: *"Let's shut up and continue business as usual !"*

Use of CFD in design

Aeronautical design

Is **CFD** a mature technology ? **(K. Sermeus & H. Deconinck, 2003)**

- Not in the way that Computational structural mechanics (**CSM**) and Computer aided design (**CAD**) are
- CFD is not yet a component of an integrated **virtual prototype environment** used by aircraft design engineer

Lock design

More or less similar, but:

- (quasi) unique prototypes
- less strict demands for accuracy

CFD practitioner

Has an extremely important role

CFD client

Has an extremely important **role** too:

- needs to give sufficient **time** to CFD practitioner
- supplies (directly / indirectly) calibration / validation **data**
- should be **critical** and should not simply trust pretty pictures *("Colourful Fluid Dynamics")*

The way to proceed ?

Hybrid methodology (J.M. Hiver, 2009 ; S. Roux & N.Badano, 2011)

- numerical models with various number of spatial dimensions and physical complexity, including **CFD**

- physical models

The way to proceed ?

Numerical modelling

Physical modelling

In situ tests & enquiries

Design criteria

PIANC WG155 ?!

"*The parable of the blind*" ("*The blind leading the blind*")

Pieter Bruegel the Elder, 1568