Coupled Flow Solution Algorithms in OpenFOAM

Hrvoje Jasak

Faculty of Mechanical Engineering and Naval Architecture, Uni Zagreb, Croatia

Wikki Ltd. United Kingdom

TOBB ETU, Ankara, 23 October 2019

Outline

Objective

- Present the activities of the CFD Research group in Zagreb and Wikki Ltd.
- Present the details of the coupled implicit solver

Topics

- CFD Group at University of Zagreb: Research Activity
- Block matrix and block linear solvers
- Pressure-based coupled implicit solver
- Coupled turbulence model
- Validation examples
  - Simple canonical flows
  - Internal and external aerodynamics
  - Turbomachinery
- NUMAP-FOAM Summer School
- Summary

Overview of Research Activity

Research Group Members, CFD Group at University of Zagreb

- CFD Research Group attached to the **Chair of Turbomachinery**
- 2 professors: Prof. Hrvoje Jasak, Prof. Željko Tuković
- 1 (+ 1) post doctoral researchers: dr. Vuko Vukčević
- 6 fully funded PhD students
- Larger group of Master Thesis and Diploma Thesis students integrated within the group: results of Master Thesis projects directly used in further research
- Regular external (foreign) visitors working with the group: 3-6 months

Communication and Activity

- **Leading developer of OpenFOAM**: `http://foam-extend.fsb.hr/`
- Integrated work effort, modern communication and data integration tools
- Private Media-Wiki: `http://spirit.local/mediawiki/`
- YouTube Channel: `8th Floor CFD@FSB`
- Public web site: `http://www.fsb.hr/cfd`
- Approximately 25 (significant) publications per year

Overview of Research Activity

Open Source Software in Research and Engineering

- Open Source tools are ideal for a research environment: industrial partner gains access not only to physical model equations but also to a working and validated implementation by expert users
- Deployment of results of research is faster and more reliable
- Proven track record of model development and delivery
- First-class students with good technical and CFD background (OpenFOAM)
- Strong multi-disciplinary group: fluids, structure multi-phase flow modelling, turbulence, optimisation. Premier source of numerics knowledge in collaboration with leading world Universities

Overview of Research Activity

Areas of Research Activity, CFD Group at University of Zagreb

- **CFD simulations in Turbomachinery**: basic validation and verification and practical industrial simulations
  - Incompressible flow: pumps and turbines, wind energy devices
  - Compressible flow: compressors and fans with pressure- and density-based CFD solution algorithms
  - Harmonic balance modelling in CFD as a general-purpose tool
- Gradient-based and gradient-free optimisation: continuous and discrete adjoint
- **CFD in naval hydrodynamics, wave and off-shore structures**
- Fundamental research in numerics: discretisation, solution techniques, HPC performance, inter-equation coupling
- Complex solid mechanics modelling: large deformation, lubricated contact, conjugate heat transfer
- Fluid-solid interaction and "multi-physics" modelling
- Detailed fuel cell modelling using CFD
- Acoustics modelling using linearised Euler equations and coupling with CFD

Block Matrix in OpenFOAM

Background

- OpenFOAM uses equation mimicking to perform field algebra and discretisation: **perfect for simple PDE-s or segregated solution algorithms**
- . . . but sometimes we use equation segregation inappropriately
- There exists a family of problems that cannot be solved efficiently without inter-equation coupling; some simulations “that work” can be performed 10-s or 100-s of times faster than with equivalent segregated algorithms

Objective

- Implement flexible and efficient block-coupled solution infrastructure
- Re-use all operator-based discretisation schemes, parallelisation and boundary condition tools already available in OpenFOAM
- Optimise top-level code for efficient execution and ease of assembly

Examples

- Incompressible steady pressure-velocity system (with turbulence)
- Compressible multi-phase free surface simulations: under-water explosions

Block Matrix in OpenFOAM

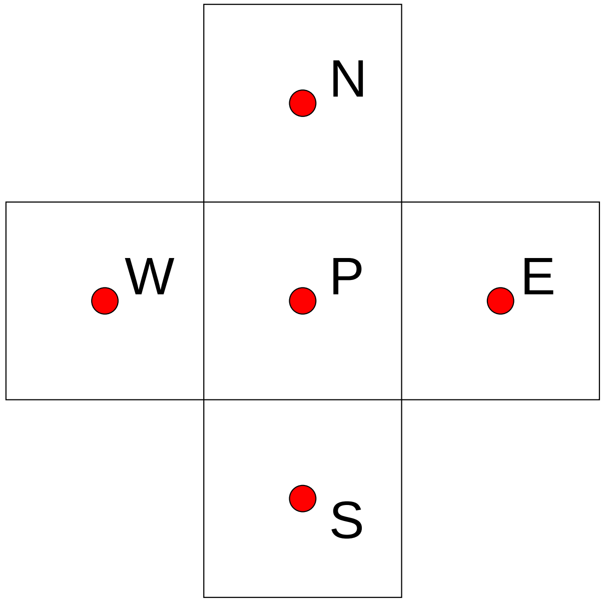
Block-Coupled Solution Algorithms

- For cases of strong coupling between the components of a vector, the components can be solved as a **block variable**: (u_x, u_y, u_z) will appear as variables in the same linear system
- In spite of the fact that the system is much larger, the coupling pattern still exists: components of $\mathbf{u}$ in cell P may be coupled to other components in the same point or to vector components in the neighbouring cell
- With this in mind, we can still keep the sparse addressing defined by the mesh: if a variable is a vector, a tensorial diagonal coefficients couples the vector components in the same cell. A tensorial off-diagonal coefficient couples the components of $\mathbf{u}_P$ to all components of $\mathbf{u}_N$, which covers all possibilities
- For **multi-variable block solution** like the compressible Navier-Stokes system above, the same trick is used: the cell variable consists of $(\rho, \rho\mathbf{u}, \rho E)$ and the coupling can be coupled by a 5×5 matrix coefficient
- Important disadvantages of a block coupled system are
  - Large linear system: several variables are handled together
  - Different kinds of physics can be present, *e.g.* the transport-dominated momentum equation and elliptic pressure equation. At matrix level, it is impossible to separate them, which makes the system more difficult to solve

Block Matrix in OpenFOAM

Matrix Connectivity and Mesh Structure

- Irrespective of the level of coupling, the FVM dictates that a cell value will depend only on values in surrounding cells



- We still have freedom to organise the matrix by ordering entries for various components of the solution variable x
- Global sparseness pattern related to mesh connectivity: easier coefficient assembly

Block Matrix in OpenFOAM

Coupling Coefficient

- Matrix implemented with **block coefficients**
- Consider general linear dependence between two vectors $\mathbf{m}$ and $\mathbf{n}$

$$\mathbf{m} = \mathbf{A}\,\mathbf{b}$$

- Component-wise coupling describes the case where m_x depends only on n_x, m_y on n_y and m_z on n_z
  1. Scalar component-wise coupling
  2. Vector component-wise coupling
  3. Full (block) coupling
- Explicit methods do not feature here because it is not necessary to express them in terms of matrix coefficients
- For reference, the linear equation for each cell featuring in the matrix reads

$$\mathbf{A}_P \mathbf{m}_P + \sum_N \mathbf{A}_N \mathbf{m}_N = \mathbf{R}$$

Coupled p-U Solver

Turbulent Steady Incompressible Flows: SIMPLE or Coupled System

- Equation set contains linear p-U and non-linear U-U coupling

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla\bullet(\mathbf{u}\mathbf{u}) - \nabla\bullet(\nu\nabla\mathbf{u}) = -\nabla p$$

$$\nabla\bullet\mathbf{u} = 0$$

- Traditionally, this equation set is solved using the **segregated SIMPLE algorithm**
  - Low memory peak: solution + single scalar matrix in peak storage
  - p-U coupling is handled explicitly: loss of convergence (under-relaxation)
  - Number of iterations is substantial; not only due to non-linearity
  - Convergence dependent on mesh size: SIMPLE slows down on large meshes
- **Block-implicit p-U coupled solution**
  - Coupled solution significantly increases matrix size: 4 blocks instead of 1
  - ... but the linear p-U coupling is fully implicit!
  - Iteration sequence only needed to handle the non-linearity in the U-equation
  - Net result: **significant convergence improvement** (steady or transient) at a cost of increase in memory usage: **reasonable performance compromise!**

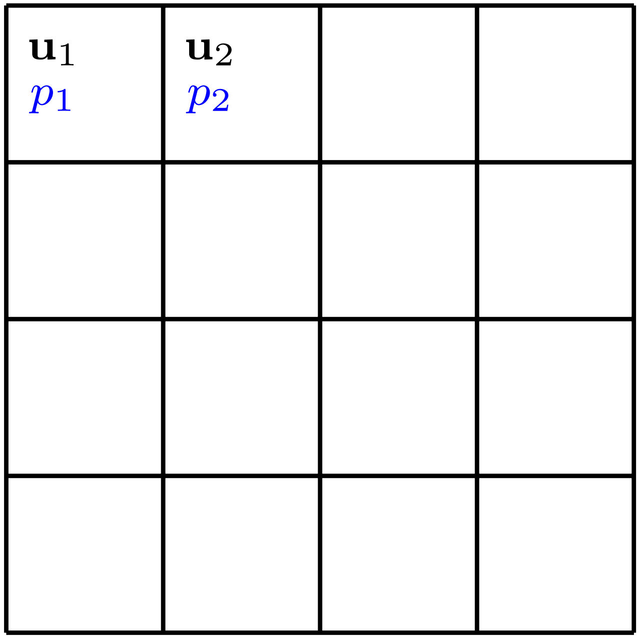
Coupled p-U Solver

SIMPLE-Based Segregated p-U Solver

```
// Momentum equation assembly and solution
fvVectorMatrix UEqn
(
    fvm::div(phi, U)
  + turbulence->divDevReff()
);
UEqn.relax();
solve(UEqn == -fvc::grad(p));

// Pressure equation assembly and solution
U = UEqn().H()/UEqn.A();
phi = fvc::interpolate(U) & mesh.Sf();
fvScalarMatrix pEqn
(
    fvm::laplacian(1/UEqn.A(), p) == fvc::div(phi)
);
pEqn.solve();
phi -= pEqn.flux();
p.relax();
```

Block-Coupled $\mathbf{u} - p$ System Matrix Structure



$$\begin{bmatrix} \begin{pmatrix} a_{P\,(\mathbf{u}\,\mathbf{u})} & a_{P\,(\mathbf{u}\,p)} \\ a_{P\,(p\,\mathbf{u})} & a_{P\,(p\,p)} \end{pmatrix} & \begin{pmatrix} a_{N\,(\mathbf{u}\,\mathbf{u})} & a_{N\,(\mathbf{u}\,p)} \\ a_{N\,(p\,\mathbf{u})} & a_{N\,(p\,p)} \end{pmatrix} & \cdots \\ . & \begin{pmatrix} a_{P\,(\mathbf{u}\,\mathbf{u})} & a_{P\,(\mathbf{u}\,p)} \\ a_{P\,(p\,\mathbf{u})} & a_{P\,(p\,p)} \end{pmatrix} & \cdots \\ \vdots & \vdots & \ddots \end{bmatrix} \begin{bmatrix} \mathbf{u}_1 \\ p_1 \\ \mathbf{u}_2 \\ p_2 \\ \vdots \end{bmatrix} = \begin{bmatrix} b_{\mathbf{u}\,1} \\ b_{p\,1} \\ b_{\mathbf{u}\,2} \\ b_{p\,2} \\ \vdots \end{bmatrix}$$

Coupled p-U Solver

Coupled Implicit p-U Solver: Source Code

```
fvVectorMatrix UEqn
(
    fvm::div(phi, U)
  + turbulence->divDevReff()
);
fvScalarMatrix pEqn
(
   - fvm::laplacian(rUAf, p) == -fvc::div(fvc::grad(p))
);
BlockLduSystem<vector, vector> pInU(fvm::grad(p));
BlockLduSystem<vector, scalar> UInp(fvm::UDiv(U));

BlockLduMatrix<vector4> A(mesh);
blockMatrixTools::insertEquation(0, UEqn);
blockMatrixTools::insertEquation(3, pEqn);
blockMatrixTools::insertBlockCoupling(3, 0, UInp, false);
blockMatrixTools::insertBlockCoupling(0, 3, pInU, true);

UpEqn.solve();
UpEqn.retrieveSolution(0, U.internalField());
UpEqn.retrieveSolution(3, p.internalField());
```

Performance of the Coupled p-U Solver

Performance Improvements and Extensions in the Coupled p-U Solver

- Improvements in performance for the coupled solver: consistency, numerics
- Developed a procedure for analysis of inter-equation coupling
- Extension to **implicit** MRF and porous media
- **Block-coupled $k - \epsilon$ and $k - \omega$ SST turbulence models**
  - Turbulence equations solved in a single block-coupled system
  - Analysis of source terms to establish favourable cross-equation coupling
  - Implemented in Diploma Thesis assignment: Robert Keser, Uni Zagreb

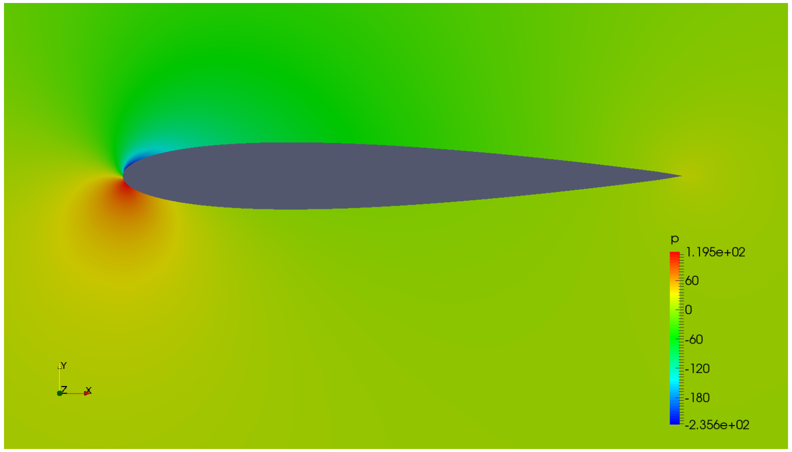
Block Algebraic Multigrid

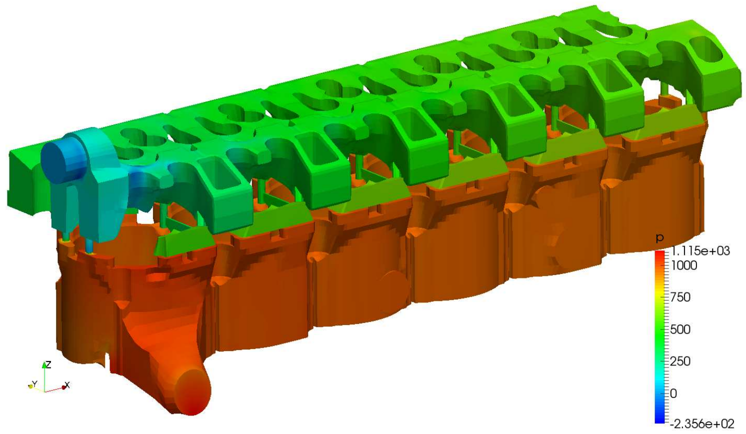
Block Version of Selective Algebraic Multigrid

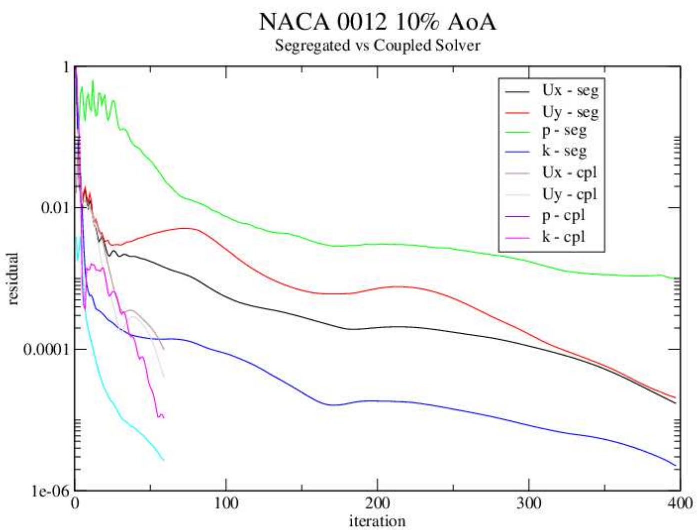
- Major performance jump: block-coupled AMG with Selective Coarsening
- The algorithm follows the principles of the classical SAMG (Stüben), but uses a primary matrix (Clees) for coarsening and calculation of interpolation
- Additionally, using additive correction (Hutchinson 1988) for solution acceleration
- Algorithm is extended to non-M-matrices and block coefficients
- New smoother based on Crout's lower-upper factorisation
- Parallelisation with the in-depth matrix fetch across coupled interfaces
- Support for non-trivial coupling: GGI interface, mixing plane

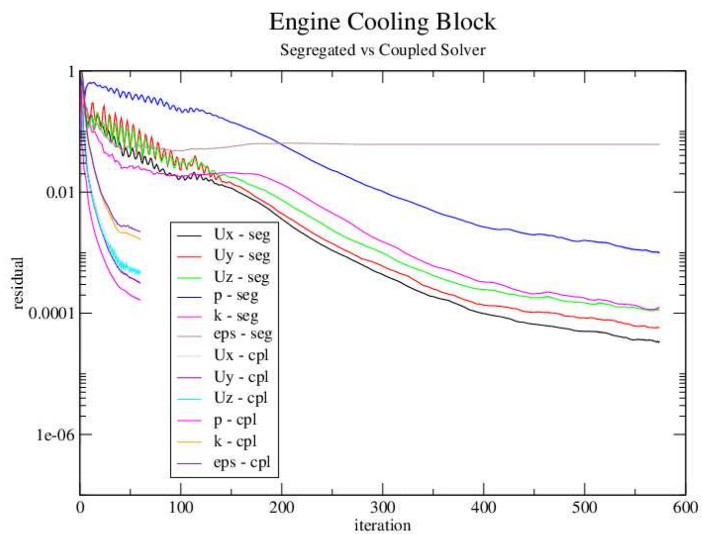
Performance of the Coupled p-U Solver

Performance of the Coupled p-U Solver: Speed and Robustness



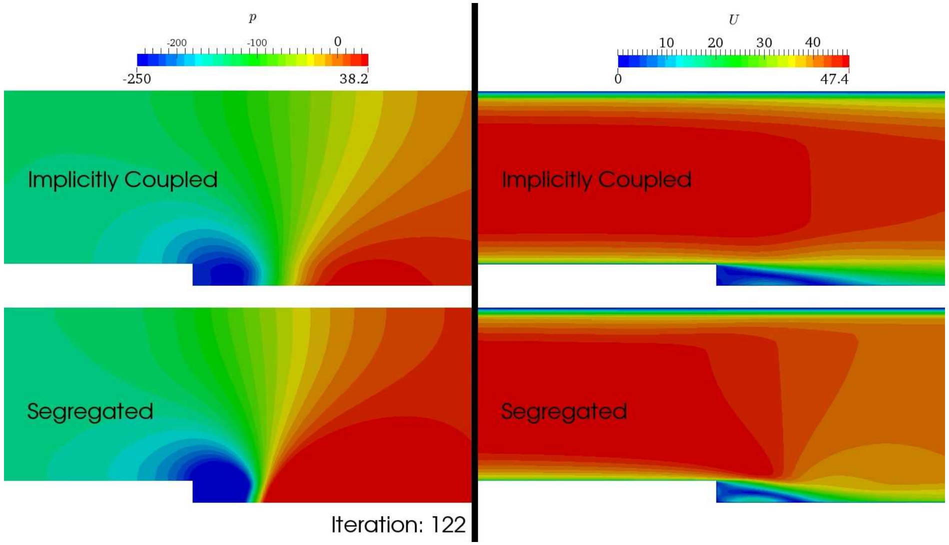


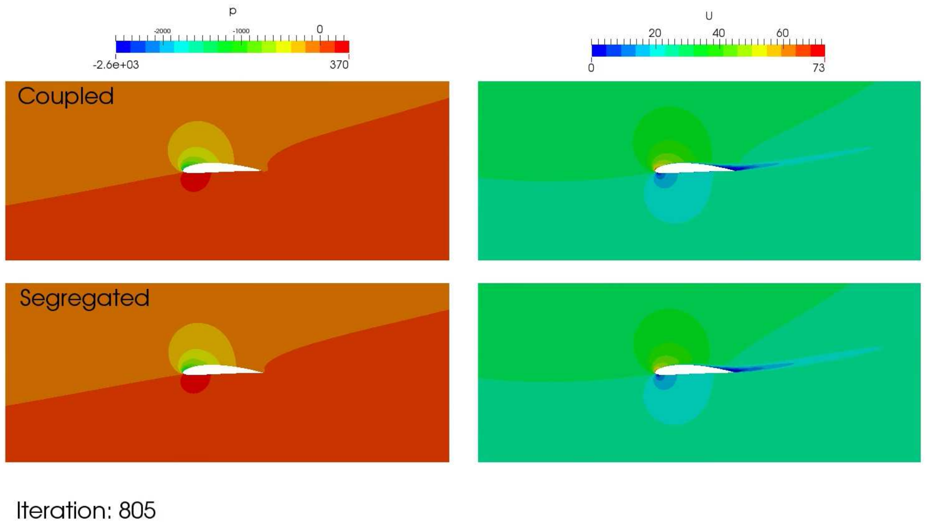




Performance of the Coupled p-U Solver

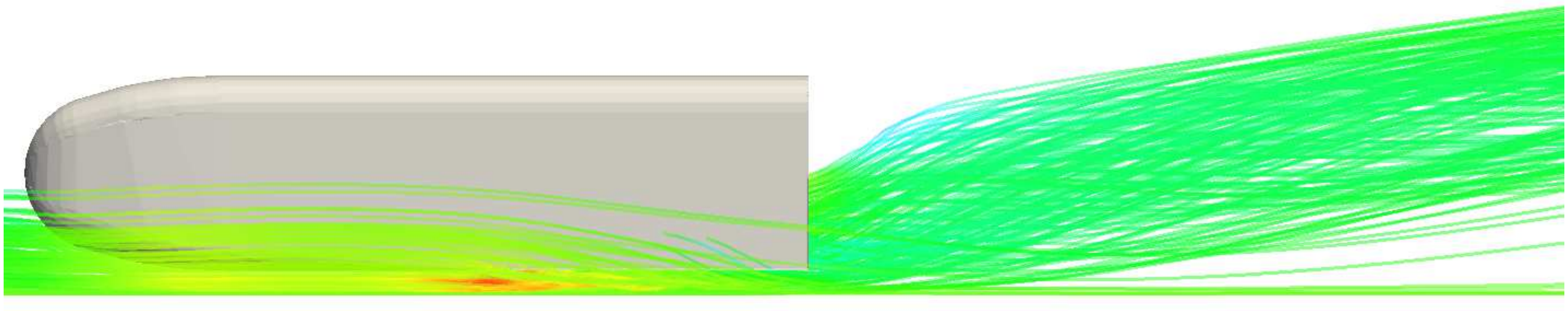
Performance of the Coupled p-U Solver: Speed and Robustness

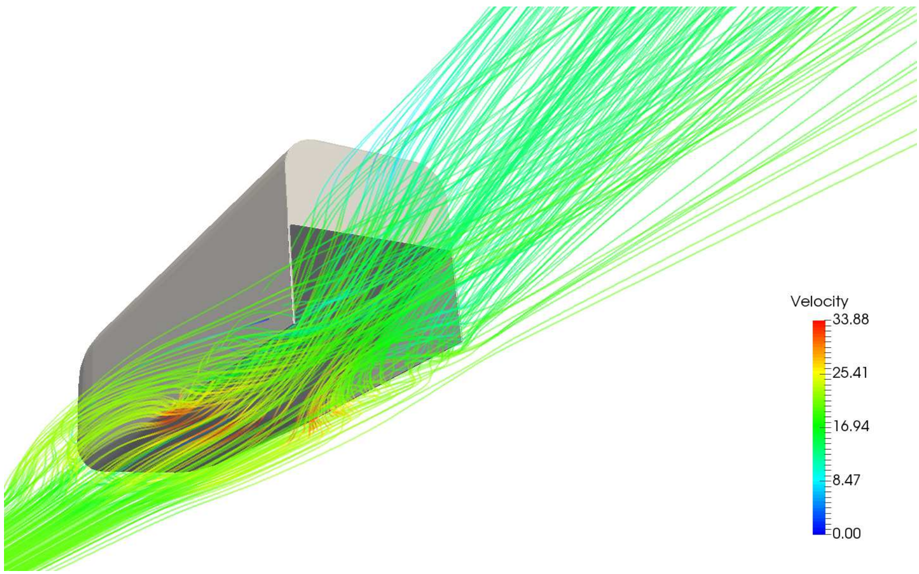


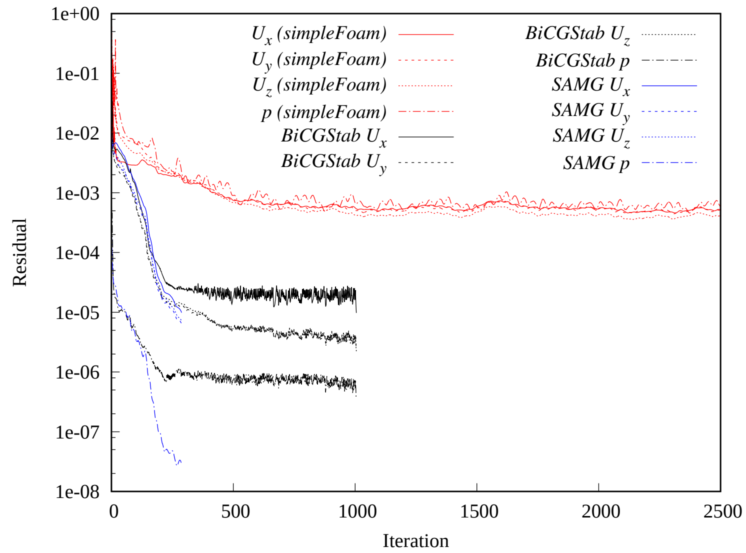


Performance of the Coupled p-U Solver

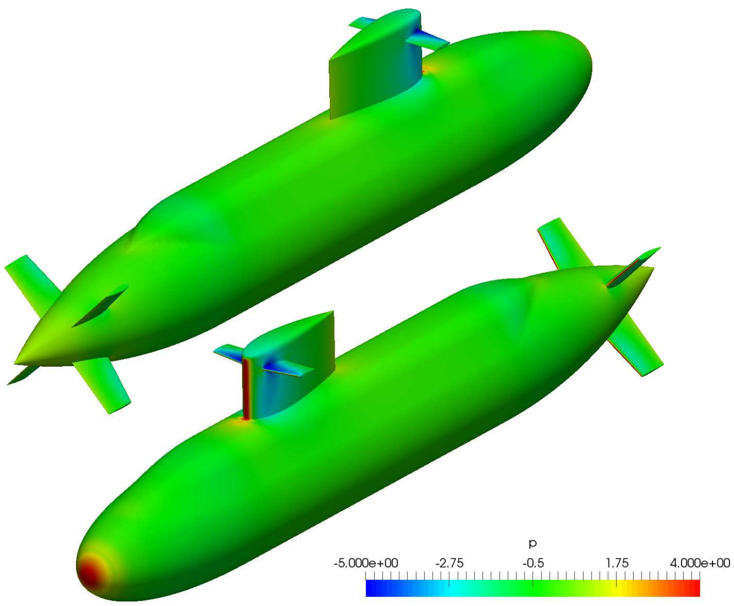
Performance of the Coupled p-U Solver: External Aerodynamics

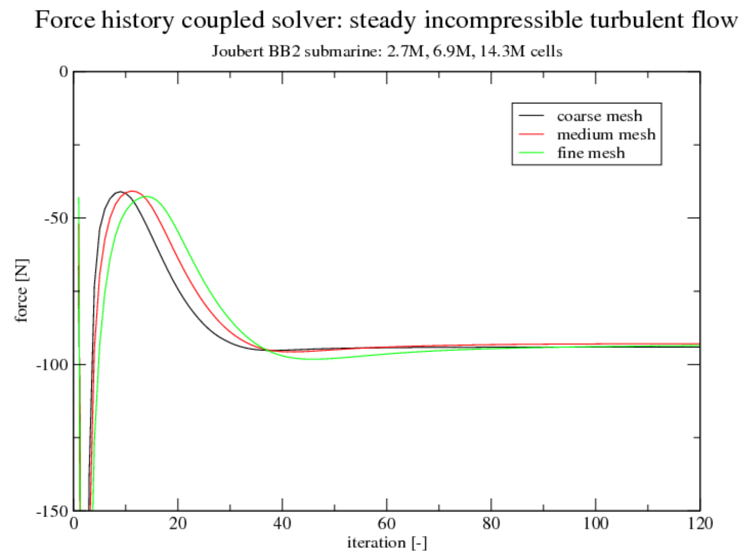


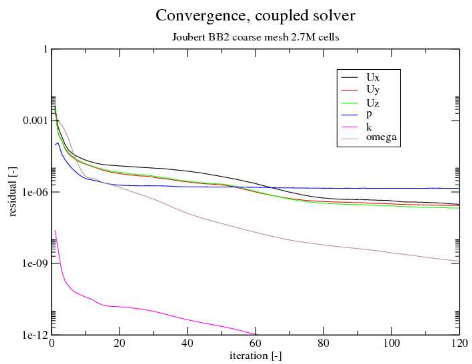


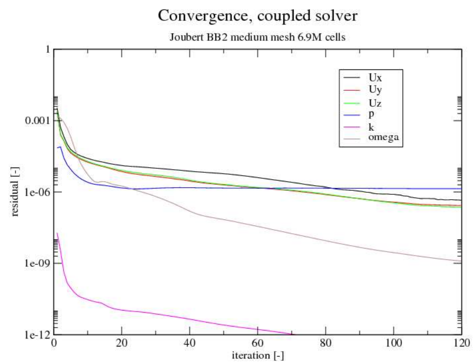


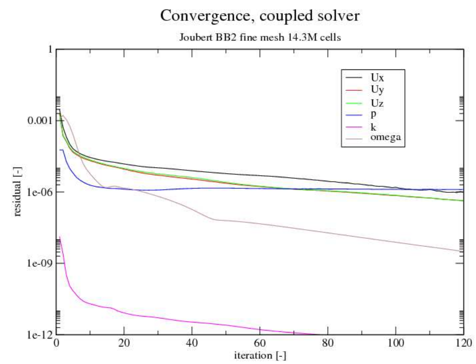
Performance of the Coupled p-U Solver: Submarine Flight, 14M Cells





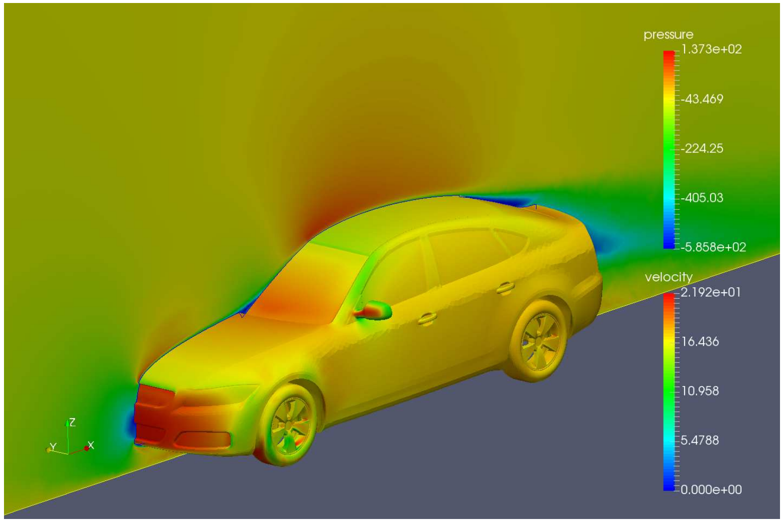


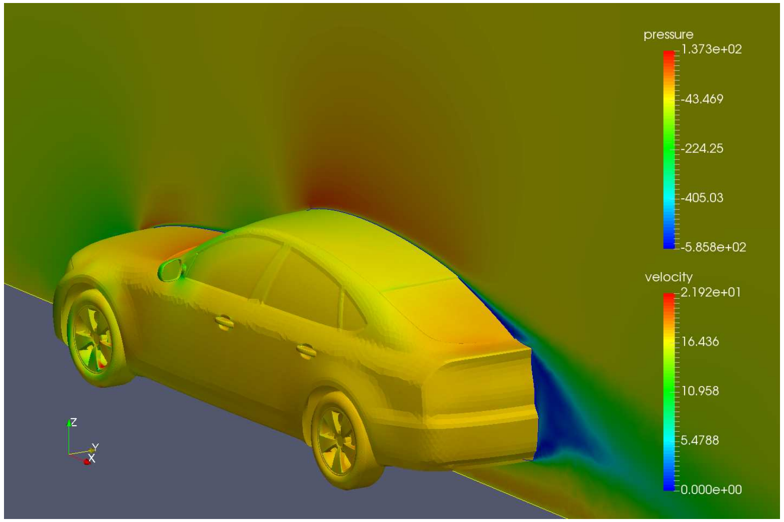


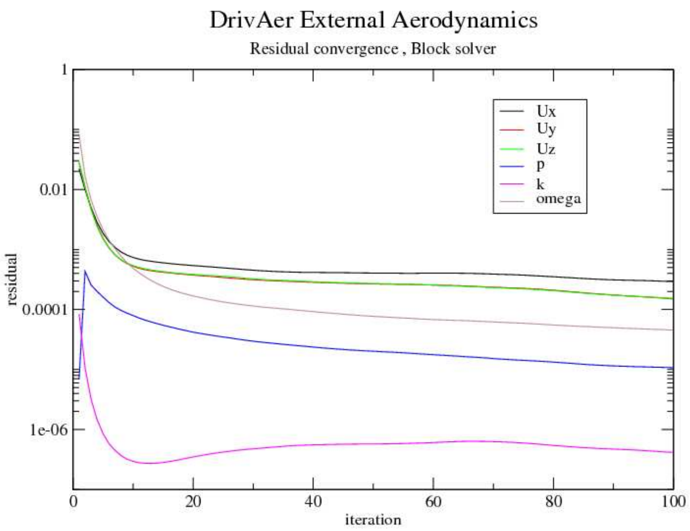


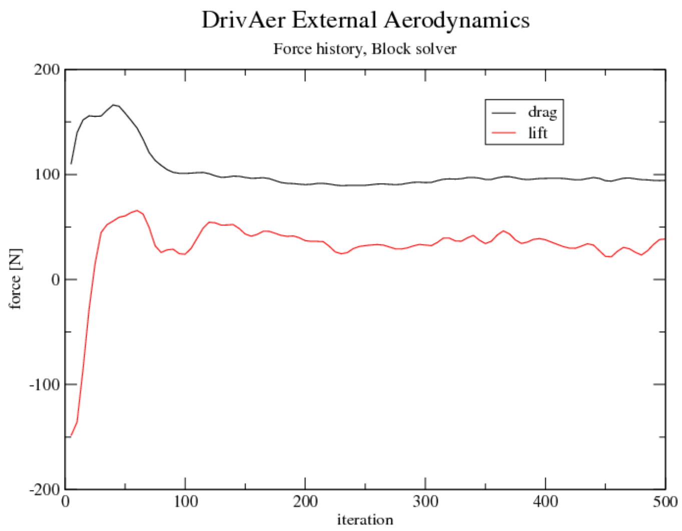
External Aerodynamics Simulations

DrivAer Geometry: External Aerodynamics, Coupled Solver, 13.2M Cells



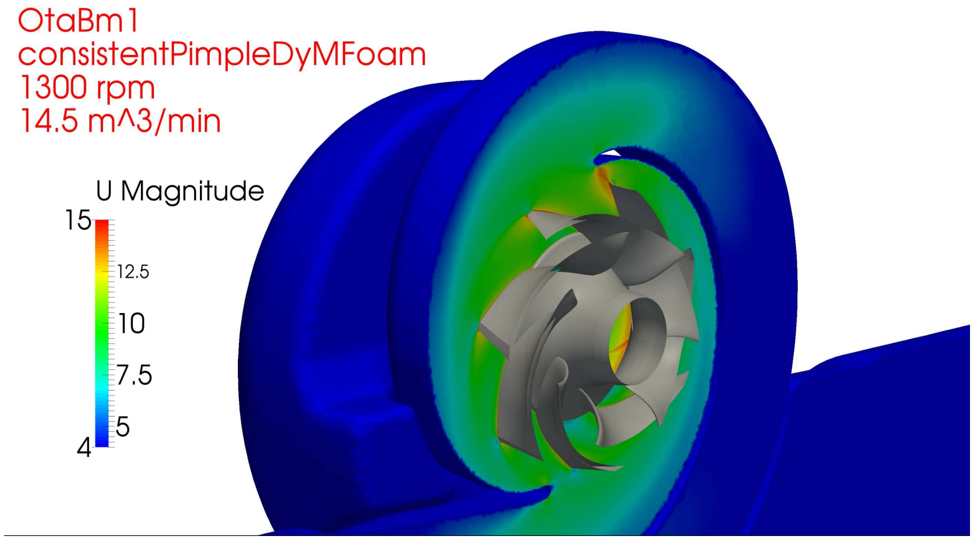


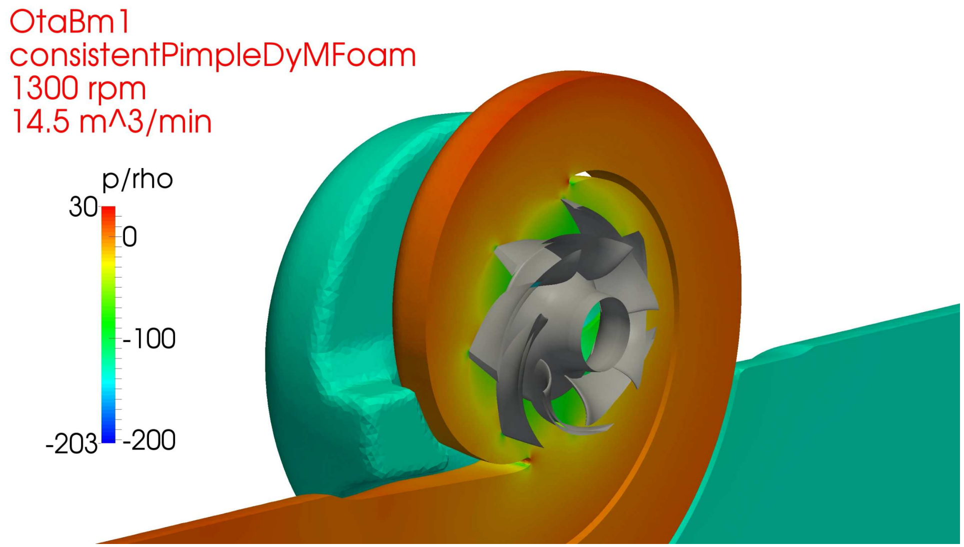




Performance of the Coupled p-U Solver

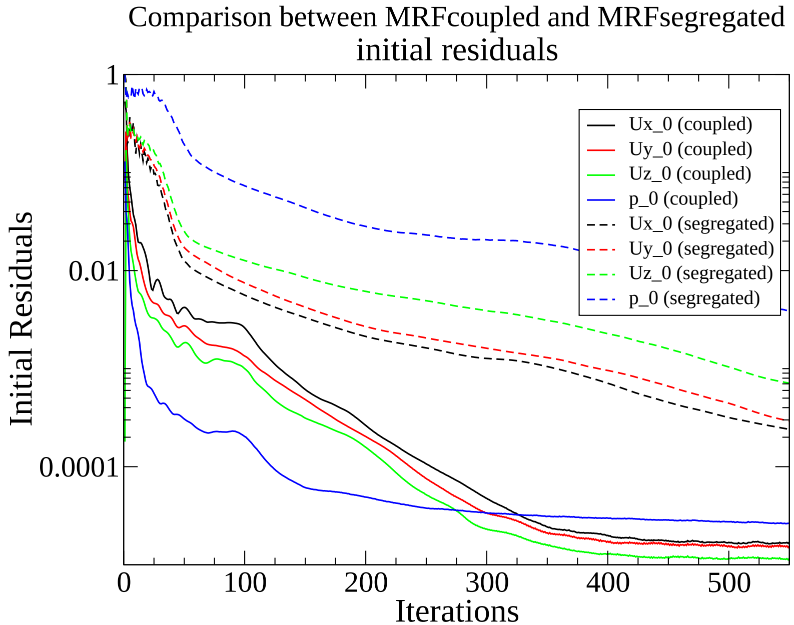
Turbomachinery: OTA-BM-1 Pump, Frozen Rotor MRF, 9M Cells

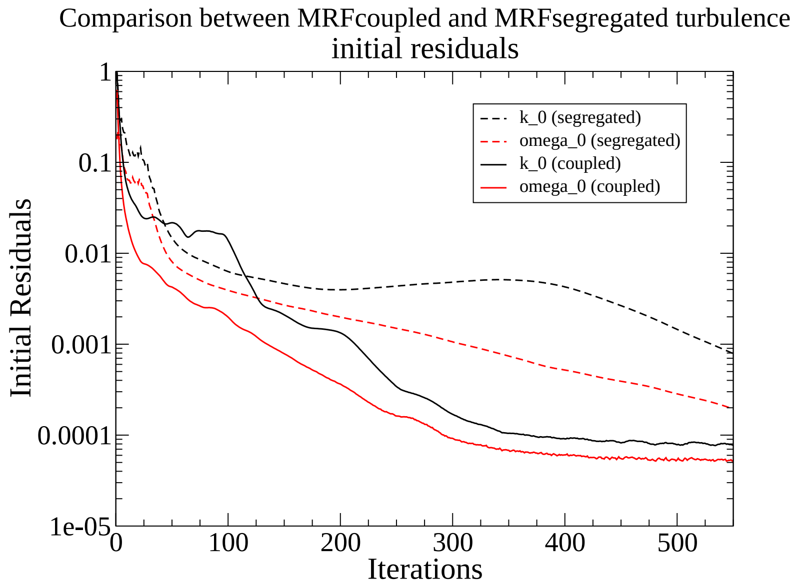


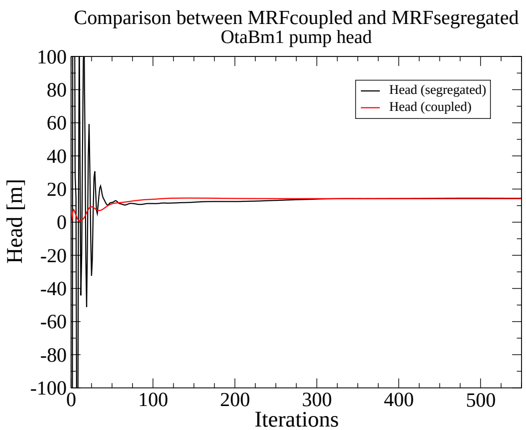


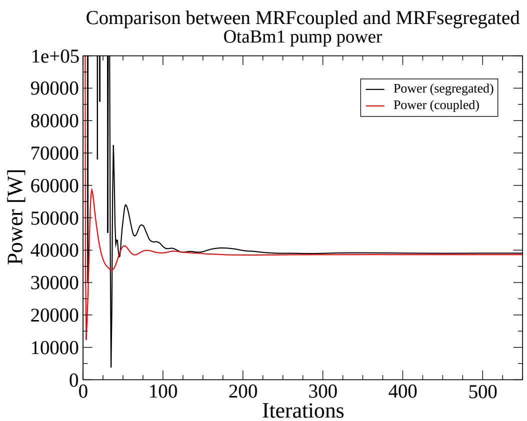
Performance of the Coupled p-U Solver

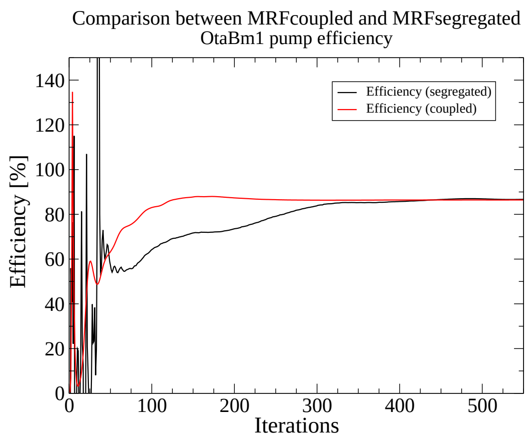
Turbomachinery: OTA-BM-1 Pump, Frozen Rotor MRF, 9M Cells











Water Jet

Water Jet Propulsor: Flow Conditions

- Six-bladed rotor, at 2000 rpm; eight-bladed stator
- Turbulent flow with steady inlet condition, $u = 11.43\,\mathrm{m/s}$
- No experimental data available: real water jet cavitates at this flow rate

Mesh Layout

- Full annulus with resolved blade tip clearance: 2,153,424 hexahedral cells
- Two domains: rotor and stator connected using a GGI interface

Frozen Rotor MRF Simulation: Coupled Solver

- Rapid and smooth convergence in 150 iterations: 4 hours on a laptop computer

Transient Simulation

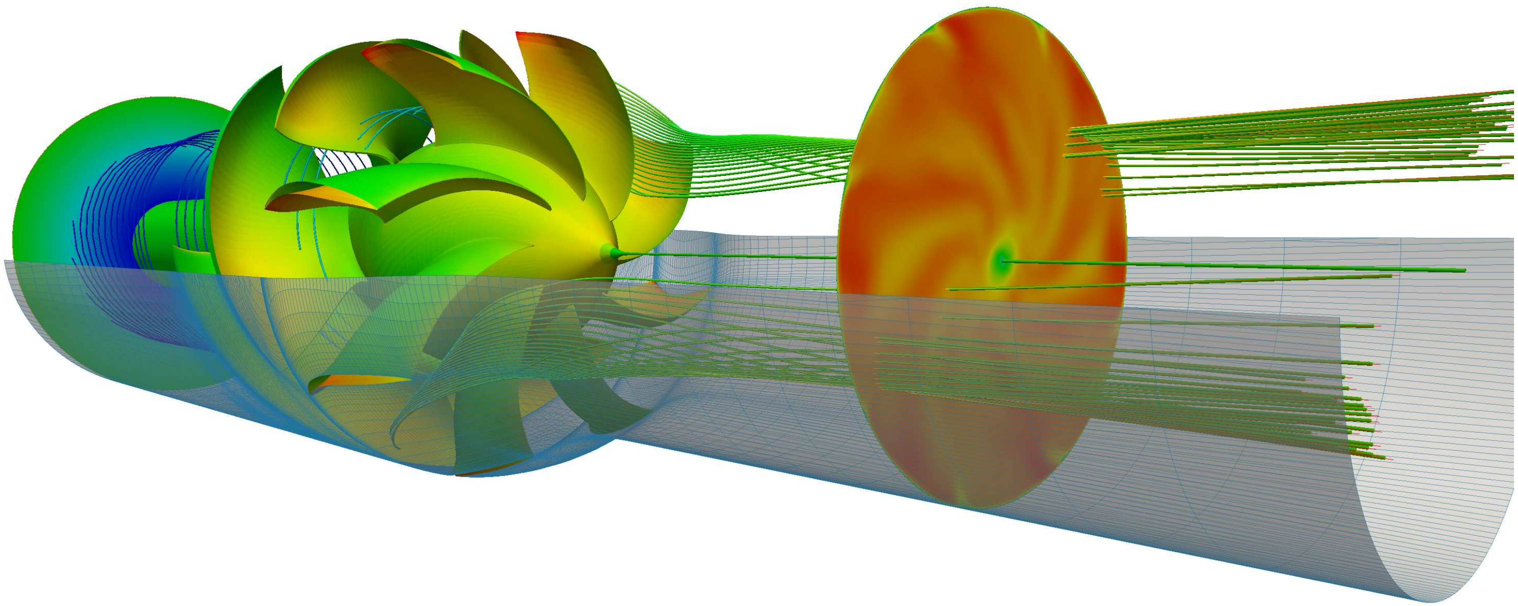
- Transient simulation completely impractical due to small mesh size at tip clearance with large velocities
- Typical $\Delta t = 1e - 07\,\mathrm{s}$; time for 1 period = $0.03\,\mathrm{s}$
- Transient run ongoing for 4 weeks on a workstation (small mesh)

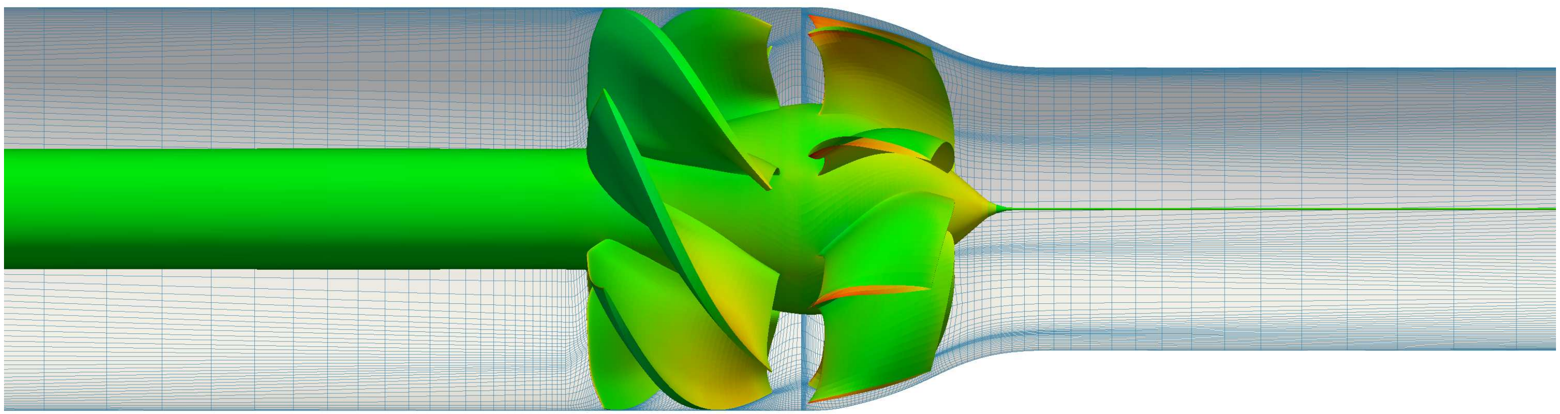
Harmonic Balance Simulation

- Performing harmonic balance simulations with 1, 2 and 7 harmonics

Water Jet

Water Jet Propulsor

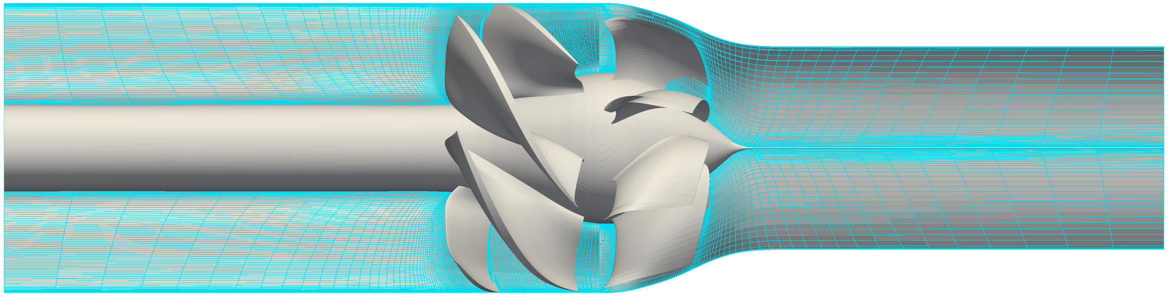


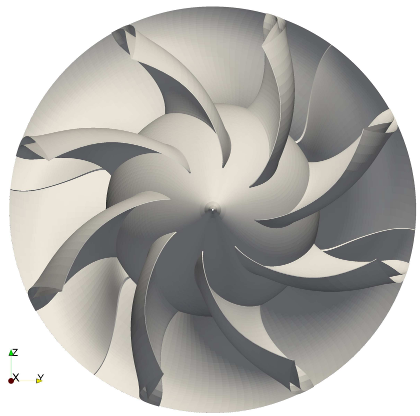


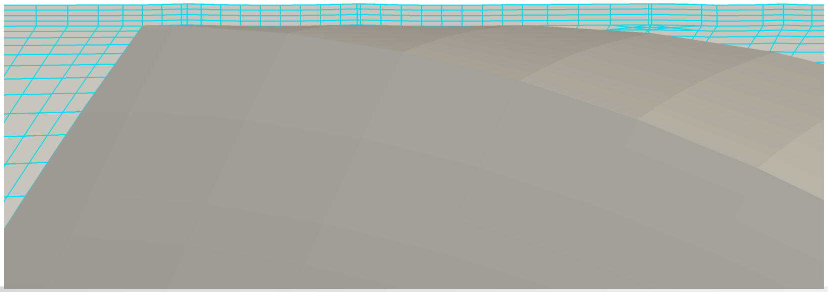
WIKKI

Harmonic Balance: Water Jet

Water Jet Propulsor: Geometry and Mesh

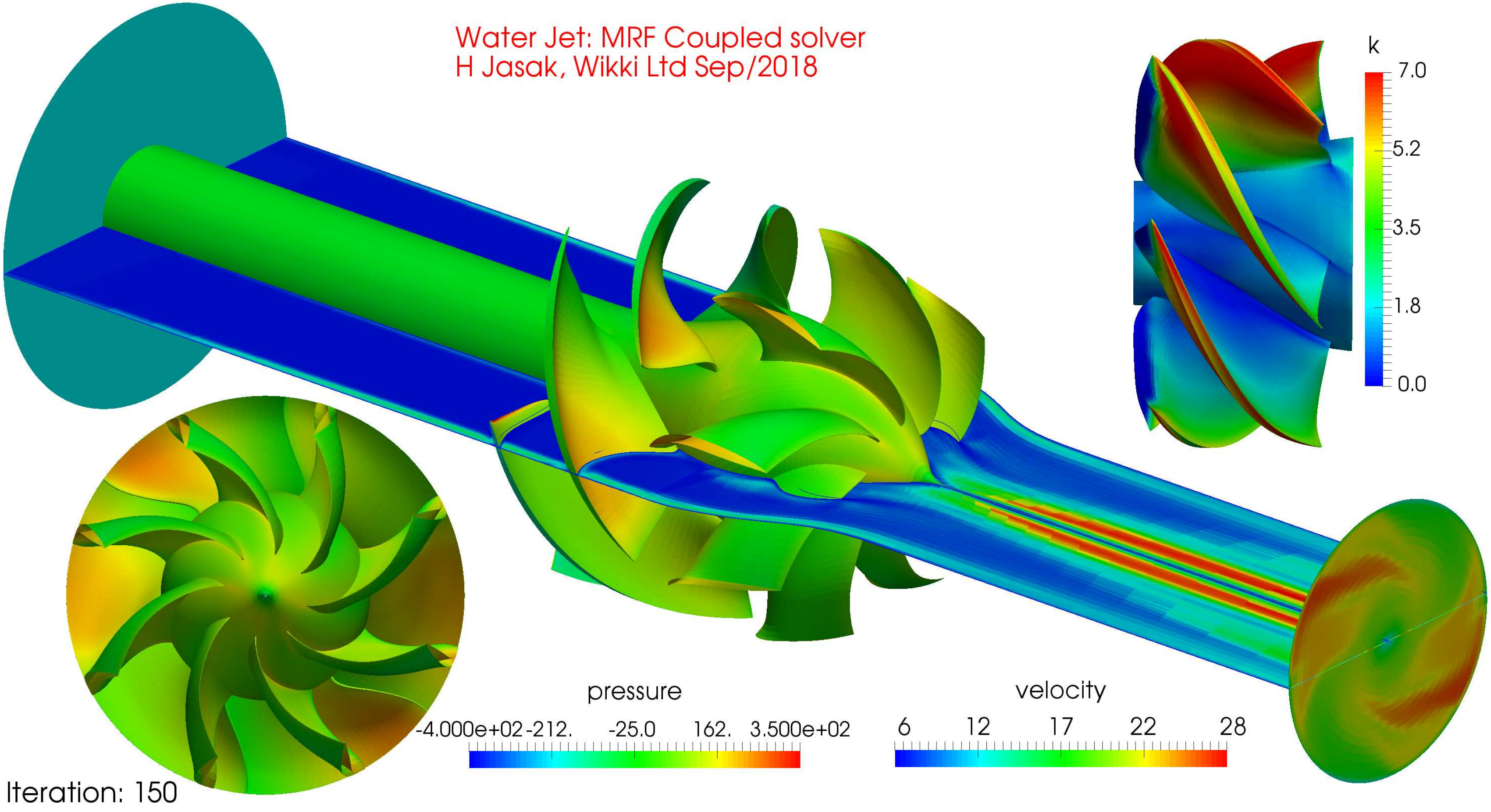






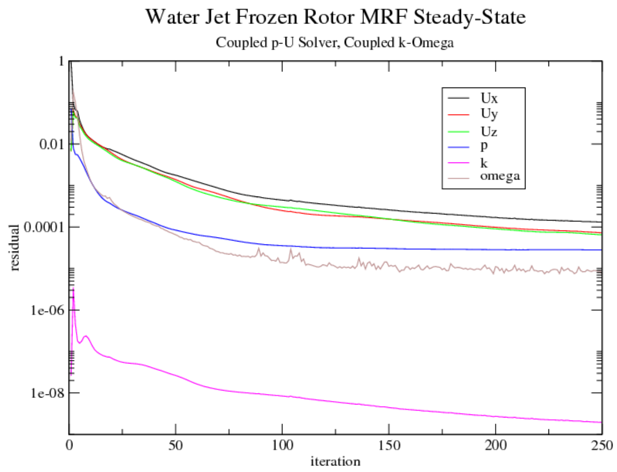
Harmonic Balance: Water Jet

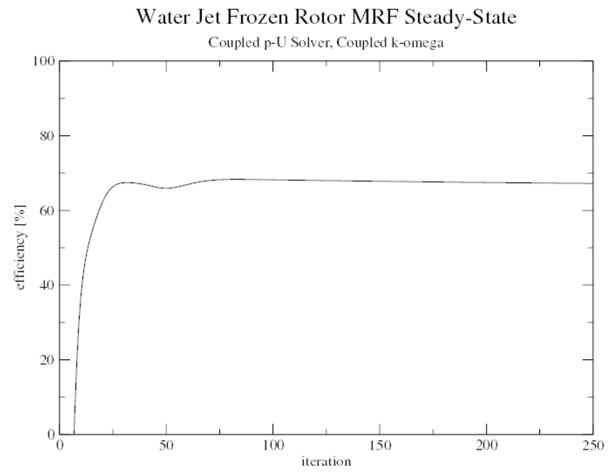
Steady-State Frozen Rotor, MRF Solution, Coupled Solver: Convergence History

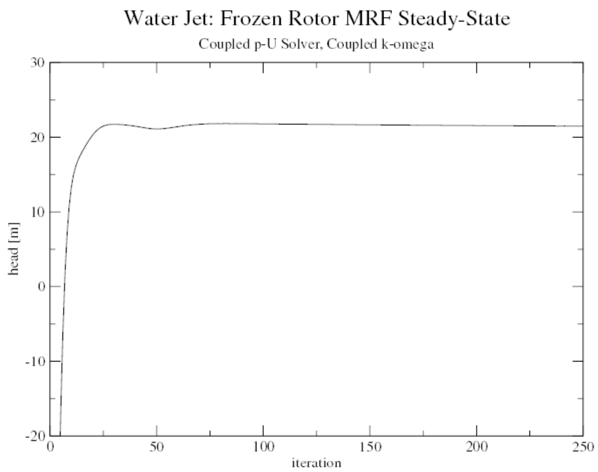


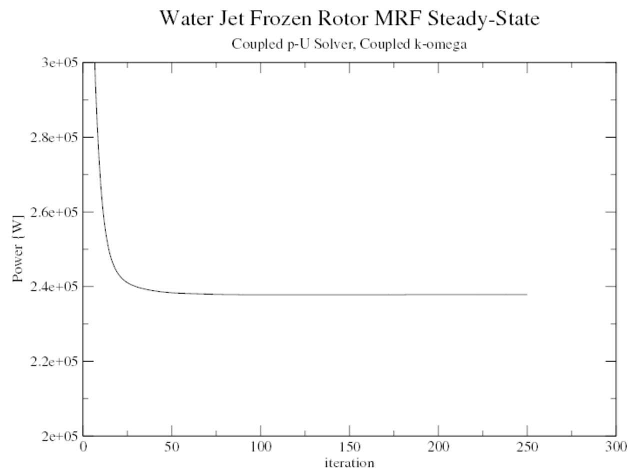
Harmonic Balance: Water Jet

Steady-State Frozen Rotor, MRF Solution, Coupled Solver



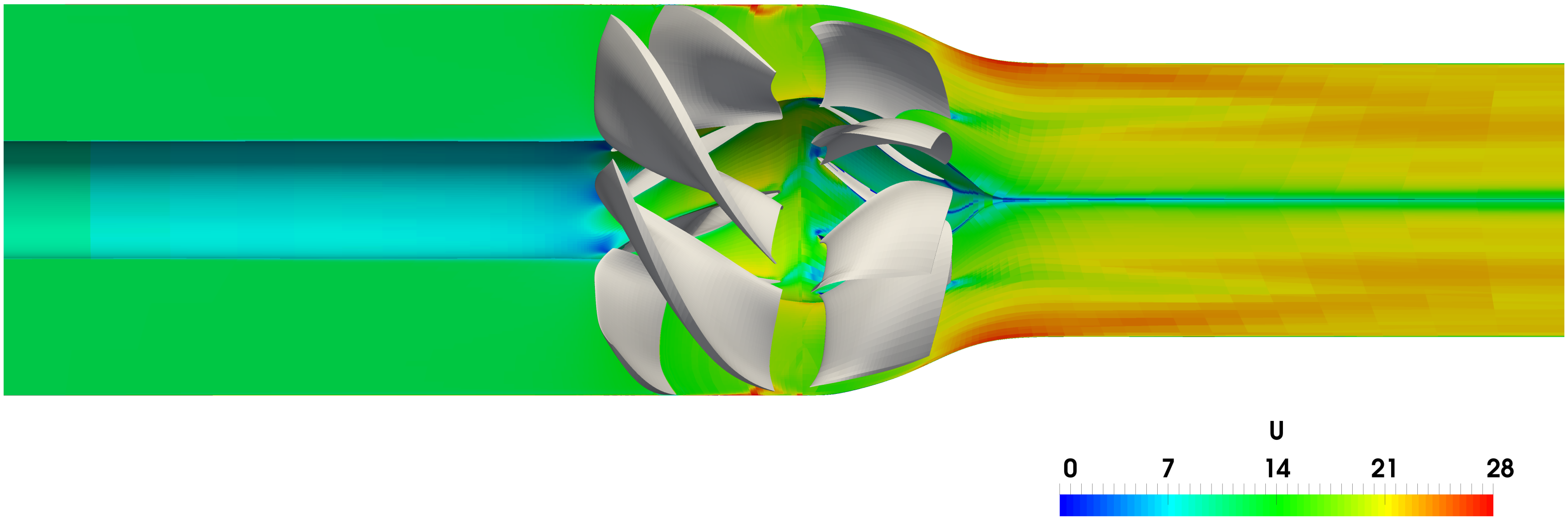


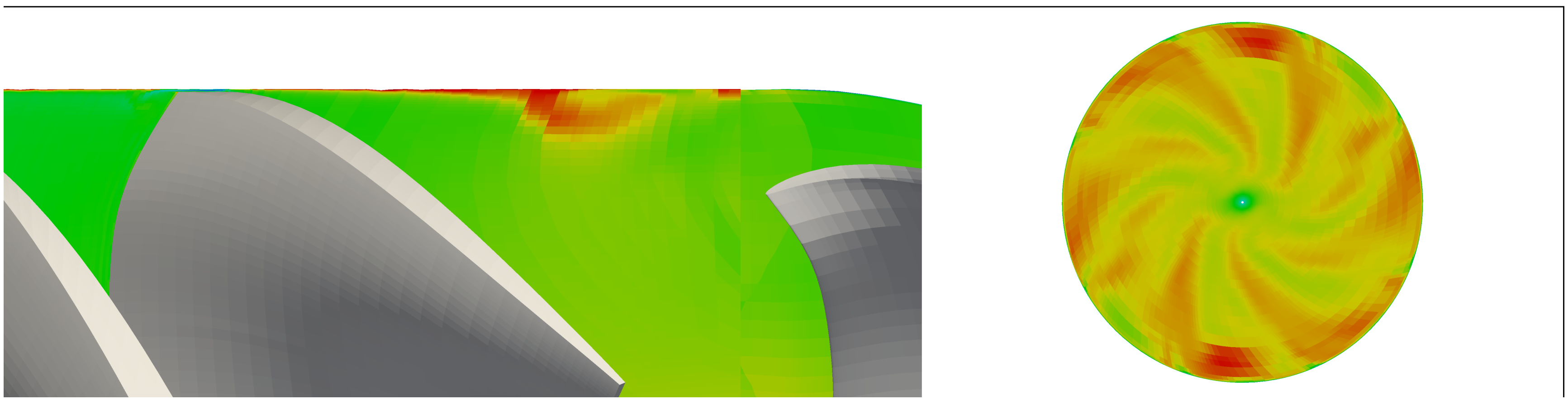




Harmonic Balance: Water Jet

Harmonic Balance for a Water Jet Propulsor

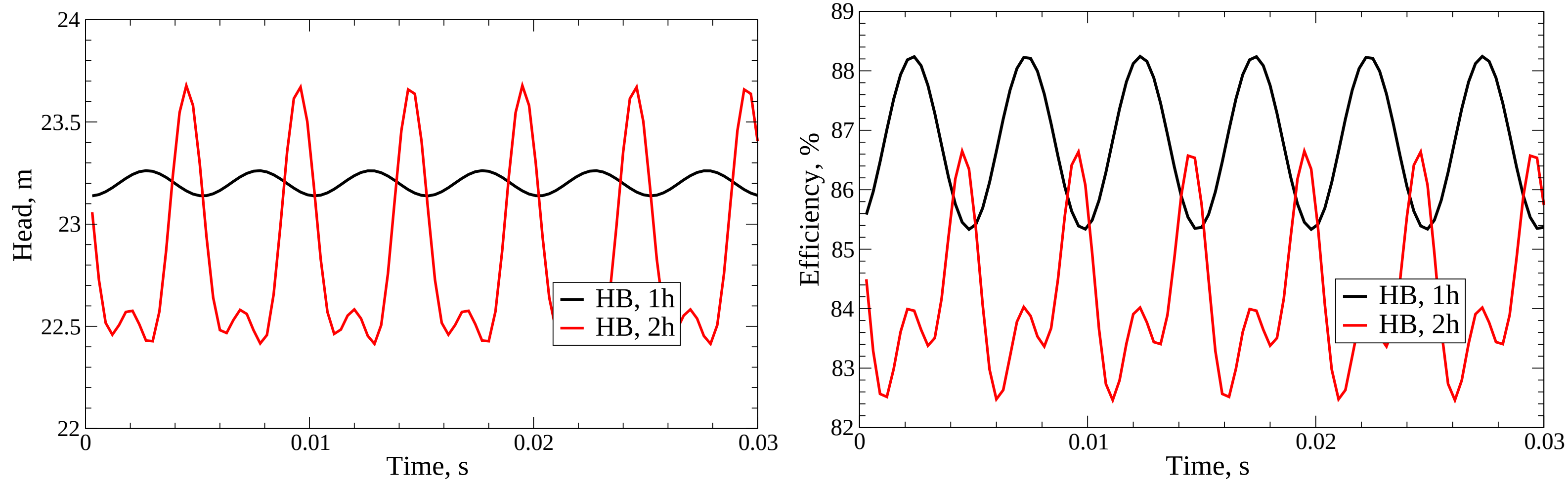




Harmonic Balance: Water Jet

Harmonic Balance for a Water Jet Propulsor

- Temporal variation of head and efficiency: 1 and 2 harmonics



Water Jet: Future Work

- Further validation & verification work ongoing
- It is possible to extend the HB model to cavitating flow

NUMAP-FOAM Summer School

NUMAP-FOAM Summer School 2019

- 13th Edition of NUMAP-FOAM Summer School: 19-30/Aug/2019
  `https://www.fsb.unizg.hr/numap`

  The idea of the Summer School is to expand the physical modelling knowledge, numerics and programming skills of attendees using OpenFOAM in their research through direct supervision and one-to-one work.

  This is NOT an introductory OpenFOAM course: significant understanding of the project and software is a pre-requisite for application.

- The School accepts 15 attendees bringing their own projects to the School over a period of 10 working days
- Work is embedded in the research group with 4–6 tutors providing daily one-to-one attention
- School is open to "young researchers" (typically PhD students) but also to industrial users, government labs and professors
- Strong follow-up collaboration and extensive publication lists
- Approx 170 attendees to NUMAP-FOAM, from the start in 2008

Summary

Summary

- Ongoing research activity at Uni Zagreb on naval hydrodynamics, basic numerics and turbomachinery CFD
- Actively looking for collaboration partners

Current Work Topics

- **Naval hydrodynamics**: added resistance in regular and irregular waves, full-scale ship simulation, self-propulsion and manoeuvring, green water and freak wave impact, modelling of irregular sea states
- **Numerics**: strongly coupled solution algorithms, Discontinuous Galerkin discretisation, Overset Mesh and Immersed Boundary
- **Turbomachinery**: quasi-periodic methods (harmonic balance), LES and instability modelling, implicit pressure- and density-based solvers, turbulence and transition
- **Solid mechanics and FSI**: coupled non-linear FSI problems
- **Optimisation**: Gradient-free and adjoint-based methods; uncertainty propagation and robust design

About Me

Hrvoje Jasak

- First degree: mechanical engineering, University of Zagreb, Croatia 1992
- PhD, Imperial College London 1993-1996: The birth of FOAM
- Senior development engineer, CD-adapco (Siemens), 1996-2000
- Technical director, Nabla Ltd. 2000-2004
- Consultant on CFD software, numerics and modelling, ANSYS Fluent 2000-2008

Current Work

- Director, Wikki Ltd: UK-based consultancy company 2004-
- Professor, University of Zagreb, Croatia 2007-
- Mercator Fellow, TU Darmstadt, 2016-
- Various software development and commercial support projects based on OpenFOAM with consultants and large industrial partners
- Coordinating open source OpenFOAM development to allow contributions from the public domain developers
- OpenFOAM workshops, lectures and seminars, visiting professorships (TU Delft, Chalmers University and others)