CFDIMPACT Conference June 30, 2015
Book of Abstracts

TECHNION Israel Institute of Technology

sgi

RAFAEL ADVANCED DEFENSE SYSTEMS LTD.

Program
CFD IMPACT Conference
June 30, 2015

CFD IMPACT CONFERENCE

08:15-09:00	Gathering and Refreshments
09:00-10:00	**Opening Session**
09:00-09:10	Opening Remarks
09:10-10:00	**Keynote Lecture:**
	Next Generation Computational Fluid Dynamics: High-Order Methods and Many-Core Hardware
	Peter E. Vincent, Department of Aeronautics, Imperial College, London, UK
10:00-10:20	Coffee Break
10:20-11:40	**1st Session - High-Order Numerics and LES Applications**
10:20-10:40	*High-Order Large Eddy Simulation using Multiblock Immersed Boundary Method for Rotorcraft Aerodynamics*
	Yann Delorme, Faculty of Mechanical Engineering, Technion, Haifa, Israel
10:40-11:00	*High-Fidelity LES/PDF Method for Turbulent Flames*
	Steven (Chaim) Frankel, Faculty of Mechanical Engineering, Technion, Haifa, Israel
11:00-11:20	*Simulation of Fluidic Oscillator as Applied for Flow Control*
	Eran Arad, Rafael, Israel
11:20-11:40	*The Impact of Large Scale Flow Structures on Turbulent Spray Flames*
	Yuval Dagan, Rafael, Israel
11:40-12:00	Coffee Break
12:00-13:20	**2nd Session - RANS and Hybrid RANS/LES Applications**
12:00-12:20	*Turbulent Flow Simulations on Unstructured Grids using Reynolds Stress Models*
	Y. Mor-Yossef, Israeli CFD Center, Caesarea, Israel
12:20-12:40	*Symmetry Breaking and Hysteresis of the Averaged Flow-Field in Stirred Tank Reactors with Radial Impellers*
	R. Ben-Nun and **M. Sheintuch**, Department of Chemical Engineering, Technion, Haifa, Israel
12:40-13:00	*RANS Modeling for Stratified Combustion*
	Steven (Chaim) Frankel, Faculty of Mechanical Engineering, Technion, Haifa, Israel
13:00-13:20	*Computational Surgery Towards Powered Fontan Hemodynamics*
	Yann Delorme, Faculty of Mechanical Engineering, Technion, Haifa, Israel
13:20-14:20	Lunch Break
14:20-15:40	**3rd Session - Numerical and Experimental Methods**
14:20-14:40	*Direct Linear Solvers for Lower Order Methods in Incompressible CFD*
	A. Yu. Gelfgat, School of Mechanical Engineering, Tel Aviv University, Tel Aviv, Israel
14:40-15:00	*Convergence Acceleration for Multiphysics Compressible Flow*
	O. Peles and E. Turkel, School of Mathematical Sciences, Tel Aviv University, Tel Aviv, Israel
15:00-15:20	*Development of a Compressible Navier-Stokes Solver Using Immersed Boundary Method and Adaptive Mesh Refinement*
	Kunal Puri, Faculty of Mechanical Engineering, Technion, Haifa, Israel
15:20-15:40	*Targeted Experiments for CFD Validation of Separated and Unsteady Turbulent Flows*
	David Greenblatt, Faculty of Mechanical Engineering, Technion, Haifa, Israel
15:40-16:00	**Final Remarks, Discussion, Tour of CFDLAB, and Adjourn**

For more information & registration please visit our website: www.cfdimpact.org

Next-Generation Computational Fluid Dynamics: High-Order Methods and Many-Core Hardware

P. E. Vincent

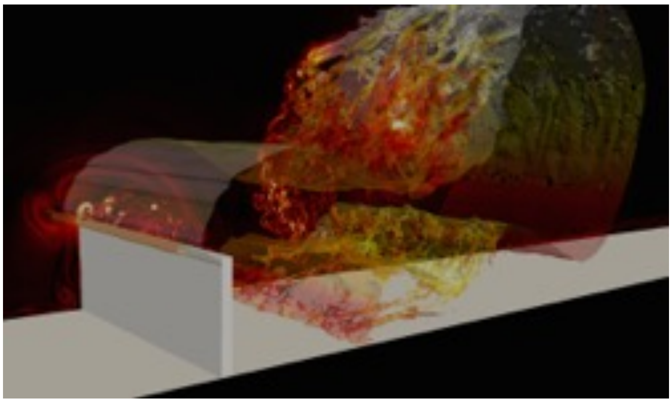
Department of Aeronautics, Imperial College London, UK

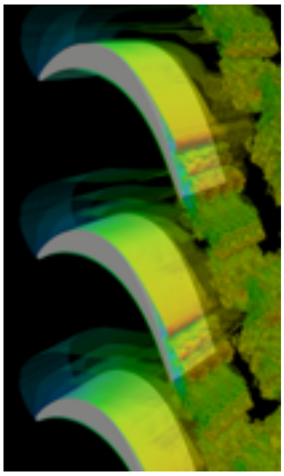
High-order numerical methods for unstructured grids combine the superior accuracy of high-order spectral or finite difference methods with the geometrical flexibility of low-order finite volume or finite element schemes. The Flux Reconstruction (FR) approach [1] unifies various high-order schemes for unstructured grids within a single framework. Additionally, the FR approach exhibits a significant degree of element locality, and is thus able to run efficiently on modern many-core hardware platforms, such as Graphical Processing Units (GPUs). The aforementioned properties of FR mean it offers a promising route to performing affordable, and hence industrially relevant, scale-resolving simulations of hitherto intractable unsteady flows within the vicinity of real-world engineering geometries. In this talk I will present PyFR (www.pyfr.org) [2], an open-source Python based framework for solving advection-diffusion type problems using the FR approach. The framework is designed to solve a range of governing systems on mixed unstructured grids containing various element types. It is also designed to target a range of hardware platforms via use of a custom Mako-derived domain specific language. The latest release of PyFR is able to solve the compressible Euler and Navier-Stokes equations on grids of quadrilateral and triangular elements in two dimensions, and hexahedral, tetrahedral, prismatic, and pyramidal elements in three dimensions, targeting clusters of multi-core CPUs, NVIDIA GPUs (K20, K40 etc.), AMD GPUs (S10000, W9100 etc.), and heterogeneous mixtures thereof. Results will be presented for various benchmark and 'real-world' flow problems, and scalability of PyFR will be demonstrated on clusters with 100s of NVIDIA GPUs. Throughout the talk the importance of algorithm-software-hardware co-design, in the context of next-generation computational fluid dynamics, will be highlighted.

References

[1] Huynh, H. T., *A Flux Reconstruction Approach to High-Order Schemes Including Discontinuous Galerkin Methods*, AIAA Paper 2007–4079, 2007.

[2] Witherden, F. D., Farrington A. M., Vincent P. E., *PyFR: An Open Source Framework for Solving Advection-Diffusion Type Problems on Streaming Architectures using the Flux Reconstruction Approach*, Computer Physics Communications, 185(11) pp. 3028–3040, 2014.





High-Order LES using Multiblock Immersed Boundary Method for Rotorcraft Aerodynamics

Yann Delorme
CFDLAB, Faculty of Mechanical Engineering,
Technion - Israel Institute of Technology, Haifa, Israel

Rotorcraft aerodyanmics involves complex turbulent high Reynolds number transonic flows featuring strong vorticity, rotating geometry and significant wake-body interactions. As a result, such flows are very challenging to simulate. Unsteady RANS CFD approaches are generally too dissipative, inaccurate, and very computational expensive due to the need for large (>100M) grid points for 3-4 revolutions of the vortex system. LES based CFD generally benefits from the use of high-fidelity (high-order and high-resolution) numerical methods but use of such methods with complex rotating geometric is challenging and such computations are prohibitively expensive for high Reynolds numbers. State-of-the-art codes, such as the US Army's Helios, employ dual grid/codes using body-fitted structured or unstructured near-body grids to handle the rotor geometry and structured Cartesian off-body grids allowing for high-order finite-different numerical methods. Complex overset grid technology is needed and generally a detached eddy simulation is used with a URANS one-equation model near-the body and inviscid Euler equations for the off-body calculations. Such simulations have been observed to capture the complex vortical structures associated with what appears to be rotor-wake instabilities. In this presentation we repent an alternative complementary approach based on fixed structured Cartesian grids and a novel multi block immersed boundary method (IBM) for LES of rotorcraft hover. Comparisons between the IBM results and Helios for the same problem with comparisons to measured data will be presented and discussed with regard to both accuracy and efficiency.

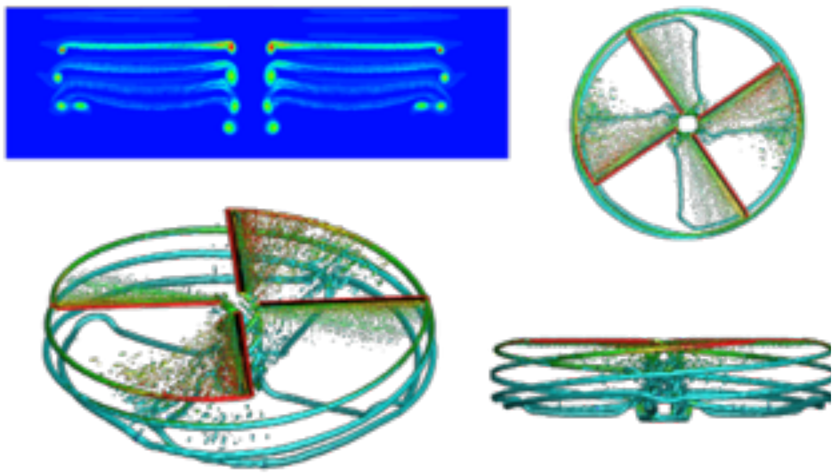


Figure 1. A snapshot showing vortical structures from LES of rotorcraft hover.

High-Fidelity LES/PDF Method for Turbulent Flames

Steven H. Frankel
CFDLAB, Faculty of Mechanical Engineering,
Technion - Israel Institute of Technology, Haifa, Israel

The focus of current turbulent combustion research is driven by the desire to design more efficient low-emission stable combustors and engines. To facilitate this, and to complement experimental studies, accurate and efficient methods for modelling and simulating turbulent flames and combustors are needed. In particular, these models should be able to capture flame dynamic events, such as ignition, extinction, lift-off, flashback, and possible instability by accurately modelling turbulent-chemistry interactions across flame regimes. While RANS is still widely use in industry, large eddy simulation (LES) is rapidly developing as a powerful tool to study combustion dynamics. The use of LES in conjunction with high-order numerical methods has the potential to strike the right accuracy versus efficiency balance. All that is needed is a subgrid-scale (SGS) combustion model that is universally accurate and applicable across combustion flame regimes from premixed to non-premixed and in between. One of the most promising regime-independent SGS combustion models in this regard are probability density function (PDF) methods. In this talk, we highlight the development and application of an in-house combustion code that employs LES to solve the low-Mach number form of the incompressible Navier-Stokes equations. A combination of high-order WENO scheme and central finite-difference methods are used to spatially discretize the governing equations on a structured Cartesian grid. The stochastic Eulerian composition PDF method is implemented as the SGS combustion model. The model was previously tested for a non-premixed hydrogen-air jet flame (see Fig. 1 below). In this talk, a reduced finite-rate chemistry scheme for methane-air combustion will be used. Preliminary results for the partially-premixed turbulent jet flame known as Sandia Flame D will be compared to measured data. A previously published direct closure SGS combustion model with the potential for significant computational savings is also implemented and will be tested for the first time.

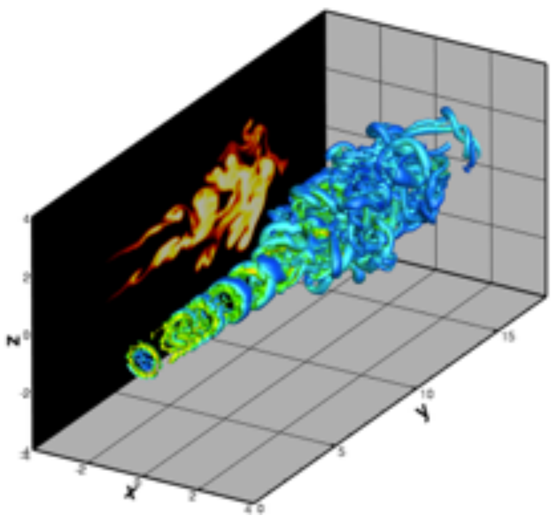


Figure 1. A snapshot showing vortical structures and flame contours from an LES/PDF of a hydrogen-air non-premixed turbulent jet flame.

Simulation of Fluidic Oscillator as Applied for Flow Control

Eran Arad, Aeronautical Systems, RAFAEL

Active flow control devices and applications received a lot of attention in the recent three decades. While synthetic jets were the preferred technique during the early stages of this practice, fluidic oscillators, also called sweeping jets actuators, were recently in the focus of several research and engineering projects. These actuators bring in the advantages of no-moving-parts and high amplitude signal. However, their control is not straight forward and many times the exact nature of their output is not fully comprehended.

The internal flow mechanism of a suction and blowing (SOAB) device was investigated using Large Eddy Simulation, followed by comparison with measurements. This analysis is a part of a large-scale combined experimental-numerical bi-national effort. Instantaneous velocity contours from the simulation of the internal domain is attached below.

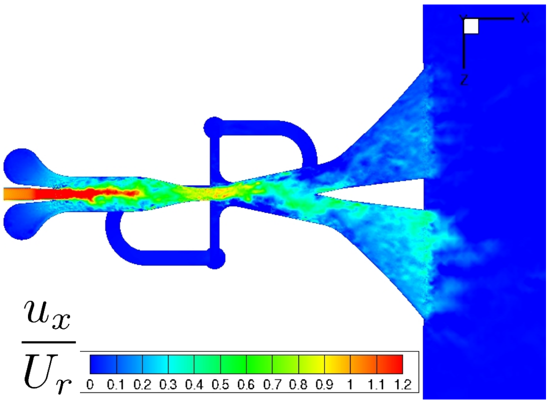


Figure 1: Instantaneous velocity contours over an horizontal cut in the flow domain, obtained by LES

The simulation results were then utilized to construct a set of functional-fit velocity profiles, to be used as time-dependent boundary conditions for the oscillatory blowing.

Towards implementation of a bank of actuators in a turbulent boundary layer, the separated effects of the actuators output, like suction alone, were investigated within a laminar and turbulent boundary layers. Large eddy simulation was the tool employed for this task also, coupled with wind-tunnels measurements. Interesting aspects of shear enhancement, interaction between suction holes wakes and onset of turbulence were observed.

The impact of large scale flow structures on turbulent spray flames

Yuval Dagan, Eran Arad and Yoram Tambour

June 4, 2015

The dynamics of spray-flame structures in the vicinity of a recirculating flow is investigated in the present study. Flame instabilities play an important role in turbulent combustion. Strong coupling with turbulent flow occurs and significantly influences the flame dynamics. Cases in which the flow is turbulent, which are inherently three-dimensional and unsteady, impose crucial unsteady characteristics on the combustion process which complicates the instability analysis. In our recent work [1], unsteady turbulent spray-flame instability was studied in concentric jet combustion chamber. A unique, two stage repetitive pattern of turbulent non-premixed spray flame, developing in the vicinity of a recirculating flow, was identified. In the current study we extend the investigation of large-scale spray flame evolution of the same configuration, using reduced order methods such as one dimensional flame tracking and proper orthogonal decomposition (POD). This analysis provides new insight into the driving mechanism behind the large-scale flame instabilities in turbulent recirculating flows.

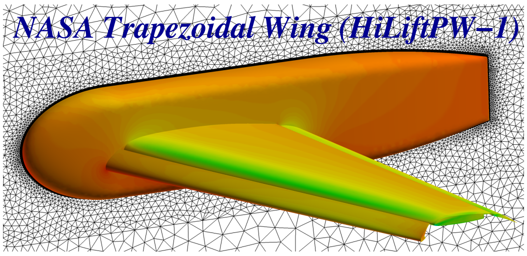
References

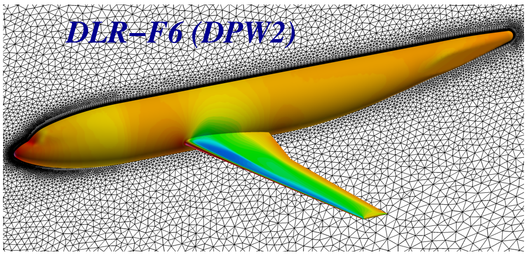
[1] Yuval Dagan, Eran Arad, and Yoram Tambour. On the dynamics of spray flames in turbulent flows. *Proceedings of the Combustion Institute*, 35(2):1657–1665, 2015.

Turbulent Flow Simulations on Unstructured Grids using Reynolds Stress Model

Y. Mor-Yossef[*]

Progress toward a stable and efficient numerical treatment for the compressible Reynolds-averaged Navier-Stokes equations with a Reynolds-stress model (RSM) using unstructured grids is presented. The mean-flow and the Reynolds stress model equations are discretized using a finite volume method that is based on second order accuracy. The time-marching approach that is chosen relies on the use of a decoupled implicit time integration method. A special design of the Reynolds stress model implicit operator significantly improves the overall flow solver robustness. The key idea is the use of the unconditionally positive-convergent implicit scheme (UPC), originally developed for two-equation turbulence models. The extension of the UPC scheme for Reynolds stress models guarantees the positivity of the normal Reynolds-stress components and the turbulence (specific) dissipation rate for any time step. Thanks to the UPC matrix-free structure and the decoupled approach, the resulting computational scheme is very efficient. Results obtained from two and three-dimensional numerical simulations demonstrate the significant progress achieved in this work toward optimally convergent solution of Reynolds stress models. Furthermore, the scheme is shown to be unconditionally stable and positive.





[*]Chief Scientist, Israeli CFD Center (ISCFDC), Caesarea Industrial Park 3088900, Israel

Symmetry-Breaking and Hysteresis of the Averaged Flow-Field in Stirred Tank Reactors with Radial Impellers

R. Ben-Nun and M. Sheintuch

Department of Chemical Engineering, Technion – Israel Institute of Technology, Haifa, Israel

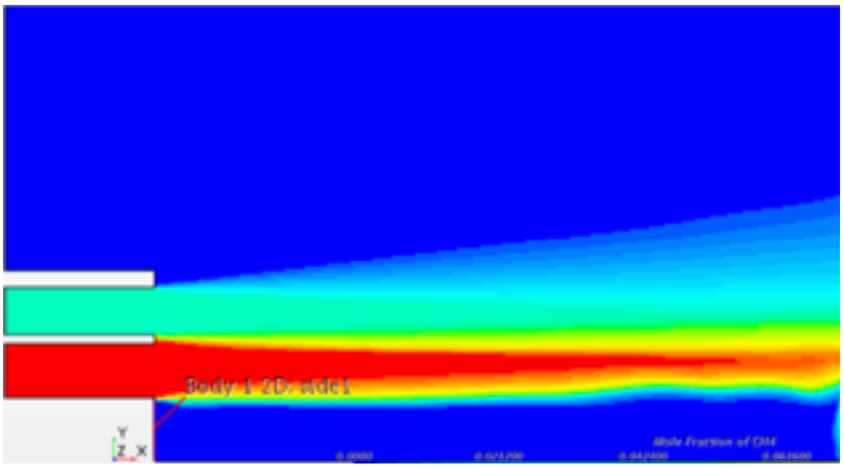
Conference topic section #3 ; Speaker is underlined.

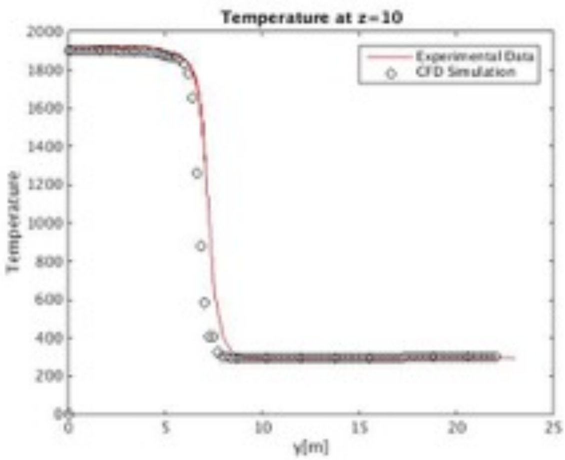
Abstract – Symmetry-breaking of the averaged turbulent flow-field in stirred tanks with single- and dual-Rushton impellers is revealed by numerical experiments. This leads to multiplicity of solutions and hysteresis of the flow-field, upon varying a parameter, in tanks with equal off-top and off-bottom impeller clearances (see Fig. 1.a). As expected, at low tank to impeller diameter ratio (T/D) the solution is symmetric relative to the horizontal mid-plane of the tank but beyond a certain threshold it loses stability and converges to an asymmetric state (see Fig. 1.b). Tracking the solutions backwards to geometries with smaller T/D ratios reveals the existence of a multiple-solutions domain. This is verified for two rotational speeds (N) and various tank heights (H), using the standard $k-\varepsilon$ turbulence model ($N = 100rpm$ and $1600rpm$, corresponding to $Re = 1.67 \cdot 10^4$ and $2.67 \cdot 10^5$, respectively). This suggests that the bifurcation to asymmetry is subcritical i.e., into unstable solutions. Finally, a general correlation for the symmetry-breaking bifurcation is successfully formulated, based on the turbulent jet properties of the impeller's jet and the system dimensions. To our best knowledge, these phenomena have not been reported before for the standard stirred tank system, neither experimentally nor numerically, since most studies employ quiescent fluid as initial condition. As we discuss in the paper, the discoveries from our numerical work portend significant industrial importance and interest.

RANS Modeling of Stratified Combustion

Steven H. Frankel
CFDLAB, Faculty of Mechanical Engineering,
Technion - Israel Institute of Technology, Haifa, Israel

Turbulent combustion is the driving force behind the majority of the worlds transportation and power generation devices including most notably automotive internal combustion engines, jet and stationary gas turbine engines, and residential and industrial power heat and power generation systems. Concerns about improved efficiency and stability and lower pollutant emissions continue to drive research in this field. The challenge of turbulent combustion modelling is related to somehow accurately accounting for the effects of the wide range of length and time scales associated with interactions between the turbulent flow field, scalar mixing, and chemical kinetics processes on the large-scale or mean flow. It is generally accepted that a turbulent combustion model that accurately accounts for turbulent-chemistry interactions across flame regimes from premixed to non-premixed and beyond is needed. In particular, a recent trend in turbulent combustion research involves a further shift away from academic flame studies such as premixed and non-premixed to partially-premixed and even more recently, stratified flames. The later are more common in practice where direct injection or unsteady flame dynamics may lead to combustion occurring within a stratified mixture. Also stratification is being considered as a means to achieve further gains towards stable, low-emission combustion engines. In this talk, we plan to provide a brief overview of stratified flames and focus on two target stratified flames with regard to availability of measured data for validation and a first attempt at applying standard RANS models available in the commercial CFD software Star-ccm+ to non-swirl and swirl stratified target flames. Preliminary methane contour plot and temperature profile for two cases below.





Computational Surgery Towards Powered Fontan Hemodyanmics

Yann Delorme
CFDLAB, Faculty of Mechanical Engineering,
Technion - Israel Institute of Technology, Haifa, Israel

Babies who are born with a single working ventricle often have to undergo a series of three staged open hear surgeries (called the Fontan procedure) during the first few years of their lives to surgically connect the inferior and superior vena cavae (IVC and SVC) to the left and right pulmonary arteries (LPA and RPA) effectively bypassing the dysfunctional right side of the heart. This creates what is called the total cavopulmonary connection or TCPC or Fontan geometry which resembles a four-way junction with the two IVC/SVC streams colliding and splitting at the junction supplying blood to the lungs via the LPA/RPA vessels. These patients experience complicated side-effects from the surgeries and often experience heart-failure like symptoms later on in life. Over the past few years, in collaboration with Dr. Mark Rodefeld at Indiana University and Riley Children's Hospital, we have developed a novel blood pump to restore normal biventricular pressure levels to these patients either on a temporary basis or via a chronic device. The chronic device would be surgically implanted effectively replacing the TCPC junction. We have recently carried out high-fidelity CFD studies exploring virtual surgical implantation of the device in a patient-specific Fontan geometry and the subsequent complex hemodynamics using an in-house high-fidelity LES code which employs a novel multi block immersed boundary method to enable simulations of rotating geometries using fixed structured Cartesian grids. Results are presented with regard to complex vortical structures, flow patterns, and particle-laden flow tracking as it relates to hepatic factor distribution and potential thrombus formation.

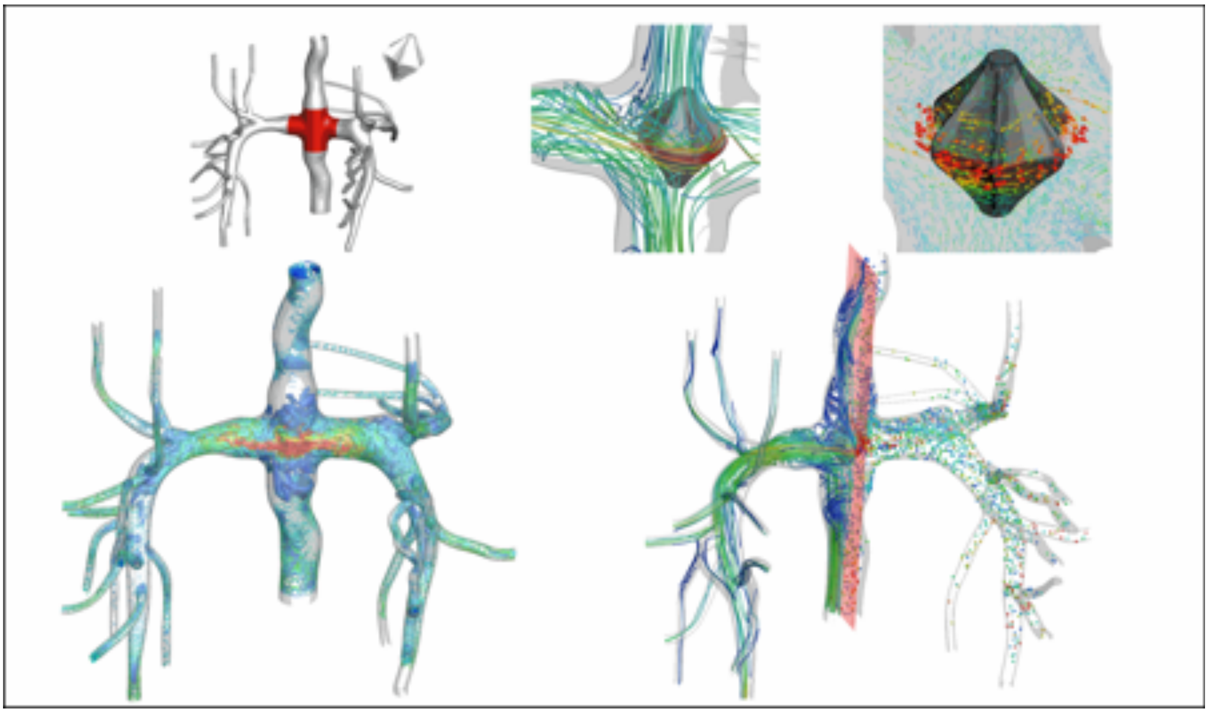


Figure 1. Images showing patient-specific Fontan geometry with chronic pump implanted and various flow pattern and structure plots.

Direct linear solvers for lower order methods in incompressible CFD

A. Yu. Gelfgat

School of Mechanical Engineering, Faculty of Engineering, Tel-Aviv University, Ramat Aviv 69978, Tel-Aviv, Israel

Most of modern CFD codes apply implicit or semi-implicit discretization in time that reduce implementation of each time step to solution of several systems of linear algebraic equations of very high order and sparse matrices. Usually these systems are being solved by iterations methods among which multigrid and Krylov-subspace iterations became most popular. At the same time several effective direct solvers for sparse matrices were developed and were implemented for direct computation of steady flows by Newton method and analysis of their linear stability, for which eigenvalues are computed by Arnoldi iteration. For latter tasks the direct methods may consume less CPU time than iterative ones. Furthermore, with increase of computational power, a well-known direct matrix inverse via the eigenvalue decomposition can be applied to three-dimensional time-dependent problems. In this presentation we review these methods and describe several successful applications to different problems starting from steady sate flows, analysis of their instabilities, and fully three-dimensional DNS.

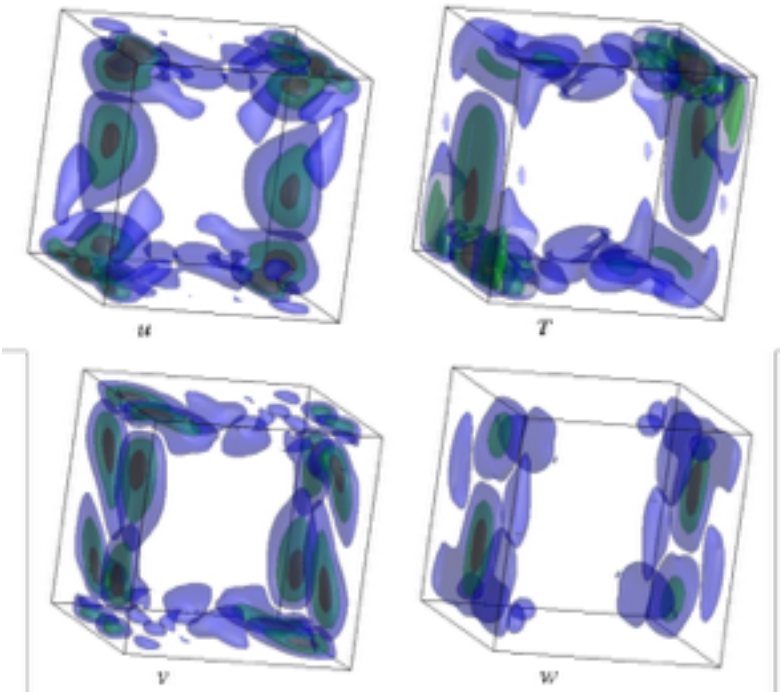


Figure 1. Patterns of most unstable disturbance (leading eigenvector) of buoyancy convective flow in a laterally heated cube with perfectly conducting horizontal and perfectly insulated spanwise boundaries.

Convergence Acceleration for Multiphysics Compressible Flow

Oren Peles and Eli Turkel (presenter)
School of Mathematical Sciences, Tel Aviv University

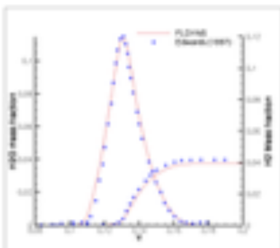
Session: High-complexity in CFD

Abstract:

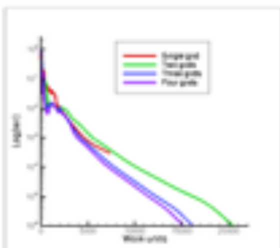
We study the numerical solution of physically complex compressible flows, including reacting and two-phase turbulent flow. We discuss both external flow, such as flow around wings, and internal flow in complex geometries e.g. inside a combustion chamber. We present solutions for both steady state solutions and time dependent ones using the compressible Reynolds-Averaged Navier-Stokes with turbulent model equations including the one equation Spalart-Allmaras model and a two equation model ($k-\omega$ SST) and including multi-species, reactive, real gas flows. Now, there are several continuity equations, one for each chemical species. The internal energy is represented by a polynomial relation with empirical polynomial coefficients taken from thermodynamic databases. An Arrhenius chemical kinetics model is used for the chemical reactions. The last extension is the coupling to an Eulerian representation of dispersed phase flow. This model represents the spreading of solid particles or liquid droplets in the computational domain. The particles have their own spatial density and advection velocity. The momentum equation is controlled by the drag force between the particles and the gas. The system is very stiff due to a wide range of velocities (from Mach number zero to supersonic flow), large scales of reaction rates of the species and strong source terms relative to the advection and diffusion together with numerical grids in boundary layers characterized by a large grid aspect ratio. This can require long computational times for convergence. We develop a method that allows for fast convergence for steady state flow. This is based on a central difference scheme, augmented by several possible dissipation models, and an explicit Runge-Kutta pseudo-time stepping for the steady state. We add an implicit residual smoothing that includes contributions from all the stiff terms. This preconditioning allows much larger time steps than that of the explicit scheme. This is extended to time dependent problems by the dual time stepping approach.

Results for a **reactive shear layer**

This example demonstrates the effect of multigrid for a case with mid-subsonic turbulent non-premixed reactive flow.



Cut-line H2O and H2 distribution at constant x = 0.44



Convergence history for reactive shear layer problem

Development of a Compressible Navier-Stokes Solver using Immersed Boundary Method and Adaptive Mesh Refinement

Kunal Puri
Faculty of Mechanical Engineering, Technion - Israel Institute of Technology, Haifa, Israel

High-fidelity compressible Navier-Stokes CFD codes are needed for a number of practical engineering problems including aerodynamics, reacting and multiphase flows. Such codes must somehow strike the correct balance between accuracy and efficiency while still being able to handle complex geometries with high-fidelity. Two modern computational methods or tools to facilitate these goals include the immersed boundary method (IBM) and adaptive mesh refinement (AMR). IBM is a class of techniques enabling CFD simulations of flow over or through complex stationary or moving geometries using fixed structured Cartesian grids. The advantage of using structured Cartesian grids is that it (a) expedites the grid generation process when dealing with complex geometries and (b) enables the use of high-order finite-difference or finite-volume numerical methods. AMR is a class of techniques for adaptively refining the computational mesh during a simulation to more accurately and affordably capture complex flow features. In this talk, efforts towards combining these two powerful computational methods into a single compressible Navier-Stokes solver are described. The finite-volume method is used to discretize the equations. The IBM implementation features a ghost-point method and the AMR solver is based on the BoxLib framework. Implementation details and test results from both the IBM and AMR methods will be presented followed by applications to several benchmark type flow problems to address accuracy vs. efficiency issues.

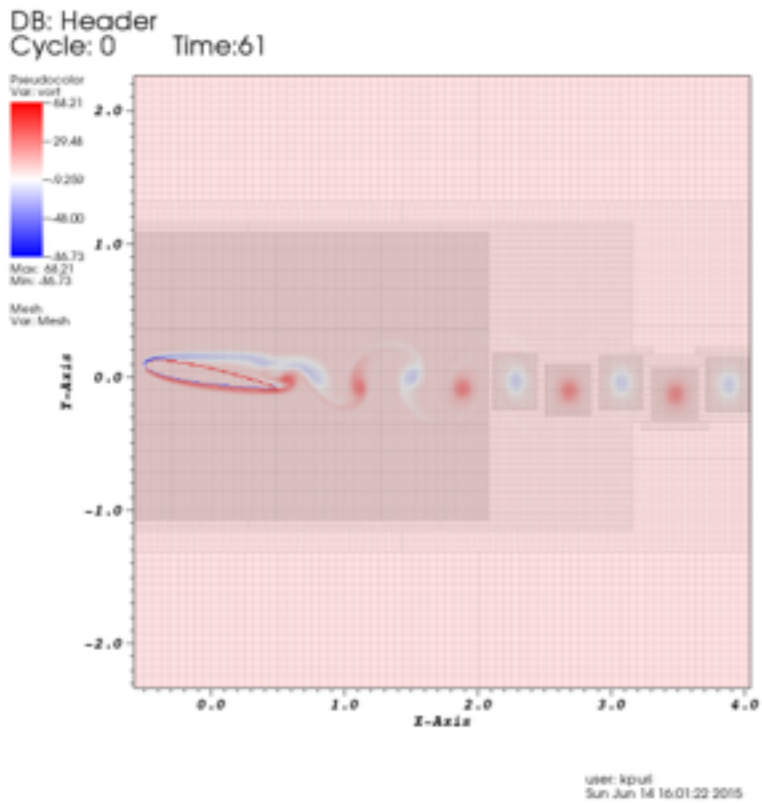


Figure 1. Snapshot showing vorticity contours from 2D flow over airfoil at 10 deg AOA from the new IBM/AMR compressible Navier-Stokes code.

Targeted Experiments for CFD Validation of Separated and Unsteady Turbulent Flows

David Greenblatt
Faculty of Mechanical Engineering, Technion - Israel Institute of Technology, Haifa, Israel

An important step in the development of CFD codes for aerodynamic applications is validation against high fidelity data. Historically, these data were generated by means of well documented experiments that comprised hot-wire anemometry and optical measurements of the flowfield, and discrete unsteady surface pressure measurements. Optical measurements included laser Doppler anemometry (LDA) and both two- and three-dimensional (stereoscopic, i.e. three velocity components in a plane) particle image velocimetry (PIV). However, for meaningful validation of LES and DNS codes that generate three-dimensional unsteady instantaneous turbulent flowfields, new validation experiments must be designed and modern measurement techniques must be exploited. In recent years, two powerful optics-based experimental techniques have emerged: tomographic PIV for volumetric three-dimensional flowfield measurements and unsteady pressure sensitive paint (PSP) for practically continuous surface pressure measurements. In this talk, I will describe a CFD validation test case that I led at NASA Langley Research Center ten years ago, that is still being used for CFD validation. Following this, I will describe a unique unsteady wind tunnel (UWT) in the Technion's Flow Control Laboratory that offers unprecedented optical access specifically to facilitate tomographic PIV and unsteady PSP. Examples data of these two techniques, acquired by us and our collaborators, will be presented and future experiments will be described.

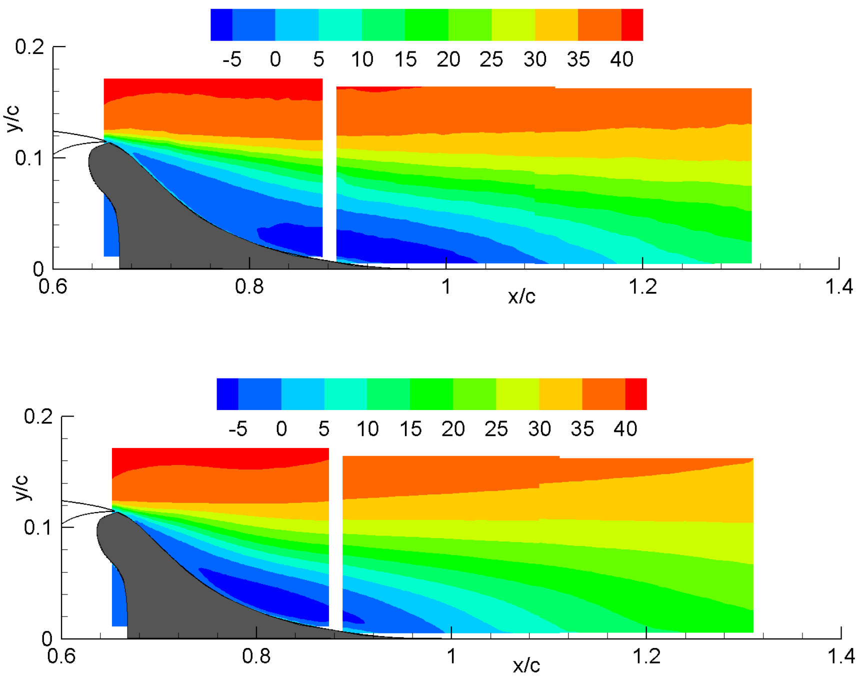


Figure 1. PIV measurements for CFD validation case for flow over a ramp without and with flow control.