ManChester CFD

World-leading research and consultancy

www.ManchesterCFD.co.uk

December 2023

INDEX

WORLD-CLASS RESEARCH & CONSULTANCY

Heat, Ventilation and Air Conditioning (HVAC)
Environmental Impact Assessment
Experimental Fluid Mechanics
Biomedical Engineering
Design Optimisations
Artificial Intelligence
Machine Learning
Multiphase Flow
Heat Transfer
Turbulence
Net-Zero

ManChester CFD

The 'ManchesterCFD' Team

The 'ManchesterCFD' team is hosted within the University of Manchester and directed by Dr Amir Keshmiri. The 'ManchesterCFD' team is multi-award winning research team with access to world-class experimental and numerical research facilities. The team is actively involved in high-profile research and industrial projects and produces several high-quality publications per year. The consultancy and training division of ManchesterCFD team has been involved in numerous industrial and academic projects and has conducted several specialist training in different aspects of modelling and simulation.

Computational Fluid Dynamics (CFD)

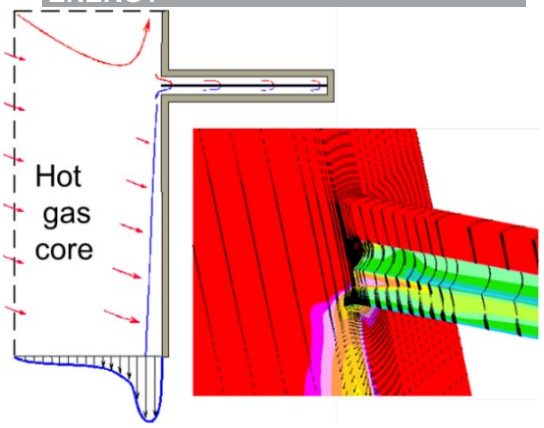
CFD is a process of representing a fluid flow problem by mathematical equations based on the fundamental laws of physics and solving those equations to predict the variation of the relevant parameters within the flow field. Usually the parameters which are solved in CFD are velocity, pressure, temperatureand other variables such as turbulence parameters and concentrations of chemical species. Therefore,CFD can provide a cost-effective and accurate alternative to scale model testing. Recent advances in computing power together with powerful graphics and interactive 3D manipulation of models mean thatmore complex flow problems can be simulated using CFD.

Extensive Industrial Experience

In addition to conducting world-class fundamental research, the ManchesterCFD regularly works with numerous companies in the UK and beyond in a wide range of sectors. Some of the companies which have recently worked on research and consultancy projects with ManchesterCFD team are listed below.

TURBULENCE & HEAT TRANSFER

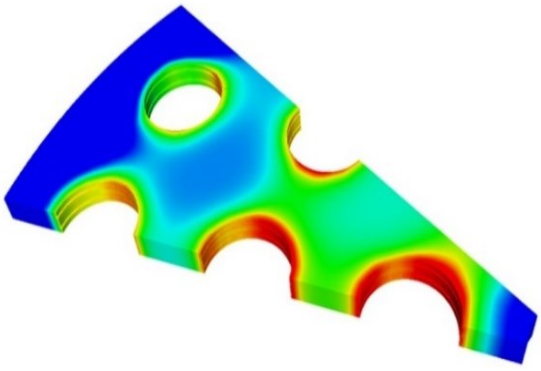
ENERGY



Buoyancy induced Flow in boiler's penetration cavity

Company: EDF Energy, UK

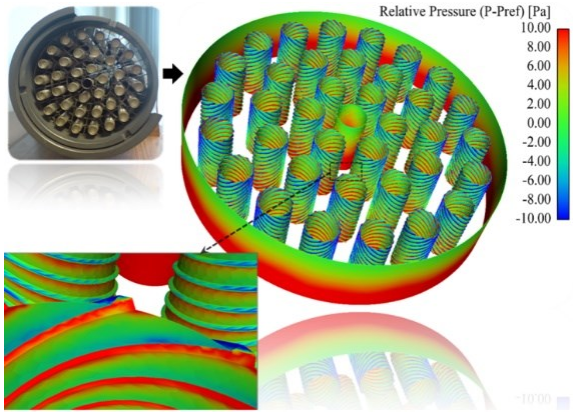
NUCLEAR



Heat transfer analysis of the CO2 coolant in reactor core flow

Company: British Energy, UK

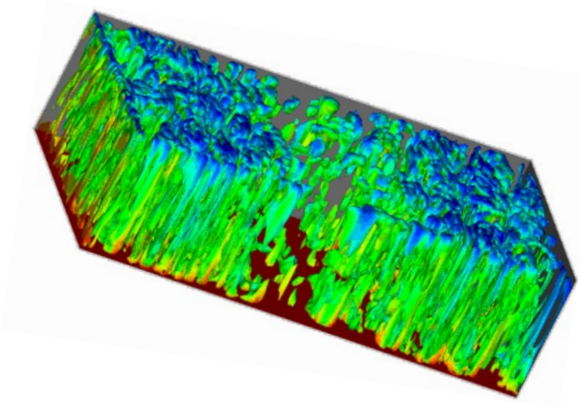
NUCLEAR



Modelling of coolant flow in advanced gas-cooled reactors

Company: Dalton Nuclear Institute, UK

ENERGY



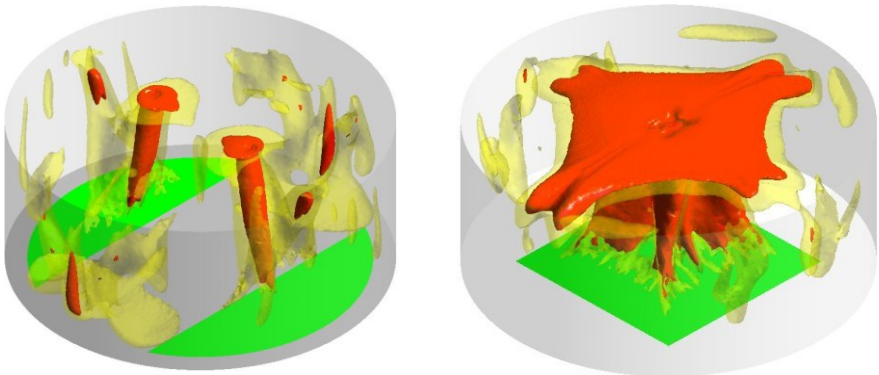
Heat transfer in an inclined cavity simulated by three different CFD Codes

Company: EDF Energy R&D, UK

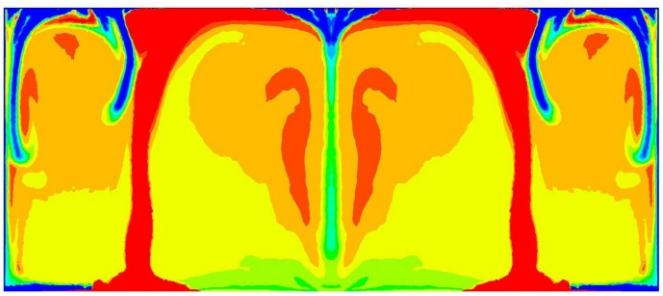
Project: CFD Simulation of Different Heating Elements in a Large Storage Tank

Company: Engie/Total, UK

Brief Description: In this project, a large storage tank was to be simulated in order to design the most efficient heating element configuration. A very viscous fluid was maintained at a high temperature with continuous flow in and out of the tank. The aim was to maximise the natural convection within the tank, find the optimum heating element configuration, reduce the energy consumption and to find the best location to place the temperature probes within the tank. The simulations were successfully conducted for a number of designs and the most efficient configuration based on optimum heat transfer and velocity distributions were selected. All the hot and cold spots within the tank were also identified to provide the best locations to place the temperature probes. In addition, the effects of external wind on the tank was also taken into consideration when deciding on the best heating element configuration, to minimise energy consumption.



Velocity iso-surfaces at 2 different heating elements



Temperature contour and natural convection at the centre of a tank

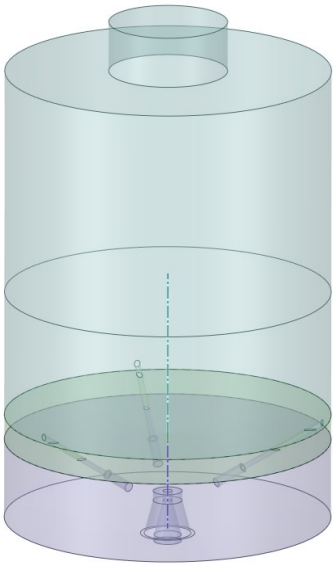
Relevant ManchesterCFD Publications:

- Boujelbene, M., Mahdi, J.M.,…, Yaïci, W., Talebizadehsardari, P. & Keshmiri, A., 'Revolutionizing heat recovery in shell-and-tube latent heat storage systems: an arc-shaped fin approach', Eng. Applications of Computational Fluid Mechanics, 17(1), 2023. DOI: 10.1080/19942060.2023.2255036
- Khedher, N.B., Khosravi, H.T., Talebizadehsardari P. & Keshmiri, A., 'Comprehensive analysis of melting enhancement by circular Y-shaped fins in a vertical shell-and-tube heat storage system', Eng. Applications of Computational Fluid Mechanics, 17 (1), 2023.
- Khosravi, K., Eisapour, A.H., Rahbari, A., Mahdi, J.M., Talebizadehsardari P. & Keshmiri, A., 'Photovoltaic-thermal system combined with wavy tubes, twisted tape inserts and a novel coolant fluid: energy and exergy analysis', Eng. Applications of Computational Fluid Mechanics 17 (1), 2023.
- Arasteh, H., Rahbari, A., Mashayekhi, R., Keshmiri, A., Mahani, R., Talebizadehsardari, P., 'Effect of pitch distance of rotational twisted tape on the heat transfer and fluid flow characteristics', Int. Journal of Thermal Sciences, Vol. 170, 2021, 106966.
- Shamsabadi, H., Rashidi, S., Esfahani, J.A. & Keshmiri, A., 'Condensation in the presence of non-condensable gases in a convergent 3D channel', Int Journalof Heat and Mass Transfer, 152, 119511, 2020.
- Darbari, B., Rashidi, S. & Keshmiri, A., 'Nanofluid heat transfer and entropy generation inside a triangular duct equipped with delta winglet vortex generators', Journal of Thermal Analysis and Calorimetry, 140, 2020, 1045-1055.
- Akbarzadeh, M., Rashidi, S., Keshmiri, A. & Shokri, N., 'The Optimum Position of Porous Insert for a Double-Pipe Heat Exchanger Based on Entropy Generation and Thermal Analysis', Journal of Thermal Analysis and Calorimetry, 139, 2020, 411-426.
- Keshmiri, A., Revell, A. & Darabkhani, 'Assessment of a Common Non-Linear Eddy-Viscosity Turbulence Model in Capturing Laminarization in Mixed Convection Flows', Int. J. Numerical Heat Transfer; Part A, Vol. 69(2), 2016, 146-165.
- Dehghan, M., Shokri, N., Keshmiri, A., Valipour M.S. and Saedodin, S., 'On the Thermally Developing Force Convection Through A Porous Material Under the Local Thermal Non-Equilibrium Condition: An Analytical Study', Int. J. Heat and Mass Transfer, Vol. 92, 2016, 815-823.
- Keshmiri, A., Osman, K., Benhamadouche, S. & Shokri, N., 'Assessment of Refined RANS Models against Large Eddy Simulation and Experimental Data in the Investigation of Ribbed Passages with Passive Heat', Int. J. Numerical Heat Transfer; Part B, Vol. 69(2), 2016, 96-110.
- Keshmiri, A., Uribe, J. & Shokri, N., 'Benchmarking of Three Different CFD Codes in Simulating Natural, Forced and Mixed Convection Flows', Int. J. Numerical Heat Transfer; Part A, Vol. 67(12), 2015, 1324-1351.

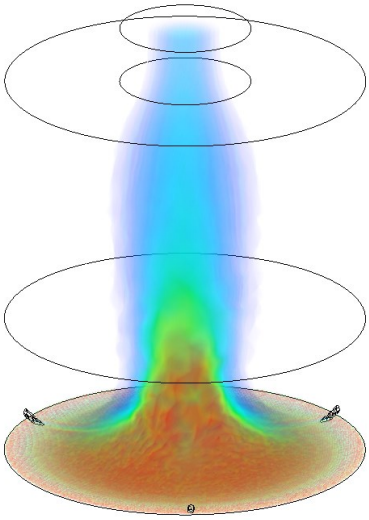
Project: Optimisation of the Ceramic Fibre Production Furnace

Company: Unifrax/Alkegen, UK.

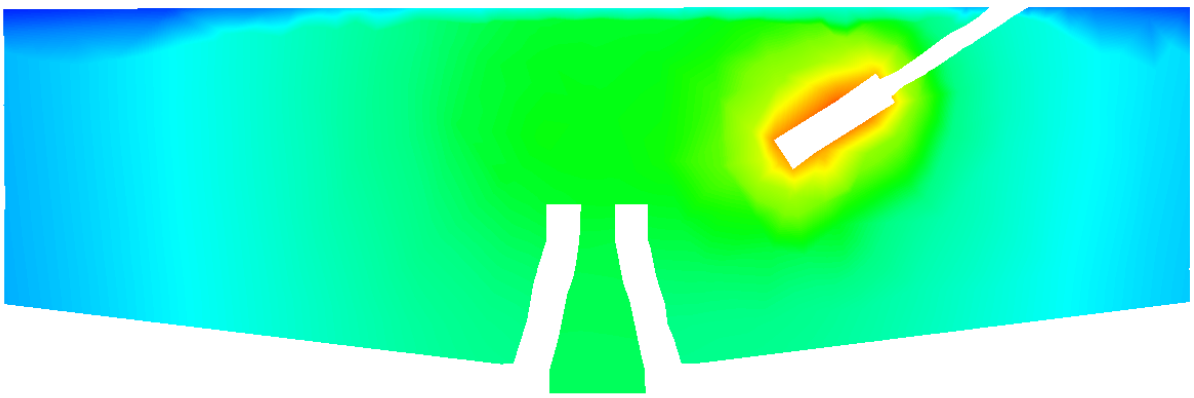
Brief Description: The aim of this project was to simulate a large industrial furnace and its operation with the purpose of identifying ways to reduce energy consumption and reduce its carbon footprint. The project was completed in 2 different stages and an accurate 3D model was built which included all the details of the furnace, heaters, nozzle output, cooling jackets and ventilation, etc. The results focused on temperature and melting profiles across the furnace and different heater locations were modelled to identify any potential savings. A parametric study was conducted to find the optimum location of the heaters. The simulations included complex heat transfer models and multiphase flow and the results were successfully compared against the experimental data provided by the client.



The full computational domain



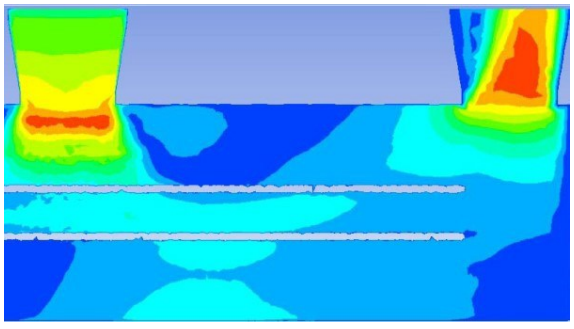
3D Velocity contour showing the ventilation



Temperature profile across a central 2D plane in the furnace

MULTIPHASE FLOWS AND POROUS MEDIA

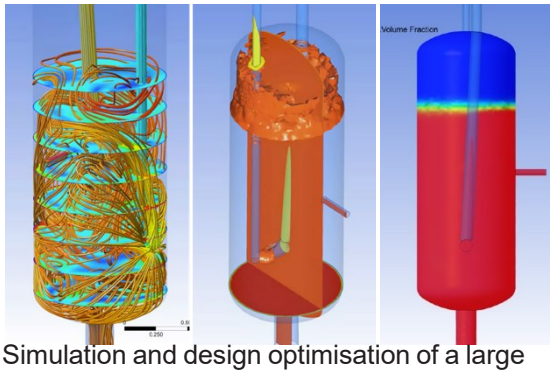
HEALTH&SAFETY



Simulation of an ultra-violet reactor designed for use in waste and clean water processing

Company: Xylem Water Solutions, UK

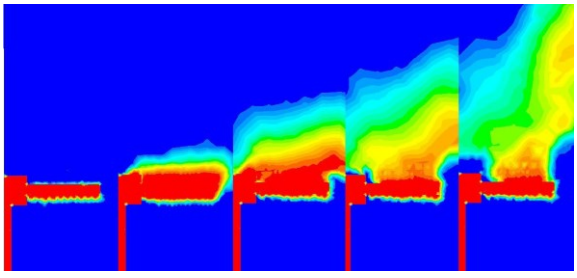
PETROCHEMICAL



Simulation and design optimisation of a large industrial mixing reactor.

Company: Ingevity, USA

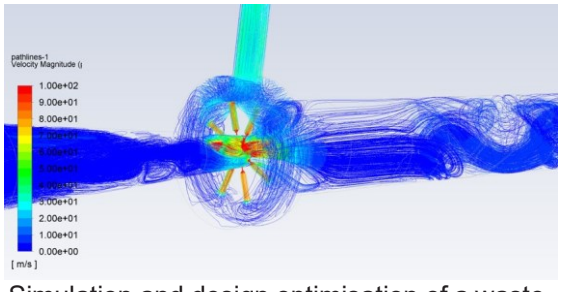
CHEMICAL



Simulation of Radial Gas Spargers in industrial bubble columns

Company: Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

WATER&UTILITIES



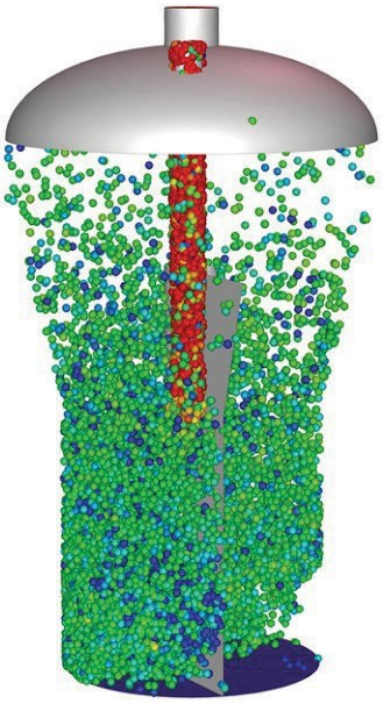
Simulation and design optimisation of a waste-water treatment line.

Company: Oxfiniti, UK

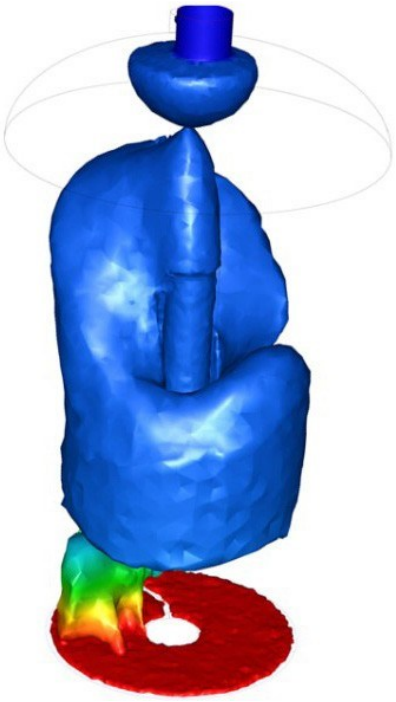
Project: Simulation and Optimisation of an Industrial-Scale Fluidised-Bed Dryer

Company: SCG Chemicals, Thailand

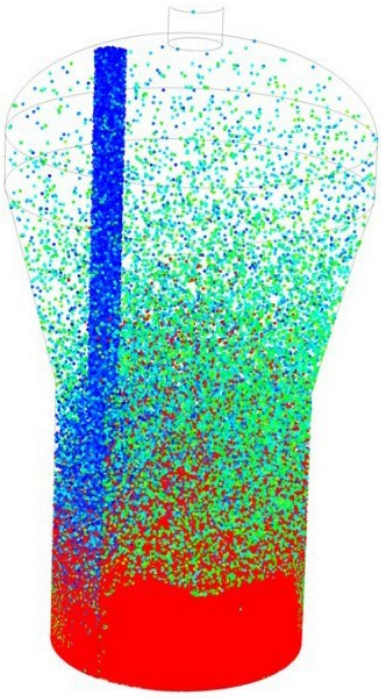
Brief Description: For the first time, an industrial-scale Fluidised-Bed Dryer (FBD) for the largest petrochemical company in Thailand was simulated with an aim of obtaining key operating parameters such as temperature and velocity distributions, evaporation rate, moisture level within solid particles and residence time. The physics involved in such a large reactor is extremely complicated and involves multiphase flow, turbulence, heat transfer, evaporation and particle interaction, etc. Most of the methods and techniques used in this project have never been tried before and this project represents the most comprehensive and successful simulation of a fluidised-bed reactor with an industrial-scale application.



Particle diameter distribution



Static pressure iso-surfaces



Particle residence time

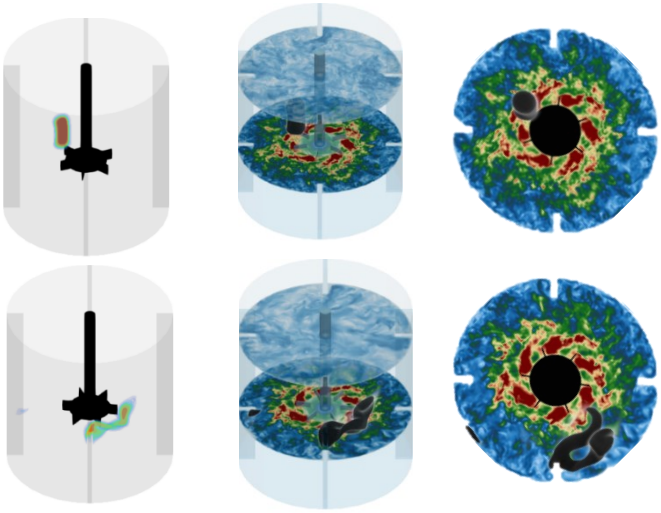
Relevant ManchesterCFD Publications:

- Shamsabadi, H., Rashidi, S., Esfahani, J.A. & Keshmiri, A., 'Condensation in the presence of non-condensable gases in a convergent 3D channel', Int Journal of Heat and Mass Transfer, 152, 119511.
- Hassani, A., Azapagic. A., D'Odorico, P., Keshmiri, A. & Shokri, N. 'Desiccation crisis of saline lakes: A new decision-support framework for building resilience to climate change', Science of the Total Environment, 703, 2020, 134718.
- Akbarzadeh, M., Rashidi, S., Keshmiri, A. & Shokri, N., 'The Optimum Position of Porous Insert for a Double-Pipe Heat Exchanger Based on Entropy Generation and Thermal Analysis', Journal of Thermal Analysis and Calorimetry, 139, 2020, 411-426.
- Rad, M.N., Shokri, N., Keshmiri, A. & Withers, P., 'Effects of Grain and Pore Size on Salt Precipitation During Evaporation from Porous Media: A Pore-Scale Investigation', J. Transport in Porous Media, Vol. 110(2), 2015, 281-294.
- Akbarzadeh, M., Rashidi, S., Keshmiri, A. & Shokri, N., 'The Optimum Position of Porous Insert for a Double-Pipe Heat Exchanger Based on Entropy Generation and Thermal Analysis', Journal of Thermal Analysis and Calorimetry, 2019 (DOI: 10.1007/s10973-019-08362-x).
- Dehghan, M., Shokri, N., Keshmiri, A., Valipour M.S. and Saedodin, S., 'On the Thermally Developing Force Convection Through A Porous Material Under the Local Thermal Non-Equilibrium Condition: An Analytical Study', Int. J. Heat and Mass Transfer, Vol. 92, 2016, 815-823.
- Norouzi Rad, M., Shokri, N., Keshmiri, A. & Withers, P., 'Effects of Grain and Pore Size on Salt Precipitation during Evaporation from Porous Media: A Pore-Scale Investigation', Petroleum Abstracts, Vol. 56(7), 2016, 101.
- Shokri, N., Zhou, P. & Keshmiri, A., 'Patterns of Desiccation Cracks in Saline Bentonite Layers', J. Transport in Porous Media, 2015, Vol. 110(2), 2015, 333-344.

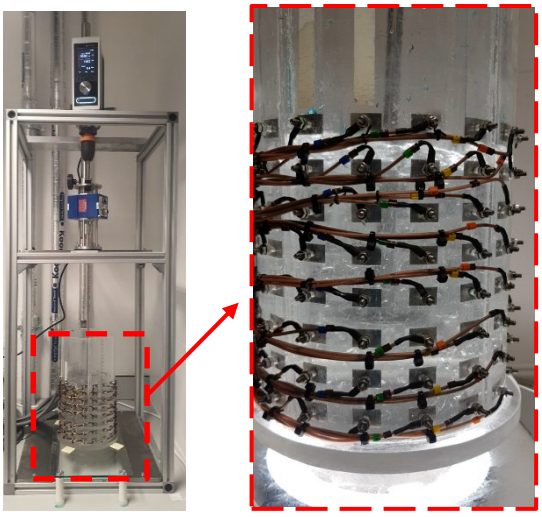
Project: Mixing of Liquids with Complex Rheological Behaviour

Company: Unilever PLC, UK.

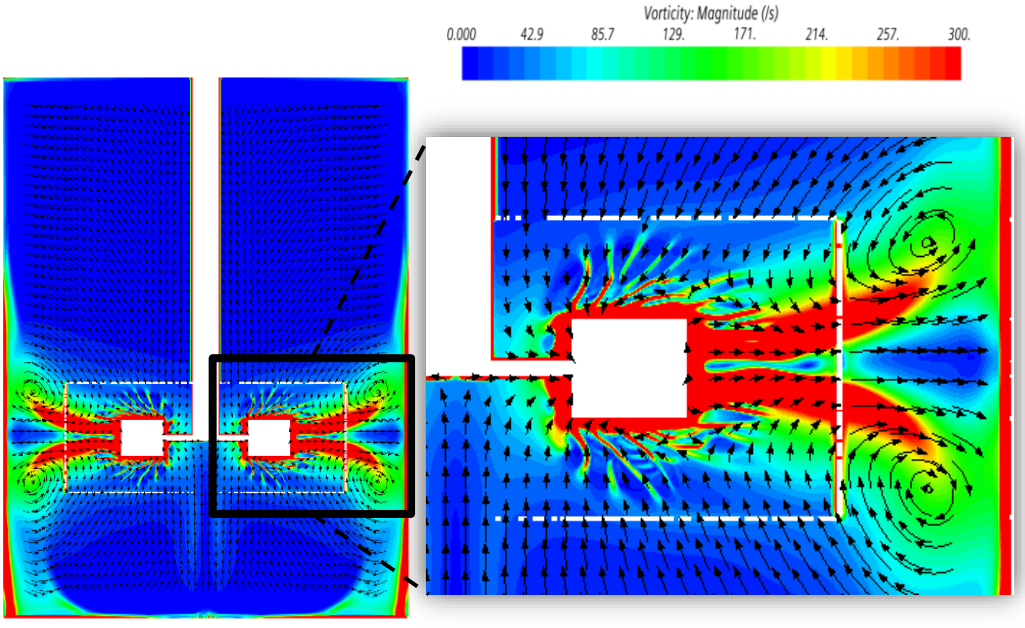
Brief Description: Mixing in stirred vessels is an everyday routine process in almost all food, consumer goods, and pharmaceutical industries. The aim of this project was to predict the flow behaviour using high-fidelity computational simulations within mixing processes in stirred vessels containing miscible liquids with high viscosity ratios. The computational framework developed by ManchesterCFD team makes use of accurate multi-phase flow models to predict the mixing time, which is a critical design parameter or assessing the mixing efficiency and is applied to compare the performance of different mixing vessels in industry. An experimental set up was also developed as part of this project using electrical resistivity tomography (ERT) technique, providing invaluable data for validating the CFD simulations. This flagship project was part of a multi-million pound project funded by Unilever and the UK Research Council.



Flow structures in an industrial-scale Rushton turbine using CFD



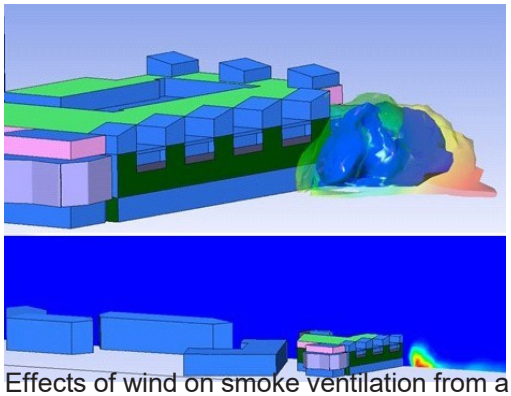
Experimental set up for validation of CFD simulations



Velocity vectors near the flat impeller of a Rushton turbine

HEATING, VENTILATION & AIR-CONDITIONING (HVAC)

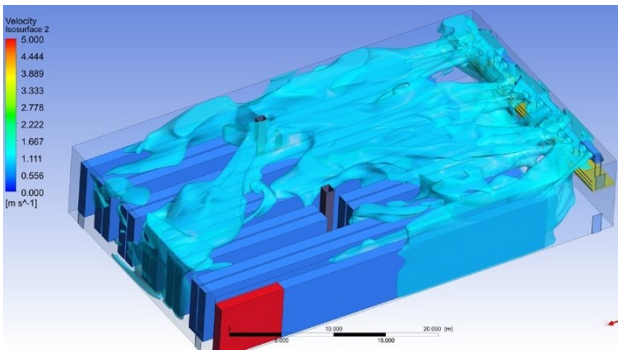
BUILDING SERVICES



Effects of wind on smoke ventilation from a large commercial carpark

Company: McAleer&McGarrity, Ireland

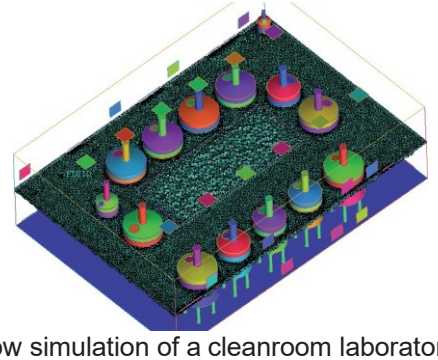
PHARMACEUTICAL



Simulation of the refrigeration system in a large cold store for pharmaceutical application

Company: Piggott&Whitfield Ltd, UK

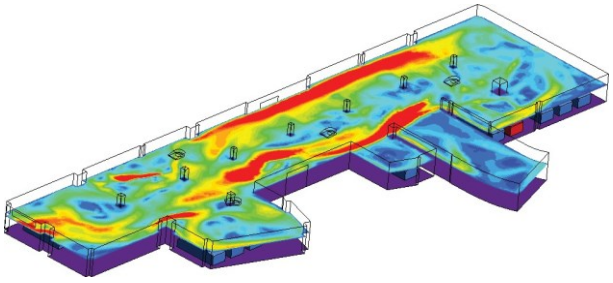
HEALTH&SAFETY



Flow simulation of a cleanroom laboratory

Company: Medical Simulation Ltd, UK

BUILDING SERVICES



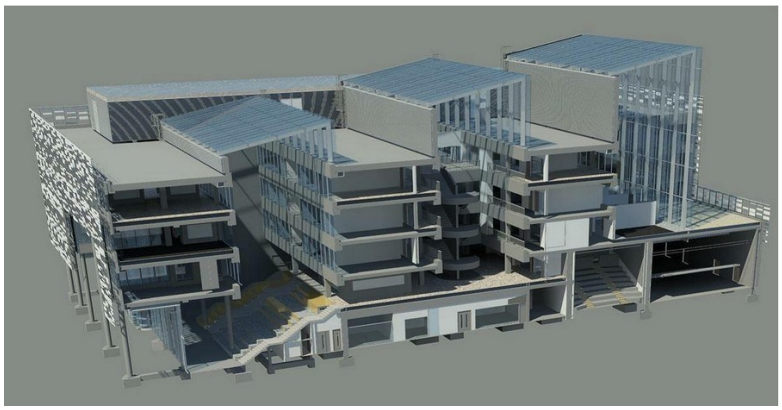
Smoke ventilation modelling in a large public car park

Company: Tourdan Ltd, UK

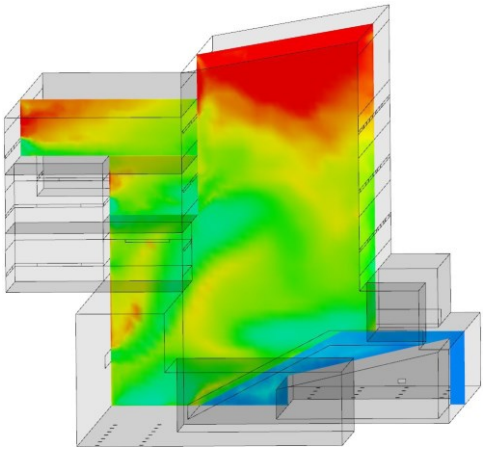
Project: Ventilation System Design for a Large Academic Building

Company: Manchester Metropolitan University, UK

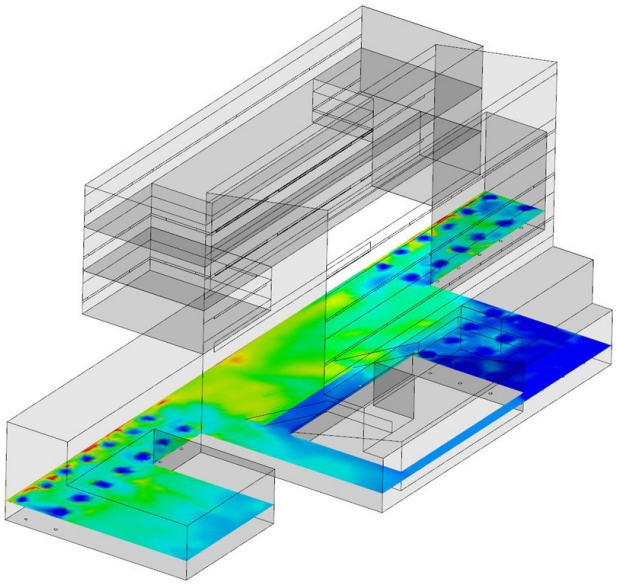
Brief Description: The ventilation system within the Birley Field Academic building (now Brooks Building) at Manchester Metropolitan University (MMU) was designed and modelled using state-of-the-art computational facilities. The aim was to reduce the energy consumption by improving the natural ventilation throughout the building using innovative designs. Our team carried out a number of detailed CFD simulations of the air flow for different atrium designs and scenarios which included different seasons and different occupancies, etc. This project involved a meticulous design of three different atria at different elevations, built to face the prevailing south-westerly wind in Manchester. Environmental Impact Assessment of this building was also evaluated using wind microclimate analysis.



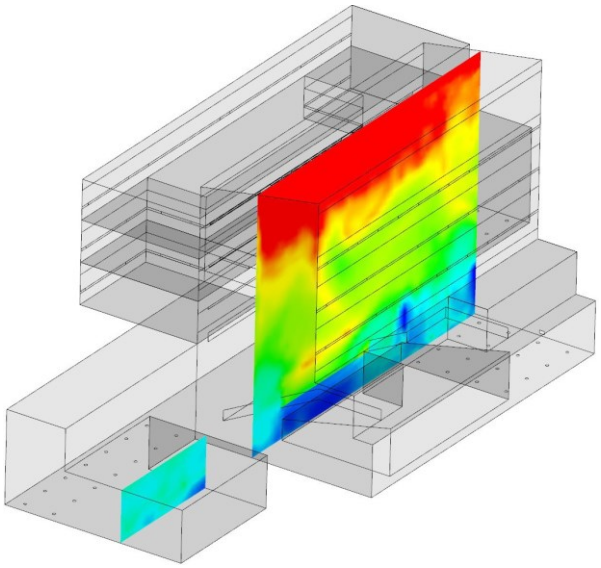
Schematic of the Birley Field academic building in Manchester



Temperature distribution and natural ventilation in an atrium



Velocity distribution within the ground floor and the main foyer

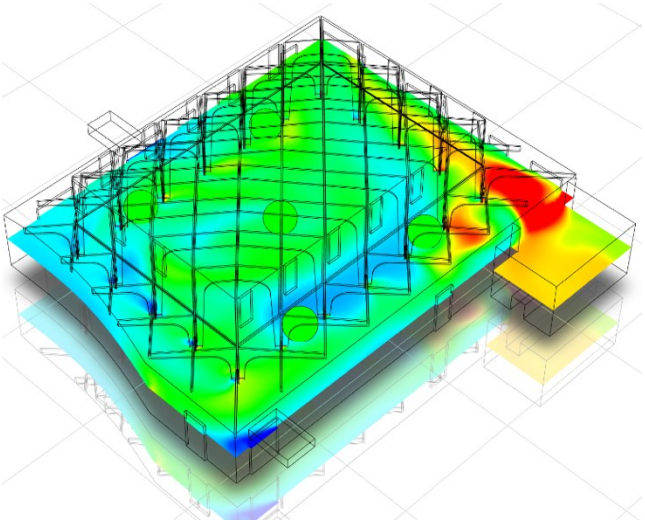


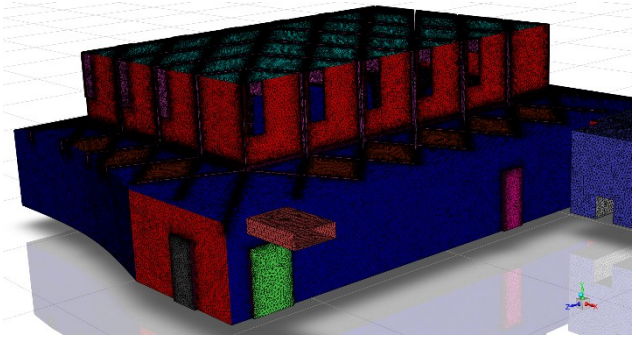
Temperature distribution within the first atrium

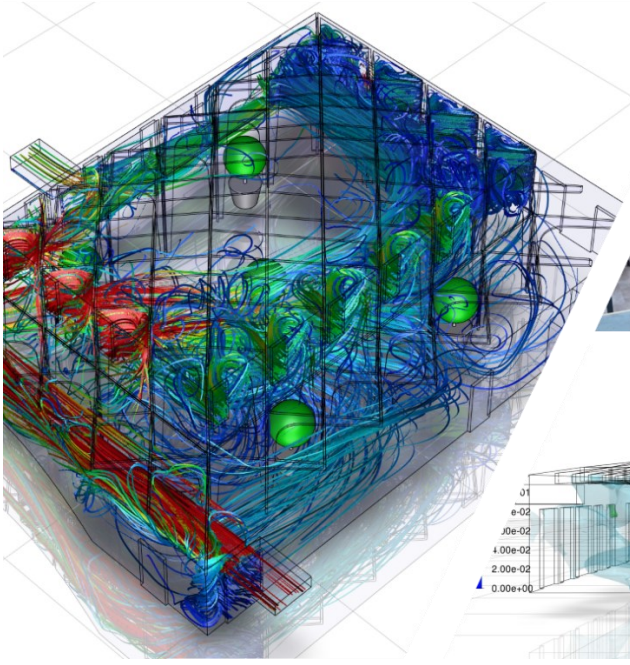
Project: Assessment of a Natural Ventilation System for COVID-19 Risk of Infection

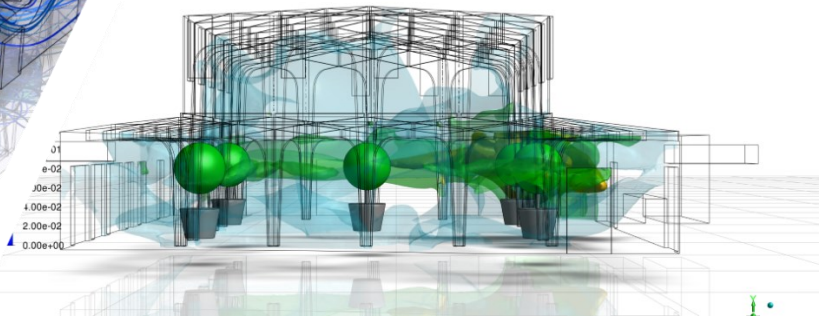
Client: St Richard's Hospice, UK

Brief Description: The St Richards Hospice cares for people with a serious progressive illness who have complex needs that cannot be met by other care services. In line with modern engineering priorities, a flagship building was designed to include the latest thinking in energy reduction technologies, such as natural ventilation. The aim of the investigation was to examine the impact and dangers of Covid-19 on the hospice's patients. Advanced computing expertise was used to help ensure that hospice patients are better protected from Covid-19 or similar future viruses following a pioneering study led by the ManchesterCFD team. The work developed a numerical framework and a new metric to assess the level of risk with high level of confidence, which can be used in all future ventilation designs for all similar large buildings. These findings will inform care managers and architects to create even safer environments as we look to manage living with Covid-19.





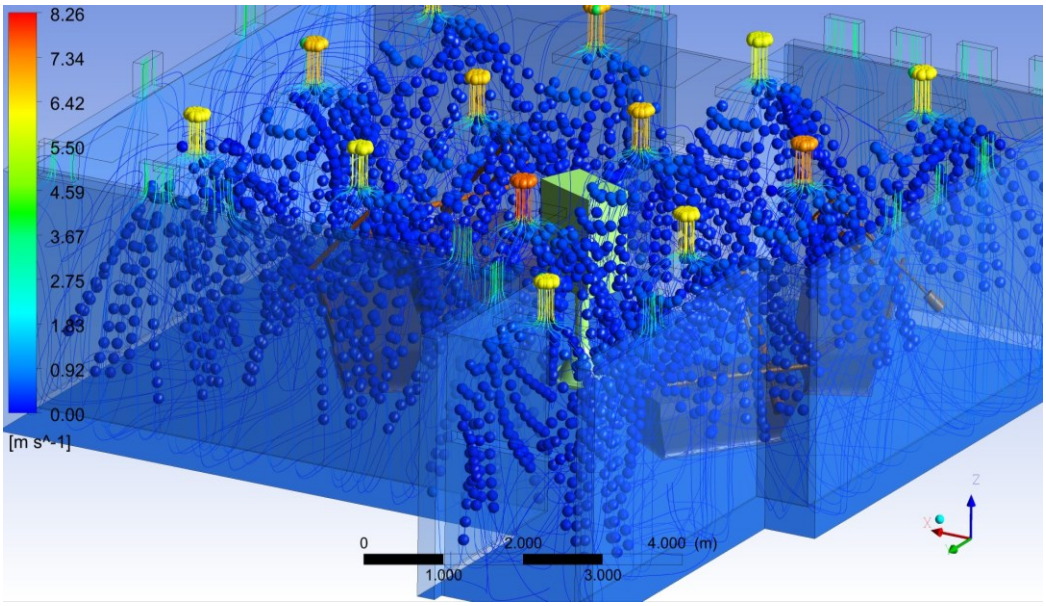




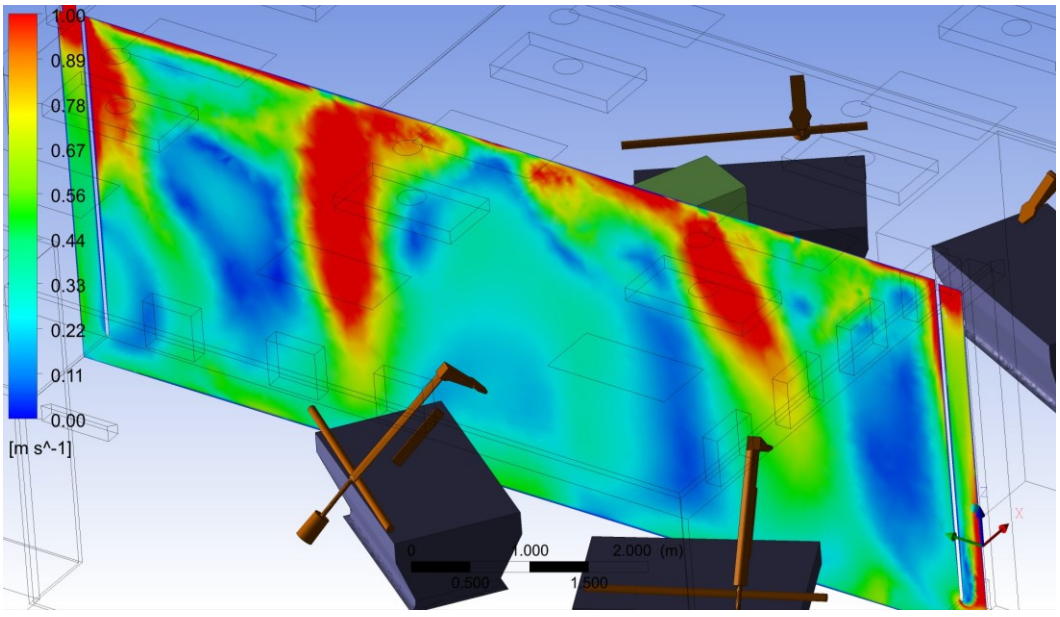
Project: Simulation and Design Optimisation of a Cleanroom Ventilation

Company: Rolls-Royce Plc, UK

Brief Description: The main aim of this project was to investigate the impact of making significant changesto the locations of various components in the ceiling of a large cleanroom. Two different scenarios were carefully modelled, designed and simulated by ManchesterCFD and the efficiency of the ventilation system for both scenarios was assessed. In order to determine the ventilation performance in a cleanroom, detailsof all filters and all inlets/outlets had to be modelled in detail and the assessment on the performance wascarried out using a range of innovative metrics including statistical analysis of individual units within the cleanroom, air freshness in different sections and particle tracking. The changes proposed by Manchester- CFD were subsequently implemented in the cleanroom by the client.



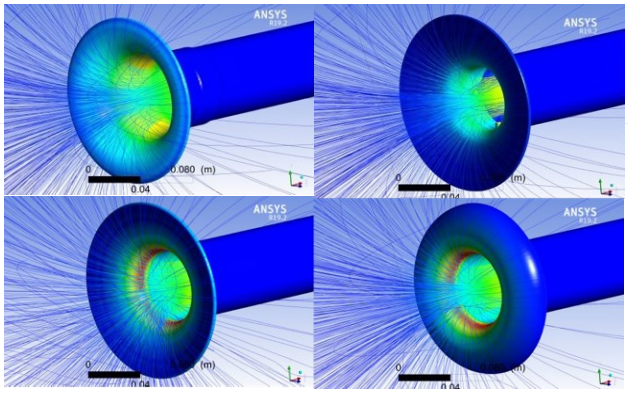
Velocity streamlines showing flow pattern from each inlet



Velocity distribution on a vertical plane across the cleanroom

DESIGN OPTIMISATION

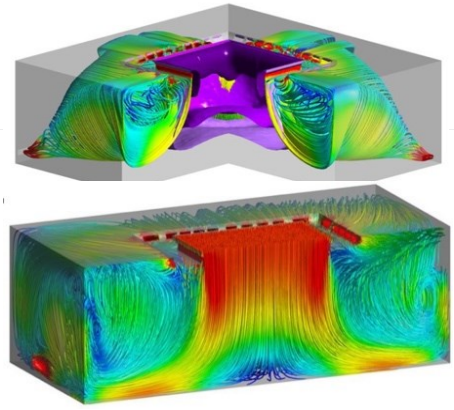
MANUFACTURING



Optimisation of a Local Exhaust Ventilation (LEV) system

Company: AEM Products, UK

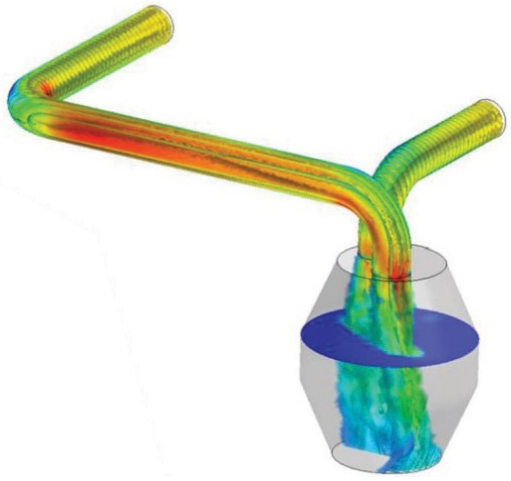
HEALTHCARE



Utilising novel fluid mechanics concept in ultraclean ventilation design for hospitals

Company: Medical Air Technologies, UK

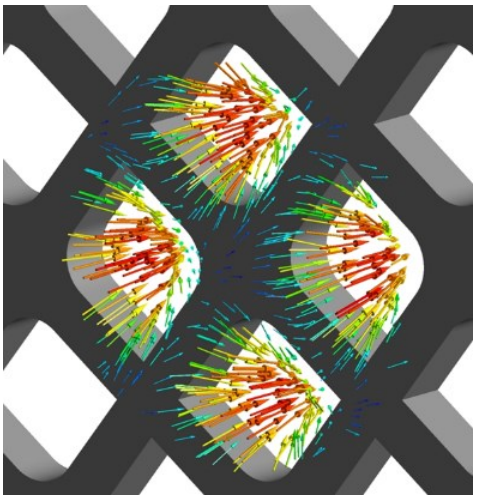
AUTOMOTIVE



Design optimisation of a dual stream engine exhaust system

Company: Ener-G, UK

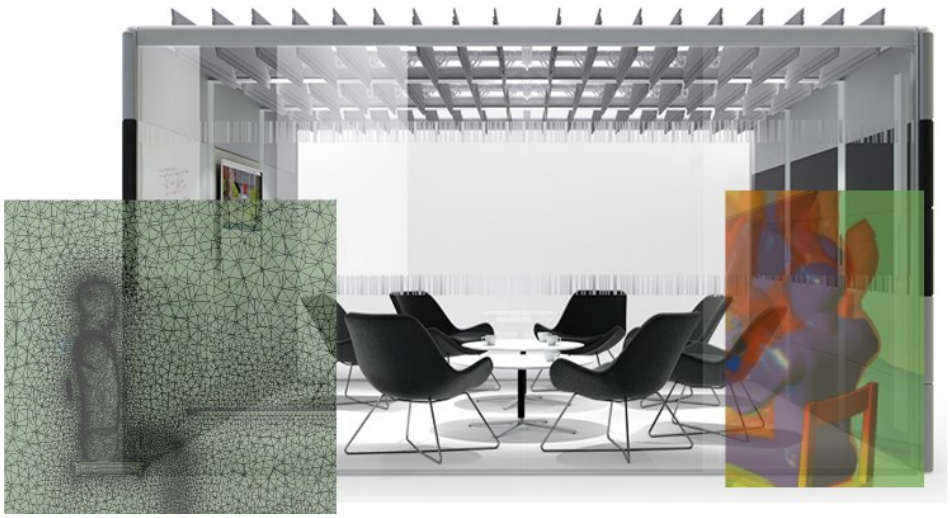
BUILDING SERVICES



Air flow in different lattices for building ventilation

Company: Native, UK

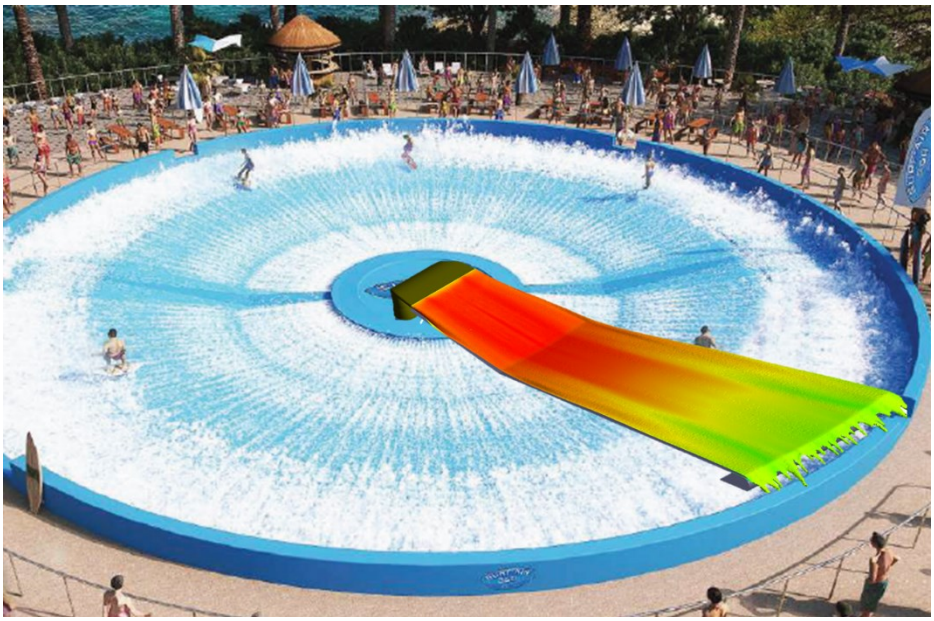
VENTILATION



Simulation and design optimisation of an innovative office pod and assessment against the latest standards

Company: OrangeBox, UK

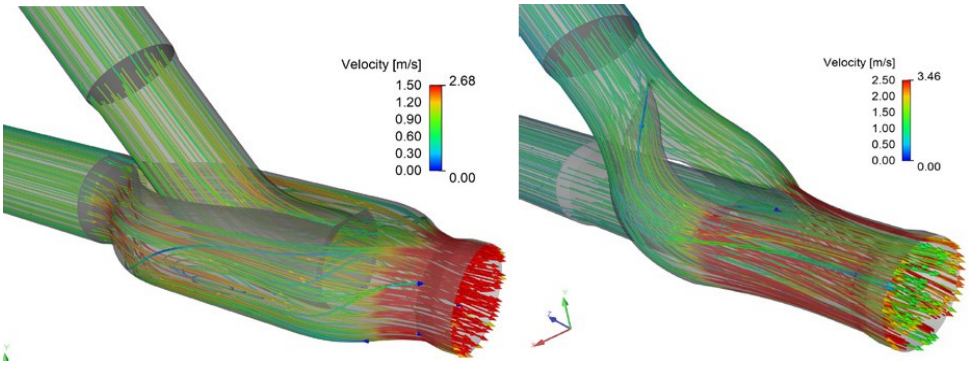
LEISURE



Design and simulation of a 360° Surf Rider

Company: Murphy Waves Ltd, UK

WATER&UTILITIES



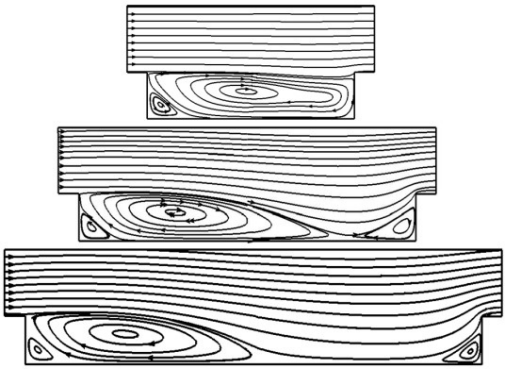
Design and optimisation of an innovative Y-Branch connection pipe to minimize head loss and waste deposition

Company: Engmaster Ltd, UK

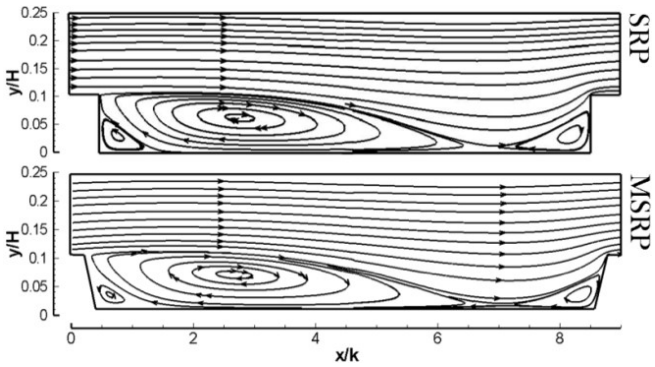
Project: Simulation of AGR Core Flows and Carbon Deposition

Company: British Energy/EDF Energy, UK

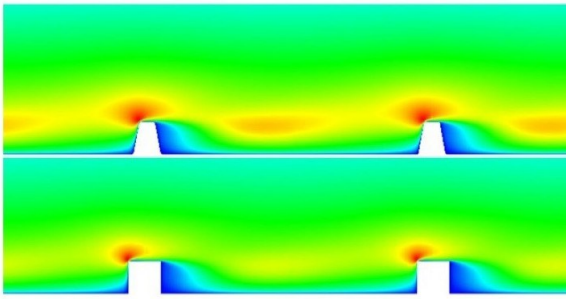
Brief Description: Advanced Gas-cooled Nuclear Reactors (AGRs) are the second generation of British gas-cooled reactors and there are currently seven of them in the UK, all owned and operated by EdF Energy. The core of AGRs consists of hundreds of fuel pins containing the nuclear fuel pellets. Thesurfaces of these fuel pins are rib-roughened to enhance heat transfer. One of the major operational problems with the AGR fuel pins is associated with the carbon particle deposition between the rib whichresults in heat transfer impairment and in turn higher fuel pin temperatures. The aim of this project wasto analyse the thermal-hydraulic effects of carbon depositions on fuel pins within AGR cores. These depositions represent a major operational problem and result in a reduction in AGR's efficiencies. The outcome of this project led to several journal publications and played an important role in British Energy's understanding of the problem and taking appropriate actions which led to a significant life extension of the AGR stations.



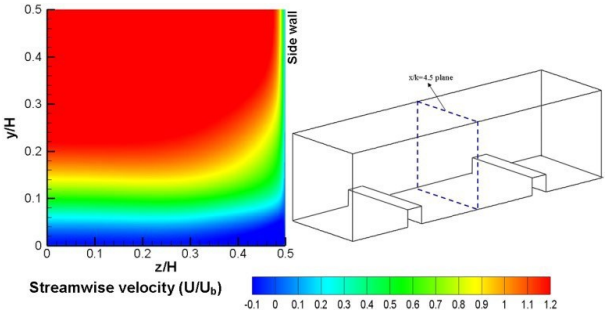
Effects of spacing between the ribs on flow and carbon deposition



Effects of carbon deposition around the rib



Turbulent Kinetic Energy distribution for 2 different configurations



Spanwise velocity distribution in the 3D ribbed channel

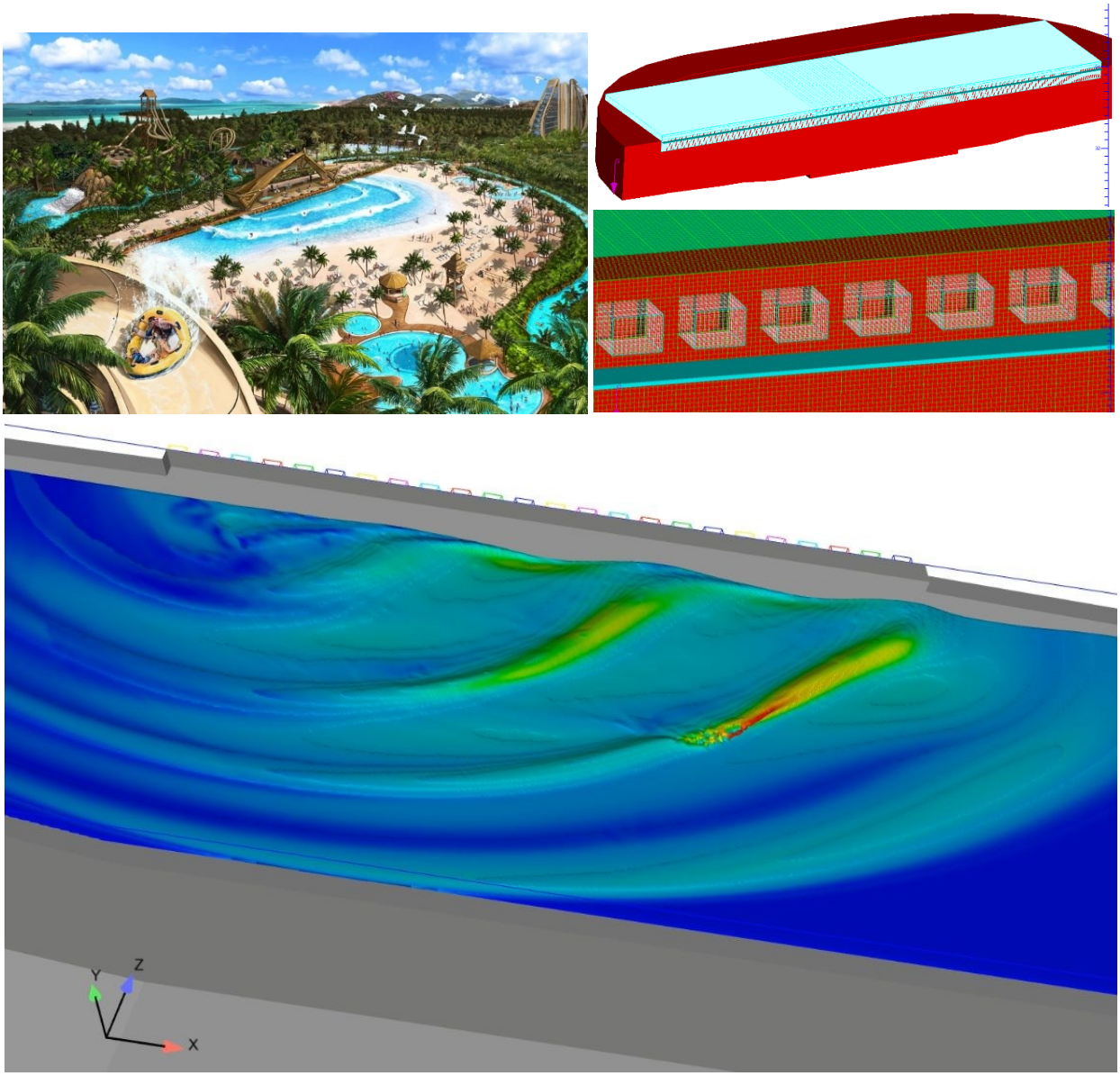
Relevant ManchesterCFD Publications:

- Boujelbene, M., Mahdi, J.M., Yaïci, W., Talebizadehsardari, P. & Keshmiri, A., 'Revolutionizing heat recovery in shell-and-tube latent heat storage systems: an arc-shaped fin approach', Eng. Applications of Computational Fluid Mechanics, 17(1), 2023. DOI: 10.1080/19942060.2023.2255036
- Xenakis, A., Ruiz-Soler, A. & Keshmiri, A., 'Multi-Objective Optimisation of a Novel Bypass Graft with a Spiral Ridge', J. Bioengineering, 10 (4), 489, 2023.
- Shahbazi, F., Jabbari, M., Nasr Esfahani, M., Keshmiri, A.. 'A Computational Simulation Platform for Designing Real-Time Monitoring Systems with Application to COVID-19', Biosens. Bioelectron. 171, 2021, 112716.
- Ruiz-Soler, A., Kabinejadian, F., Slevin, M.A., Bartolo, P.J. & Keshmiri, A. 'Optimisation of a Novel Spiral-Inducing Bypass Graft Using Computational Fluid Dynamics', Nature (Scientific Reports) 7:1865, 2017.
- McElroy, M., Ruiz-Soler, A. & Keshmiri, A., 'Left Ventricular Assist Devices: Impact of Flow Ratios on The Localisation of Cardiovascular Diseases Using Computational Fluid Dynamics', J. Procedia CIRP, Vol. 49, 2016, 163-169.
- Keshmiri, A., Ruiz-Soler, A., McElroy, M. & Kabinejadian, F., 'Numerical Investigation on The Geometrical Effects of Novel Graft Designs for Peripheral Artery Bypass Surgery', J. Procedia CIRP, Vol. 49, 2016, 147-152.
- Erfani, R., Keshmiri, A., Erfani, T. & Kontis, K., 'Multiple Encapsulated Electrode Plasma Actuators Effect on Aerofoil-Wake Interaction', AIAA Journals; Plasmadynamics and Lasers Series, 2013, 1-9, DOI: 10.2514/6.2013-2884.
- Keshmiri, A., 'Numerical Sensitivity Analysis of 3- and 2-Dimensional Rib-Roughened Channels', J. Heat and Mass Transfer, Vol. 48, 2012, 1257-1271.
- Keshmiri, A. & Gotts, J., 'Thermal-Hydraulic Analysis of Four Geometrical Design Parameters on Fuel Pins in AGR Core Flows', Int. J. Numerical Heat Transfer; Part A, Vol. 60, 2011, 305-327.
- Keshmiri, A., 'Effects of Various Physical and Numerical Parameters on Heat Transfer in Vertical Passages at Relatively Low Heat Loading', ASME J. Heat Transfer, Vol. 133, 2011, p. 092502-1.

Project: Design and Simulation of a Large Shore Breaker for Surfing

Company: Murphy Waves Ltd, UK

Brief Description: The aim of this project was to develop an accurate CFD model for a full-scale pneumatic wave generator which would create waves of up to 2m in height for surfing. Developing such a design by taking into account all the constraints and limitations of the mechanical component is extremely challenging and requires conducting a detailed and careful parametric study. Another challenge associated with such full-scale projects is the computational cost of running domains with tens of million of cells in a transient mode. This project utilised the ManchesterCFD team's exclusive High Performance Computing (HPC) facilities, which enabled running several simulations on hundreds of CPUs, resulting in a much shorter turnaround time, representing an essential aspect of this particular project. This successful design will be built in multiple locations in the world in the near future.



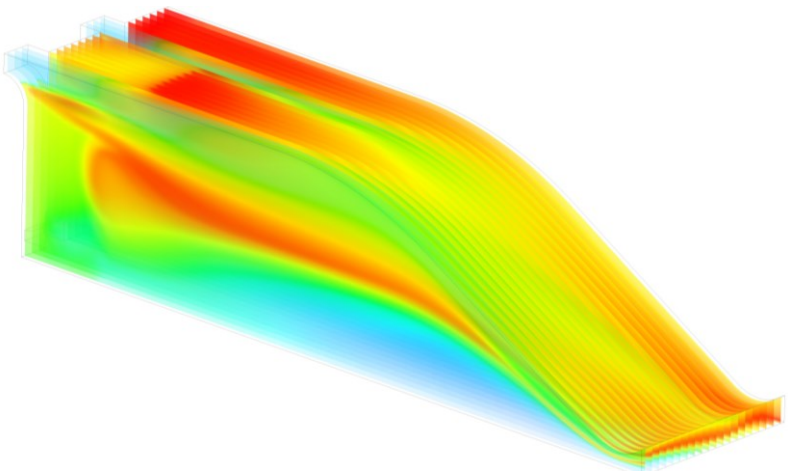
Full 3D simulations of the shore breaker with accurate prediction of wave formation and break

Project: Design and Optimisation of a Novel MEMS Microcombustor

Company: URA Thrusters, UK

Brief Description: The Iridium Catalysed Electrolysis CubeSat Thruster (ICE-Cube Thruster) is a novel micro-scale rocket engine which is being developed by URA Thrusters. The thruster utilises the propellants of hydrogen and oxygen produced by the electrolysis of water and is designed to produce a thrust of only 4.5mN. Designing a Micro-Electrical Mechanical Systems (MEMS) thruster to function at this scale is a unique challenge and requires a very different approach compared to the typical rocket engines. The ManchesterCFD team conducted a series of parametric studies consisting of geometrical and operational parameters using high-fidelity simulations and obtained critical design data including temperature, pressure and velocity in different parts of the combustion chamber, which enabled the company to successfully improve the design of the thruster. In addition, a bespoke combustion efficiency was developed and a series of reaction equations were implemented by the ManchesterCFD team in order to enhance the design and obtain the main combustion species.

The Iridium Catalysed Electrolysis (ICE) CubeSat Thruster



Velocity magnitude obtained using CFD in the proposed micro-combustor

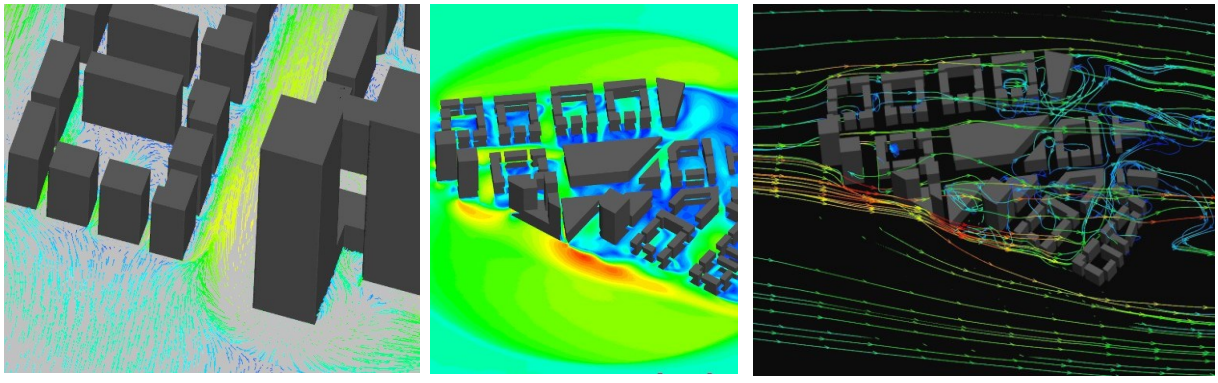
ENVIRONMENTAL IMPACT ASSESSMENTS

ENVIRONMENTAL

Modelling Urban Heat Island (UHI) effects in built-up districts and cities

Company: AECOM, UK

URBAN DESIGN



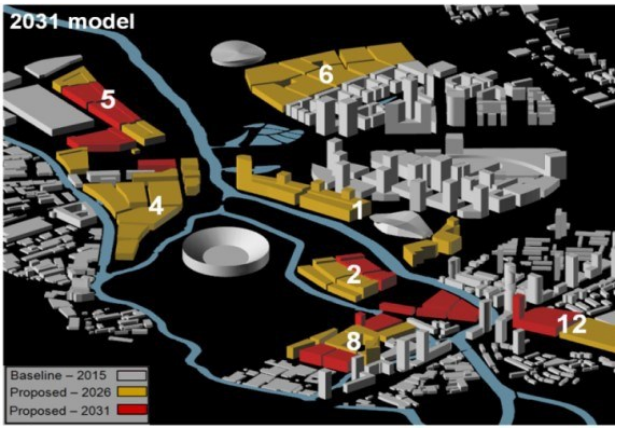
Wind simulation to assess pedestrian comfort in a built-up area

Company: Manchester City Council, UK

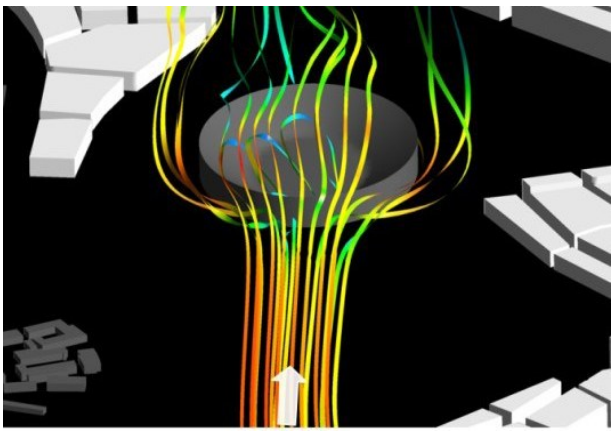
Project: Large Scale Ventilation and Wind Microclimate Analysis of London Legacy Development Masterplan

Company: London Legacy Development Corporation, UK

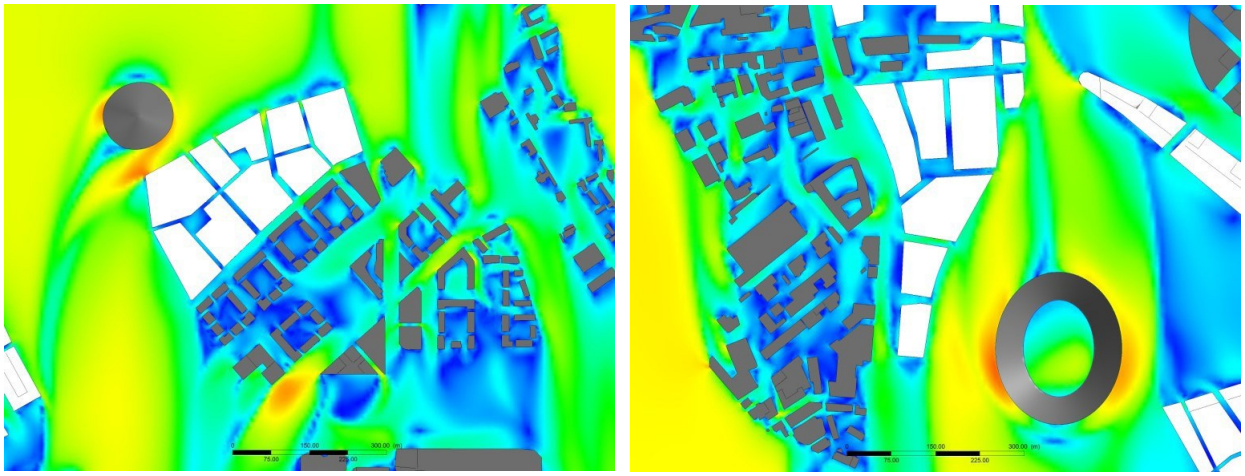
Brief Description: It is somewhat inevitable that, with the construction of new developments, the wind microclimate in the vicinity of the site(s) will be altered. Where new buildings are significantly different in size, form, orientation or height from those in the immediate vicinity, winds can be introduced which may cause discomfort to pedestrians. This project simulates the large scale ventilation and wind micro-climate around the Queen Elizabeth Olympic Park and its legacy master plan which includes numerousbuildings and developments across a large area. The computational model in this project represents one of the most complicated configurations ever attempted by computer simulations. The Team successfullysimulated this masterplan and provided important input into the urban design in this large part of London in order to minimise environmental impact and pedestrian discomfort.



Schematic of the Queen Elizabeth Olympic Park Legacy master plan



Streamlines within the main structures to identify locations to apply mitigating techniques against strong wind levels



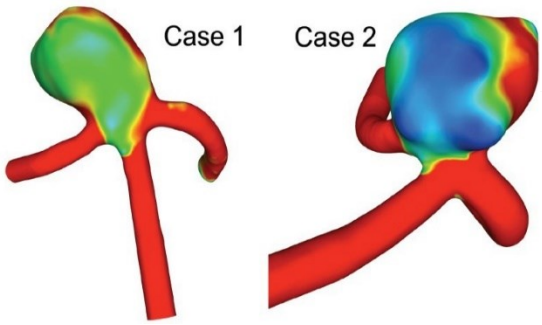
Large scale wind microclimate and extreme wind levels around several developments and structures

Relevant ManchesterCFD Publications:

- Hosseinzadeh, A., Bottacin-Busolin, A. & Keshmiri, A., 'A parametric study on the effects of green roofs, green walls and trees on air quality, temperature and velocity', Buildings, 12(12), 2159, 2022.
- Hosseinzadeh, A. & Keshmiri, A., 'The Role of Turbulence Models in Simulating Urban Microclimate', Advances in Heat Transfer and Thermal Engineering, 2021, 675-680.
- Hassani, A., Azapagic. A., D'Odorico, P., Keshmiri, A. & Shokri, N. 'Desiccation crisis of saline lakes: A new decision-support framework for building resilience to climate change', Science of the Total Environment, 703, 2020, 134718.
- Hosseinzadeh, A., Shokri, N. & Keshmiri, A, 'The Role of Turbulence Models in Simulating Urban Microclimate', Proceedings of the 16th UK Heat Trans-fer Conference (UKHTC2019), 8-10 Sep 2019, Nottingham, UK.

BIOMEDICAL ENGINEERING

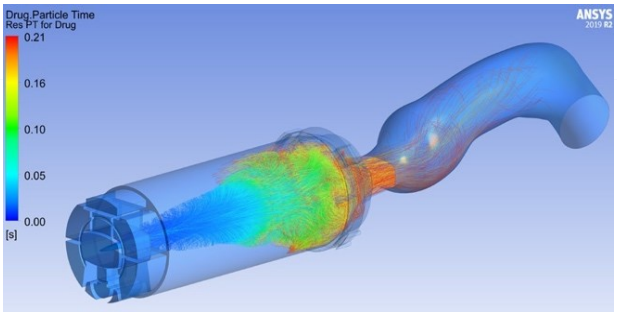
MEDICAL DEVICE



Development of a new biocompatible treatment for intracranial aneurysms

Company: Cardiovascular Institute of Cataluña, Spain

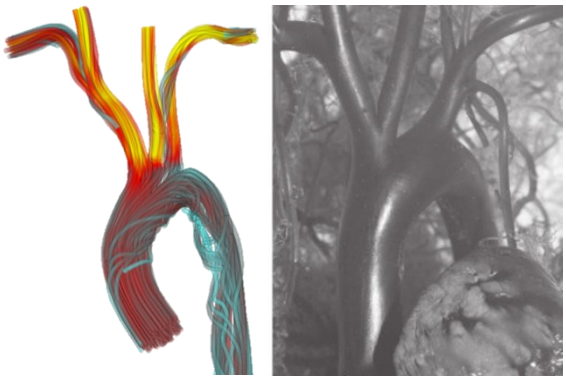
MEDICAL DEVICE



Simulation of a Valved Holding Chamber (HVC) to be used with inhalers

Company: Inhaler+, UK

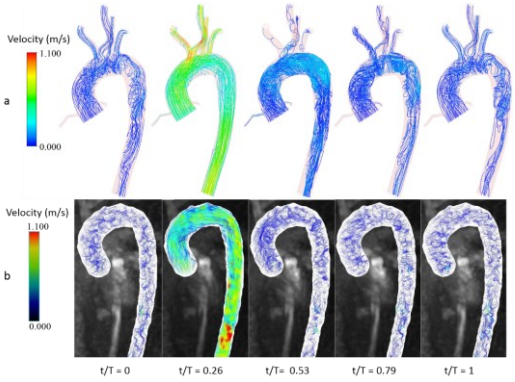
CARDIOVASCULAR



Predicting regions prone to atherosclerosis in animal models using CFD

Company: Manchester Royal Infirmary, UK

CARDIOVASCULAR



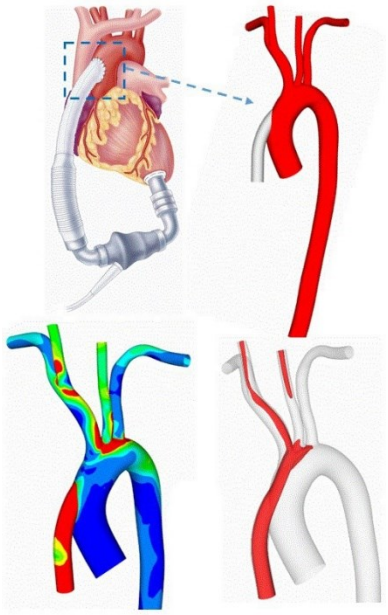
Modelling Atrial Fibrillation (AF) using CFD and comparison against 4D MRI

Company: Manchester Royal Infirmary, UK

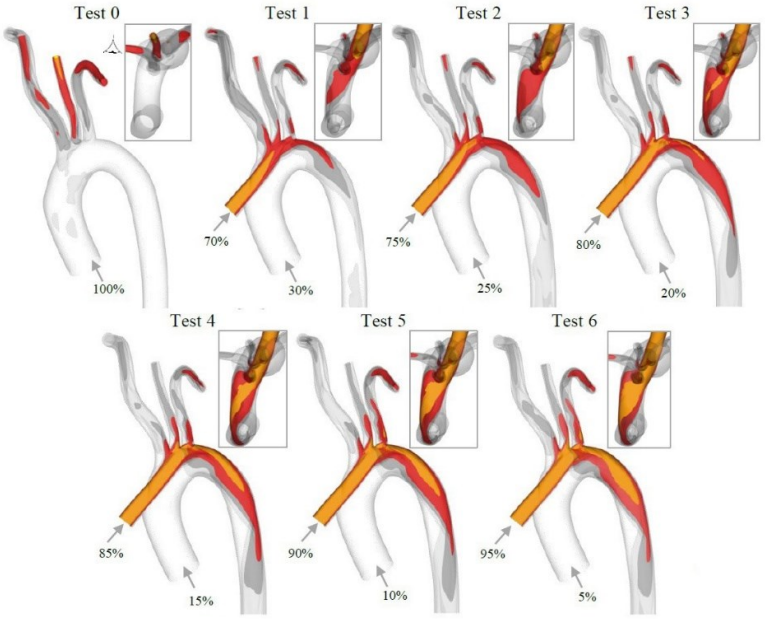
Project: Optimisation of a Left Ventricular Assist Device Cannula

Company: Medical Simulation Technologies Limited, UK

Brief Description: Heart failure is the leading cause of hospitalisation in people over 65. A left ventricular assist device (LVAD) is an option to provide mechanical circulatory support as a 'bridge to cardiactransplantation' or 'destination therapy'. The aim of this project was to assess the effects of different LVAD connection on the blood flow in order to find the optimum cannula connection. The other aim of this work was to assess the effects of LVAD on the severity of heart failure. In this project, a patient specific configuration, obtained by MRI was converted into a computational model and accurate CFD simulations were carried out for different degree of heart failure. This project resulted in important findings about the surgical configuration and the design of LVAD cannulas.



The ventricular assist device configuration



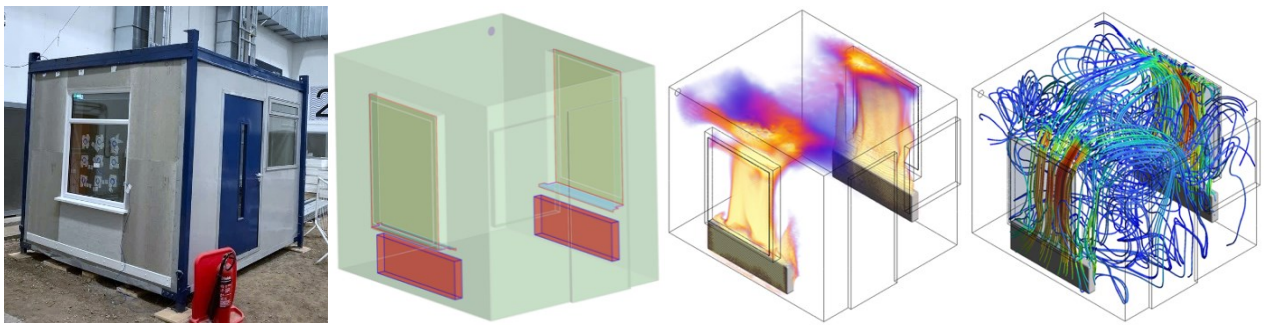
Time-Averaged Velocity (VAT) distribution for different heartfailure severity

Relevant ManchesterCFD Publications:

- Deyranlou, A., Revell, A. & Keshmiri, A., 'Exergy Destruction during the Atrial Fibrillation, an Index for the Heart Ageing', J of Theoretical Biology, 111623, 2023.
- Xenakis, A., Ruiz-Soler, A. & Keshmiri, A., 'Multi-Objective Optimisation of a Novel Bypass Graft with a Spiral Ridge', J. Bioengineering, 10 (4), 489, 2023.
- Lopez-Santana, G.B., De Rosis, A., Grant, S.W., Venkateswaran, R. & Keshmiri A., 'Computational Fluid Dynamics as a Surgical Tool to Optimise the Positioning of the LVAD Outflow Graft for Reducing Aortic Regurgitation', The J. Heart and Lung Transplantation 42 (4), S416, 2023.
- Hosseini, S.E. & Keshmiri, A., 'Experimental and numerical investigation of different geometrical parameters in a centrifugal blood pump', Research on Biomedical Eng., 2022, DOI : 10.1007/s42600-021-00195-8.
- McElroy, M., Kim, Y., … Gijsen, F, Johnson, T., Keshmiri, A., White, S., 'Identification of the haemodynamic environment permissive for plaque erosion', Nature (Scientific Reports), 11:7253, 2021.
- Deyranlou, A., Naish, J.H., Miller, C.A., Revell, A. & Keshmiri, A., 'Numerical Study of Atrial Fibrillation Effects on Flow Distribution in Aortic Circulation', Annals of Biomedical Eng., 48 (4), 2020, 1291-1308.
- Shahbazi, F., Souri, M., Jabbari, M. & Keshmiri, A.. 'Flow Control Techniques for Enhancing the Bio-Recognition Performance of Microfluidic-Integrated Biosensors', Applied Sciences, 11, 2021.
- Shahbazi, F., Jabbari, M., Nasr Esfahani, M., Keshmiri, A.. 'A Computational Simulation Platform for Designing Real-Time Monitoring Systems with Application to COVID-19', Biosens. Bioelectron. 171, 2021, 112716.
- Satta, S., McElroy, M., Langford-Smith, A.W., Keshmiri, A., et al. 'High-Level Nrf2 Activation Promotes Endothelial Detachment-Implications for Acute Coronary Syndromes Triggered by Endothelial Erosion of Plaques', Journal of Vascular Surgery, 56, 2019, 1-71.
- McElroy, M. & Keshmiri, A., 'Impact of Using Conventional Inlet/Outlet Boundary Conditions on Haemodynamic Metrics in a Subject-Specific Rabbit Aorta', Journal of Engineering in Medicine, 232(2), 2018, 103-113.
- Ruiz-Soler, A., Kabinejadian, F., Slevin, M.A., Bartolo, P.J. & Keshmiri, A. 'Optimisation of a Novel Spiral-Inducing Bypass Graft Using Computational Fluid Dynamics', Nature (Scientific Reports) 7:1865, 2017.
- Kabinejadian, F., Ruiz-Soler, A., McElroy, M., Leo, H.L., Slevin, M., Badimon, L. & Keshmiri, A., 'Numerical Assessment of Helical/Spiral Grafts to Improve the Hemodynamics of Distal Graft Anastomoses', Plos One, Vol. 11 (11), 2016, 1-22.
- McElroy, M., Ruiz-Soler, A. & Keshmiri, A., 'Left Ventricular Assist Devices: Impact of Flow Ratios on The Localisation of Cardiovascular Diseases Using Computational Fluid Dynamics', J. Procedia CIRP, Vol. 49, 2016, 163-169.
- Keshmiri, A., Ruiz-Soler, A., McElroy, M. & Kabinejadian, F., 'Numerical Investigation on The Geometrical Effects of Novel Graft Designs for Peripheral Artery Bypass Surgery', J. Procedia CIRP, Vol. 49, 2016, 147-152.

NET-ZERO & SUSTAINABILITY

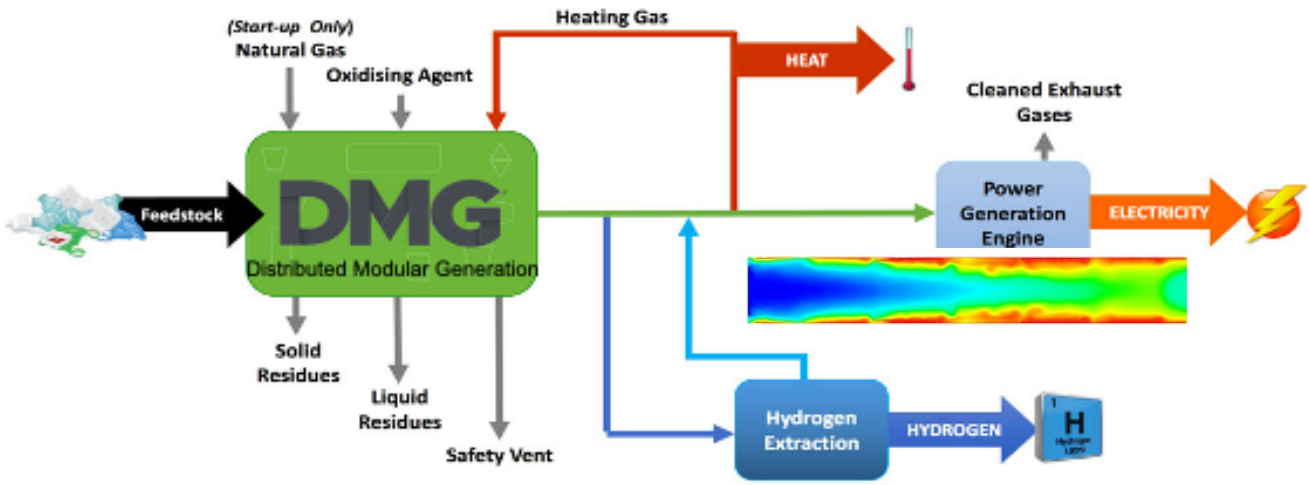
HEAT DECARBONISATION



CFD and experimental analysis of decarbonising heat in homes

Company: Salford Energy House, UK

HYDROGEN SEPARATION



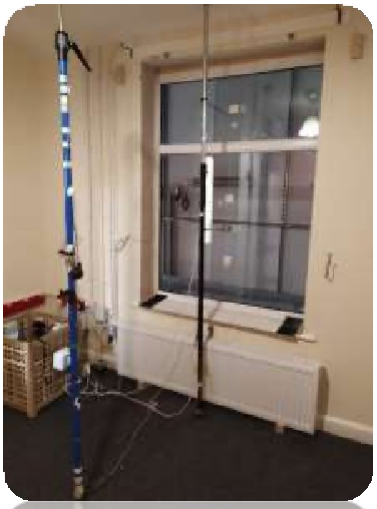
Hydrogen separation/extraction from a recycling waste

Company: Powerhouse Energy Plc, UK

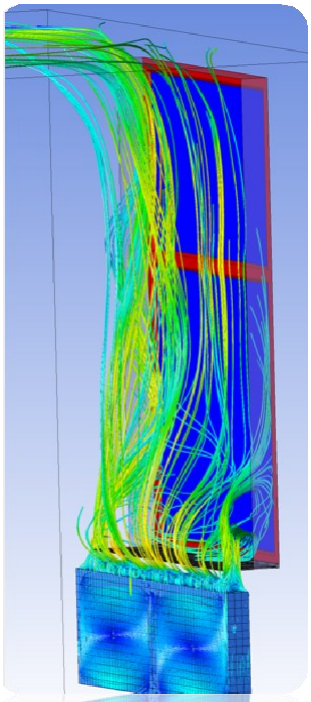
Project: Simulation, Validation and Design Optimisation of *Thermocill™*

Company: Thermocill, UK

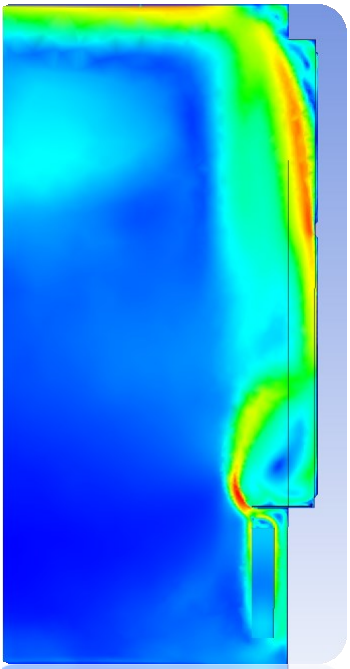
Brief Description: Thermocill is an energy savings product that is designed for installation under the window board and above the radiator in a room. It is made from recycled plastic materials and can be retrofittedto existing homes as well as new builds. In its operation, the product tends to direct the natural convectionfrom the radiator to create a wall of warm air immediately in front of the internal side of the glazed window. In this project, the performance of the Thermocill was investigated based on the previously designed experimental condition. The main aim of the CFD simulation was to assess Thermocill's effectiveness in a typical bedroom consisting of a radiator, a double-glazed window and a door. Several high-fidelity and transient simulations were conducted to reproduce the experimental conditions and to provide an insight into the details of Thermocill's operation, particularly near the window. Subsequently, a design optimisation on several geometrical aspects of Thermocill were carried out. The CFD simulations successfully reproducedthe experimental data and led to major changes in Thermocill's design. Products such as Thermocill play an important role in decarbonisation of heating in homes in the UK, which is one of the priorities of the UK government in achieving net-zero emissions.



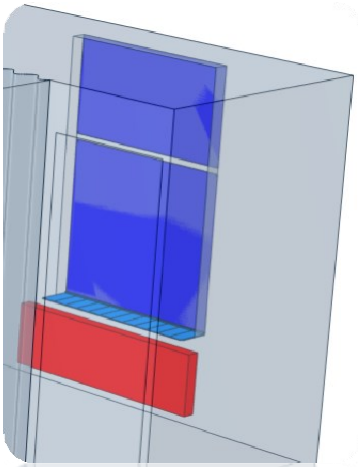
Experimental setup



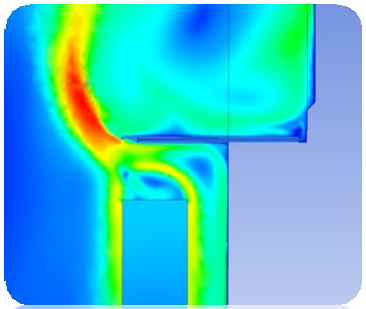
Velocity streamlines around Thermocill



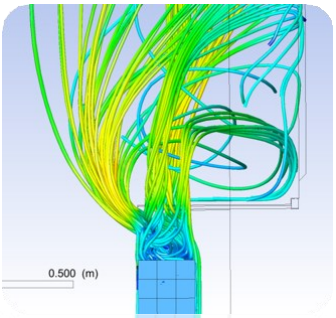
Computational simulationsof the hot air velocity and formation of a thermal layer on the window



Computational Model



Velocity Distribution

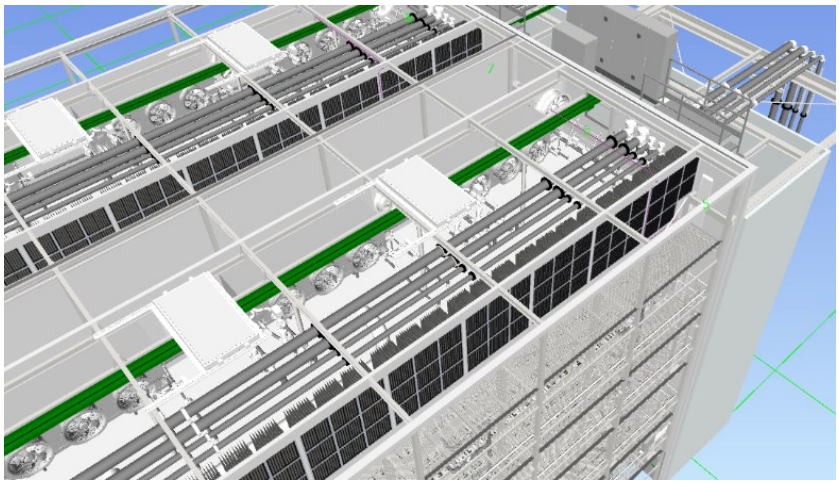


Velocity Streamlines

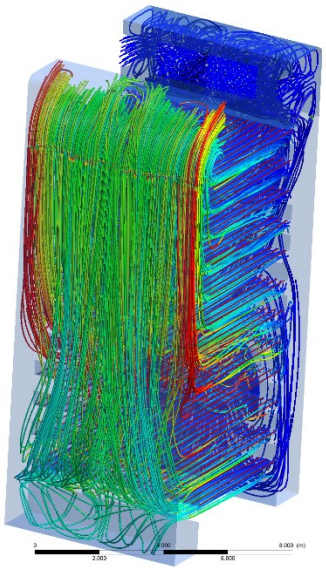
Project: Simulation of a Large Vertical Farming Unit

Company: Grow Up Farms Limited, UK

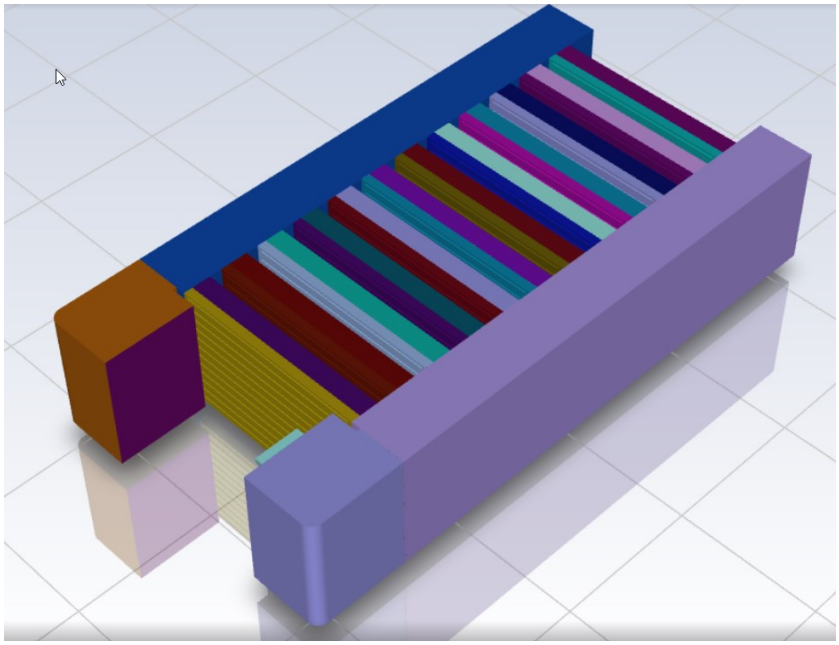
Brief Description: The aim of this project was to develop a detailed modular numerical framework to simulate the flow and heat transfer in a large-scale vertical farming unit, also known as a climate cell. The numerical model was developed to allow flexibility and scalability and to assess the impact of various parameters on the performance of the climate chamber. Following a successful development of the model and running a series of simulations, the effect of heat transfer, velocity and pressure on the crops were well captured and presented qualitatively by contours, streamlines and vectors and quantitatively by averaged values and uniformity index. The simulation carefully accounted for radiation, convention, buoyancy, moisture, turbulence and porosity of the crops, and provided a great tool for any type of design optimisation and assessment for the client. Vertical farming including this state-of-the-art facility plays an important role in achieving a sustainable future/net-zero and significantly contributes to food security.



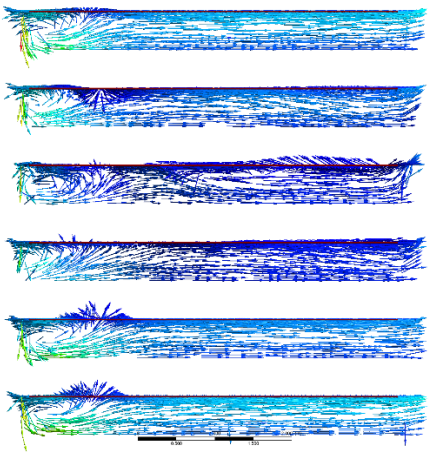
The detailed 3D model of the full-scale unit



Velocity contour in sections of the climate cell



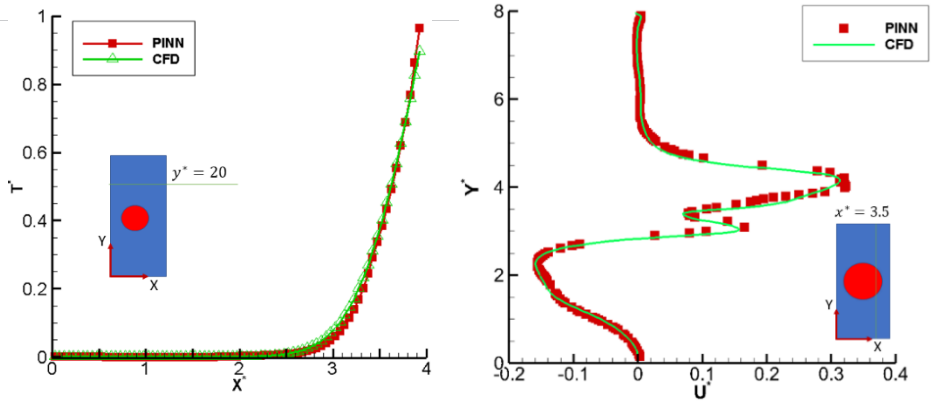
The generated 3D model for the simulations

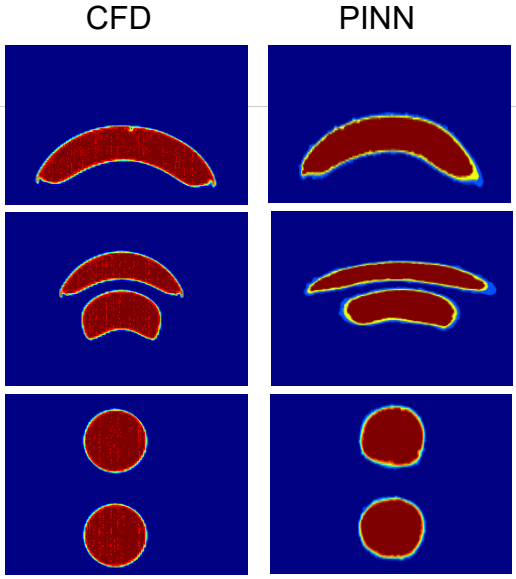


Detailed air flow patterns in different levels of the climate cell

ARTIFICIAL INTELLIGENCE & MACHINE LEARNING

Physics-Informed Neural Network (PINN)



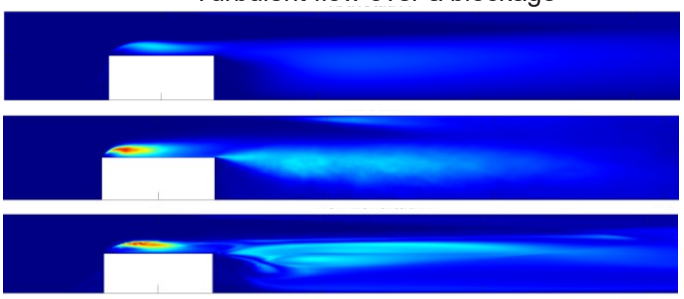


Coaxial coalescing

ManchesterCFD team has developed novel machine learning algorithms, known as PINN, capable of replacing conventional CFD techniques through a comprehensive training dataset, therefore reducing the computational time by a few orders of magnitude. This is ideal for parametric and optimisation studies.

Tensor Basis Neural Network (TBNN)

Turbulent flow over a blockage

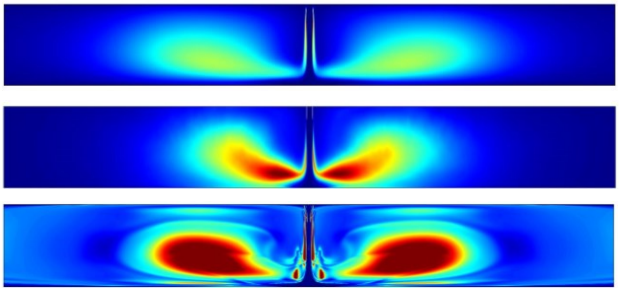


Low Resolution CFD

High Resolution CFD

TBNN

Turbulent flow for impingement jet



ManchesterCFD team has developed another technique known as Tensor Basis Neural Network (TBNN) similar to PINN is able to predict the flow and heat transfer in different flow problems, obtaining great accuracy but at a fraction of time compared to conventional CFD techniques. Such techniques are currently applicable to simple case studies, but they are currently being developed to work for a wide range of problems in near future, again ideal for design optimisation purposes.

Relevant ManchesterCFD Publications:

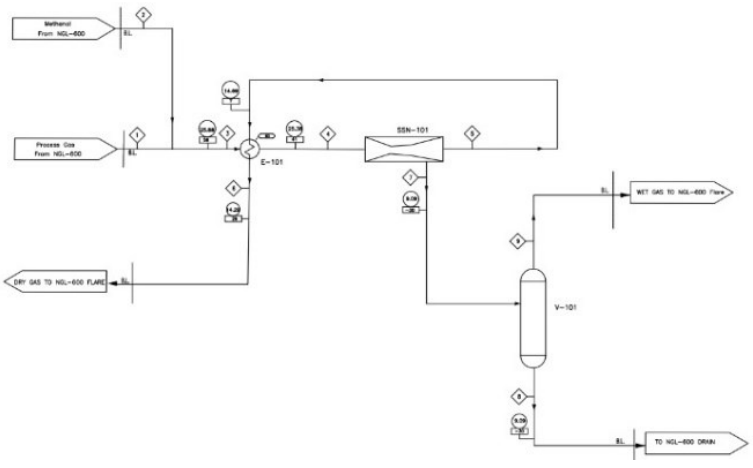
- Jalili, D., Jan, S., Jadidi, M., Giustini, G., Keshmiri, A., & Mahmoudi, Y., 'Physics-Informed Neural Networks for Heat Transfer Prediction in Two-Phase Flows', Int. J. Heat Mass Transfer, 2023.
- Man, A., Jadidi, M., Keshmiri, A., Yin, H., & Mahmoudi, Y., 'A Divide-and-Conquer Machine Learning Approach for Modelling Turbulent Flows', Physics of Fluids 35 (5), 2023.
- Man, A., Jadidi, M., Keshmiri, A., Yin, H. & Larimi, YM., 'Optimising a Machine Learning Model for Reynolds Averaged Turbulence Modelling of Internal Flows', 16TH International Conference on Heat Transfer, Fluid Mechanics and thermodynamics, 2022.

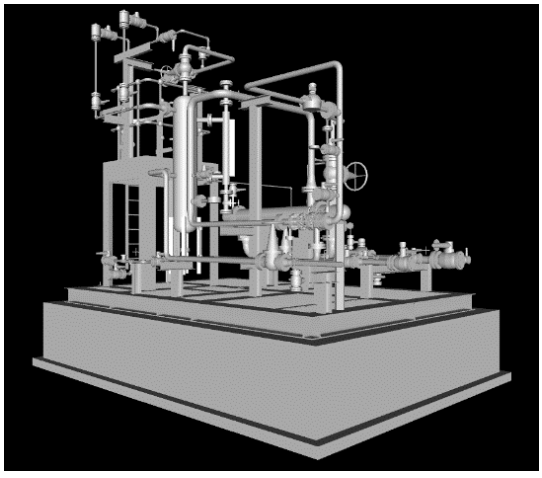
DESIGN OF CHEMICAL PROCESSES & FEED STUDY

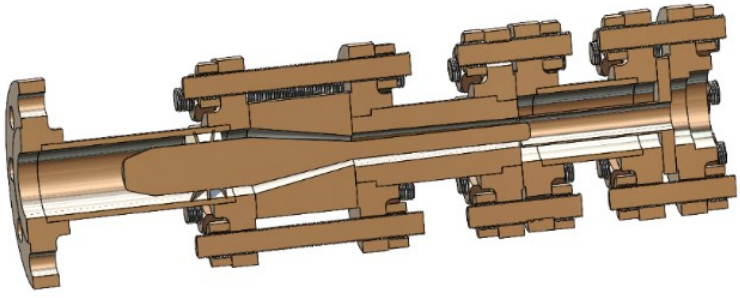
Project: Process Simulation and Technoeconomic Analysis of a Gas Dehydrator

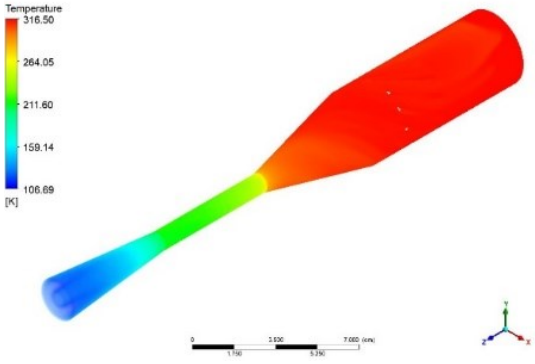
Company: Hydrogennovate, UK

Brief Description: The natural gas should be processed before deployment to the transmission pipelines to set the dewpoint and remove the impurities. This project professes a new process for setting both the water and hydrocarbon dewpoint of the natural gas which needs at worst case scenario 40% of energy in conventional methods. This project consisted of PFD and P&ID Preparation and included equipment, design, sizing and design optimisation as well as HAZOP and HAZOP+ Risk assessment.



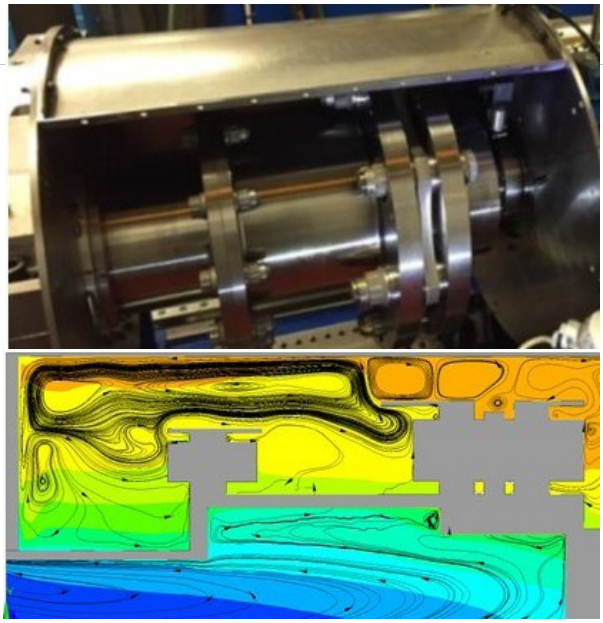






EXPERIMENTS & VALIDATION

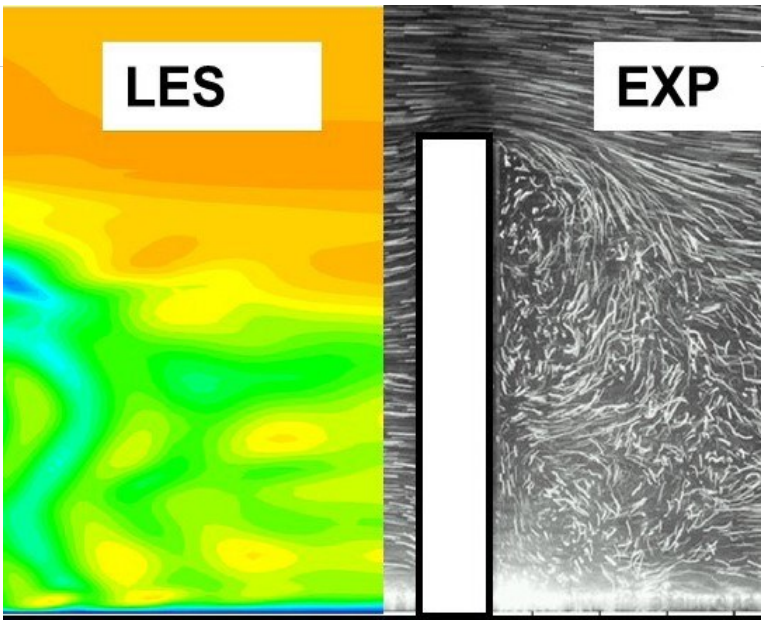
AUTOMOTIVE



Modelling of automotive coupling systems andvalidation against experimental data

Company: John Crane, UK

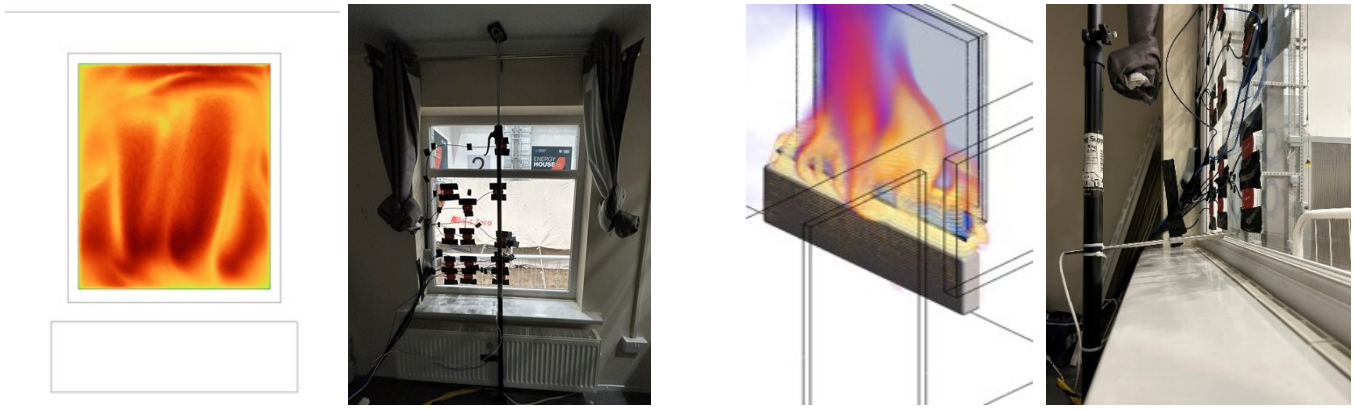
ENERGY



Computational and experimental analysis of heat exchanger baffles

Company: Couette Ltd, UK

ENERGY



Calculation of Heat losses under different conditions through different windows using experiments and CFD

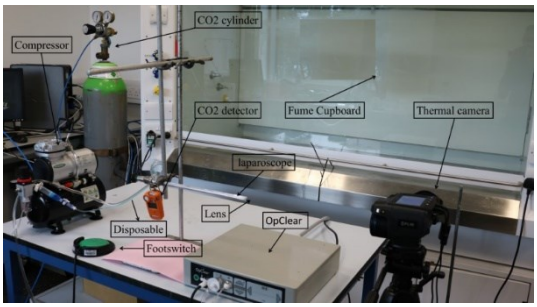
Company: Building Research Establishment (BRE) Group, UK

Project: Optimisation of a Disposable Design for Cleaning Laparoscope Lenses

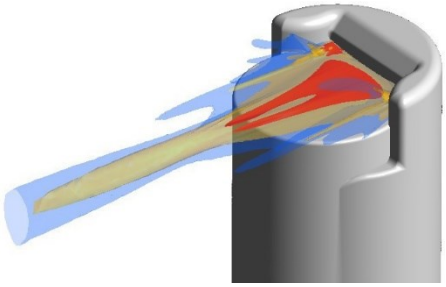
Company: Cipher Surgical, UK

Brief Description: Laparoscopic surgery requires continuously clear vision throughout a procedure. Cipher Surgical has designed a unique device that will clear a laparoscope lens of all fogging, soiling and fluid without removing the scope from the patient for cleaning. The device is a single use disposable sheath and tube set that attaches to the laparoscope. The Disposable is connected to the OpClear® Control Unit that directs carbon dioxide (CO2) and saline into the sheath and across the tip of the laparoscope. ManchesterCFD has conducted several computational simulations and have validated their results using different experimental methods with an aim of improving the performance of the disposable and design new types of disposable to fit on a wide range of laparoscopes. The simulations in this project required meticulous assessment of the saline and CO2 with accurate boundary conditions. Different image processing techniques used to provide a validation tool for the computational simulations.

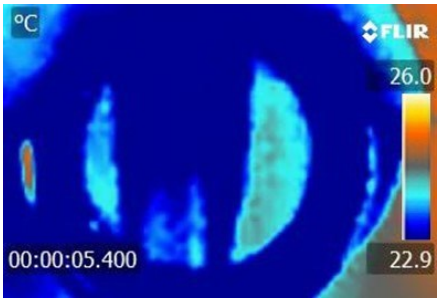
Schematic of the product modelled and optimised by the ManchesterCFD team



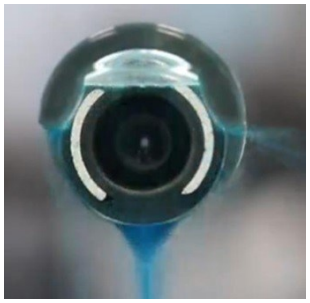
The experimental setup used for validation tests



Computational simulations showing velocity iso-surfaces on the surface of the lens



Flow pattern analysis using thermal cameras



Flow analysis using high-speed cameras

High-resolution image processing techniques using customised computer codes

Relevant ManchesterCFD Publications:

- Lopez-Santana, G.B., Kennaugh, A., & Keshmiri, A. 'Experimental techniques against RANS method in a fully developed turbulent pipe flow: Evolution of experimental and computational methods for the study of turbulence', Fluids, 2022.
- Hosseini, S.E. & Keshmiri, A., 'Experimental and numerical investigation of different geometrical parameters in a centrifugal blood pump', Research on Biomedical Eng., 2022, DOI : 10.1007/s42600-021-00195-8.
- Keshmiri, A., Osman, K., Benhamadouche, S. & Shokri, N., 'Assessment of Refined RANS Models against Large Eddy Simulation and Experimental Data in the Investigation of Ribbed Passages with Passive Heat', Int. J. Numerical Heat Transfer; Part B, Vol. 69(2), 2016, 96-110.
- Thompson, A., Zhai, T., Palazzolo, A. & Keshmiri, A., 'Coupling Guard Temperature and Windage Power Loss: CFD Analysis and Experiments', Proc. 45th Turbomachinery and 32nd Pump Symposia, Houston, Texas, 12-15 Sep 2016.
- Keshmiri, A., Uribe, J. & Shokri, N., 'Benchmarking of Three Different CFD Codes in Simulating Natural, Forced and Mixed Convection Flows', Int. J. Numerical Heat Transfer; Part A, Vol. 67(12), 2015, 1324-1351.
- Keshmiri, A., 'Verification and Validation of three Different CFD Codes in Simulating Natural and Mixed Convection Flows using Two Advanced Eddy-Viscosity Models', Proc. 15th Int. Heat Transfer Conference, 'IHTC-15', Kyoto, Japan, 10-15 Aug 2014, pp. 1-14.
- Darabkhani, H.G., Oakey, J., Zhang, Y. & Keshmiri, A., 'Study of the Flame Structure and Dynamics Using Non-intrusive Combustion Diagnostic Techniques', AIAA Journals; Fluid Dynamics Series, 2013, 1-8, DOI: 10.2514/6.2013-2605.

INNOVATIVE COMMERCIAL PRODUCTS

Project: Development of a Revolutionary Bypass Graft (HeliRidgeGraft)

Brief Description: This multidisciplinary project led by ManchesterCFD team has integrated fluid mechanics, biomechanics and biology with cardiovascular surgery to develop a ground-breaking biomedical device, inspired by the nuclear engineering sector. Instead of conventional straight tubes, this novel graft design is based on inducing rotational/helical flow in the bypass graft through a combination of out-of-plane helicity and an internal ridge. The induced helical flow significantly enhances the hemodynamic condition around the junction, hence improving the longevity of the bypass operation. This design improves the longevity of the graft as it tends to retain the natural 'spiral flow' in the artery which is caused by the rotational compressive pumping of the heart and supported by the tapered, curved andnon-planar geometry of the arterial system. This project attracted global interest and has received a number of prestigious awards.

The New Zealand Herald
Artificial arteries a game changer for heart surgery - research

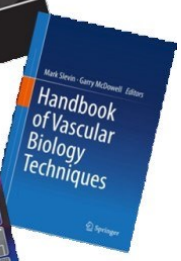
BT
New hope for 'gun barrel' arteries
Artificial arteries designed like gun barrels could revolutionise heart bypass surgery and save the NHS millions, researchers claim.

BBC RADIO

Daily Mirror
Gun-barrels op 'can save NHS millions'

Manchester Evening News
Manchester scientists' heart surgery breakthrough will help patients and 'save the NHS millions'
Experts at Manchester Metropolitan University say longer-lasting 'grafts' could benefit 20,000 people a year

Guernsey Press
New hope for 'gun barrel' arteries



The Press and Journal
Gun barrel can save millions

New hope in 'gun barrel' arteries
The Daily Telegraph

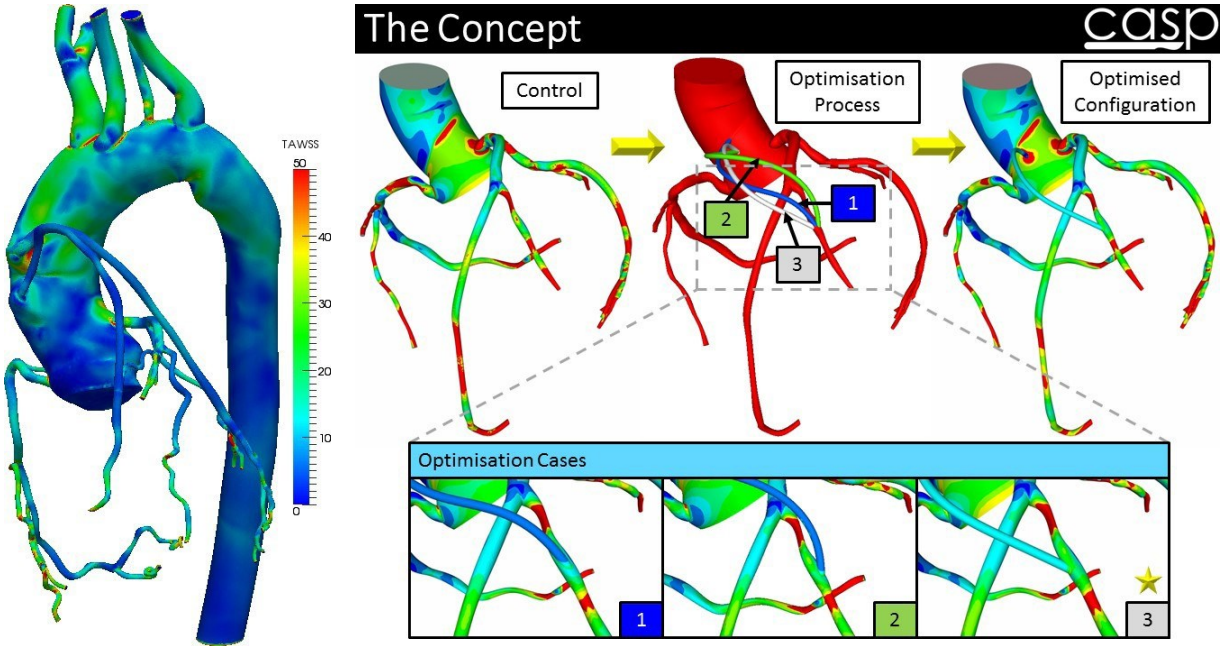
YAHOO! NEWS
New hope for 'gun barrel' arteries

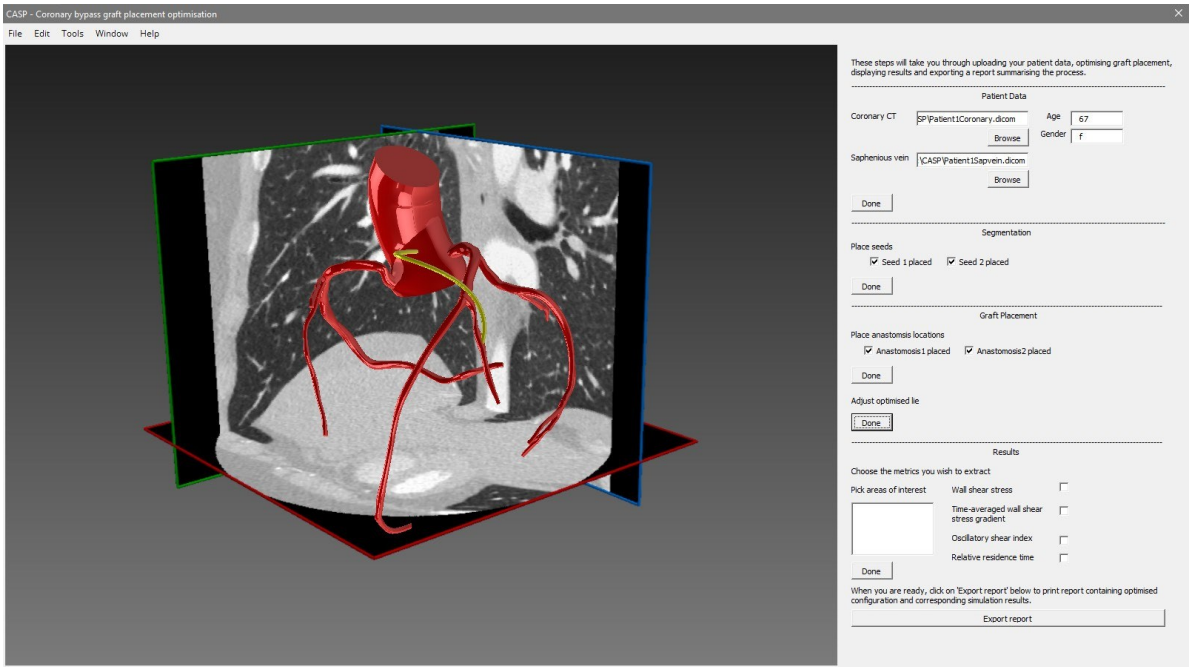
THE AUSTRALIAN
THE HEART OF THE NATION
New hope in 'gun-barrel' arteries

www.HeliRidgeGraft.com

Project: Developing the 'First Computer-Aided Surgical Planning' (CASP) Platform

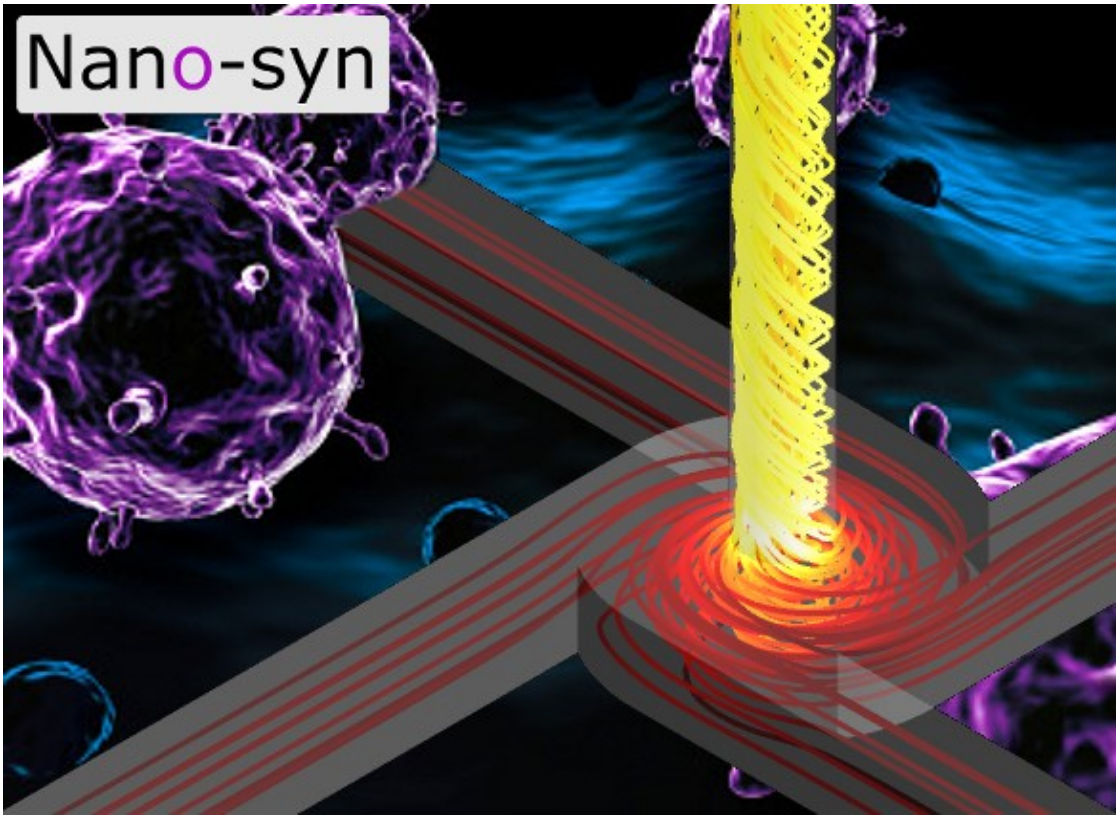
Brief Description: ManchesterCFD has developed a computer software called CASP that uses advanced CFD and medical image processing tools to assist cardiologists and cardiothoracic surgeons with daily clinical practice when planning the optimal placement of grafts used during heart bypass surgery. CASP Technologies Limited was set up in 2018, as a University spinout from research carried out by ManchesterCFD in collaboration with University Teaching Hospital Trust. The software enables tailored treatment for patients and significantly improves the clinical outcomes. This project has so far received grants from prestigious funding organisations such as UK Research Councils (MRC & EPSRC) and has attracted a lot of interest from the medical and engineering communities.

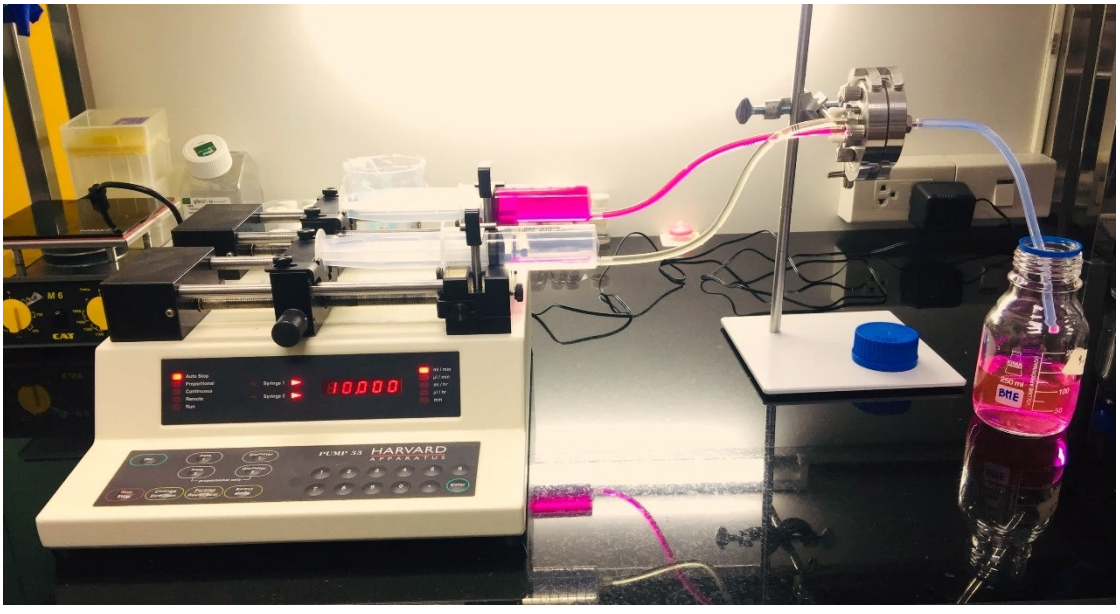




Project: Nanoparticle Synthesis for Antimicrobial Resistant Infection Treatment (NANO-SYN)

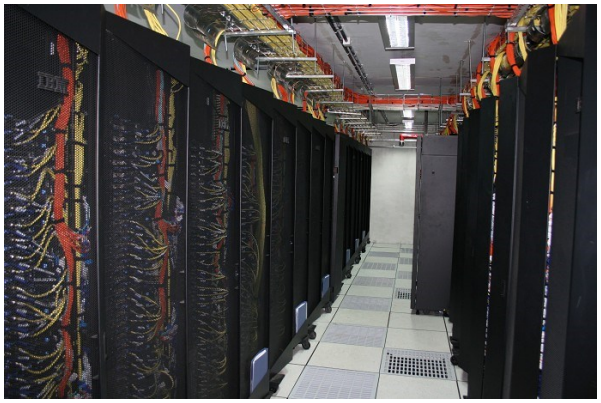
Brief Description: In collaboration with researchers from Oxford University Clinical Research Unit (OUCRU) in Vietnam and Prince of Songkla University (PSU) in Thailand, ManchesterCFD team led a novel project with an aim of developing technologies to make synthesis of nanoparticles for pharmaceutical applications. This was achieved through the design and testing of a new macroscale Multi-Inlet Vortex Reactor (MIVR) which could generate large quantities of functional nanoparticles in a both efficient and economical way. Given the commercialisation potential of this project, all 3 universities involved as well as the industrial stakeholder in both Thailand and the UK formed a spin-out (Nano-Syn Limited).



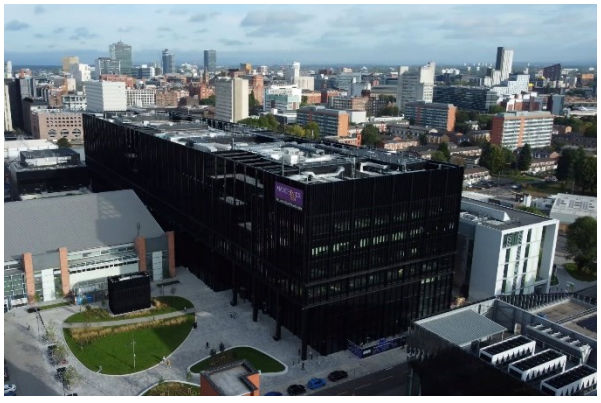


EXPERIMENTAL FACILITIES & TRAINING

In addition to extensive High Performance Computing (HPC) facilities, our group also has exclusive access to wind tunnel facilities at the University of Manchester, which cover the speed range from subsonic to hypersonic flow. Our team also has access to the instrumentation and technical skills to undertake both force, and high-quality point, surface, and field measurements for various aerodynamics applications. We are based at one of the largest engineering complex in Europe, which supports our world-class research.



The High-Performance Computing facilities available to ManchesterCFD



The Engineering Building which hosts ManchesterCFD

The Boundary Layer Tunnel

The “Bob” Wind Tunnel

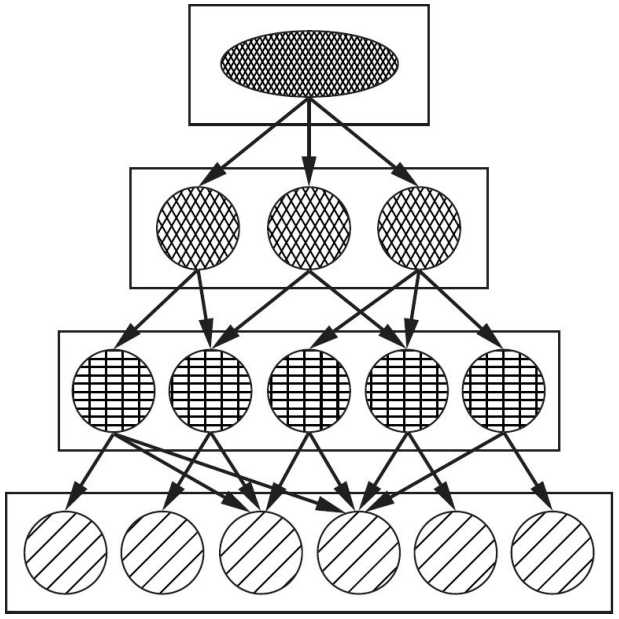
ManchesterCFD has organised and attended numerous national and international scientific workshops and events in the broad area of CFD and its industrial applications. In addition, ManchesterCFD team offers a range of intensive and customised training courses for companies in various simulation technologies. Some of the software and codes used by our team for both consultancy and training are as follows:

- Ansys Fluent
- Ansys CFX
- Ansys Mechanical
- STAR-CCM+
- OpenFOAM
- COMSOL
- DualSPHysics
- FLOW3D
- ASPEN+

Examples of previous conferences, training and workshops attended or organised by ManchesterCFD in different countries.

Building-Block Approach:

Because of the infeasibility and impracticality of conducting true validation experiments on most complex systems, the standard approach adopted by ManchesterCFD for quality assurances is to use a 'building- block approach', as shown below. In this approach, we normally break down a complex engineering system of interest into three, or more, progressively simpler tiers, to ensure accuracy in our results and estimate errors/uncertainty. The approach is clearly constructive in that it recognises that there is a hierarchy of complexity in systems and simulations. It also recognises that the quantity and accuracy of information that is obtained from experiments varies radically over the range of tiers.



Tier1: 'Complex System'
(e.g. Reactor, Engine, Boiler, Atmospheric Tank, Cardiovascular system, etc)

Tier2: 'Sub-System'
(e.g. parts, sections, elements, organs, etc)

Tier3: 'Benchmarking/Validation Cases' (e.g. heat transfer in a pipe, turbulent mixingin a vessel, multi-phase flow in a channel,chemical reaction in a combustor, blood flowin an artery, etc)

Tier4: 'Unit Problem'
(e.g. Navier-Stokes equation, energy equation, reaction equations, etc)

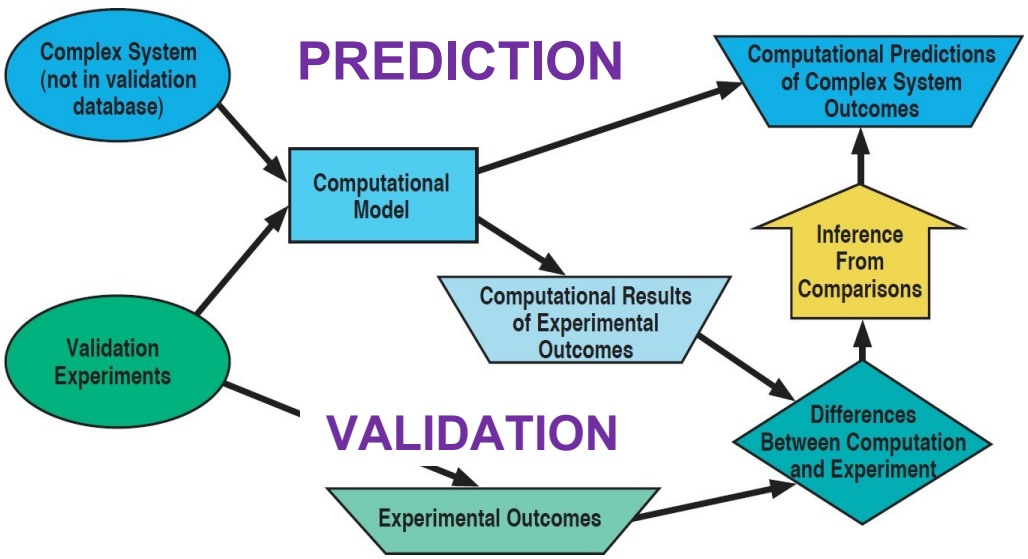
Validation & Verification (V&V):

The most crucial step in the building-clock approach adopted by ManchesterCFD is 'Tier3' which consists of carefully validated CFD cases. The two well-established concepts in benchmarking CFD and to assess the uncertainty in the computational results are:

'Verification': The process of evaluating the products of a software development phase to provide assurance that they meet the requirements defined for them by the previous phase.

'Validation': The process of testing a computer program and evaluating the results to ensure compliance with specific requirements.

The ManchesterCFD's dedicated V&V activities for CFD particularly in the Energy, Healthcare and Environment sectors have featured in a number of highly recognised and cited publications.



PROFESSIONAL MEMBERSHIPS

Through official membership of the Mechanical Engineering Institutes in UK and US, all our projects and procedures at ManchesterCFD follow the code of conduct/ethics set out by the UK Institution of Mechanical Engineers (IMechE) and American Society of Mechanical Engineers (ASME).

UNIQUE FEATURES OF OUR SERVICES

Here is a list of the main advantages of working with 'ManchesterCFD' compared to other consultancy companies:

- World-Leading Scientists: A number of multi-award winning engineers and scientists at the ManchesterCFD team will be working on this project, all with numerous world-leading publications and patents.
- Highly Technical/Scientific Report: ManchesterCFD will provide a highly technical/scientific report which can later be used for patents, auditing, marketing and other R&D purposes.
- Future Funding Opportunities: ManchesterCFD is heavily R&D driven and therefore can act as a research partner for future funding opportunities (e.g. Innovate UK, EU, etc) to develop further research, if required.
- Access to Original Simulation Files: All the simulation files and original datasets could be provided to the Company following the completion of the project which would allow the Company to run further tests in-house, should they wish.

Dr Amir Keshmiri and ManchesterCFD team members have received numerous prestigious awards including:

- Winner of ‘Making a Difference’ Award – University of Manchester (May 2019)
- Winner of 'Frontiers of Development in Engineering' Award – Royal Academy of Engineering (Dec 2018)
- Winner of the 'Collaborate to Innovate' Award - The Engineer (Aug 2017)
- Elected as the 'Council Member - Fellow' – Institution of Mechanical Engineers (IMechE) (May 2017)
- Elected as the 'Fellow of IMechE - FIMechE' – Institution of Mechanical Engineering (July 2016)
- Winner of the ‘Best Supervisor of 2015’ Award – MMU Teaching Awards (June 2015)
- Winner of the ‘Young Mechanical Engineer of the Year’ – IMechE (Apr 2015)
- Winner of the ‘Rising Star’ Staff Award – MMU Staff Awards (June 2014)
- Winner of ‘Thomas Andrew Common' Award – IMechE (May 2013)
- Winner of the ‘Best Photo for the Art of Science’ competition – MMU (Apr 2012)

ROYAL ACADEMY OF ENGINEERING

Manchester Metropolitan University

TEACHING AWARDS

UKRI Engineering and Physical Sciences Research Council

Institution of MECHANICAL ENGINEERS

MANCHESTER 1824 The University of Manchester

MAKING A DIFFERENCE

theENGINEER

Dr Amir Keshmiri *BEng MSc PhD CEng FHEA FIMechE MIPEM*

- *Associate Professor in Computational Fluid Dynamics at the University of Manchester*
- *Director of ManchesterCFD Team at the University of Manchester*
- *Head of Faculty Business Engagement at the University of Manchester*

Dr Amir Keshmiri is a Reader/Associate Professor in Computational Fluid Dynamics in the Department of Mechanical, Aerospace and Civil Engineering (MACE). He is also the Head of Business Engagement in the Faculty of Science&Engineering at the University of Manchester, UK.

Amir received his BEng (2005), MSc (2006) and PhD (2010) in Mechanical Engineering from the University of Manchester in collaboration with British Energy and Électricité de France (EdF) R&D in Paris. Following the completion of his PhD, Amir received a prestigious research fellowship to extend his collaboration with industrial partners in the field of heat transfer and thermofluids. He was then invited to lead a CFD team at AECOM, one of the largest engineering consultancy companies in the world. Amir returned to academia as a Lecturer/Senior Lecturer at Manchester Metropolitan University (MMU) andin 2016, he returned to the University of Manchester as an Assistant Professor and Director of Business Engagement.

To date, Amir has been the recipient of numerous prestigious awards and honours, some of the recent examples include the ‘UK Research Council’s (EPSRC) Doctoral Prize’ in 2010, ‘Institution of Mechanical Engineers (IMechE) Thomas Common Award’ in 2013, University’s ‘Rising Star’ award in 2014, and ‘Best Supervisor of the Year’, IMechE’s ‘Young Mechanical Engineer of the Year’ award in 2015, ‘Collaborate to Innovate’ award in 2017 by ‘The Engineer’ magazine, the ‘Frontiers of Innovation in Engineering' by Royal Academy of Engineering (RAEng) in 2018 and the ‘Making a Difference’ award by the University of Manchester in 2019.

Amir is a ‘Chartered Engineer’ and ‘Fellow’ of IMechE. In 2017, in the general election, he was elected as a 'Council Member' of the IMechE for 3 years. IMechE is one of the largest Mechanical Engineering Institutes in the world with over 130,000 members in over 80 different countries. Amir is also an associate editor of the Journal of Research on Biomedical Engineering, published by Nature Springer.

Tel: +44 (0) 161 820 1686
Web: www.ManchesterCFD.co.uk
Email: info@ManchesterCFD.co.uk
Address: Office30, Core2, 4th Floor, Engineering Building, Booth E St, Manchester, M13 9SS, UK