LONDON SOUTH BANK UNIVERSITY

1st IIR International
Cold Chain and Sustainability Conference

CFD in Refrigeration

Introduction to Mathematical modelling

Alan Foster

alan.foster@lsbu.ac.uk

Contents

- Overview of CFD
    - What it does
    - Equations/models
    - How you use it
- CFD codes
- Application examples
- Future

What is CFD?

- Computational Fluid Dynamics
- Predict

  Fluid flow – convection in air, water etc

  Natural convection – buoyant flow

  Heat transfer – through solids/fluids

  Radiation

  Others – combustion, particle tracking etc

Why use CFD?

- Reduce testing – expensive and time consuming
- Impossible to test – catastrophic failures
- Colourful visualisation enhances understanding and stimulates ideas

Navier Stokes equations

- Conservation of:
- Mass (continuity)

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_j) = 0$$

Mass in = mass out (incompressible)

- Momentum (Newton's 2nd law of motion)

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial}{\partial x_j}\left[-p\delta_{ij} + \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \rho g_i$$

rate of increase of momentum of fluid element = sum of forces on element

- Energy

$$\frac{\partial}{\partial t}(\rho CT) + \frac{\partial}{\partial x_j}(\rho u_j CT) - \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial T}{\partial x_j} \right) = s_T$$

rate of increase of energy of fluid element = rate of heat added

Buoyant flow

- Buossinesq approximation
- Density differences can be neglected except where they cause buoyant flow

$$\rho = \rho_{ref}[1 - \beta(T - T_{ref})]$$

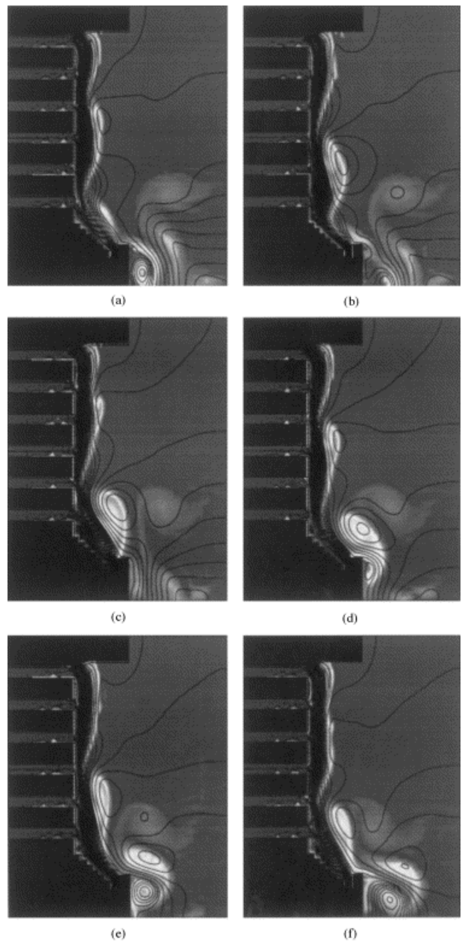
- Cannot be used at high ΔT
- Otherwise ideal gas equation (more numerically complex)

Turbulence

- Complex – 3D, unsteady, many scales
- Large effect on flow and heat transfer
- High Re
- Possible to predict from N-S equations (DNS), however, range of l and t is such that enormous computer resources required for a few s of flow

Turbulence models

- RANS
  - Statistical models
  - Average out fluctuations
  - Mean flow and fluctuating values
  - Numerically simple
  - Many models to choose from
- LES – Large eddy simulation
  - Implicity calculate large eddies
  - Numericaly between DNS and RANS
- DES - Direct Eddy simulation
  - Switch between LES and RANS



RANS models

- k-ε most popular (industry standard)

  Confined flows – good

  Wakes, jet entrainment, swirling – not good

- Mixing length

  Easy and cheap

  Cannot do seperation and re-circulation

- RSM - Reynolds stress model

  Accurate complex flows

  Numercialy complex

- Many others

Other models

- Porous media

  Complex geometry – HEs, stacks of product)

  Flow straightening

  Darcy Forchheimer equation

$$\frac{\partial \rho}{\partial x} = -\frac{\mu}{K} v + \rho C v^2$$

- Non-Newtonian fluids

  Shear thinning or thickening behaviour

  Yoghurt, soup, milk etc

  Power law, Bingham, Herschel-Bulkley models

Wall functions

- Boundary layer
  - Laminar – linear (v with x)
  - Turbulent – log-law
  - Laminar sub-layer
- Wall mesh
  - Consider Y^+ and model
- Important
  - Confined flows
  - Impingement
  - Attachment

Other models

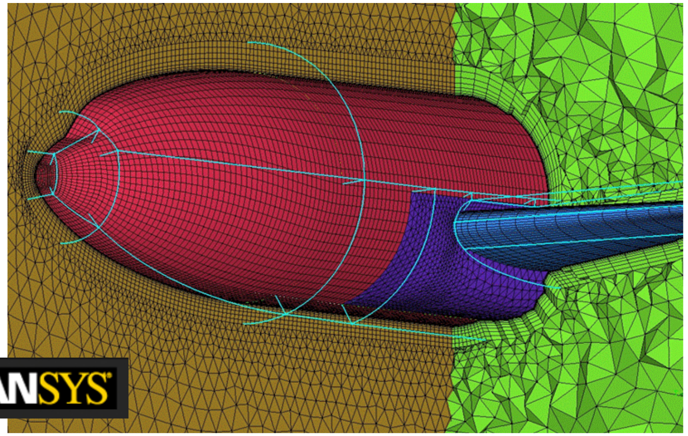
- Radiation
  - Monte-Carlo method – photons, fluid transparent
  - P1 and Rosseland - Combustion
- Multi-phase
  - Evaporation, humidity, mass transfer – Lewis Relationship
  - Vapour Refrigerant cycle?
- Particle transport
  - Lagrangian

How do you use CFD?

- Geometry/mesh
- Boundary conditions – inlets/outlets velocities and temperatures
- Convection schemes
- Solution
- Visualise output

Geometry/Mesh

- 2D or 3D
  - Half length = 4 x cells (2D) 8 x cells (3D)
  - More cells = more memory and more time = more expensive
- Symmetry, periodicity
- Structured/un-structured/hybrid
- Sliding mesh
- Adaptive meshes
- Refinement and shape near wall



Boundary conditions

- Define the numerical domain
- Boundary conditions

  Temperature, heat transfer, velocity, k and ε

  May require separate model to evaluate k_i and ε_i
- Initial conditions
- Fluid and solid properties

  ρ, μ, c_p, k, β

Convection schemes

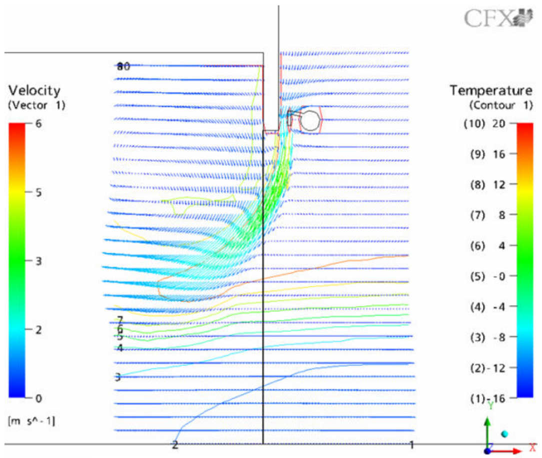
- Method to solve (converge)
- Convect solution from boundary conditions into the domain
- Different schemes offer different accuracy, time to solve and robustness
  - Central – 2nd order, high directional convection
  - Upwind – 1st order, flow direction
  - Hybrid – best of both, less stable
  - QUICK – 3rd order, CPU
  - Others

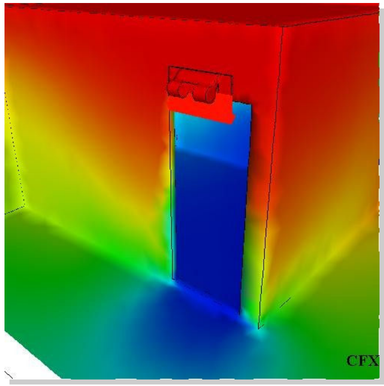
Solution

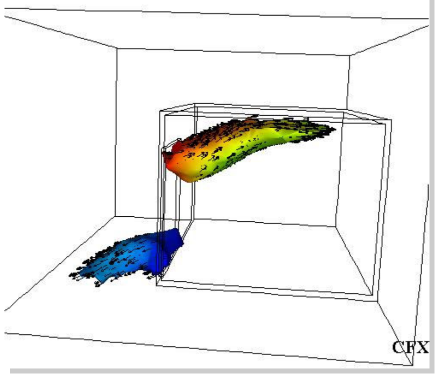
- Converged
  - Maximum residual
  - Not changing with more iterations (monitor points)
  - Grid independent
- Compare with measured data (specific points)

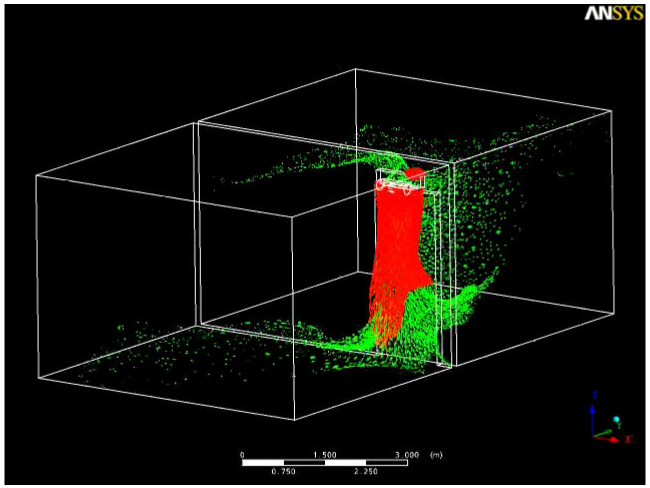
Visualise output

- Does it look right?
- Anything unexpected (double check)
- Vectors
- Colour contours
- Iso-surfaces
- Streamlines (3D)
- Videos
- VRML
- Export to other software
- B&W and greyscale!









CFD codes

- CFX (ANSYS)– Air blast chilling, vertical display cabinets, vacuum cooling, weight loss
- Fluent (ANSYS) – Refrigerated trucks, display cabinets, domestic refrigerators, humidity
- Star-CD – Refrigerators, air side hc fin and tube HEs
- Phoenics (CHAM) – Pallets of food during shipping
- CFDesign
- In house
- Shareware

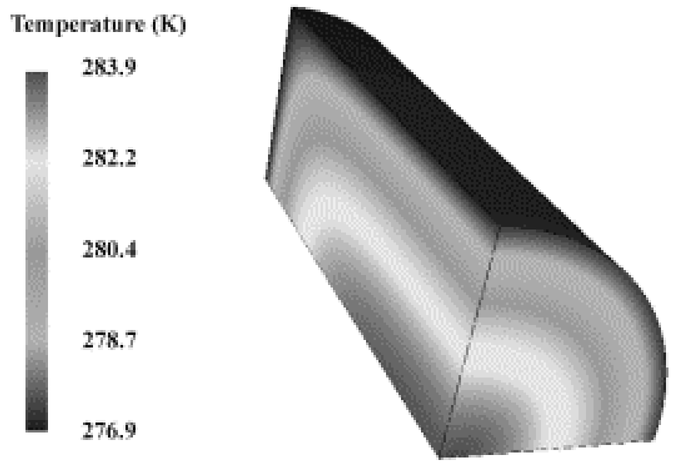
Examples

- Food processing – Vacuum cooling process
- Transport - Refrigerated truck with pallets
- Cold storage
- Display– 3D vertical display cabinet
- Domestic frost-free refrigerator
- Louvered fin-and-flat-tube heat exchangers
- Unsteady Flows in a Scroll Compressor

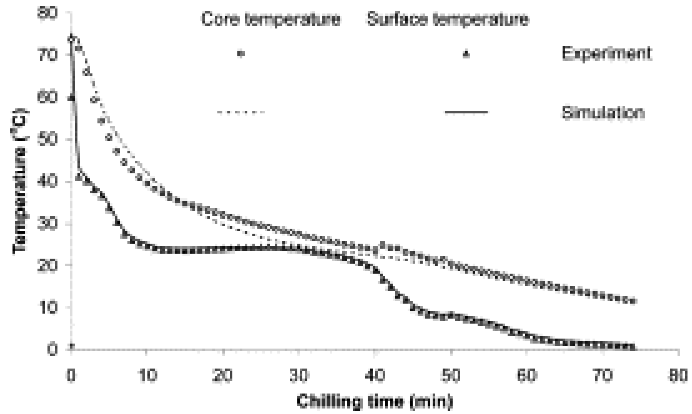
CFD simulation of coupled heat and mass transfer through porous foods during vacuum cooling process

Z. Hu, D. Sun (2004) Int. J. Ref, 27:8,1009

- Predicts - temperature distribution, weight loss, moisture content
- Effect of - pressure, density and water content, thermal shrinkage, and anisotropy of the food



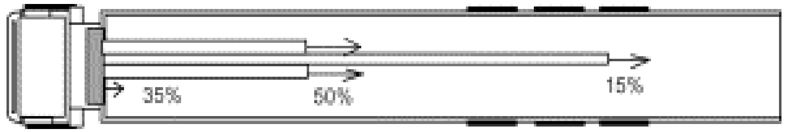
Cylindrical ham at 73 min chilling time



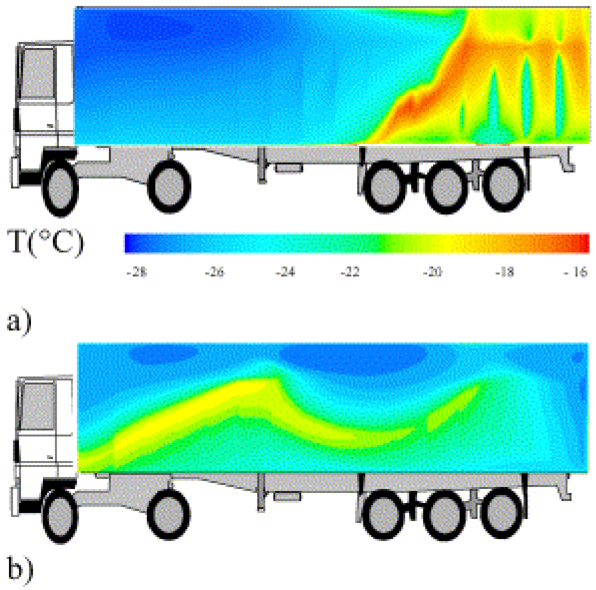
Airflow pattern and temperature distribution in a typical refrigerated truck configuration loaded with pallets

J. Moureh, D. Flick (2004) Int. J. Ref., 27:5,464-474

- Optimise air-distribution in refrigerated vehicles throughout palletised cargos
- Fluent, the Reynolds stress model (RSM)



air ducts

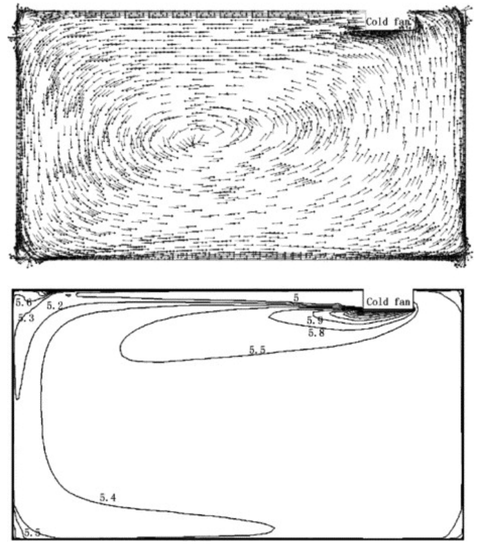


Contours of iso-temperatures with and without air ducts.

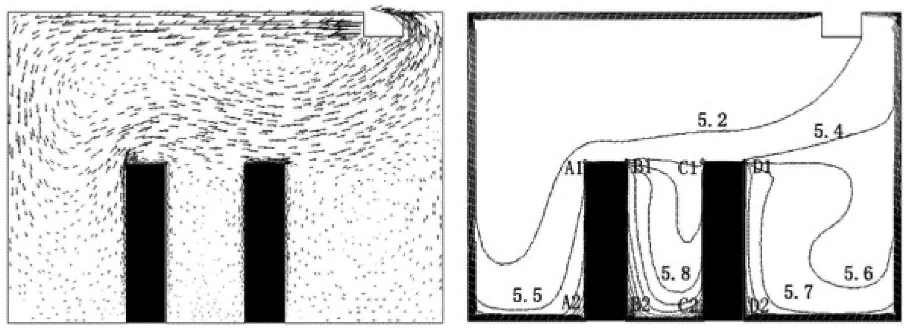
Effects of design parameters on flow and temperature fields of a cold store by CFD simulation

J. Xie et al. (2006) J. Food Eng. 77:2, 355-363

- Airflow and temperature distribution
- Analyse corner baffle and the stack mode of foodstuffs



Flow field and temperature field

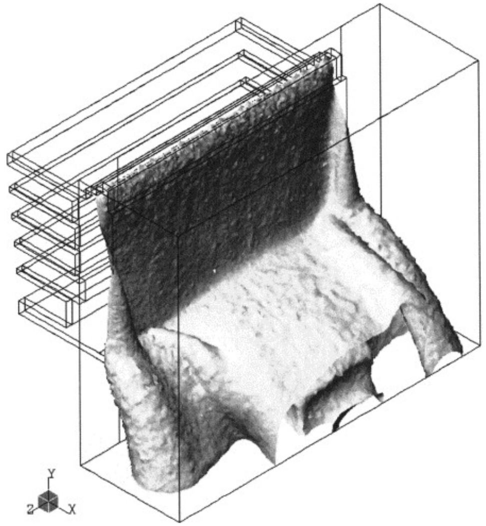


With product

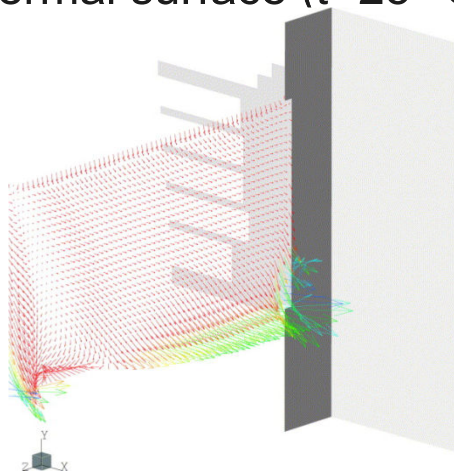
Two- and three-dimensional CFD applied to vertical display cabinets simulation

P. D'Agaro et al. (2006) Int. J. Ref. 29:2,178-190

- Frozen food vertical display cabinet
- Short cabinets - 3D secondary vortices at the side walls provide the most important mechanism for hot air entrainment



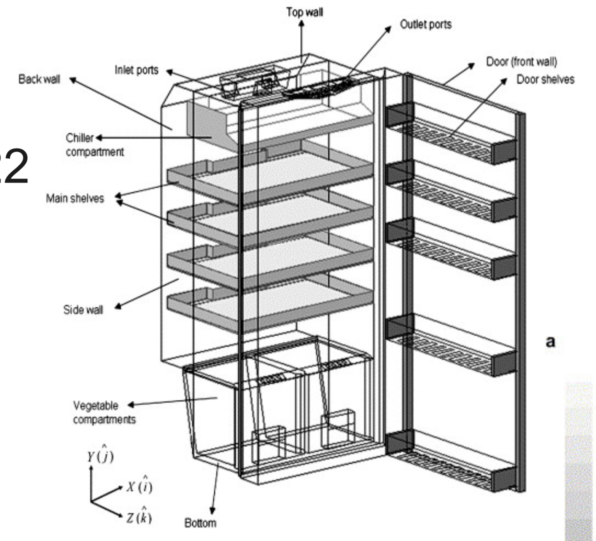
Isothermal surface (*t*=25 °C)



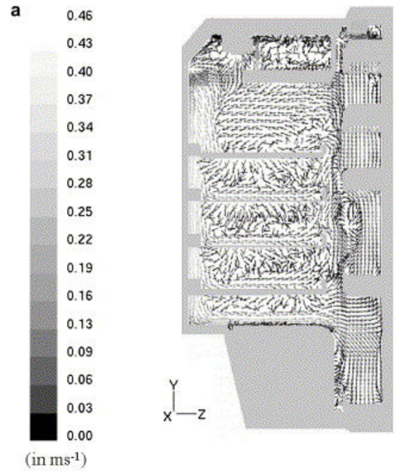
Velocity vectors (temperature-based colour) with 0.2 m s^{-1} from left to right

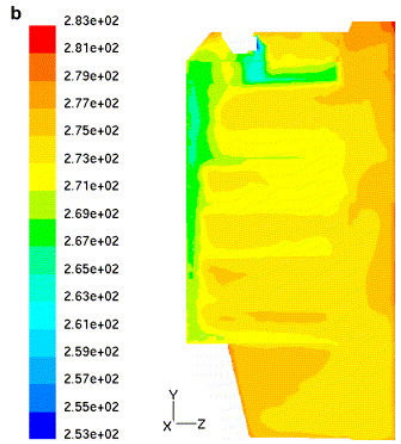
Modeling of a domestic frost-free refrigerator

J Gupta et al. (2007) Int. J. Ref. 30:2,311-322



- Qualitatively similar, perceptible offset
- Unaccounted heat transfer rates through the door gaskets and the compressor

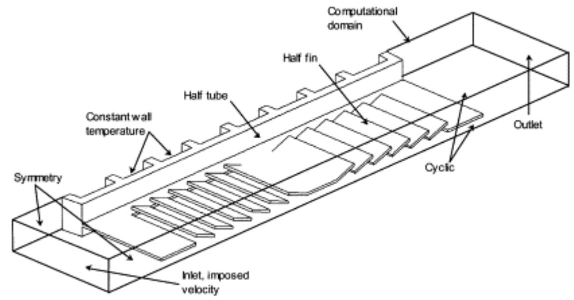


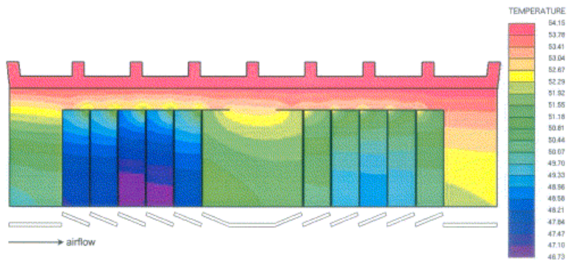


Thermal-hydraulic CFD study in louvered fin-and-flat-tube heat exchangers

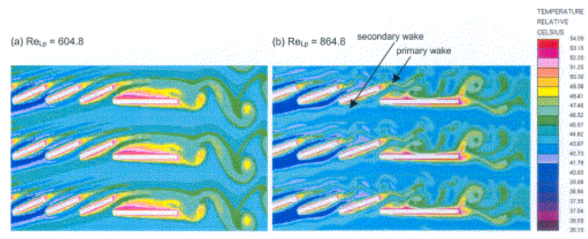
T Perrotin et al. (2004) Int. J. Ref. 27:4,422-432

- Louvered fin automotive condenser
- 3D models, tube effects, conjugate heat transfer and conduction through the fin good agreement with the experimental results





Fin temperature distribution

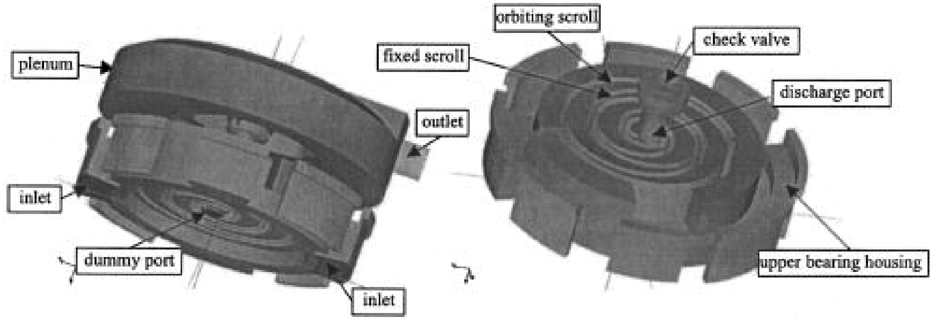


Temperature field for different Re

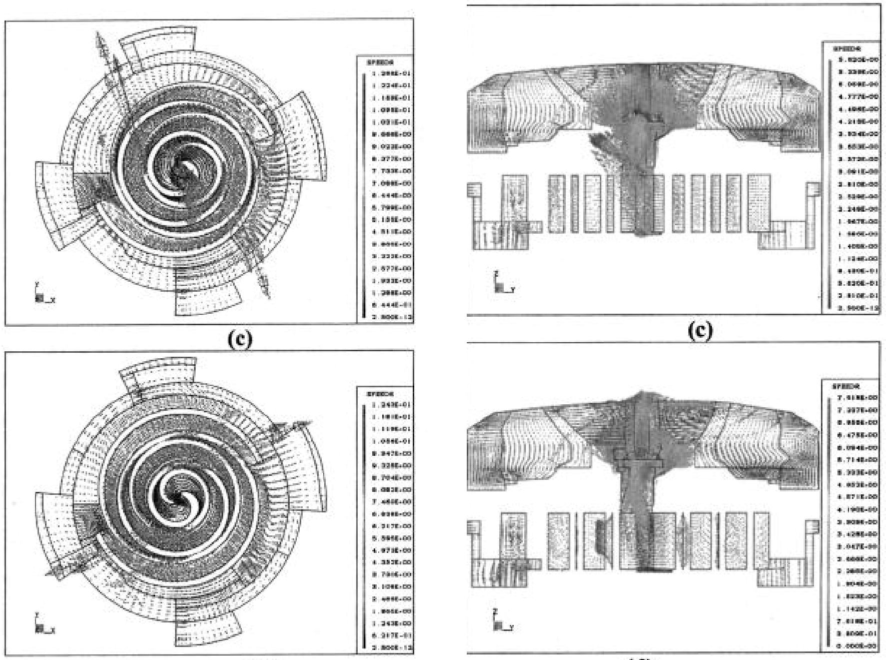
Numerical Study of Unsteady Flows in a Scroll Compressor

M Cui et al. (2006) J. Fluids Eng. 128:5,947

- Change in volume and moving boundary
- Unsteady flows inside and between the gas pockets



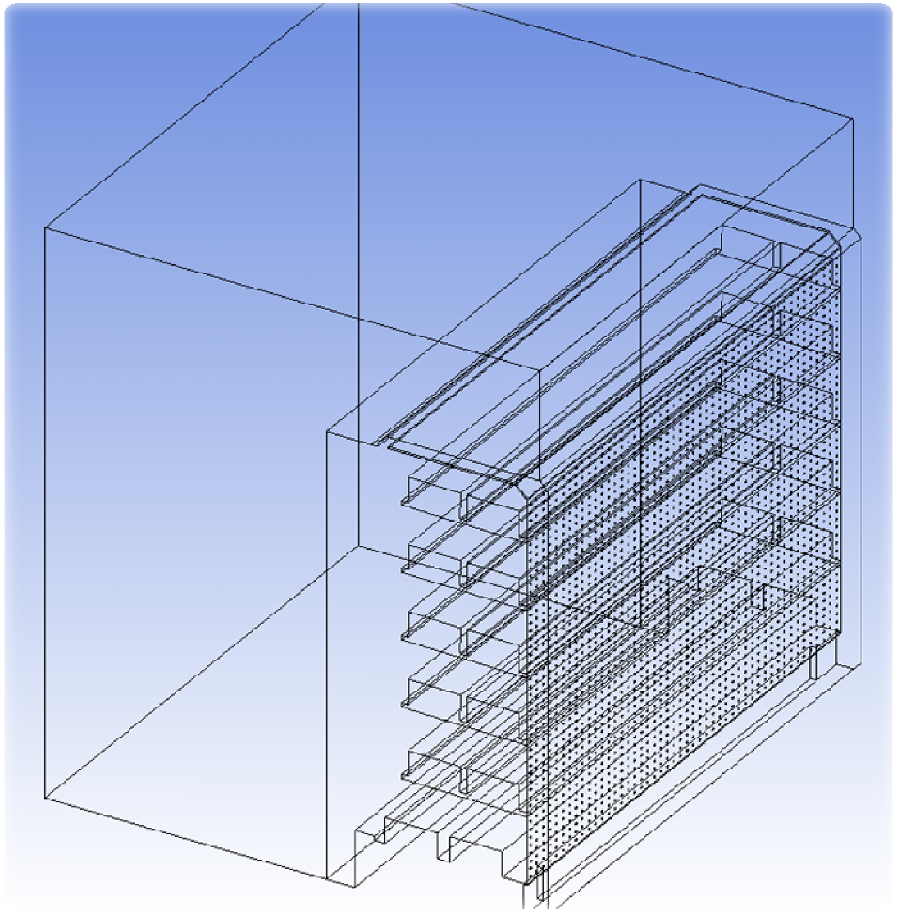
(a) (b)


(c) (c)
(d) (d)

Future of CFD (like to see)

- All in one package
  - User friendly
  - Accurate
  - Stable
  - Good visualisation
- CAD to CFD in one step
- Rapid changes
- Cheaper licenses
- More tailored (simpler) software
- Universal models – turbulence etc

When can we model this?



- Holes 10 mm (1 mm mesh)
- 10 to 50 mm mesh
- Mesh elements 50,647,848
- 1st iteration 5 days