MULTI-PHYSICS MODELLING AND SIMULATION: CHALLENGES AND OPPORTUNITIES

M. Cross, T.N. Croft, D. McBride, A.K. Slone, and A.J. Williams

Centre for Civil and Computational Engineering

School of Engineering

University of Wales, Swansea

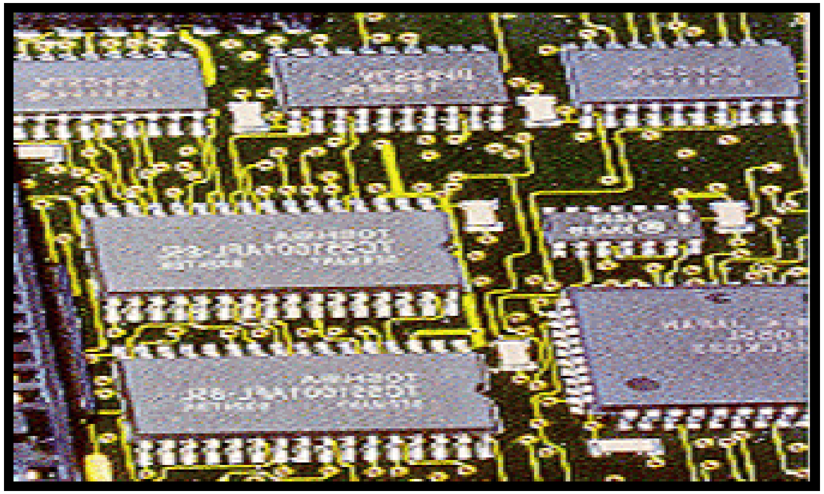
m.cross@swan.ac.uk

Multi-Physics 2006, Maribor
December 2006

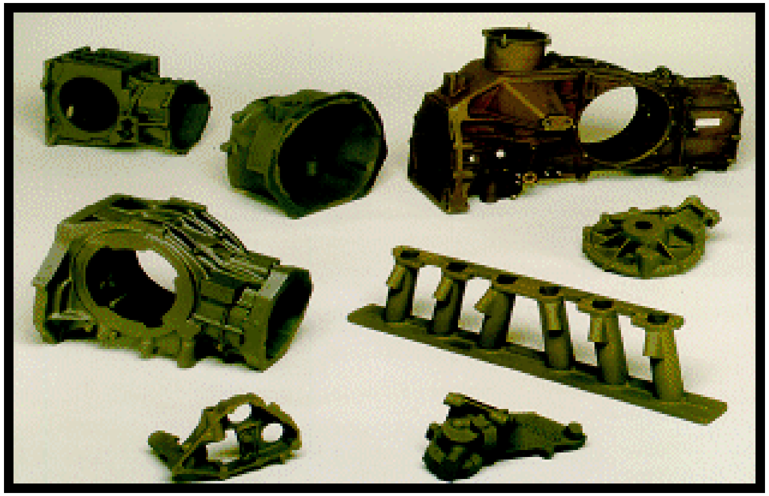
Why Multi-Physics?

- Large number of real-world problems are multi-physics.

Electronic Packaging

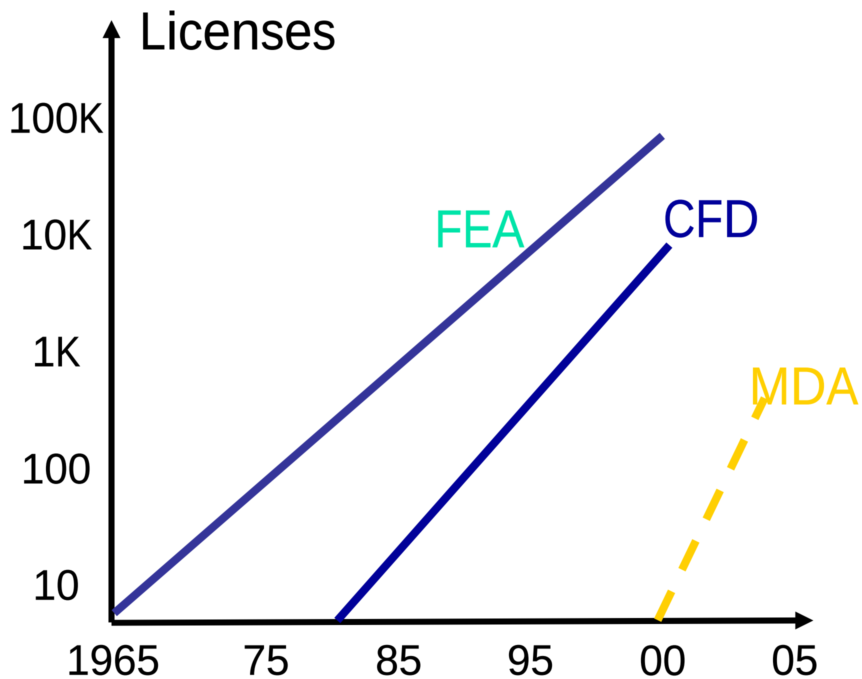


Castings Components



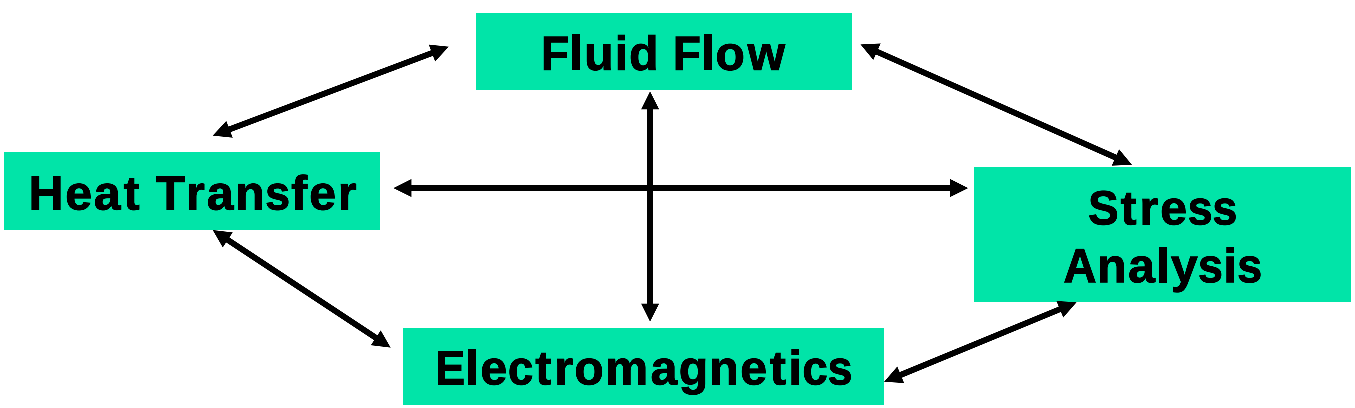
CAE analysis tools market history

- **FEA** started mid 60's with
  - NASTRAN, Abaqus, ANSYS, etc as major players
- **CFD** started 1980 with
  - FLUENT,CFX, PHOENICS and STAR-CD as major players
- **MDA** started mid 1990's
  - Coupling codes
  - MDICE, Spectrum, PHYSICA



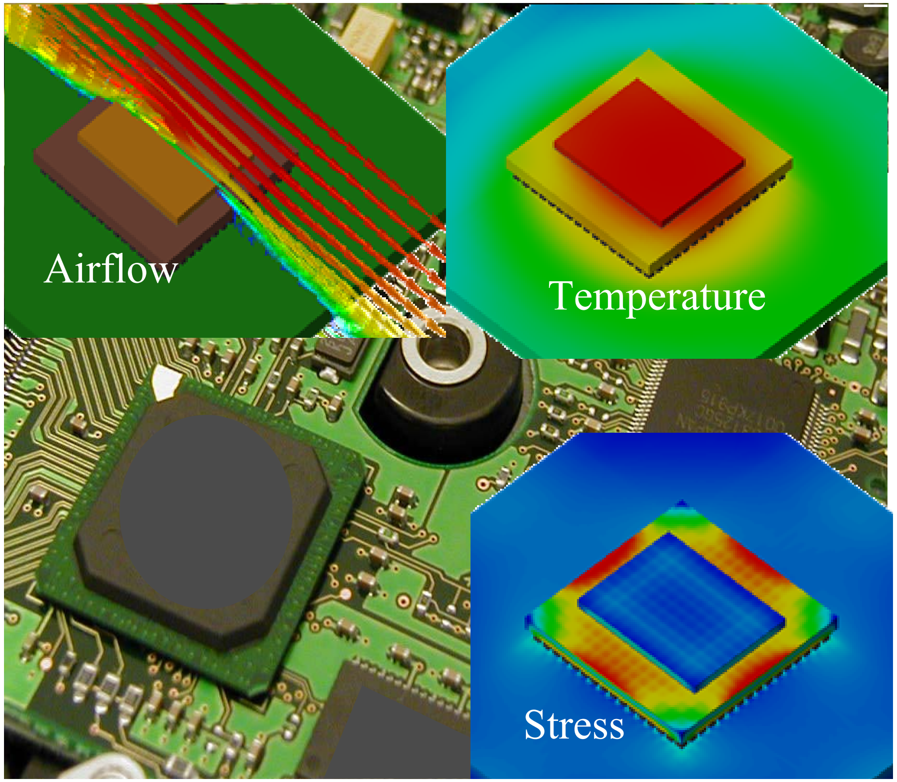
Why Multi-physics Modelling ?

- Large number of real world problems require multi-physics simulation tools.
- Examples
  - Solidification problems – Solder Joints
  - Fluid-Structure interaction – Flutter in aircraft wings
- Need to solve for integrated physics
- Ensure two-way coupling



Commercial Software – Multi-physics

- Number of products claiming to be multi-physics
- ANSYS/Multi-physics
  - http://www.ansys.com/
- PHYSICA
  - http://www.physica.co.uk/
- COMSOL
  - http://www.comsol.com/
- Algor
  - http://www.algor.com/
- DYNA
  - http://www.lsc.com
- ADINA
  - http://www.adina.com
- Flomerics
  - http://www.flomerics.com/

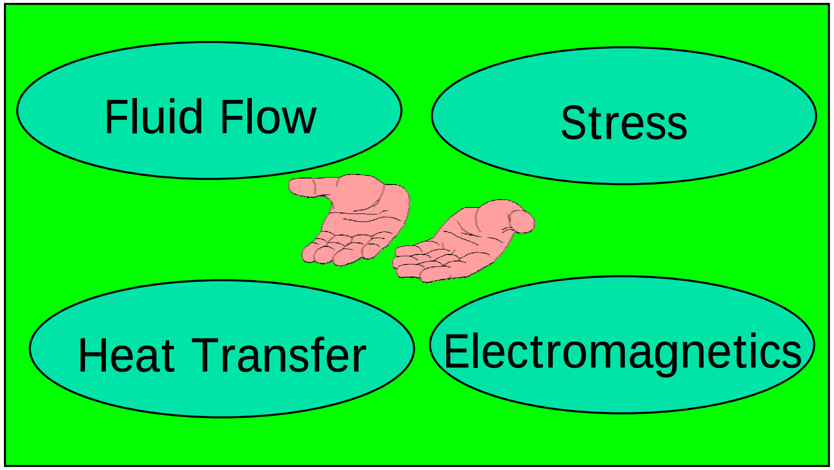


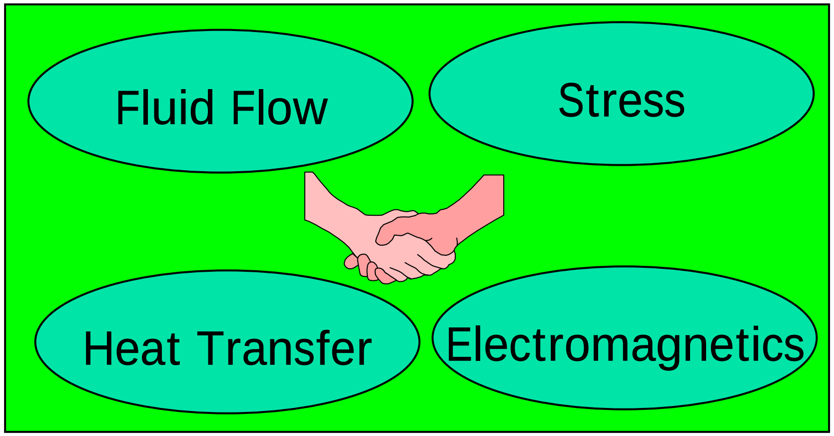
Classifying multi-physics

- What most vendors advertise is ***multi-physics***
- What most vendors offer is ***multi-disciplinary***
- ***Multi-disciplinary*** – using data generated by one code as input into another – *loose or one way coupling* (e.g. electric field loading a thermal calculation)
- ***Multi-physics*** – two way exchange of information, which could involve implicit convergence within a time-step (e.g. thermo-mechanical)
- ***Closely coupled multi-physics*** – time and space accurate exchange of data (e.g. dynamic fluid-structure interaction)

MDA vs. MULTI-PHYSICS

- Must distinguish between MDA and multi-physics:
  - one loosely coupled, other tightly coupled
  - one significant challenge, other major new technology development
- Multi-physics analysis always involves challenging flow analysis, so must be designed to compete well with leading edge CFD tools
- Limited CFD => limited multi-physics
- Limited parallel scalability => limited multi-physics





Multi-physics Modelling

- Physics Requirements
  - Fluid Flow
  - Heat transfer
  - Solidification/phase change
  - Stress
  - Electro-magnetics

→ MULTI-PHYSICS

- Geometry
  - Complex

→ UNSTRUCTURED

- Large simulations

→ PARALLEL

Key issue: CFD capability

Simulation technologies

- Key players for thermo-fluid based models:
  - CFX
  - FLUENT
  - STAR-CD
- Key players for thermo-mechanical models:
  - ANSYS
  - ABAQUS
  - NASTRAN
- Key player for electro-magnetics:
  - OPERA & CONCERTO, Vector fields

Computational approaches

- Most ‘leading’ CFD codes use FV methods on unstructured mesh
- All CSM codes based upon FE methods with a wide variety of element types
- CEM usually based on FE (and sometimes BE) methods
- **Handling the physics interaction – the challenge!**

What do you need for multi-physics simulation?

- Necessities:

  - phenomena specific solver software that can accept boundary data, volume source data and modifications to property data from other codes

  - good filters to exchange boundary and volume source data from one solver module to another

  - solver strategies which are compatible

- Practical demands:

  - Compatibility of the mesh structure

  - Very good filters for mapping numerical information from one solver to another

  - Avoid opening and closing files – read numerical information directly from one solver by another; a common memory database is desirable

  - Parallel scalability is necessary for the large problems

Practicalities of multi-physics simulation

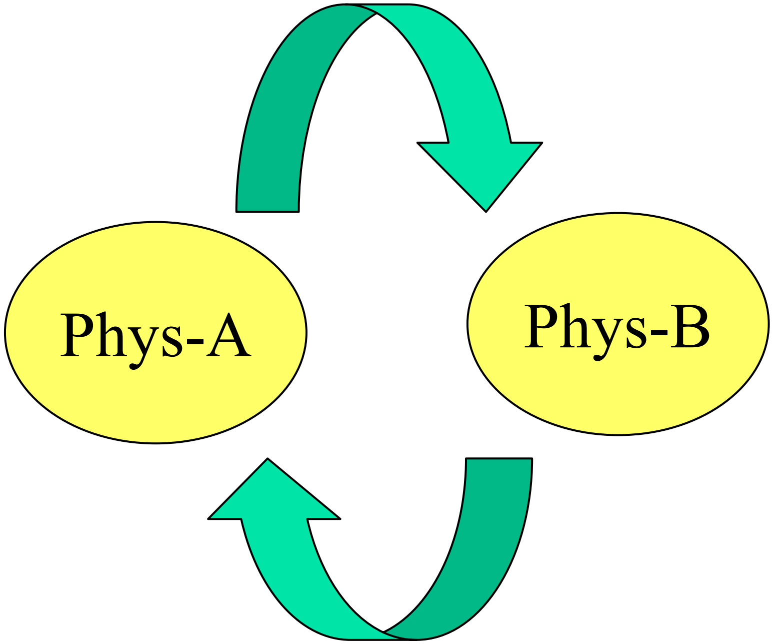
- **Good numerical filters** to map data from one solver into another

- Interpolation from one set of variables to another => **compatibility of mesh**

- **Single database** of mesh data & simulation variables

- **Solver strategy**
  - Direct vs Iterative
  - Eulerian vs Lagrangian

- Is coupling strategy compatible with **scalable parallelism**, EVEN if software components are parallel?



MpCCI – a tool for code interoperability

- Coupled physics implies coupling of separate phenomena codes:

  - without opening/closing files

  - operate in a parallel context
- Emerged from an EU project – public domain OPEN SOURCE tools
- www.scai.fraunhofer.de/mpcci.0.html
- Applications to fluid-structure interaction:

  - ABAQUS + FLUENT for DFSI

  - STAR-CD + NASTRAN for DFSI
- **BUT exchanging data does NOT necessarily mean coupling of the physics that is time or space accurate**

Alternative approach: Single Software Framework

- **Key route to closely coupled multi-disciplinary (multi-physics) simulation**
- Basic requirements of a SSF:

- **consistency** of mesh for all phenomena

- **compatibility** in the solution approaches to each of the phenomena

- **single database & memory map** so that no data transfer & efficient memory use between programs

- facility to enable **accurate exchange** of boundary or volume sources (e.g. body force)

- enables **scalable parallel operation** for all physics interactions

Attempts at SSF for multi-physics

- **COMSOL** – FEMLAB

  - Originally based on MATLAB as a suite of FE discretisation routines
- **OEFELE** – Open Engineering

  - An FE based solver framework
- **FOAM**

  - solver framework for FE and FV discretisations
- **PHYSICA**

  - FV based tools for multi-physics

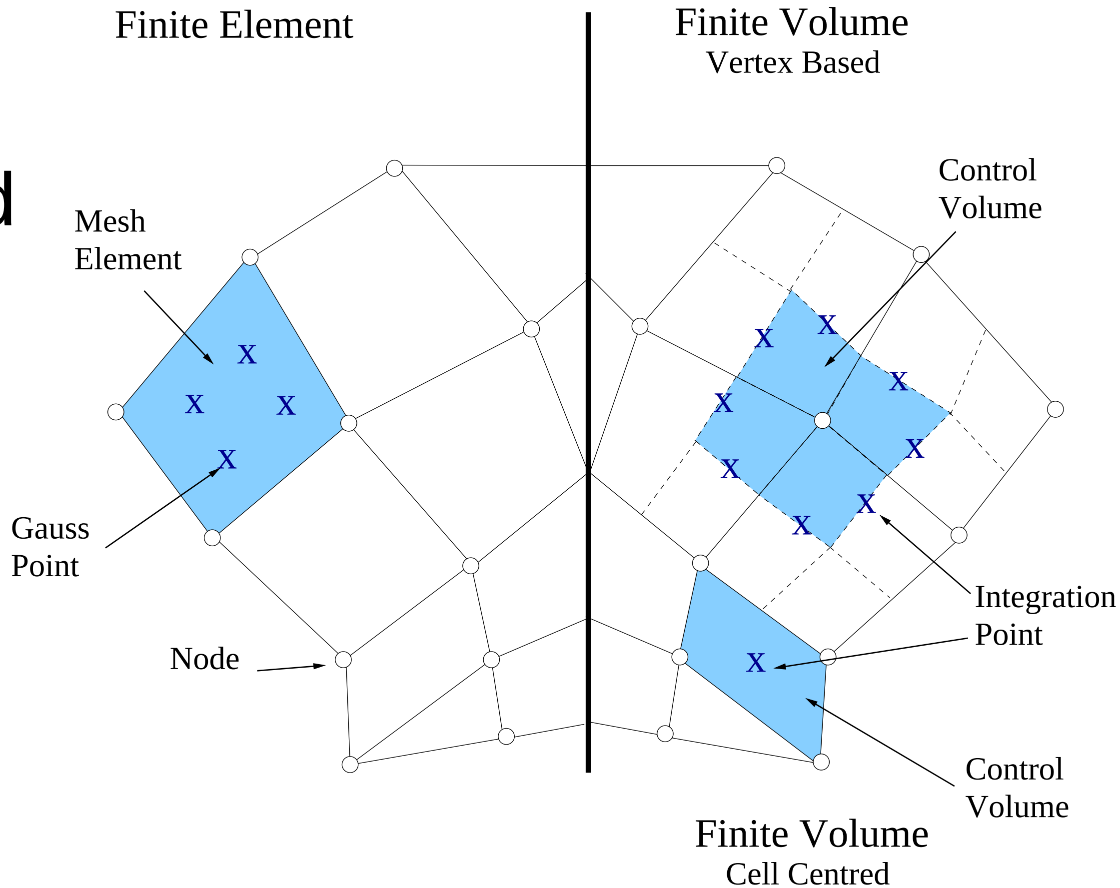
PHYSICA – Multi-physics Framework

- Work started in late 1980s at University of Greenwich
- Based upon FV methods on unstructured mesh (FV-UM)
- **Conservative approach:**

  ****FV-UM discretisation used for everything****
- Flow/ electro-magnetics/ heat transfer procedures from FV-SM -> FV-UM
- Solid mechanics developed from scratch
- Prototypes moved from:
  a) 2 ->3D and
  b) scalar -> parallel
- Key issue was to ensure FLOW worked well in all contexts
- Solidification processes a key target

Spatial Discretisation in PHYSICA

- Unstructured mesh
- CSM
  - Vertex based
  - FV
- CFD
  - Cell centred
  - FV



Finite Volume Method

- Domain divided into a number of finite size control volumes (CV)
- Conservation equation integrated over each CV and time
- Approximations to each term yields a linear system in the unknown values of the variable ϕ,

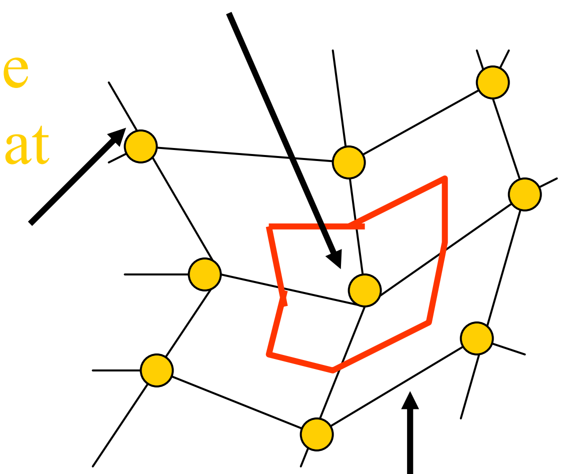
$$\frac{\partial(\rho\phi)}{\partial t} + \nabla\cdot(\rho\phi\underline{\mathbf{u}}) = \nabla\cdot(\Gamma\nabla\phi) + S_\phi$$

Discretisation Techniques for FV Methods

Vertex-Based (VB)

CV Constructed around Mesh Vertex

Variable Solved at Mesh Vertex



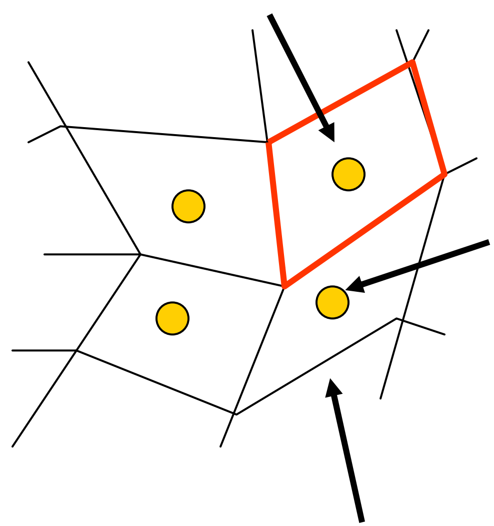
Shape Function Approximations

SOLID MECHANICS

Cell-Centred (CC)

CV Associated with Element

Variable Solved at Element Centre



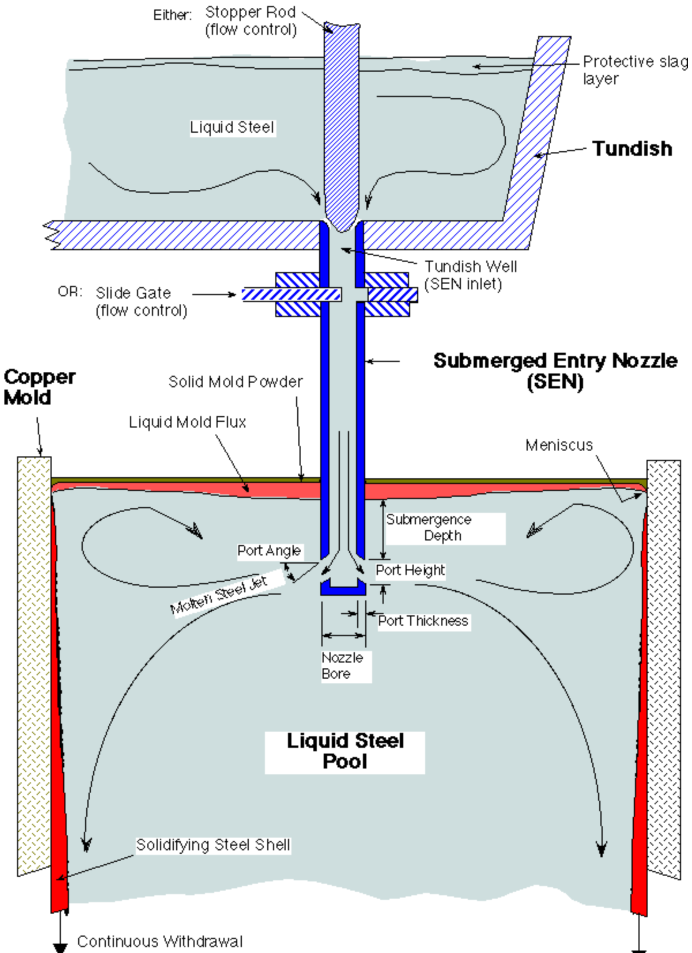
Finite Volume Approximations

THERMO-FLUID MECHANICS

Continuous casting process: example of CFD based multi-physics

- Mixture of liquid steel and argon injected into rectangular mould
- Liquid metal flux sits on top of mould
- Water cooled mould extracts energy forming a solid steel shell
- Continuous withdrawal

Schematic of continuous casting tundish, SEN, and mold



B.G. Thomas

Multi-phase equations

- **Mass and momentum**

$$\frac{\partial}{\partial t}(\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = \nabla . (\mu \nabla \mathbf{u}) - \nabla p + \mathbf{S}$$

$$\frac{D(\ln \rho)}{Dt} + \nabla \cdot (\mathbf{u}) = 0$$

- **Energy**

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\rho h \mathbf{u}) = \nabla . (k \nabla T) + \mathbf{S}_h$$

- **Density**

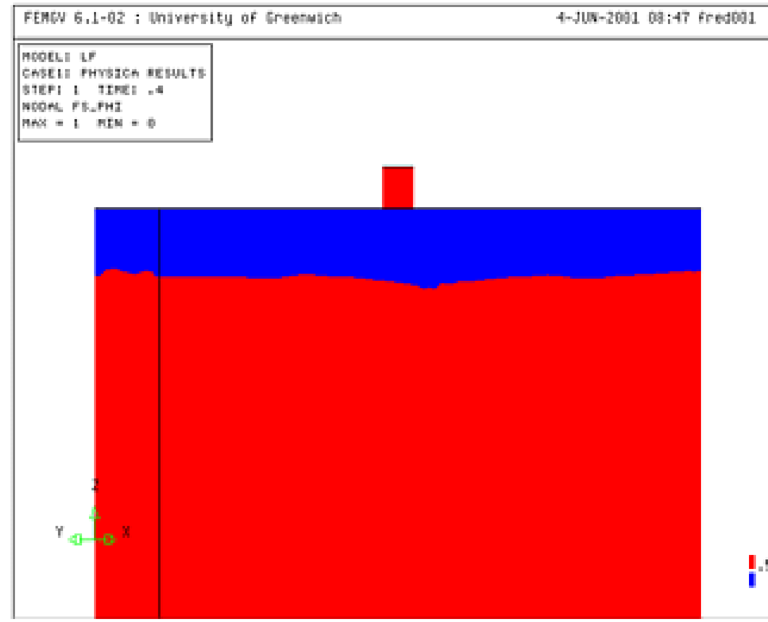
$$\rho = \rho(\phi_{metal}, \phi_{gas}) \quad or \quad \rho = \rho_{flux}$$

Free surface (SEA)

- **Solves:**

$$\frac{\partial \phi}{\partial t} + \mathbf{u}.\nabla \phi = 0$$

  - where ϕ is the fraction of metal in a cell



- van Leer scheme used to reduce smearing of interface
- continuity equation solved for volume not mass
- properties a linear combination of phases present

Solidification

- **Release of energy due to phase change**

$$S^h = -\frac{\partial}{\partial t}\left(\phi\rho_m f_L L\right) - \nabla\cdot\left(\phi\rho_m \mathbf{u} f_L L\right)$$

$$f_L = \begin{cases} 1 & T > T_L, \textit{the liquidus temp} \\ \left(\dfrac{T - T_S}{T_L - T_S}\right) & T_S \le T \le T_L, \textit{in the mushy zone} \\ 0 & T < T_S, \textit{the solidus temp} \end{cases}$$

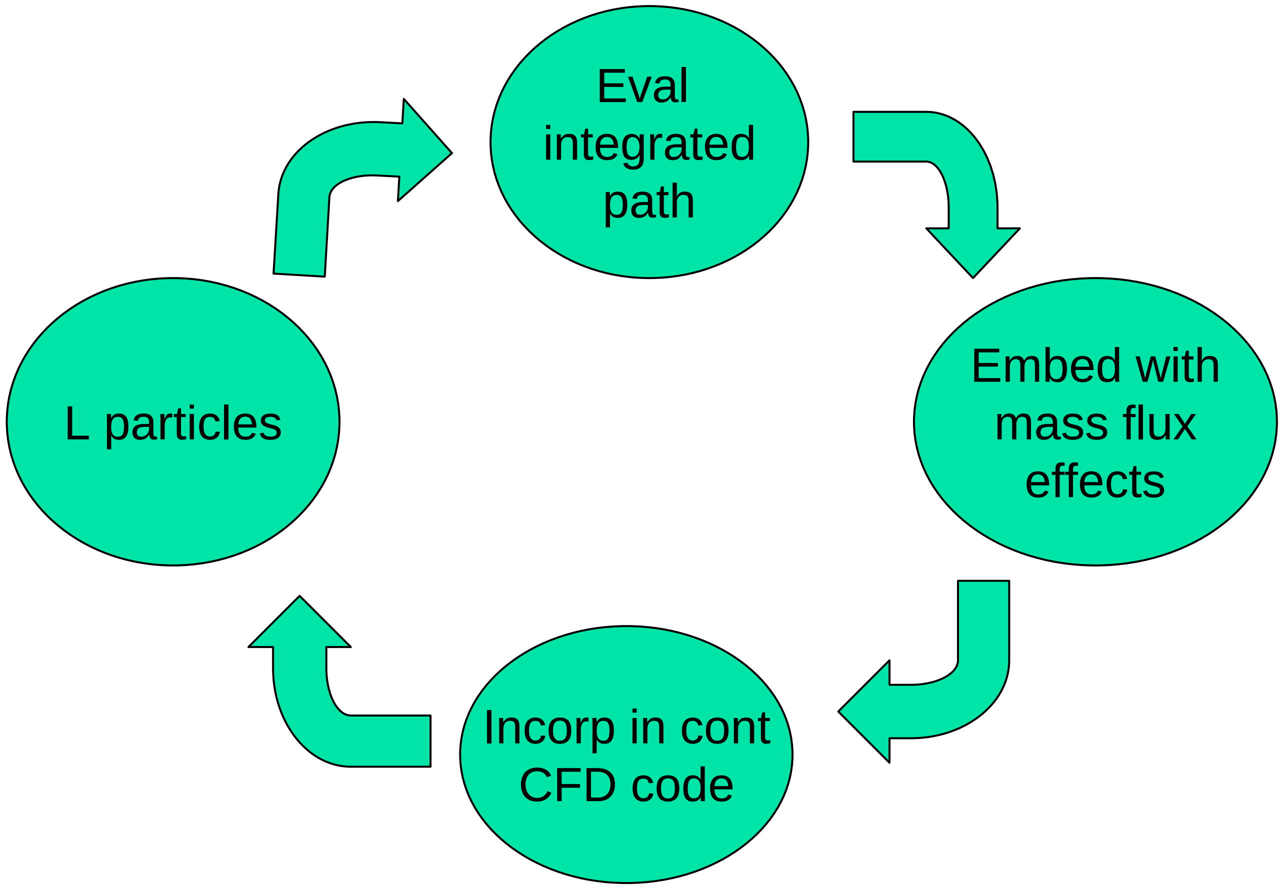
- **Darcy source for momentum equations**

Argon Bubbles

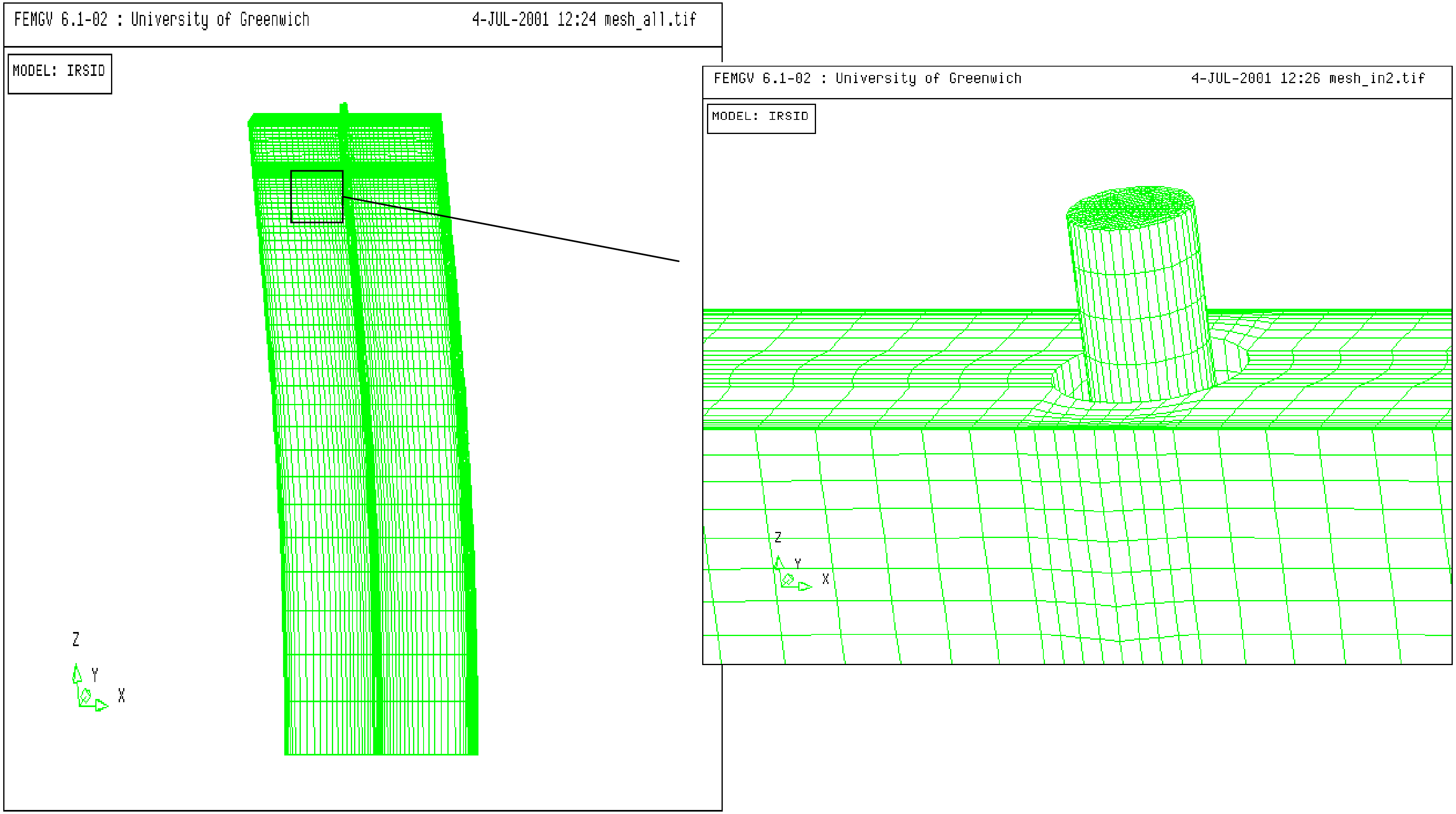
- CFD - a mixed Lagrangian – Eulerian calculation:
  - calculate the flow field using CFD procedure
  - use the flow field to influence the particle movement

- B/D/P equations (Argon bubbles in this case) are solved explicitly in Lagrangian framework.
- New position of each particle at given time-step computed from the particle equation of motion.
- Particle is subjected to a drag force C_d and buoyancy but no turbulence feedback.
- Drag force is an empirical function of the "slip" Reynolds number between particle and surrounding fluid.
- Account is taken of
  - the particles entering and leaving each computational cell
  - the time taken between entry and exit.
- Giving the instantaneous volume fraction of Argon in each cell, which is used to adjust the average density or other cell properties

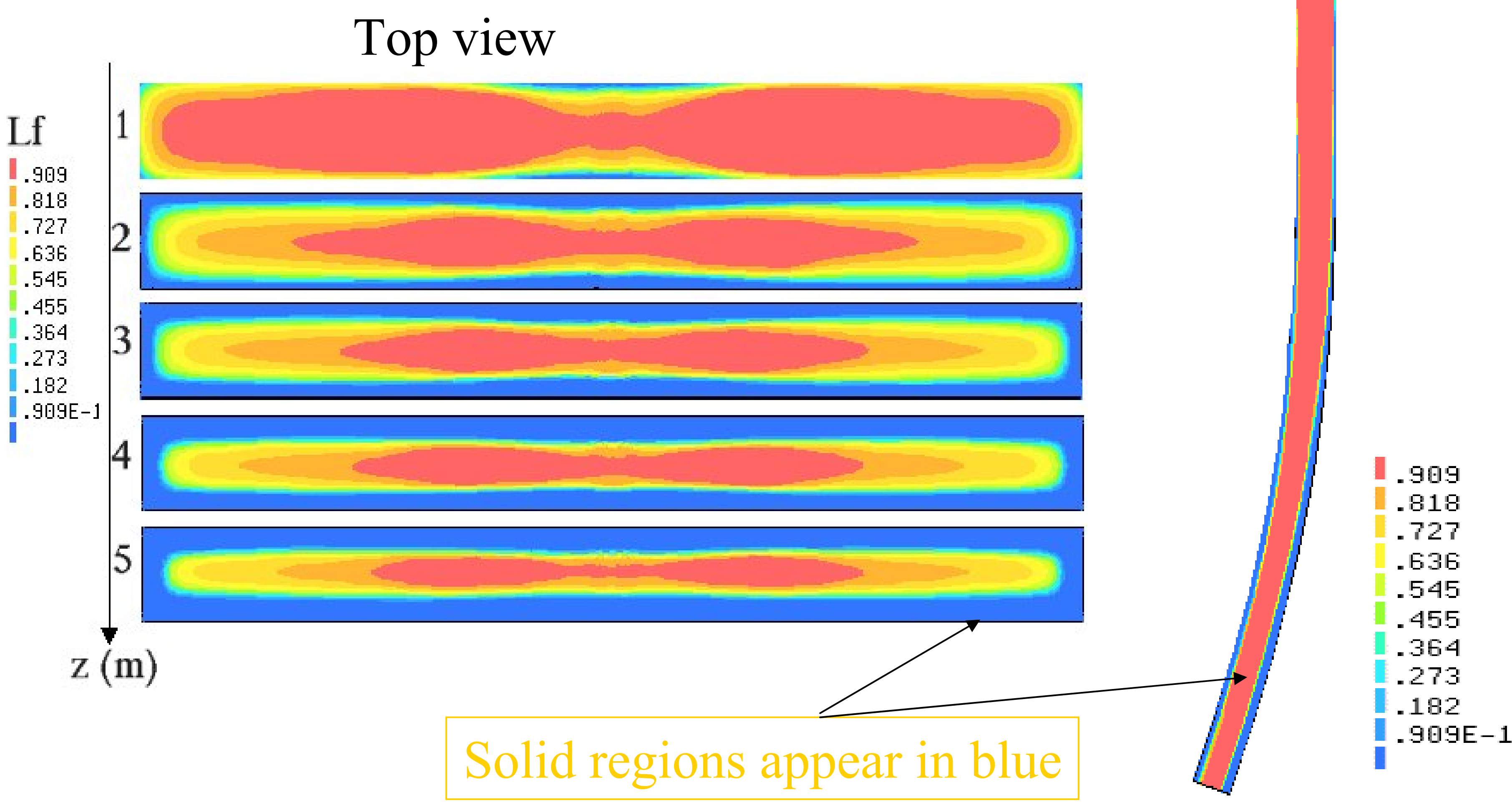
Argon bubble injection: closely coupled L-E approach



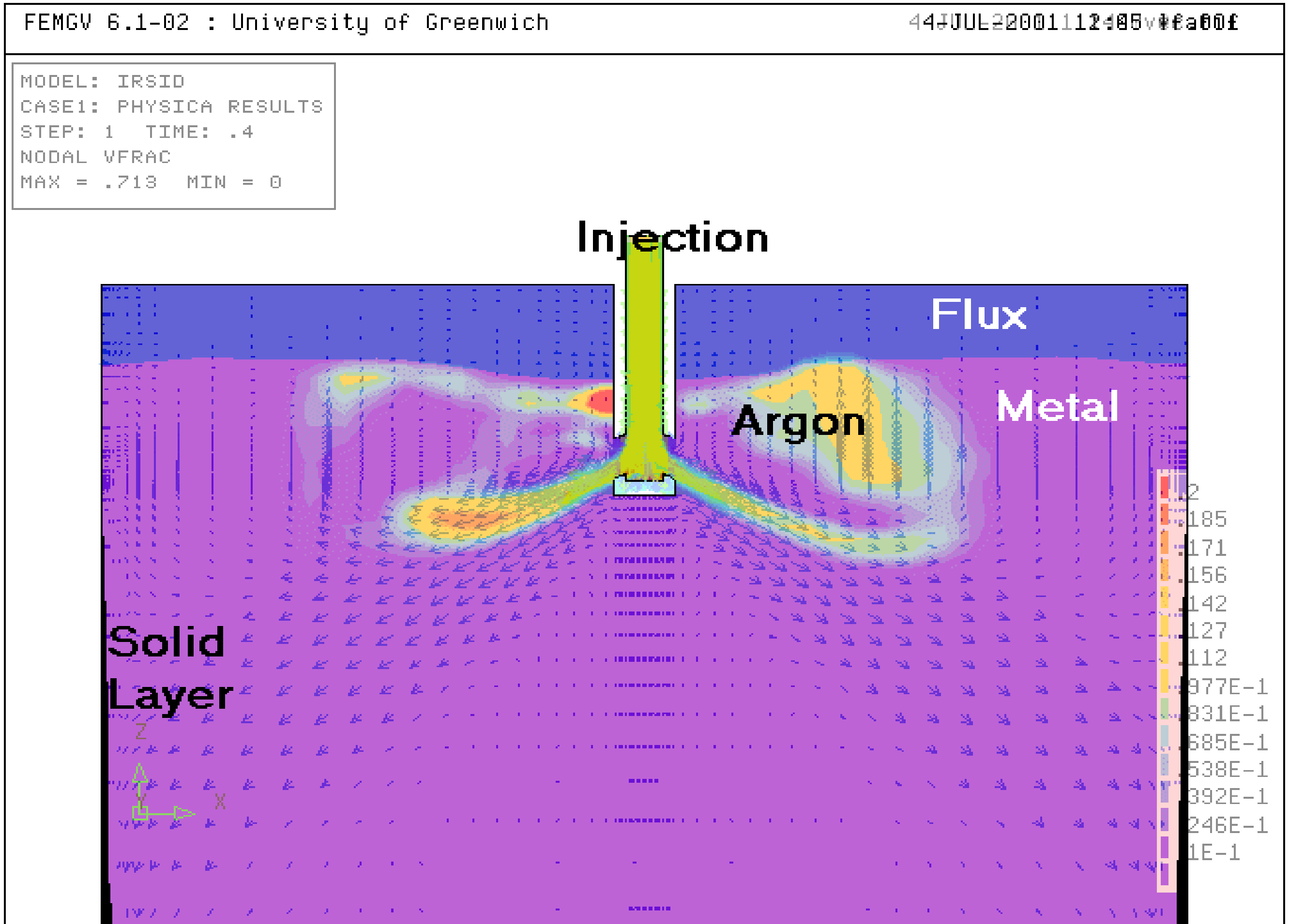
Solution domain



Solidification Strand



Clustering of argon bubbles



Solved using PHYSICA

Coupled EM-flow calculations

For most practical calculations in metals processing:

- The EM field influences the flow and thermal fields
- BUT the thermo-fluid phenomena has little influence of the EM fields
- Hence, essentially one way coupling
- So calculate the EM field and calculate the thermal and flow loads in the CFD calculation

Example: Electromagnetic brake simulations

Computations were also performed to estimate the effects of EMB on the free surface . For this the Maxwell equations were solved, which with the usual MHD assumptions, lead to:

Continuity of magnetic flux: $\nabla . \underline{B} = 0$

Ohm's Law for conducting metals

$$\underline{J} = \sigma(\underline{E} + \underline{U} \times \underline{B}),\ where\ \underline{E} = -\nabla\phi$$

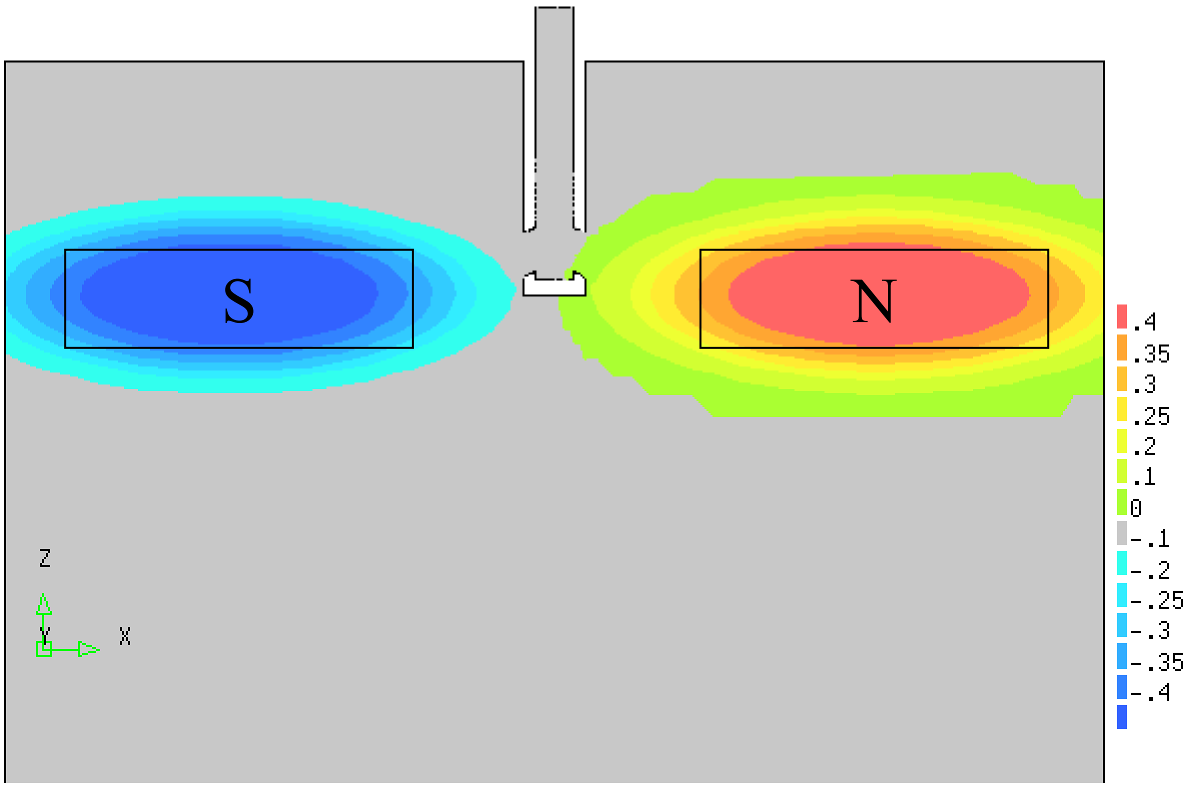
Magnetic Transport, or Induction equation

$$\frac{\partial \underline{B}}{\partial t} = \nabla \times (\underline{U} \times \underline{B}) + \eta \nabla^2 \underline{B} \quad where, \eta = \frac{1}{\sigma \mu_m}$$

Lorentz force: $F_L = J \times B$

Note: Terms containing the velocity $\underline{U}$, are only important when R_m $(= LU/\eta) > 1$

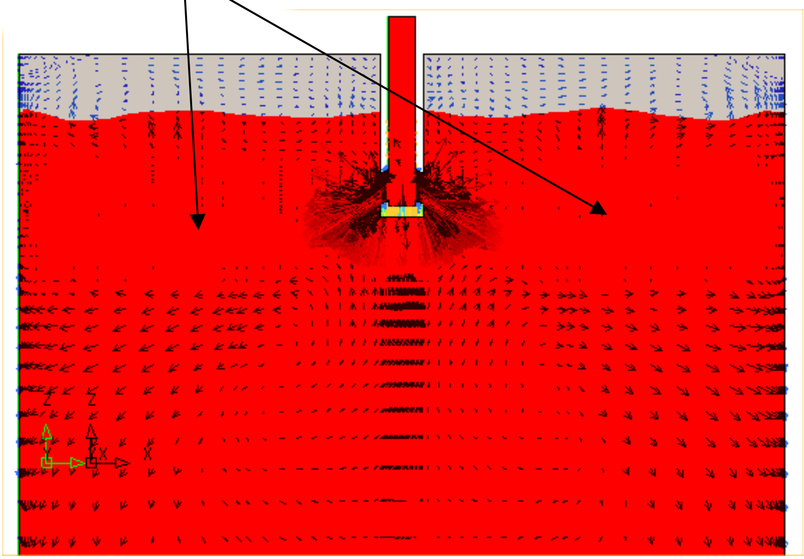
Brake arrangement



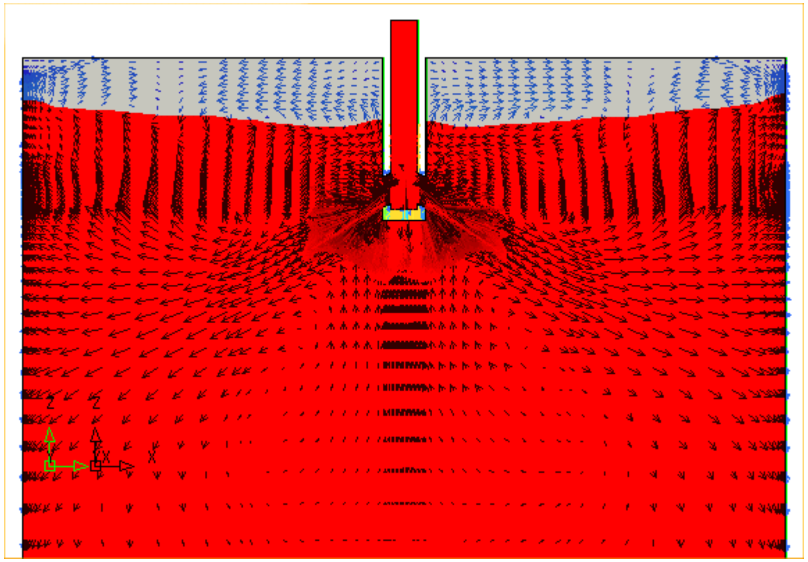
Two electromagnets of opposite polarity ($B_y = \pm 0.4T$) placed in the jet region to reduce velocity and hence, surface deformation

Fluid behaviour under EMB conditions

Flow suppressed here



B=0.4T

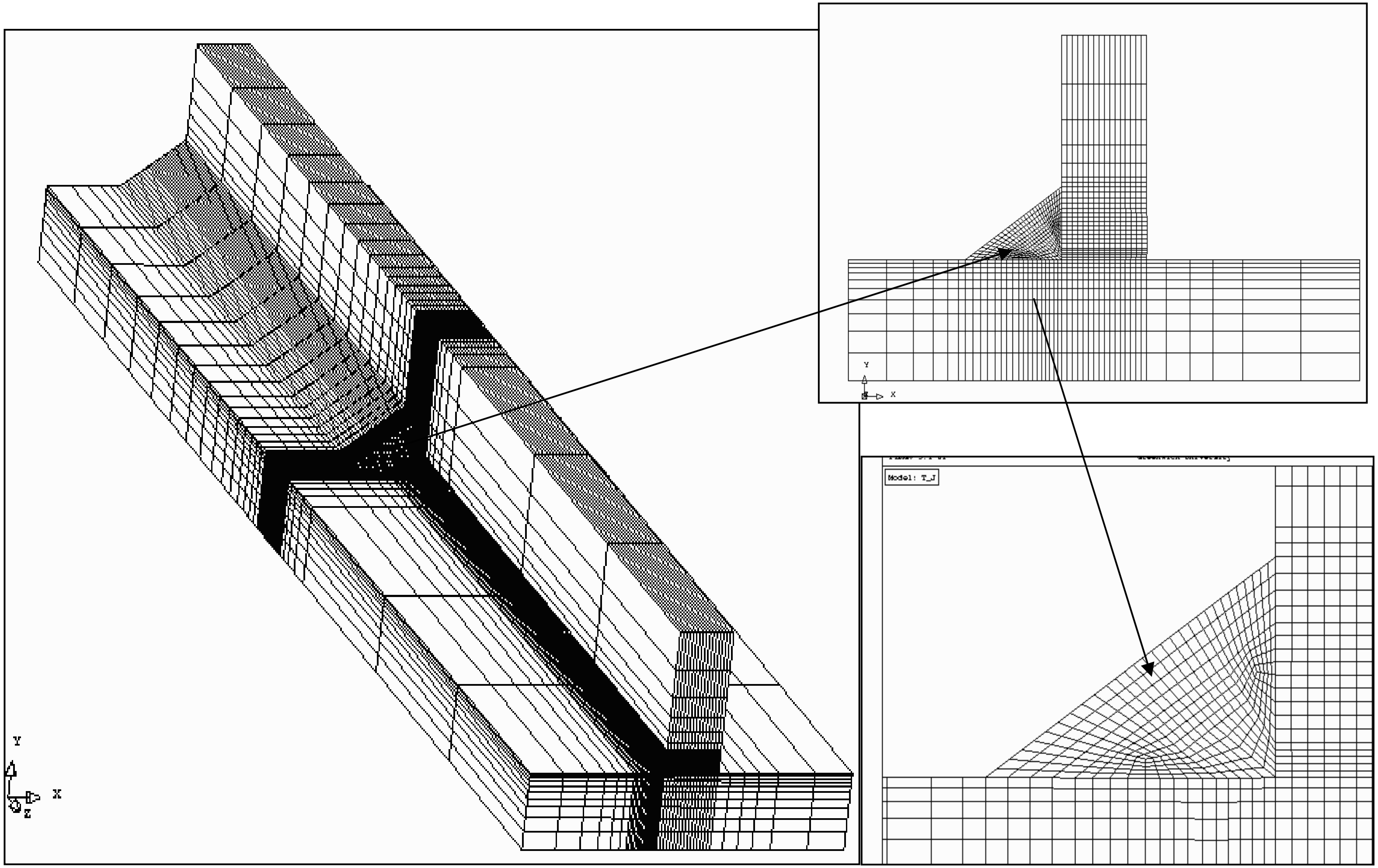


B=0T

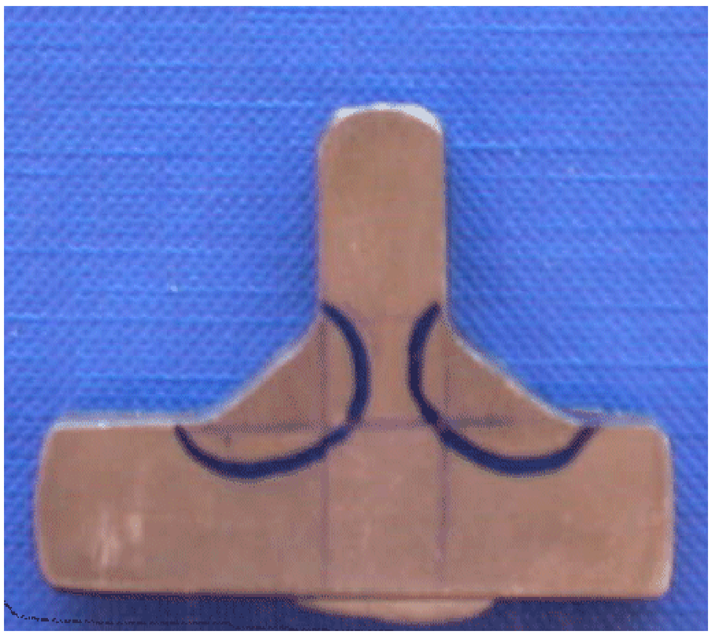
Welding processes simulation - natural multi-physics

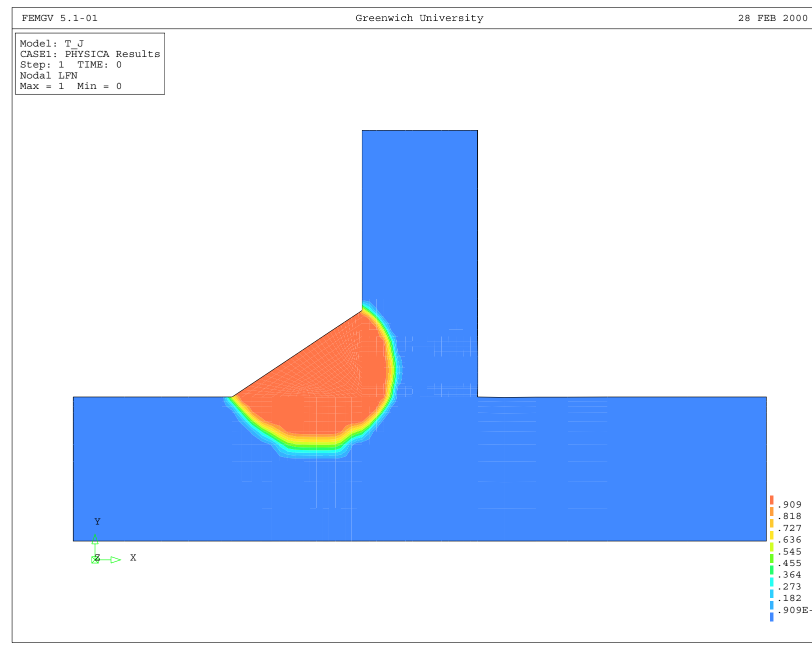
- Processes involve:
  - free surface flow
  - electromagnetic forces
  - heat transfer with solidification/melting
  - development of non-linear stress
- Ideal candidate for multi-physics modelling

T-Junction arc weld simulation



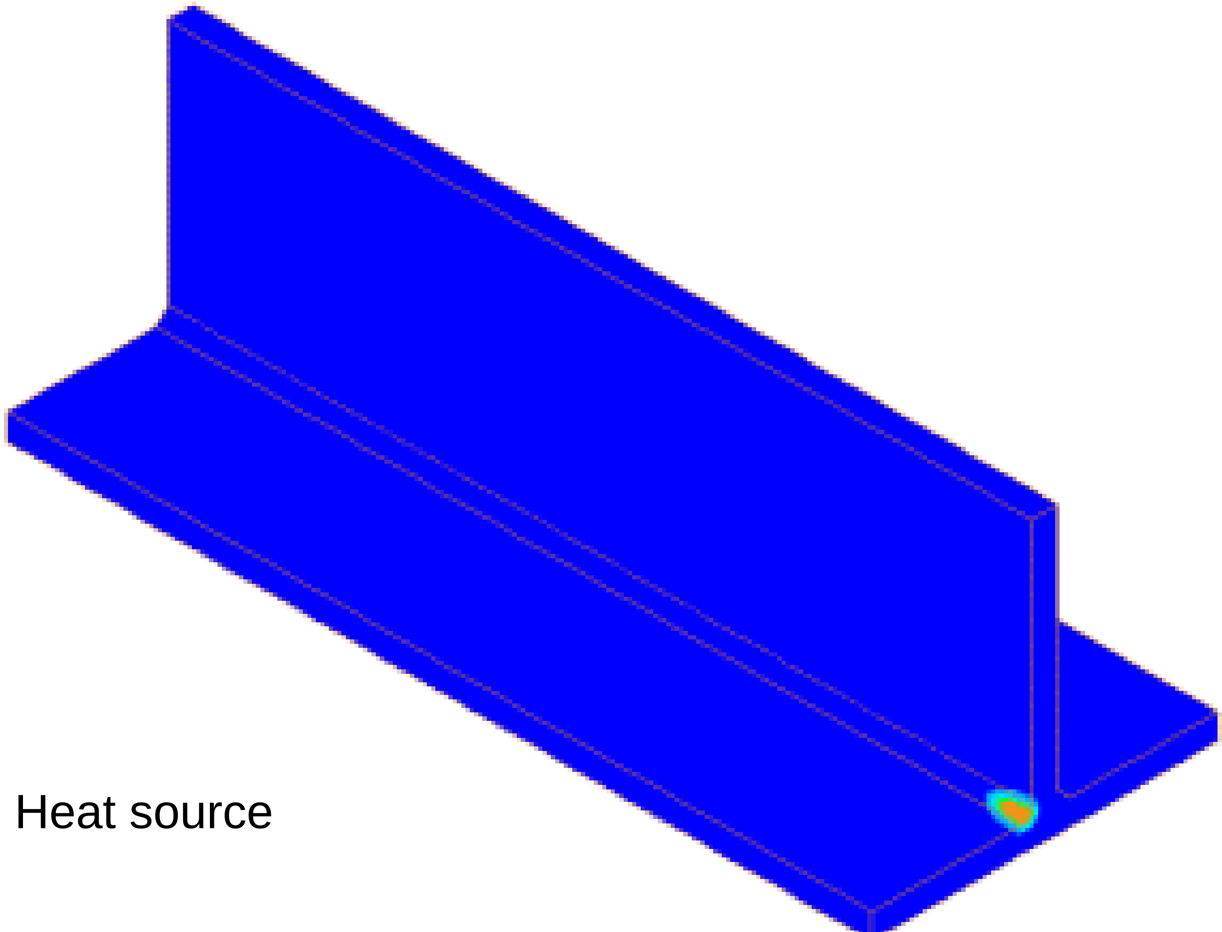
Experiment and simulation





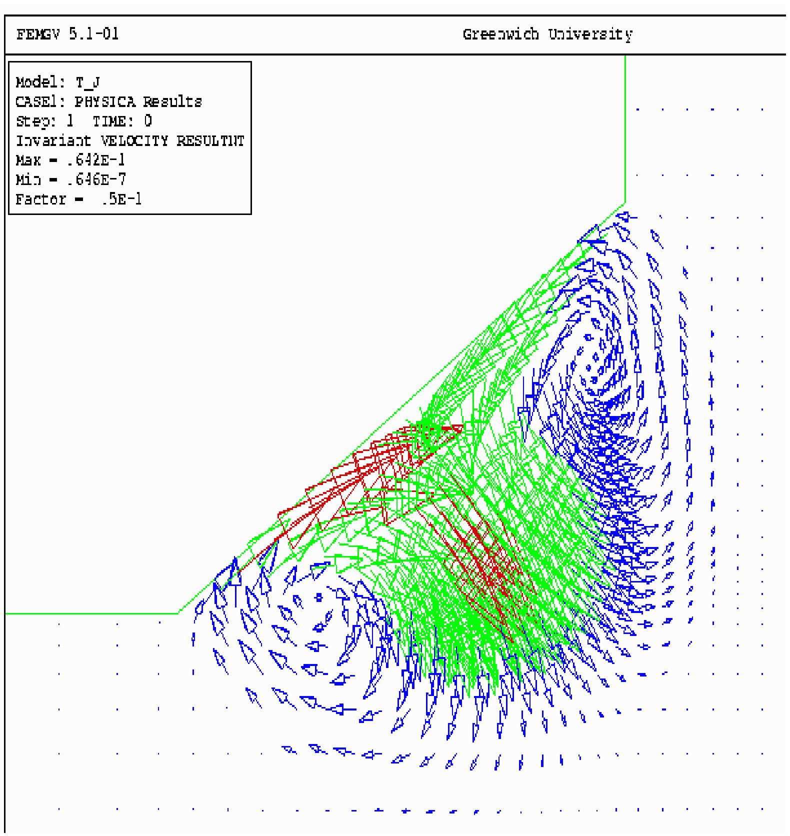
T-junction section, highlighting HAZ region

Distortion of T-junction due to heat source

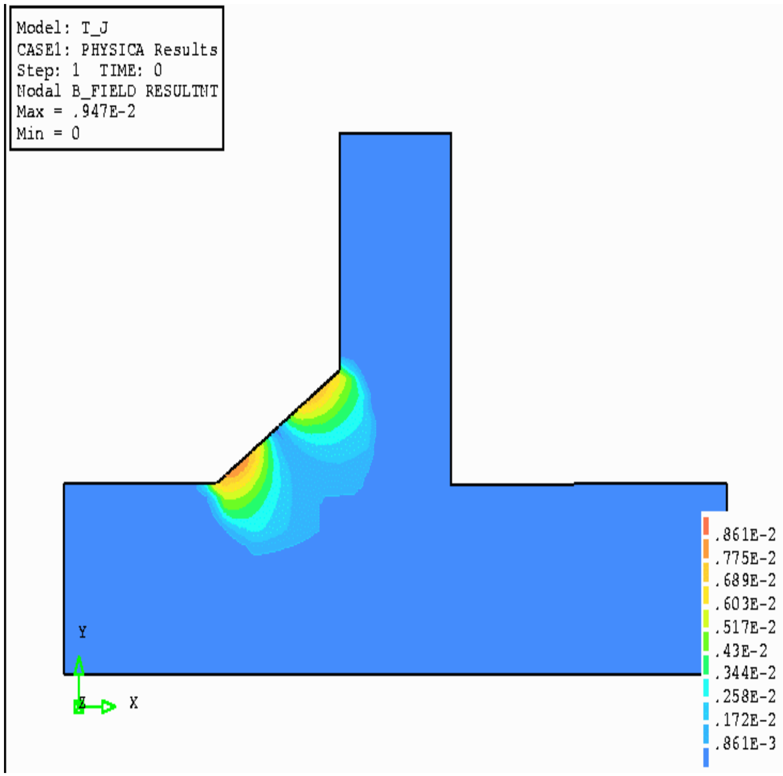


Heat source

Weld pool dynamics

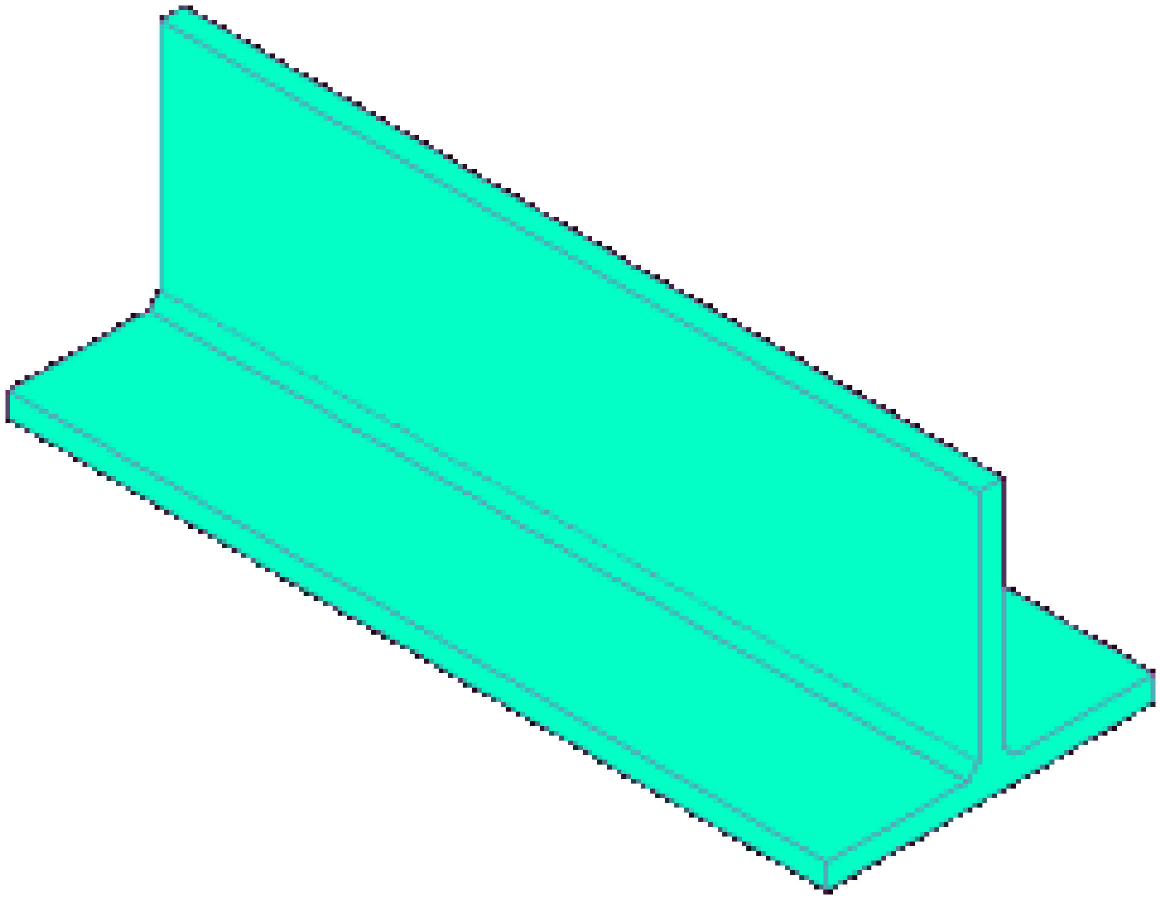


Velocity vectors in crossection



Lorentz force distribution in the weld-pool

Distortion of T-junction due to heat source



Welding – multi-physics BUT . .

- Welding involves:
  - free surface fluid flow
  - heat transfer and solidification/melting
  - electro-magnetic fields
  - non-linear stress

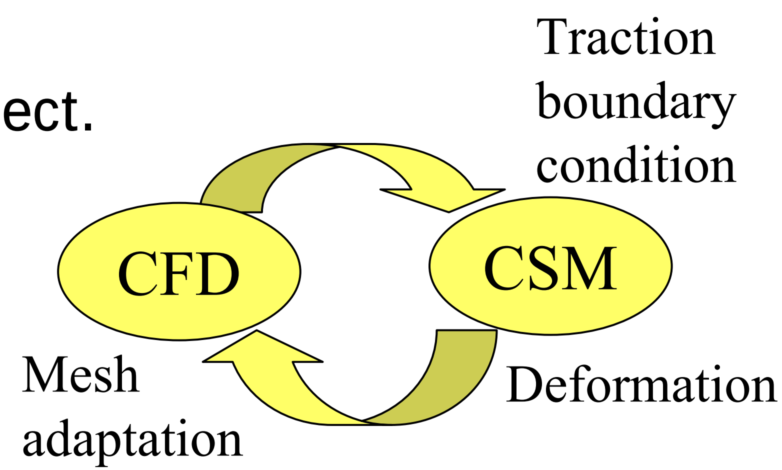
BUT . . no coupling back:

- from thermo-fluids to EM field
- from stress calculation to thermo-fluids

SO . . reasonably loosely coupled

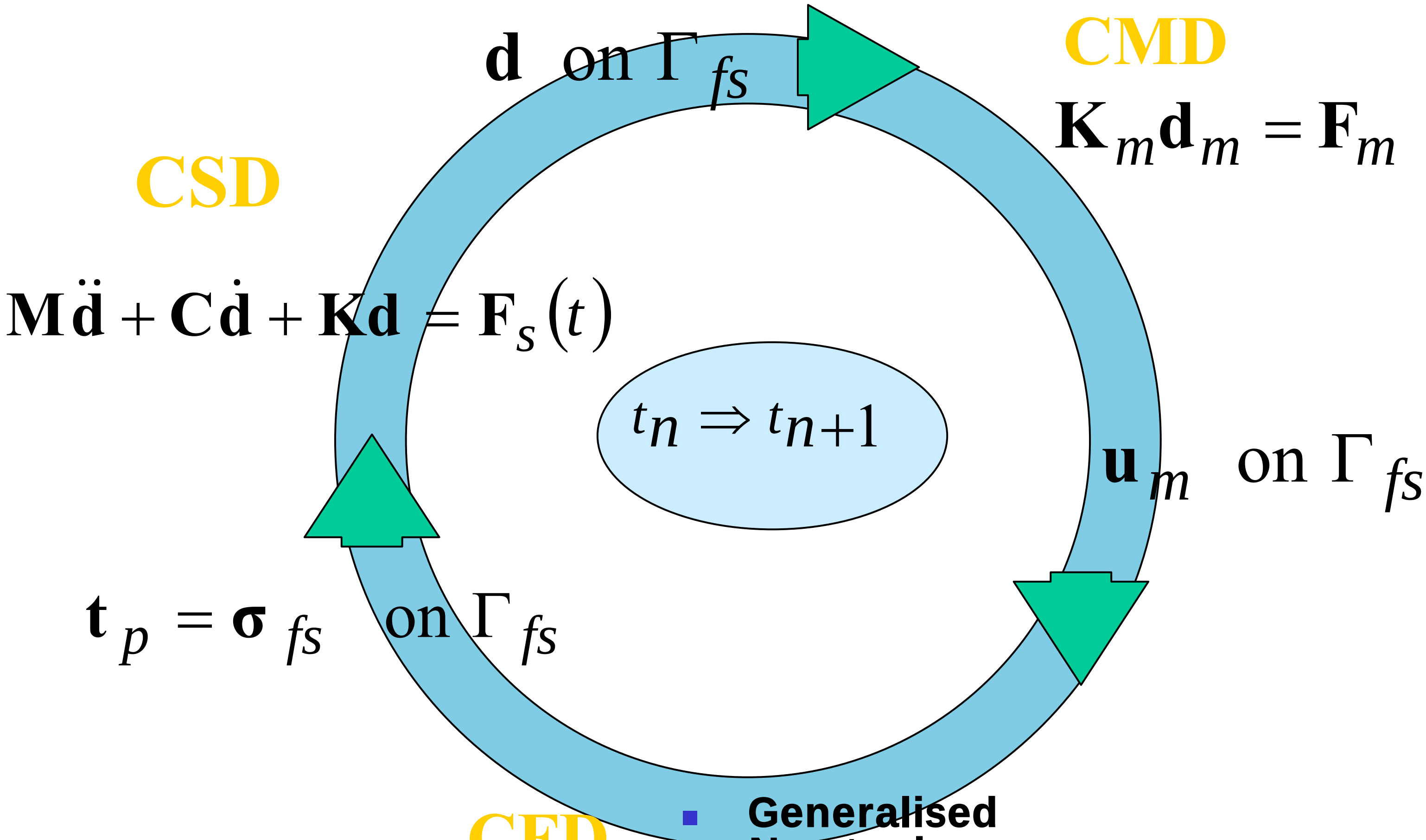
Generic Dynamic Fluid Structure Interaction

- Closely coupled multi-disciplinary problem
  - Time & space accurate
  - Very challenging in every respect.
  - Issue of GCL



  - Implementation of boundary conditions.
- Features of single software framework:
  - Consistency of mesh.
  - Single database & memory map.
  - Compatibility in the solution approaches FV-UM.

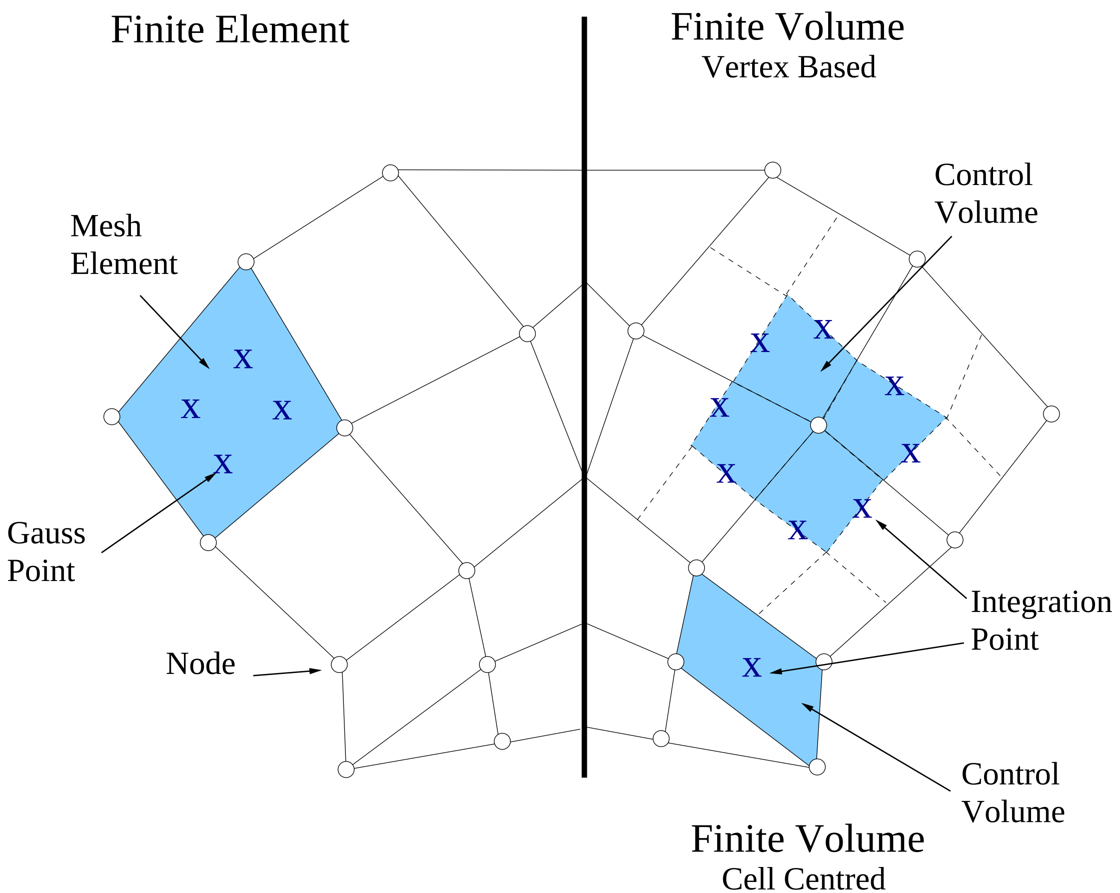
Three Phase Approach



- Generalised Newtonian
- Continuity

Spatial Discretisation for closely coupled multi-physics

- Unstructured mesh
- CFD
  - Cell centred
  - Or mixed CC- VB
  - FV
- CSM
  - Vertex based
  - FV/FE



Structural Dynamics

- Equilibrium Equation $\mathbf{L}^{\mathbf{t}}\sigma + \mathbf{b} - \rho \dfrac{d^2}{dt^2}(\mathbf{d}) = 0$

- Method of Weighted Residuals
  - Greens 1st theorem
    - where $\mathbf{d} \approx N_j \hat{d}_j$ evaluated at nodes

$$\int_\Omega \mathbf{W}_i^T \rho N_j \frac{d^2 \hat{d}}{dt^2}\,\mathrm{d}\Omega + \int_\Omega (\mathbf{L}\mathbf{W}_i)^T \mathbf{D}\mathbf{L}N_j \hat{d}\,d\Omega - \oint_{\Gamma_d} \mathbf{W}_i^{\mathrm{T}} \mathbf{T}\mathbf{D}\mathbf{L}N_j \hat{d}\,\mathrm{d}\Gamma$$

$$= \int_\Omega \mathbf{W}_i^T \mathbf{b}_0\,\mathrm{d}\Omega + \oint_{\Gamma_{t_i}} \mathbf{W}_i^{\mathrm{T}} t_p\,d\Gamma + \int_\Omega (\mathbf{L}\mathbf{W}_i)^T \sigma_{\mathbf{0}}\,\mathrm{d}\Omega - \oint_{\Gamma_{d_i}} \mathbf{W}_i^{\mathrm{T}} \mathbf{T}\sigma_{\mathbf{0}}\,\mathrm{d}\Gamma$$

Dynamic Structural Mechanics

- Compact matrix form of equilibrium equation

$$\mathbf{M}\frac{d^2}{dt^2}\left(\hat{\mathbf{d}}\right)+\mathbf{C}\frac{d}{dt}\left(\hat{\mathbf{d}}\right)+\mathbf{K}\hat{\mathbf{d}}=\mathbf{f}$$

  - where C is the damping matrix

$$\mathbf{M}_{ij}=\int_{\Omega_i}\mathbf{W}_i^T\rho N_j\;d\Omega$$

$$\mathbf{K}_{ij}=\int_{\Omega_i}\left(\mathbf{L}\mathbf{W}_i\right)^T\mathbf{D}\mathbf{L}N_j\;d\Omega-\oint_{\Gamma_{d_i}}\mathbf{W}_i^T\mathbf{T}\mathbf{D}\mathbf{L}N_j\;d\Gamma$$

$$\mathbf{f}_\mathbf{i}=\int_{\Omega_i}\mathbf{W}^T\mathbf{b}_0\;d\Omega+\oint_{\Gamma_{d_i}}\mathbf{W}_i^T\mathbf{t}_p\;d\Gamma+\int_{\Omega_i}\left(\mathbf{L}\mathbf{W}_i\right)^T\boldsymbol{\sigma}_0\;d\Omega-\oint_{\Gamma_{d_i}}\mathbf{W}_i^T\mathbf{T}\boldsymbol{\sigma}_0\;d\Gamma$$

  - traction boundary condition on fluid – structure boundary

$$t_p=-p\delta_{ij}+\mu\left(\frac{\partial \mathrm{u}_\mathrm{i}}{\partial \mathrm{x}_\mathrm{j}}+\frac{\partial u_j}{\partial x_i}\right)-{}^2\!/_3\,\mu\nabla\cdot\mathbf{u}\,\delta_{ij}\text{ on }\Gamma_s$$

CSM Spatial Formulation

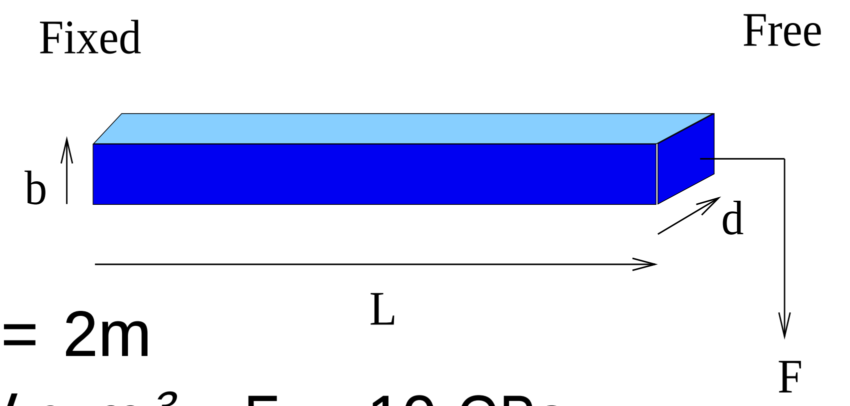
- Essential difference between FE & FV Weighting Functions
  - FE $\mathbf{W}_i = N_i$
    direct association between
    N_i & element
  - FV $\mathbf{W}_i = \mathbf{I}$
    within cv
    zero elsewhere
- Mass matrix
  - FE $$\mathbf{M}_{ij} = \int_{\Omega_i} N_i^T \rho N_j \mathrm{d}\Omega$$
  - FV $$\mathbf{M}_{ij} = \int_{\Omega_i} \rho N_j \mathrm{d}\Omega$$

CSM Spatial Formulation

- Stiffness matrix
  - FE $$\mathbf{K}_{ij} = \int_{\Omega_i} \left(\mathbf{L}N_j\right)^T \mathbf{D}\mathbf{L}N_j \, d\Omega$$
  - FV $$\mathbf{K}_{ij} = -\oint_{\Gamma_i} \mathbf{T}\mathbf{D}\mathbf{L}N_j \, \mathrm{d}\Gamma$$
- Load vector
  - FE $$\mathbf{f} = \int_{\Omega} N_i^{\,T} \mathbf{b}_0 \, \mathrm{d}\Omega + \oint_{\Gamma_{t_i}} N_i^{\,\mathrm{T}} t_p \, d\Gamma + \int_{\Omega} \left(\mathbf{L}N_i\right)^T \boldsymbol{\sigma}_\mathbf{0} \, \mathrm{d}\Omega$$
  - FV $$\mathbf{f} = \int_{\Omega} \mathbf{b}_0 \, \mathrm{d}\Omega + \oint_{\Gamma_{t_i}} t_p \, d\Gamma - \oint_{\Gamma_{d_i}} \mathbf{T} \boldsymbol{\sigma}_\mathbf{0} \, \mathrm{d}\Gamma$$

Comparison of FE and FV performance

- 3D cantilever



  - L = 20m , b = d = 2m
  - $\nu = 0.2,\ \rho = 2600$ *kg m^{-3}* , E = 10 GPa
  - F = 2000 N
- Mesh
  - 80x8x8
  - 5120 elements and 6561 nodes

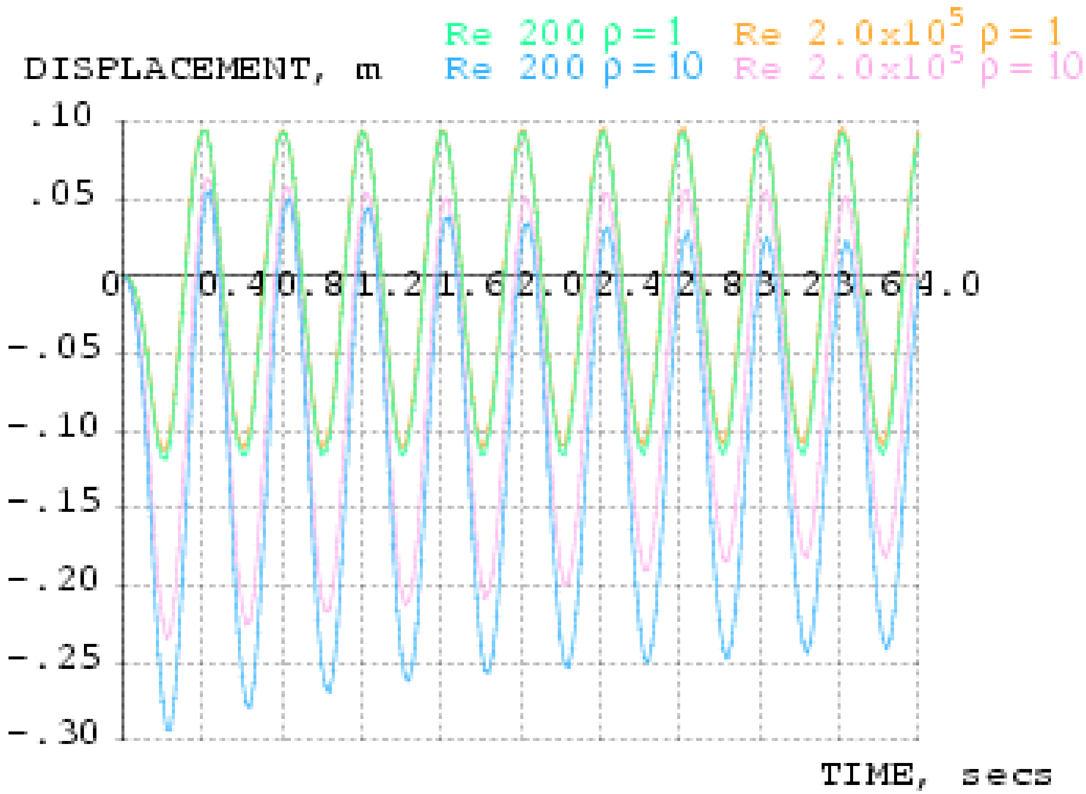
- Analytic 2d solution Fenner ν = 0

Comparison of FE and FV performance

- 3D cantilever, static results on Dec Alpha 466 MHz processor

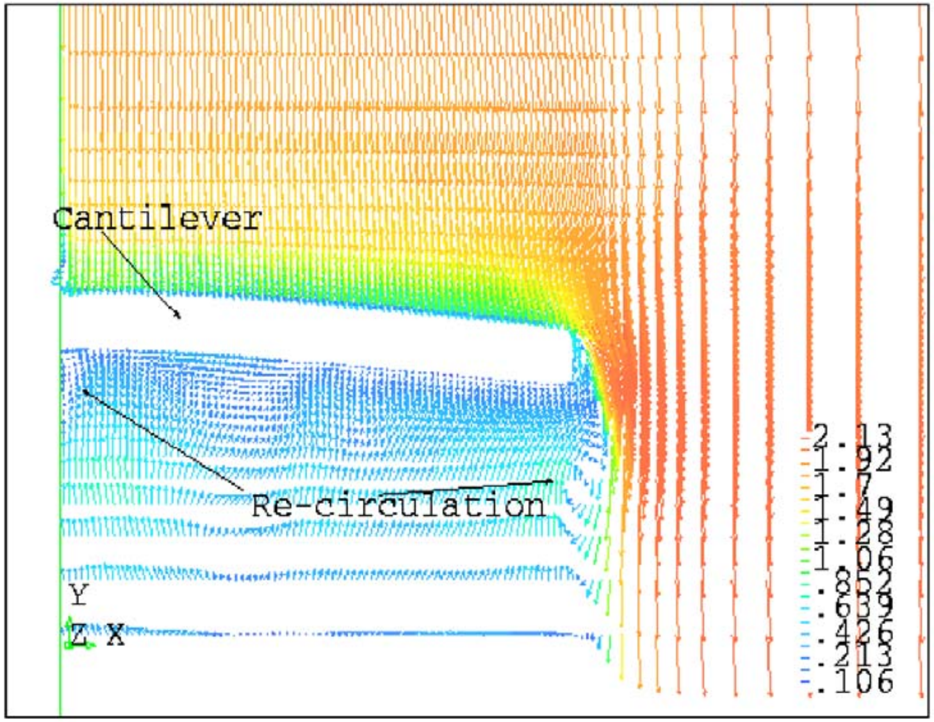
	Iterations			Run times, second		
ν	FE JCG	FE-BICG	FV-BICG	FE JCG	FE-BICG	FV-BICG
0.3	539	540	544	48	95	98
0.2	483	483	483	44	85	88
0.0	438	438	437	38	78	80

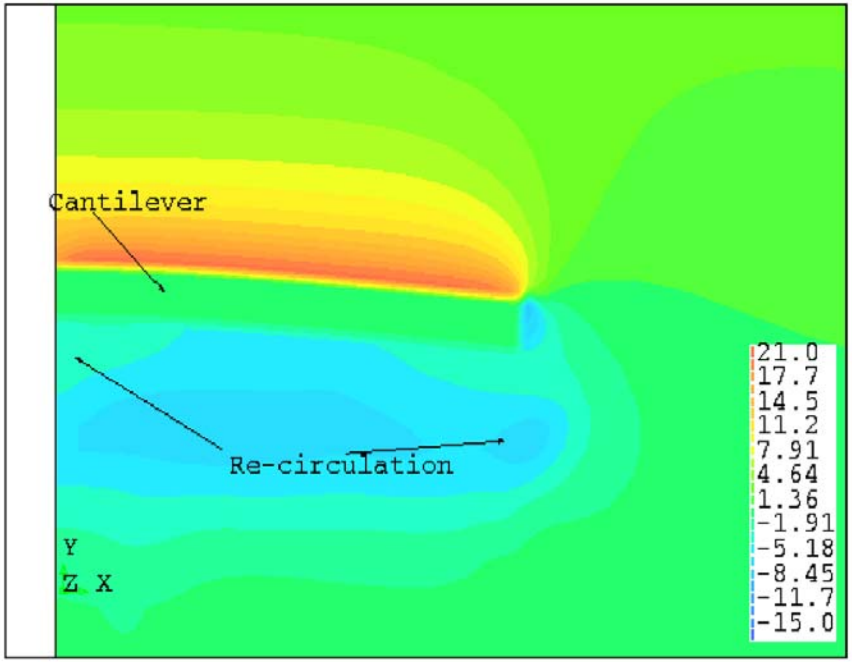
Cantilever Dynamic Displacement



Fluid velocity & pressure fields

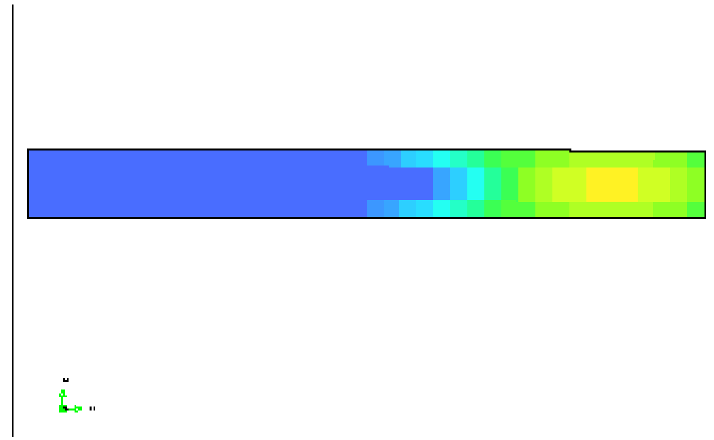
Re 4000



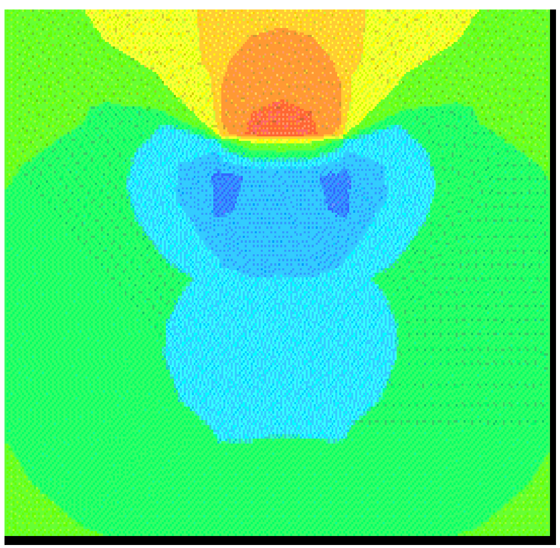


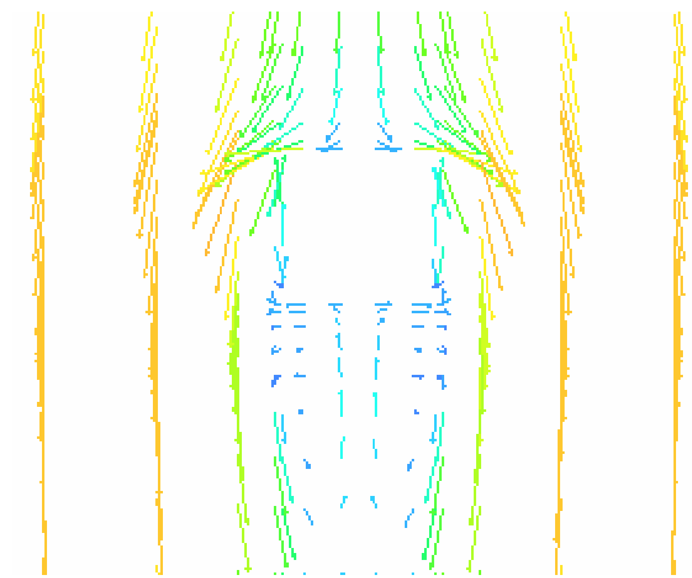
Cantilever interaction

- Neutral z plane shear *xy* stress



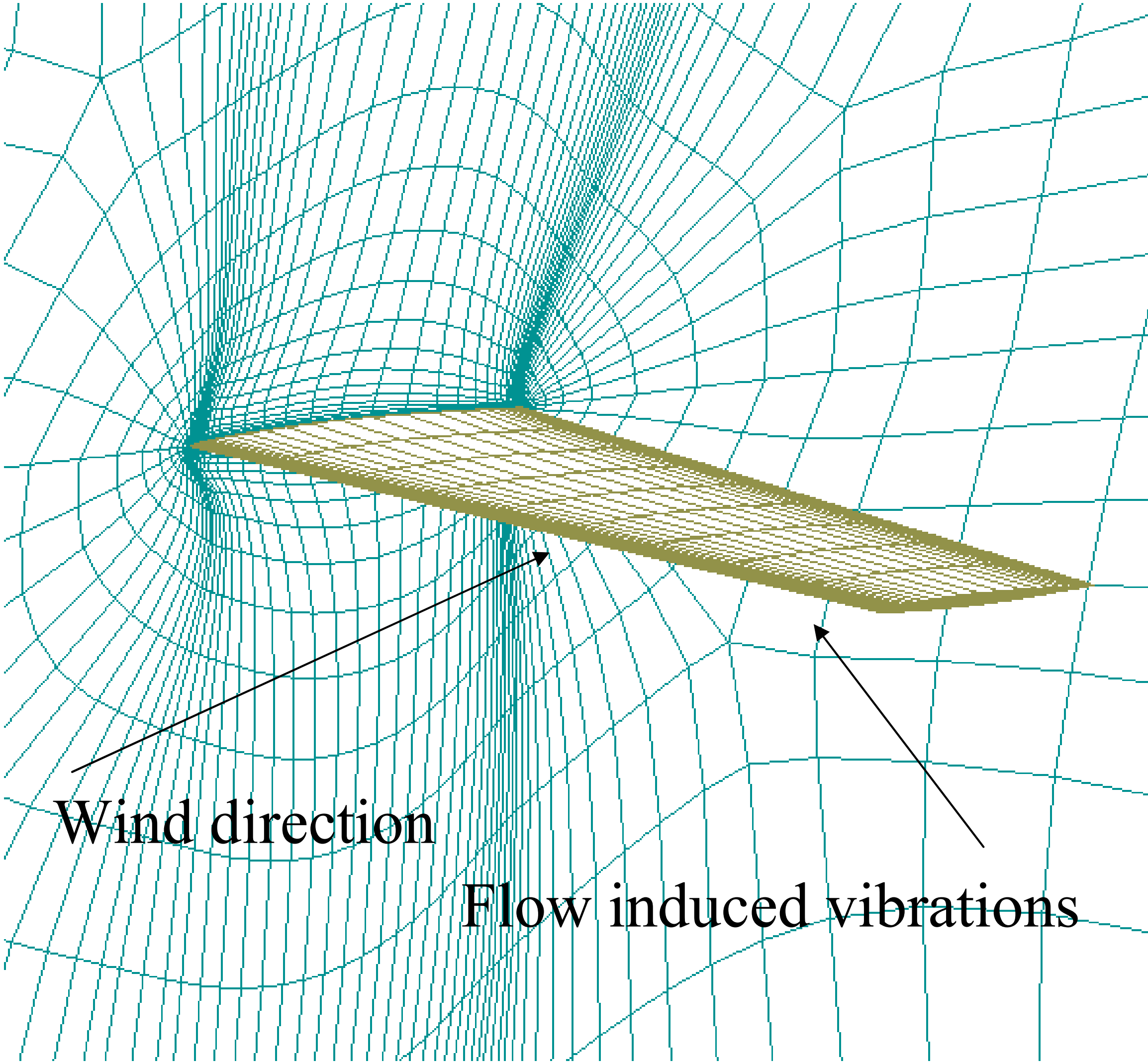
- Centre of cantilever length



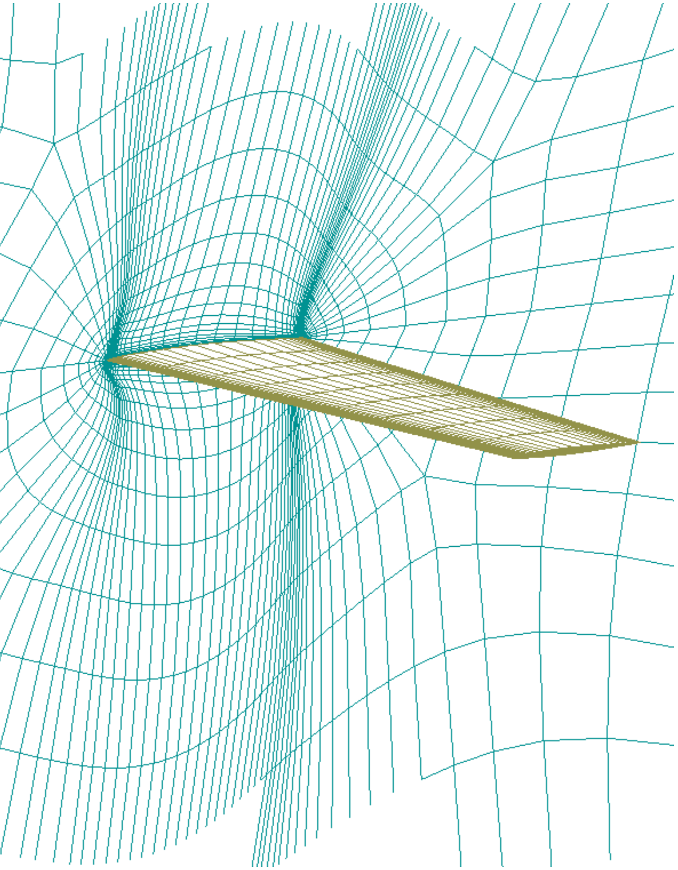


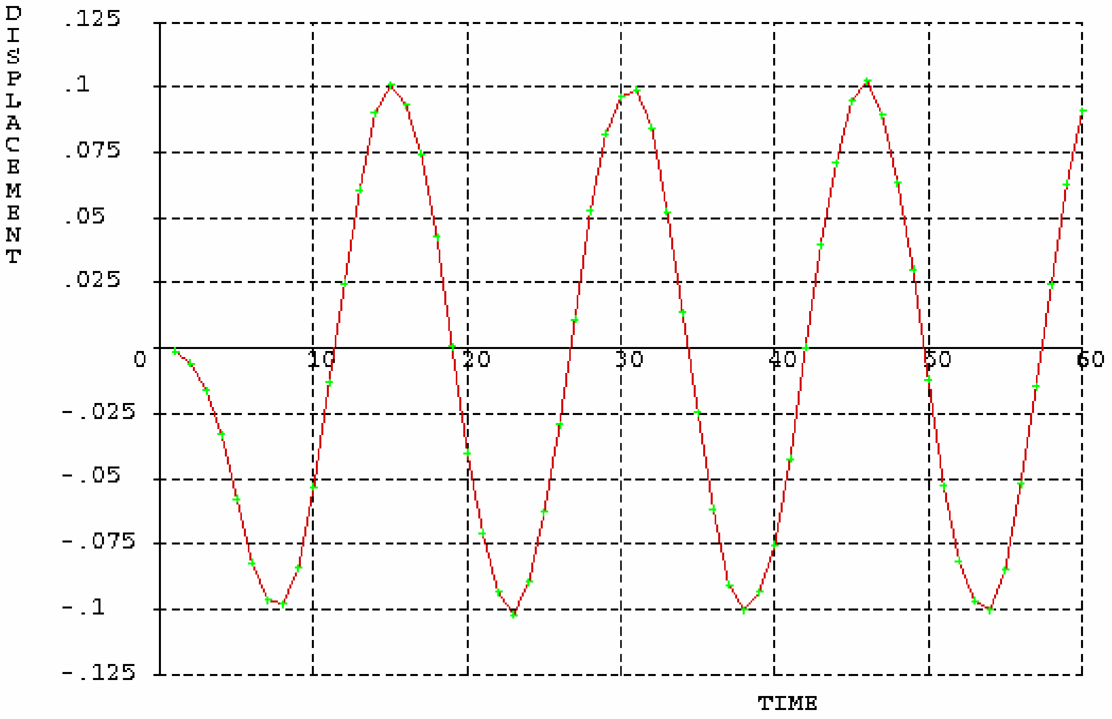
Dynamic fluid-structure interaction

- Targeted at problems involving flow induced vibrations
- Use dynamic structural equations and Navier-Stokes flow equations
- Objective: move to VB flow and FE based dynamics

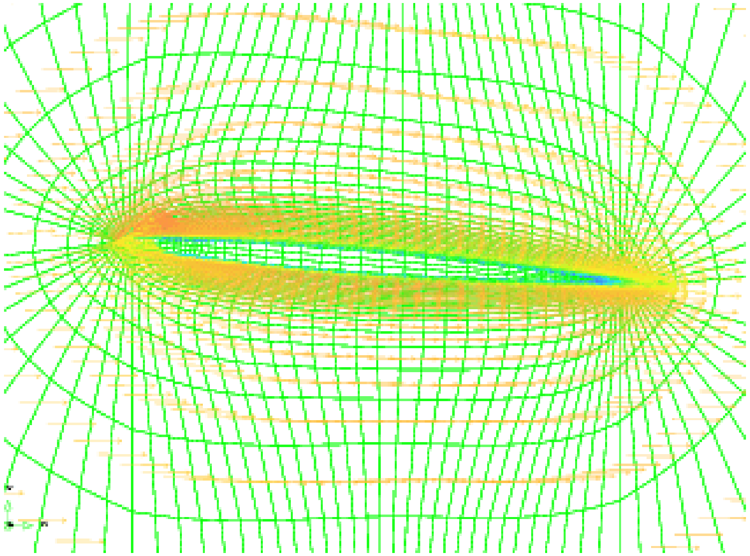


Dynamic response of structure without flow

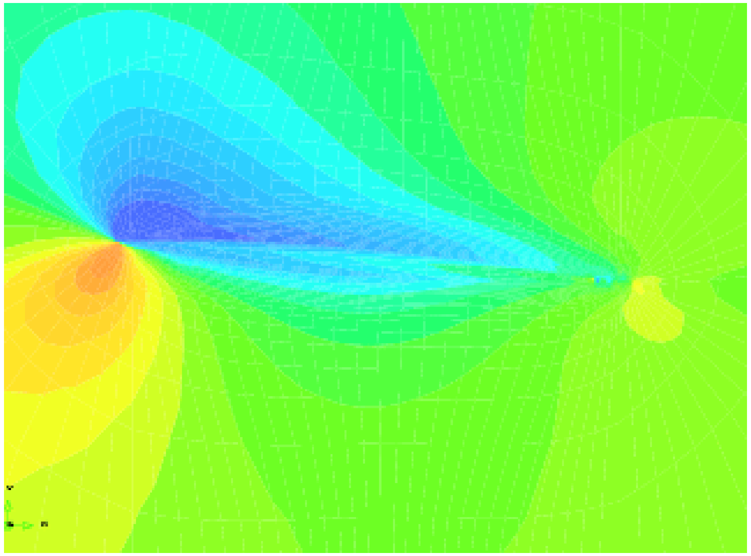




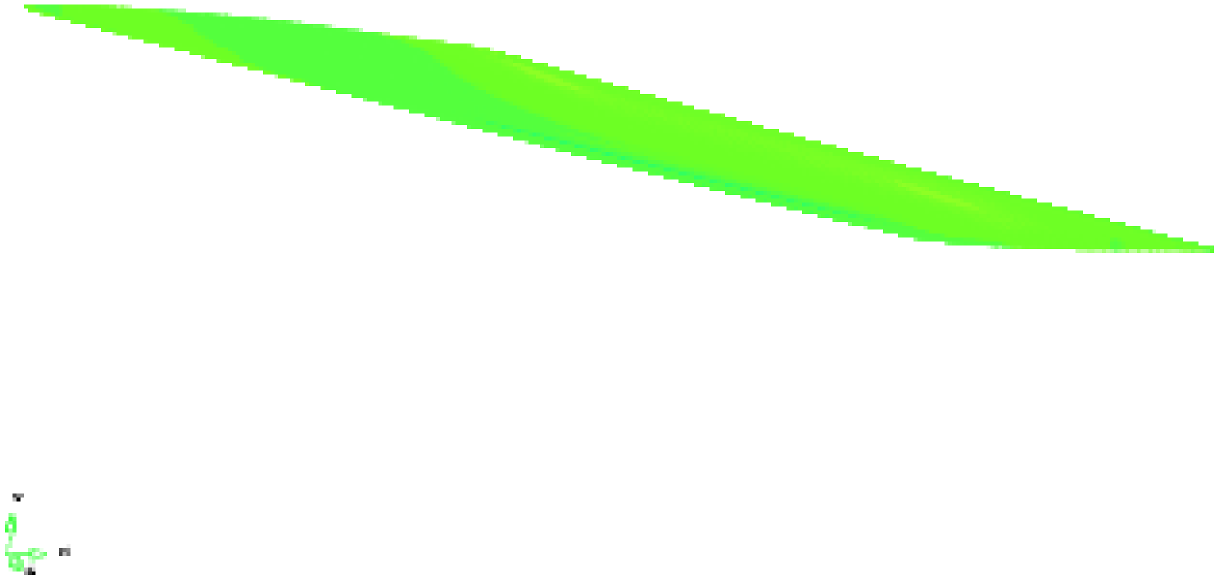
Fluid Velocity and Pressure Movies



At tip of wing



Shear Stress σ_{xy} Movie



Parallel Multi-Physics Modelling

Multi-Physics 2006, Maribor
December 2006

Multi-physics compute demands:
secs per node(elt) / time step per problem class

- Unstructured Mesh analysis = 3* Structured mesh analysis
- Performance on a Compaq alpha 466Mhz

  - Heat Transfer (HT) + Solidification (Sol) = 2. 10-3
  - Fluid Flow (FF) + HT + Sol = 6.10-3
  - HT + Sol + Stress = .09
  - FF + HT + Sol + Stress = .14

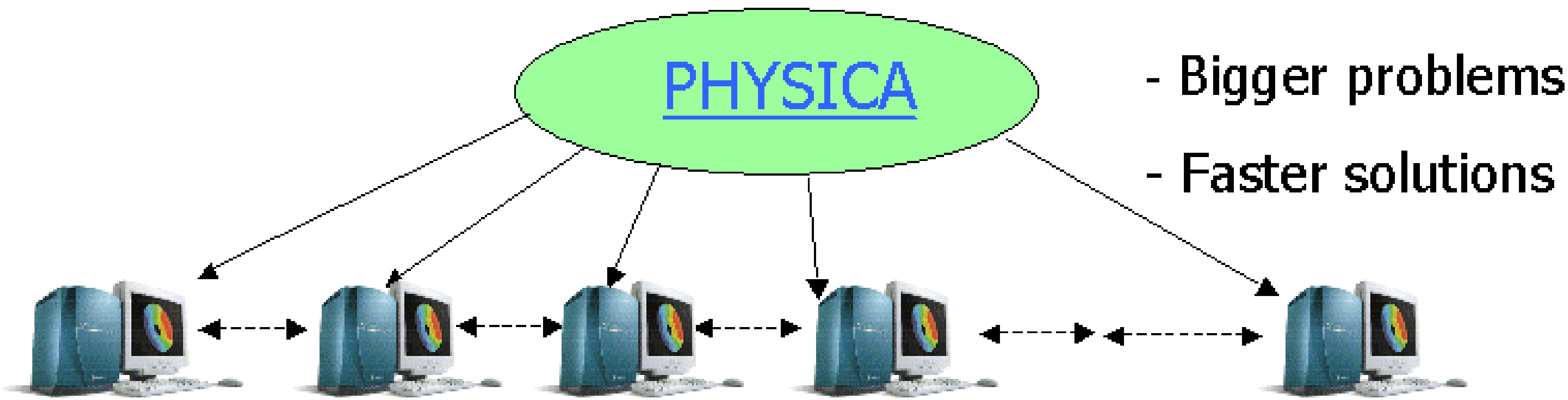
=> a casting simulation with 100K nodes, and 100 time steps is 300+hrs!

- We need simulation times 100x faster

PARALLEL – WITH CHANGING PHYSICS

Parallel Strategy -PHYSICA

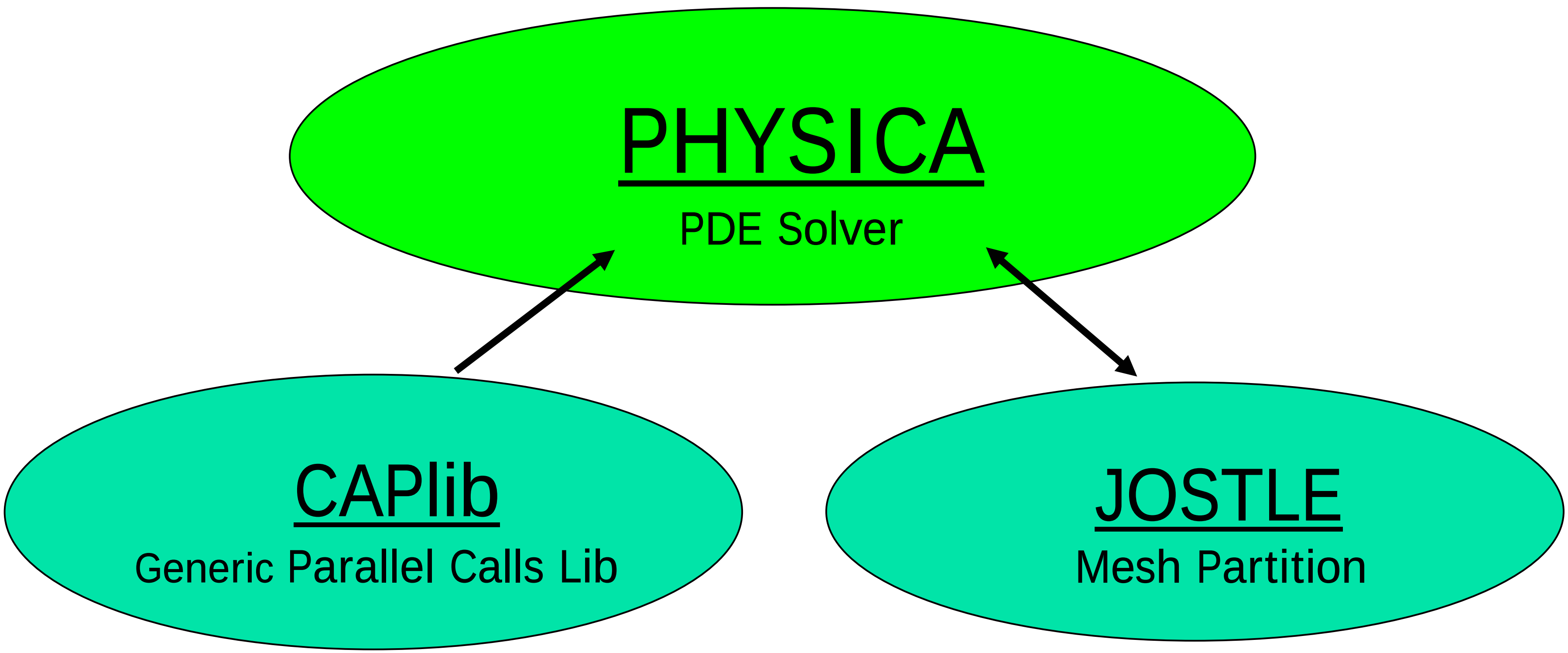
- Single Program Multiple Data (SPMD)



- Program resident on each processor
- Mesh Partitioned across processors.
- Minimise communication times.

Parallel Multi-Physics Framework

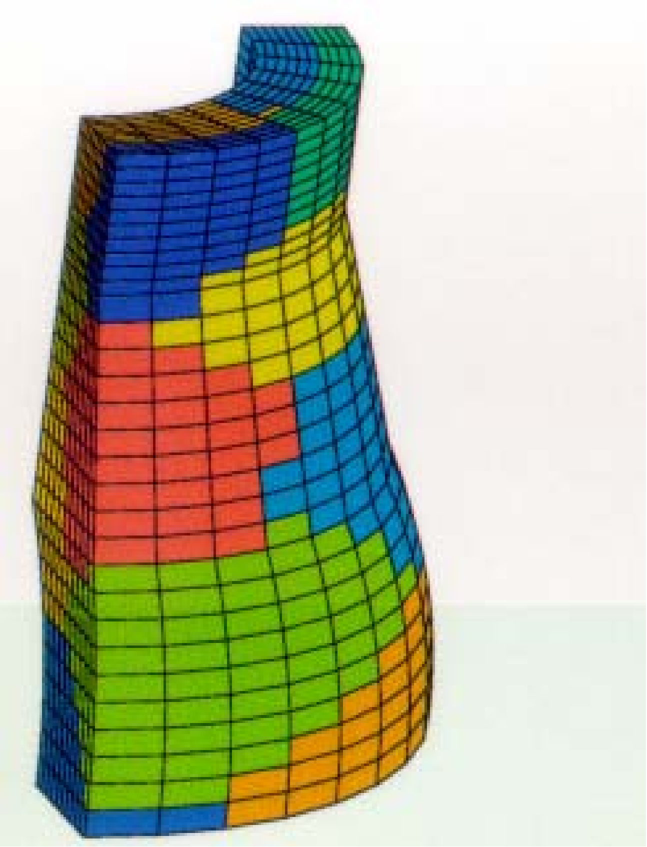
Simulations very Time Consuming – need Parallel capability



Parallelisation approach uses mesh partitioning
SPMD strategy with non-uniform workload

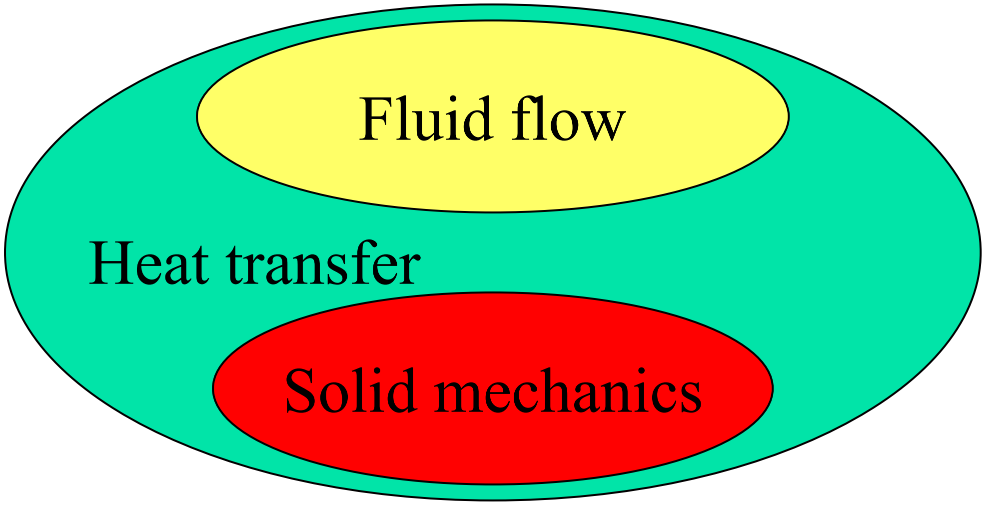
Partition of 3D unstructured mesh by JOSTLE assuming a homogeneous load balance across the mesh:

- **load balanced** (even no of cells per node)
- **minimises** sub-domain interface elements
- sub-domain connectivity **matches processor topology** of the parallel system



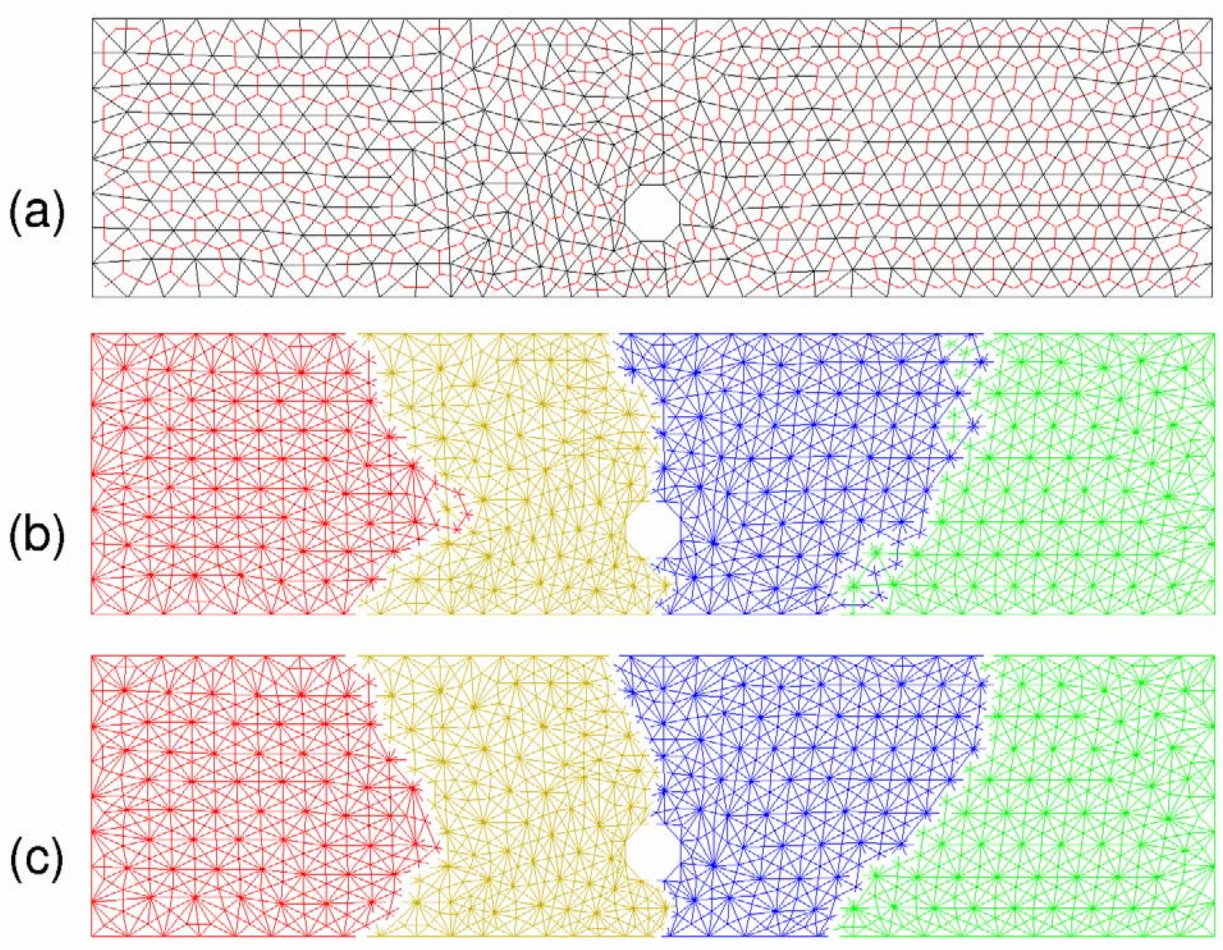
Multi-physics Simulation parallel issues

- Sub-domains have specific physics so partition must reflect this:
  - **non-uniform load/node**
- Distinct physics uses distinct discretisation procedures:
  - **2ndary partitions**
- Also, sub-domains may change as problem develops:
  - **dynamic load balance**



Strategy needs to address all the above issues

Primary & secondary partitions



Primary & secondary meshes

Good primary & poor Secondary partition

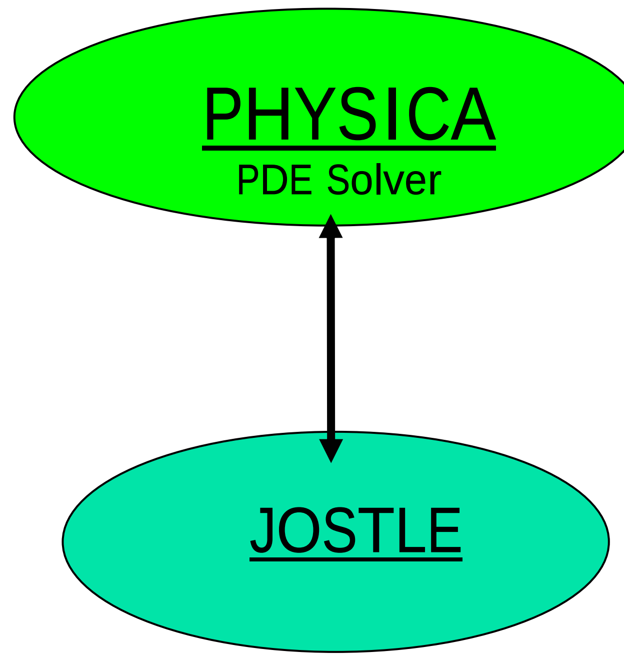
Good primary & Secondary partitions from JOSTLE

Parallel multi-physics: two level approach

- **Implement a generic parallel version of Multi-physics code / MDA codes**

  - without regard to in-homogeneity of the computational work over the mesh(es) defining the analysis domain

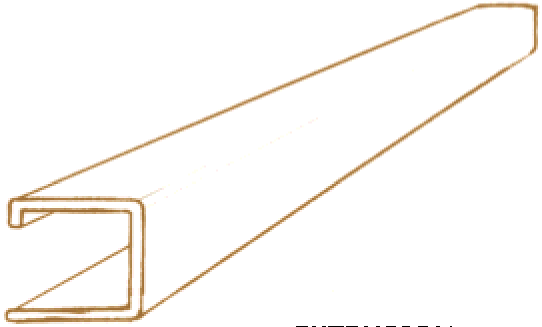
- **Dump the load balancing into the mesh (re)partitioning task** - JOSTLE_DLB
- **Process as straightforward as possible**



Mesh Partition

Metal Forming - Extrusion

- Involves large scale deformation of metal work-piece through interaction with one or more dies
- Multi-physics problem
  - Flow/deformation of work-piece
  - Heat transfer generated by internal friction
  - Stress/strain in die(s)



Mixed Eulerian-Lagrangian Approach

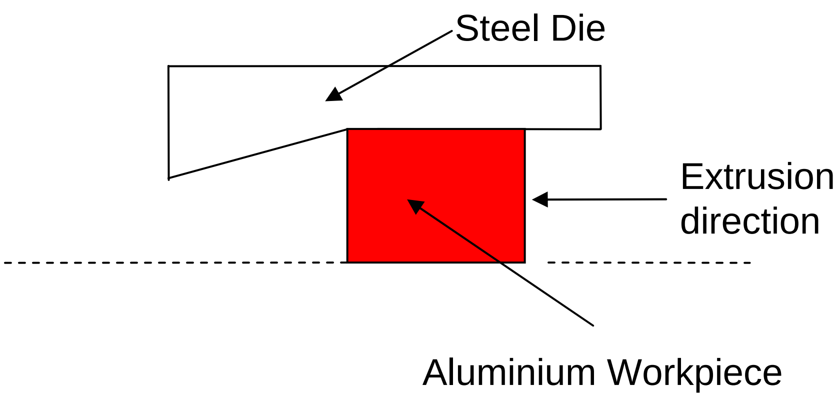
Workpiece

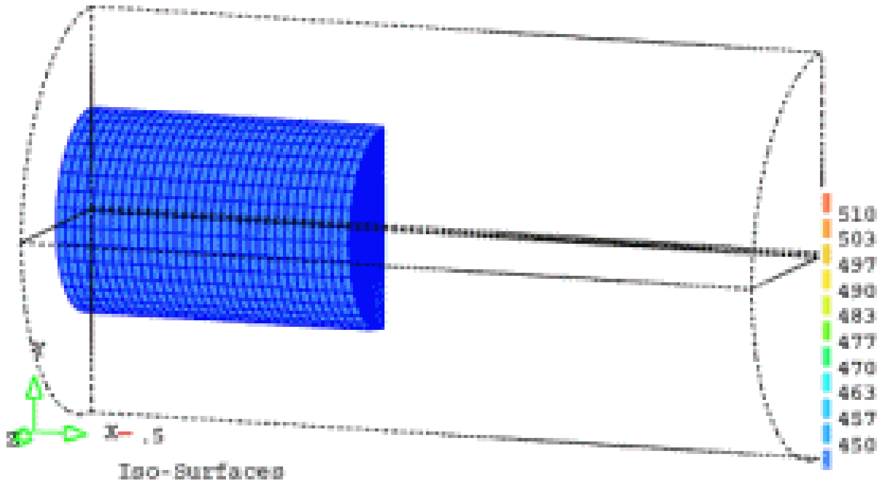
- Eulerian mesh
- Free-surface algorithm to track deformation
- Non-Newtonian material model
- Heat transfer plus energy generated by internal friction

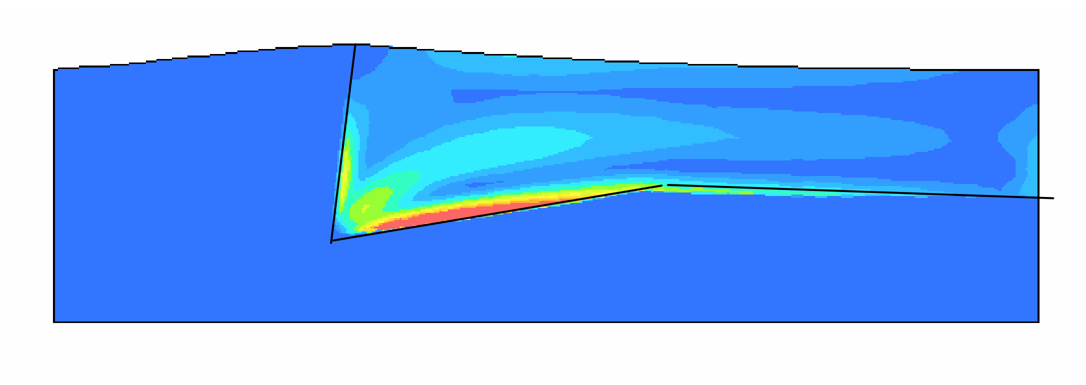
Die

- Lagrangian mesh
- Mechanical behaviour coupled with:
  - Thermal behaviour in workpiece
  - Fluid traction load from workpiece

Extrusion through a conical die







Governing Equations - Extrusion

- Coupled Thermo - mechanical problem
  - Heat transfer significant factor in deformation process
- CFD
  - Non-Newtonian viscosity model – Plastic Norton Hoff law
  - Heat Transfer - Friction between die and workpiece.
  - Free Surface - Van Leer method
- CSM
  - Static equilibrium equation – linear elastic solid.
- Coupling at the workpiece/die boundary:
  - Die subject to fluid traction boundary condition.
  - Workpiece subject to a die velocity boundary condition.
  - Dynamic meshes – GCL. Fluid velocity relative to mesh movement.

Governing Equations

- Free Surface
  - Scalar Equation Method ~ marker ϕ used to track free surface

$$\frac{\partial \phi}{\partial t} + \nabla(\boldsymbol{u} \cdot \phi) = 0$$

  - Advection Scheme - Van-Leer

$$\phi_{face} = \phi_u + \frac{1}{2}\frac{\Delta\phi}{\Delta n}\left(d_{ud} - |(\boldsymbol{u} \cdot \boldsymbol{n})_{face}|\,\Delta t\right)$$

    - $\Delta\phi/\Delta n$ dependant on value of ϕ for upwind-upwind element
  - Density Gradients – GALA algorithm
- Coupled thermo-mechanical problem
  - Heat transfer significant factor in deformation process.
  - Energy entered into thermal equation as:

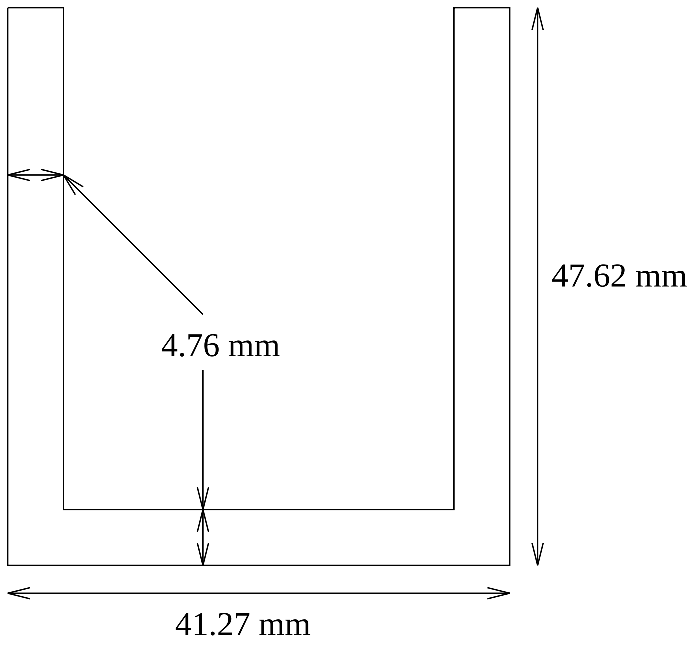
$$\frac{\partial}{\partial t}\left(\rho c_p T\right) + \nabla \cdot \left(\rho c_p \mathbf{u} T\right) = \nabla \cdot \left(k \nabla T\right) + \dot{r}$$

  - Temperature development dependant on energy dissipation at rate:

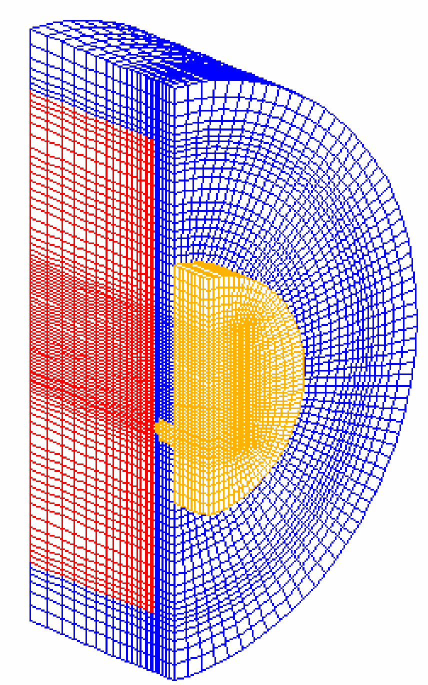
$$\dot{r}_{ij} = \beta \sigma_{ij} \dot{\varepsilon}_{ij}$$

β is proportion of plastic deformation energy dissipated as heat in solid material.

Extrusion through U-shaped die



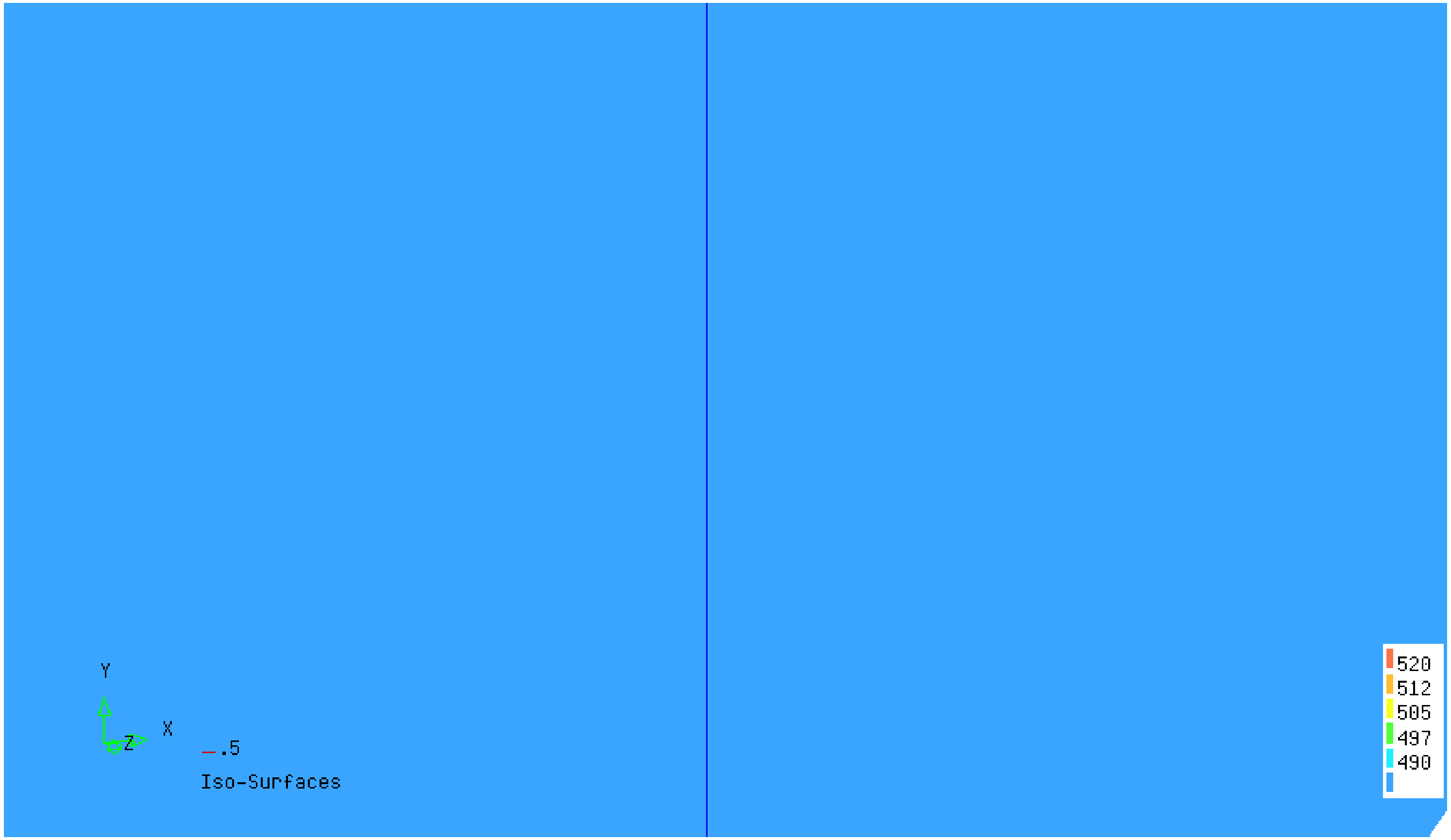
- **Initial diameter = 200mm**
- **Bearing length = 2.5mm**
- **Punch speed = 5.85E-3m/s**



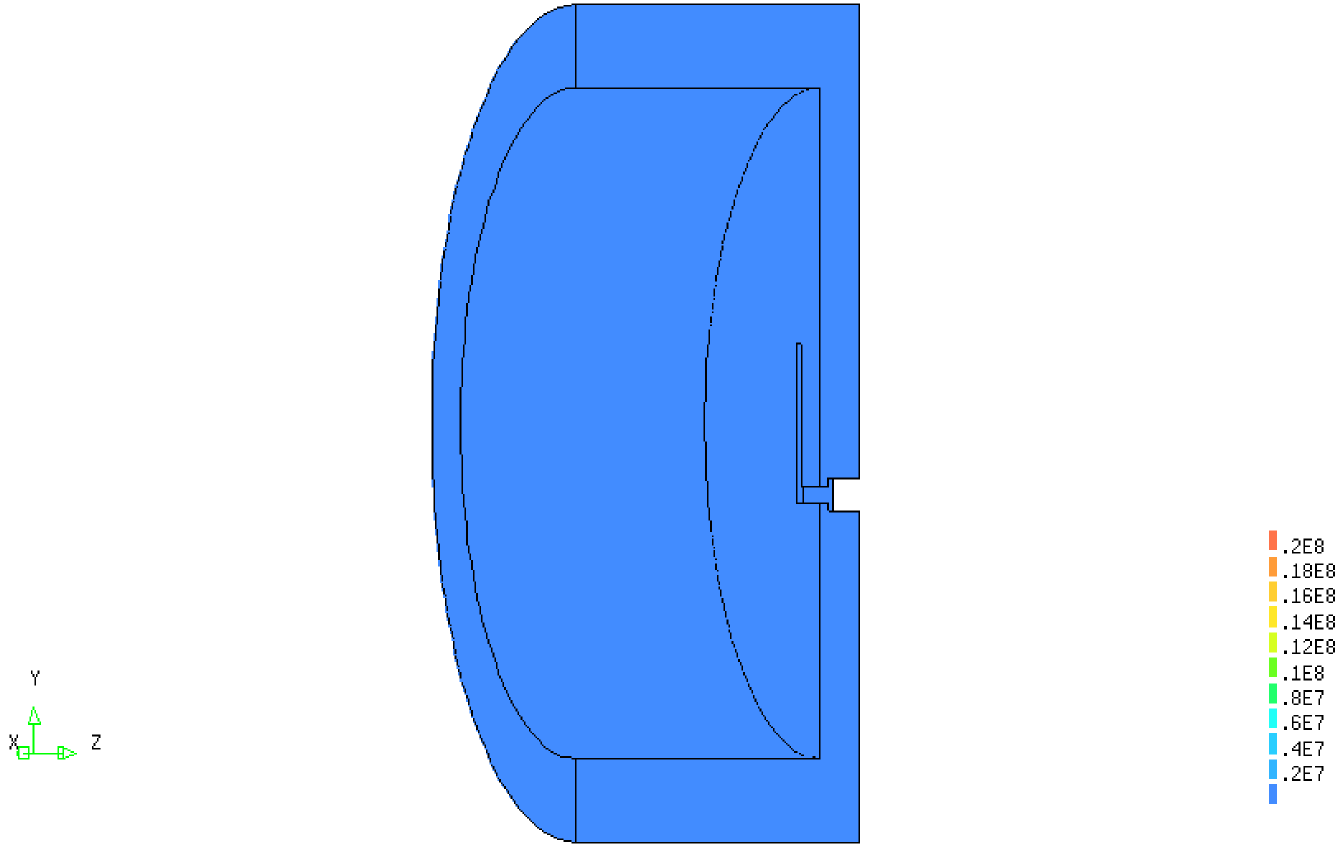
- 63220 elements
- 69507 nodes

- **Workpiece = 470°C**
- **Die = 450°C**
- **Air = 30°C**

Temperature contours in extruding work-piece

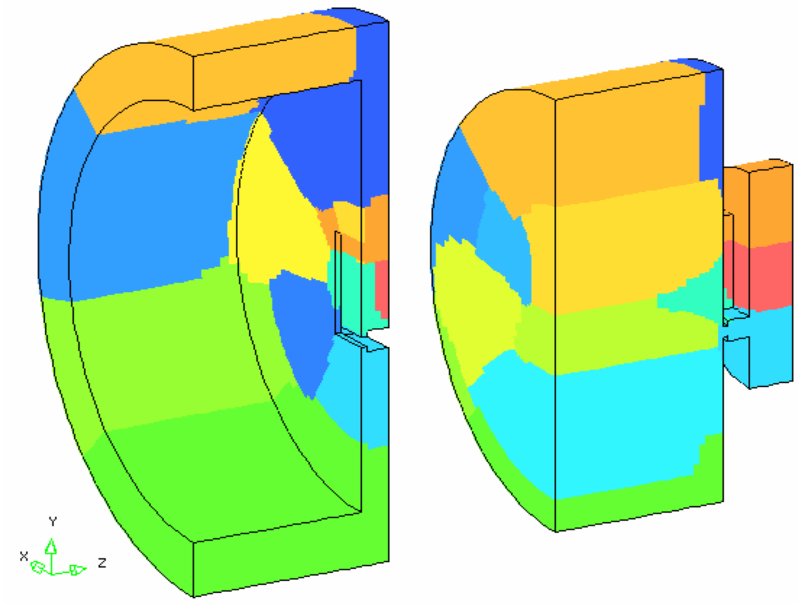


Effective stress contours and deformation of die



Parallel results

Processors	Run time (hours)	Speed-up
1	81.9	1
4	18.3	4.48
8	10.2	8.03
12	7.5	10.92
16	6.1	13.43



Single phase mesh partitions on 16 processors

- **Itanium IA 64 cluster running Linux OS**
- **Eight nodes, two 733MHz processors per node**
- **Each node with 2 Gb memory & 2Gb swap space**

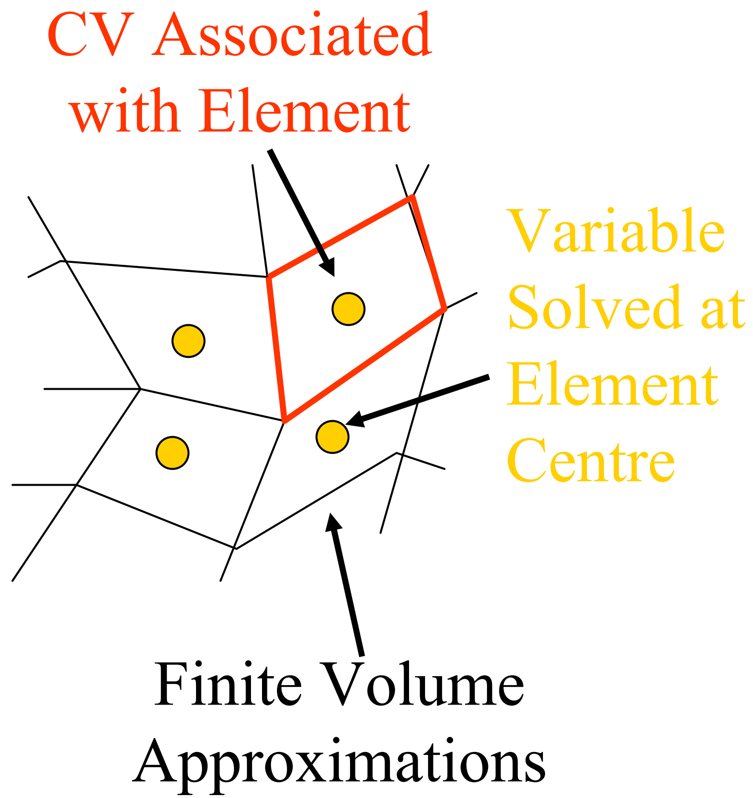
Finite Volume Methods for CFD: CC

Strengths and weaknesses:

- Cell centred methods:
  - memory efficient
  - fast

  BUT
  - accuracy fades rapidly as mesh quality degrades
  - **fails to converge with poor quality meshes**
  - correction terms help
    - slows convergence
    - stability

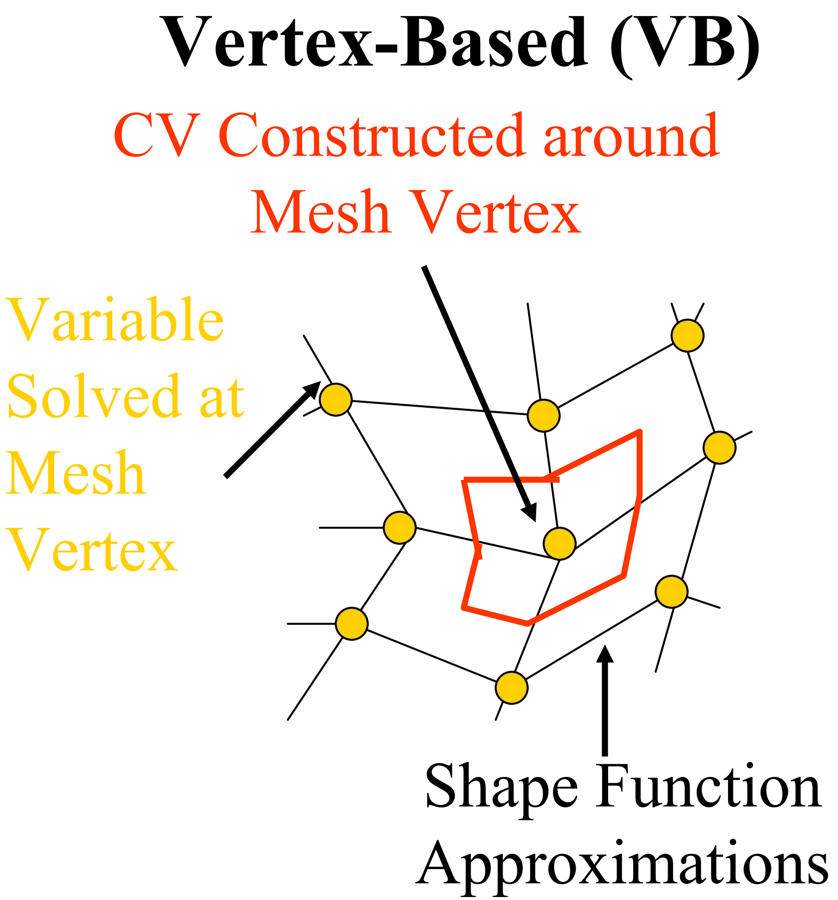
Cell-Centred (CC)



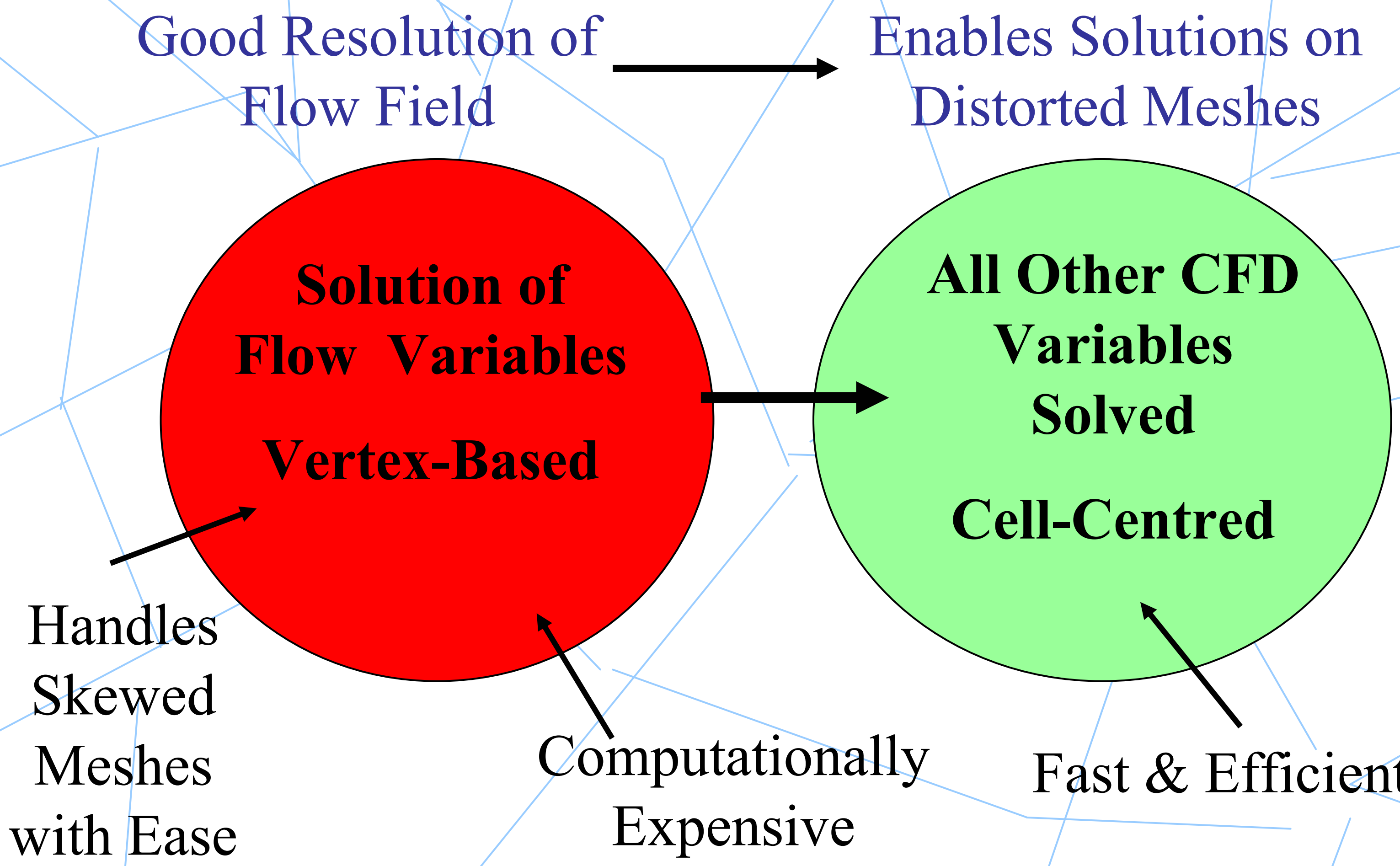
FV Methods for CFD: VB assessment

Strengths and weaknesses:

- Vertex centred methods:
  - heavy on memory
  - relatively compute intensive
  - good accuracy as mesh quality degrades
  - converges with almost any kind of mesh, no matter how poor its quality

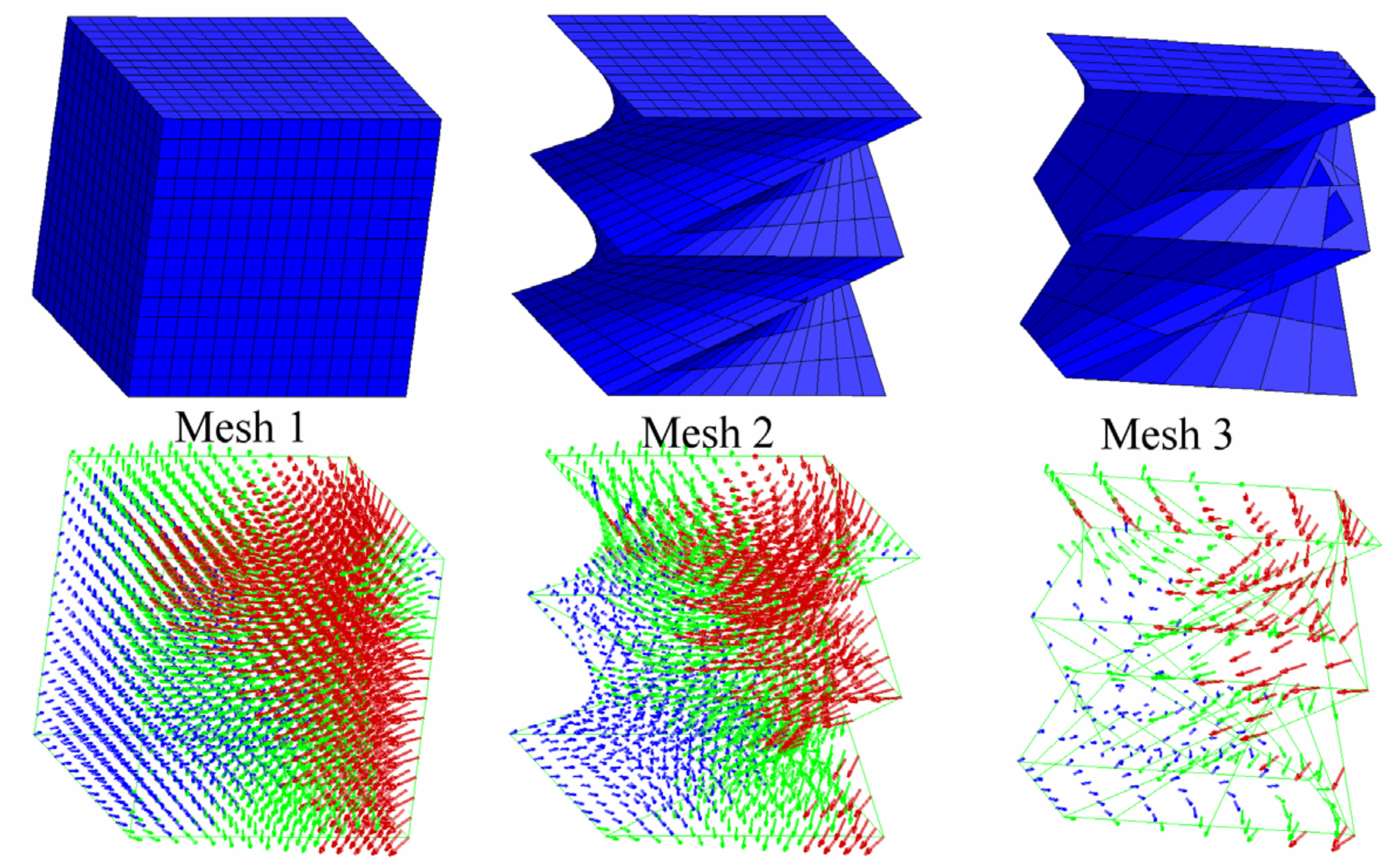


Concept Of Approach for VB CFD



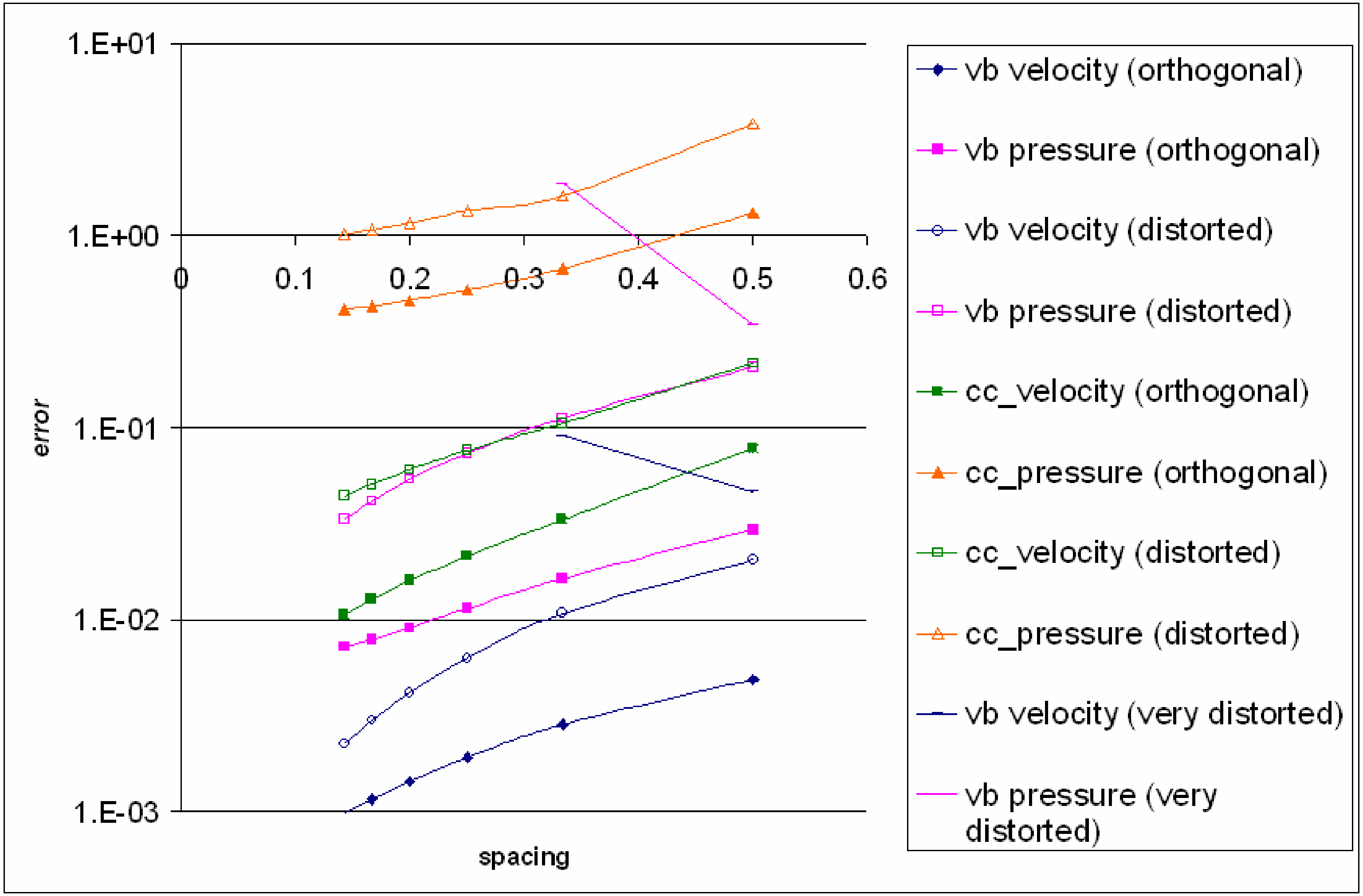
Motivation - physics rich CFD module using CC methods for all transport variables

Increasingly skewed meshes for VB



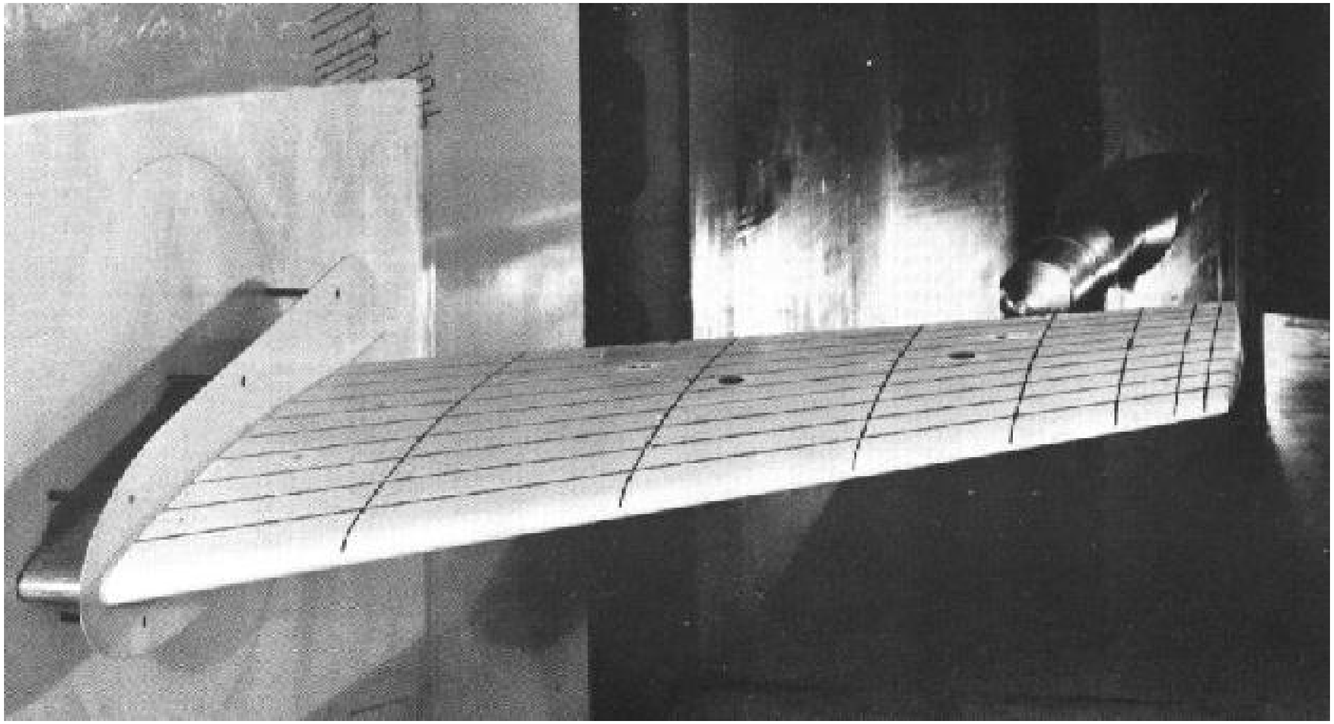
Beltrami problem – 3D benchmark with analytical solution

Measured numerical errors for VB, CC and combinations



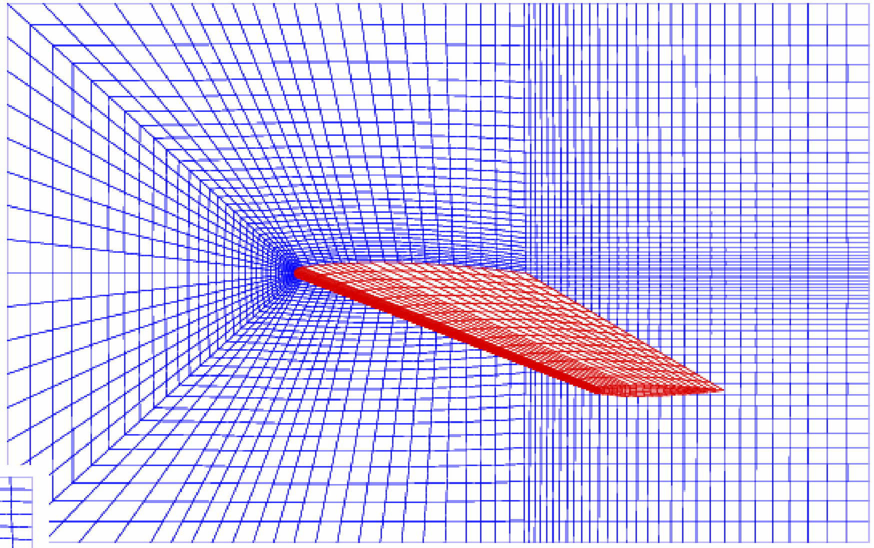
Flow over an ONERA M6 Wing

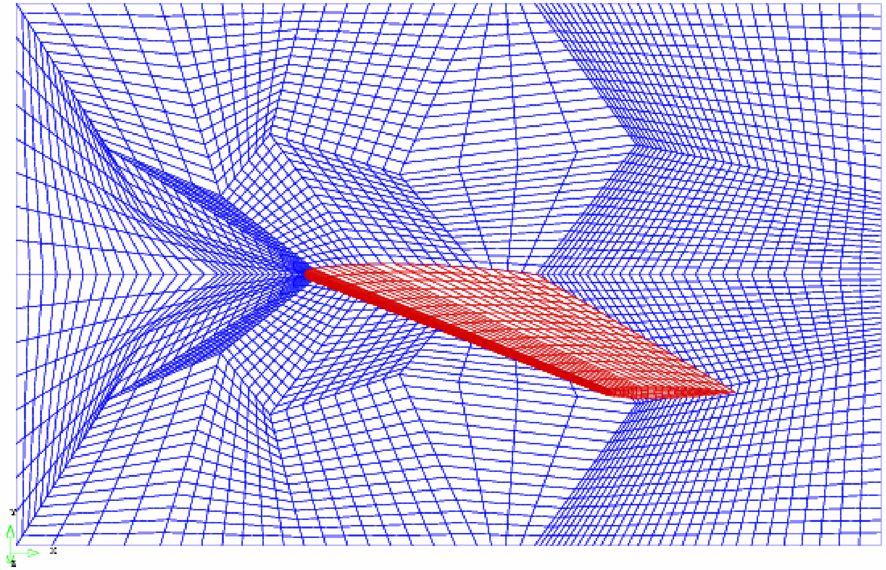
- Flow speed equivalent to Mach 0.3
- k-ε turbulence model



Meshes

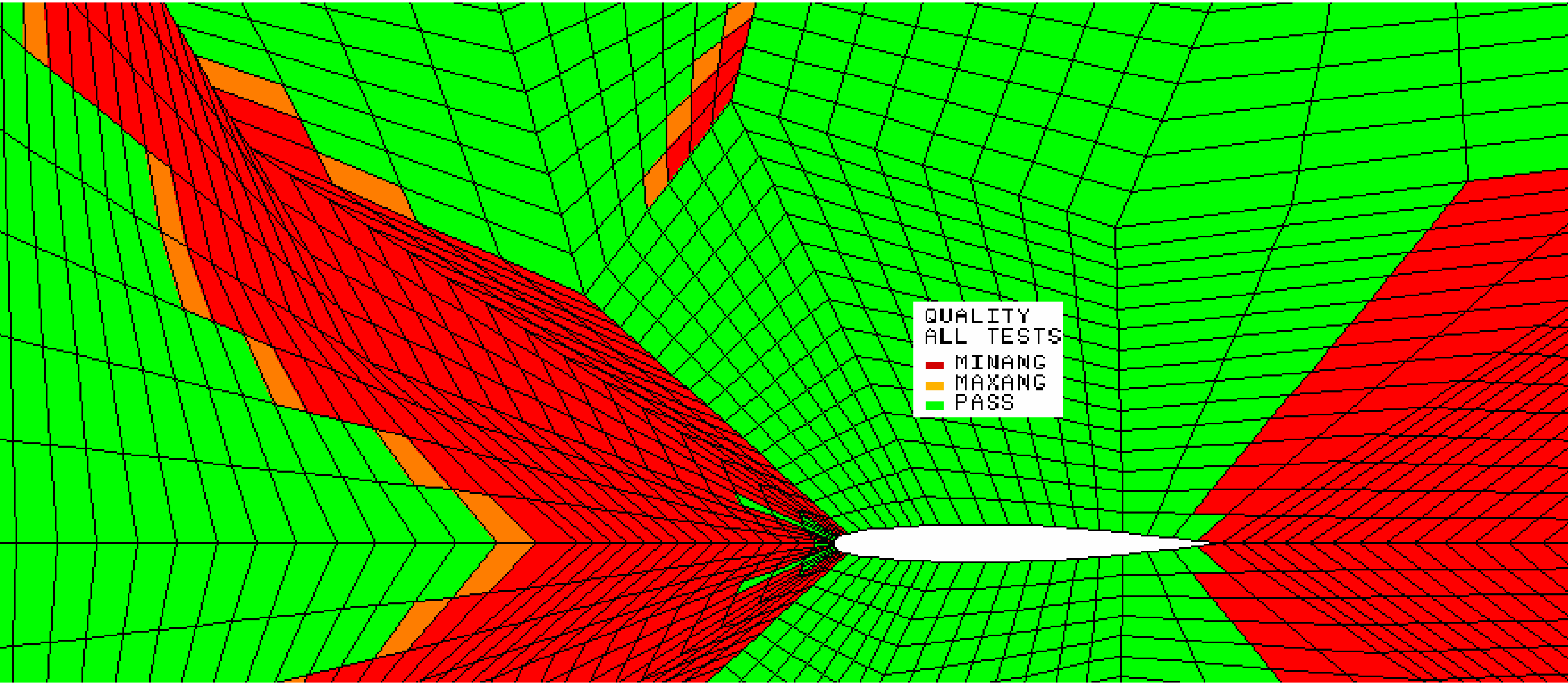
- C-mesh





- Distorted C – Mesh

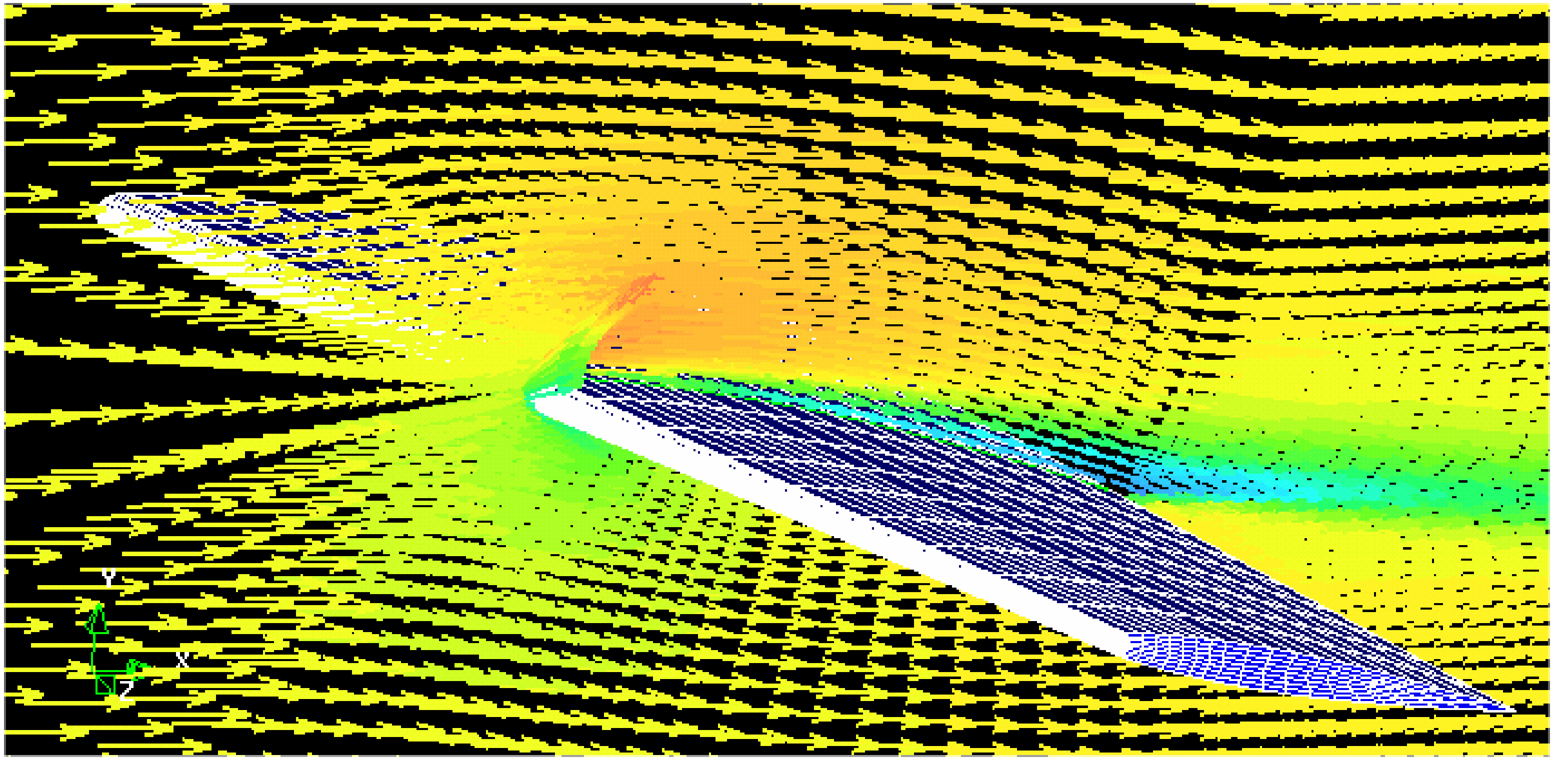
Mesh element quality



Angle: Below 30º : Above 150º

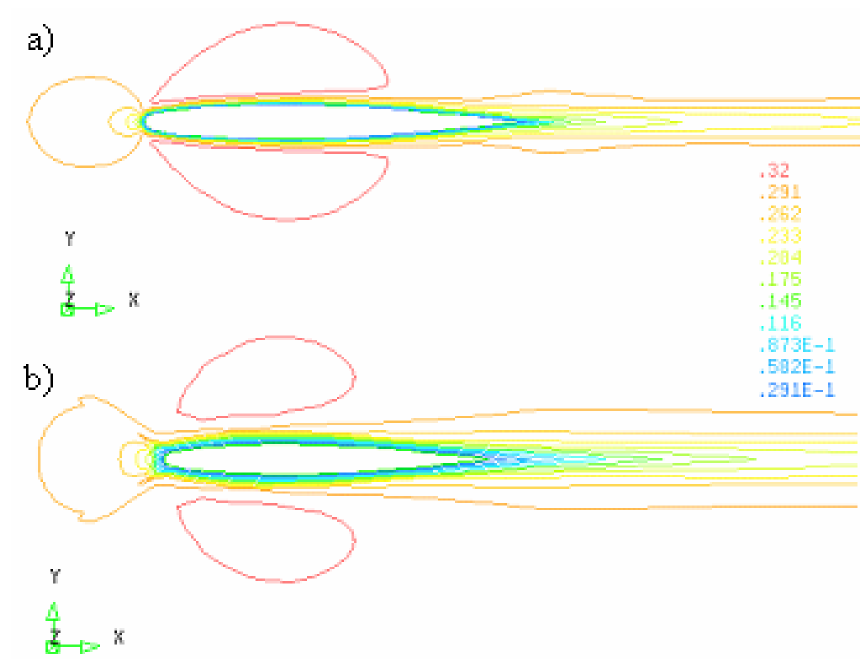
Model

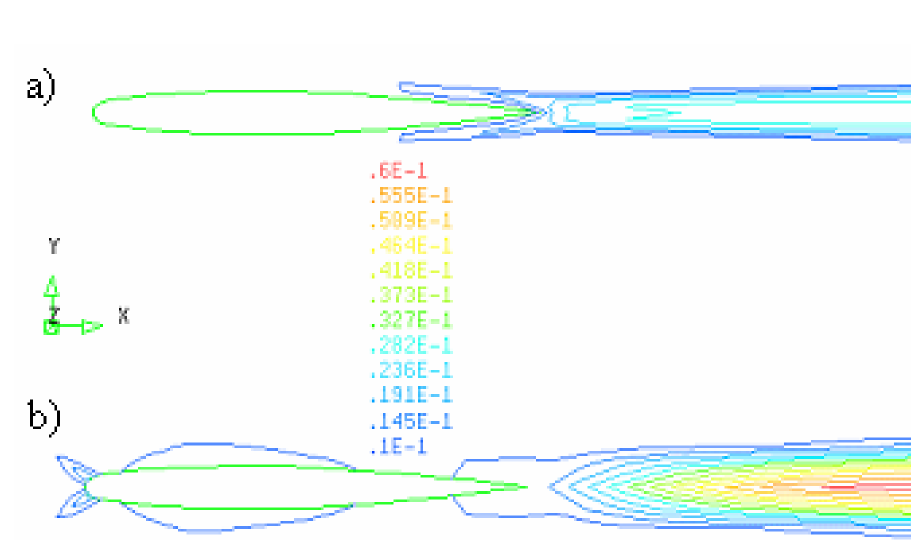
- 118,314 Vertices, 101412 Elements
- Flow Variables (u, p) solved Vertex Based
- Turbulence k-e solved cell centred



Results

- Mach Number
- Turbulent Viscosity





a) C-mesh results   b) distorted C-mesh results

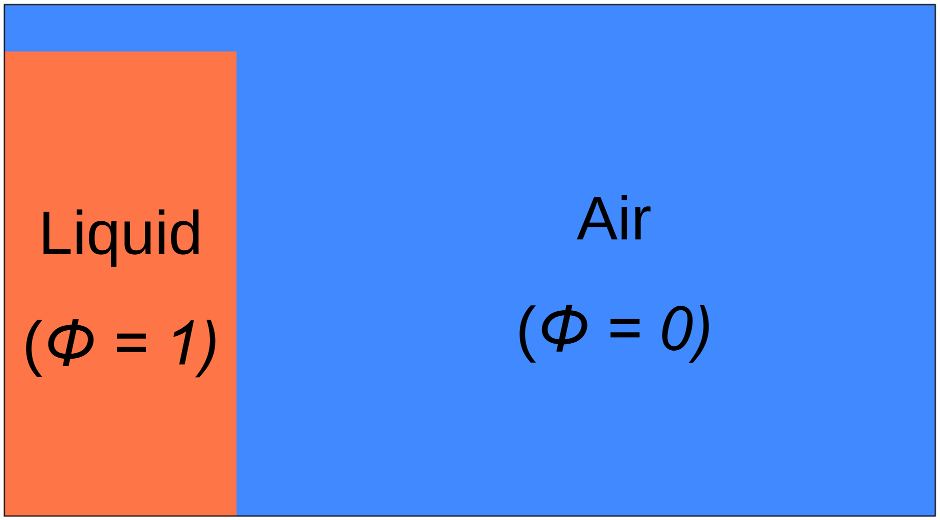
Computational Requirements

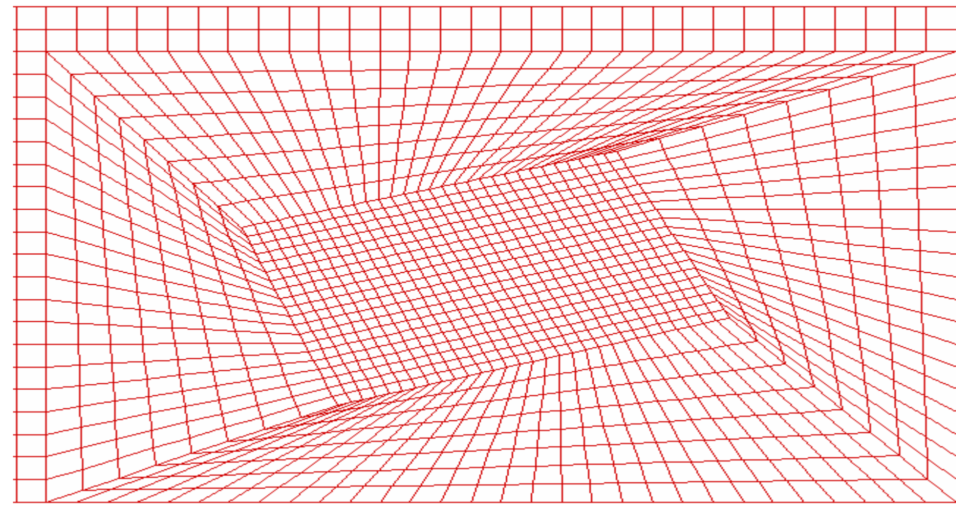
- Memory per Solution Point
  - Vertex-Based -> 373 Bytes
  - Cell-Centred -> 42 Bytes
- Seconds per Iteration / per Solution Point
  - Vertex-Based -> 3.3×10^{-5}
  - Cell-Centred -> 7.0×10^{-6}
- Number of Iterations
  - C-Mesh -> 254
  - Distorted Mesh -> 302

Example of VB-CC calculation: free surface capture

- SEA solves for the whole domain as a two component fluid and tracks the free surface development
- Uses D-A and van Leer schemes to sharpen surface capture
- Implemented using CC discretisation
- Procedure re-implemented using VB velocity components which are interpolated onto cell faces, and then hooked into conventional SEA

Simple test problem: collapsing liquid column



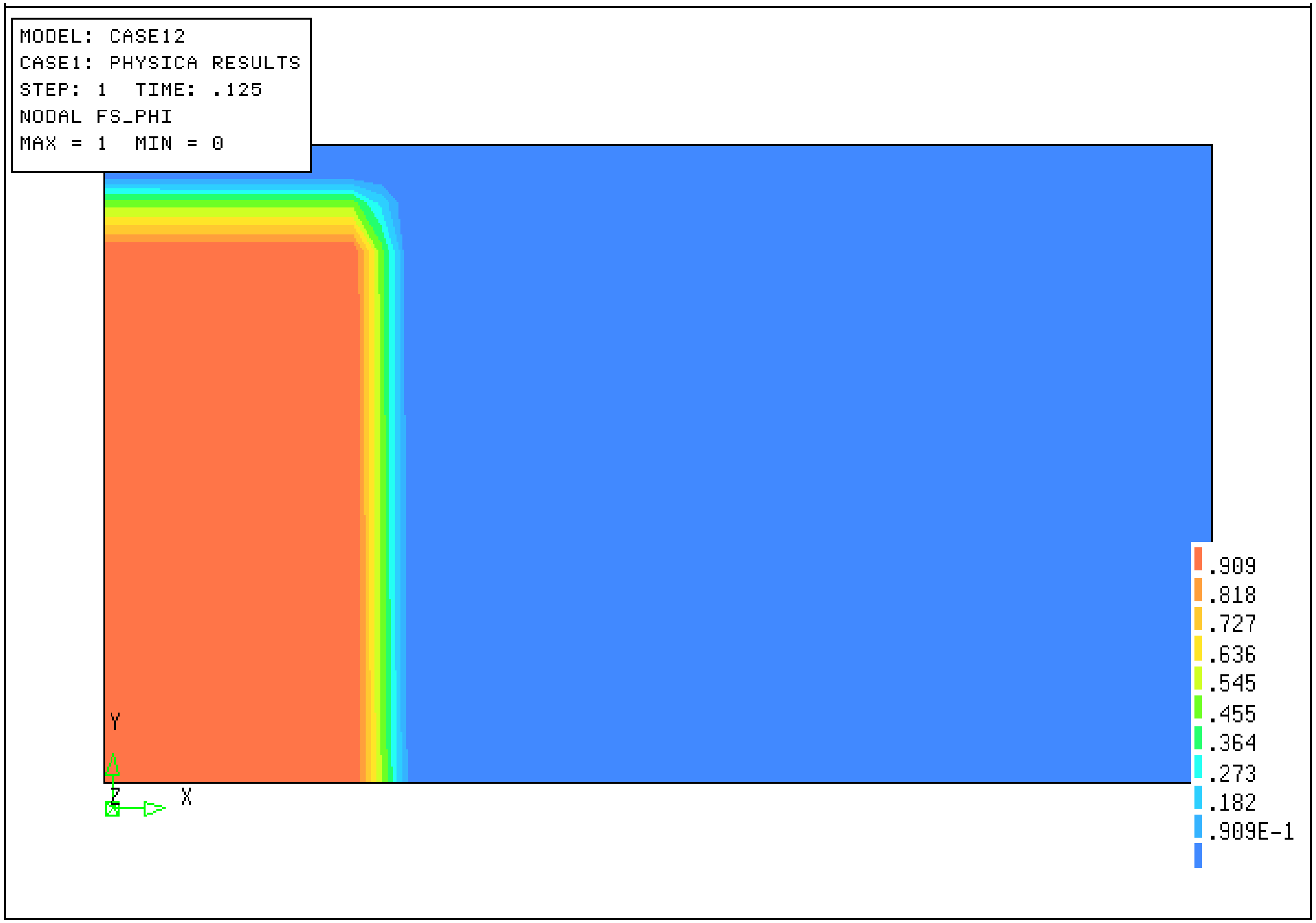


Key issue here is to test free surface procedure as the mesh quality is reduced

Mesh quality – non orthogonality ranges from 7 to 175 deg

CC has no chance of converging – question how does VB-CC method converge & how does accuracy degrade?

Simple test application: 2D collapsing column

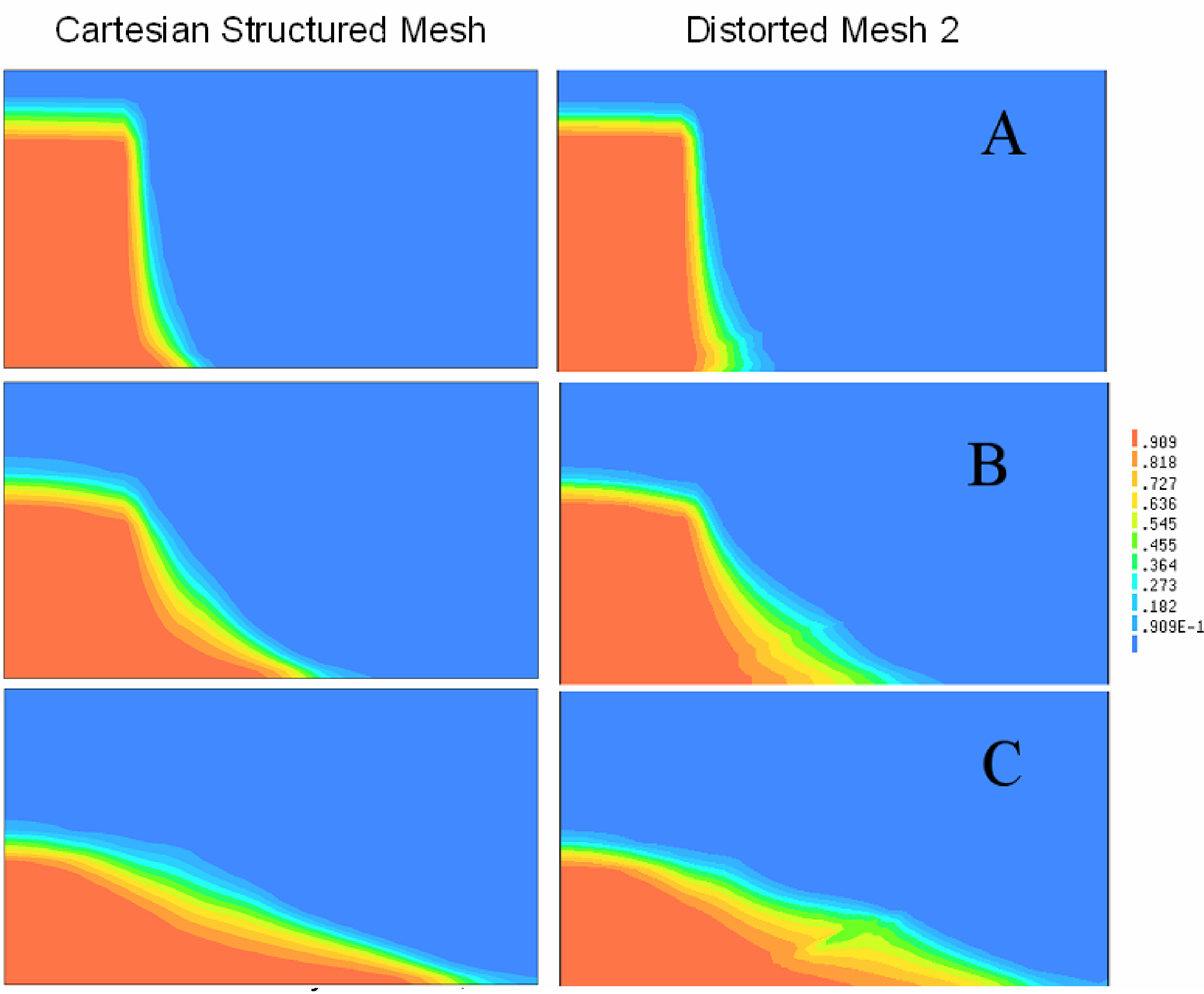


Comparison with cartesian mesh

Key issues:

• Convergence
-Good

• Accuracy degradation
- localised

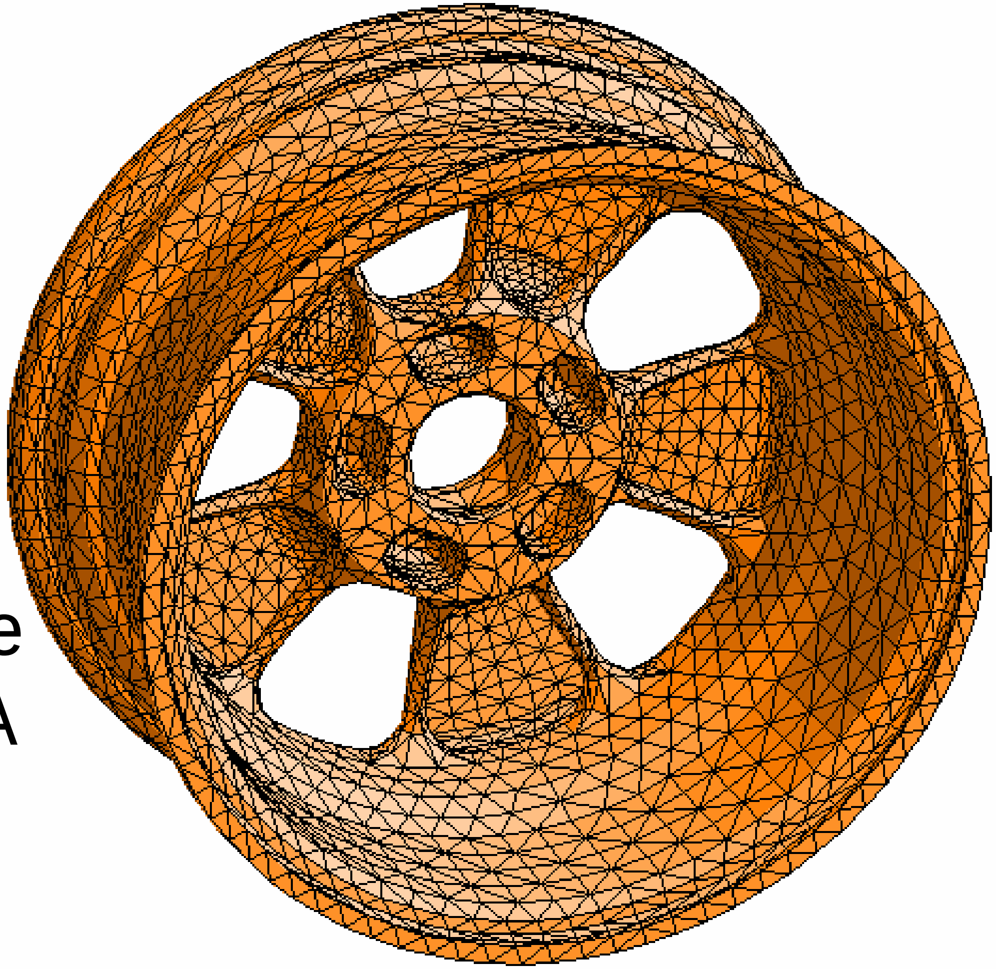


Some conclusions

- Orthogonal Mesh
  - CC fast & efficient
  - No advantage in VB method
- Distorted (Non-orthogonal) Mesh
  - CC fails OR contains significant errors
  - Coupled VB-CC Method
    - Good global resolution of flow field
    - Enables solution of other transported quantities CC
    - Easily coupled with other well-established CC algorithms such as Scalar Equation Algorithm for free surface flow.

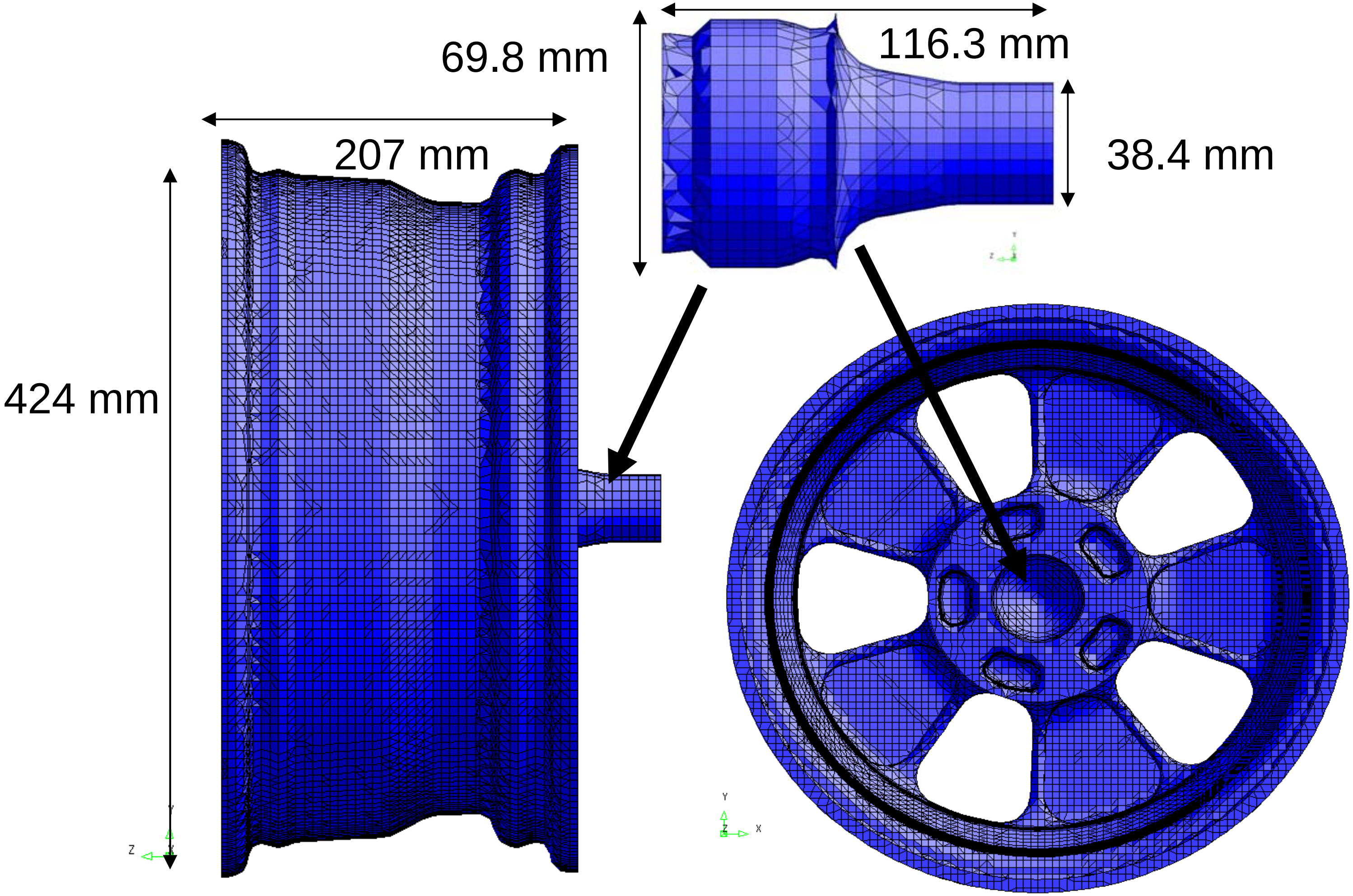
Challenging example

- Supplied by collaborator as an example of wheel
    - mixed elements
    - 91415 Elements
    - 55877 nodes

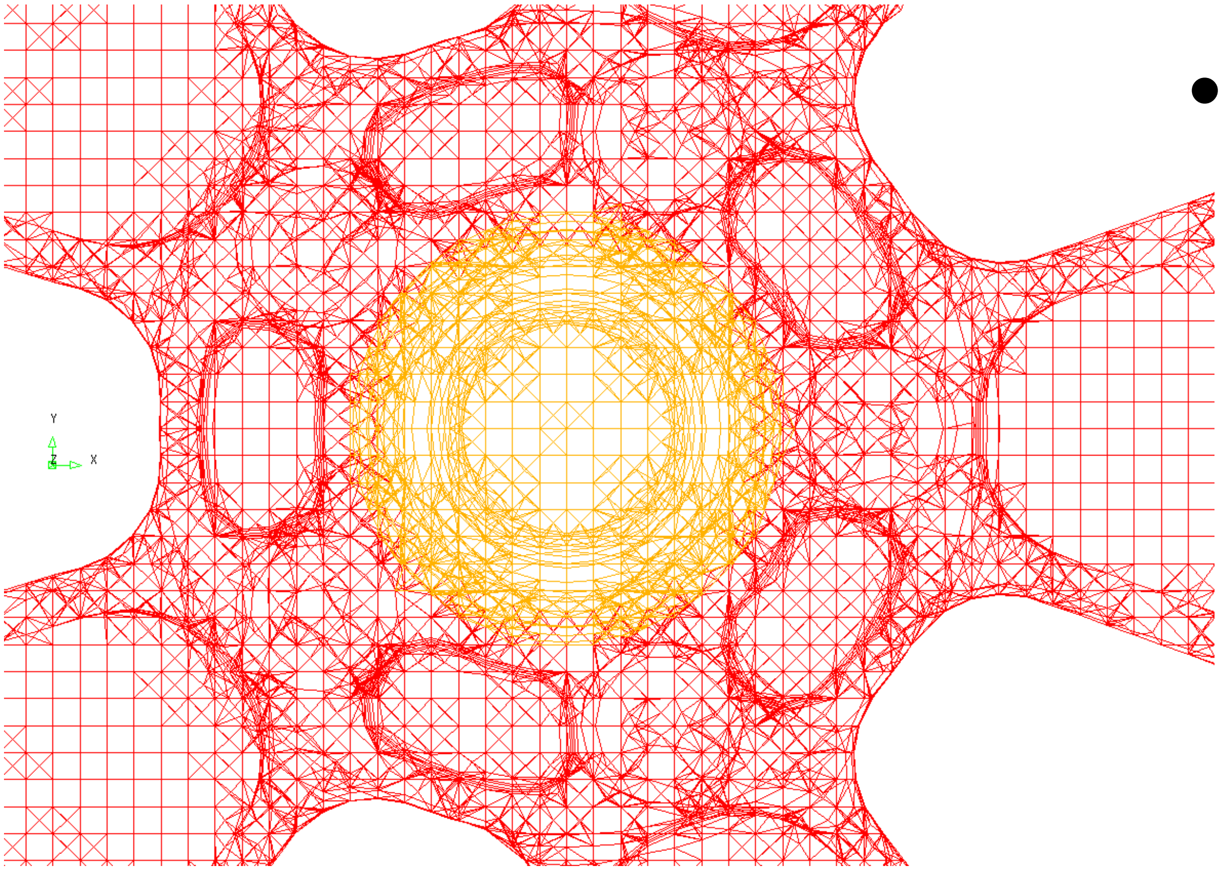


- No solution with the CC free surface SEA procedure

Application to a real case: the wheel



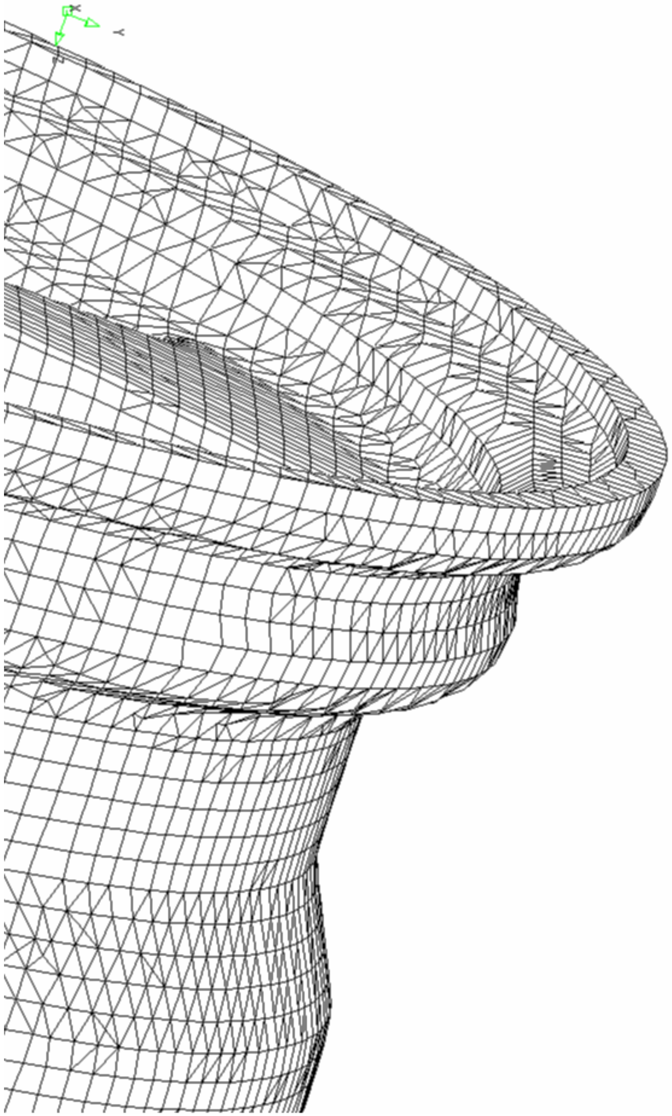
Mesh element types and complexity

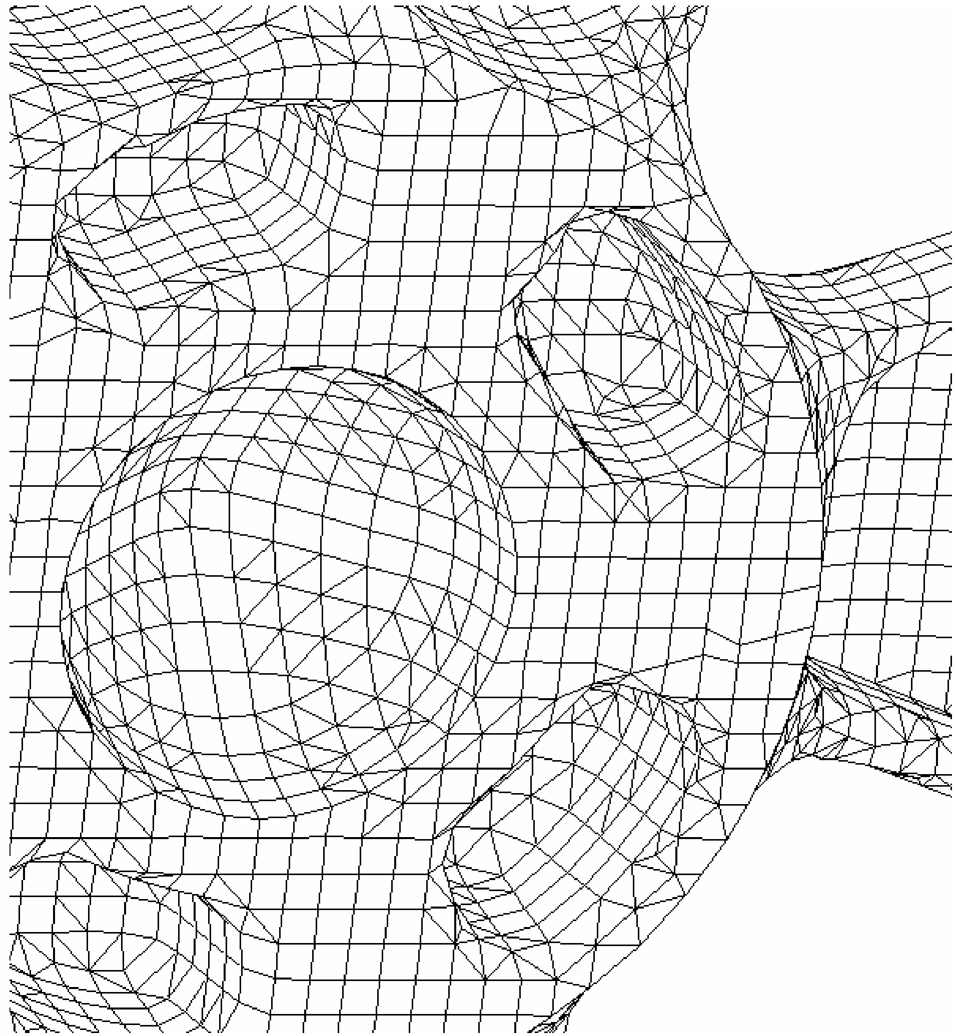


- 91415 Elements
- 55877 Vertices

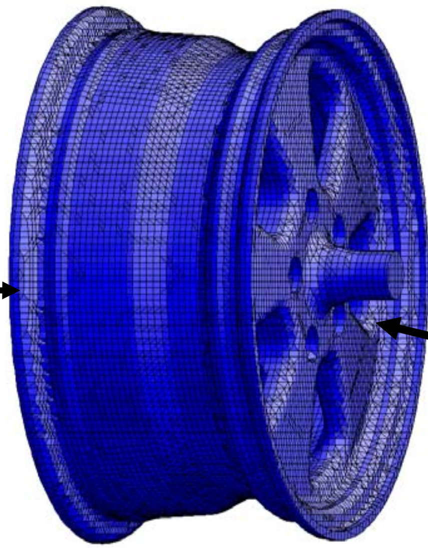
  - 36296 Pyramids
  - 24013 Tetrahedrals
  - 11390 Pentahedrals
  - 19716 Hexahedrals

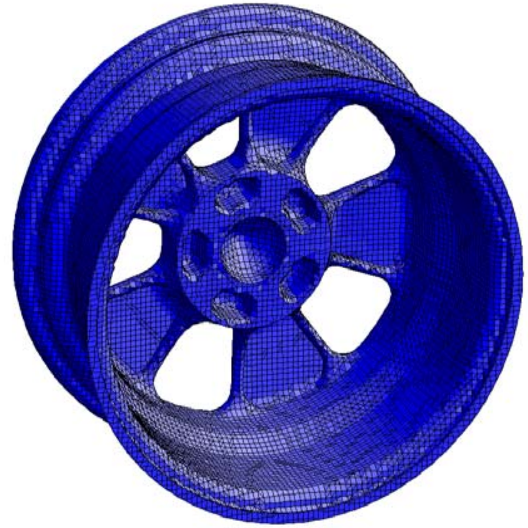
Problems with mesh quality

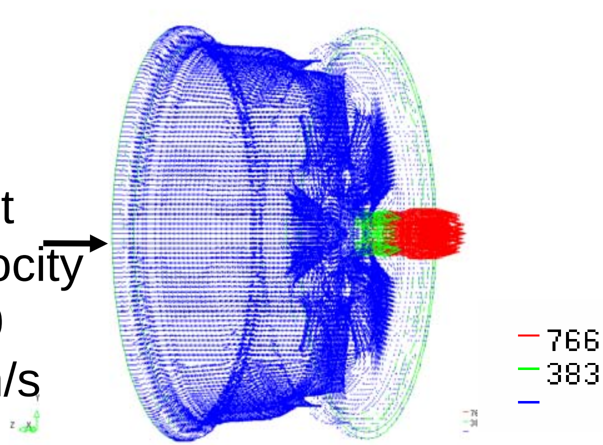


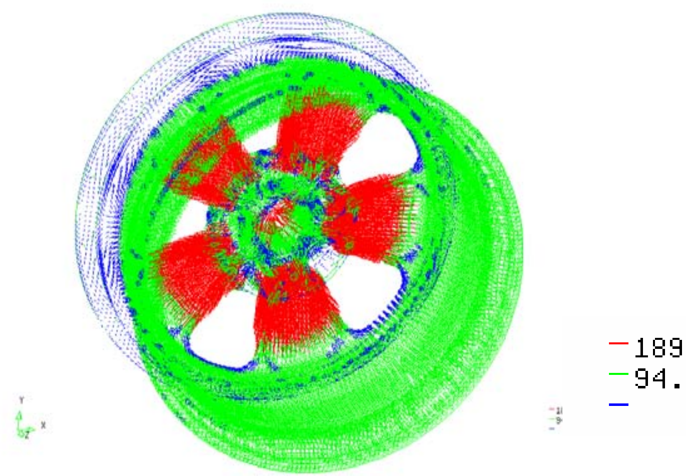


Boundary conditions & flows

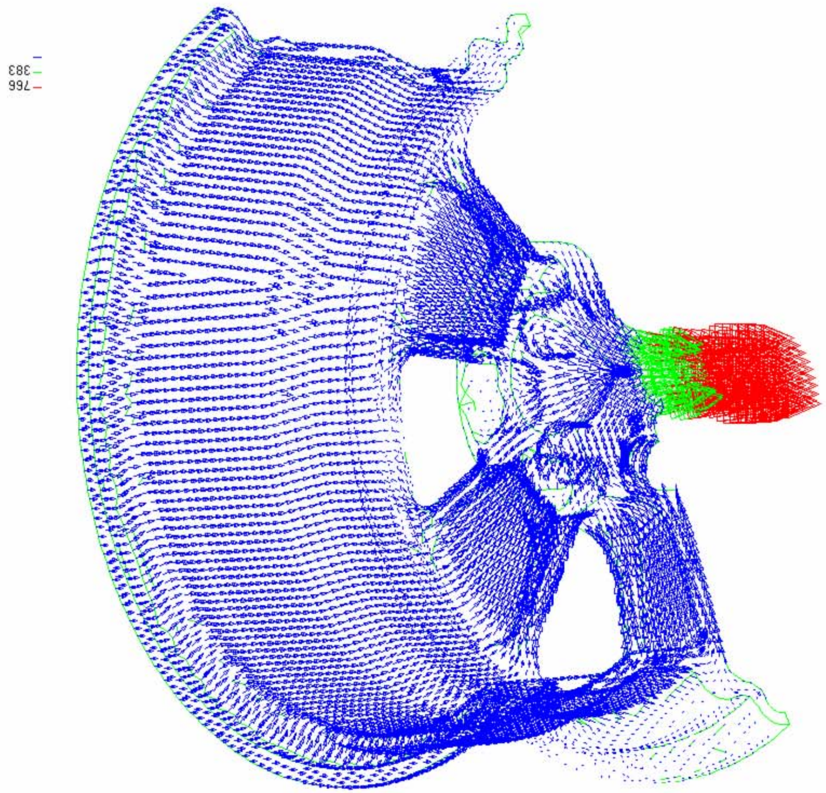


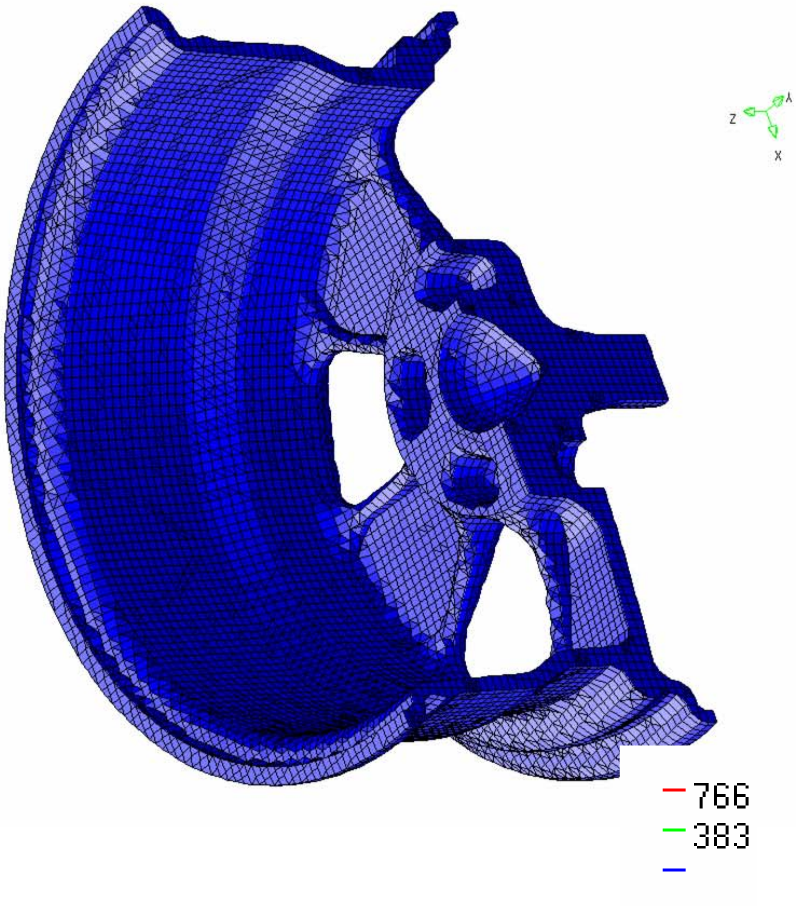




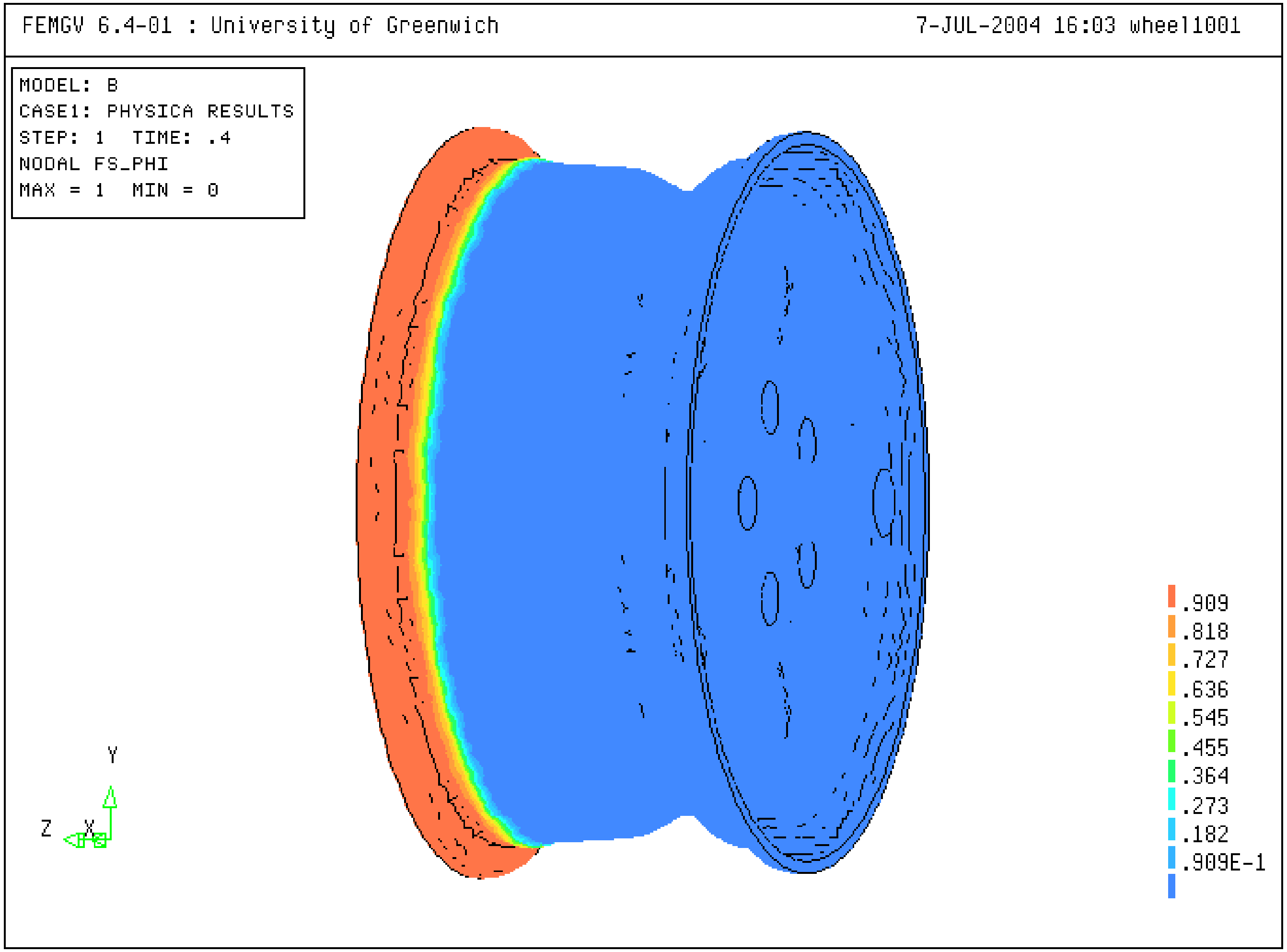


X-Section View – flow velocities





The filling process



Run time data

- Velocity is solved using a variant of SIMPLE with outer time step
- Free surface marker is tracked explicitly using smaller time steps
- Solved for 460 outer time steps to capture 8 seconds of real time
- Run on an Intel Pentium 4 2.53 Ghz processor with 83.62 MB memory
- Scalar run time - 12 Hours
- Now implemented in parallel.

Conclusions

- **Multi-physics simulation demanding of compatibility in specific phenomena solvers**

- Key features of multi-physics simulation:
  - CFD capability
  - Fluid –Structure Interaction (FSI)
  - Parallel framework

- Our initial work very conservative in its initial – FV methods on unstructured meshes for all phenomena
  - FV-CC for flow has limitations on mesh quality – use VB-CC hybrid methods
  - FV for stress means reinvention of all FE stress solvers – why?

- **Key challenges**
  - coupling complex flow physics into multi-physics solvers
  - coping with extreme deformation with DFSI (e.g. parachute opening)
  - coupling distinct physics (e.g. DEM with CFD)