Digital Twin for Continuous Casting Modelling

Pavel E. Ramirez Lopez
SWERIM AB
pavel.ramirez.lopez@swerim.se

Research Fund for Coal & Steel

A bit of history...

- Model starts in 2003-2004 (MSc) focused on flow dynamics
- Added heat transfer and solidification in 2005-2006

- Added mould oscillation and slag infiltration (PhD) 2007-2008
- Slag solidification and rim formation (2009)
- Discovered evidence of Oscillation Marks (2010)

- Adding Argon Injection as Discrete Phase (2012)
- Adding Electromagnetics (2016-2019)
- Adding Curved casters (2018-2019)
- Adding taper effects & friction (2019-2020)

Q1) In general, how has been your experience with modelling?

a) Excellent: we use it everyday at our company/institution to create industrial value

b) Good: We used it often and we have obtained some good results but something is missing

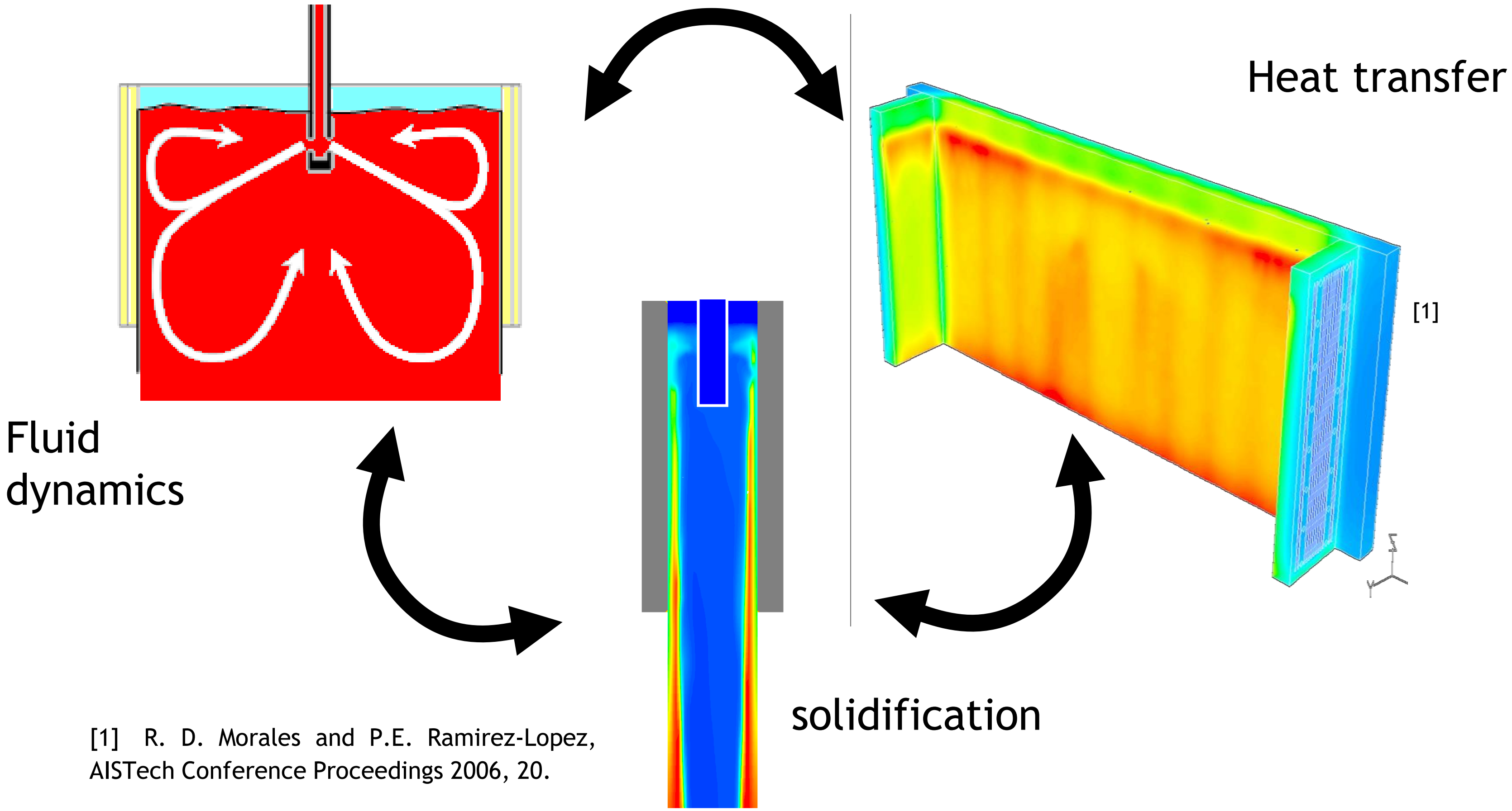
c) Regular: We have used it, but results have been mixed and its usability is limited

d) Poor: Bad experiences with modelling with no industrial value

Modelling approach

Coupled phenomena in CC

CONTINUOUS CASTING → MULTIPLE PHENOMENA



[1] R. D. Morales and P.E. Ramirez-Lopez, AISTech Conference Proceedings 2006, 20.

Model theory*

* based on ANSYS-FLUENT ©

Flow dynamics through Navier-Stokes eqs.+ turbulence effects (RANS or LES models):

$$\frac{\partial \rho}{dt} + \nabla(\rho \vec{v}) = S_m$$

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \mu \nabla^2 \vec{v} + \rho g + F$$

$+$

$$\nabla(\rho k \vec{v}) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla \varepsilon\right] + G_k - Y_k$$

$$\nabla(\rho \varepsilon \vec{v}) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right] + G_\varepsilon - Y_\varepsilon$$

- Multiphase (steel-slag) through Volume of Fluid model and Surface tension effects:

$$\rho_{mix} = \alpha_p \rho_p + (1-\alpha_q)\rho_q$$

$$\mu_{mix} = \alpha_p \mu_p + (1-\alpha_q)\mu_q$$

$$\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}) = S_{\alpha_q} + \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp})$$

$$\frac{\partial}{\partial t}(\rho_{mix}\vec{v}) + \nabla \cdot (\rho_{mix}\vec{v}\vec{v}) = -\nabla p + \nabla[\mu_{mix}(\nabla \vec{v} + \nabla \vec{u})] + \rho_{mix}\beta \Delta T g - S_s + S_\sigma$$

$$S_\sigma = \sigma_{pq} \frac{\rho_{mix}\kappa_q \nabla \alpha_q}{\frac{1}{2}(\rho_p + \rho_q)}$$

- Heat transfer through the slag and mould + shell Solidification with the Lever Rule:

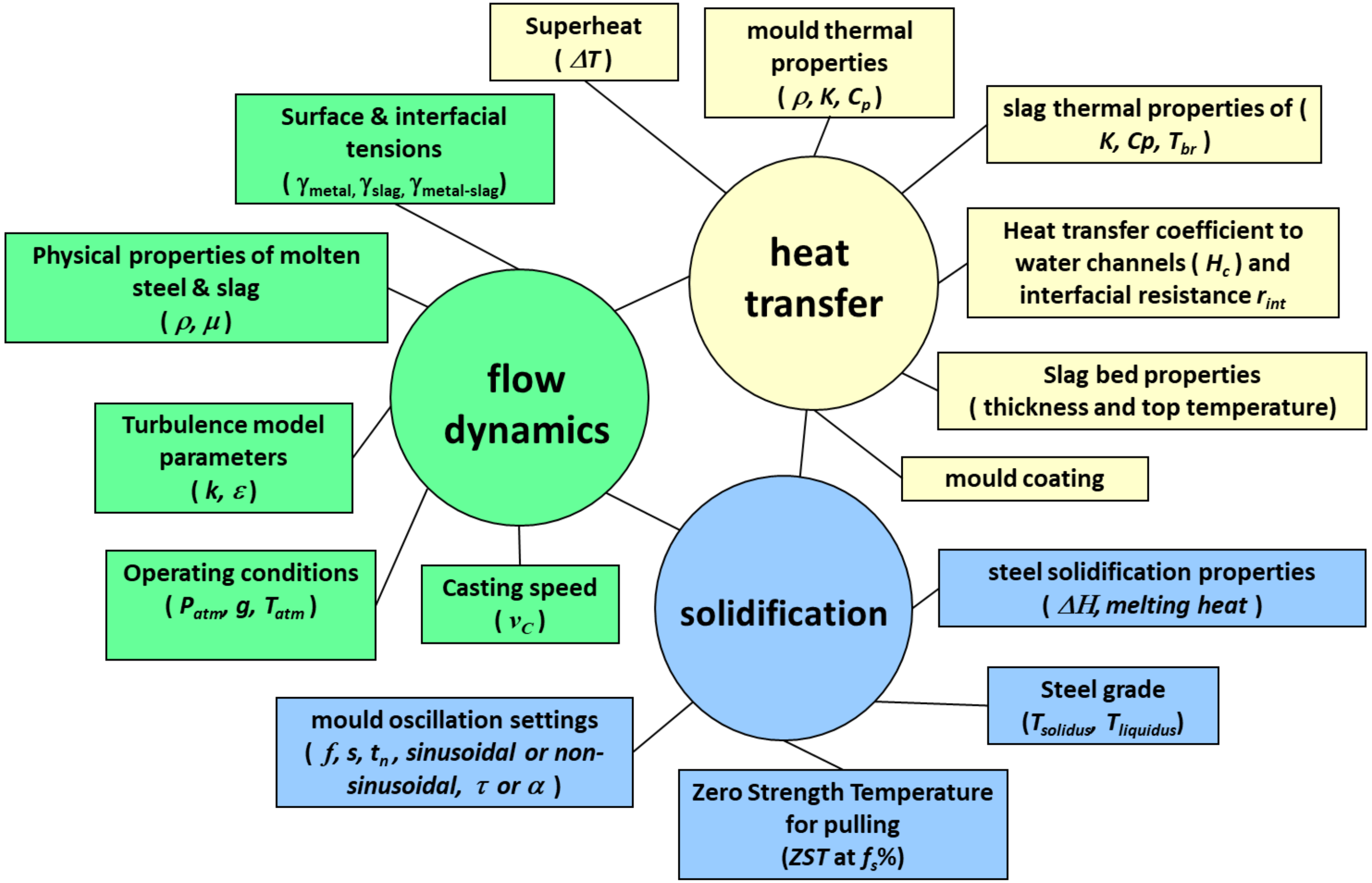
$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\vec{v} \rho h) = \nabla \cdot (K_m \nabla T)$$

$$f_l = \frac{T_q - T_{q,solidus}}{T_{q,liquidus} - T_{q,solidus}}$$

$$S_M = \frac{(1-f_l)^2}{(0.001 + f_l^2)} A_{mush}(\vec{v} - v_c)$$

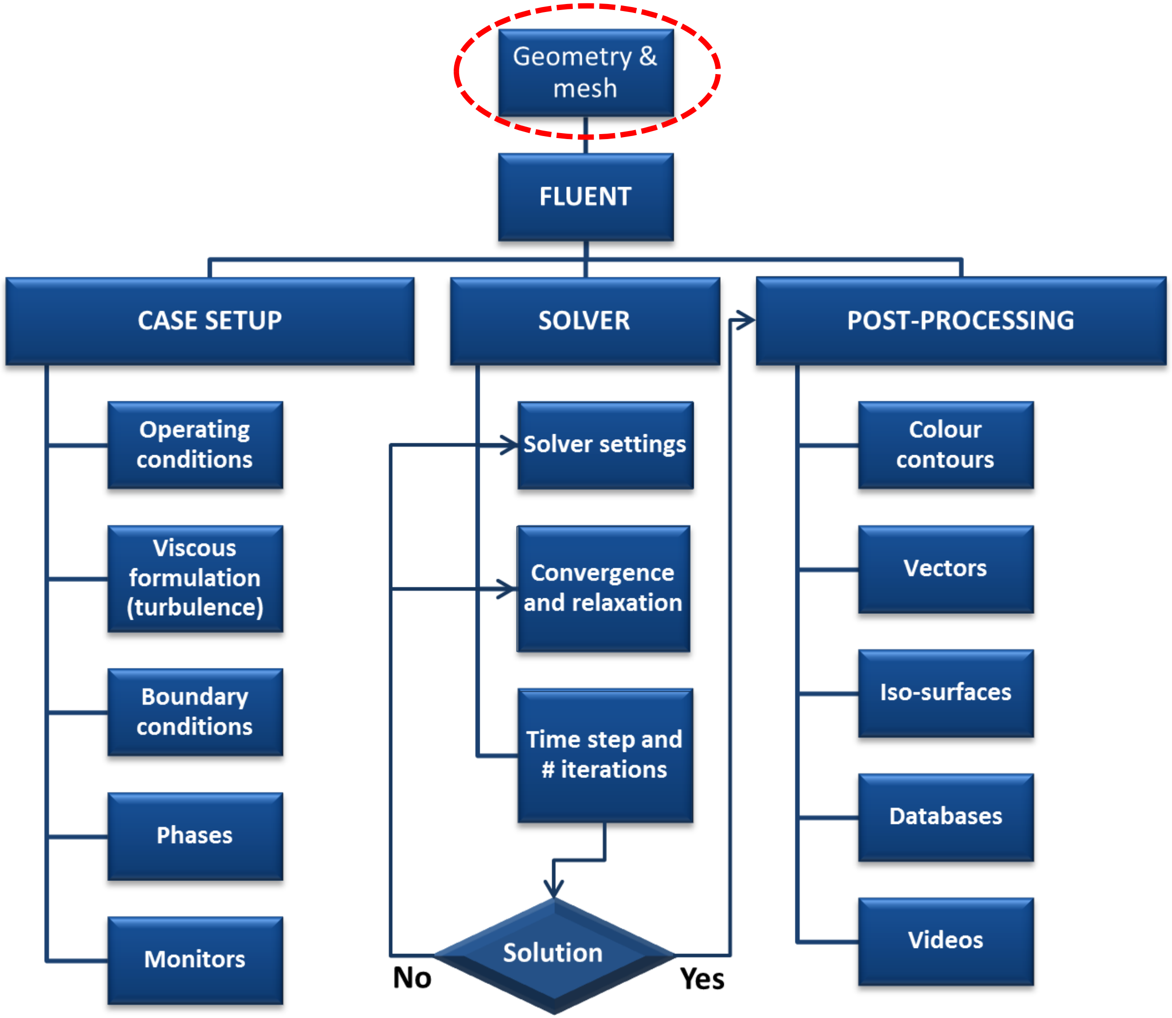
- Specific operation practices (e.g. oscillation, mould coating, etc) through User Defined Functions (UDF's)

Coupled phenomena



Model setup

Geometry and Mesh

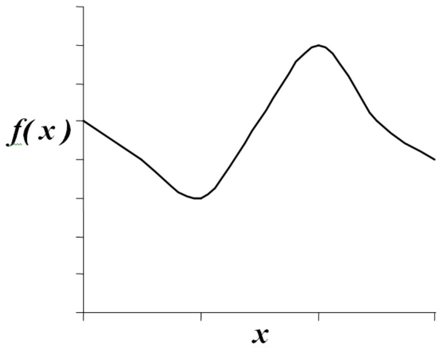


Discretization

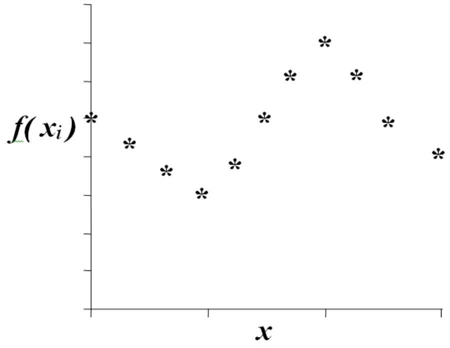
- We must assume N-S equations are partial differential equations with “possible analytical solutions” for all the flow field.*

- Theoretical solutions (analytical) are continuous throughout the whole function domain. See $f(x)$

- But this solution is not feasible for the complex N-S equations, which have to be rather solved for an specific (discrete) number of points (i.e. discretization)



Analytical solution for $f(x)$

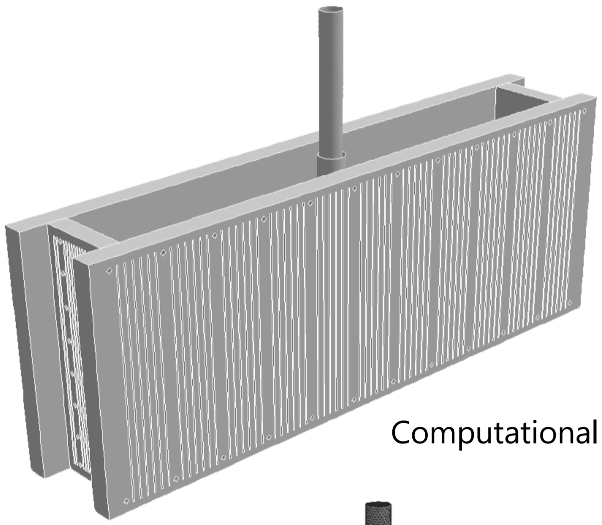


Discrete solution for $f(x)$

** See Navier–Stokes existence and smoothness problem & Millenium Prize ($1,000,000).*

Geometry and mesh

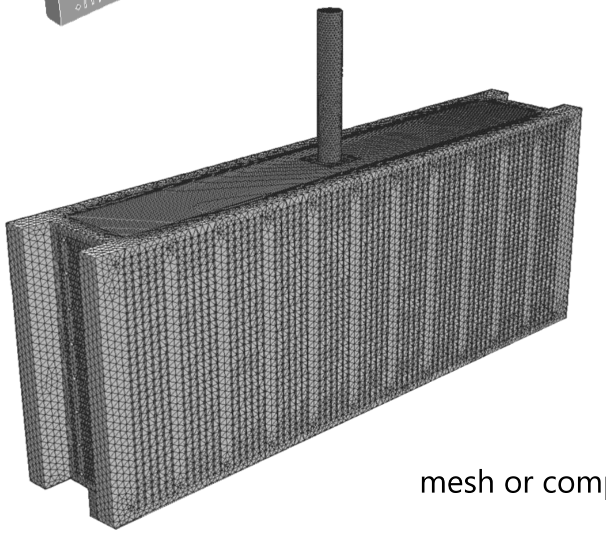
In contrast to analytical solutions, numerical methods (e.g. models) only provide results on specific points of the flow field, so the constitutive equations (e.g. Navier-Stokes Eqs.) must be generated for such limited number of points.



Computational domain (a.k.a. geometry)

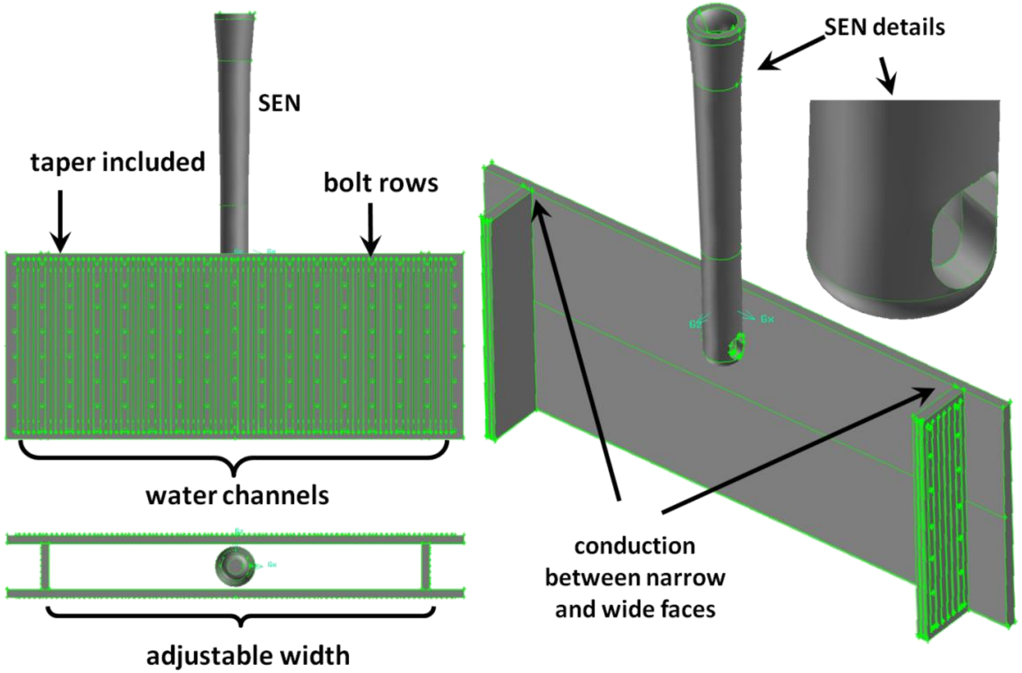
This process begins by establishing a geometry (obviously an area in 2D or a volume for 3D calculations) known as ***computational domain***.

The process of subdividing the domain in nodes, elements or volumes (e.g. FD, FEM or FV) to allocate the constitutive equations is known as ***mesh or grid generation.***

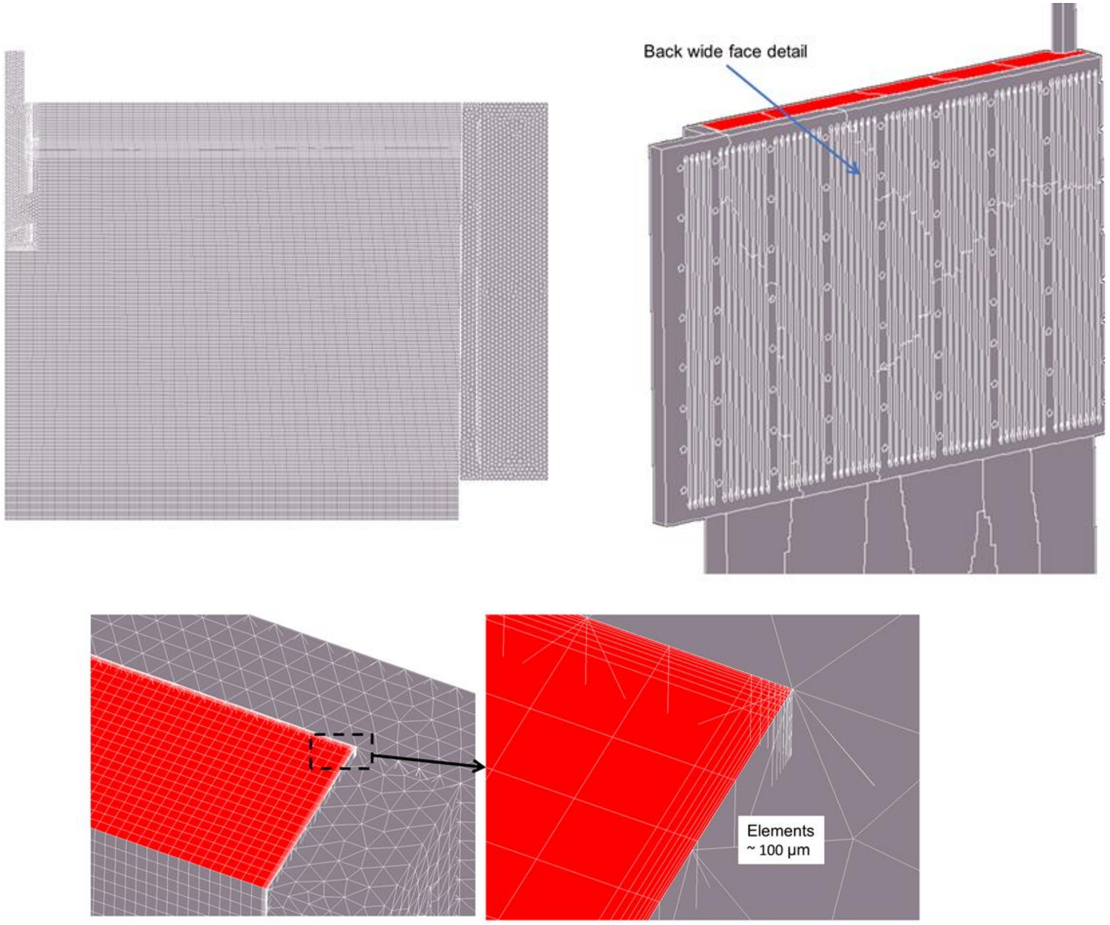


mesh or computational grid

Geometry & mesh for CC models

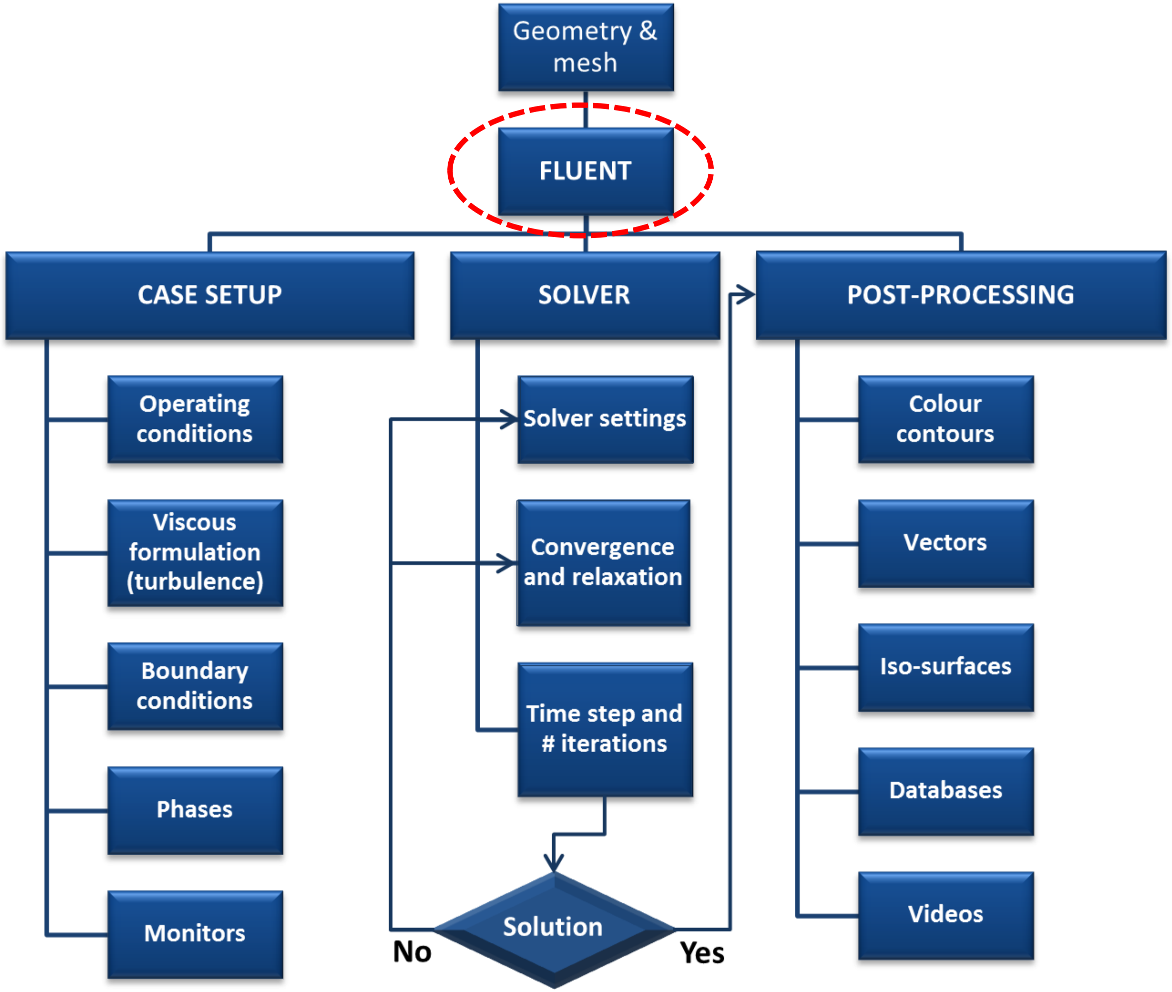


Geometry



Mesh or computational grid

Discretization Methods



Discretization methods

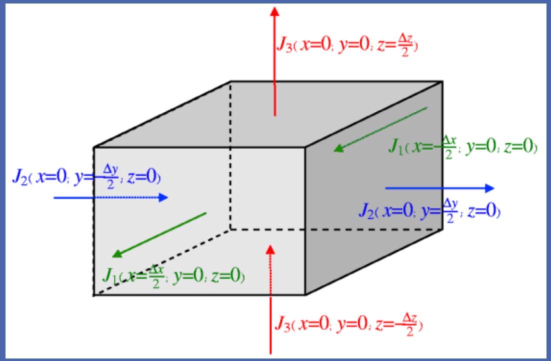
Discretization methods are grouped into:

- **Finite differences (FD)**
- **Finite volume (FV or FVM)**
- **Finite Element (FE or FEM)**

Finite Volume is the most common method for Computational Fluid Dynamics (CFD) applications. In FV, volume integrals in a partial differential equation are converted to surface integrals, using the divergence theorem. These terms are then evaluated as fluxes at the surfaces of each finite volume or cell (e.g. in a mesh).

http://wwwf.imperial.ac.uk/ssherw/spectralhp/papers/HandBook.pdf

Divergence theorem

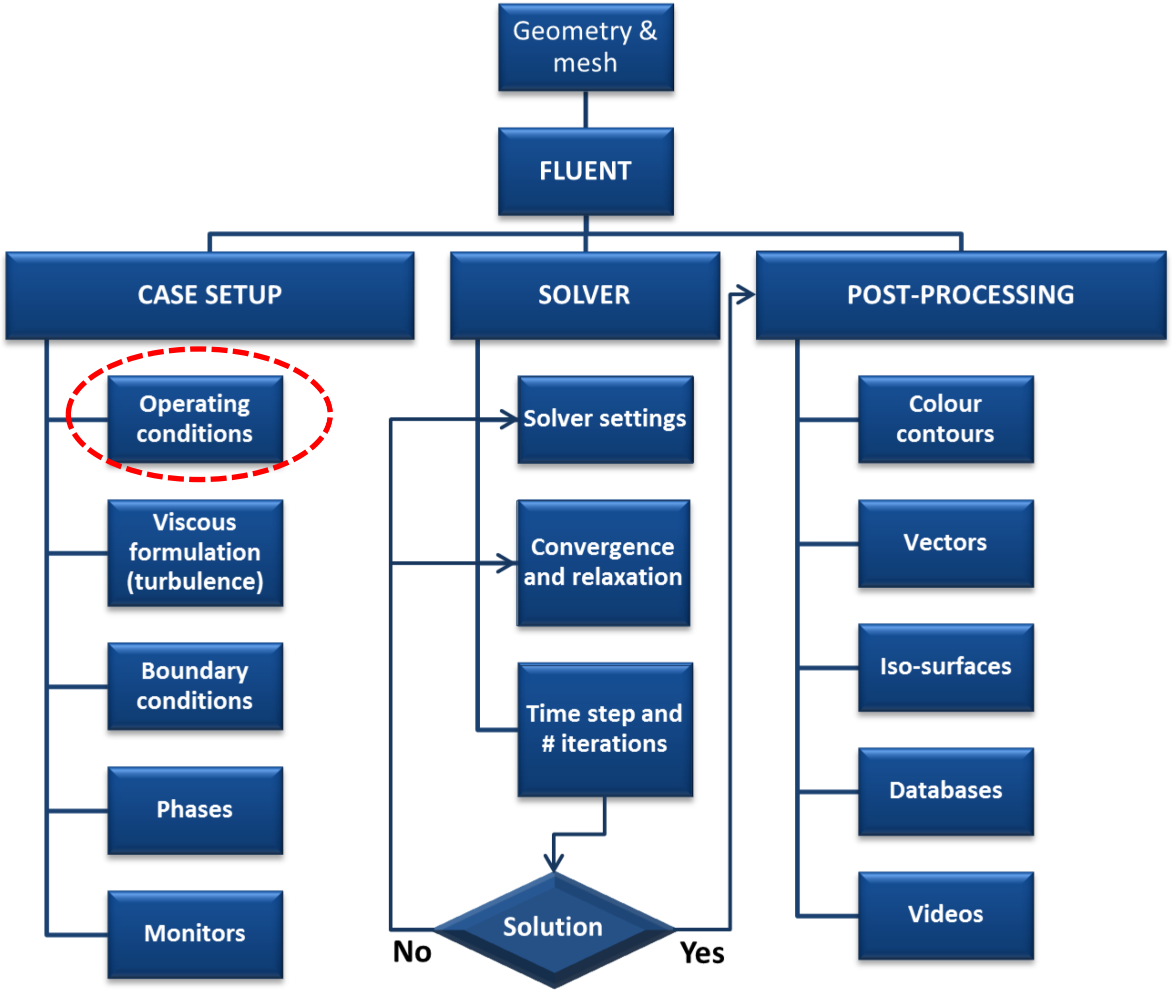


The divergence theorem states that the outward flux of a vector field through a closed surface is equal to the volume integral of the divergence over the region inside the surface. *Intuitively, it states that the sum of all sources minus the sum of all sinks gives the net flow out of a region.*

$$\frac{\partial}{\partial t}\int_V U dV = -\oint_S \vec{F}.d\vec{S} + \int_V Q dV$$

http://pruffle.mit.edu/3.016-2005/Lecture_16_web/node2.html

Operating Conditions



Transient vs steady state schemes

For transient simulations, the governing equations (N-S) must be discretized in both space and time. The spatial discretization for the time-dependent equations is identical to the steady-state case. Temporal discretization involves the integration of every term in the differential equations over a time step ΔT. For instance:

$$\frac{\partial(v)}{\partial t} = \mathrm{F}(v)$$

where $\mathrm{F}(v)$ incorporates any spatial discretization. If the time derivative is discretized using backward differences, the first-order temporal discretization is:

$$\frac{v^{n+1} - v}{\Delta t} = \mathrm{F}(v)$$

Where, $n + 1$= value of v at the next time level $(t + \Delta t)$ and n= value of v at the current time. Once the time derivative has been discretized, a choice remains for evaluating $\mathrm{F}(v)$: in particular, which time level values of v should be used in evaluating F.

*ANSYS-FLUENT v.12.0 theory guide

Steady state:
Flow characteristics at any given point in space are constant in time, e.g. v = *f(x,y,z).*

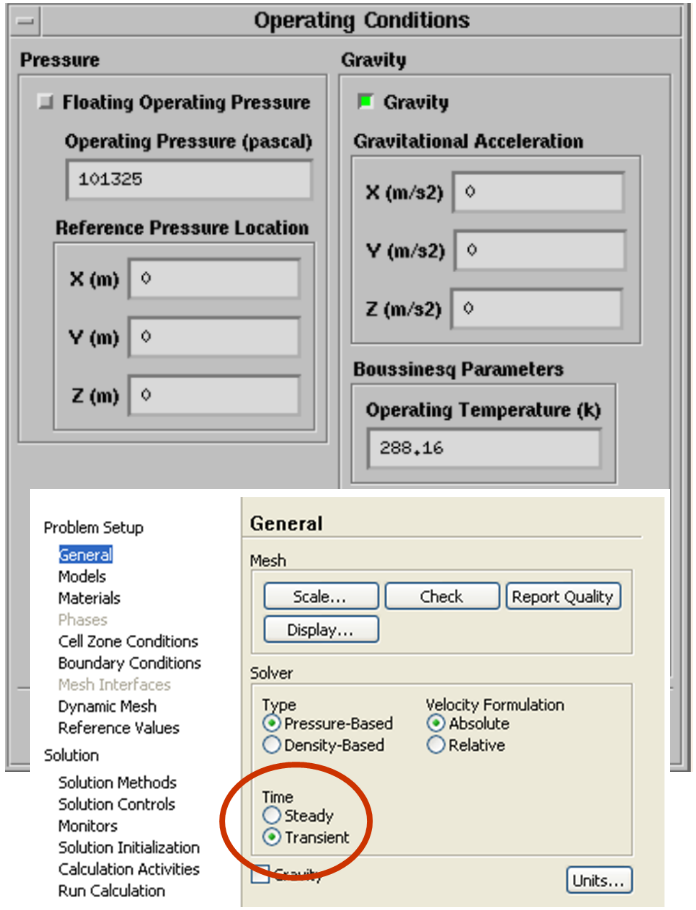
$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

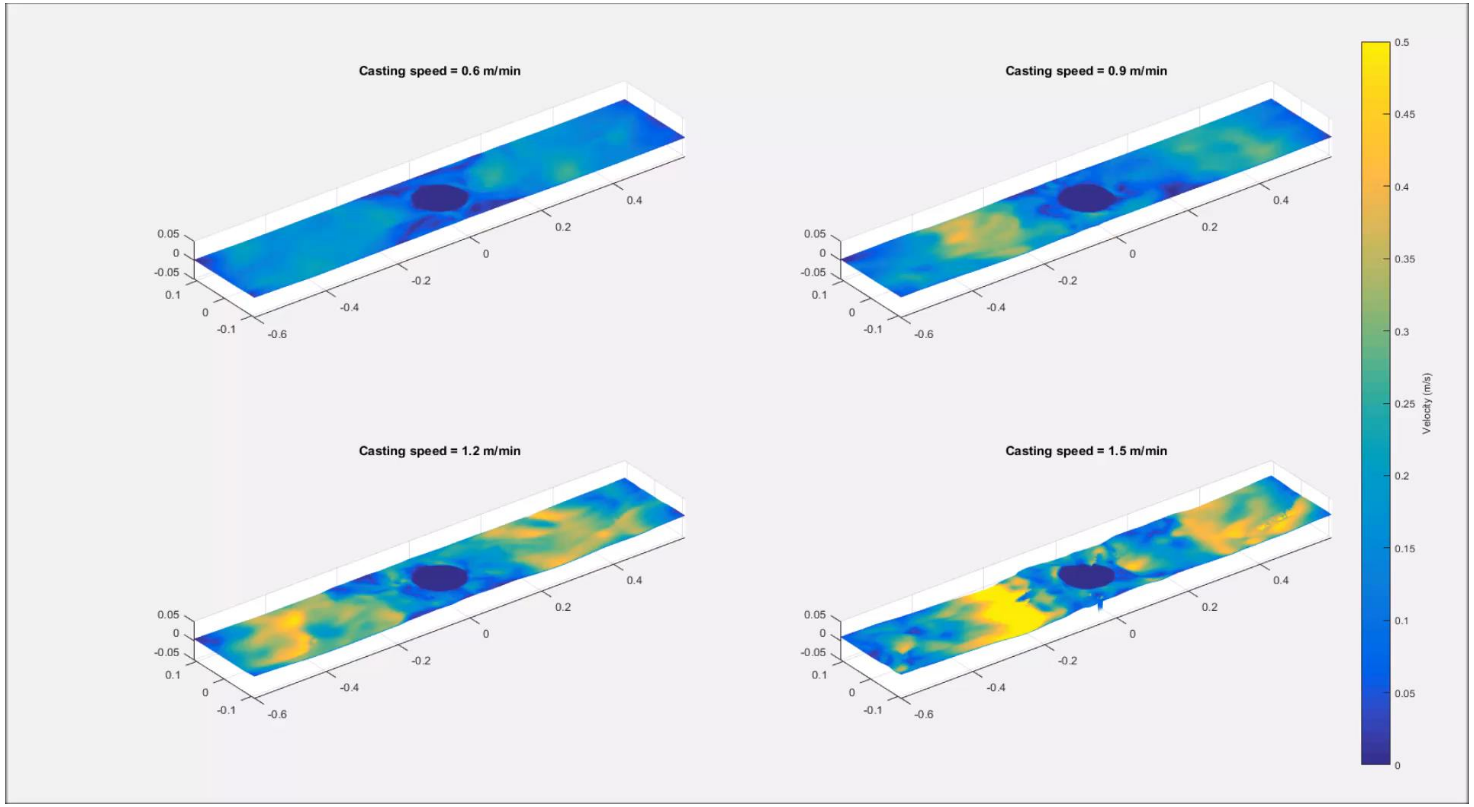
(0)

Unsteady state:
Flow characteristics at any given point in space change with time, e.g. *f = f(x,y,z,t).*

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

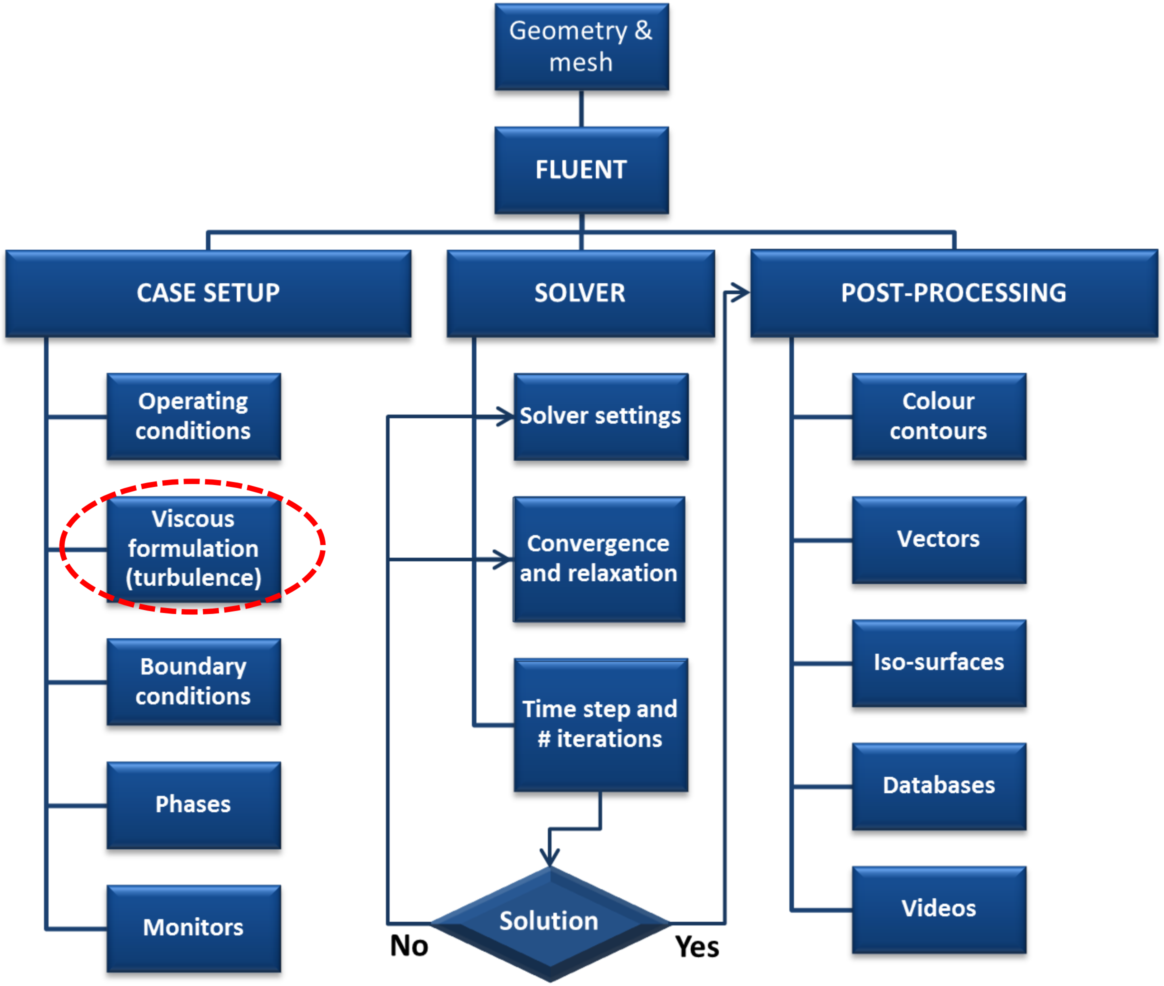
Operating Conditions





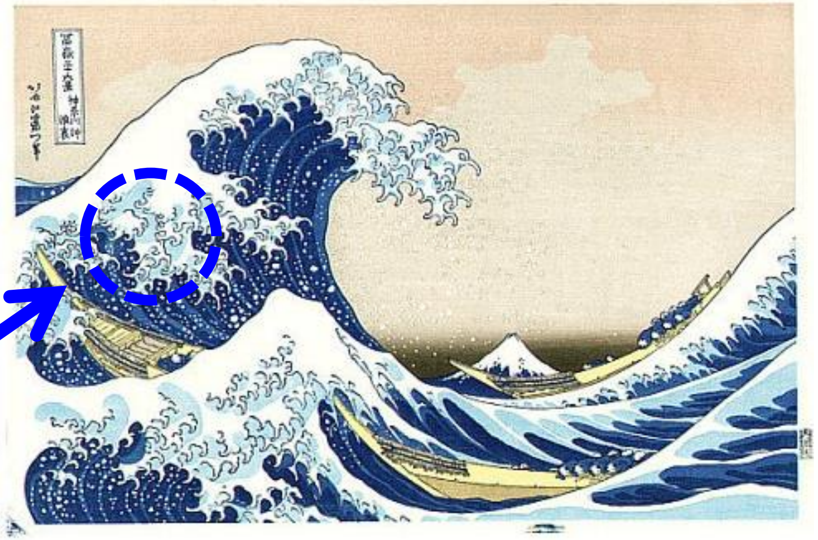
* *ANSYS FLUENT user's guide*

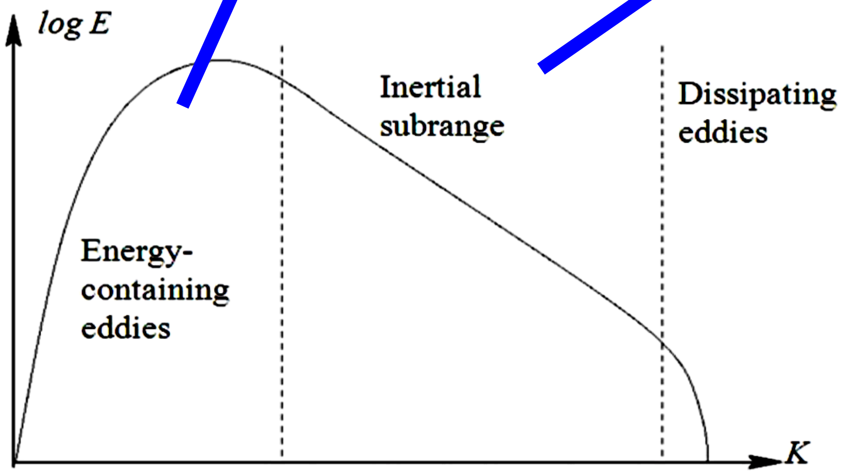
Case setup

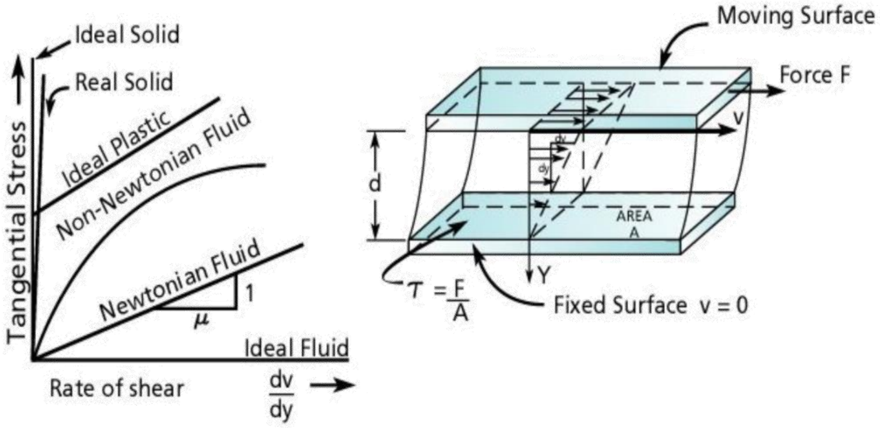


Geometry & mesh
FLUENT
CASE SETUP
SOLVER
POST-PROCESSING
Operating conditions
Viscous formulation (turbulence)
Boundary conditions
Phases
Monitors
Solver settings
Convergence and relaxation
Time step and # iterations
Solution
No
Yes
Colour contours
Vectors
Iso-surfaces
Databases
Videos

Turbulent scales…

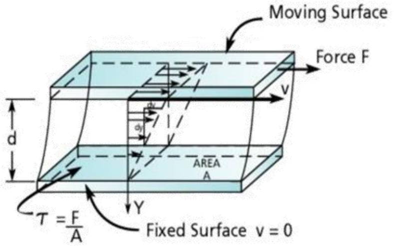


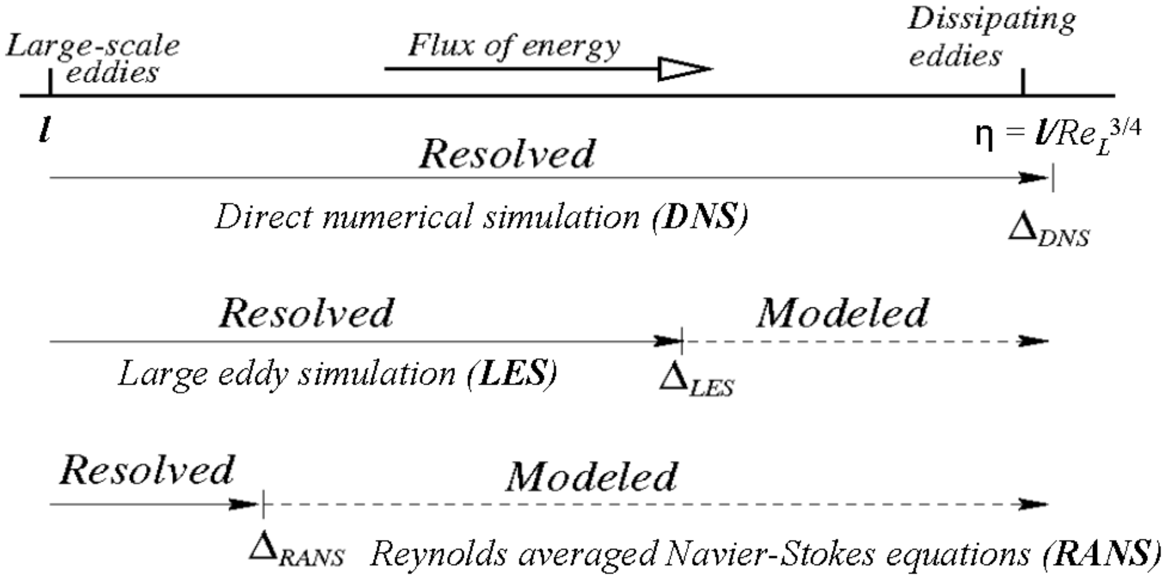




http://www.pumpfundamentals.com/about_fluids.htm

Turbulence modelling





http://www.bakker.org/dartmouth06/engs150/10-rans.pdf

Choosing the right model for turbulent viscosity

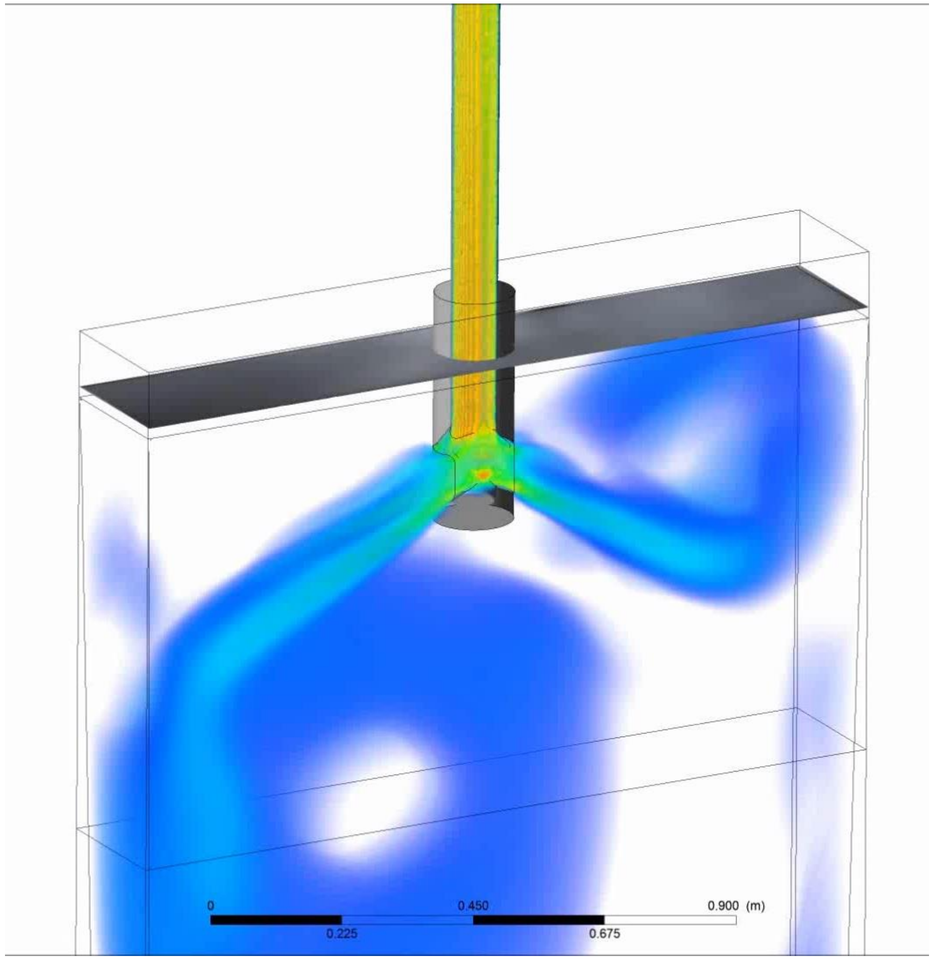
- Mixing length
- Spalart-Allmaras
- Standard k-ε
- k-ε RNG model
- Realizable k-ε
- k-ω model
- Etc…

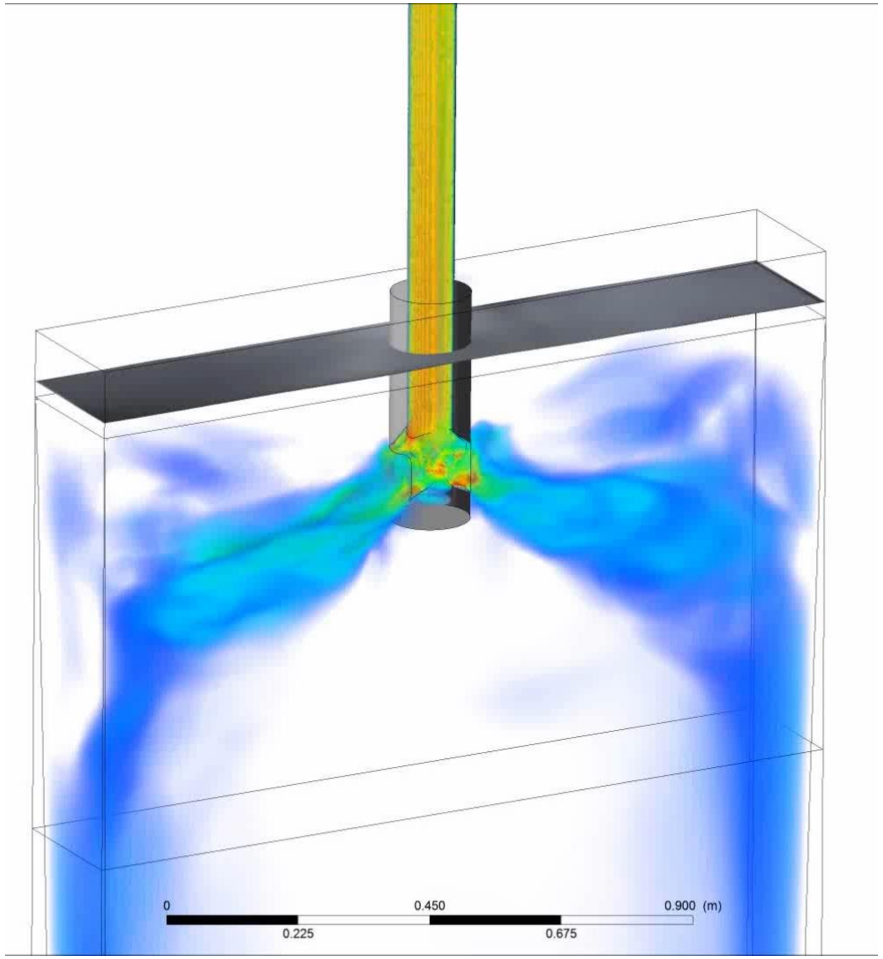
The election of an "appropriate" turbulence model depends on the problem since: *"turbulence is a property of the flow itself, and not of the fluid"**

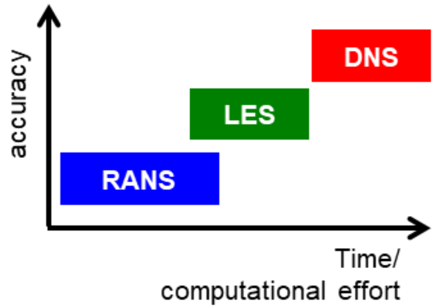
*Lectures in Turbulence for the 21st Century. Prof. William K. George, Department of Aeronautics, Imperial College London, UK & Department of Applied Mechanics, Chalmers University of Technology, SWEDEN. http://www.turbulence-online.com

Choosing the right turbulence model...

k-ε turbulence model	**Large Eddy Simulation (LES)**

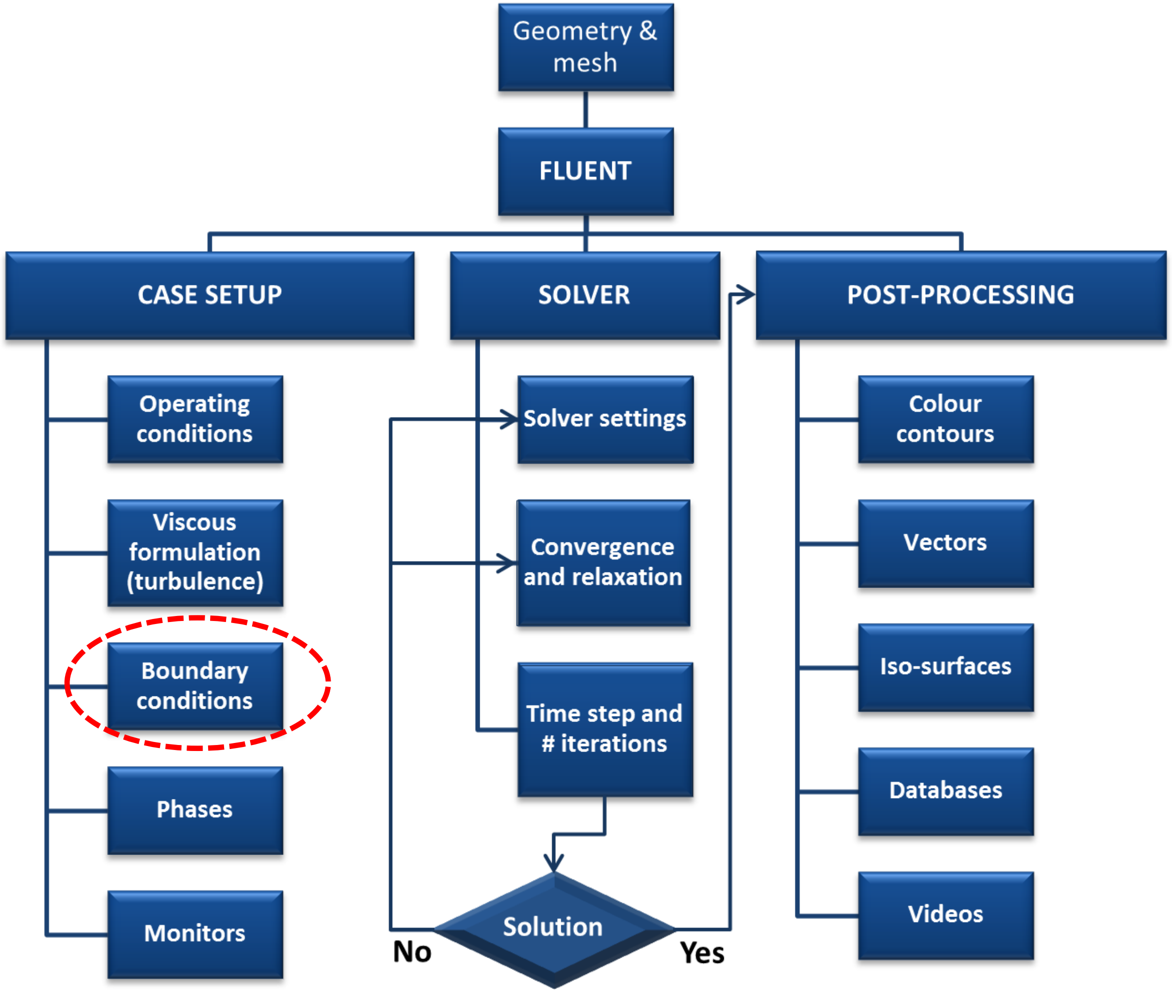






SWERIM
sidenor
RINA
Materials Processing Institute
BFI

Boundary Conditions



Initial and boundary conditions

Initial conditions:

ρ, v and T must be given at time t=0 in all the computational domain (e.g. Initialize).

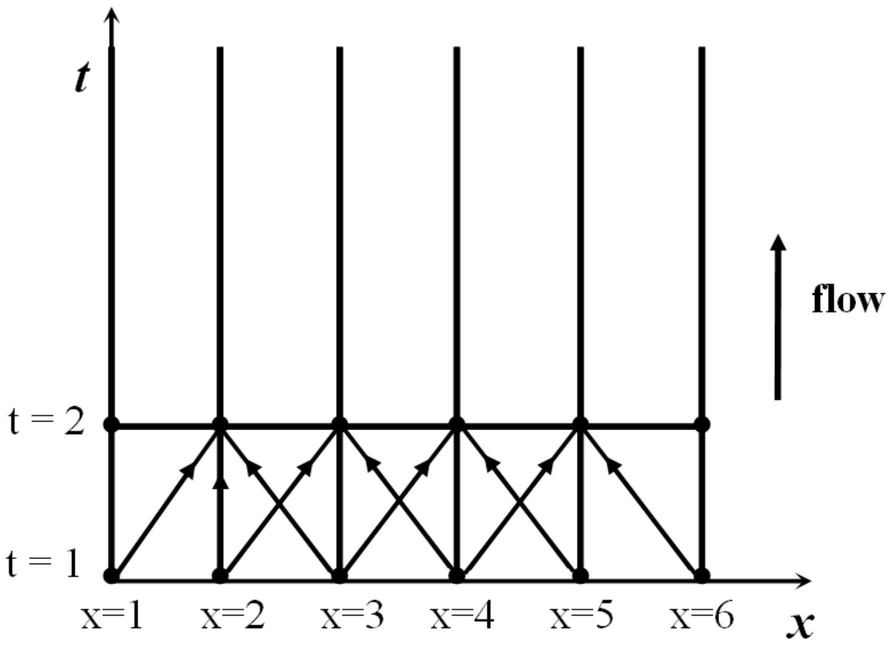
Boundary conditions:

On solid walls:

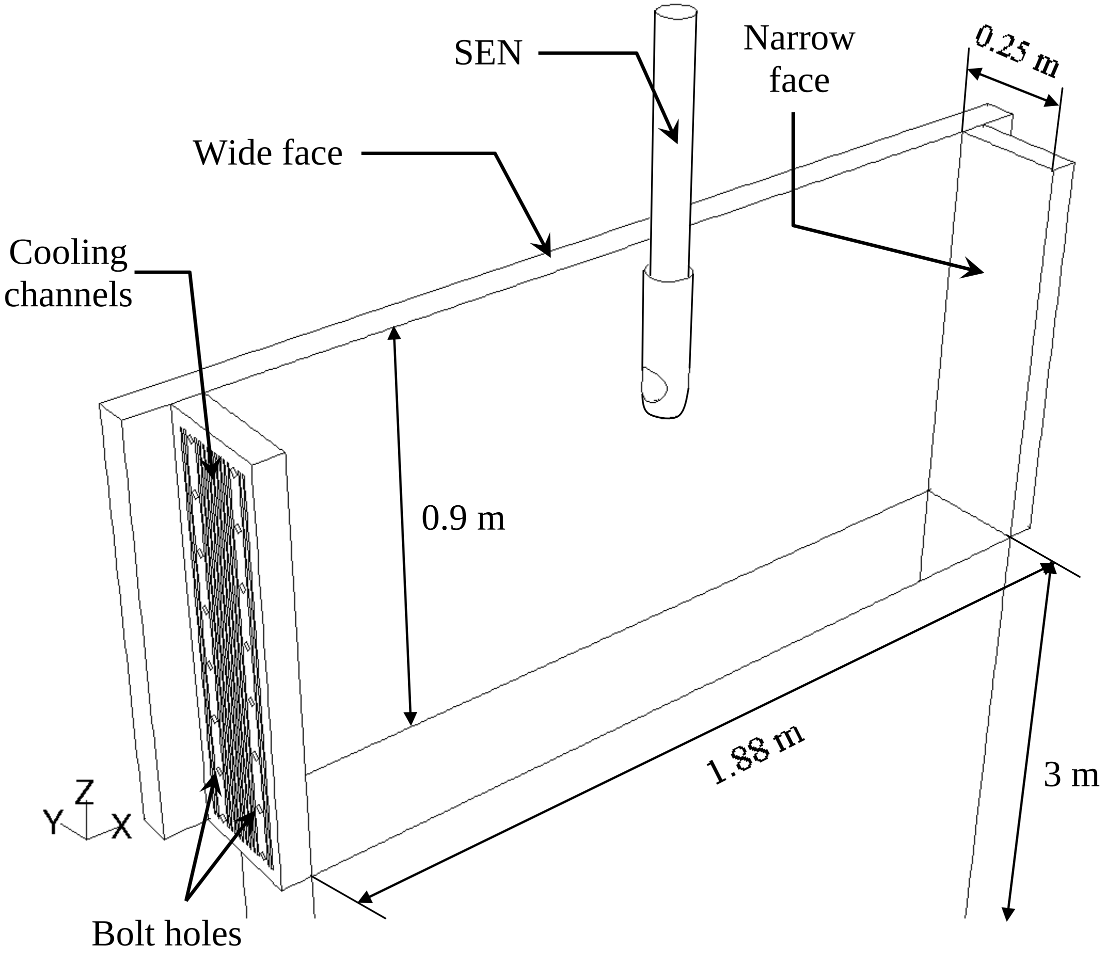
- $v = v_{wall}$ (no-slip condition).
- $T = T_{wall}$ (fixed temperature) or
- $k\partial T/\partial n = -q_{wall}$ (fixed heat flux).

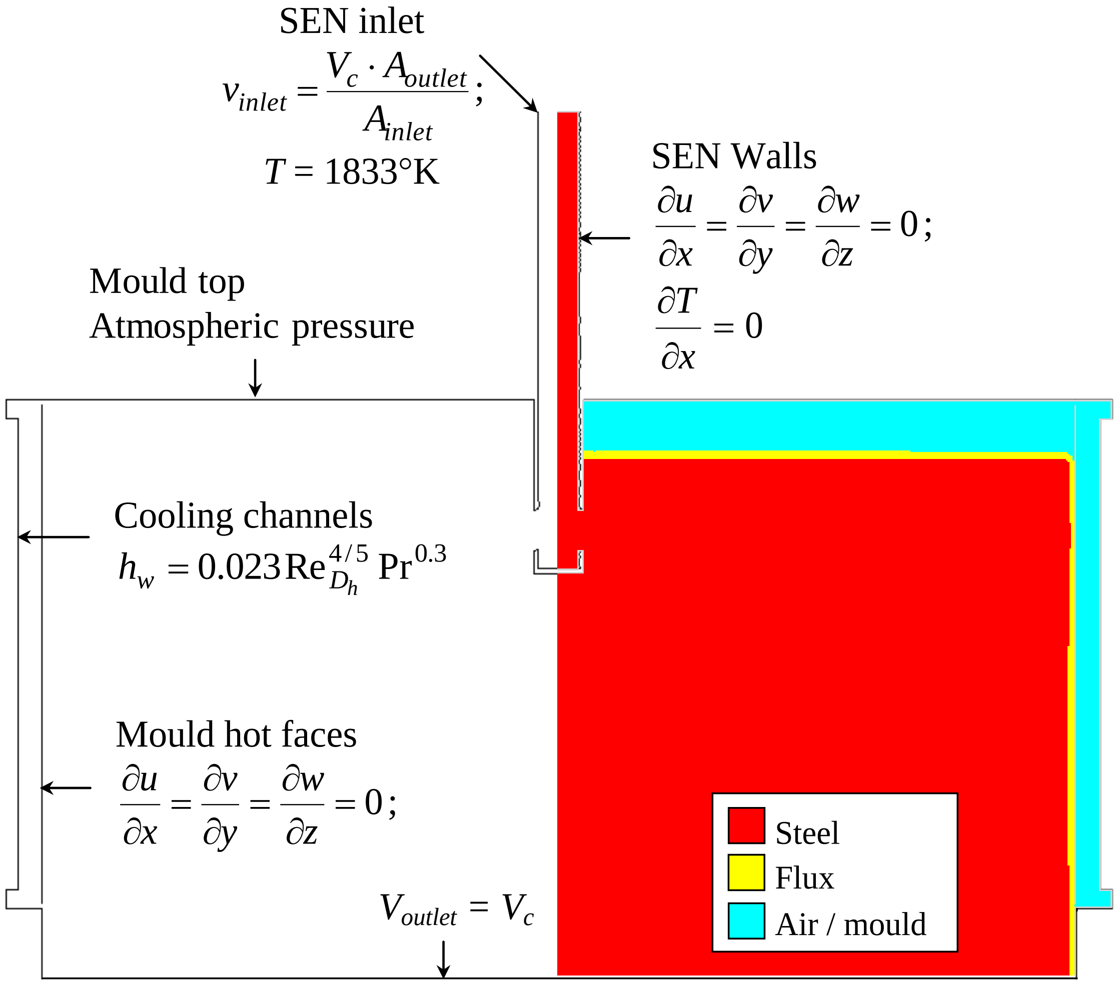
On fluid boundaries:

- Inlet: ρ, v and T must be known as a function of position.
- Outlet: $-p+\mu\,\partial v_n/\partial n = F_n$ and $\mu\,\partial v_t/\partial n = F_n$ (stress continuity). (F is the given surface stress.

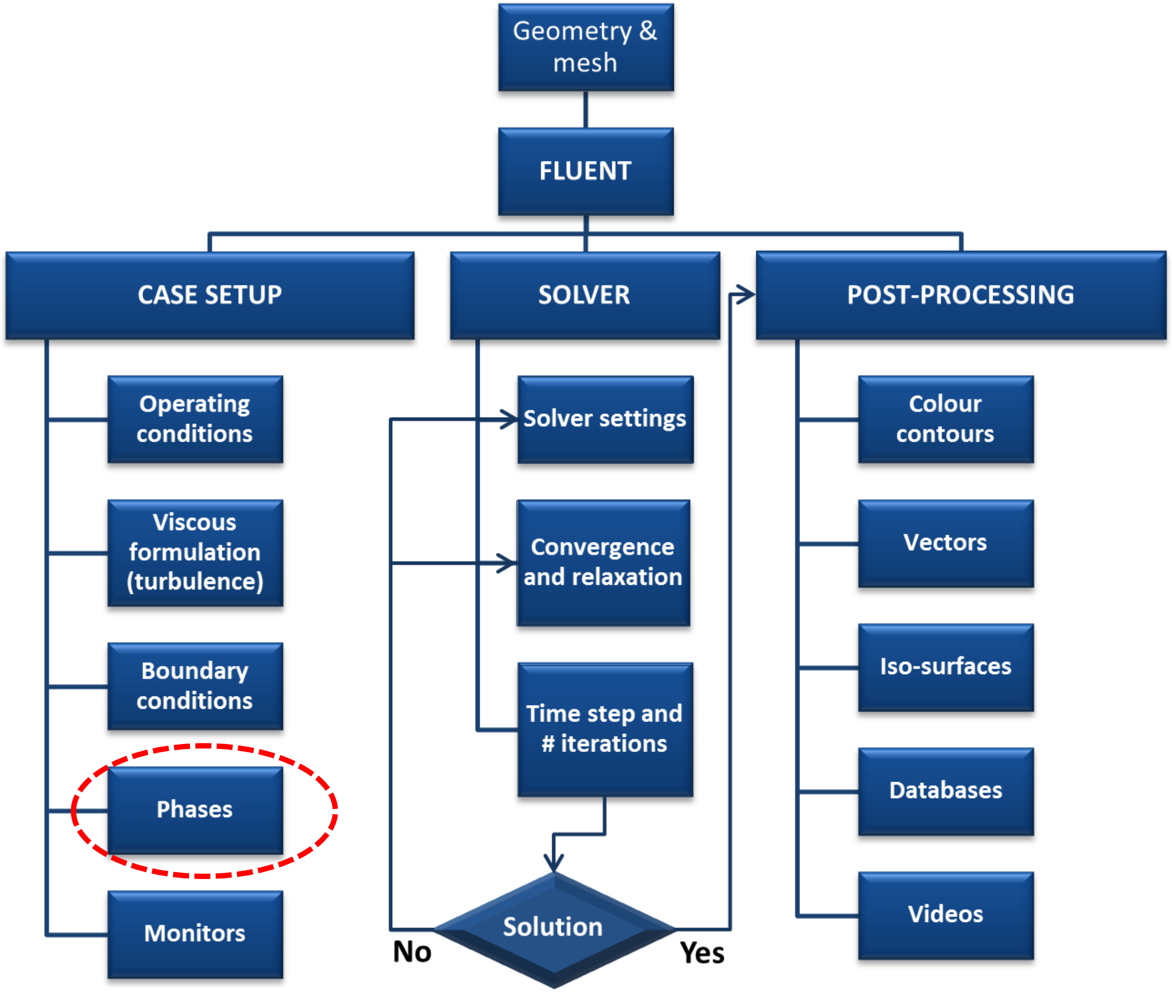


Initial and Boundary Conditions

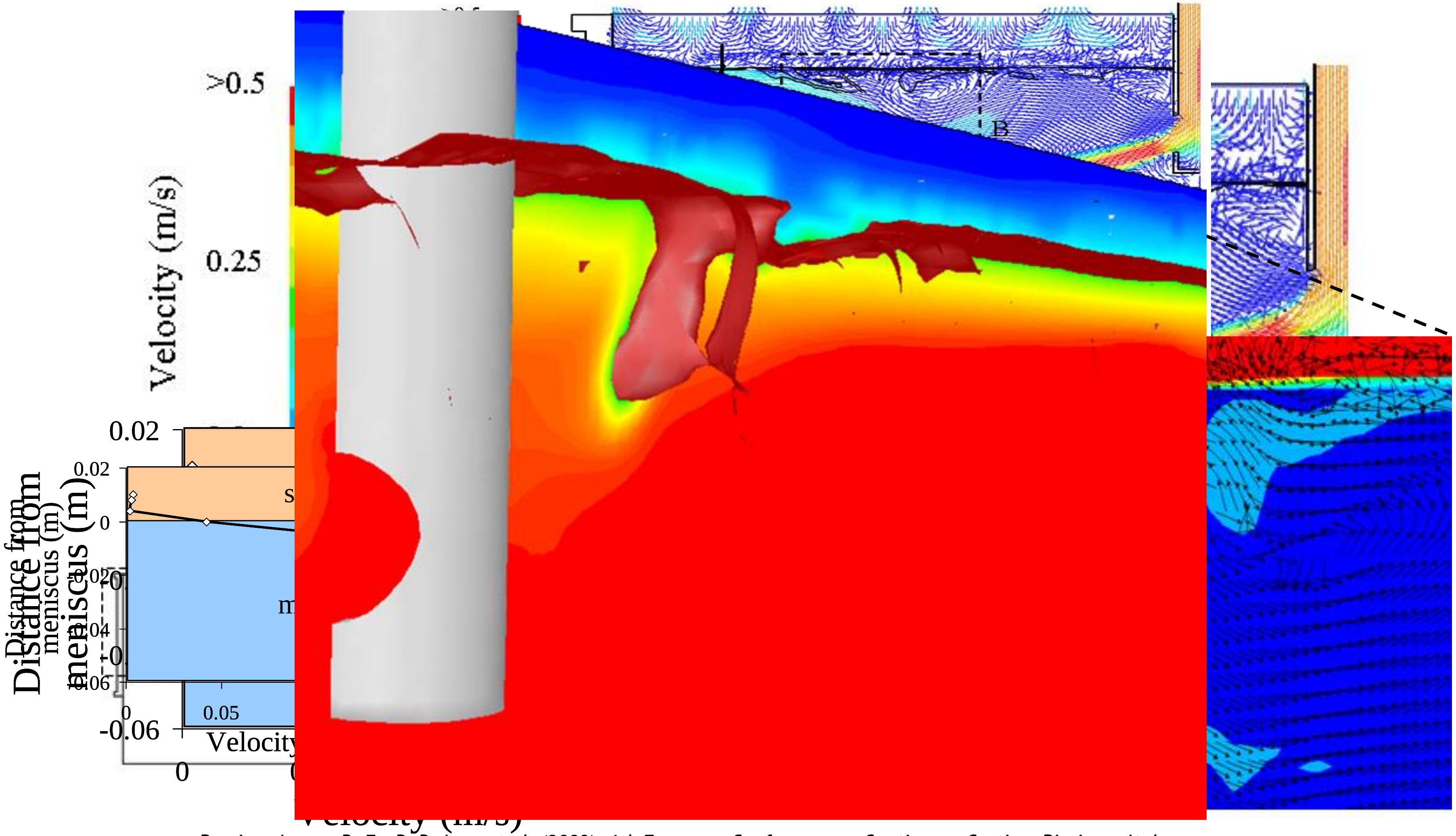




Phases definition

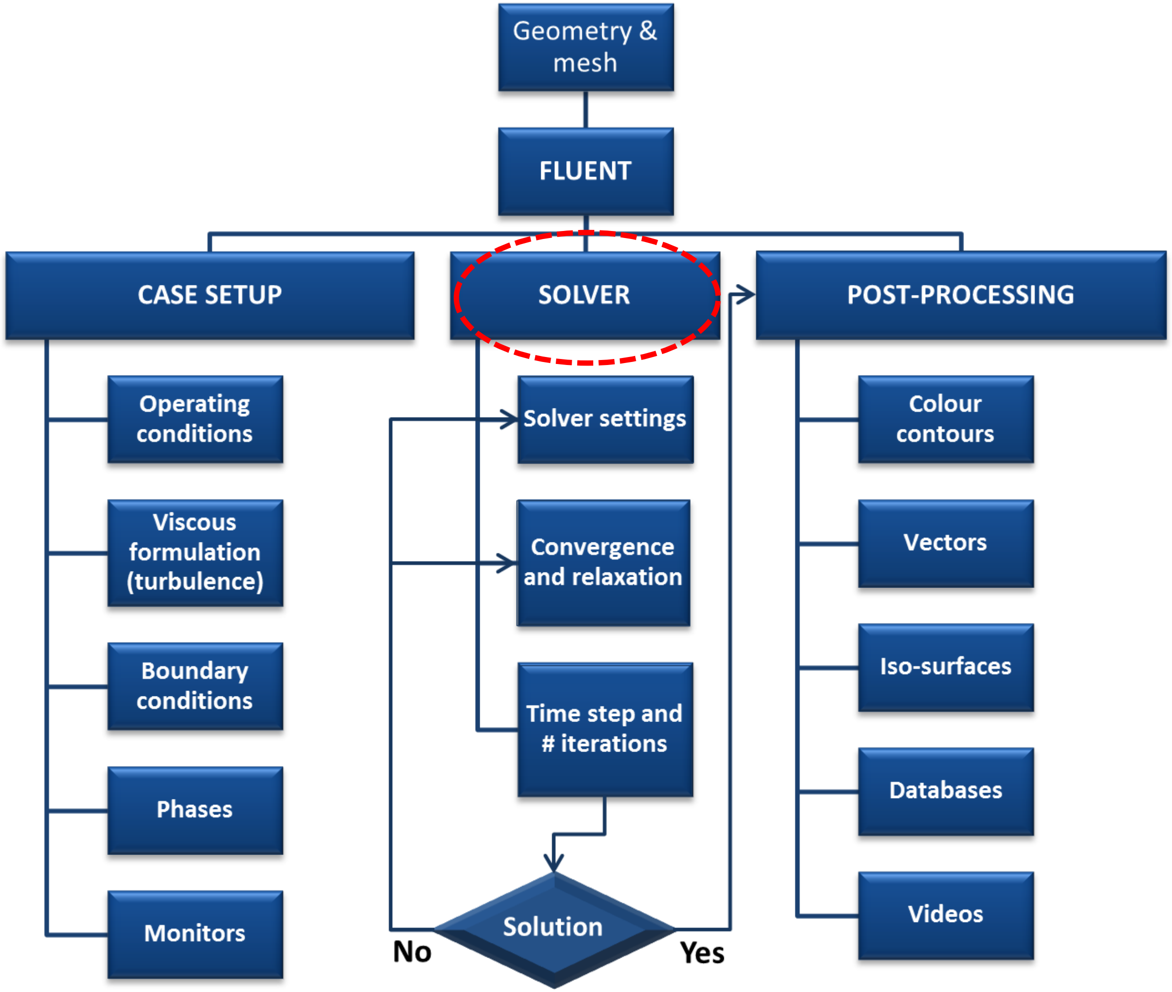


Solving multiple phases

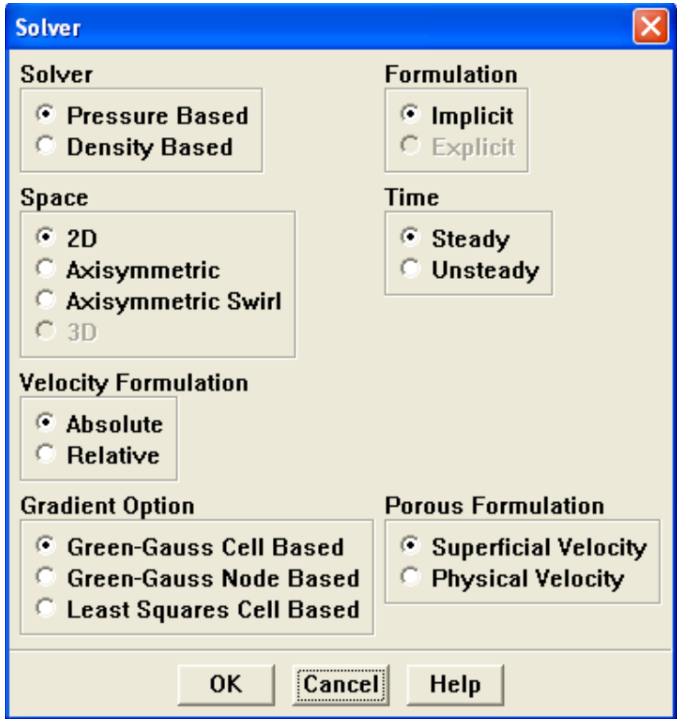


Ramirez-Lopez, P. E., P. D. Lee, et al. (2008). 6th European Conference on Continuous Casting. Riccione, Italy

Solver


Geometry & mesh
FLUENT
CASE SETUP
SOLVER
POST-PROCESSING
Operating conditions
Viscous formulation (turbulence)
Boundary conditions
Phases
Monitors
Solver settings
Convergence and relaxation
Time step and # iterations
Solution
No
Yes
Colour contours
Vectors
Iso-surfaces
Databases
Videos

Solver: Pressure vs density based



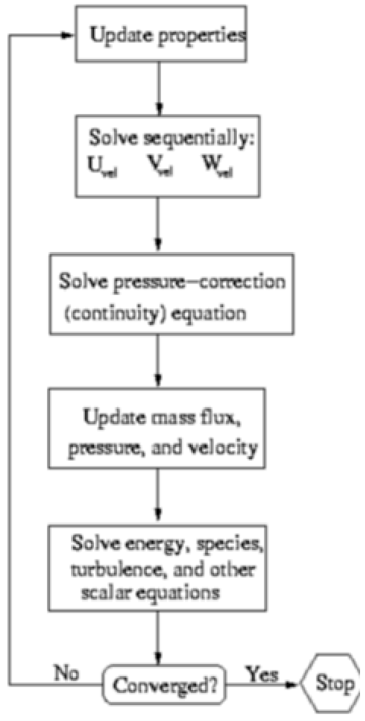
The **pressure-based** solver employs an algorithm where mass conservation is achieved by solving an extra pressure (or pressure correction) equation. This eq. is derived from the continuity and momentum equations in a way that the velocity field, corrected by the pressure, satisfies continuity. Since the governing equations are nonlinear and coupled to one another , the solution process involves and iterative process where the entire set of governing equations is solved repeatedly until the solution converges.

The **density-based** solver solves the governing equations coupled together, while additional scalars will be solved afterwards and sequentially. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained.

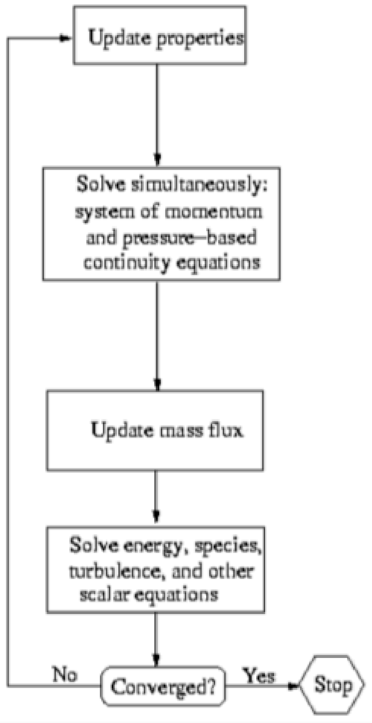
** ANSYS FLUENT user's guide*

Solver settings*

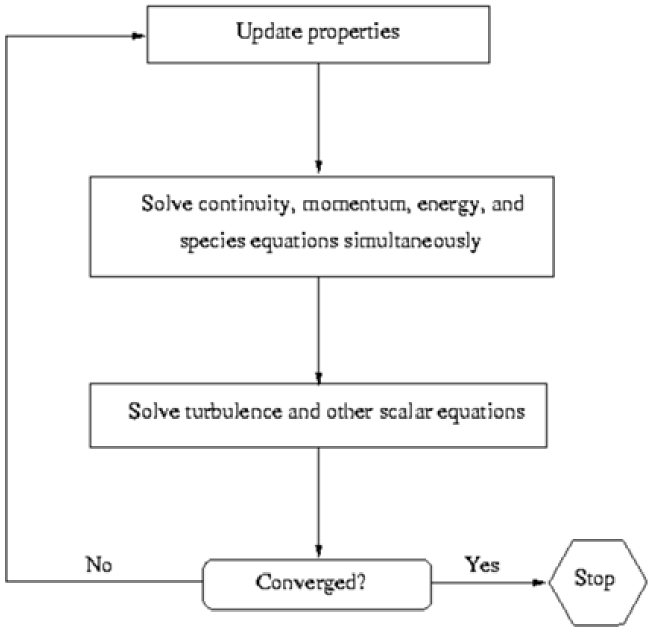
Pressure based segregated solver



Pressure based coupled solver



Density based solver

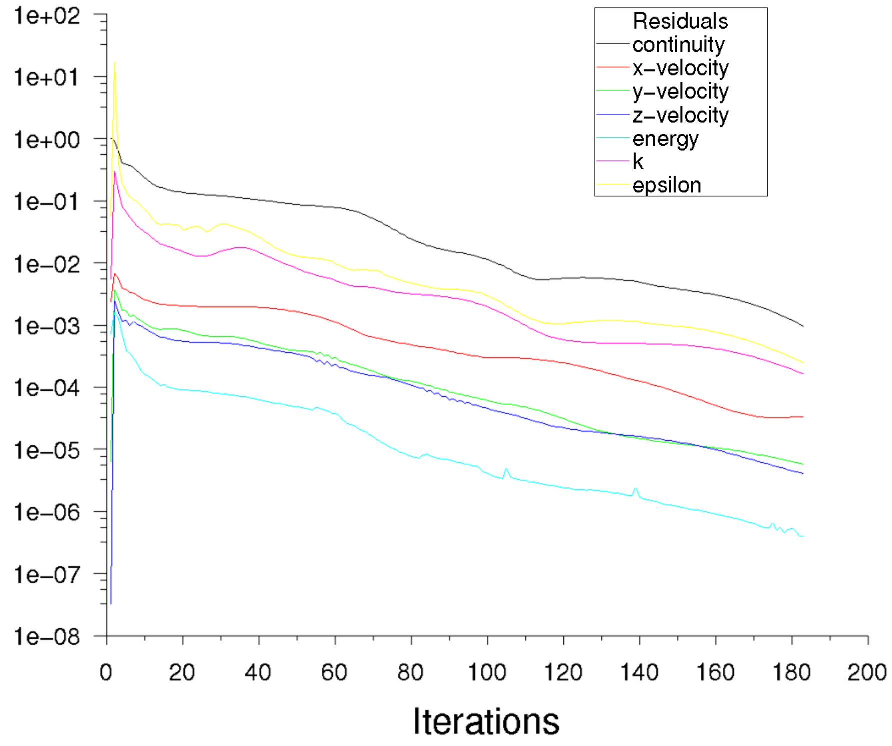


** The pressure-based solver has been traditionally used for incompressible and mildly compressible flows, while the density-based approach was originally designed for high-speed compressible flows.*

Stability & convergence

In general sense, ***convergence*** is the capability of a set of discretized equations to represent the analytical solution of a problem (e.g. the model converges when the numerical solution approaches a fixed value and the error is below the threshold defined by the user). Naturally, this occurs when the grid is extremely fine and cell size approaches zero.

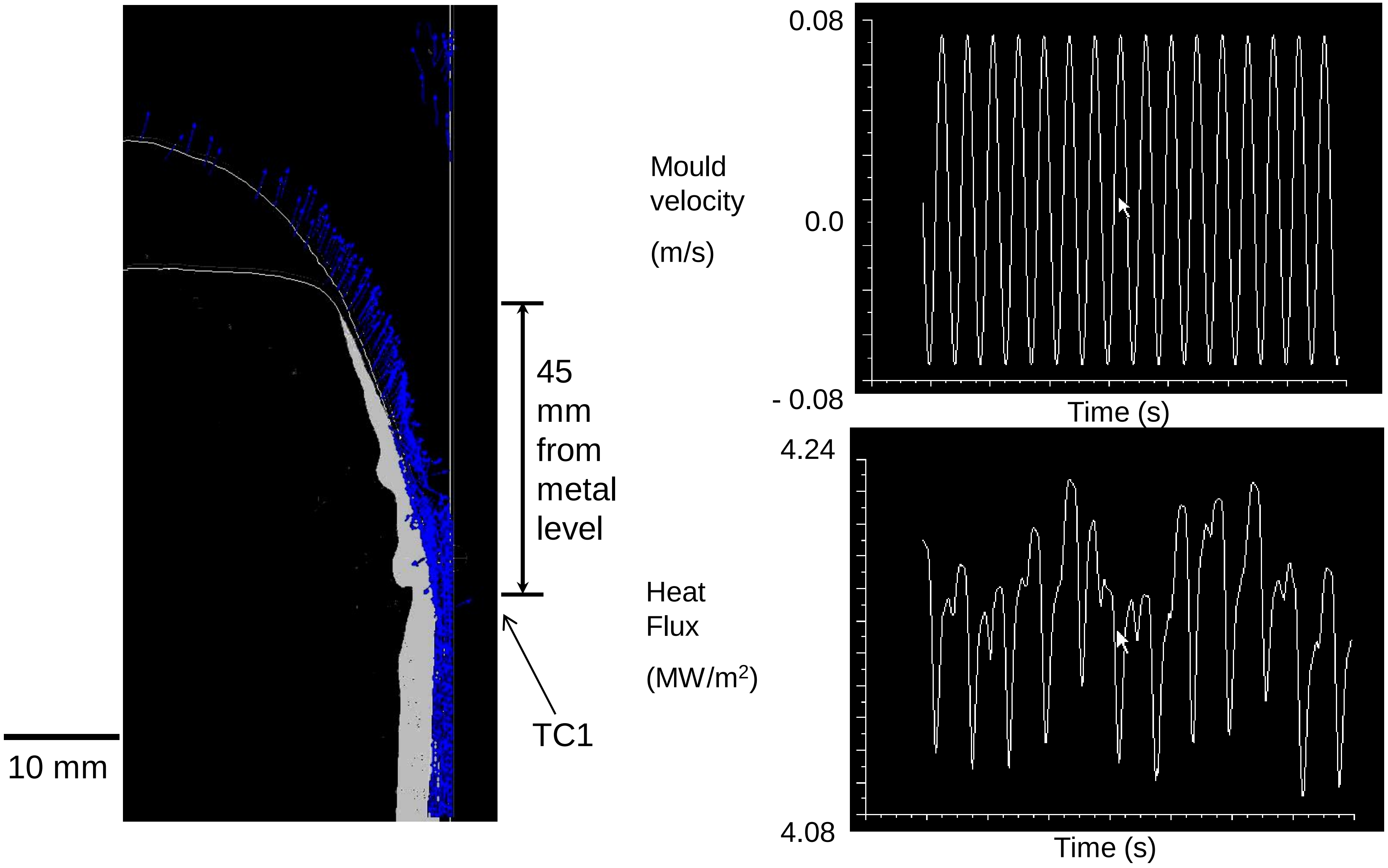
Similarly, the discretized model is **stable** if the result approaches the analytical solution despite any local or discrete deviation during the iterative process



iter	continuity	x-velocity	y-velocity	z-velocity	k	epsilon	time
190	1.21547e-03	3.56444e-04	6.45467e-04	7.21348e-04	1.27988e-03	8.79871e-03	24:30
191	1.12456e-03	3.37457e-04	6.43543e-04	7.06742e-04	9.46545e-04	8.46544e-03	24:52
192	9.92476e-04	3.24787e-04	6.41548e-04	6.56742e-04	9.07976e-04	8.06747e-03	25:13
solution converged							
192	9.92476e-04	3.24787e-04	6.41548e-04	6.56742e-04	9.07976e-04	8.06747e-03	25:13

Post-processing & application examples

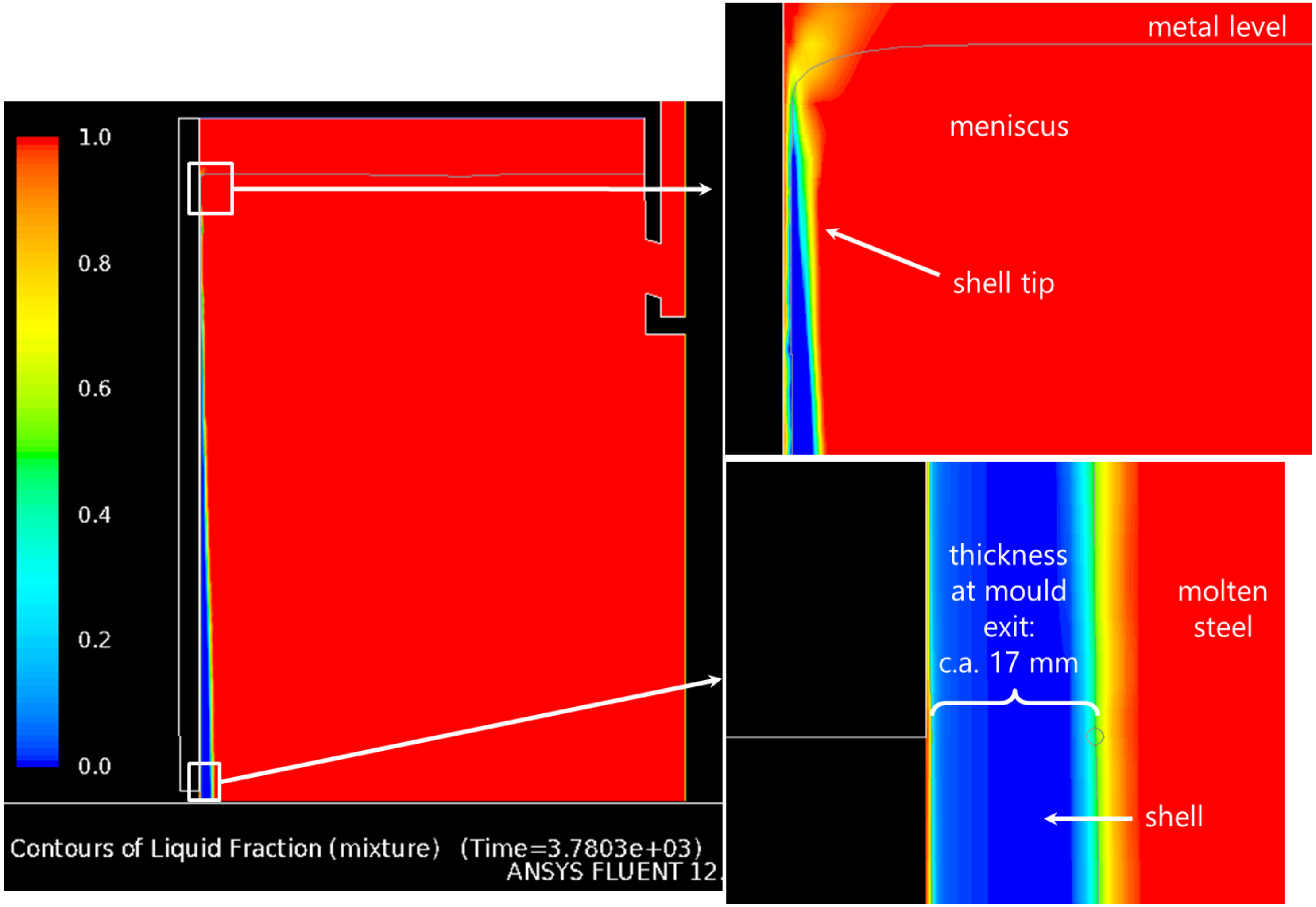
heat flux tracking during shell formation & oscillation



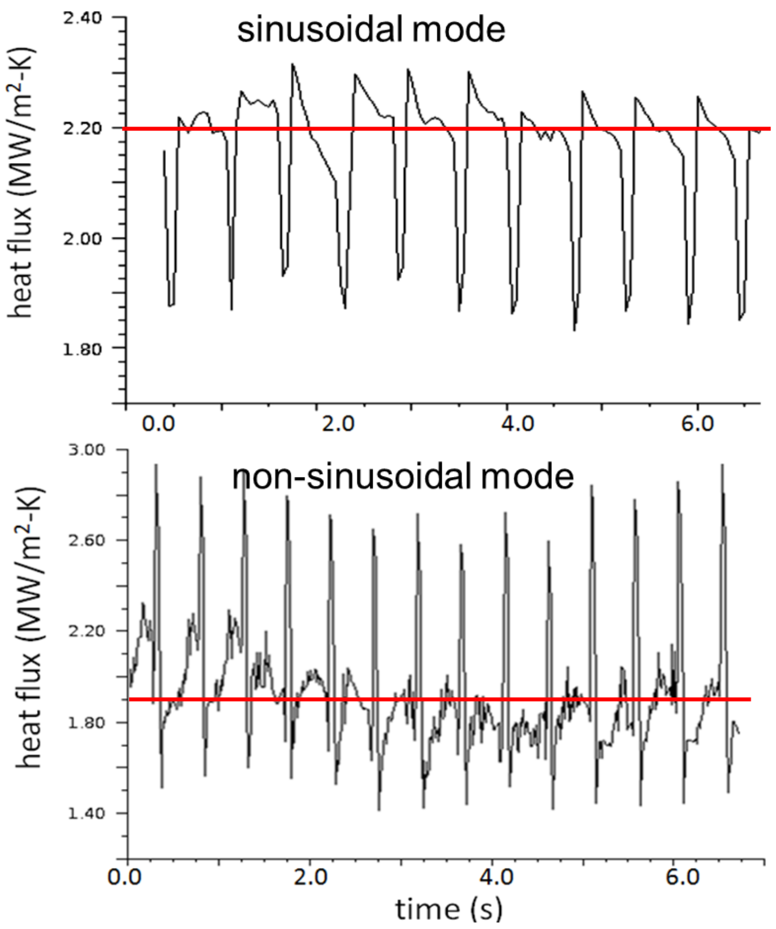
P. E. Ramirez Lopez, P. D. Lee and K. C. Mills: *2nd International Symposium on Cutting Edge of Computer Simulation of Solidification and Casting (CSSC2010), Sapporo, Japan, University of Tokio, 2010, pp.*

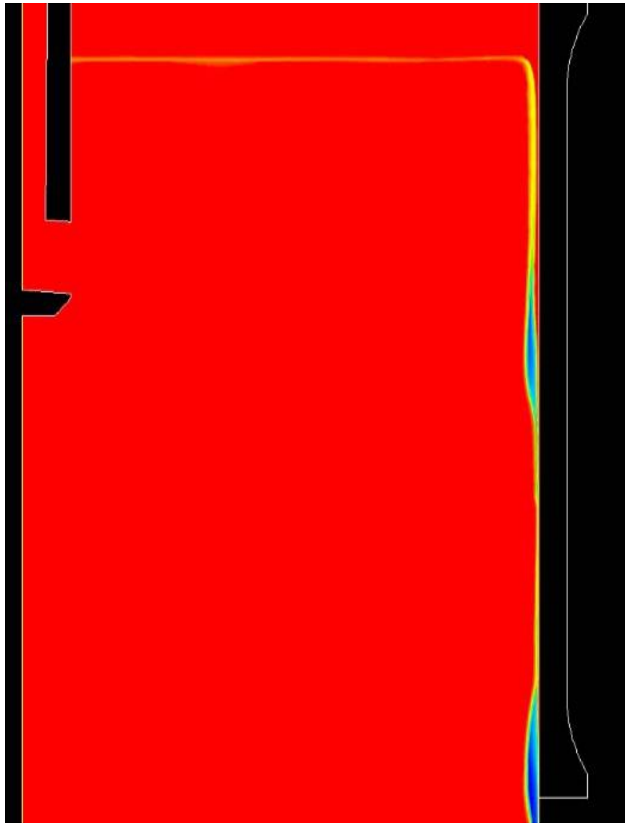
Mould oscillation studies in 2D

- Optimization of Oscillation settings
- Maintaining same lubrication level with Non-sinusoidal modes which give better quality.



Sinusoidal vs non-sinusoidal oscillation



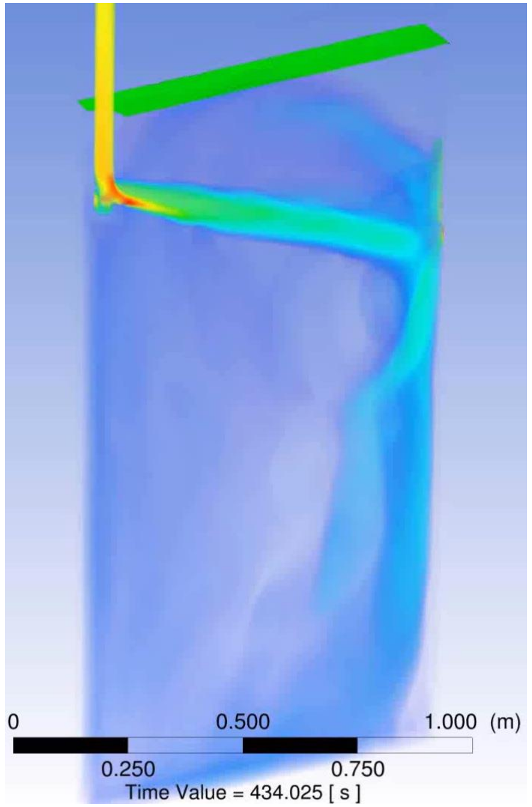


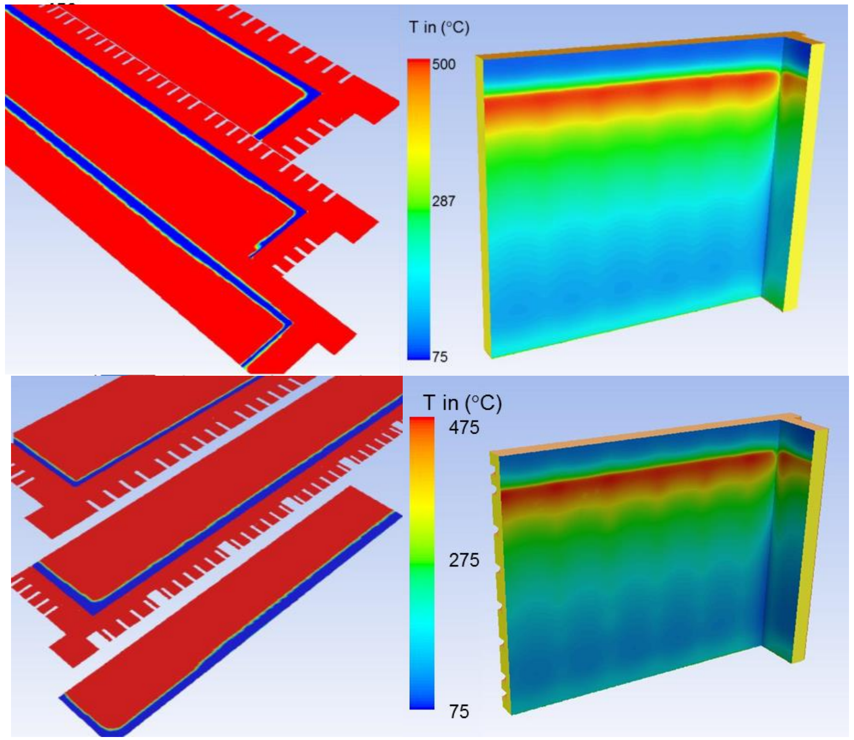
Breakout
(sin to non-sin)

Recovery
(non-sin to sin)

Digital Twin model for slabs

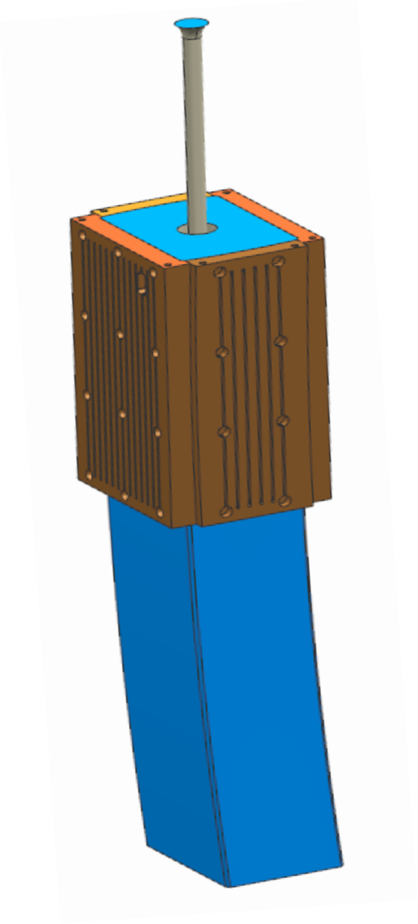
- First models to account flow on solidification for high resolution turbulence
- Optimization of flow, taper and casting conditions for different mould sizes.
- Discovery of nozzle design effects on solidification in narrow face which affect its strength (gutters)

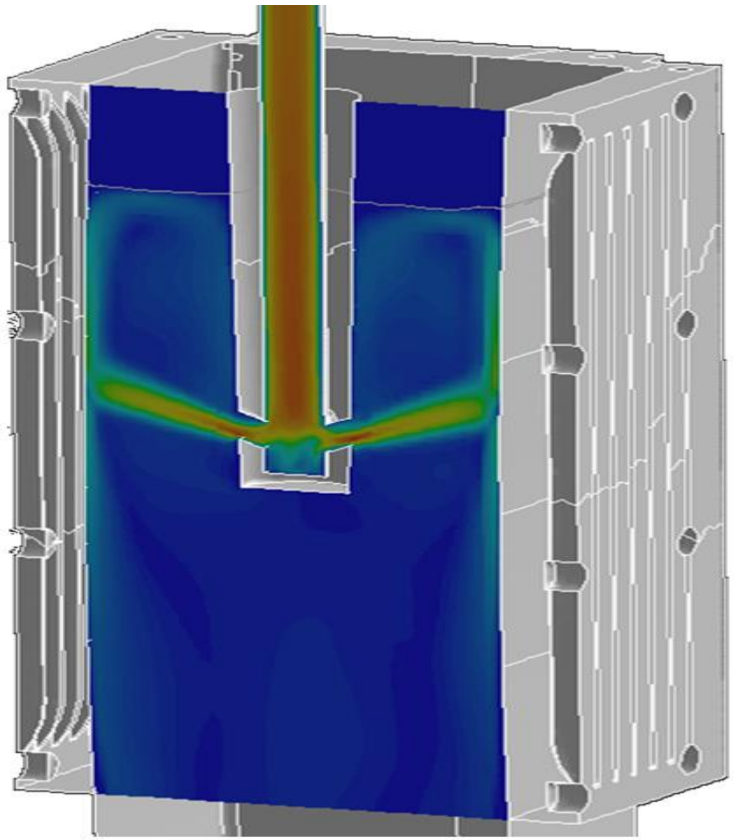




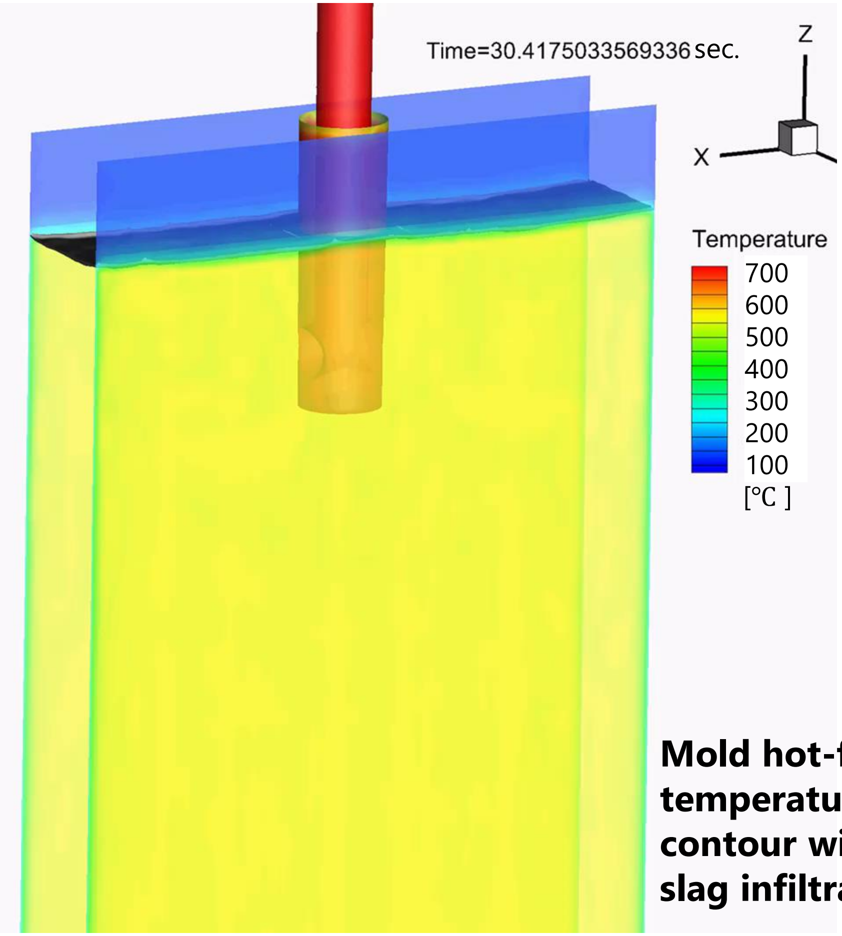
Full 3D slag infiltration modelling for blooms

- First 3D model of bloom for Ni-based alloys
- Model predicted accurately the behaviour of special casting powder
- Flow and lubrication issues were found to be related to depressions in blooms
- New project to optimize flow

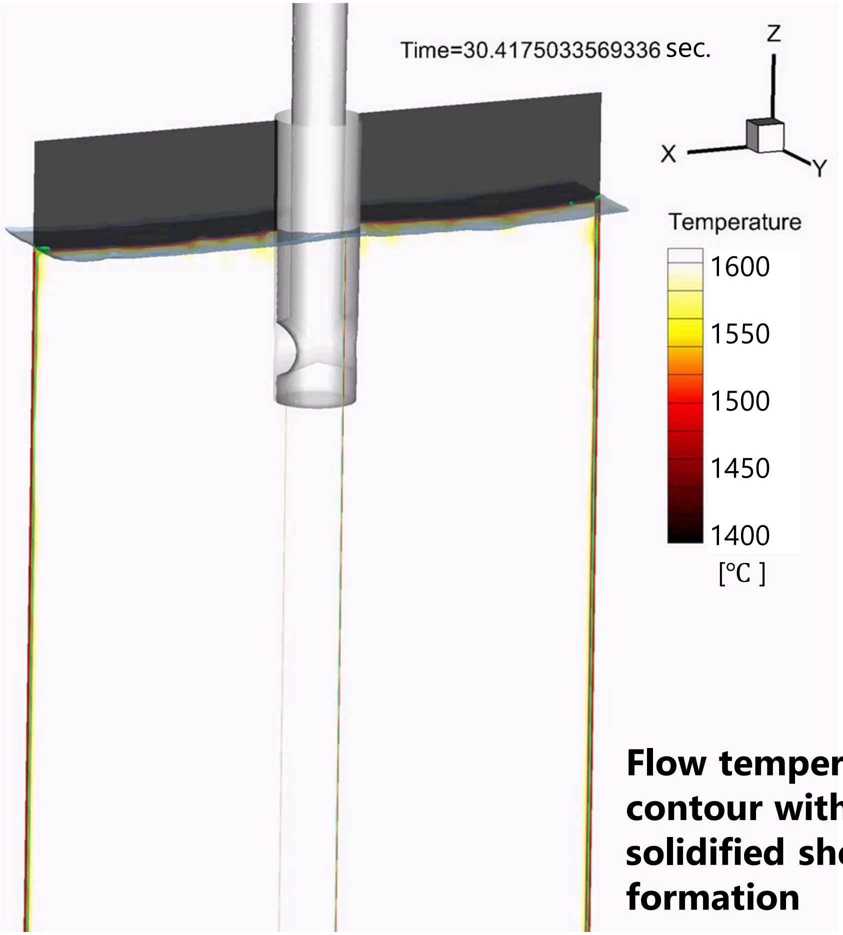




Digital Twin for Pilot caster



Mold hot-face temperature contour with slag infiltration



Flow temperature contour with solidified shell formation

Concluding remarks

Q2) What are the limitations you see to apply modelling in your job?

a) Software Cost

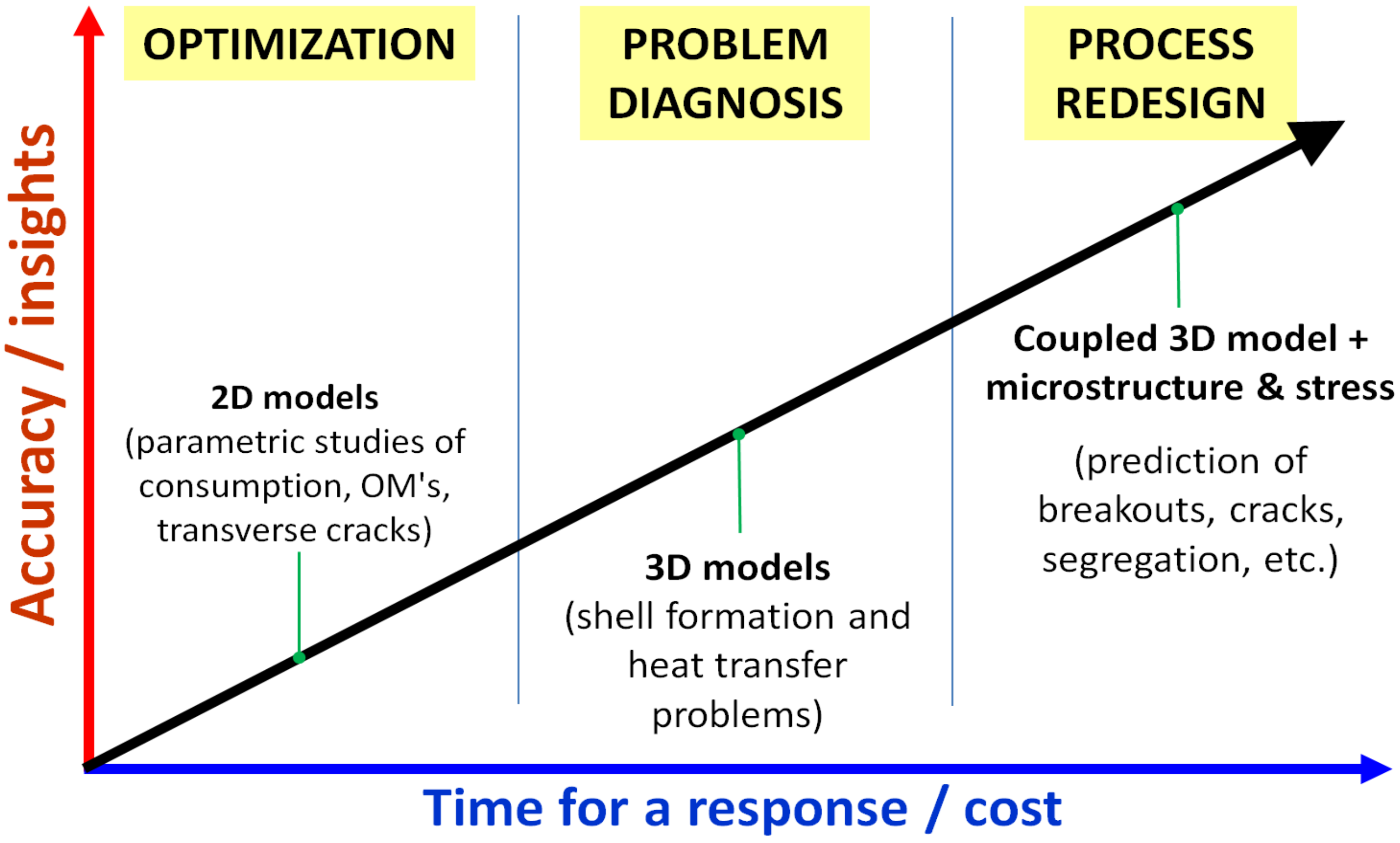
b) Hardware Cost

c) Lack of especialized training

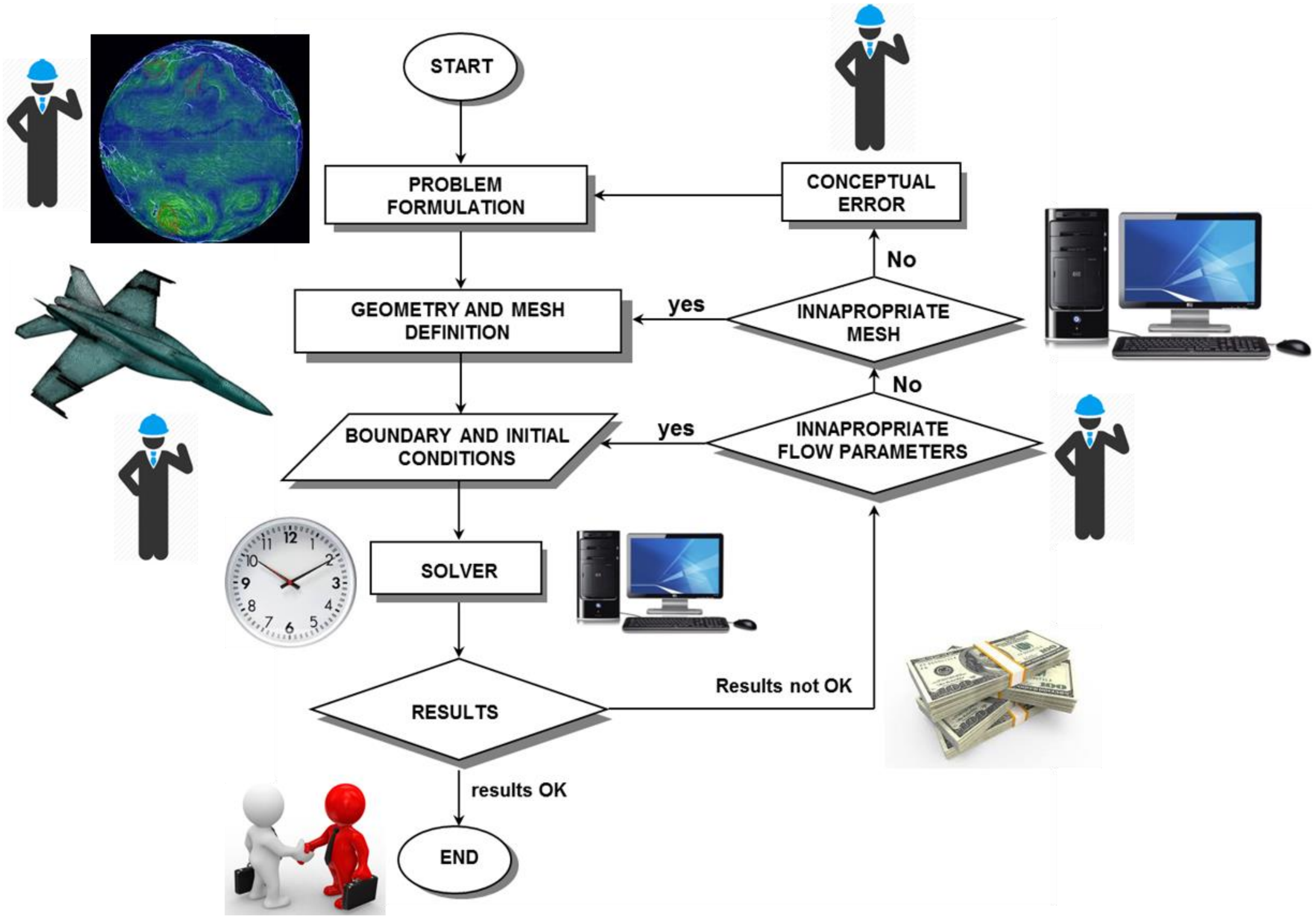
d) Lack of support/understanding in your organization

e) Not reliable results

The right model for the right job



Modelling Trade-off



Acknowledgements

The research leading to these results has received funding from the European Union's Research Programme of the Research Fund for Coal and Steel (RFCS) for the following projects:

RFCS DDT- Direct Defect Toolbox

RFCS SUPPORT-CAST

RFCS NNEWFLUX

RFCS OPTILOCALQH

VALCRA

Thanks for the attention!

Stay informed

http://valcra.eu/

https://www.linkedin.com/company/european-continuous-casting-network

VALCRA linkedin group (linkedin.com/groups/13794289/)

SWERIM

sidenor

RINA

Materials Processing Institute

BFI

European Commission

Research Fund for Coal & Steel