Customer Training Material

Lecture 4

Natural Convection

Heat Transfer Modeling using ANSYS FLUENT

Outline

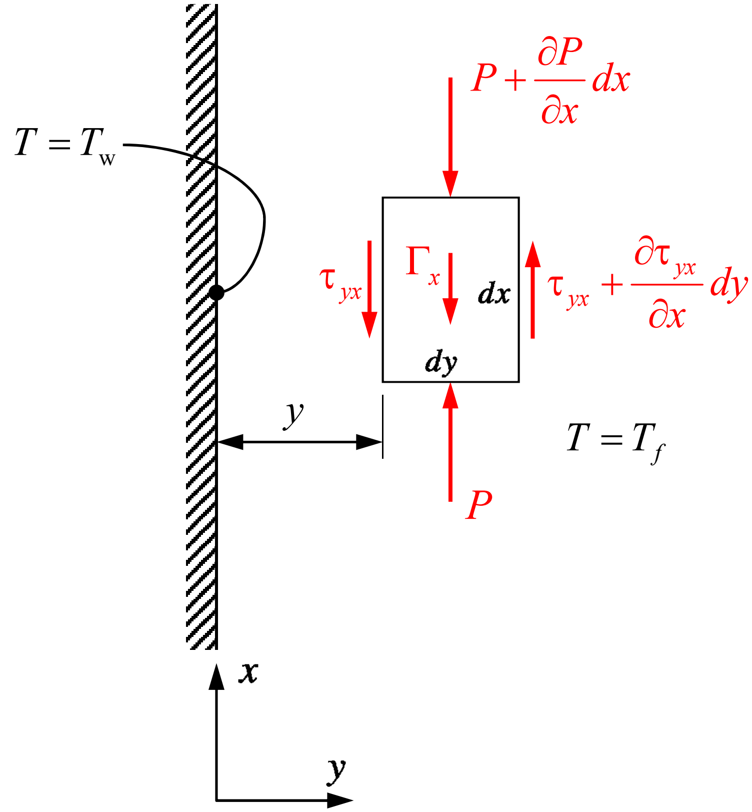
1. Theory/Definition
   a. Phenomena
   b. Transition to turbulent flow
2. Turbulence
   a. Dynamic vs. thermal boundary layer
   b. Full buoyancy effect
3. Modeling tips
   a. Pressure discretization
   b. Time stepping for unsteady simulation
4. Model setup in FLUENT
   a. Reference density and temperature
   b. Boussinesq vs. incompressible ideal gas
5. Examples
   a. Validation case – Tall cavity
   b. Baseline cases
      i. Closed domain with high Rayleigh number
      ii. Plume with high Rayleigh number
   c. Industrial case – Glass tank TC21
6. References
7. Appendix

Natural convection:

Theory & Definitions

Phenomena

- **In natural convection, fluid motion is generated due to density difference (buoyancy) in the fluid caused by temperature gradients.**
- **Body forces**
  - **Typically gravitational**
  - **Centrifugal (rotating machinery)**
  - **Coriolis (atmospheric and oceanic vortical motion)**
- **For this class of problems, flow and energy are strongly coupled.**



Forces acting on a fluid particle in natural convection.

Laminar to Turbulent Transition

- In natural convection, the Reynolds number no longer characterizes the flow.
- With an appropriate reference velocity, it is possible to determine a critical value of the Rayleigh number (Ra_L).
  – Experiments show that the critical Rayleigh number, Ra_c, is around 10^9.
  – The transition zone is quite large as Ra varies between 10^6 and 10^{10}.

Buoyancy force

$$Ra_L = Gr_L\,Pr = \frac{\boxed{\beta\, g\, L^3\, \Delta T}}{\nu\,\alpha}$$

Kinematic viscosity
Thermal diffusivity

wher
e

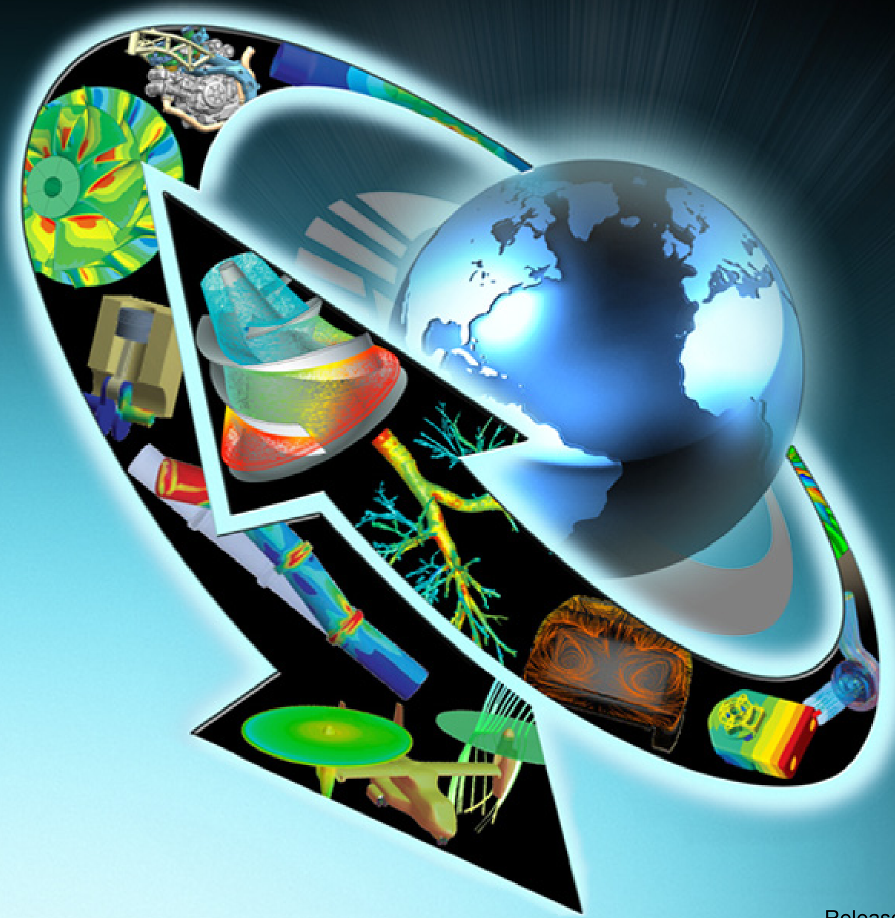
$$Pr = \frac{\nu}{\alpha}$$

Kinemati
c
viscosity

Thermal
diffusivit
y

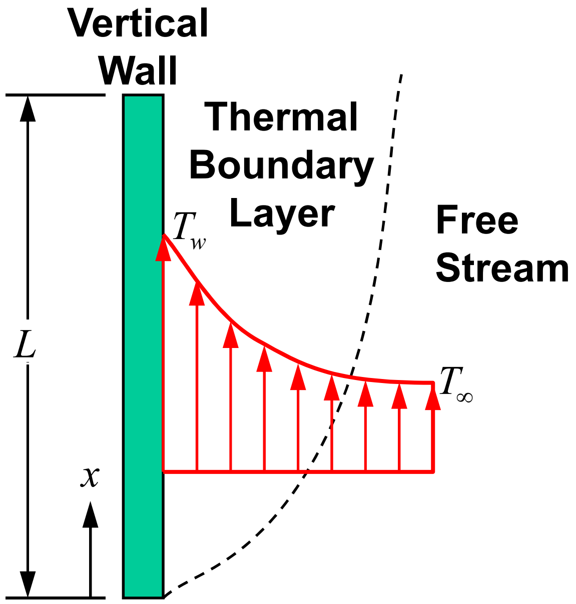
Customer Training Material

Turbulent Boundary Layers



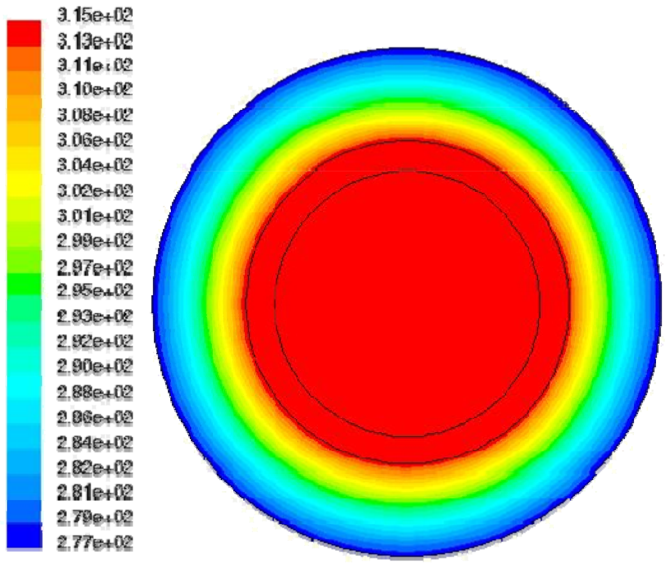
Boundary Layers

- **Impact on numerical modeling for turbulent flow**
  - **Energy and momentum equations are strongly coupled.**
    - **It is recommended to construct the mesh such that $y^+ < 1$ in order to correctly resolve both the momentum AND thermal viscous sublayers.**
    - **This is straightforward for Pr ~ 1 or Pr < 1.**
    - **When Pr > 1, the thermal sublayer is much thinner than the momentum viscous sublayer.**
    - **This behavior is relatively insensitive to grid resolution, provided that the momentum boundary layer structure is accurate ($y^+ \leq 1$ for the first cell layer and at least 10 cells between $1 < y^+ < 30$).**

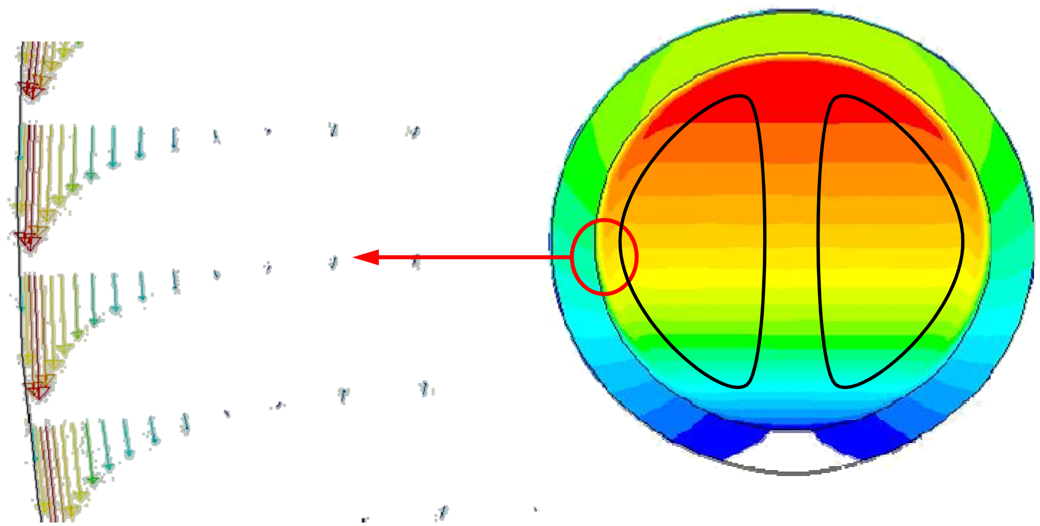


Boundary Layer Grid Generation

- **Start with a 2D test case – This is a good way to confirm what are the characteristic integration time steps and mesh size required for the desired physics.**



Initial temperature (K)

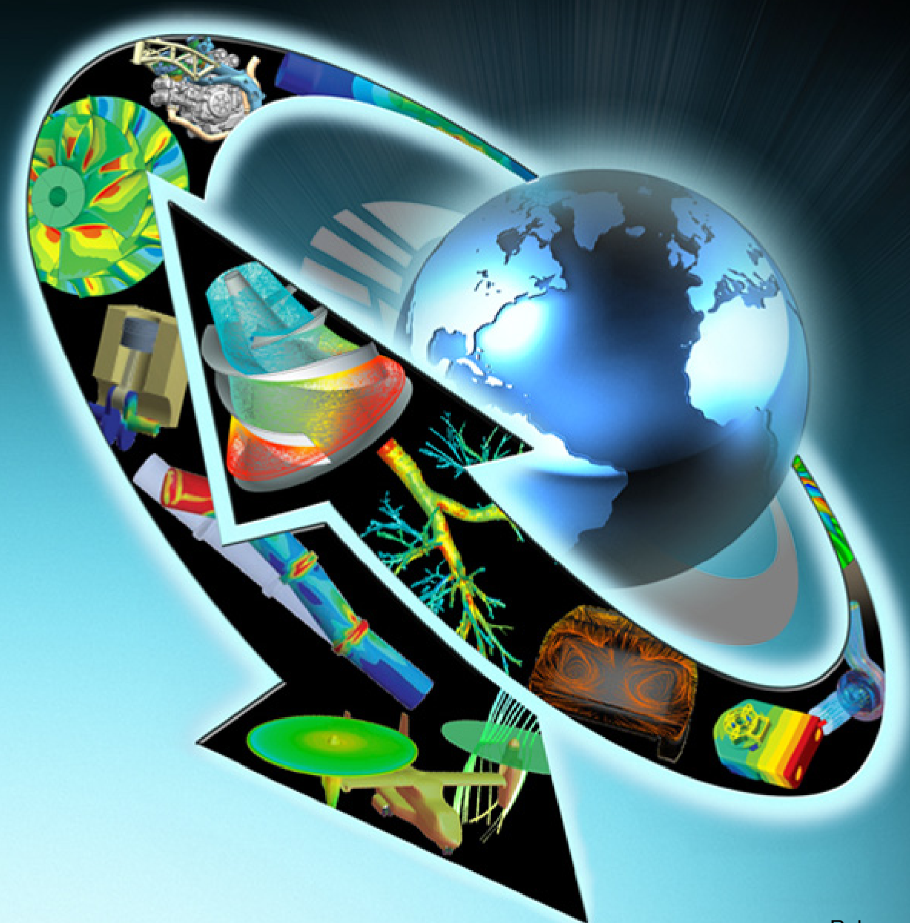


Velocity field **Temperature field**

Expected flow pattern during cool down in pipe cross section due to buoyancy forces

Turbulent Flow Heat Transfer

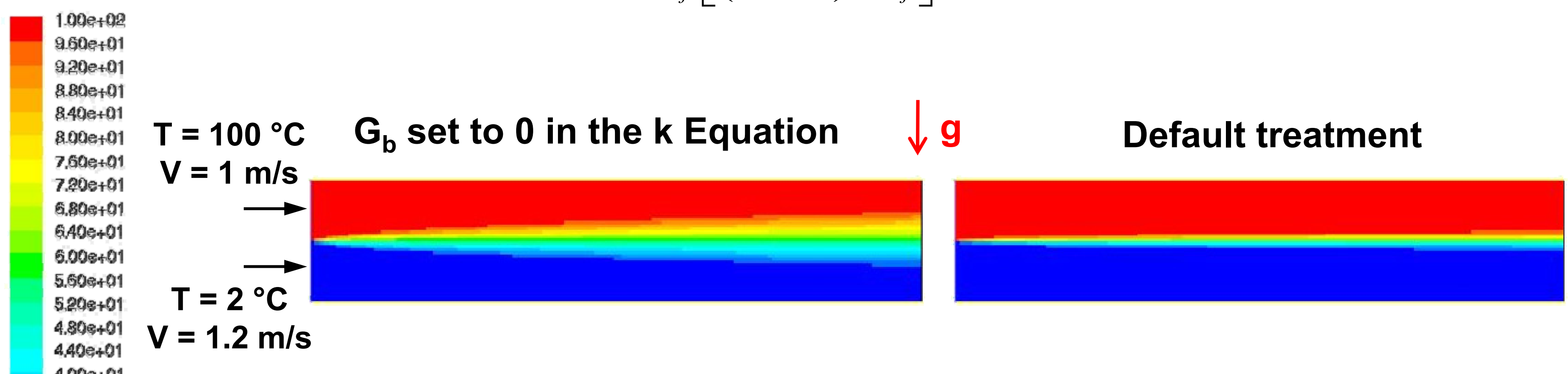
The Full Buoyancy Effects Option



Turbulence Generation due to Buoyancy

- **The importance of the buoyancy term (G_b) can be seen in a mixing layer example using the standard k–ε turbulence model.**

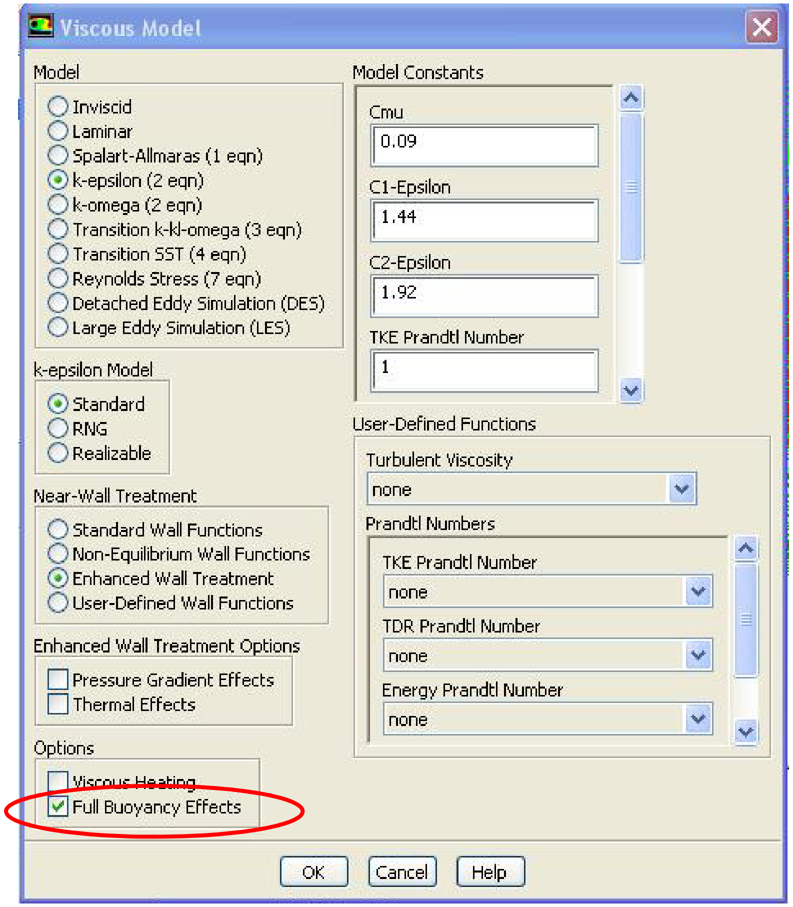
$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho u_i k)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k$$



G_b is a sink term for stable stratification. Turbulent mixing decreases

The *Full Buoyancy Effects* Option

- To include buoyancy effects on ε, you must enable the Full Buoyancy Effects option under Options in the Viscous Model panel.
- This option is available for the three k–ε models (SKE, RKE, RNG) and for the Reynolds stress model (RSM).
- Available for k–ω models via UDF (shown in the Appendix).



Customer Training Material

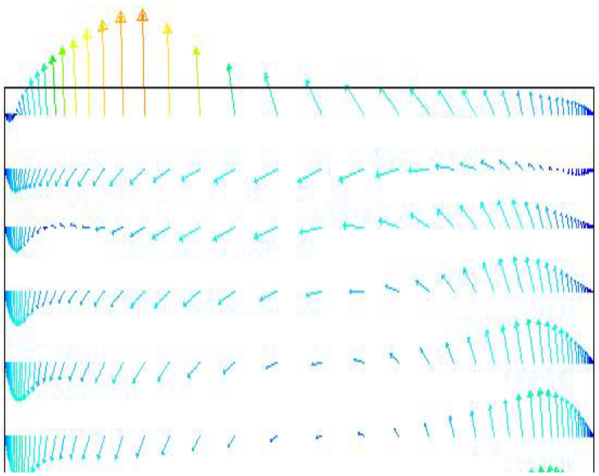
Tips and Tricks

Numerical Discretization
Unsteady Simulations

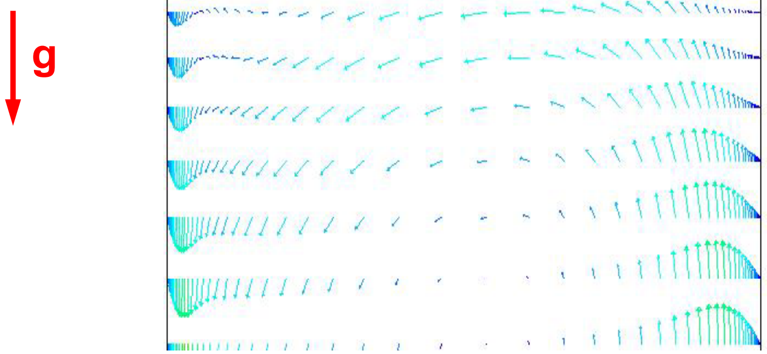
Release 13.0
December 2010

Numerical Discretization

- **Pressure interpolation scheme**
  - **Recommended to use either Body Force Weighted or PRESTO!**
  - **Standard pressure discretization (the default) can give rise to incorrect velocity near the wall.**



Standard
(Non-physical velocity near wall)



PRESTO! or Body Force Weighted
(Correct near-wall velocity)

Unsteady Simulations

- **Estimating the time step size for unsteady simulations:**
  - **Estimate the time constant from**

$$\tau = \frac{L}{U} \approx \frac{L^2}{\alpha\sqrt{\mathrm{Ra}\,\mathrm{Pr}}} = \frac{L}{\sqrt{\beta\, g\, L\, \Delta T}}$$

  - **Use a time step of Δt = τ / 4.**

- **For conjugate heat transfer problems where you are only interested in the steady solution, the density and heat capacity must be reduced (by factor of 1000 for instance) for the solid material to neglect the thermal inertia of the unsteady term in the energy equation.**

Reference Density

- **Momentum equation along the direction of gravity (z in this case)**

$$\frac{\partial(\rho W)}{\partial t}+\nabla\cdot(\rho \mathbf{U} W)=\mu\nabla^2 W-\frac{\partial P}{\partial z}+\rho g$$

- **In FLUENT, a variable change is done for the pressure field since gravity is enabled.**

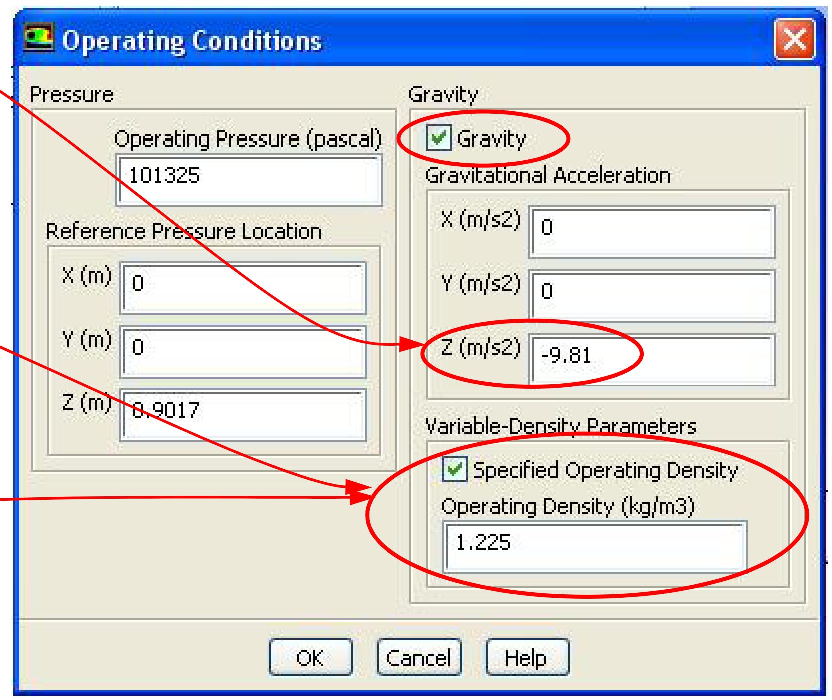
$$P' = P-\rho_0 g z$$

  – **Hydrostatic head is removed from pressure field**

- **Momentum equation becomes**

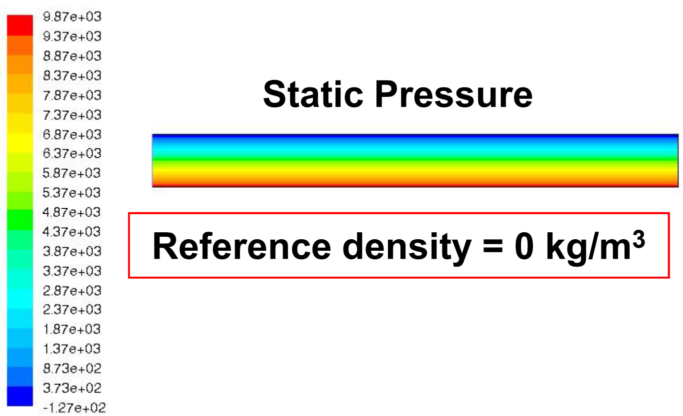
$$\frac{\partial(\rho W)}{\partial t}+\nabla\cdot(\rho \mathbf{U} W)=\mu\nabla^2 W-\frac{\partial P'}{\partial z}+(\rho-\rho_0)g$$

where P' is the static pressure used by FLUENT for boundary conditions and post-processing. This avoids round off error and simplifies the setup of pressure boundary conditions.

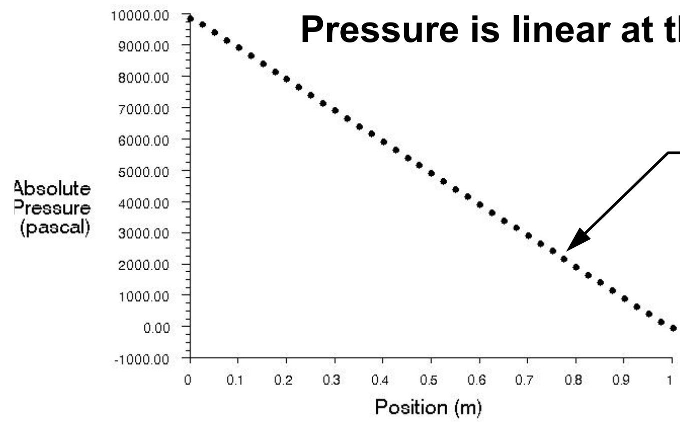


Reference Density

- **Example – Pipe flow with water.**

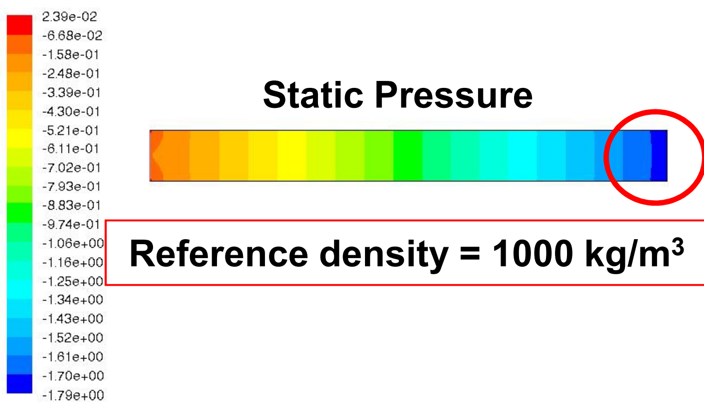


Reference density = 0 kg/m^3



Pressure is linear at the outlet.

If pressure outlet is used instead of outflow, the static pressure would have been specified with a profile (UDF or File)



Reference density = 1000 kg/m^3

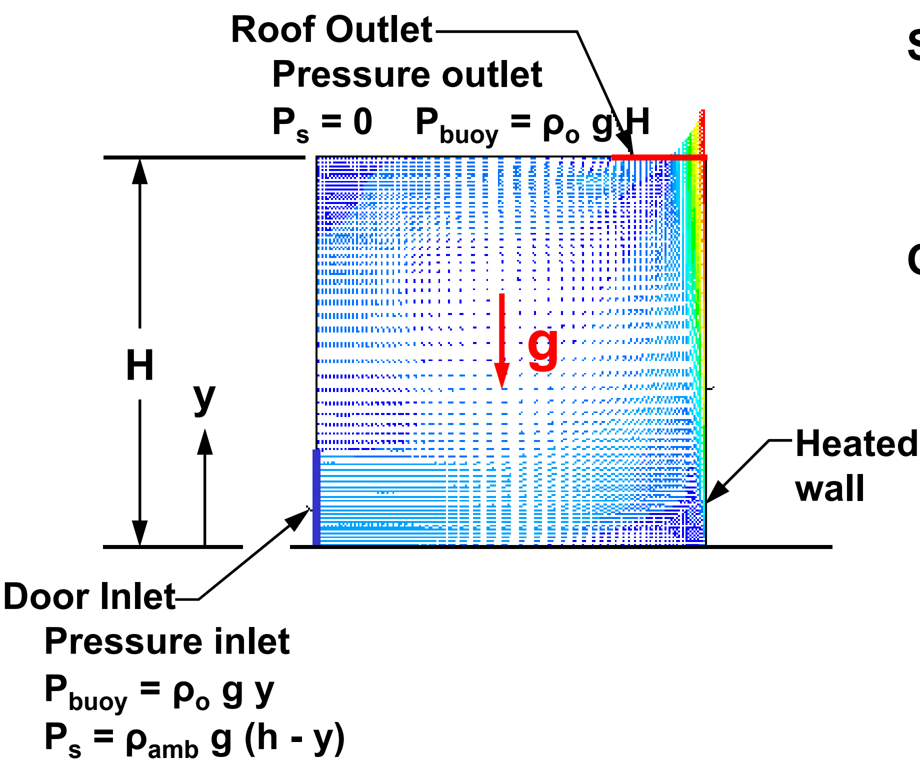
The static pressure is nearly uniform at the outlet.

Pressure outlet with constant gauge static pressure of 0 is correct.

<u>Note:</u> The above pressure field can be reconstructed using custom field functions

Selecting the Reference Density

- **Example – Door and roof vents on a building with heated wall**
  - **The roof static pressure is set to 0 while the door static pressure must be given a hydrostatic head profile based on the height of the building.**



So, the correct pressure BCs are:

$$\left(P_s'\right)_{\mathrm{top}} = \rho_0\, g\, H$$

$$\left(P_s'\right)_{\mathrm{bot}} = \rho_0\, g\, y + \rho_{\mathrm{amb}}\, g\left(H - y\right)$$

Or, equivalently,

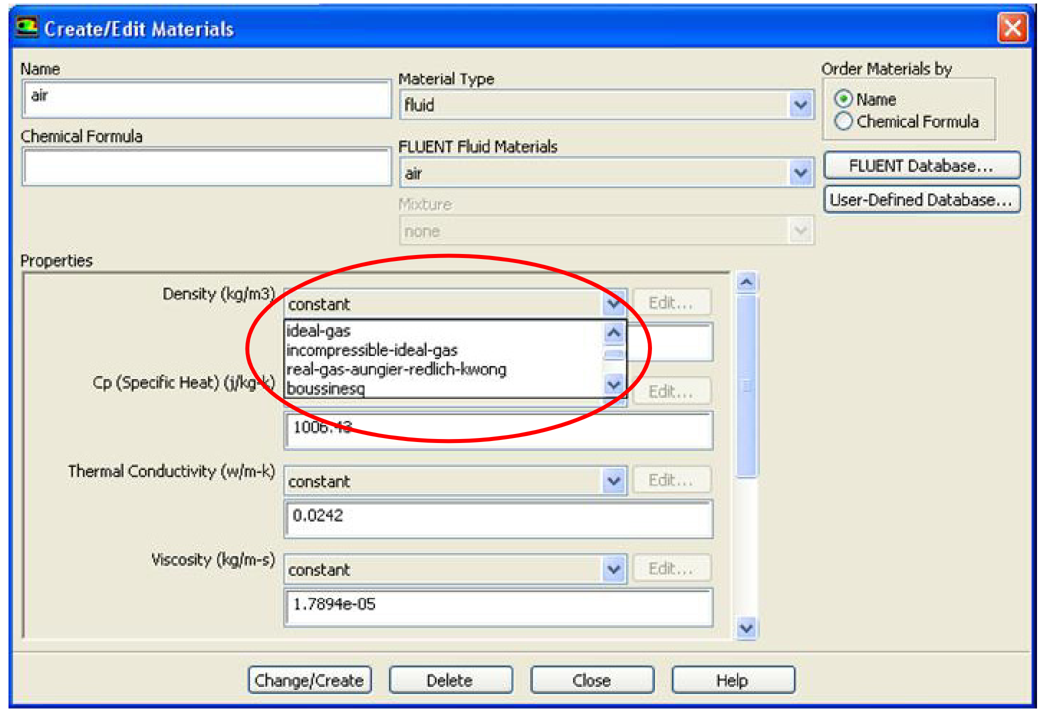
$$\left(P_s'\right)_{\mathrm{top}} = 0$$

$$\left(P_s'\right)_{\mathrm{bot}} = \left(\rho_{\mathrm{amb}} - \rho_0\right) g \left(H - y\right)$$

Note: In this case, if you can set the reference density equal to the external ambient density then the hydrostatic component can be ignored:

Density Methods

- **Fluid density can be set up as a function of temperature using a number of different methods.**
  - **Ideal gas**
  - **Incompressible ideal gas**
  - **Boussinesq**
  - **Piecewise linear**
  - **Piecewise polynomial**
  - **Polynomial**
  - **User-defined**



- **In FLUENT, the body force is always calculated as**

$$(\rho - \rho_0)g$$

  - **If density is constant, this term vanishes.**

- **In natural convection cases, this is the driving mechanism for fluid motion.**

The Boussinesq Approximation

- **Boussinesq model assumes the fluid density is constant in all terms of the momentum equation except the body force term.**

Constant (operating) density

$$\frac{\partial(\rho_0 W)}{\partial t} + \nabla \cdot (\rho_0 \mathbf{U} W) = \mu \nabla^2 W - \frac{\partial P'}{\partial z} + (\rho - \rho_0) g$$

Variable (local) density

- **In the body force term, the fluid density is linearized.**

$$(\rho - \rho_0) g = -\rho_0 \beta (T - T_0) g$$

  - **For many natural convection problems, this treatment provides faster convergence than other temperature-dependent density descriptions.**
  - **The assumption of constant density reduces nonlinear nature of the governing equations.**

- **The Boussinesq assumption is valid when density variations are small.**
- **Cannot be used with species transport or reacting flows.**

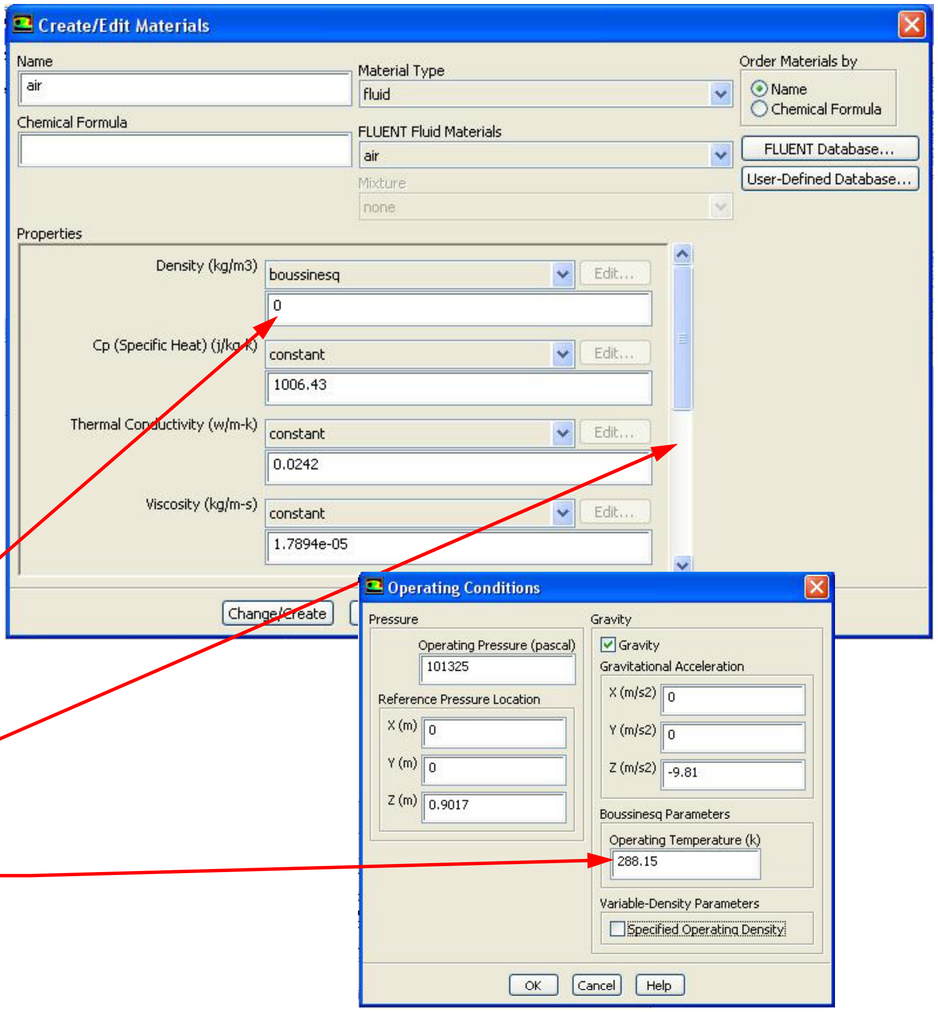
Boussinesq Setup

- **Open the 2 following panels in the following order:**

  Define → Operating Conditions…

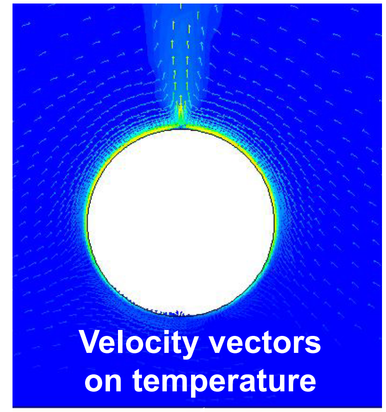
  Define → Materials…

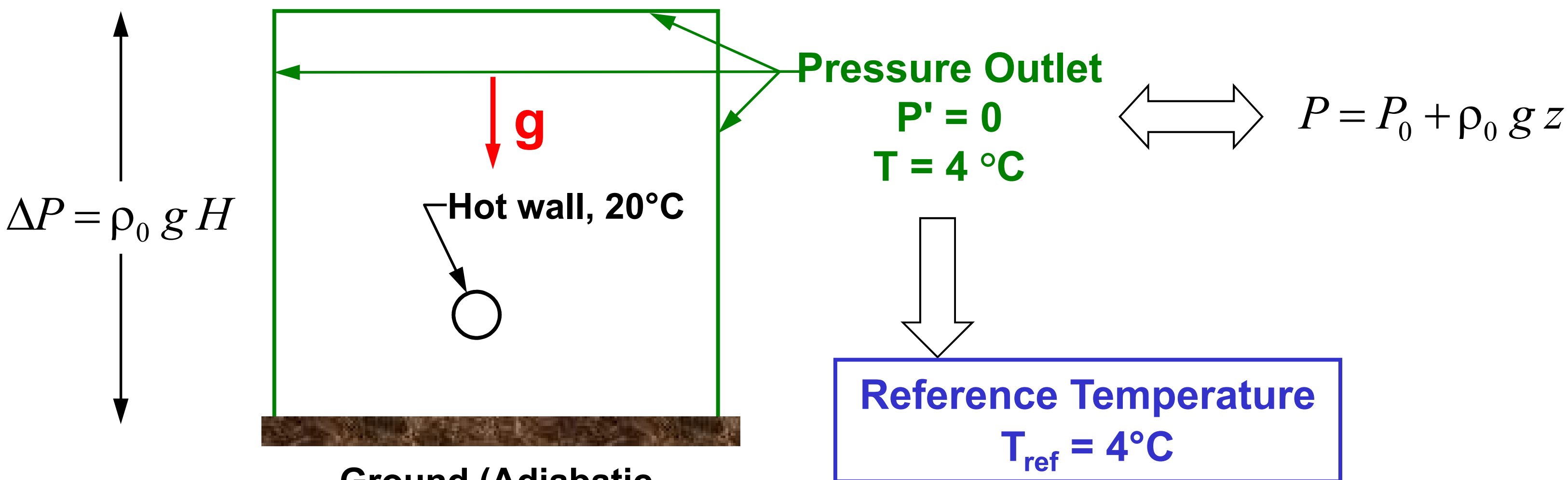
- **Define density model**
  - **In the Materials panel, select Boussinesq as the density method and assign a constant value, ρ_0.**
  - **Set the Thermal Expansion Coefficient, β.**
- **Set Operating Temperature, T_0**



Boussinesq Density Model Example

- **Hot pipeline in the sea:**
  - **Open systems:** $T_0 = T_\infty$ and $\rho_0 = \rho_\infty$
- **Thermal expansion coefficient, β:**
  - **For water, β = 0.0002 K^{-1}**
- **Density is around 1000 kg/m^3**



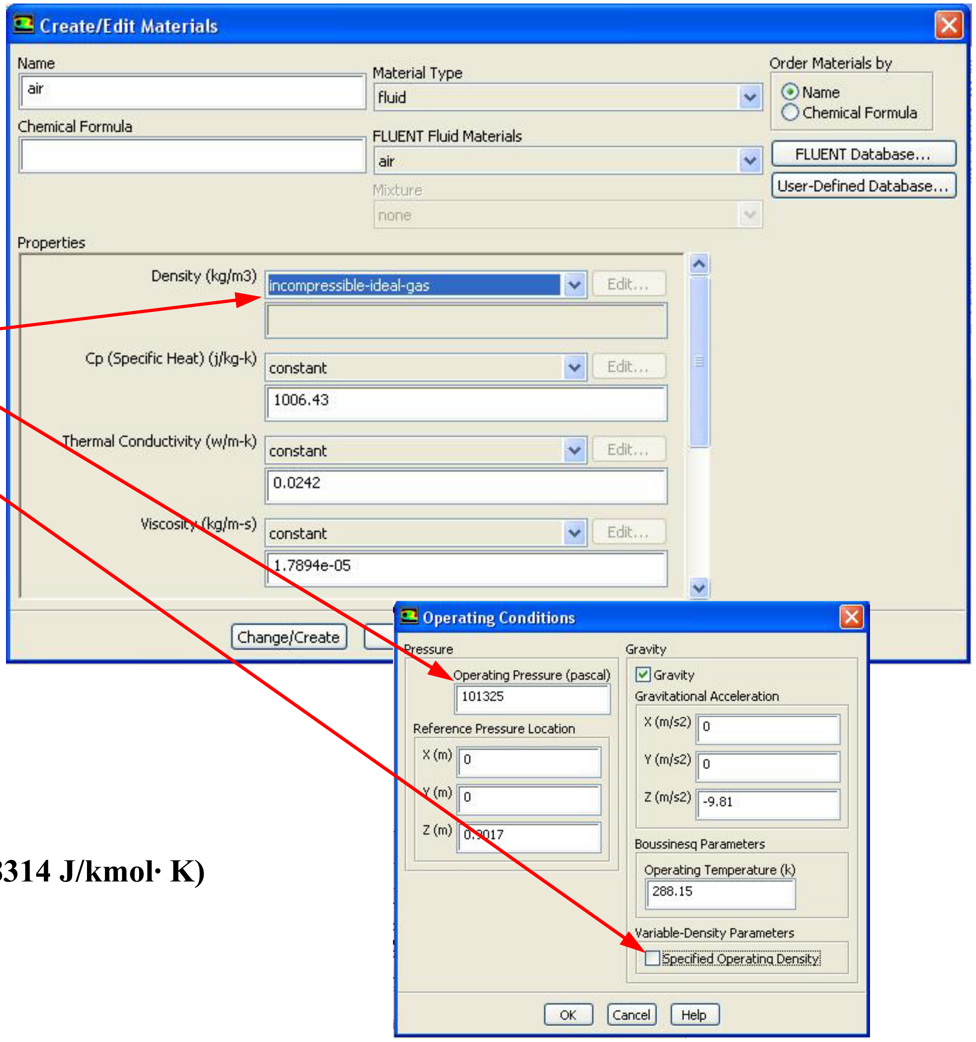


Ground (Adiabatic wall)

If you choose a different reference temperature, a pressure profile needs to be specified at the boundaries

Incompressible Ideal Gas Setup

- **Open the following two panels:**
  - Define → Operating Conditions…
  - Define → Materials…
- **Define density model**
  - **In Materials panel, specify** incompressible-ideal-gas**.**
  - **Specify** Operating Pressure
  - **Set the** Operating density **(if desired). If not specified, FLUENT will calculate ρ_0 from a cell average (default, every iteration).**



$$\rho_0 = \frac{P_{op} M}{R T}$$

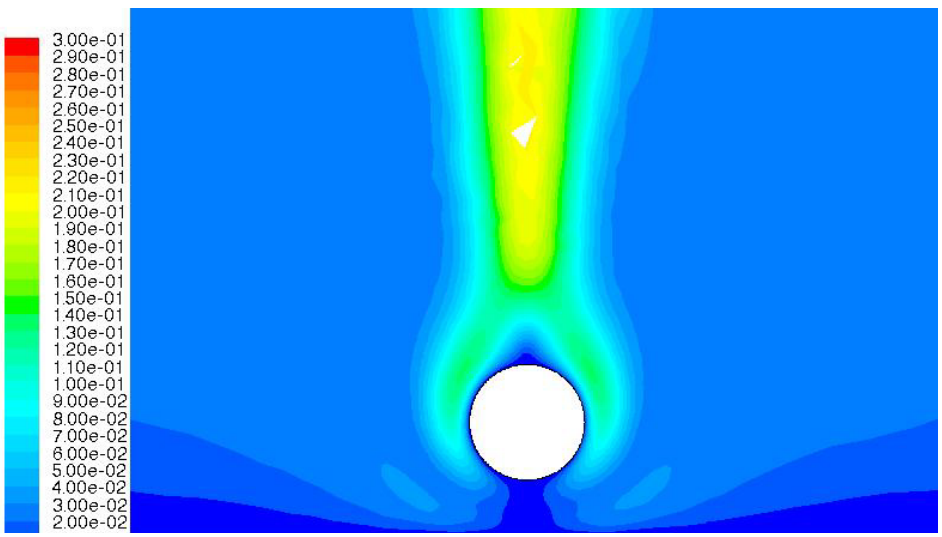
R = Universal gas constant (8314 J/kmol· K)

M = Molecular weight

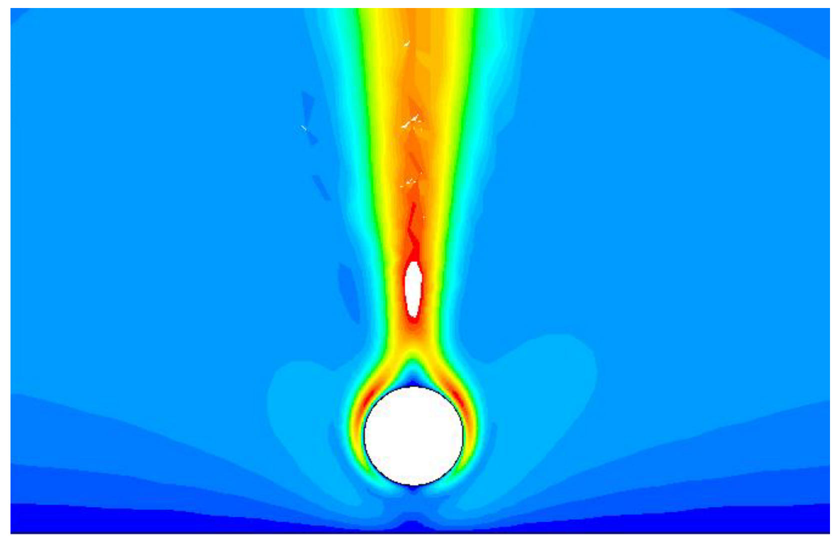
P_{op} = Operating pressure

Boussinesq vs. Incompressible Ideal Gas

- **When the fluid is a gas, either the Boussinesq or the incompressible-ideal-gas option can be selected.**
  - **Boussinesq is not appropriate when the in-situ density variation implies a strong velocity variation (to conserve momentum).**
  - **Recommendation is to use Boussinesq when there is no more than 20% change in fluid density throughout the domain.**
- **Example where Boussinesq is not valid – Hot wire at T = 540 K in ambient air at 270 K.**



Incompressible ideal gas



Boussinesq

Natural Convection in a Closed Domain

- **For a closed domain, mass conservation depends on density treatment.**
  - **This is automatically done when using the Boussinesq approximation (density is assumed to be constant).**
  - **Density only depends on temperature with incompressible ideal gas law since the operating pressure is constant (mass is not conserved).**

$$(\rho - \rho_0)\, g = -\rho_0\, \beta\, (T - T_0)\, g$$

- **In reality, pressure in the domain changes in such a way that mass is conserved. Numerically,**
  - **Density is a function of local pressure (ideal gas law).**
  - **Floating operating pressure (UDF required).**

Natural Convection in a Closed Domain

- For steady solver
  - Initial condition is merely a guess for the final solution.
  - Final solution does not necessarily correspond to any particular initial mass.
  - No EOS is imposed in order to conserve total mass
    - Boussinesq model must be used.
    - The constant density, ρ_0, properly specifies the mass of the domain

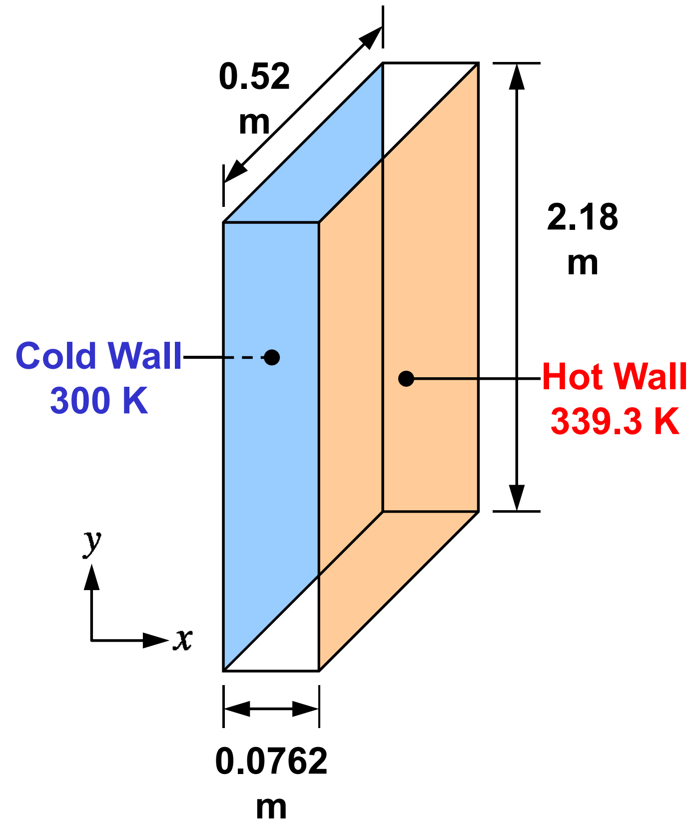
- For unsteady solver
  - Boussinesq model or ideal gas law can be used.
  - Initial conditions prescribe the mass in the domain.

Example Case #1

Natural Convection in a Tall Cavity

Natural Convection in a Tall Cavity

- **Description**
  - **The flow being modeled is the turbulent natural convection of air in a tall heated rectangular cavity.**
  - **Experiments were performed with a 39.3 K temperature differential between the plates. This yields a Rayleigh number (based on plate spacing) of 1.43×10^6.**
  - **From the experiment, the mean temperature and vertical velocity distribution are measured.**



- **Reference**
  - **P.L. Betts and I.H. Bokhari (2000), "Experiments on Turbulent Natural Convection in an Enclosed Tall Cavity,"** ***Int. J. Heat & Fluid Flow*****, Vol. 21, pp. 675-683.**

Natural Convection in a Tall Cavity

- **Modeling Strategy**
  - **2D, 4800-cell quad mesh (30×160)**
  - **Steady flow**
  - **Energy equation**
  - **Pressure-based solver, double precision**
- **Turbulence modeling**
  - **Standard and RNG k–ε with enhanced wall treatment**
  - **Standard and SST k–ω with transitional flow option enabled**
- **Operating conditions**
  - **Gravity enabled, 9.81 m/s^2 in negative y-direction.**
  - **Operating conditions: atmospheric pressure, 320 K.**
- **Discretization**
  - PRESTO! **pressure interpolation scheme**
  - SIMPLE **pressure-velocity coupling**
  - 2nd Order Upwind **for momentum, turbulence, and energy equations.**

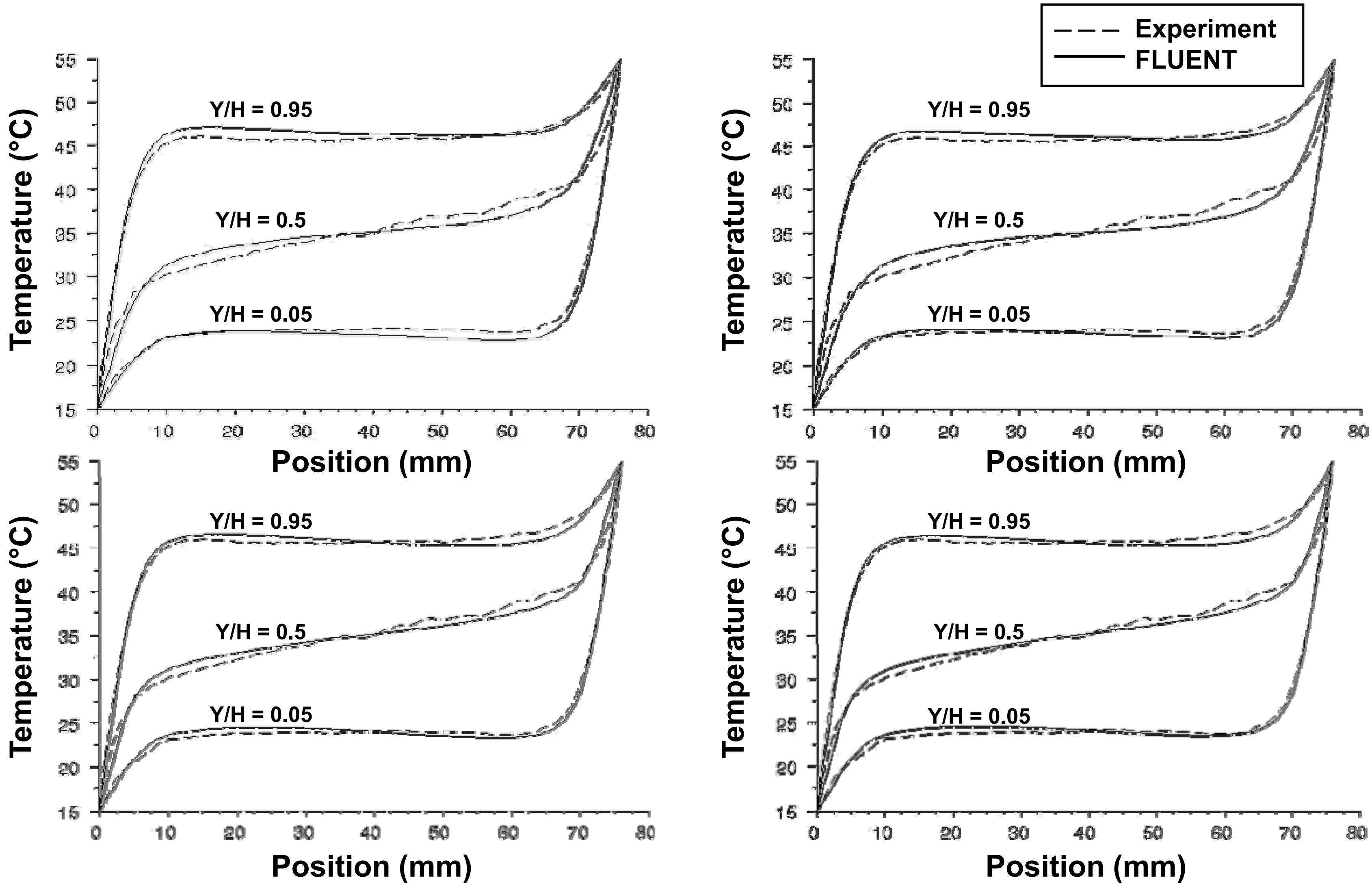
Natural Convection in a Tall Cavity

- **Computational strategy**
  - **Physical properties**

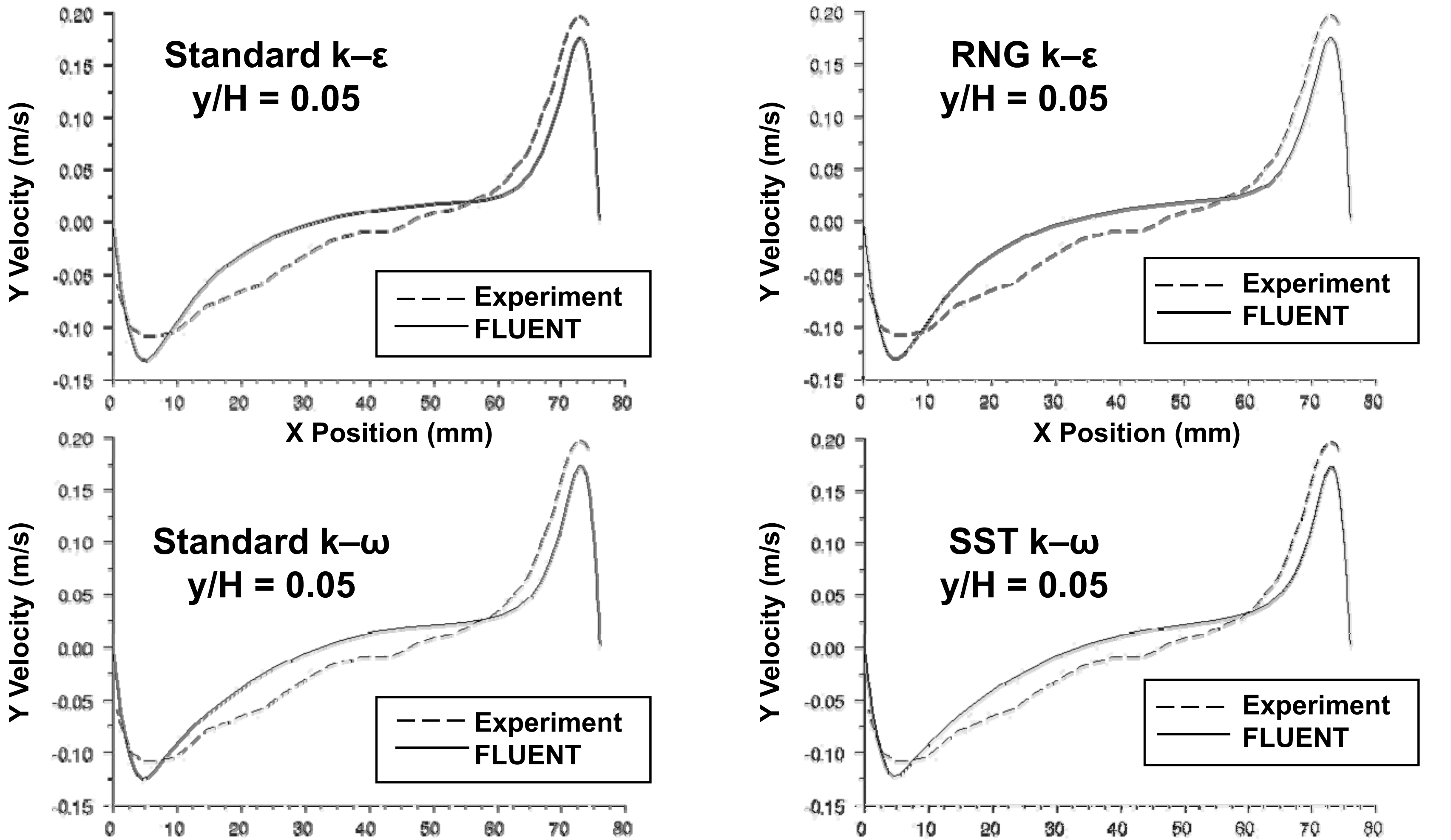
Fluid	Air
Density	Boussinesq, $\rho_0 = 1.22$ kg/m^3
Thermal Expansion Coefficient	$\beta_0 = 0.003125$ K^{-1}
Specific Heat (Cp)	1006.43 J/kg·K
Thermal conductivity	0.0242 W/m·K
Viscosity	1.7894×10^{-5} kg/m·s

- **Boundary Conditions**
  - **Hot Wall set to 328 K**
  - **Cold Wall set to 288 K**

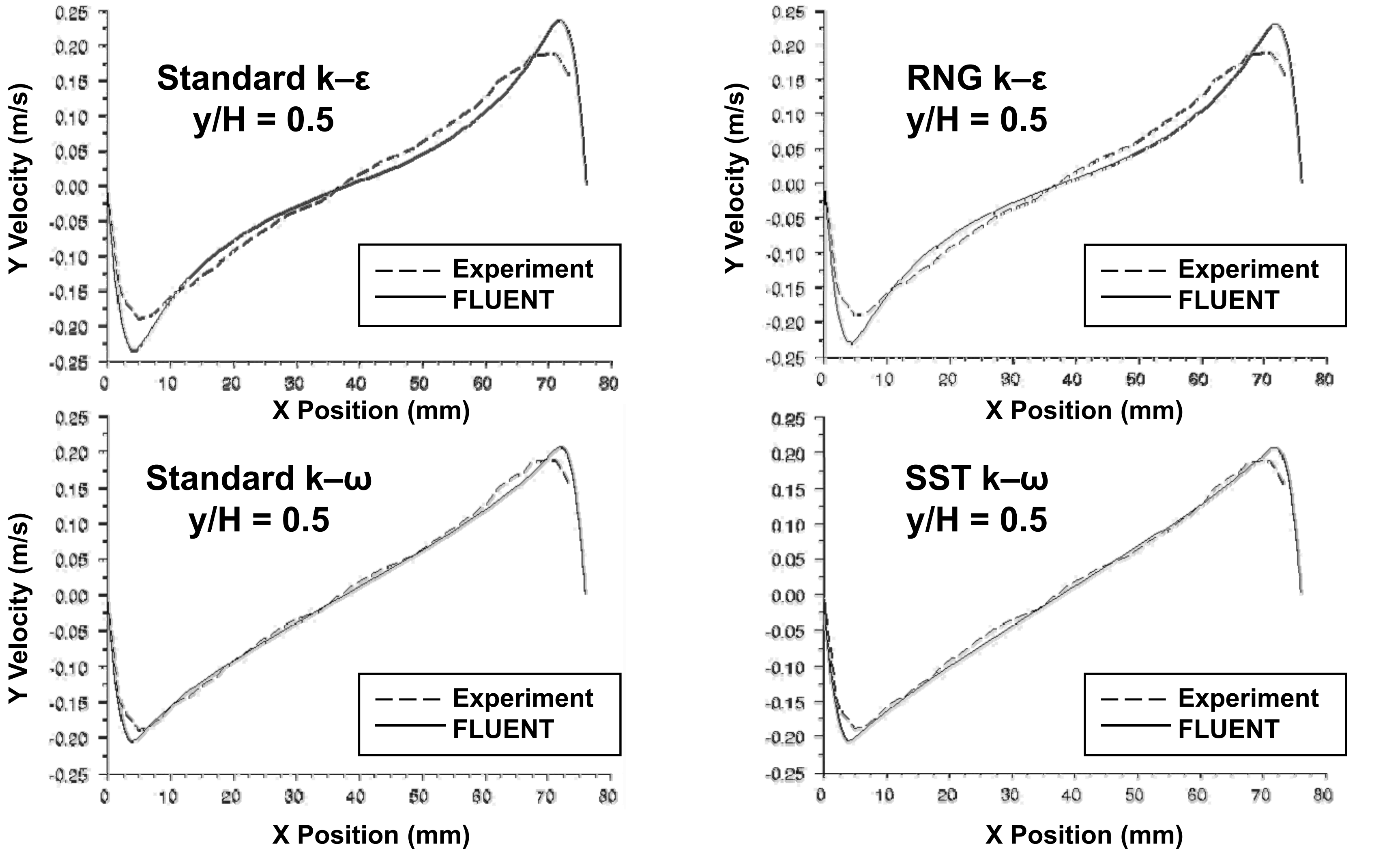
Natural Convection in a Tall Cavity



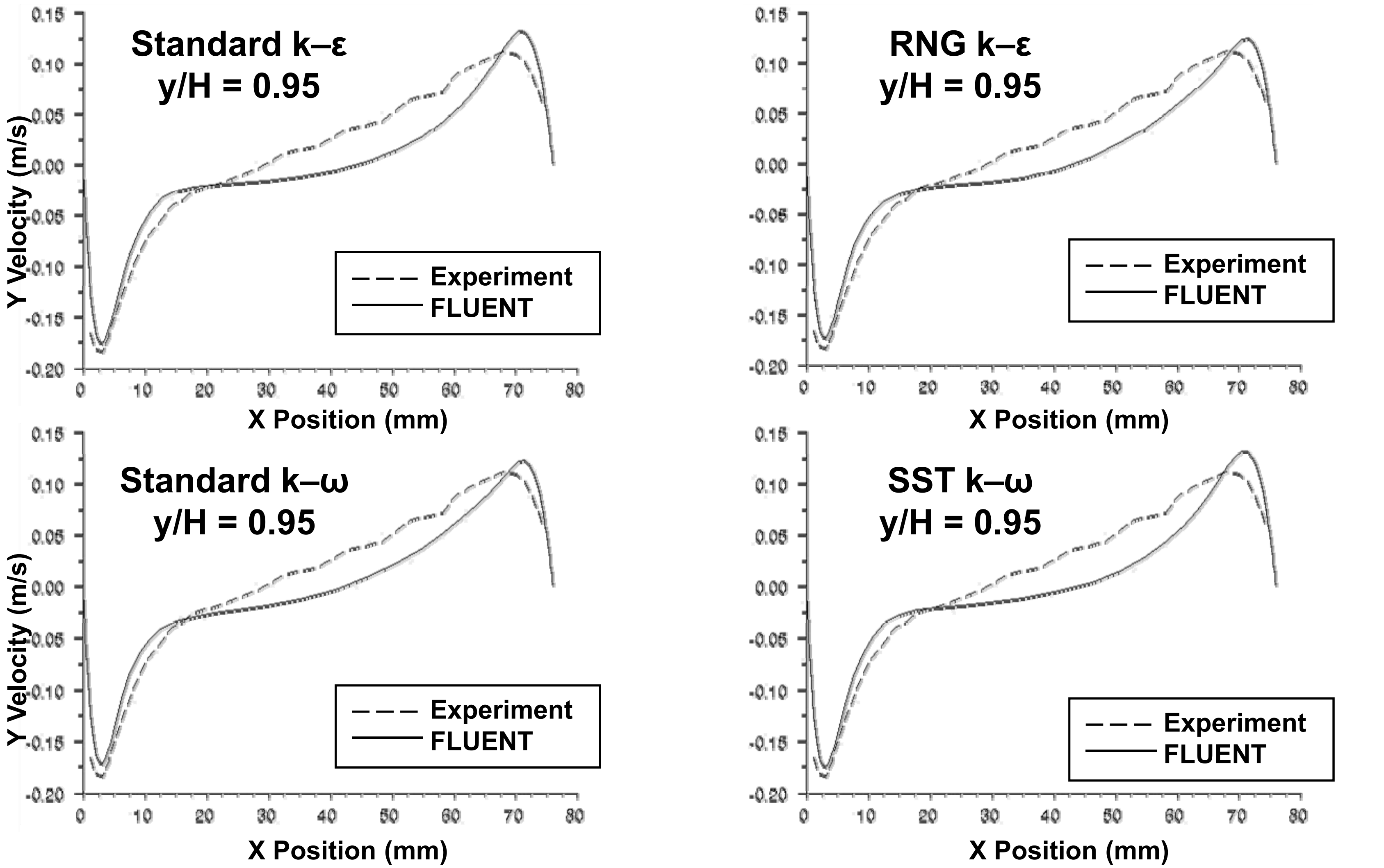
Natural Convection in a Tall Cavity



Natural Convection in a Tall Cavity



Natural Convection in a Tall Cavity

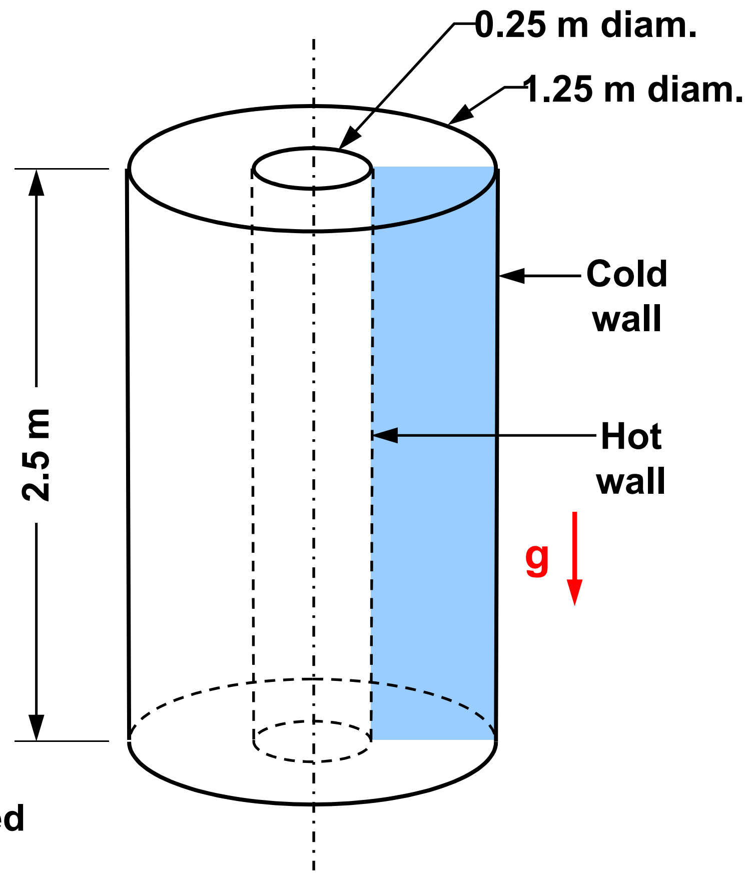


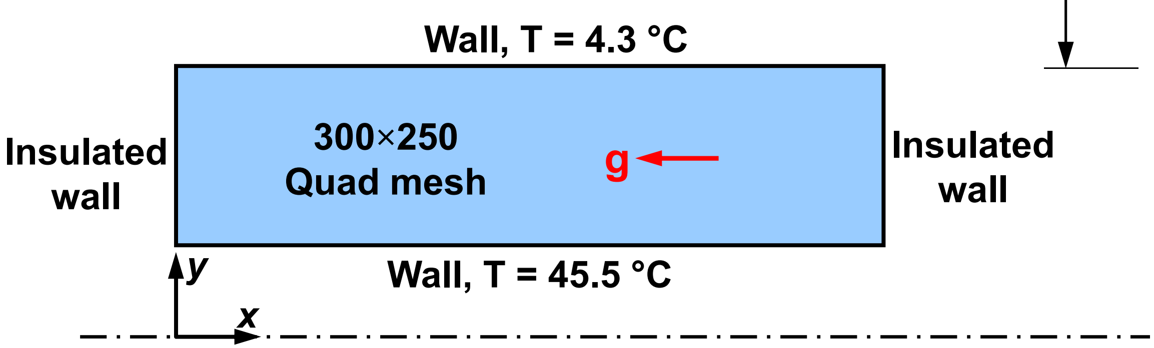
Example Case #2

Subsea Pipe Connection Box

Subsea Connection Box

- **The flow being modeled is turbulent natural convection of water in an annular fluid region (subsea connection box).**
- **Simulations were performed using a temperature differential of 41.2 K (between inner and outer walls).**
  - **Prandtl number is 7.2**
  - **Rayleigh number is around 10^{12}.**
  - **2D axi-symmetric model with gravity.**





Subsea Connection Box

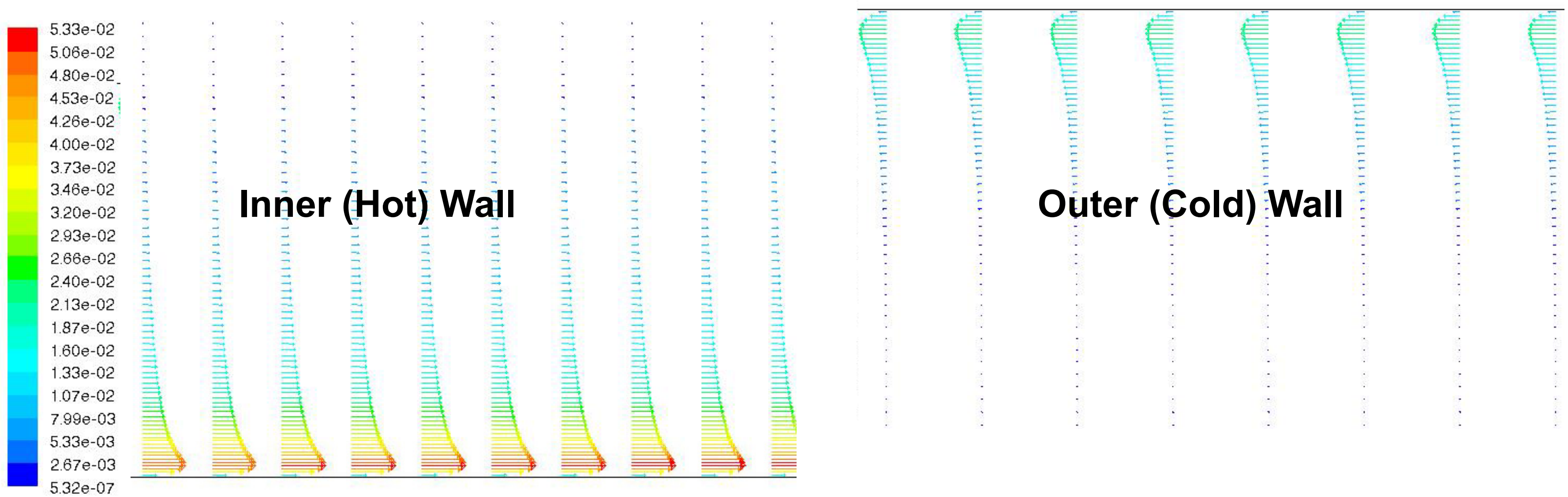
- **Mesh:** **75,000 cell hexahedral mesh ($y^+ < 2$)**
- **Solver:** **FLUENT 6.3 (pressure-based, double precision)**
- **Conditions:** **P_{op} = 101325 Pa; T_{op} = 298 K; g_x = -9.81 m/s^2**
- **Models:** **Standard k–ε with enhanced wall treatment, Energy activated**
- **Solver Controls:** **Default under-relaxation**
  SIMPLE scheme for pressure-velocity coupling
  PRESTO! pressure discretization
  2nd Order Upwind discretization for *u*, *v*, *T*, *k*, ε
- **Fluid Properties:**

Density	Boussinesq; ρ_0 = 1020 kg/m^3
Thermal Expansion Coefficient	β = 0.000294 K^{-1}
Cp (J/kg-K)	4000 J/kg·K
Thermal conductivity	0.5 W/m·K
Viscosity	0.0009 kg/m·s

- **Boundary Conditions**
  - **Hot wall set to 45.5 °C**
  - **Cold wall set to 4.3 °C**

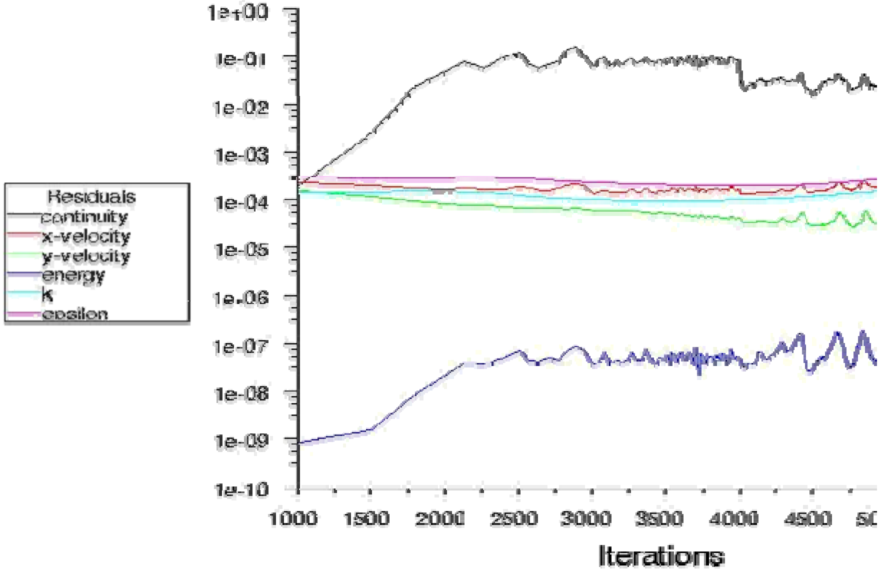
Subsea Connection Box

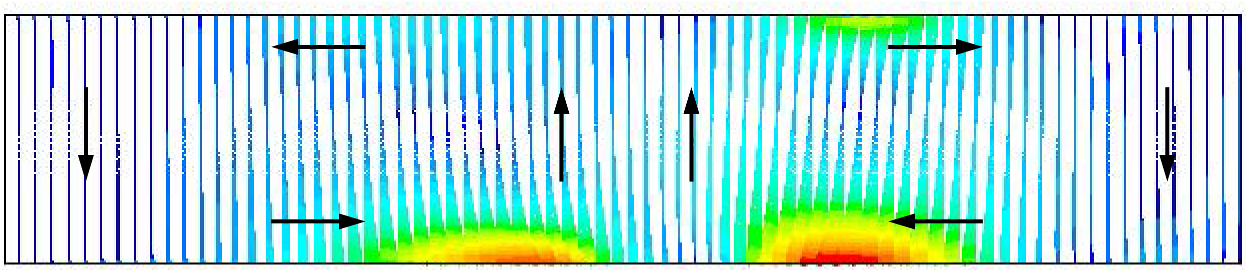
- **Results**
  - **Bottom of the geometry (adjacent to hot wall) – development of a boundary layer on the heated steel pipe.**
  - **Heat transfer from the bottom of the internal pipe to the fluid zone of the connection box.**
  - **Top of the geometry (adjacent to cold wall) – Laminar boundary layer appears near the wall.**



Subsea Connection Box

- **Using the steady solver for this problem is unstable!**
  - **No possibility to converge.**
  - **Large residuals**
  - **Predicted flow field is not physical (see below)**
  - **Heat transfer fluxes not balanced**

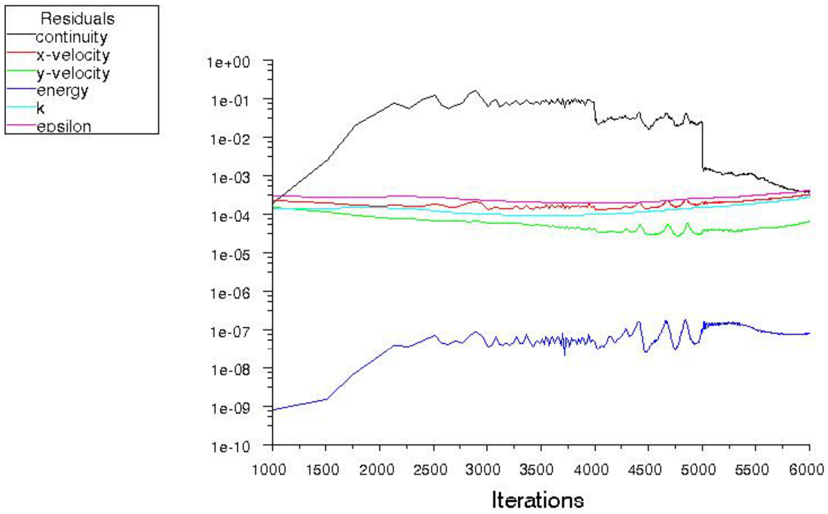


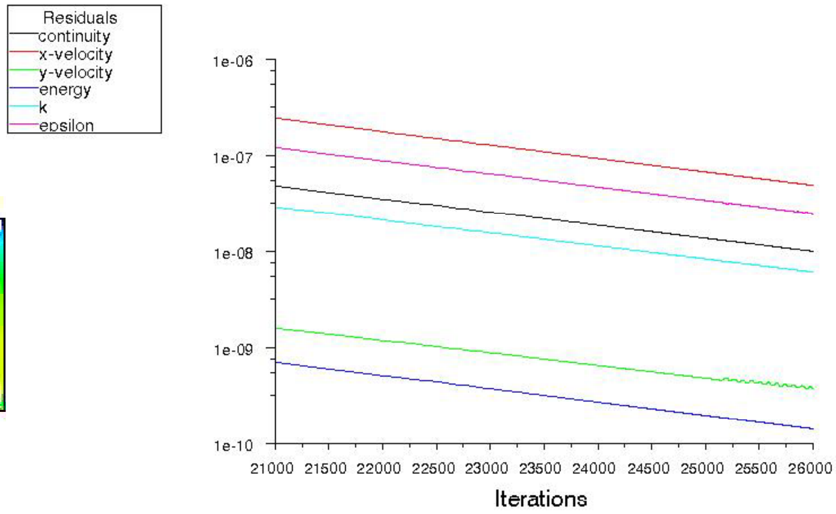


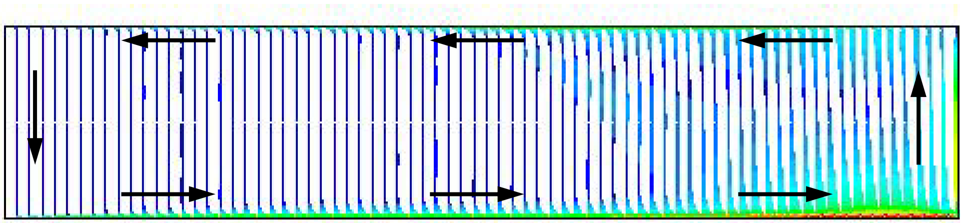
Velocity Vectors

Subsea Connection Box

- **Transient approach**
  - **Convergence achieved**
  - **Low residuals**
  - **Flow physical**
  - **Heat transfer fluxes well balanced**



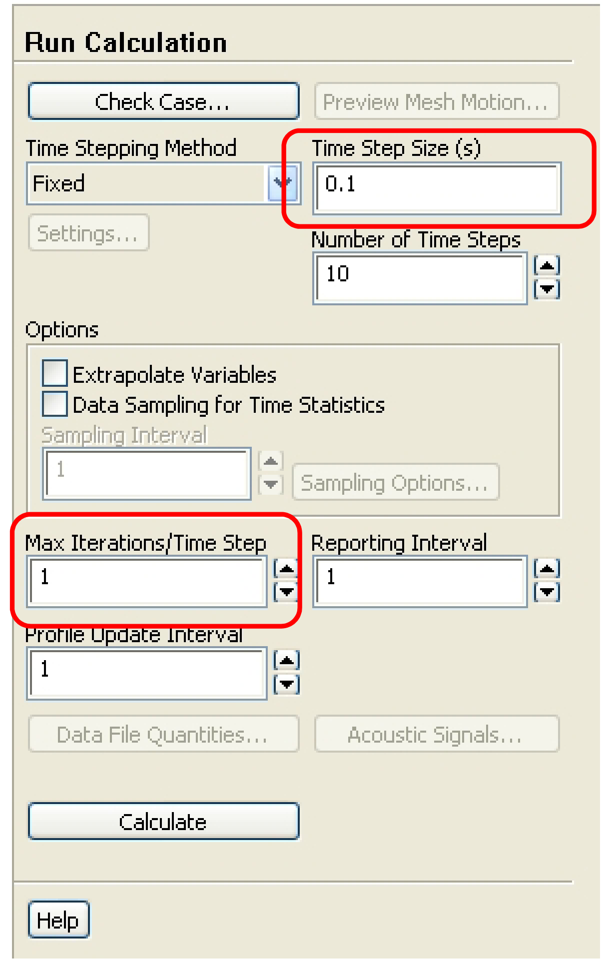




Velocity Vectors

Subsea Connection Box

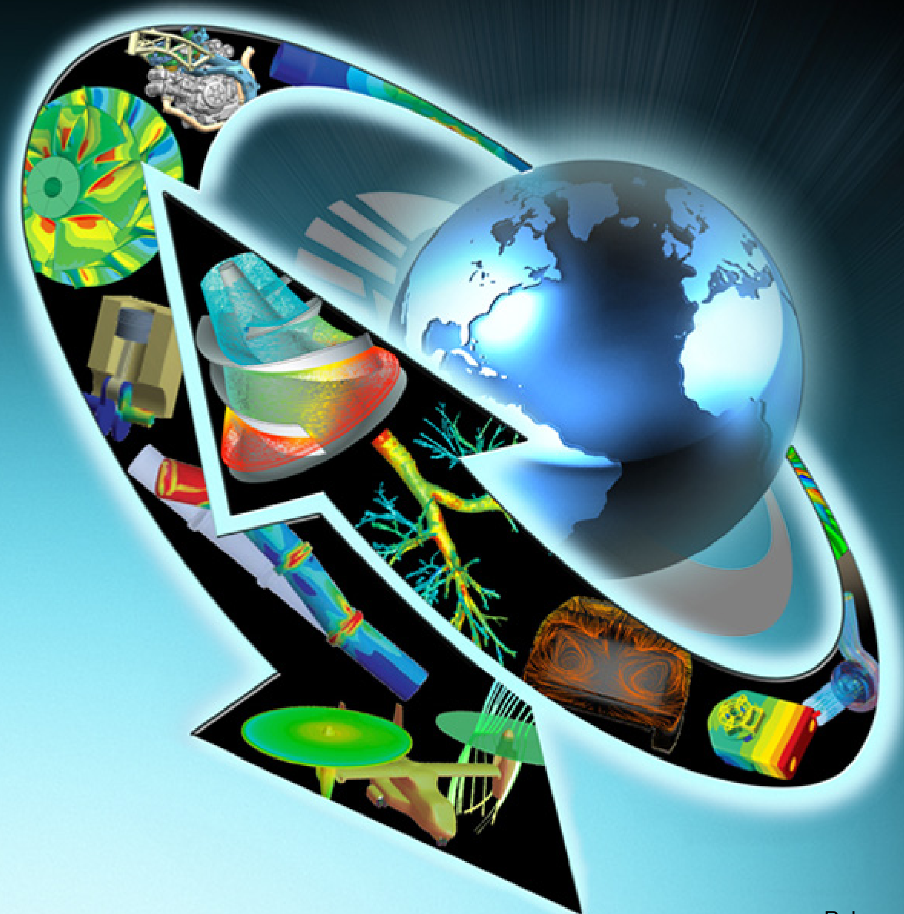
- **Transient term acts like an under-relaxation factor for the coupling between flow and energy equations (we are not interested in the transient nature of the flow).**
- **Transient solution approach**
  - **Start with a large time step size (~10,000 seconds).**
    - **Not realistic but allows information to convect rapidly.**
  - **Perform only one iteration per time step.**
  - **Gradually decrease the time step size to refine/improve the solution.**
    - **Successively decrease by one order of magnitude until Δt ~ 1 second.**
    - **This is close to a realistic time step size for this Rayleigh number.**



Customer Training Material

Example Case #3

High Rayleigh Number Natural Convection Past a Cylinder



Release 13.0
December 2010

High Rayleigh Number Flow Past a Cylinder

- **Computational strategy**
  - **Grid – 6,877 cells**
  - **Solver**
    - **Pressure-based, double precision**
    - **Both steady and unsteady**
  - **Turbulence Models**
    - **RNG k–ε model with enhanced wall treatment**

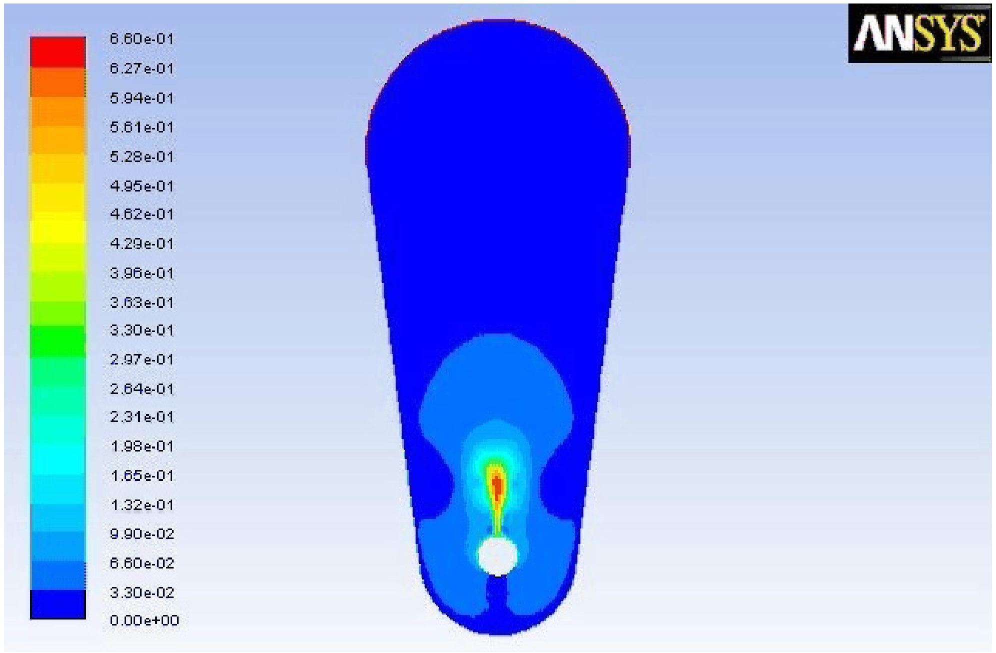
High Rayleigh Number Flow Past a Cylinder

- **The steady approach method described is not applicable in the case of natural convection around an obstacle.**

- **Time step evaluation for unsteady simulation**
  - **Estimate the time constant from**

$$\tau = \frac{L}{U} \approx \frac{L^2}{\alpha\sqrt{\mathrm{Ra\,Pr}}} = \sqrt{\frac{L}{\beta\, g\, \Delta T}}$$

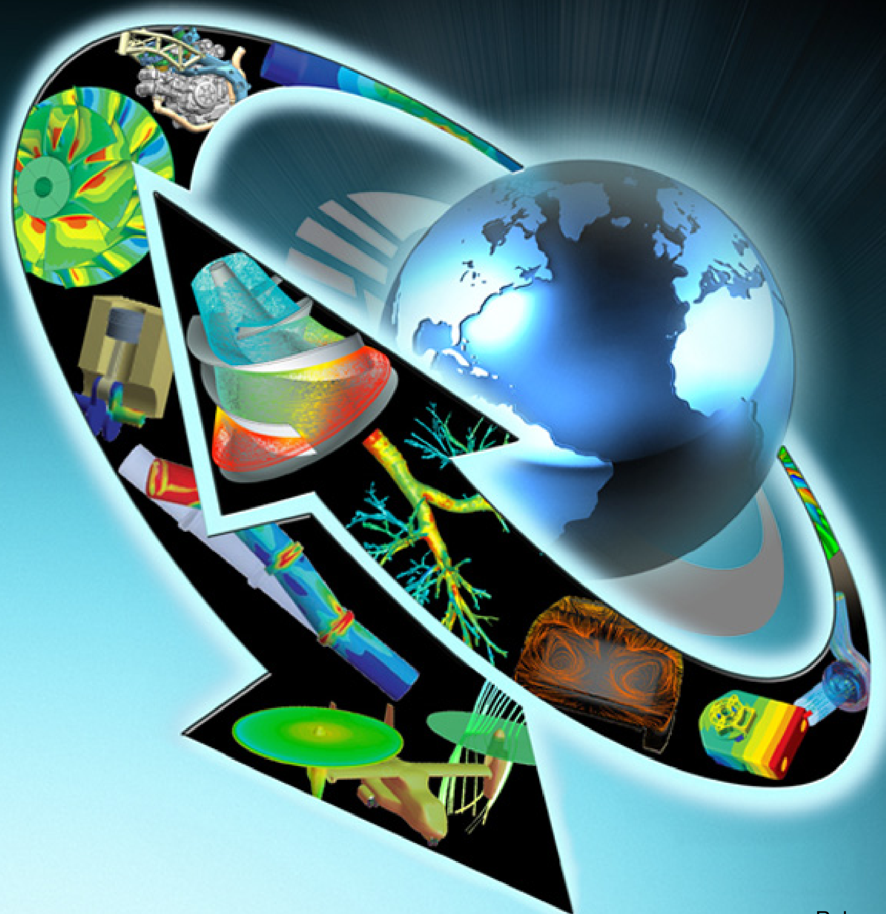
  - **Use a time step of $\Delta t = \tau / 4$**



Customer Training Material

Example Case #4

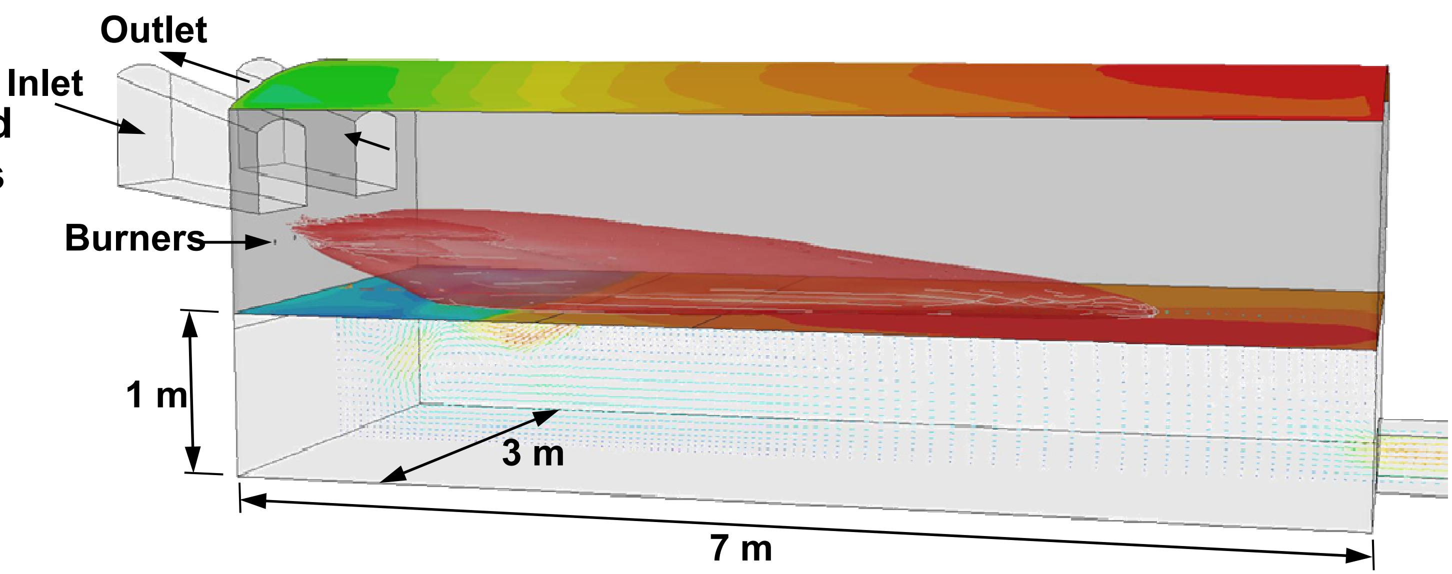
Natural Convection in Glass Processing Furnace



Glass Furnace

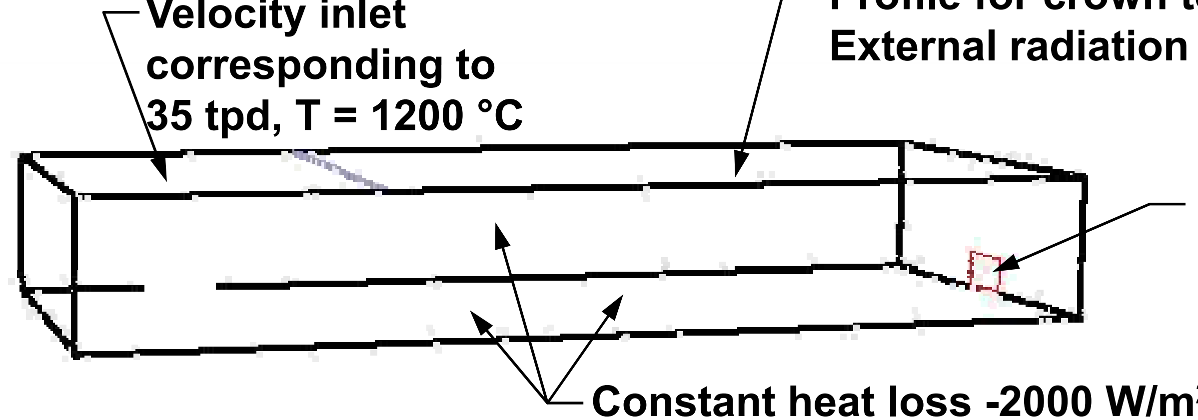
- **Problem description**
  - In the glass furnace application, melted glass is heated by radiating flame and furnace crown. Cold batch (raw material) enters on top of the glass free surface where it reacts and melts.
  - The test case of interest concerns the glass tank. Boundary conditions are simplified. The convection rolls within the tank have a direct impact on glass quality metrics.
- **Reference: TC21 Round Robin Tests**

Raw material is fed on this side and is floating on top of the glass surface



Glass Furnace

- **Boundary Conditions**



- **Computational strategy**
  - **168,000 hexahedral elements**
  - **Solver**
    - **Pressure-based**
    - **Double precision**
    - **Steady**
    - **Energy activated**
    - **Laminar flow**

Glass Furnace

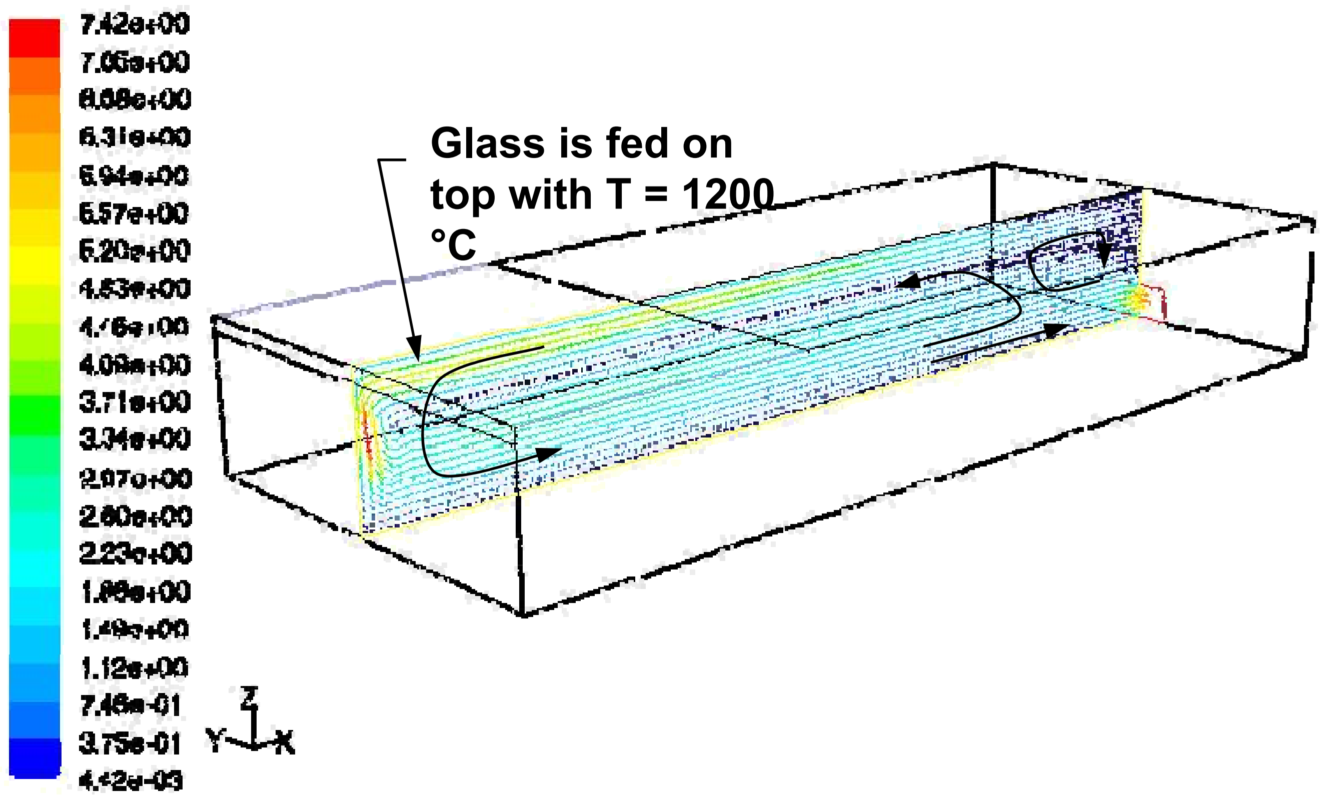
- **Material Properties**

Fluid	Glass
Density	Boussinesq, ρ_0 = 2300 kg/m^3
Thermal Expansion Coefficient	6.1×10^{-5} K^{-1}
Cp	1300 J/(kg·K)
Thermal conductivity	30 W/(m·K)
Viscosity (UDF)	

- **Computational strategy**
  - **Models**
    - **Boussinesq model for density**
  - **Operating Conditions**
    - **Gravity of -9.81 m/s2 in the z-direction.**
    - **Operating temperature input for the Boussinesq model set to 1473 K**
  - **Discretization**
    - **Body force weighted interpolation scheme for pressure**
    - **SIMPLE for pressure velocity coupling**
    - **QUICK for momentum and energy equations**

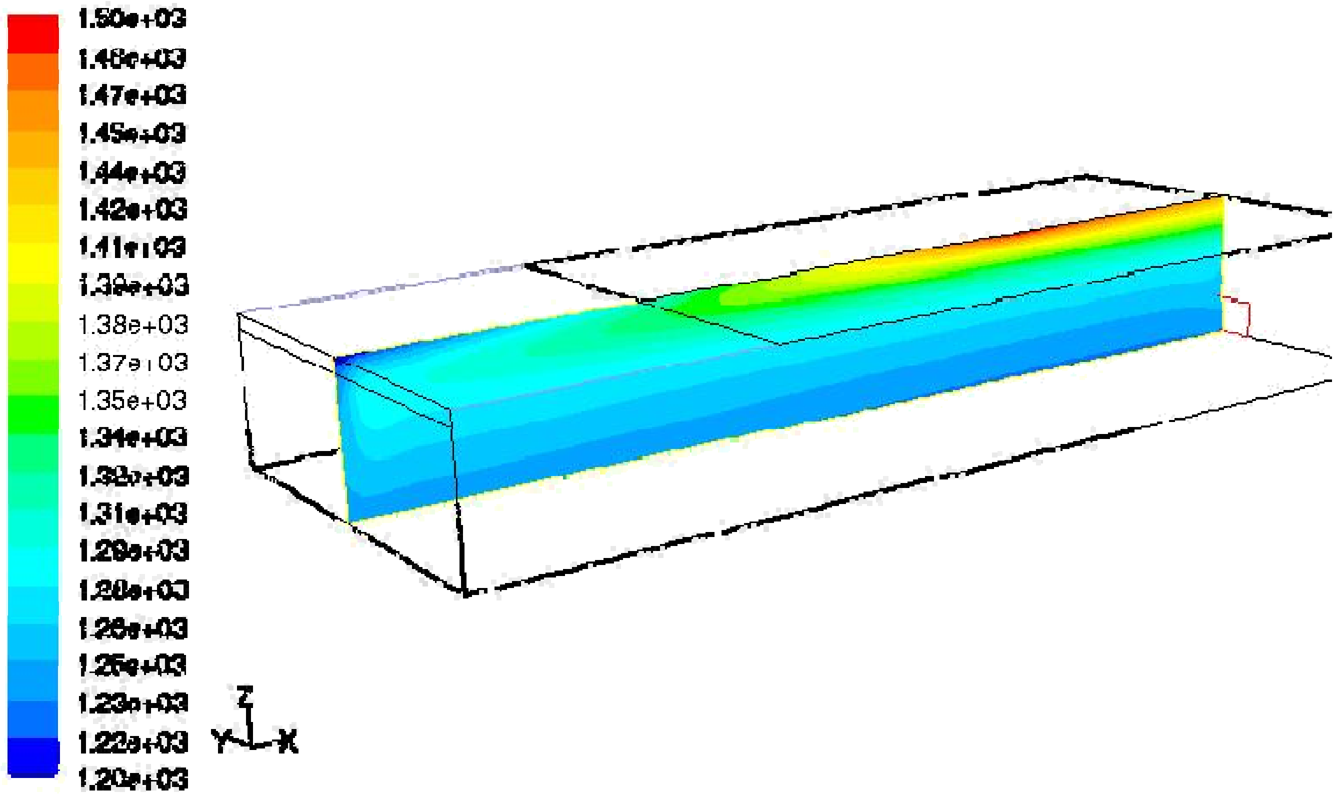
Glass Furnace

- **Results – Convection rolls**



Velocity Magnitude (m/h)

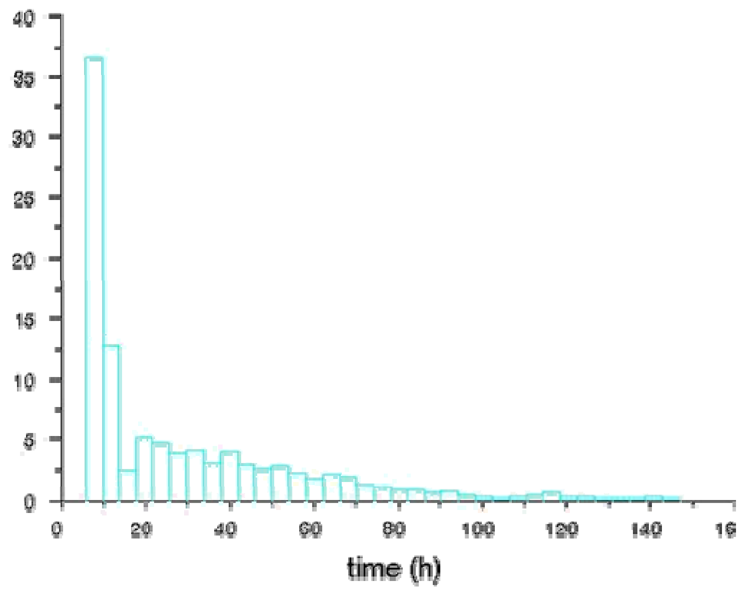
Glass Furnace

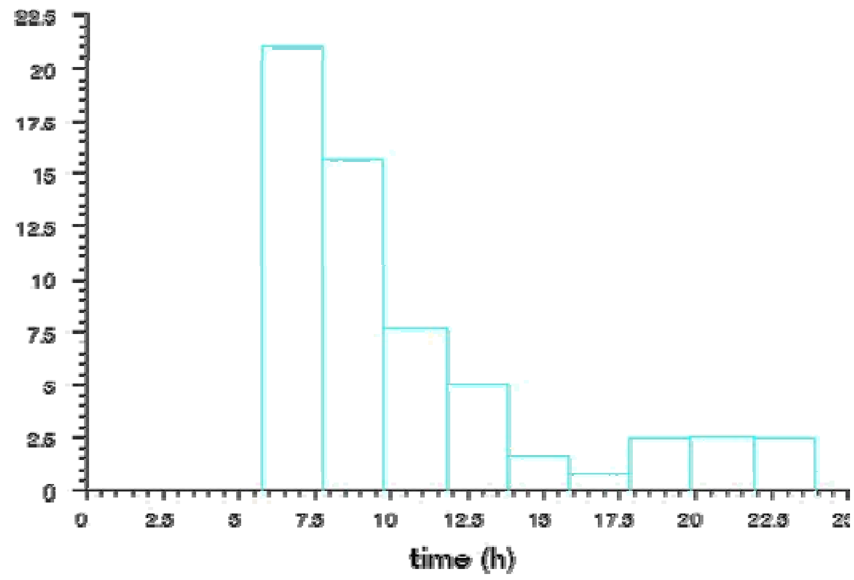


Temperature (ºC)

Glass Furnace

- **Residence time distribution is the expected result of such simulation:**
  - **It is influenced by the intensity of convection rolls.**

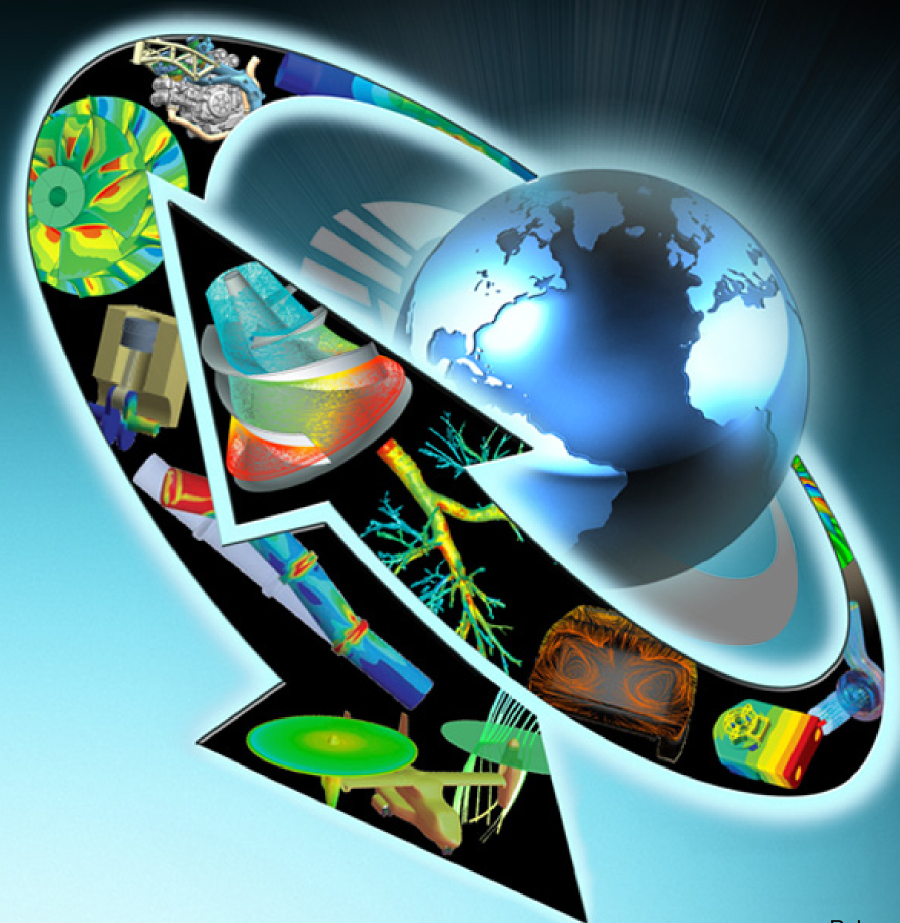




Minimum residence time 5.77 h
Outlet temperature 1243°C

Customer Training Material

Appendix



Lecture 4

Natural Convection

Grashof Number

- **Grashof number (Gr_L)**
  - **Flow regime in natural convection is governed by Grashof number**
  - **Indicates the relative importance of buoyancy forces to viscous (damping) forces**

Gravitational acceleration

Coefficient of thermal expansion

Characteristic length

Maximum temperature differential

$$\beta = \frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_P$$

$$Gr_L = \frac{\beta\, g\, L^3\, \Delta T}{\nu^2}$$

Kinematic viscosity

Back

Richardson Number

- To determine whether the flow is driven by natural convection, forced convection, or both, we examine the Richardson number.
- Richardson number (Ri) represents the relative magnitude of natural convection effects to forced convection effects.

Grashof number

$$\mathrm{Ri} = \frac{Gr}{\mathrm{Re}^2} = \frac{\beta\, g\, L\, \Delta T}{U_0^2}$$

Reynolds number

Bulk velocity

Ri = 1	$\Rightarrow$	**Free and Forced convection effects must be considered**
Ri << 1	$\Rightarrow$	**Free convection effects may be neglected**
Ri >> 1	$\Rightarrow$	**Forced convection effects may be neglected**

Back

Rayleigh number

- **Indicates the relative importance of buoyancy forces to viscous dissipation and diffusion forces.**
- **Ra is the main characteristic of natural convection flows.**
- **Large Rayleigh numbers indicate strong natural convection effects.**
- **Product of Grashof number and Prandtl number**

Buoyancy force

$$\mathrm{Ra}_L = \mathrm{Gr}_L\,\mathrm{Pr} = \frac{\boxed{\beta g L^3 \Delta T}}{\nu\alpha} \quad \textbf{where} \quad Pr = \frac{\nu}{\alpha}$$

Kinematic viscosity

Thermal diffusivity

Kinemati
c
viscosity

Thermal
diffusivit
y

Back

Transition Conduction / Convection

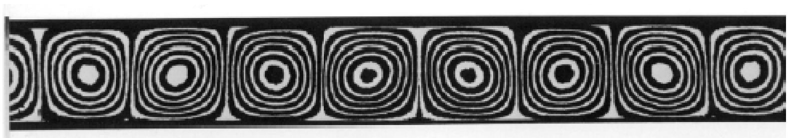
- **In a heated, closed domain, a flow particle will rise only if the buoyancy force is more important than the viscous forces and the thermal diffusion.**

- **There is a critical Rayleigh number from which the buoyancy force is high enough for the particle to start moving.**

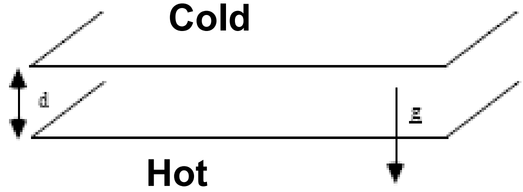
- **Experimentally, it can be shown that free convection settles since Ra > Ra*.**
  - **In a closed cavity, Ra* ~ 1700.**
  - **If Ra > Ra*, then there is no motion and heat transfer occurs only due to conduction.**

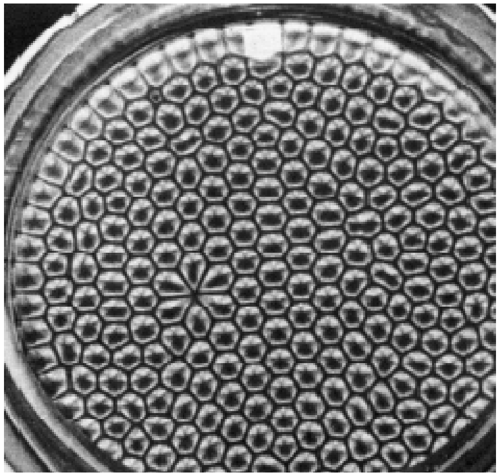
Transition Conduction / Convection

- The experiments described above is the Rayleigh Bénard problem (deals with free convection).
  - Fluid heated at the bottom expands and rises due to buoyancy forces.
  - At the top, fluid is cooling and then sinks.
- Some longitudinal rolling can be observed.
  - For high viscous material (silicon oil), these rolls are parallel as in the image below.



  - If the top is a free surface, we can observe hexagons due to the surface tension effects.

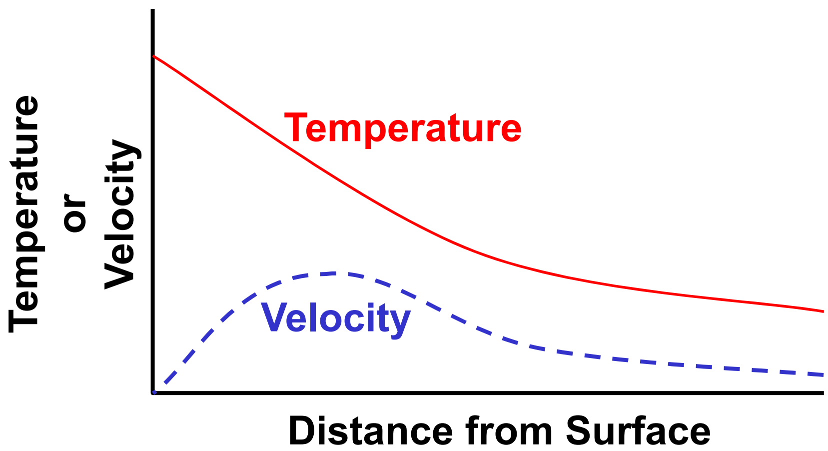




Free Convection Boundary Layers – Example

- **Consider a vertical heated wall (temperature T_w) in contact with a cold fluid. The fluid temperature far from the wall is T_∞.**

- **Close to the plate, there is a disrupted zone called a free convection boundary layer. The thickness and heat flux in this layer varies with the vertical coordinate.**



Profiles of Velocity and Temperature Perpendicular to a Heated Plate

The Full Buoyancy Effects Option

- **Generation of turbulent kinetic energy due to buoyancy (G_b) is by default neglected in the ϵ equation.**

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho u_i\varepsilon)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_T}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k+C_{3\varepsilon}G_b\right)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon$$

- **The buoyancy effects on the generation of ε are less clear than those for *k*.**
- **The degree to which ε is affected by buoyancy is determined by the constant $C_{3\varepsilon}$**
- **In FLUENT, $C_{3\varepsilon}$ is calculated according to the following relation:**

$$C_{3\varepsilon}=\tanh\left|\frac{v}{u}\right|$$

v **is the component of flow velocity parallel to the gravity vector**
u **is the component of flow velocity perpendicular to the gravity vector**

$C_{3\varepsilon}$ = 1 for flow direction aligned with gravity
$C_{3\varepsilon}$ = 0 for flow direction that is perpendicular to the direction of gravity

Turbulence Generation Due to Buoyancy

- **The generation of turbulent kinetic energy due to buoyancy (G_b) is, by default, always included in the TKE equation.**
  - **G_b is always included in the k equation for SKE, RNG, SKE, and RSM turbulence models.**
  - **Buoyancy effects can be included in the k–ω models (UDF is required, see Appendix).**

- **Note that both Gravity and Energy must be enabled.**

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho u_i k)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k$$

Turbulent viscosity

$$G_b=\beta g_i \frac{\mu_T}{\mathrm{Pr}_T}\frac{\partial T}{\partial x_i}$$

Turbulent Prandtl number for energy

- **The buoyancy effects on the generation of k are relatively well understood.**
  - **Turbulent kinetic energy tends to be augmented (G_b > 0) in unstable stratification.**
  - **For stable stratification, buoyancy tends to suppress turbulence (G_b < 0).**

Reference Temperature and Expansion Coefficient

- **In the Boussinesq approach, the reference temperature is specified instead of the reference density.**
- **Reference density is implicitly specified in the Materials panel.**
- **The reference temperature should be selected based on this density, as follows:**
  - **For a closed system (cavity):** $T_0 = \dfrac{T_{\max} + T_{\min}}{2} \quad \Rightarrow \quad \rho_0 = \dfrac{\rho_{T_{\max}} + \rho_{T_{\min}}}{2}$
  - **For an open system:** $T_0 = T_\infty \quad \Rightarrow \quad \rho_0 = \rho_\infty$
- **Thermal expansion coefficient**
  - **In general,** $\beta = -\dfrac{\left(\rho_{T_{\max}} - \rho_{T_{\min}}\right)}{\rho_0\left(T_{\max} - T_{\min}\right)}$
  - **For an ideal gas,** $\beta = \dfrac{1}{T}$

Back

UDF for Computing k–ω Turbulence Source Term


```
/* A source term to the k equation could be invoked through a  */
/* DEFINE_SOURCE UDF to include this effect in k-omega model   */
#include "udf.h"
#include "sg.h"
#include "models.h"

/* UDF to compute buoyancy production in TKE equation         */
/* In order to use UDF, the user must go to TUI and select YES */
/* for “keep temporary memory from being freed?               */
/* in /solve/set/expert                                       */

/* During the first iteration, the temperature gradient will   */
/* still not be stored and the error message will be displayed.*/
/* It should not appear again.                                 */

DEFINE_SOURCE(tke_gb, c0, t0, dS, eqn)
{
   real rho, beta, tke, tdr, mu_t, temp;
   real prod1,source;

   real pr_t = M_keprt;
   real Gravity[ND_ND];

   dS[eqn] = 0.0;
   source = 0.0;
```

UDF for Computing k–ω Turbulence Source Term


```
/* Compute Buoyancy Production */
 if(rp_seg){
            NV_V(Gravity, =, M_gravity);
            rho = C_R(c0,t0);
            tke = C_K(c0,t0);
            tdr = C_D(c0,t0);
            mu_t = C_MU_T(c0,t0);
            temp = C_T(c0,t0);
/* This assumes ideal gas behavior. More general     */
/* implementation would query beta from the solver   */
   beta = 1./temp;
            if(NNULLP(T_STORAGE_R_NV(t0, SV_T_G)))
            {
                      prod1 = beta*mu_t/pr_t*NV_DOT(Gravity,C_T_G(c0,t0));
            }
            else Message0("Error, temperature gradient not stored\n");
            source = prod1;
   }
   else
   {
             Message0("This udf is NOT VALID for the DENSITY-BASED SOLVER\n");
   }
   return source;
```