SD Numerical Simulation Technique for Hydrodynamic Flow Gas-Solids Mixing

Presented by:

Irla Mantilla N.
irlamn@uni.edu.pe

National University of Engineering, Lima - Perú

COMSOL CONFERENCE
BOSTON 2013

Abstract

We formulate a new mathematical model for a combustion chamber hydrodynamic fluid bed system (CFB) in thermal coal or solid waste power plants.

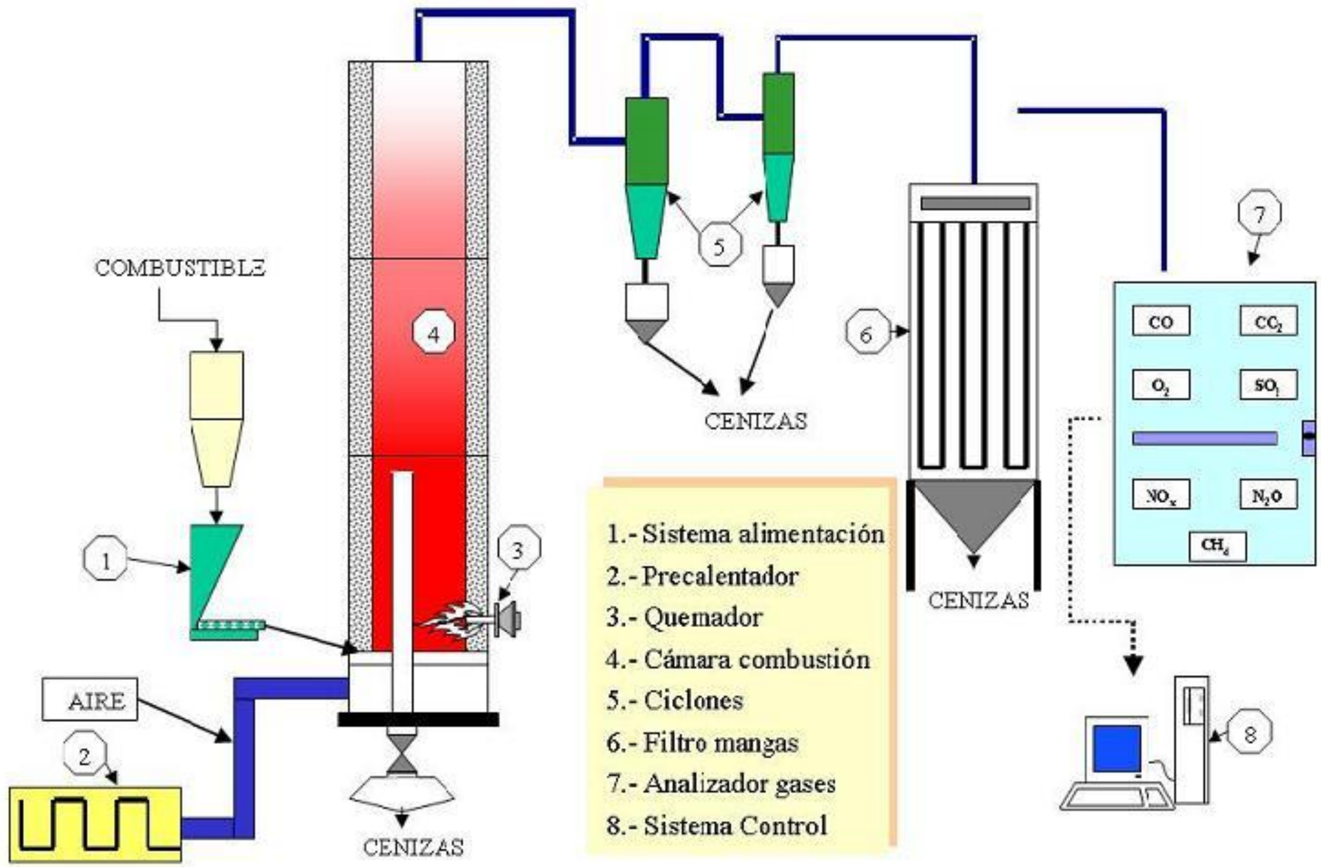
This mixture model is based in conservation equations (mass and momentum). This model gas - solid is obtained from two-phase hydrodinamic model, which takes into account a parameter ε (ratio densities gas/solid), it generates a free boundary problem.

Making an asymptotic adjustment and uncoupling of the dependent variables, then this problem has solution. The numerical simulation in 2D is implemented with COMSOL Multiphysics.

Content

1. Problem Formulation
2. Theoretical Analysis-Contribution
3. Numerical Resolution: Using COMSOL MULTIPHYSIC
4. Results

INTRODUCTION
SQUEME SYSTEM CFB [6]



1. Problem Formulation

Antecedent: The interphase momentum transfer between the two phases represented by the drag force, play an important role in any multiphase flow approach. Due to its high relevance, this phenomenon was frequently investigated in the literature. The ultimate goal of these work was to get an optinum drag model for betters fluidized bed hydrodynamics.

The volume fractions conservation equations are related as:

$$\varepsilon_s + \varepsilon_g = 1$$

Equations two phases of Gidaspow, Syamlal & O'Brien

- Mass conservation equations

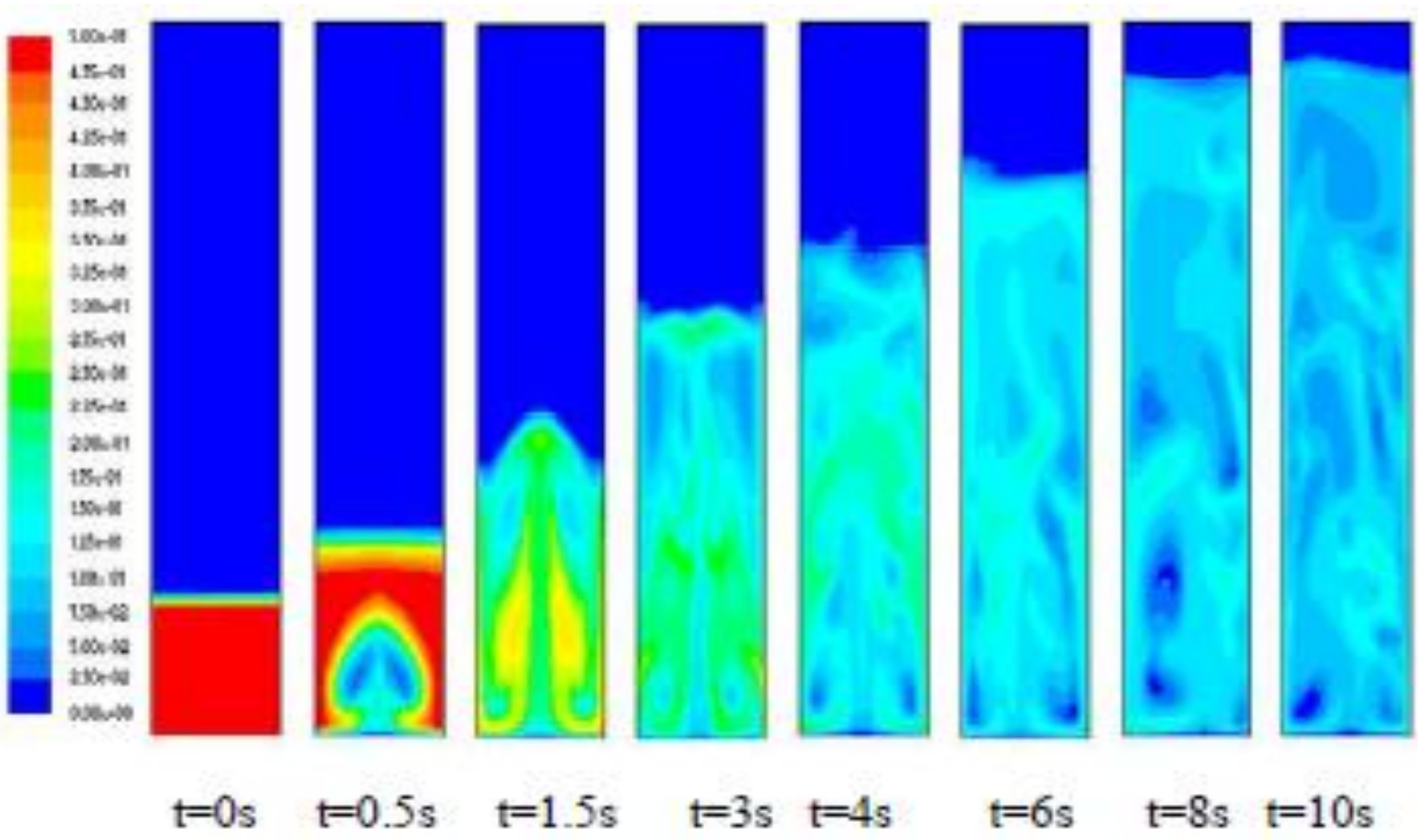
$$\frac{\partial(\varepsilon_g\rho_g)}{\partial t} + \nabla.(\varepsilon_g\rho_g u_g) = 0$$
$$\frac{\partial(\varepsilon_s\rho_s)}{\partial t} + \nabla.(\varepsilon_s\rho_s u_s) = 0$$

- Momentum conservation equations

$$\frac{\partial(\varepsilon_g\rho_g u_g)}{\partial t} + \nabla.(\varepsilon_g\rho_g u_g u_g) = \nabla.\left(\tau_g\right) - \varepsilon_g\nabla P - \beta(u_g\text{-}u_s) + \varepsilon_g\rho_g \mathrm{g}$$
$$\frac{\partial(\varepsilon_s\rho_s u_s)}{\partial t} + \nabla.(\varepsilon_s\rho_s u_s u_s) = \nabla.(\tau_s) - \varepsilon_s\nabla P - \nabla P_s - \beta(u_g\text{-}u_s) + \varepsilon_s\rho_s \mathrm{g}$$

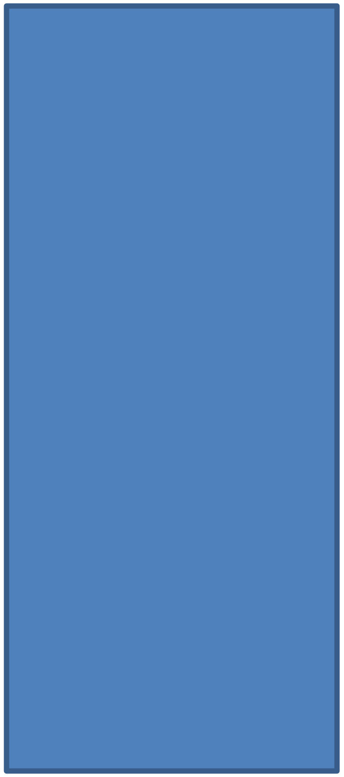
The simulation results showed that the drag models of Gidaspow and Syamlal & O'Brien highly overestimate the gas-solid drag force for the CFB the particles could not predict the formation of dense phase in the fluidized bed [2].

The conditions are characteristic of fast fluidization [1], [4]



A gas injection grid of Chamber CFB

Grid of pipes



Inlet

Two Phases Model Drew [2]

- *Phase Gas:*
- $\partial_t n + div(nu) = 0$ (1)
- $\partial_t(nu) + div(nu \otimes u + p_g I) = div(2\nu_g n D(u)) + ng - qm(u-v)$ *(2)*
- Phase Particle
- $\partial_t m + div(mu) = 0$ (3)
- $\partial_t(mv) + div(mv \otimes v + p_p I) = div(2\nu_p m D(v)) + mg + qm(u-v)$ *(4)*
- $\varepsilon = \rho_g/\rho_p$, $\rho = \rho_p \alpha$; $n = n(\varepsilon)$, $m = m(\varepsilon)$, $u = u(\varepsilon)$, $v = v(\varepsilon)$
- $D(w) = ½[grad(w) + (grad(w))^T]$

2. Theoretical Analysis and Contribution

Assuming the existence of an indicator that measures the ratio of proportionality between the densities of the two phases, in particular the parameter ε *such that* $0<<\varepsilon<1$.

$\varepsilon = \rho_g/\rho_p$, $\rho = \rho_p \alpha$; $n = n(\varepsilon)$, $m = m(\varepsilon)$, $u = u(\varepsilon)$, $v = v(\varepsilon)$.

When $\varepsilon \rightarrow 0$, result the following *mathematical model which is compressible apparently.*

$$\partial_t \rho + div(\rho v) = 0 \quad (5)$$

$$\partial_t (\rho v) + div(\rho v \otimes v) + \nabla P = div(\rho D(v)) + \rho g \quad (6)$$

$$\nabla p_h = -\rho q(\rho)(u-v) \quad (7)$$

$$div((1-\rho)u + \rho v) = 0 \quad (8)$$

where $P = p_c + p_h$.

$D(w) = ½[grad(w)+(grad(w))^T]$ (9)

- **Equation of state**
- $p_c(\rho)=\rho^{\gamma o}exp[k\rho/(\rho^*-\rho)], \gamma_o \geq 1, 0 \leq \rho \leq \rho^* < 1$ (10)

- **Equation for the drag force** between phases:
- $q(\rho) = C_q/(1-\rho)^s, s > 0, s \in [1.4, 3.6]$ (11)

- Let Ω_t an open subset of $[R^3_+ \times [0,\infty>]$,
- $\Gamma_0 = \{\underline{x} = (x_1, x_2, t) \in R^3 / t > 0\}$, (2.1)
- $\Omega_t = \{ (x_1, x_2, t) \in R^3 / (x_1, x_2) \in \Omega, 0 \leq t < \infty\}$, (2.2)

$$\Omega \subset R^2$$

- The problem is to find the volume fraction of particles $\rho \in C^1(\Omega_t) \cap C^0(\Omega_t)$, velocity of the

- solid particles velocity $v \in C^{2,1}(\Omega_t) \cap [C^{1,0}(\Omega_t)]^d$, and gas velocity represented by $u \in [C^{1,0}(\Omega_t)]^{\mathrm{d}} \cap [C^{1,0}(\Omega_t)]^d$.
- The problem is considered hydrodynamic pressure $p_h \in C^{1,0}(\Omega_t) \cap C^0(\ _t)$, from the state equation (5)- (9), to d= 2, d is the dimension of the space of the dependent variables, this vector functions that vary in space and time, which satisfy the system of equations

Conservative Form Two phase Compresible Model

$$\bar{\varphi} = \begin{pmatrix} \rho \\ \rho v_1 \\ \rho v_2 \end{pmatrix} = \begin{pmatrix} \varphi_1 \\ \varphi_2 \\ \varphi_3 \end{pmatrix}$$

$$\bar{F}(\bar{\varphi}) = \left(\begin{pmatrix} \rho v_1 \\ p_c + \rho v_1^2 \\ \rho v_1 v_2 \end{pmatrix} \begin{pmatrix} \rho v_2 \\ \rho v_1 v_2 \\ p_c + \rho v_2^2 \end{pmatrix} \right)^T ,$$

$$\bar{G}(\bar{\varphi}) = \frac{2\mu}{3\,\mathrm{Re}} \begin{pmatrix} 0 & 0 \\ 2\frac{\partial v_1}{\partial x} - \frac{\partial v_2}{\partial y} & 2\frac{\partial v_1}{\partial y} - \frac{\partial v_2}{\partial x} \\ 2\frac{\partial v_1}{\partial y} - \frac{\partial v_2}{\partial x} & 2\frac{\partial v_2}{\partial y} - \frac{\partial v_1}{\partial x} \end{pmatrix}$$

$$\mu = \rho \nu$$

$$\bar{S}(\bar{\varphi}) = \begin{pmatrix} 0 \\ \rho\left(\frac{q(\rho)}{1-\rho}\left(M(t) - \frac{\partial p_h}{\partial x} \right) - g_x \right) \\ \rho\left(\frac{q(\rho)}{1-\rho}\left(M(t) - \frac{\partial p_h}{\partial y} \right) - g_y \right) \end{pmatrix}$$

If v=(v_1,v_2)=0

- $\nabla p_c = \rho g$ *(colitional pression gradient)*
- $\nabla p_h = -\rho q(\rho) u$ *(hydrodinamic pression gradient)*

- $div((1-\rho)u) = 0$
- $M = ((1-\rho)u)$
- $P = 0$

Contribution 1: Non conservative of the mixture model

$$\boldsymbol{U}(x,y,t) = (u_1 = \rho,\ u_2 = \rho v,\ u_3 = \rho u)$$

$$R_t + div RU = 0$$

$$(RU)_t + div(RU \otimes U) + grad(\varepsilon(1-\rho)\frac{p_g}{\rho_g} + \rho\frac{p_p}{\rho_p}) =$$

$$div(\varepsilon(1-\rho)\frac{\tau_g}{\rho_g} + \rho\frac{\tau_p}{\rho_p}) - div(\varepsilon(1-\rho)(u-v)\otimes(u-v)) +$$

$$R\vec{g};\ R = \rho + \varepsilon(1-\rho);\ \varepsilon \to 0, R = \rho,$$

$$0 < \varepsilon = \frac{\rho_g}{\rho_p} << 1$$

- **Boundary conditions**
- $[(1-\rho)u+\rho v)].\mathbf{n} = M>0 \in C^0(\Gamma_0 \times [0,\infty>)$ (2.3)
- $[\rho v].\mathbf{n} = m_0 \in C^0(\Gamma_0 \times [0,\infty>)$ (2.4)
- $[\rho v \otimes v + PI - \rho D(v)].\mathbf{n} = 0 \in C^0(\Gamma_0 \times [0,\infty>)$

Is this a boundary free problem

- **Initial conditions**
- $\rho(\underline{x},0) = \rho^0(x,y) \in C^0(R^2_+ \times \{0,T\})$ (2.5)
- $v(\underline{x},0) = v^0(x,y) \in [C^0(R^2_+ \times \{0,T\}]^2$ (2.6)

Cauchy problem

Contribution 2: Conditions to solve

$$grad(P_h) = grad(\varepsilon(1-\rho)\frac{p_g}{\rho_g} + \rho\frac{p_p}{\rho_p}) = \frac{1}{Fr}\frac{R}{(1-R)^m}(u - \mathrm{v})$$

$$Slip = (u - v)$$

$$R\vec{g} = \frac{1}{Fr}Rg\begin{pmatrix}0\\1\end{pmatrix},\ M = U + \frac{1-R}{R}\rho(u-v)$$

$$div(M) = 0$$

3. Numerical Analysis

- The work consists of the construction of a numerical model for the quantitative study of the problem. This includes formulation of decoupling techniques. The solution of the variational problem in space-time, singularized the discreta inestability in time during the process computational.
- To overcome this difficulty we have used the Galerkin method with a numerical technique to capture the discontinuities in the Stream Lines Difussion (SD) with finite elements of type P1 + P2 ([5], [6]).

In the two-dimensional case, after a process dimensionless introducing a vector function of states , thus the Conservative system in variational form convective-diffusive-reactive flow in the domain located in a rectangular geometry region $\Omega = ((0,L)x(0,H)))x[0,T)$

Boundary condition: Inlet (imput) and Wall

Initial condition: Step

Stabilization : SD Numerical Method , this is expressed by:

Application: Discretization Stream Diffusion capturing Method ([3], [6])

$$Set\, U(x, y, t^0)$$

$$\phi(\mathrm{v}(x, y, t^0)) = \mathrm{v} + \delta(\mathrm{v}_t + \psi(\mathrm{v}))\partial_i \mathrm{v})$$

$$Find\ :\ U_h^n \in \prod_{n=0}^{N-1} H_0^{1+2}(S_n)$$

$$B(U, \phi) = L(\phi)$$

$$B(U_h, \phi) = \sum_{n=0}^{N-1} \left\{ U_h{}^n + \partial_R f(U_h{}^n)\partial_i U_h{}^n{}_i, \mathrm{v}^{\mathrm{n}} + \delta(\mathrm{v}_{\mathrm{t}}{}^{\mathrm{n}} + \psi(\mathrm{v}^{\mathrm{n}}))\partial_i \mathrm{v}^{\mathrm{n}}) \right\} +$$

$$(U_+ - U_-, \phi_+) + \int_{\Gamma n} U_+ \phi_+ dt; U_{+,-} = \lim_{s \to 0^{+,-}} U(x, y, t+s)$$

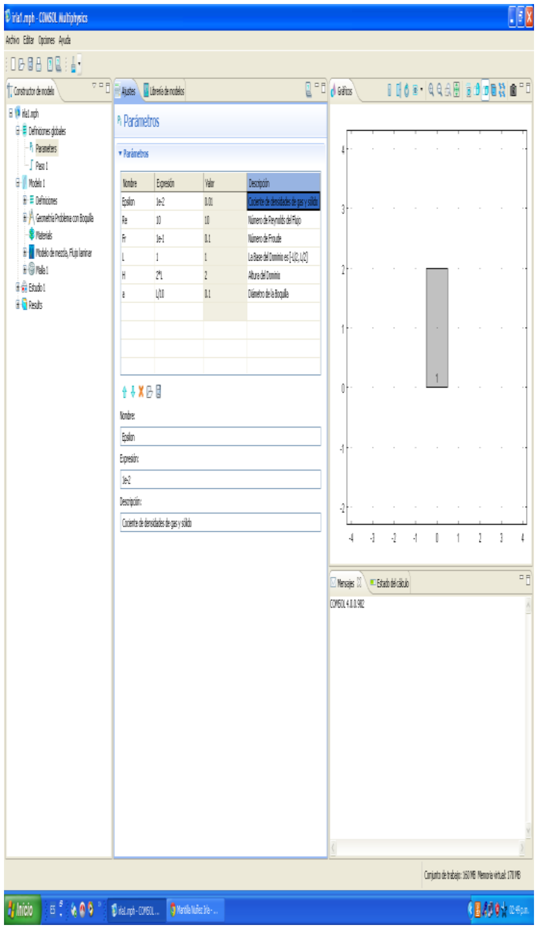
$$L(\phi) = (U^0, \phi_+) + \int_{\Gamma n} U_+ \phi_+ dt$$

$$U_h = \left\{ u_i \in H^{1+2}(\Omega_i^n) / U_K \in P^{1+2}{}_K(x, y), K \in \Im^n{}_h \right\}$$

Numerical Resolution: Using COMSOL Multiphysics

- In the two-dimensional case approximates the solution of the problem, then the method Galerkin stabilized stream Difussion (SD) and a difference scheme (BDF) for the variable explicit Capture and temporal discontinuities of singularities in the streamlines of the convective flow, can be improved with a remesh evolutive with h= $\{10^{-4}, 10^{-3}\}$ side length element maximum and minimum and with a resolution of 0.25 of curvature.

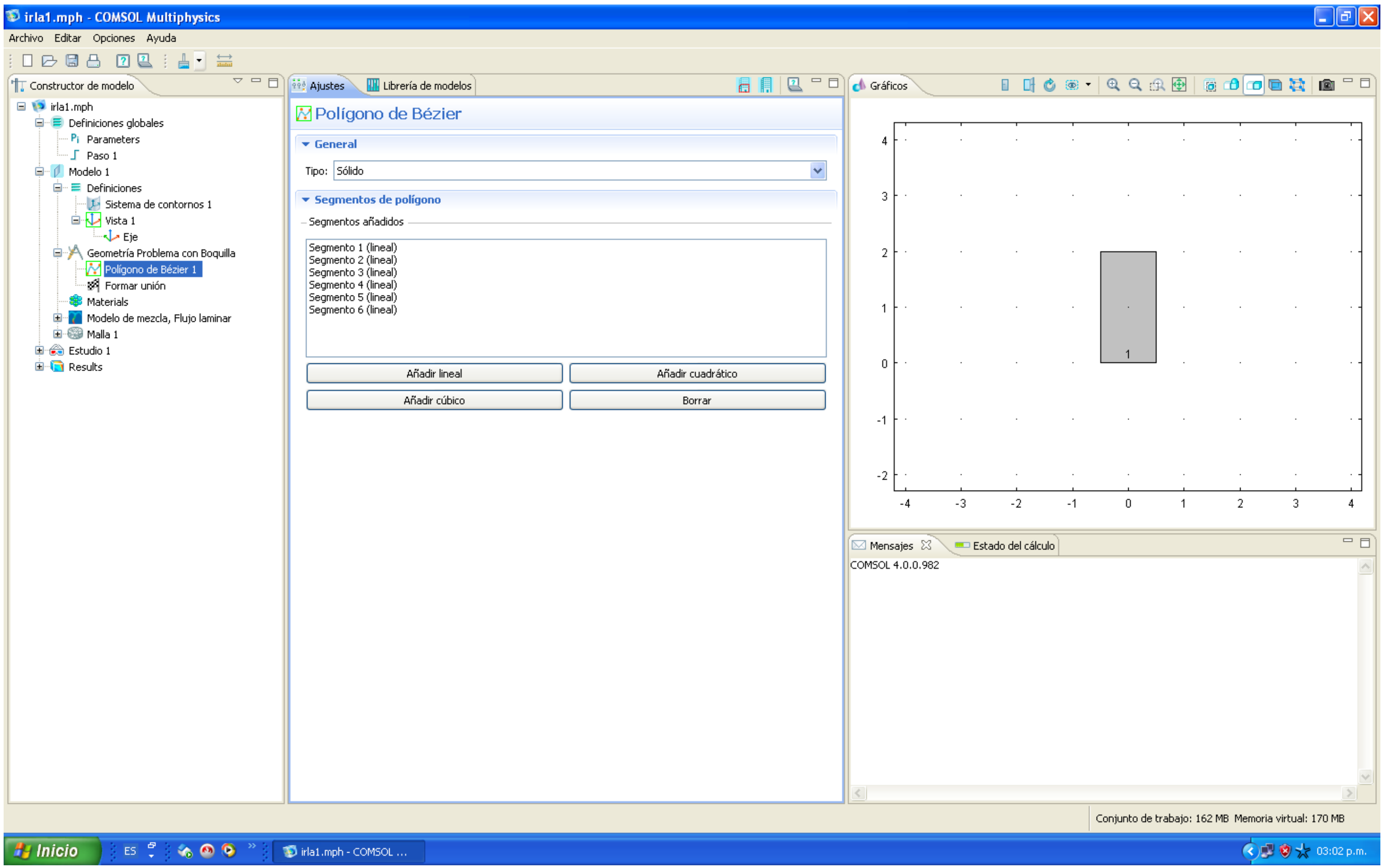
4. RESULTS



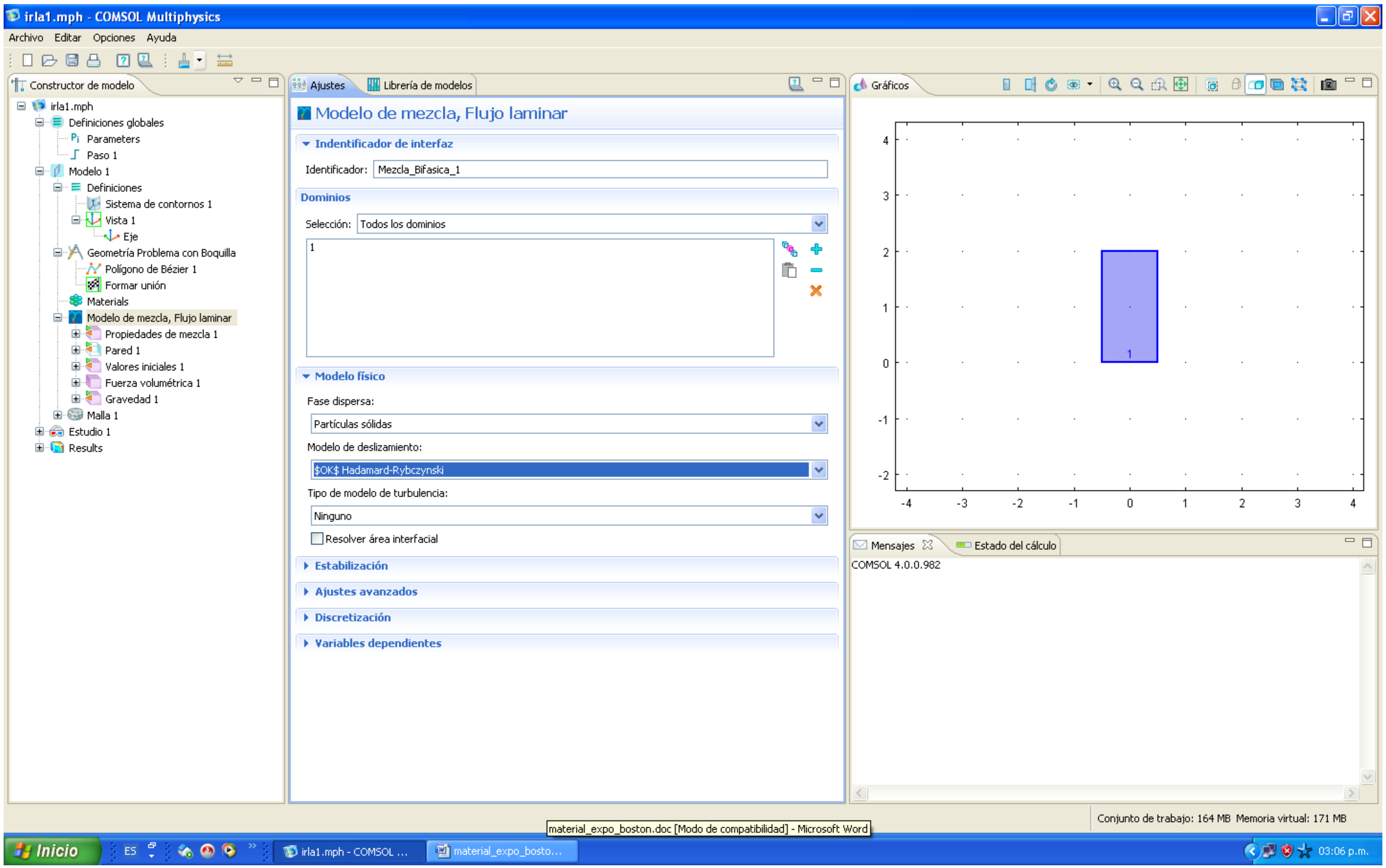
DATOS	Valor	Descripción
h0	1	Unidad de Longitud (Altura del Lecho en Reposo)
L	1/h0	Base Adimensional [-L/2, L/2]
H	2*L	Altura Adimensional
Db	L/10	Diámetro Adimensional de la Boquilla
Epsilon	1e-2	Cociente entre Dendidades de las Fases
Ng_Np	1	Cociente entre las Viscosidades Cinemáticas (gas_sólido)
Fr	1, 5, 10	número Froude
Re	1e+2, 4x1e+2, 5x1e+2	número Reynolds
Dp	1e-4	Diámetro de partícula
ro_max	pi/6	Fracción Volumétrica de Sólidos Máxima
Vsoplado	0.2	vel. de soplado

Parameters

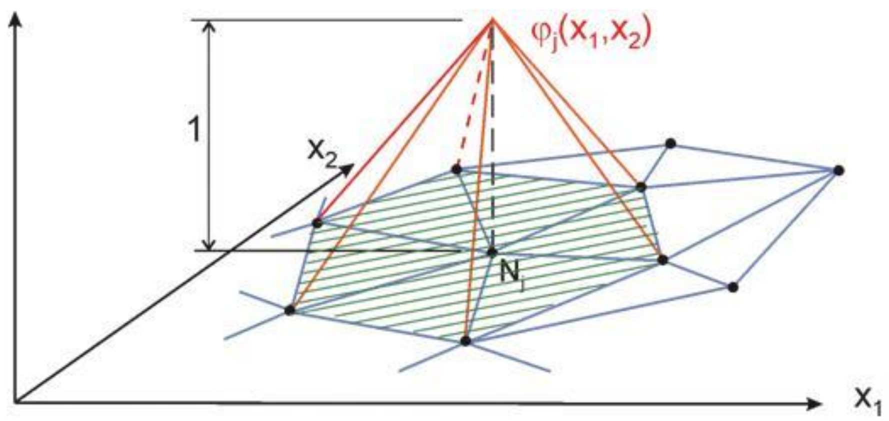
Geometry

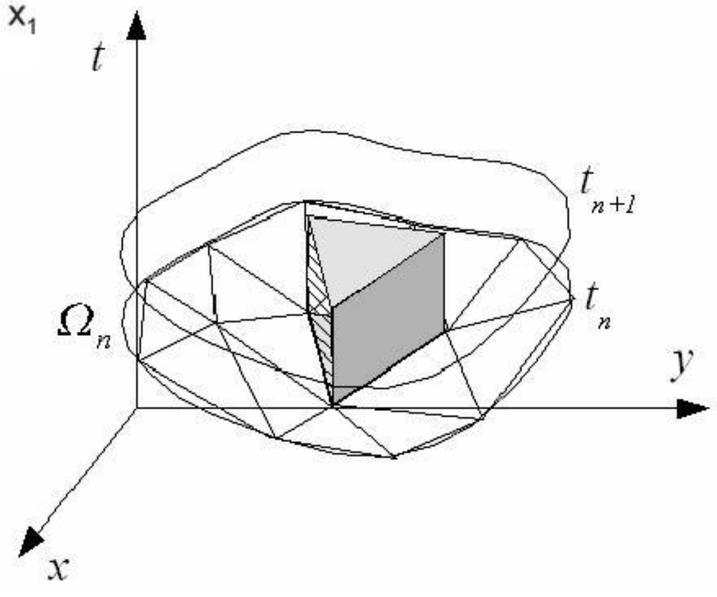

irla1.mph - COMSOL Multiphysics
Archivo Editar Opciones Ayuda
Constructor de modelo
irla1.mph
Definiciones globales
Parameters
Paso 1
Modelo 1
Definiciones
Sistema de contornos 1
Vista 1
Eje
Geometría Problema con Boquilla
Polígono de Bézier 1
Formar unión
Materials
Modelo de mezcla, Flujo laminar
Malla 1
Estudio 1
Results
Ajustes
Librería de modelos
Polígono de Bézier
General
Tipo: Sólido
Segmentos de polígono
Segmentos añadidos
Segmento 1 (lineal)
Segmento 2 (lineal)
Segmento 3 (lineal)
Segmento 4 (lineal)
Segmento 5 (lineal)
Segmento 6 (lineal)
Añadir lineal
Añadir cuadrático
Añadir cúbico
Borrar
Gráficos
Mensajes
Estado del cálculo
COMSOL 4.0.0.982
Conjunto de trabajo: 162 MB Memoria virtual: 170 MB
Inicio
irla1.mph - COMSOL ...
03:02 p.m.

Hadamard R. Model Type

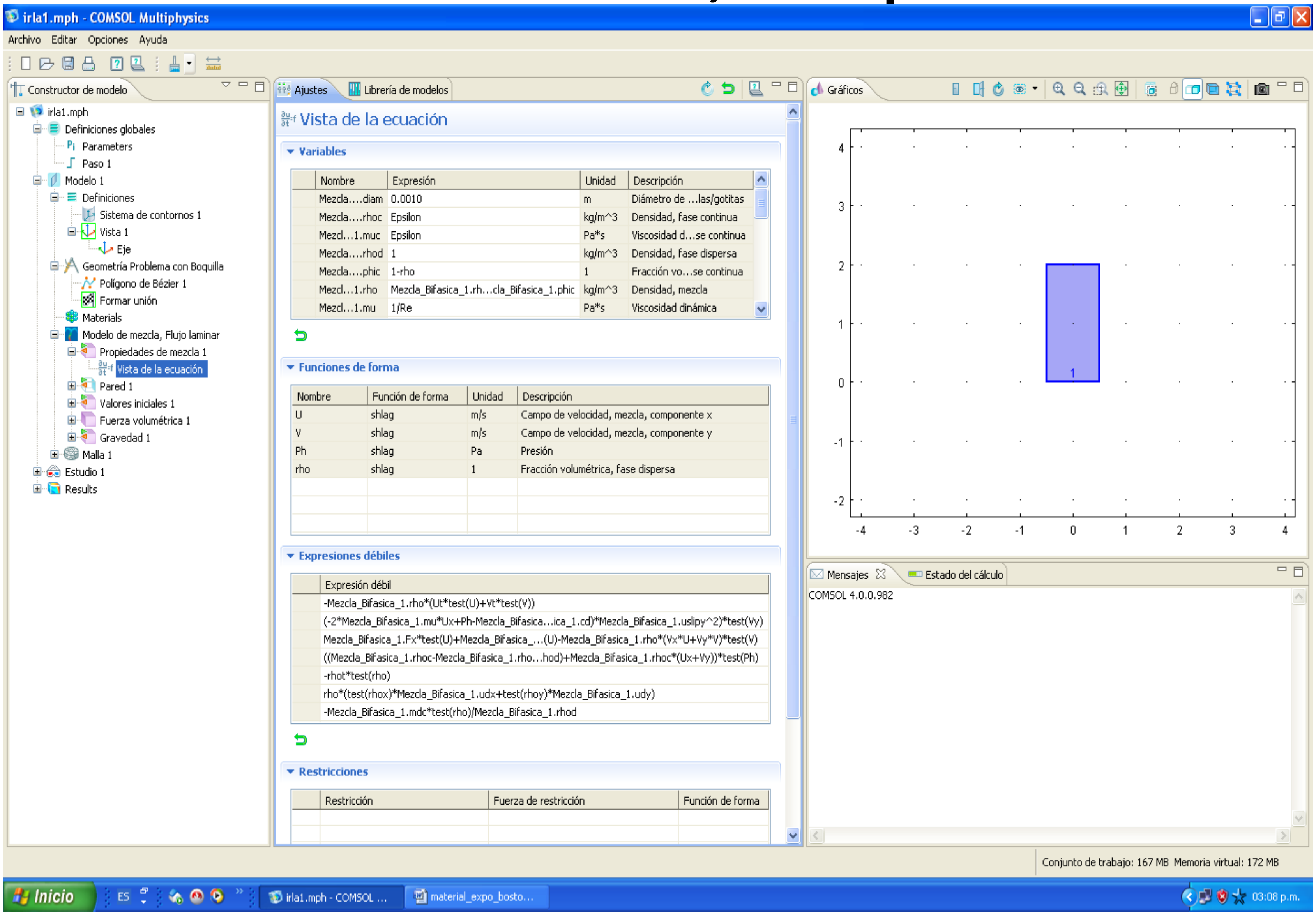


Finite Element type P1+P2

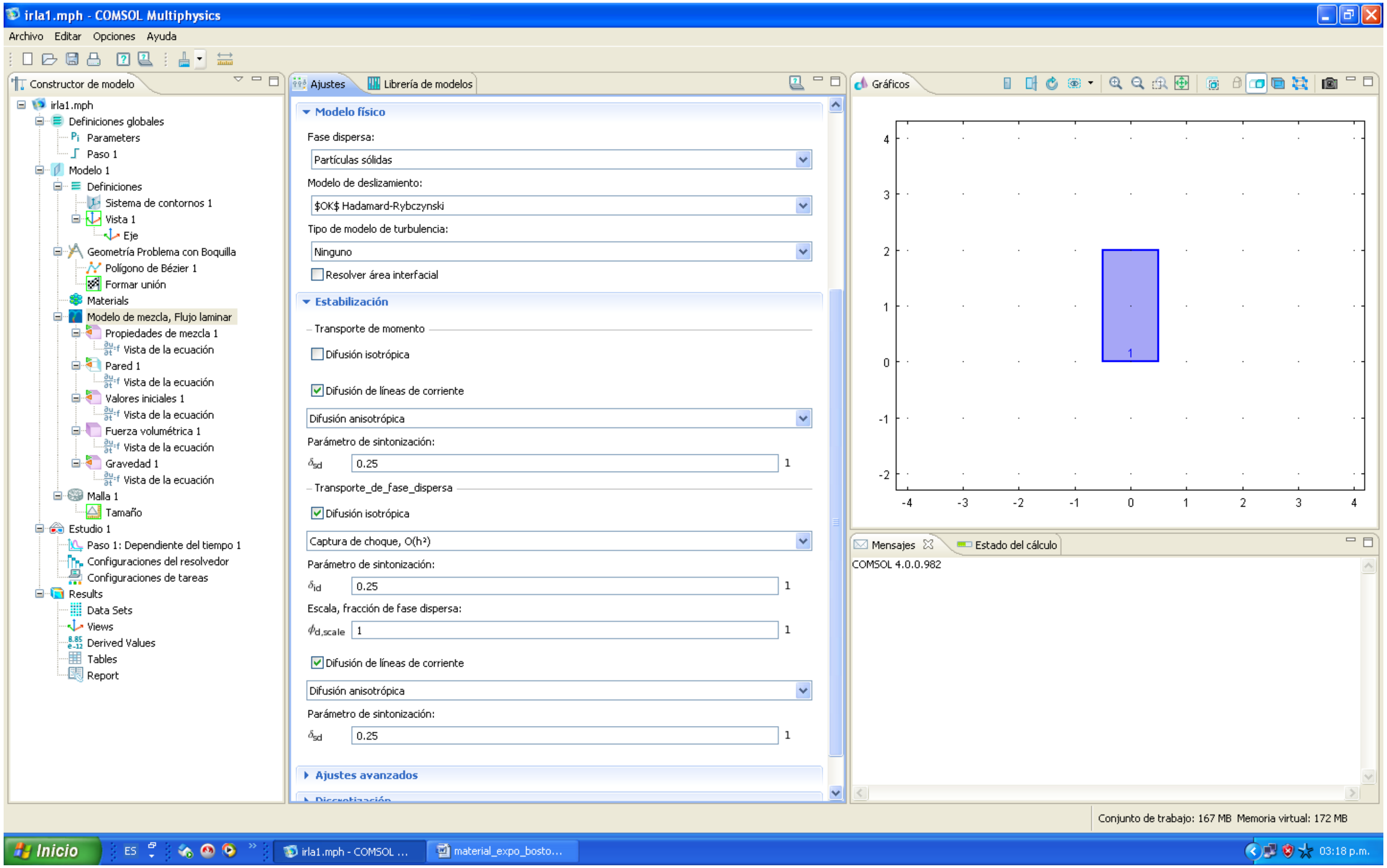




Finite Elements, shape function

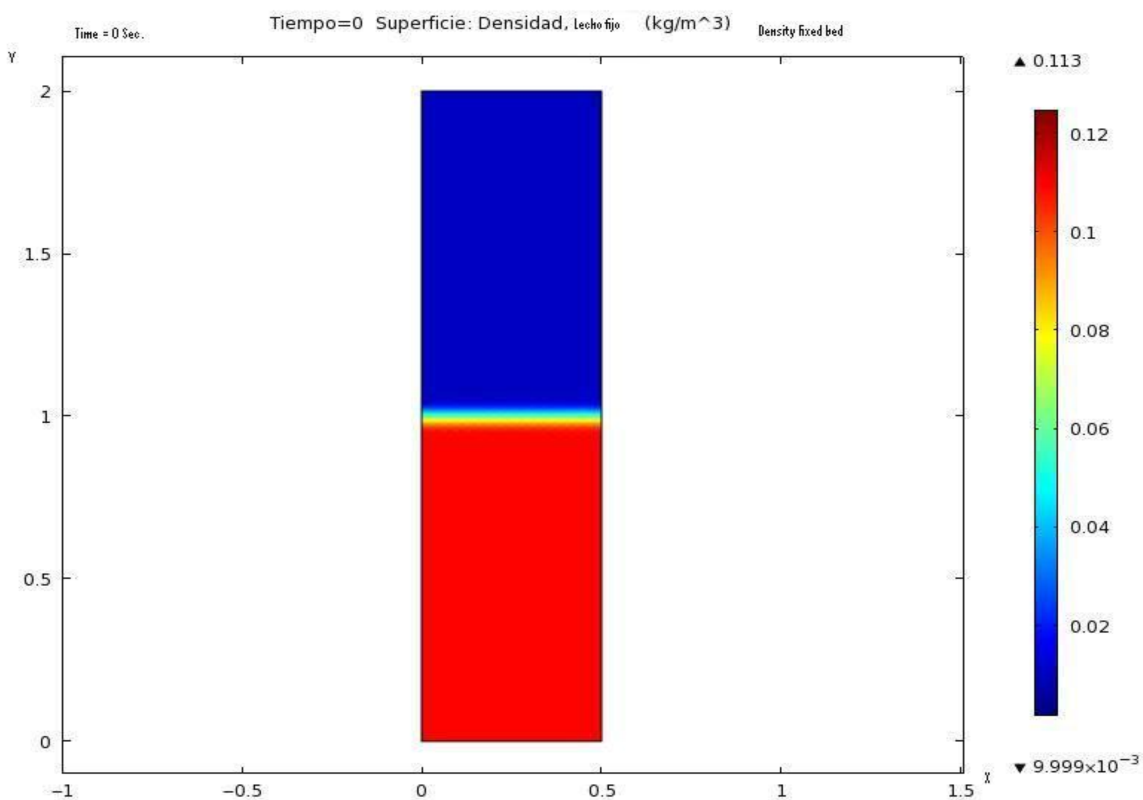


Stabilization SD

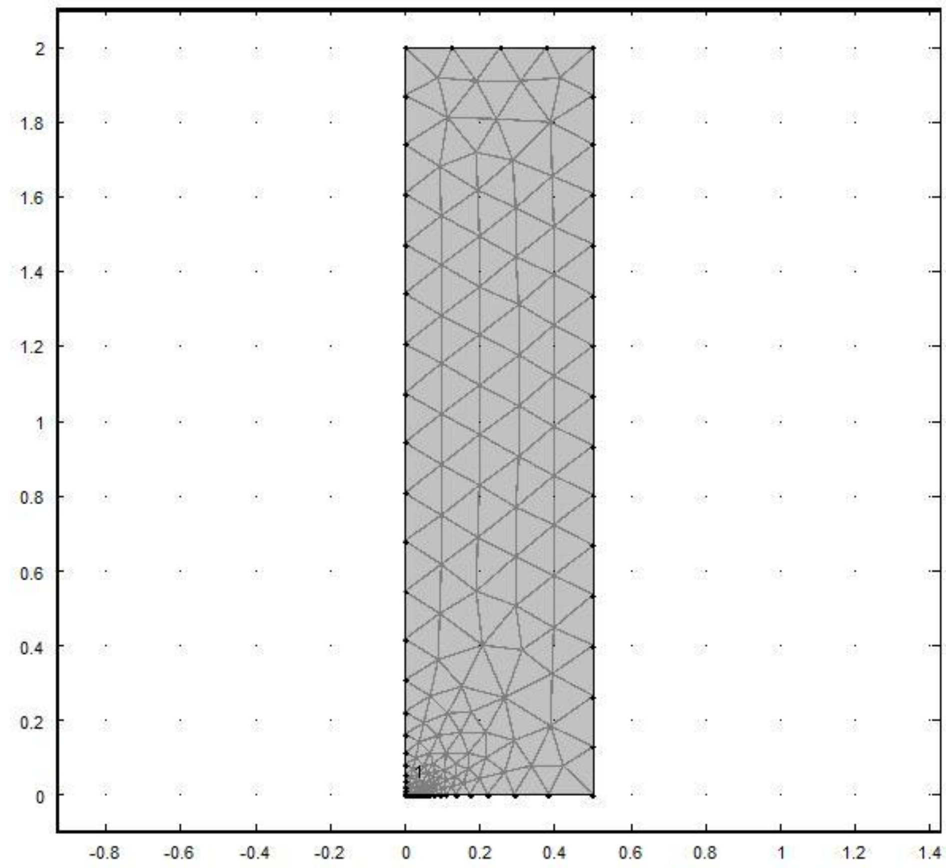


Domain fixed CFB

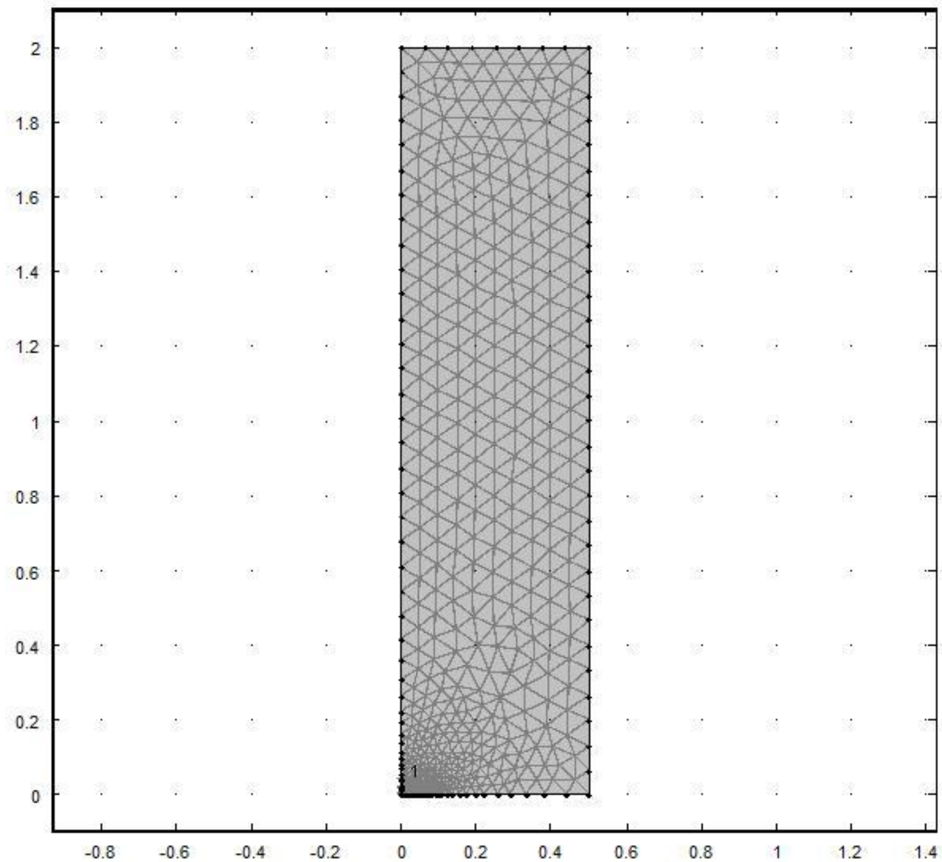
The axial section of the is represented in the XY plane.



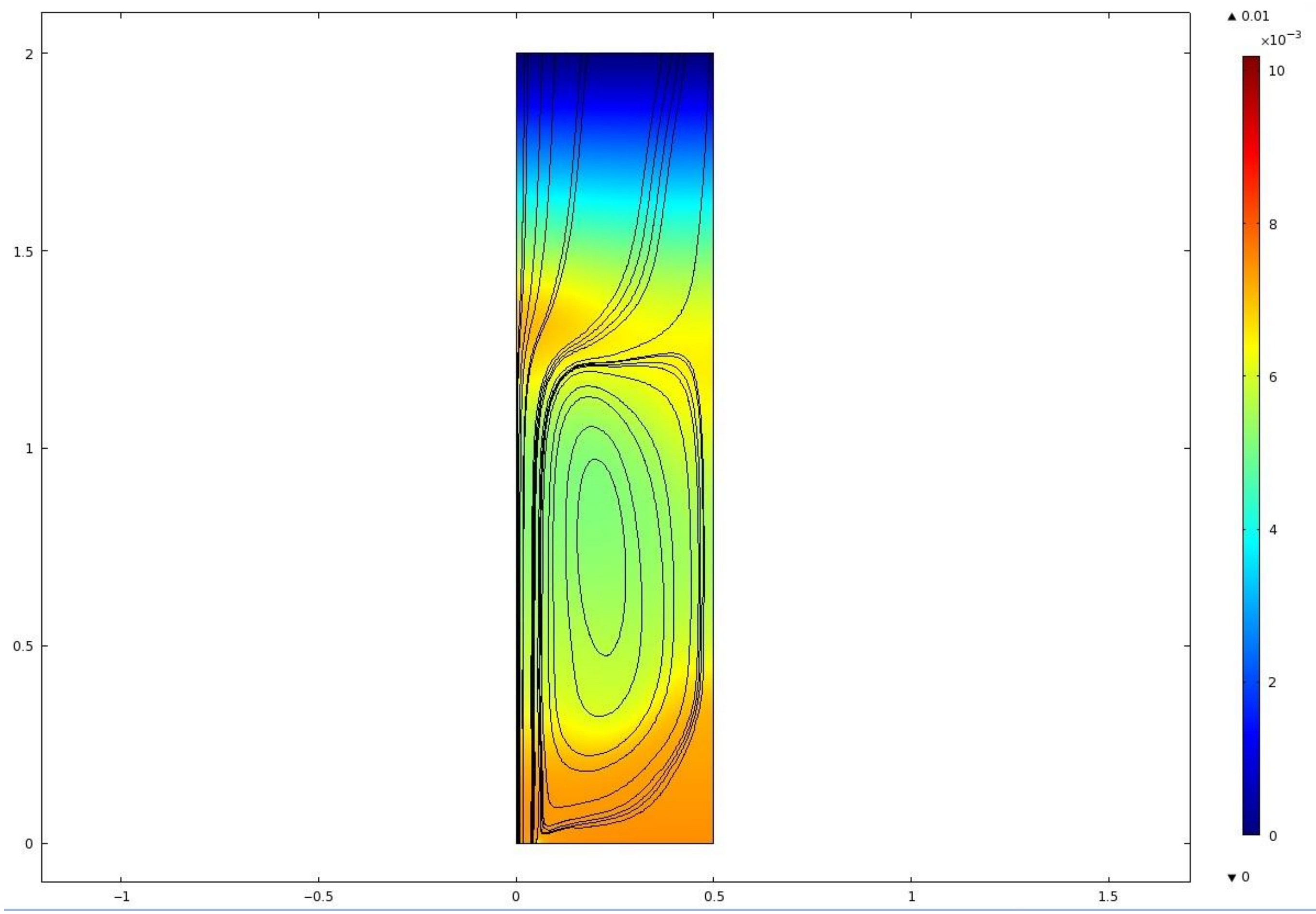
Initial mesh



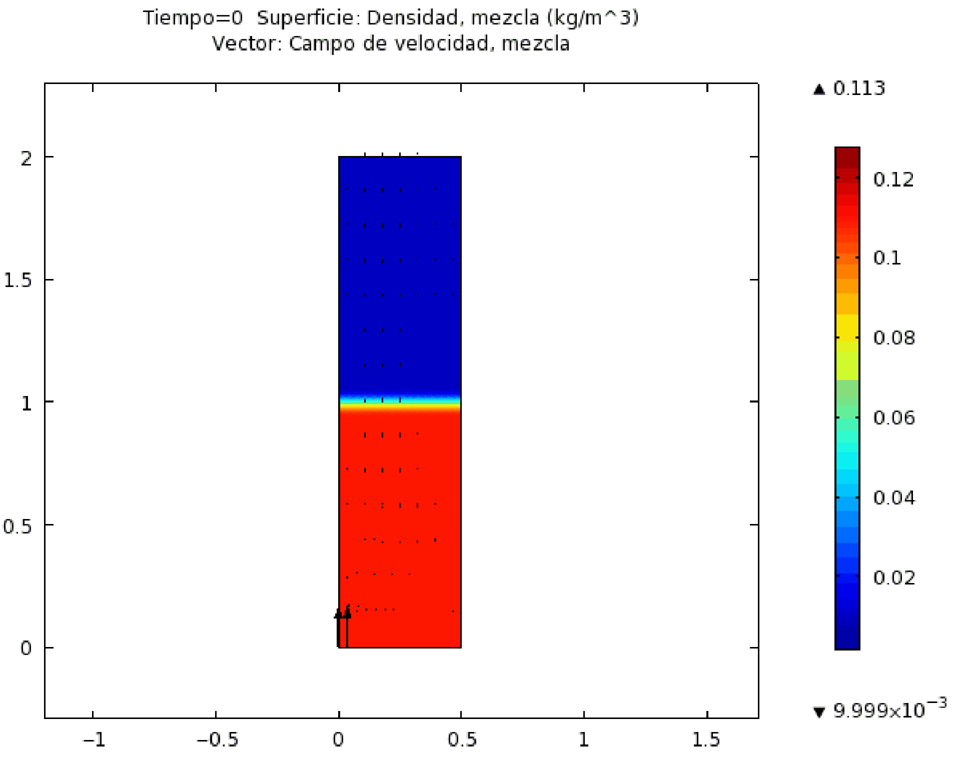
Remeshing finite Element



Pressure isolines

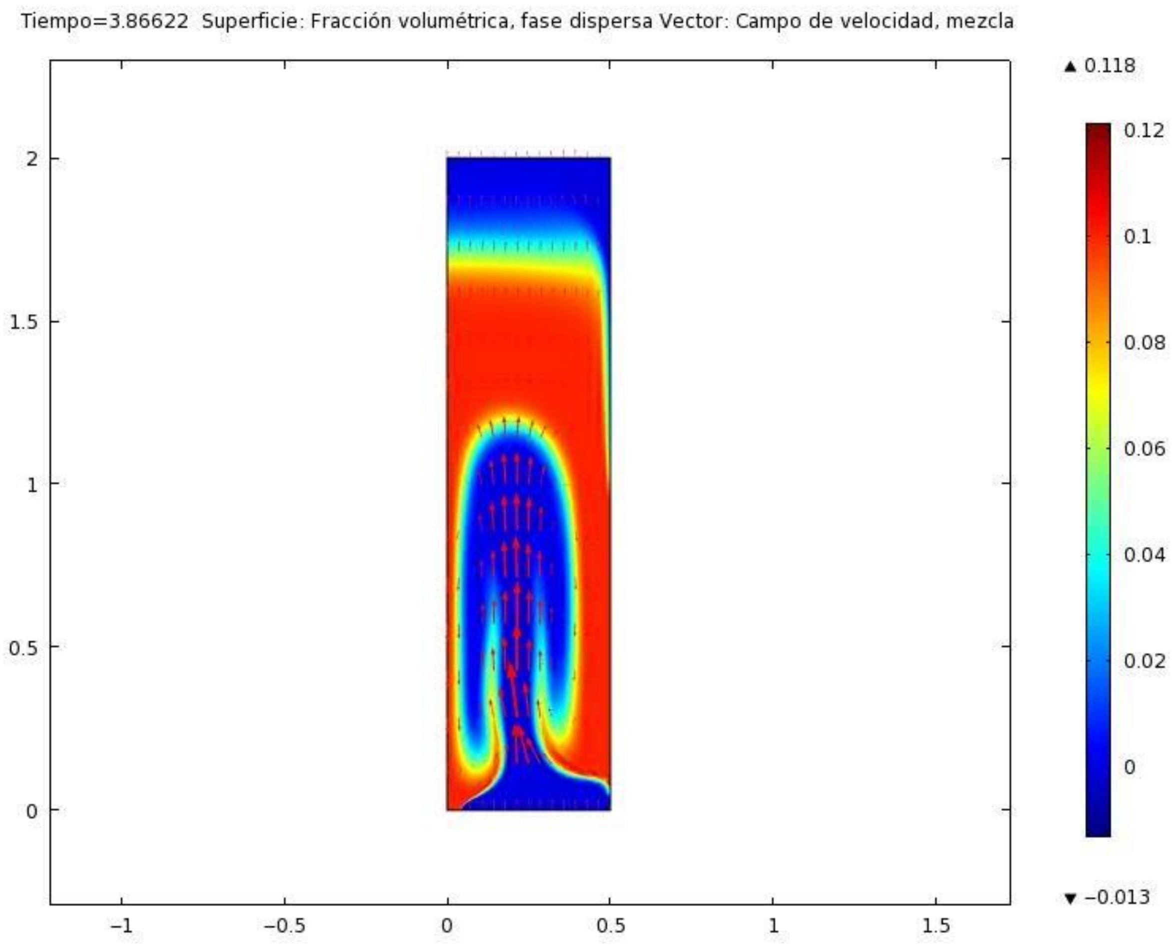


A nozzle

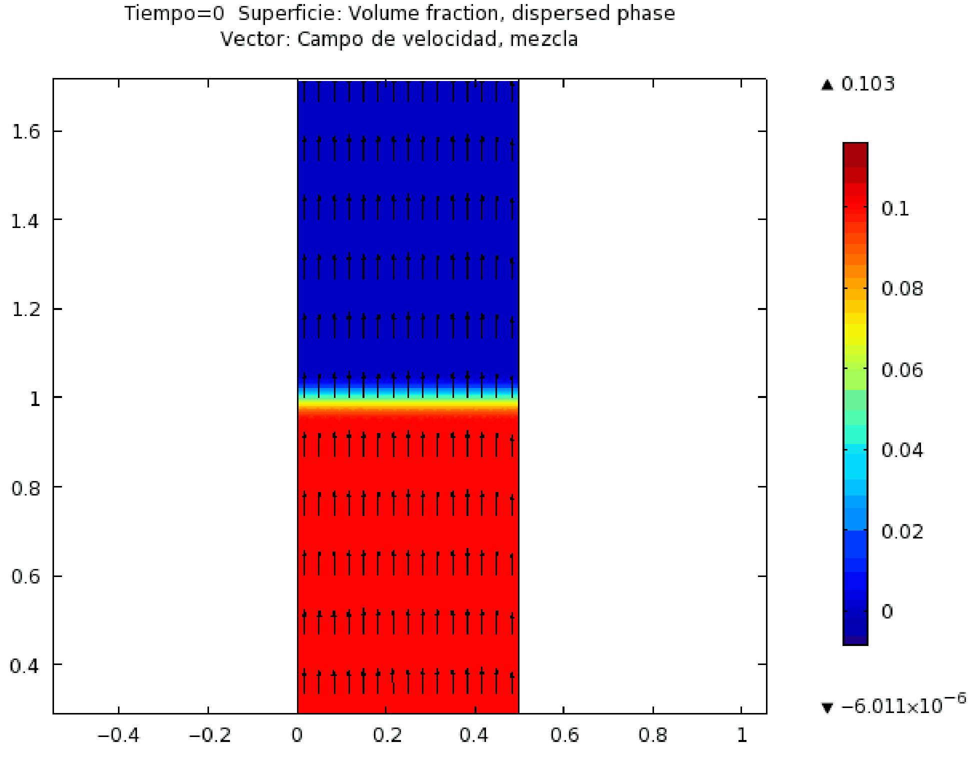


Tiempo=0 Superficie: Densidad, mezcla (kg/m^3)
Vector: Campo de velocidad, mezcla
2
1.5
1
0.5
0
−1
−0.5
0
0.5
1
1.5
▲ 0.113
0.12
0.1
0.08
0.06
0.04
0.02
▼ 9.999×10−3

Multiple nozzles



Multiple nozzles



Conclusion

1. The spectrum of the color palette, particles (red) and only gas flows (blue), a speed minimum fluidization of the results observed with the increase in the flow in bed, manifests a state of suspension caused by the upward flow gas by one and multiple nozzles. This flow creates drag force (inertial force) which balances gravity and terminal velocity which is manifested in the rate of free of the disperse phase.
2. The minimum speed is observed when bubbling the first bubble ppears, this is important because it causes the homogenity mixing Solid -Gas.

3. The convergence criteri is obtained when there expansion homogeneous mixture, ie.

$$\delta = c * h, \; h \neq 0$$

$$c = \|U\| \frac{\Delta t}{h} \to 1$$

$$\varepsilon = 10^{-4}$$

$$Fr \approx 10^{-3}$$

$$\mathrm{Re} = (400{,}500)$$

References

1. J.R. Grace, G. Sun, *Influence of particle size distribution on the performance of fluidized bed reactors*, *Journal*, Chem. Eng., Volume. 69 (5), pages 1126-1134 (1991).
2. Drew, D.A, *Mathematical modelling of two-phase Flow*. Annual Review *Fluid Mechanical*, Volum**e** 15, pages 261-291 (1983).
3. Claes Johnson, *Numerical solution of partial differential equations by the finite element*, pages 182-187, Cambridge University Press, Sweden, (1994).
4. Zimerman, S and F. Taghipour. CFD Modeling of the Hydrodinamics and Reaction Kinetics of FCC Fluidized Bed Reactors. Ind. Eng. Chem. Journal. Volume 44, pages. 9818 – 9827. 2005
5. COMSOL MULTIPHYSICS, *User's guide*. Version 4.0, pages 290-298, *Module CFD*, pages 260-308 (2010).
6. Mantilla, Irla, *Mathematical Contribution to Simulate the Numeric Behavior of the Mixture Flow Gas – Solid***,** Doctoral Thesis in National University of Engineering, 2012.

- Thank you very much