Computational Fluid Dynamics (CFD)

A cura di:

Marco Lombardelli

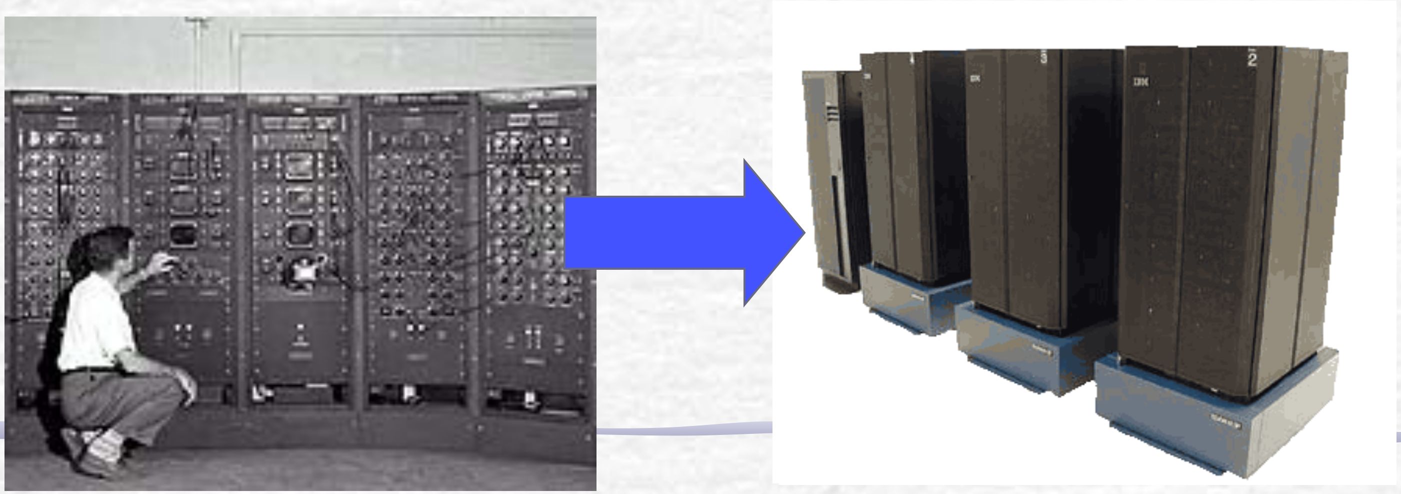
Punti fondamentali

1. CFD: Cosa? Come? Perché? Quando?
2. Modelli
3. Richiami teorici
4. Esempi

Cosa è il CFD?

- CFD è una sigla che indica un metodo numerico,discreto ed approssimato finalizzato ad affrontare problemi di termofluidodinamica.
- Storicamente il metodo CFD è stato preceduto dagli approcci AFD (Analytical Fluid Dynamics) ed EFD (Experimental Fluid Dynamics) .
- Il CFD come il FEM è stato possibile solo dopo l’inizio della “rivoluzione informatica”.

(500 flops, 1947→20 teraflops, 2003)

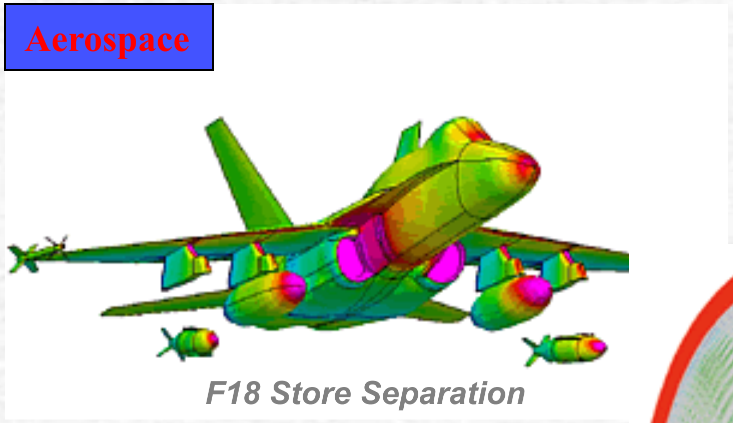


Perchè usare il CFD?

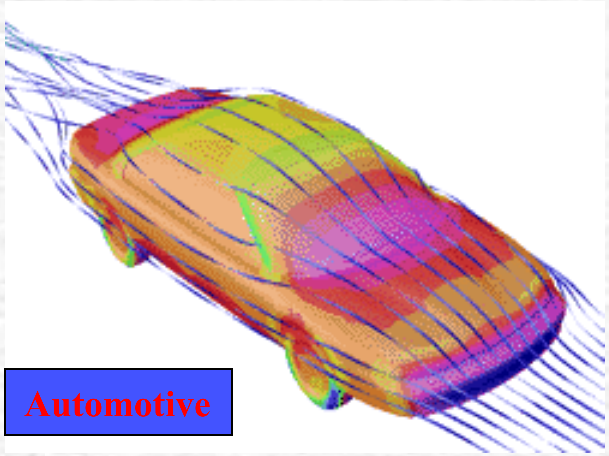
- Progettazione di prototipi
  1. “Simulation-based design” invece di “build & test”
     - ☐ Più economico e rapido
     - ☐ Possibilità di effettuare confronti diretti tra diverse soluzioni.
  2. Simulazione di problemi difficili o impossibili da trattare sperimentalmente
     - ☐ Grandi scale (navi, aerei, edifici)
     - ☐ Effetto delle condizioni atmosferiche (vento)
     - ☐ Situazioni pericolose (esplosioni, twister)
     - ☐ Problemi di interesse aereospaziale (ionosfera, mesosfera ecc.)
- Interpretazione della fisica dei problemi

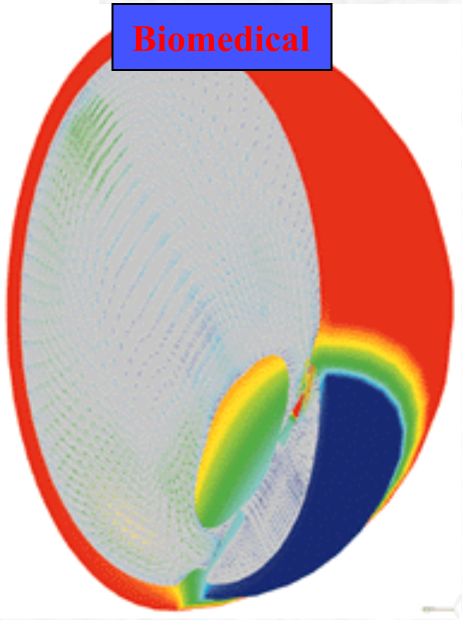
CFD:In quali settori?

- Settori?
    - ***Aerospaziale***
    - ***Automotive***
    - ***Biomedico***
    - Processi chimici
    - Ergonomia
    - Idraulica civile/ industriale
    - Marittimo
    - Oil & Gas
    - Energia
    - Sport



F18 Store Separation

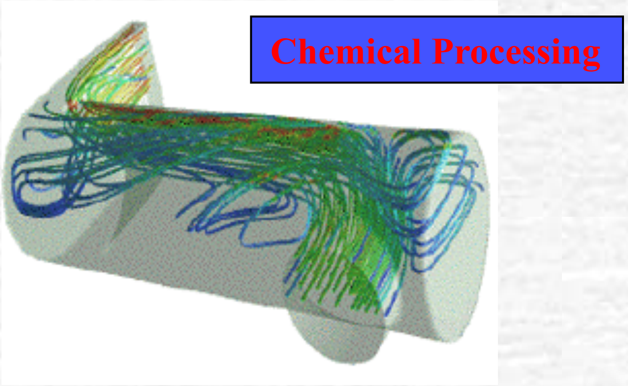

Automotive


Biomedical

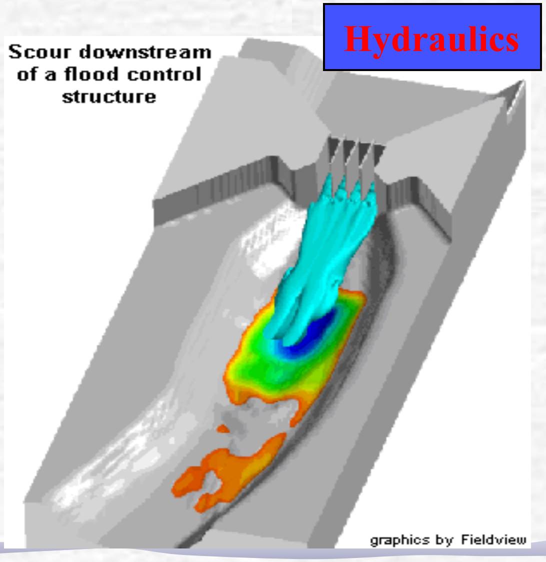
Temperature and natural convection currents in the eye following laser heating.

CFD: In quali settori?

- Settori?
  - Aerospaziale
  - Automotive
  - Biomedico
  - ***Processi chimici***
  - ***Ergonomia***
  - ***Idraulica civile/ industriale***
  - Marittimo
  - Oil & Gas
  - Energia
  - Sport


Chemical Processing

Polymerization reactor vessel - prediction of flow separation and residence time effects.

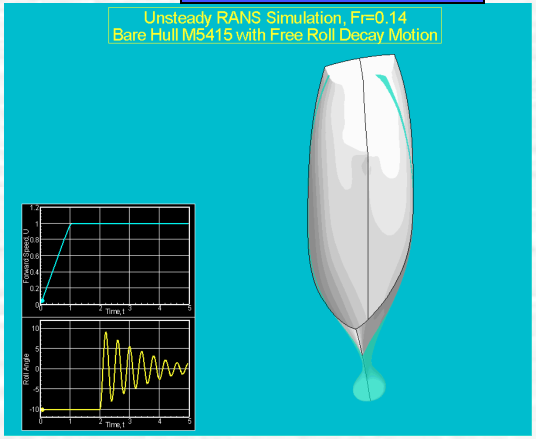

Hydraulics
Scour downstream of a flood control structure
graphics by Fieldview

Streamlines for workstation ventilation

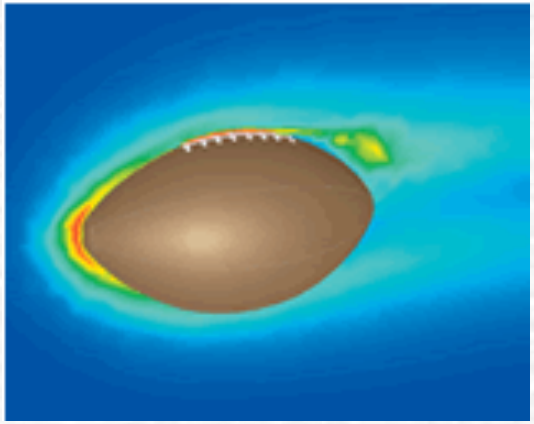
CFD: in quali settori?

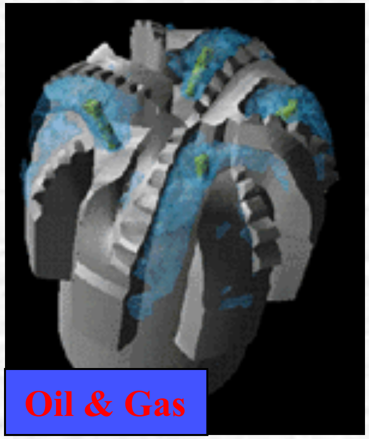
- Settori?
  - Aerospaziale
  - Automotive
  - Biomedico
  - Processi chimici
  - Ergonomia
  - Idraulica civile/ industriale
  - ***Marittimo***
  - ***Oil & Gas***
  - ***Energia***
  - ***Sport***

Marine (movie)


Unsteady RANS Simulation, Fr=0.14
Bare Hull M5415 with Free Roll Decay Motion
Forward Speed, U
Time, t
Roll Angle
Time, t

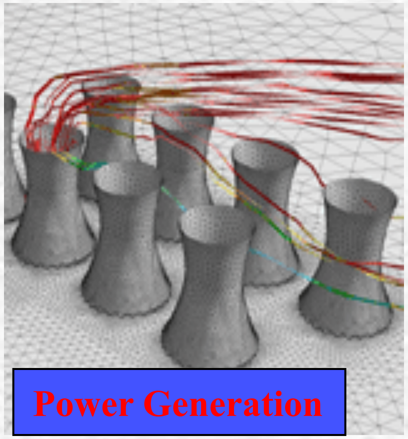
Sports





Oil & Gas

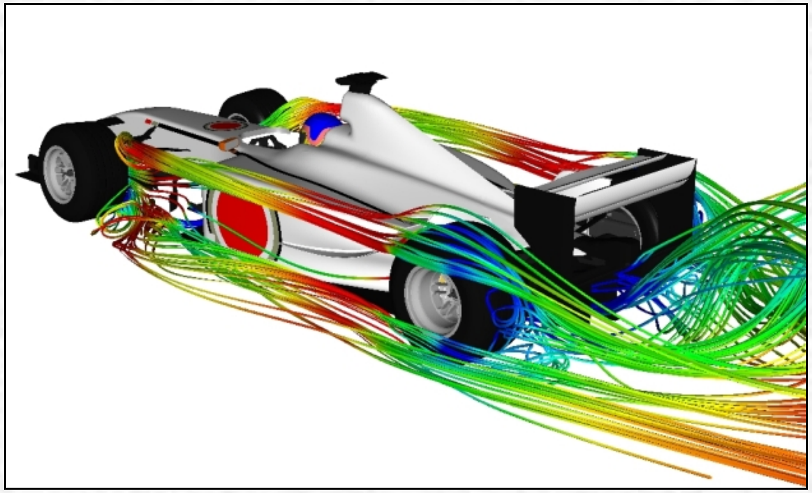
Flow of lubricating mud over drill bit

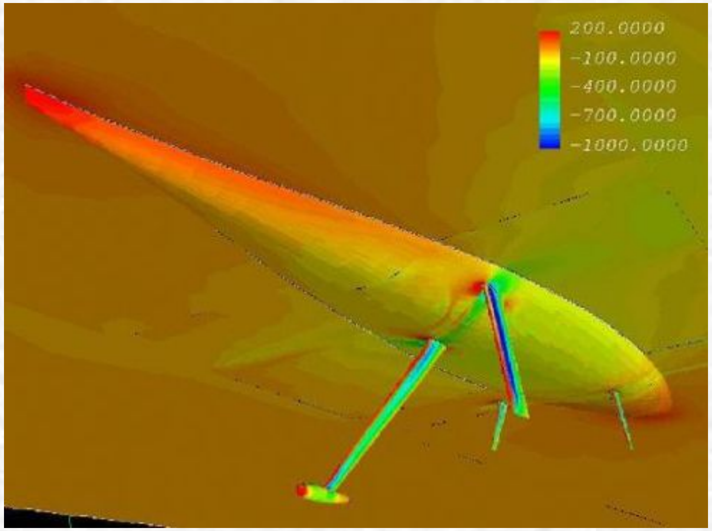


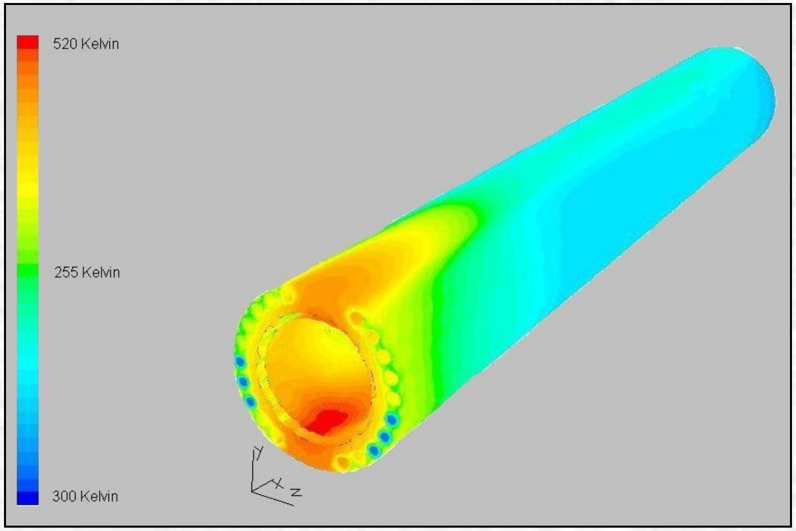
Power Generation

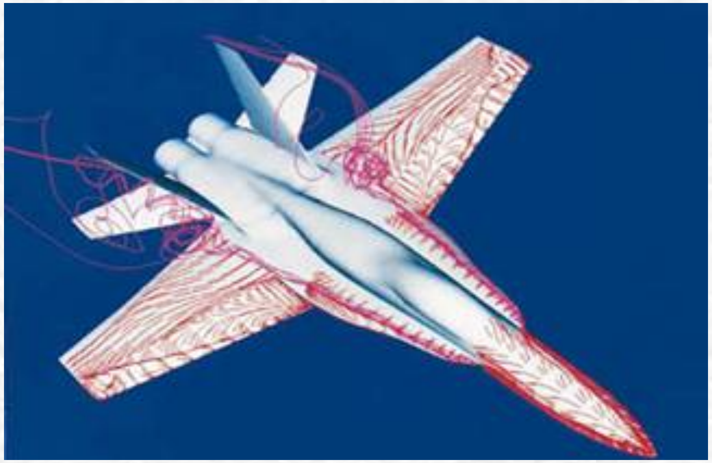
Flow around cooling towers

Le applicazioni più diffuse…




200.0000
-100.0000
-400.0000
-700.0000
-1000.0000


520 Kelvin
255 Kelvin
300 Kelvin
y
z

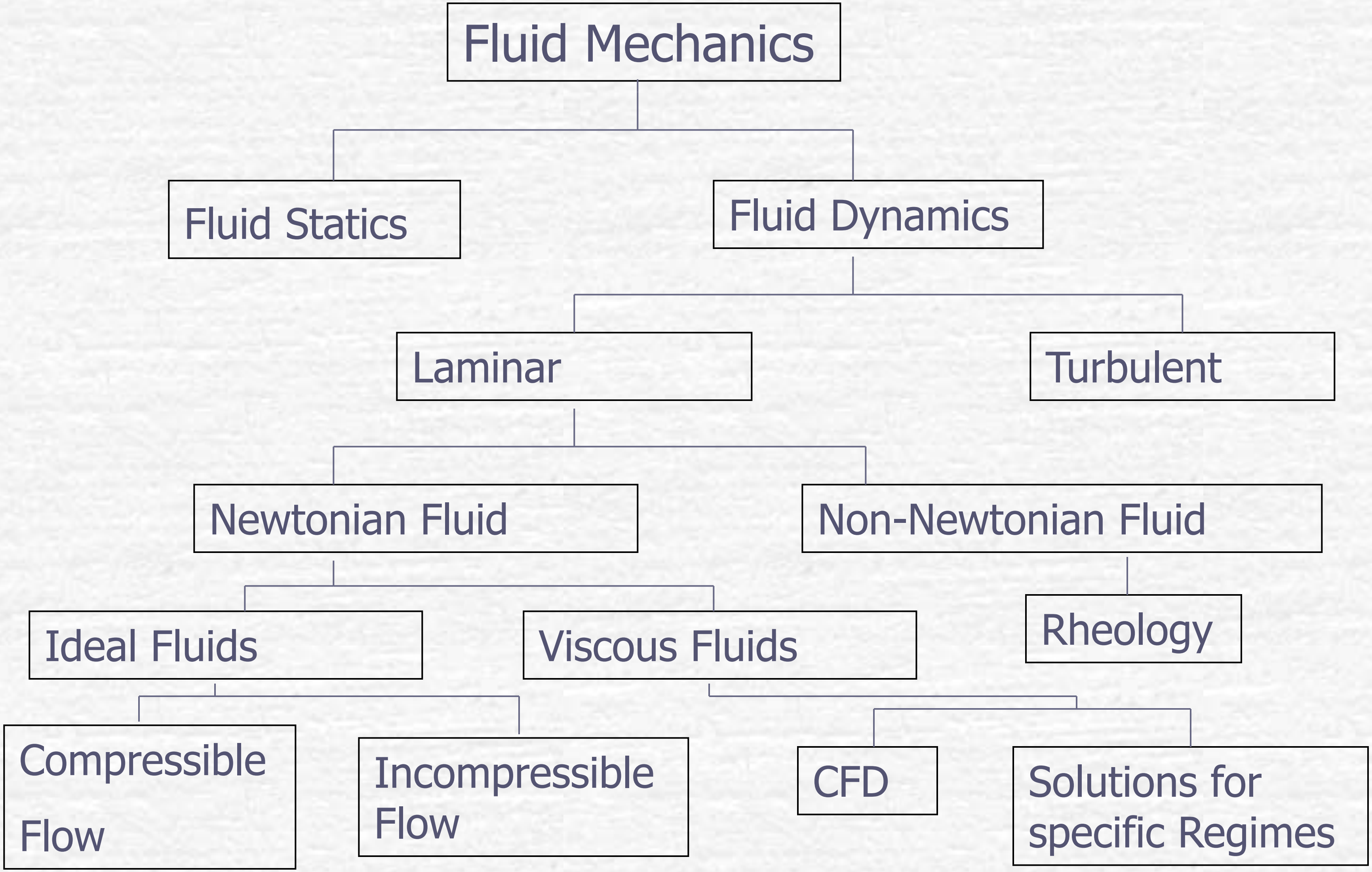


Modelli

- Il Modello è l'insieme delle correlazioni matematiche che vengono impiegate nel codice di calcolo, che ha il compito di "integrare" le equazioni del modello

- Il caso più generale è quello termofluidodinamico:
  -Equazioni di equilibrio
  -Equazioni di congruenza
  -Equazioni del legame costitutivo
  -Equazioni di continuità
  -Equazioni del moto fluido
  -Equazioni dei gas (perfetti e non)
  -Equazioni di trasmissione del calore

Richiami teorici:Concetti di base



Fluid Mechanics

Fluid Statics

Fluid Dynamics

Laminar

Turbulent

Newtonian Fluid

Non-Newtonian Fluid

Ideal Fluids

Viscous Fluids

Rheology

Compressible Flow

Incompressible Flow

CFD

Solutions for specific Regimes

Richiami teorici (equazioni)

- Navier-Stokes equations (3D in Cartesian coordinates)

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}+\rho w\frac{\partial u}{\partial z}=-\frac{\partial\hat{p}}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right]$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}+\rho w\frac{\partial v}{\partial z}=-\frac{\partial\hat{p}}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right]$$

$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}=-\frac{\partial\hat{p}}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right]$$

Local acceleration

Convection

Piezometric pressure gradient

Viscous terms

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

Continuity equation

$$p=\rho RT$$

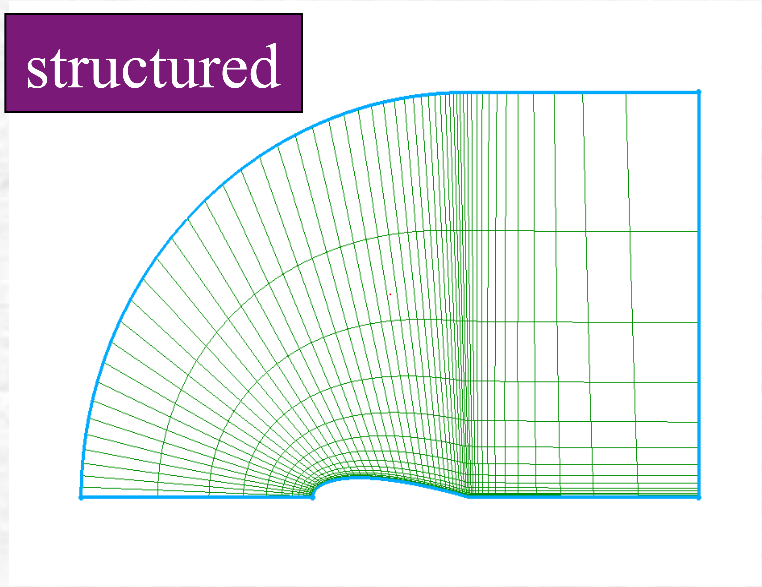
Equation of state

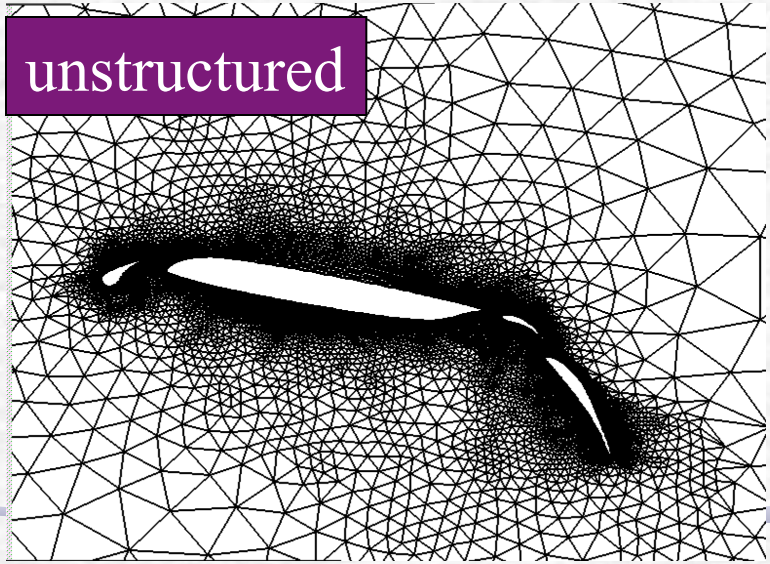
$$R\frac{D^2R}{Dt^2}+\frac{3}{2}\left(\frac{DR}{Dt}\right)^2=\frac{p_v-p}{\rho_L}$$

Rayleigh Equation

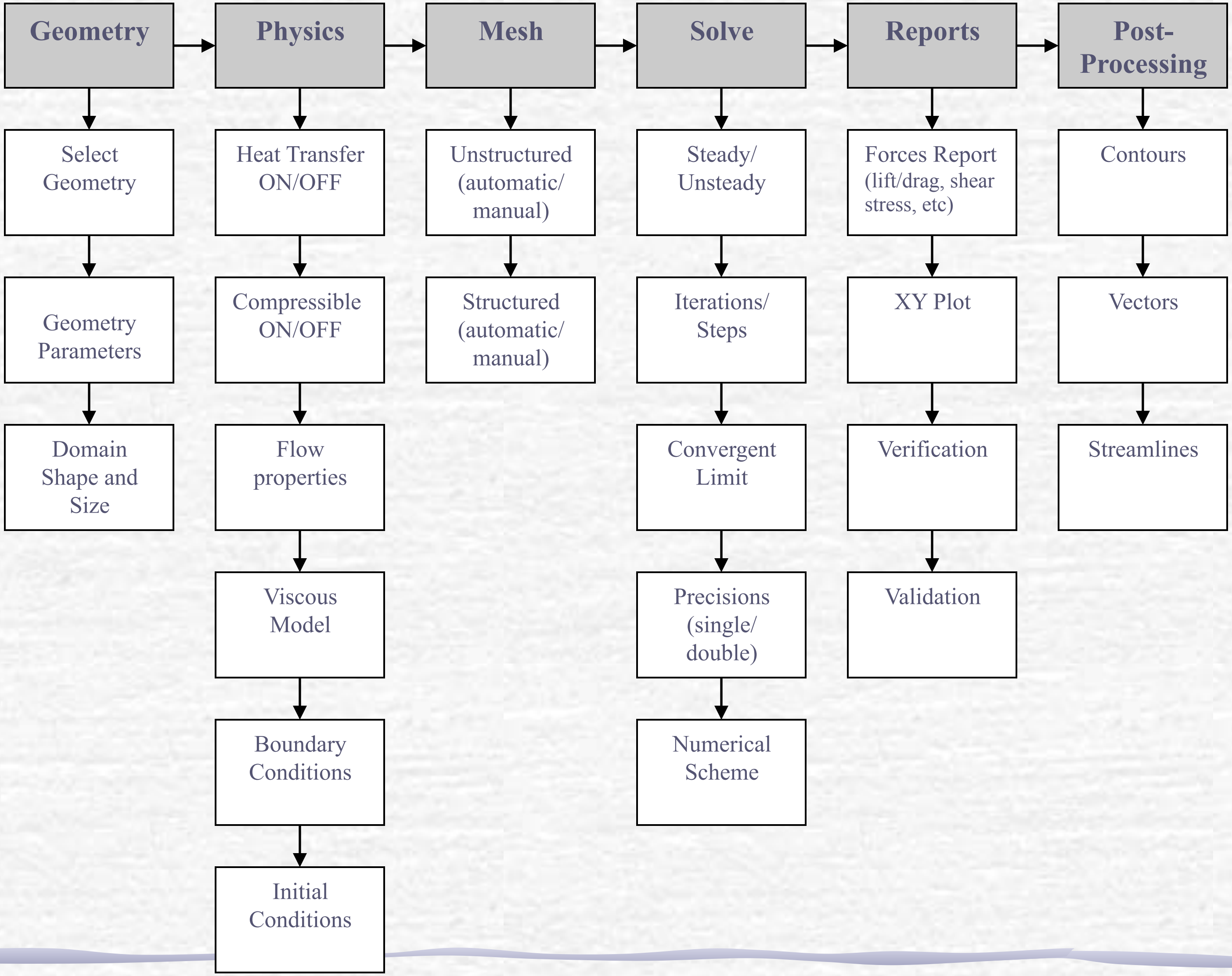
Metodi numerici (mesh)

- Il fatto che il metodo sia “discreto” significa che da una realtà continua si passa ad una realtà con un numero finito di parametri: il numero di nodi non può essere infinito!
  - Schemi di calcolo
    - ❑ Differenze finite: mesh strutturata
    - ❑ Volumi finiti: mesh strutturata/non strutturata
  - Applicazioni
    - ❑ Geometrie semplici: differenze finite
    - ❑ Geometrie complesse: volumi finiti


structured

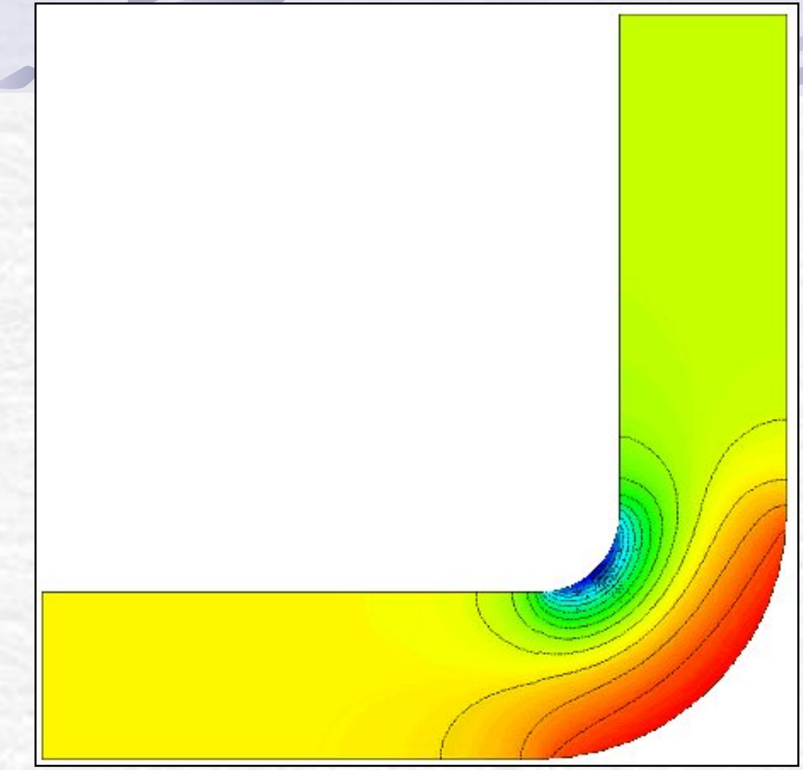

unstructured

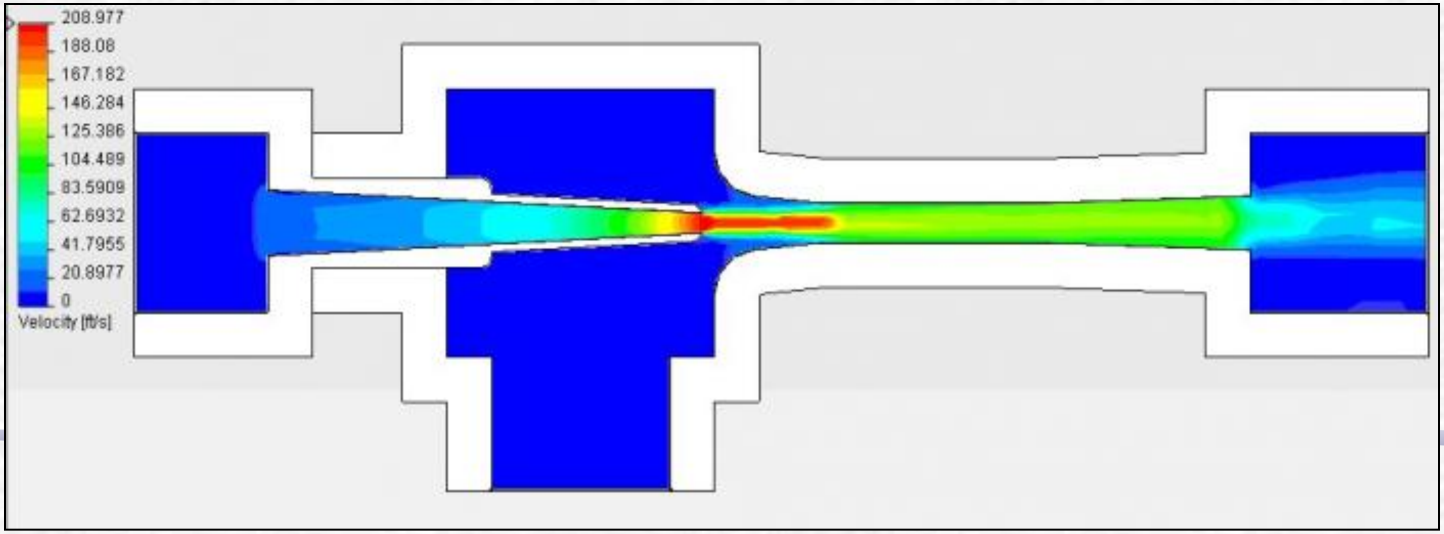
CFD:Procedimendo



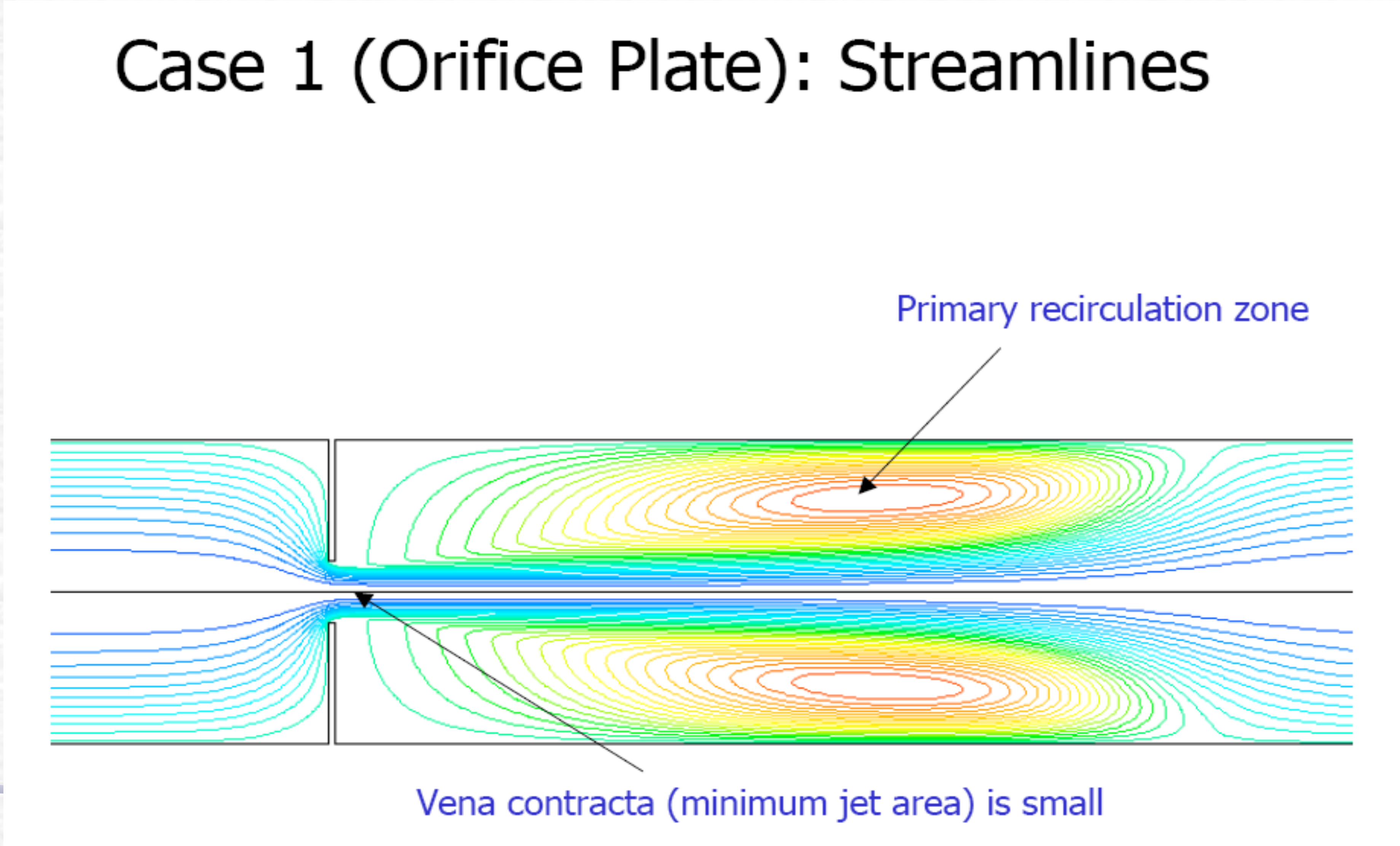
- **Geometry**
  - Select Geometry
  - Geometry Parameters
  - Domain Shape and Size
- **Physics**
  - Heat Transfer ON/OFF
  - Compressible ON/OFF
  - Flow properties
  - Viscous Model
  - Boundary Conditions
  - Initial Conditions
- **Mesh**
  - Unstructured (automatic/ manual)
  - Structured (automatic/ manual)
- **Solve**
  - Steady/ Unsteady
  - Iterations/ Steps
  - Convergent Limit
  - Precisions (single/ double)
  - Numerical Scheme
- **Reports**
  - Forces Report (lift/drag, shear stress, etc)
  - XY Plot
  - Verification
  - Validation
- **Post-Processing**
  - Contours
  - Vectors
  - Streamlines

Esempi di base




208.977
188.08
167.182
146.284
125.386
104.489
83.5909
62.6932
41.7955
20.8977
0
Velocity [ft/s]

Esempio condotto 1

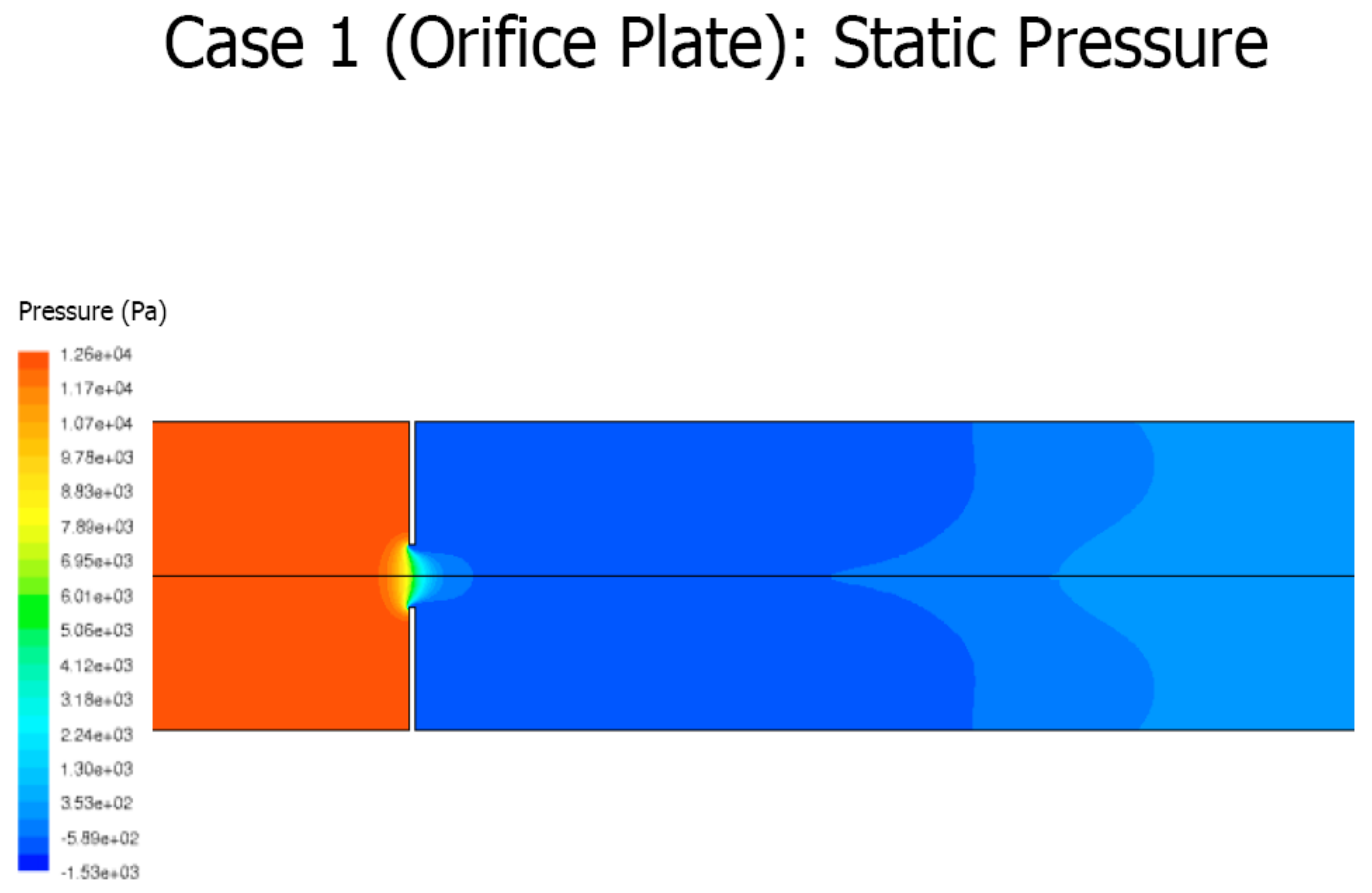


Case 1 (Orifice Plate): Streamlines

Primary recirculation zone

Vena contracta (minimum jet area) is small

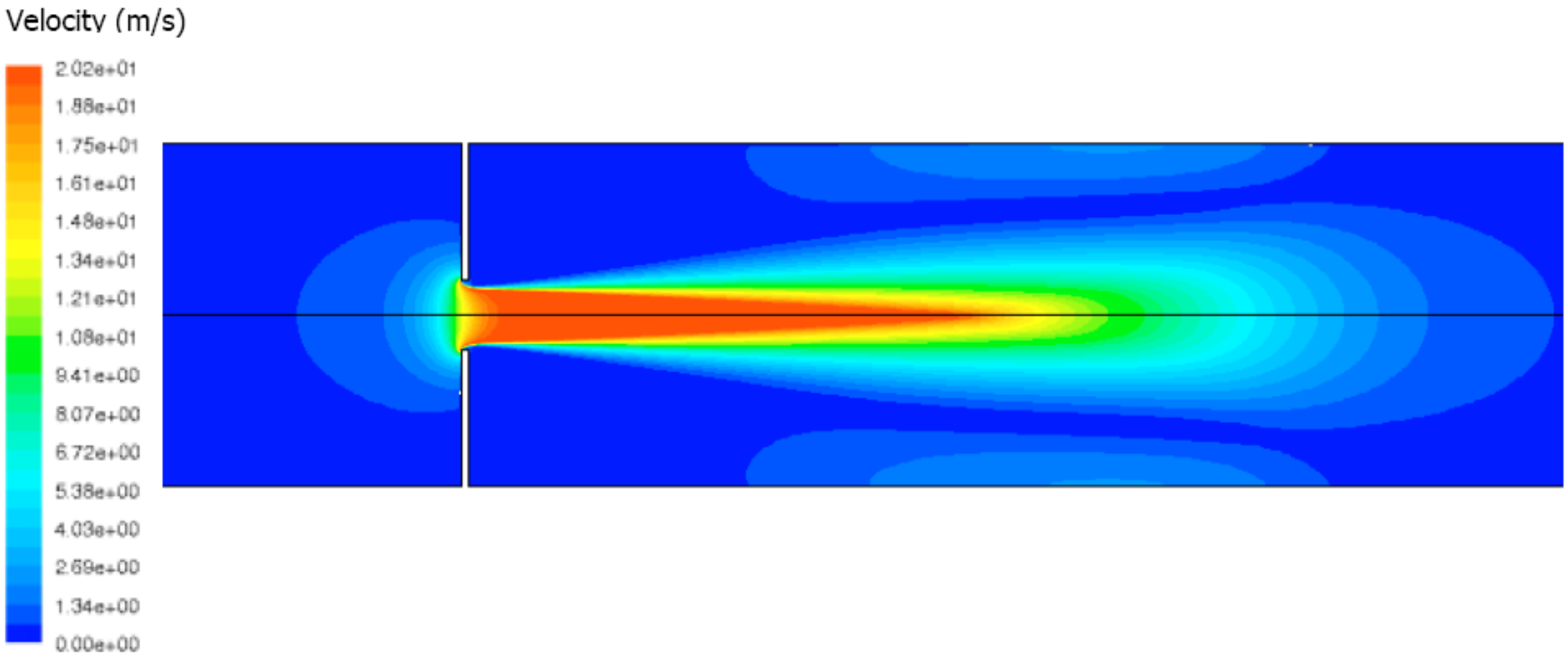
Esempio condotto 1



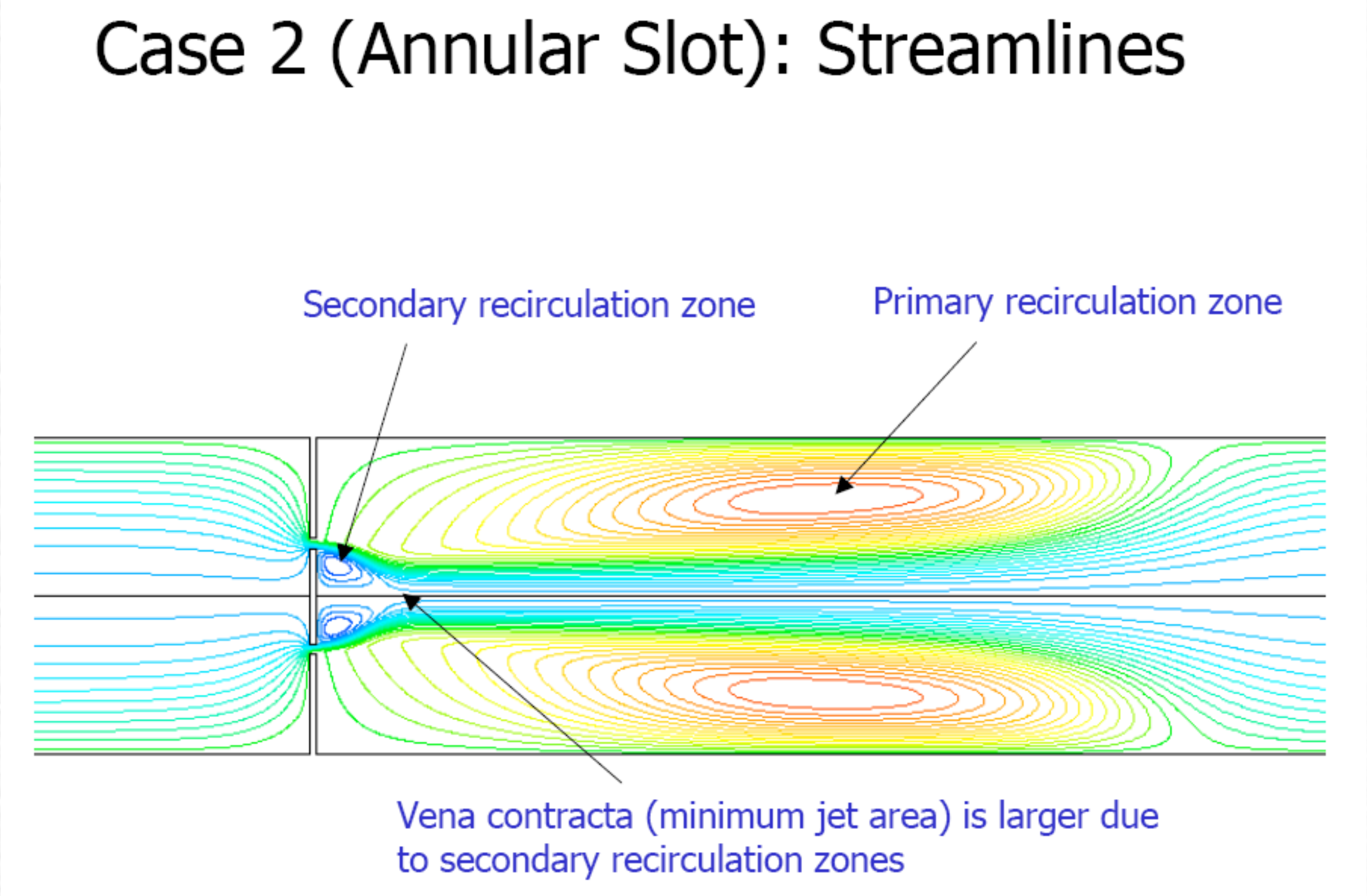
Case 1 (Orifice Plate): Static Pressure

Esempio condotto 1

Case 1 (Orifice Plate): Velocity Magnitude



Esempio condotto 2



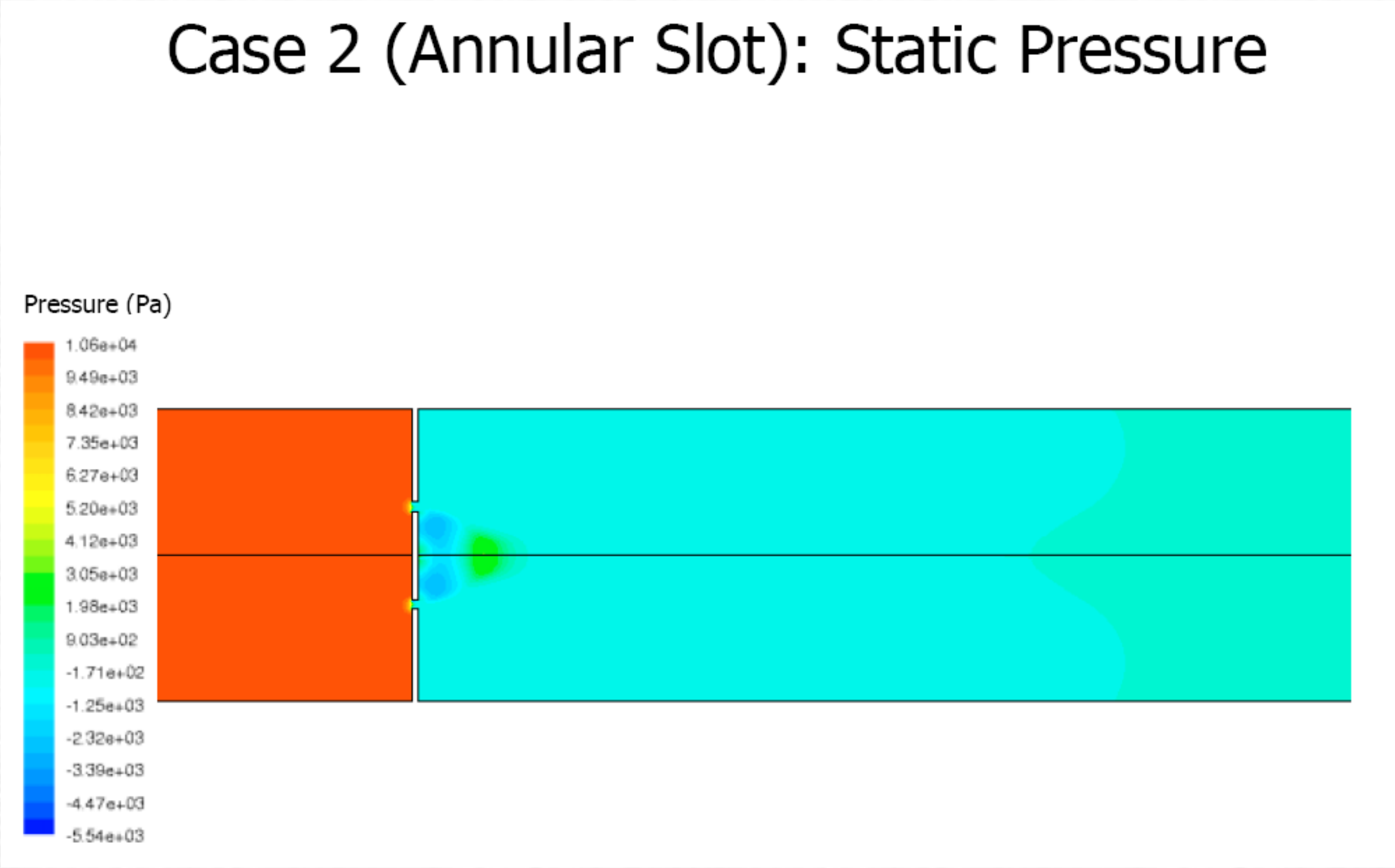
Case 2 (Annular Slot): Streamlines

Secondary recirculation zone

Primary recirculation zone

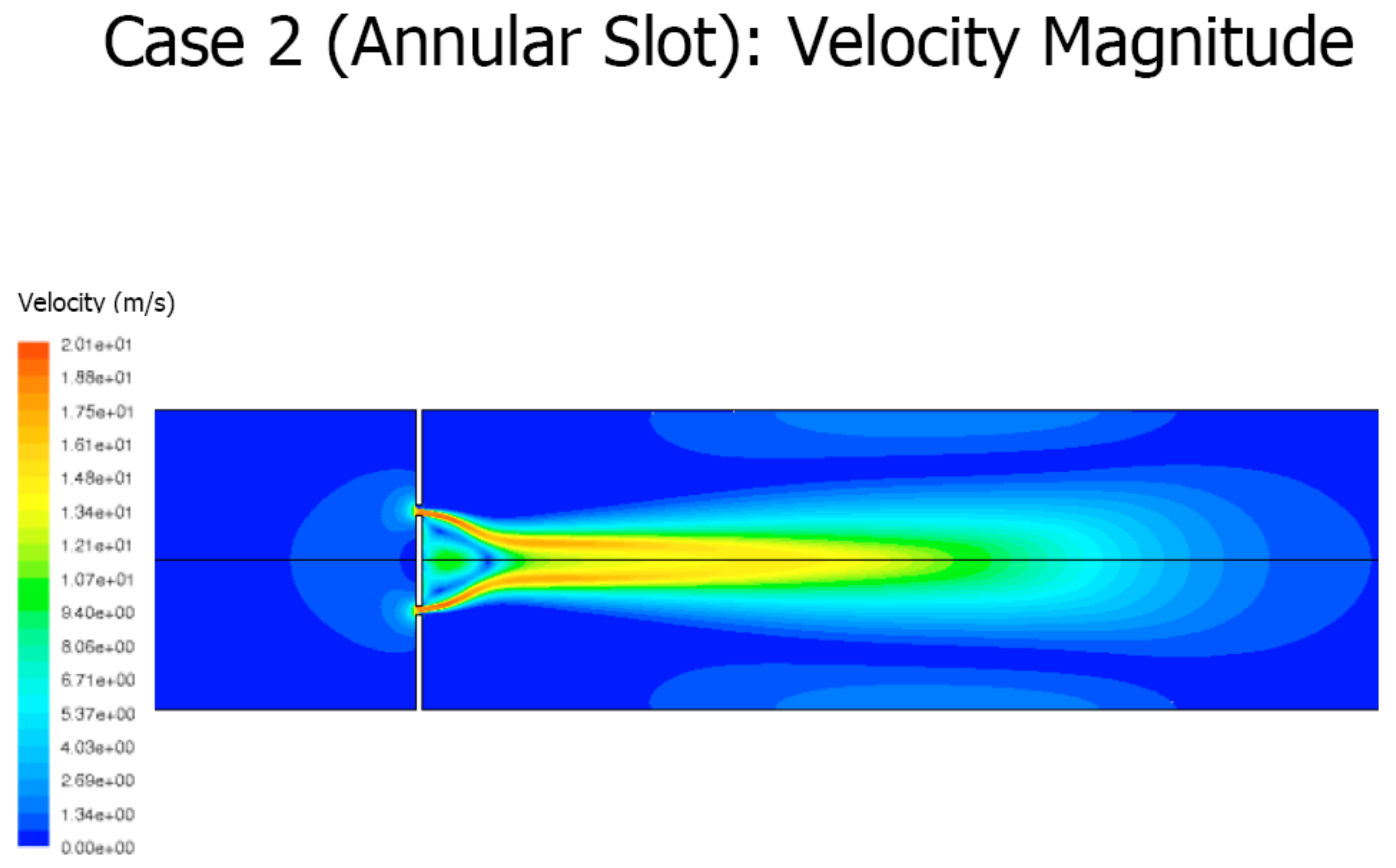
Vena contracta (minimum jet area) is larger due to secondary recirculation zones

Esempio condotto 2



Case 2 (Annular Slot): Static Pressure

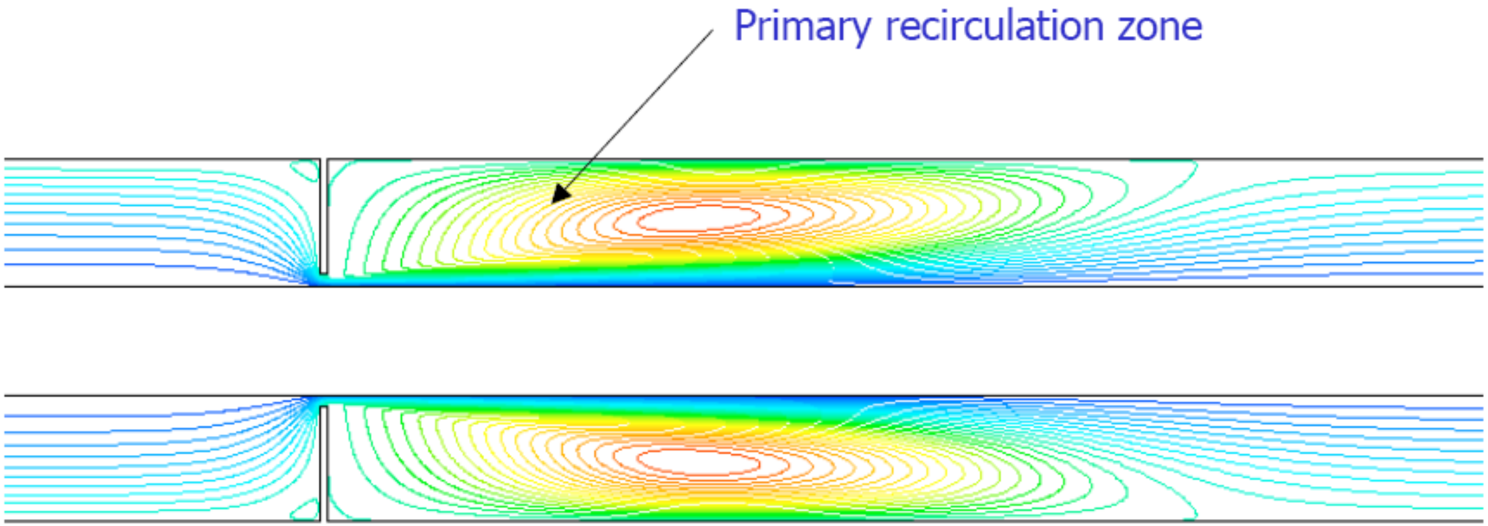
Esempio condotto 2



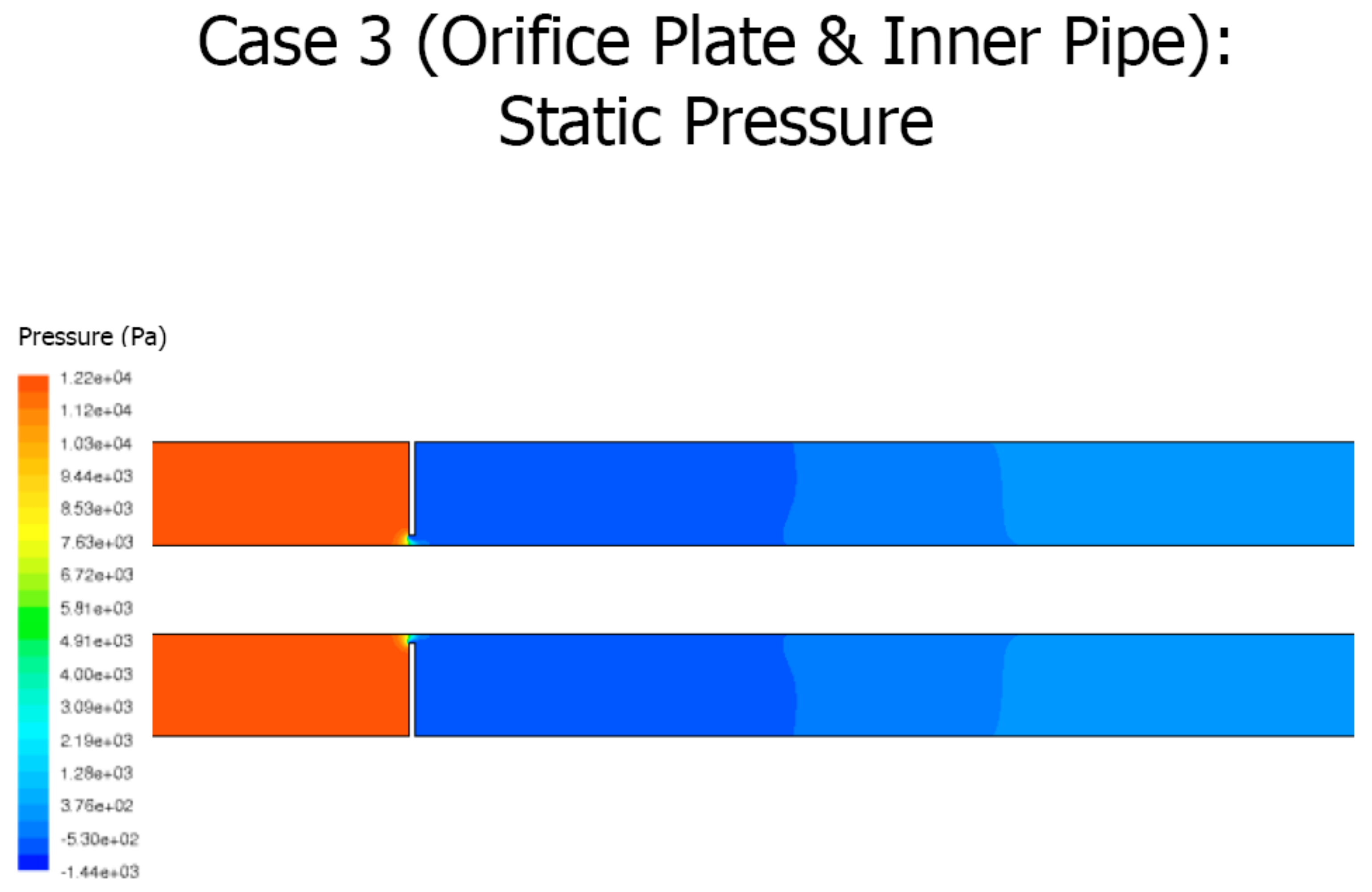
Case 2 (Annular Slot): Velocity Magnitude

Esempio condotto 3

Case 3 (Orifice Plate & Inner Pipe): Streamlines

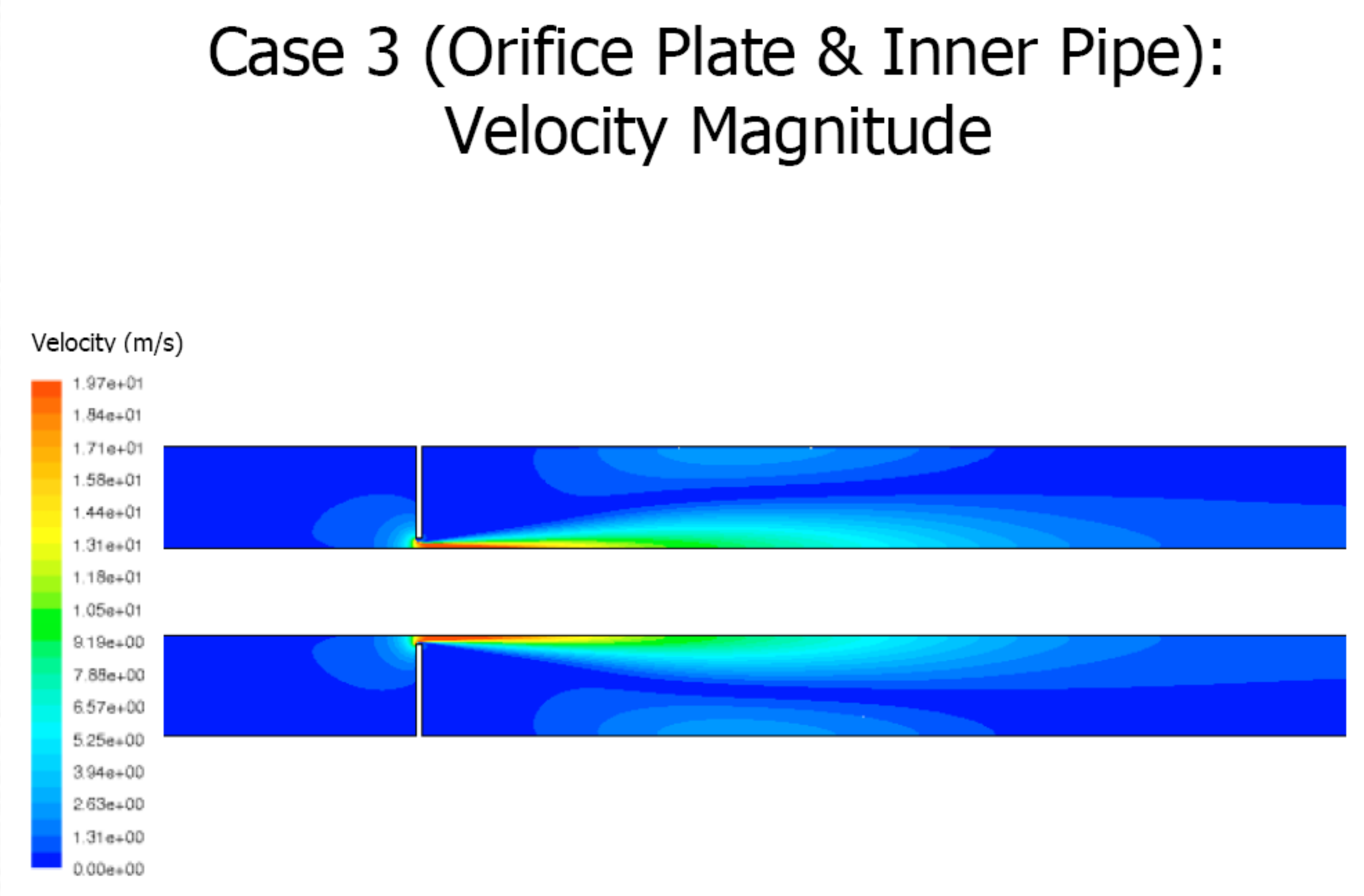

Primary recirculation zone

Esempio condotto 3



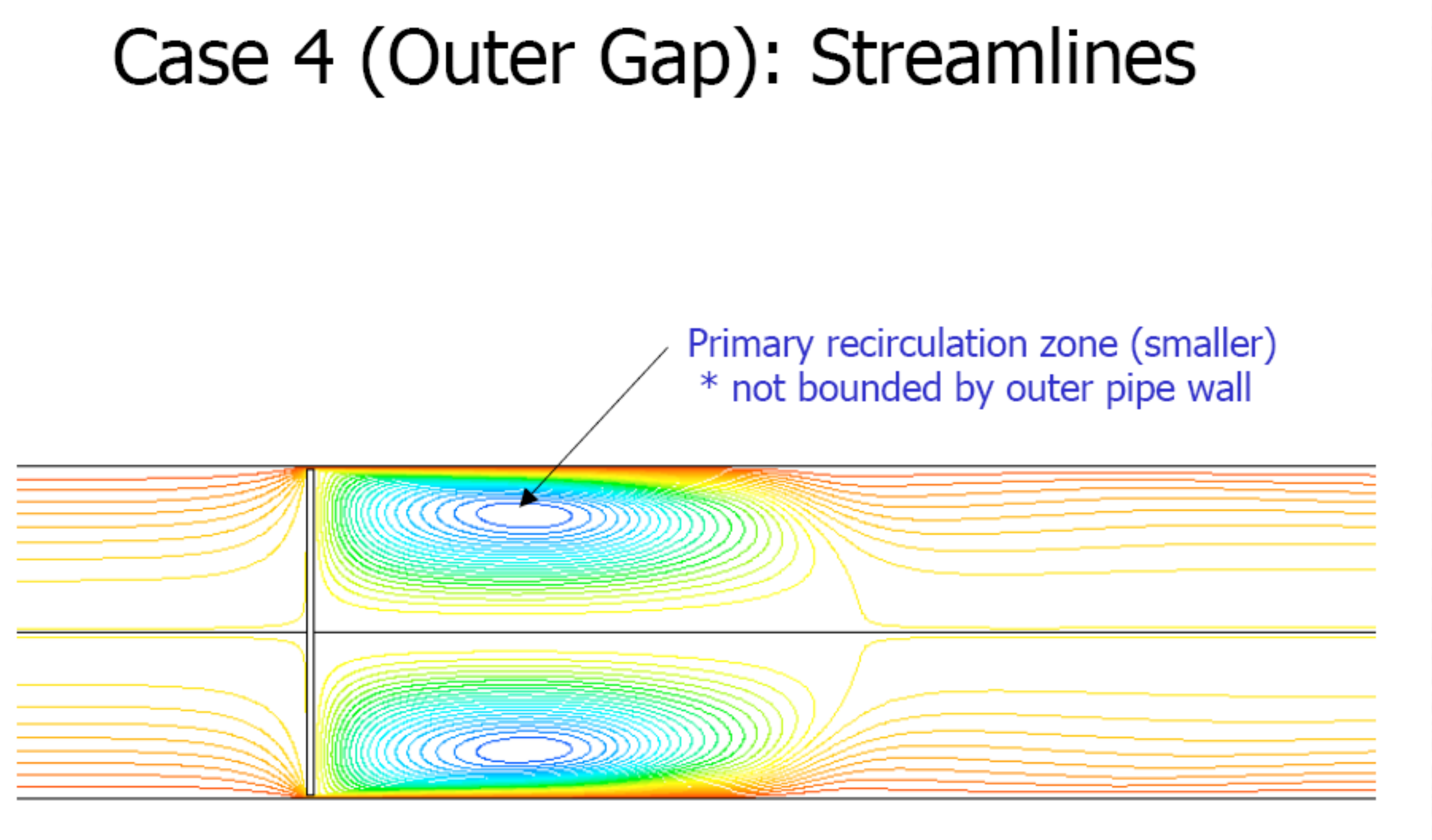
Case 3 (Orifice Plate & Inner Pipe):
Static Pressure

Esempio condotto 3



Case 3 (Orifice Plate & Inner Pipe):
Velocity Magnitude

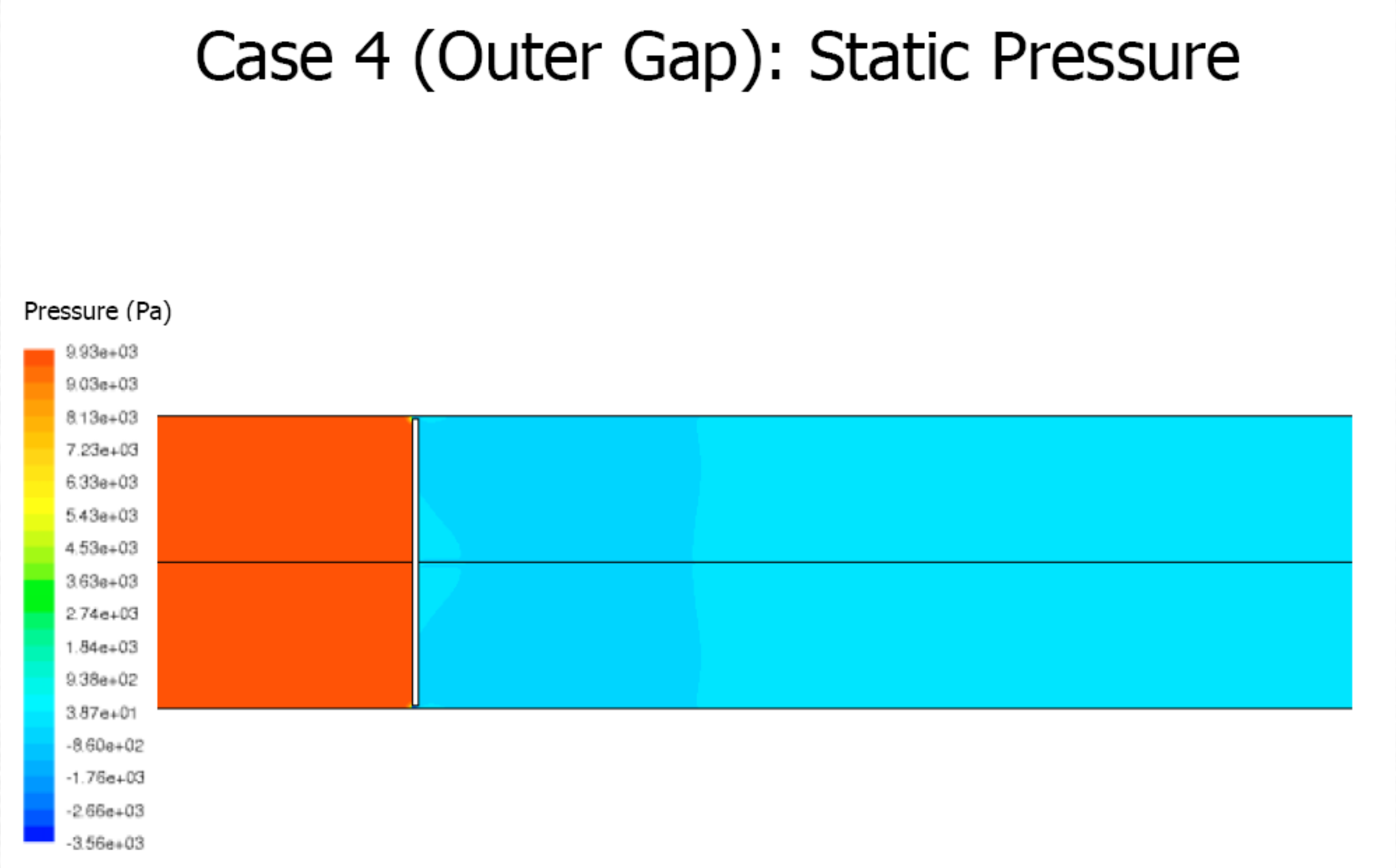
Esempio condotto 4



Case 4 (Outer Gap): Streamlines

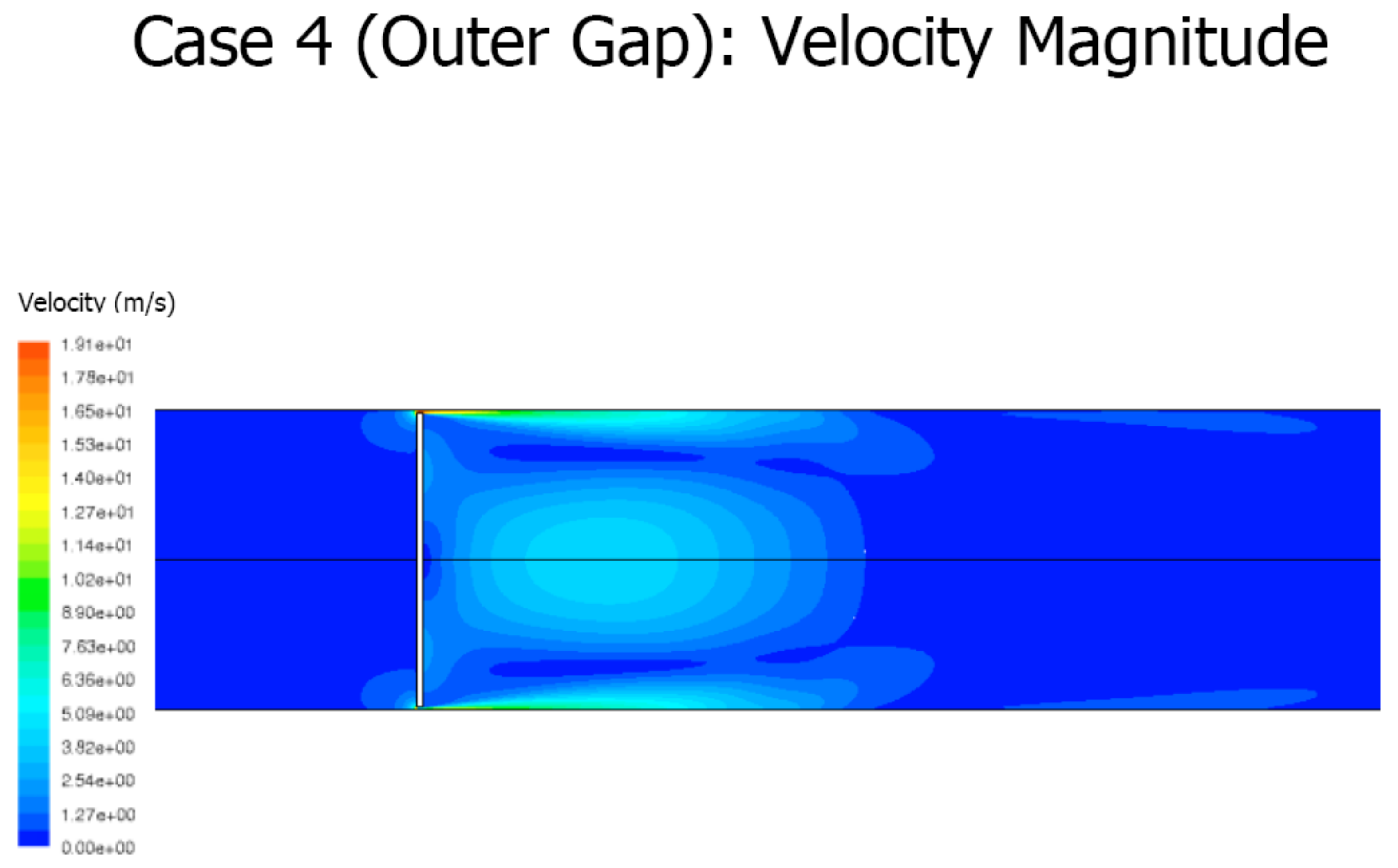
Primary recirculation zone (smaller)
* not bounded by outer pipe wall

Esempio condotto 4



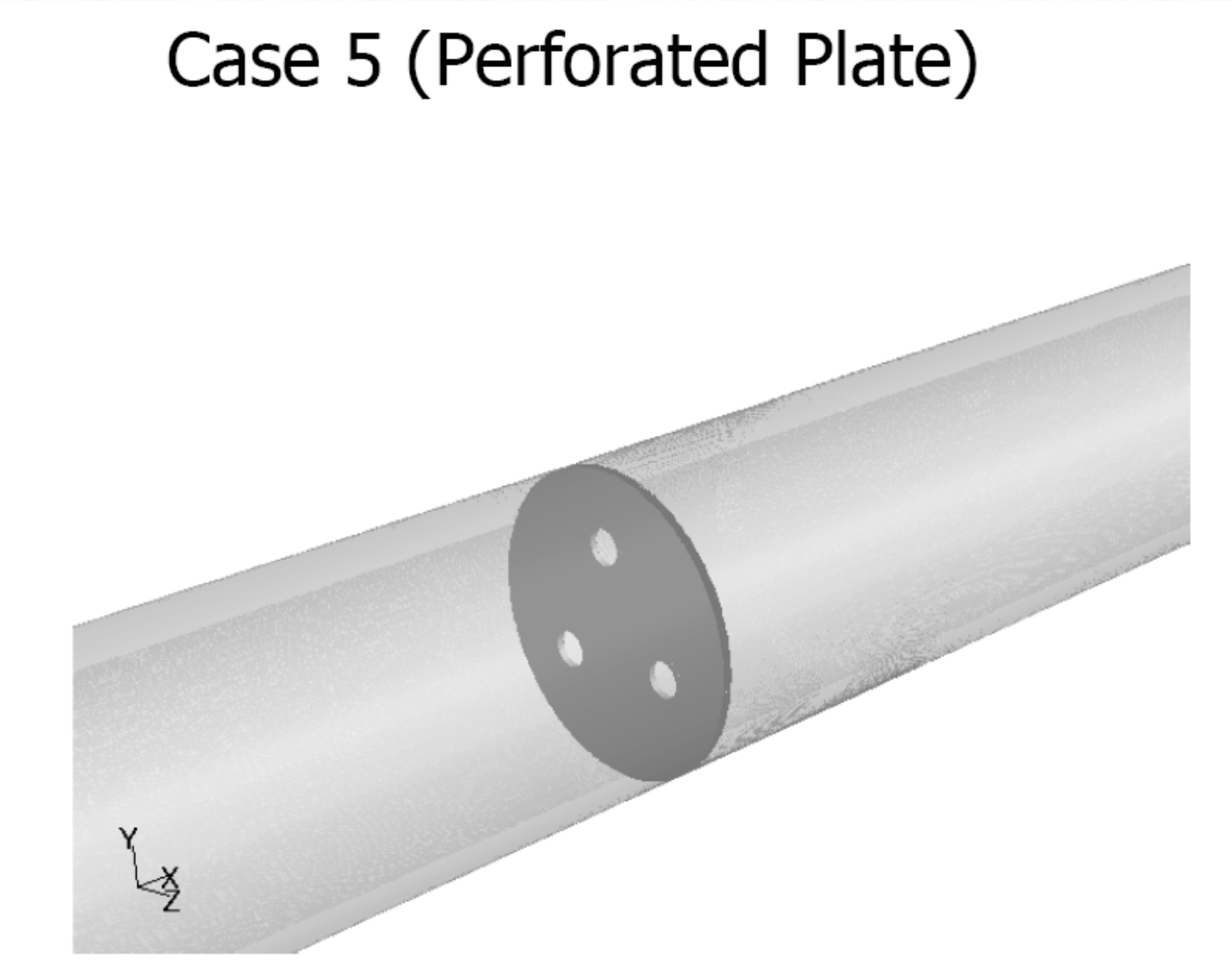
Case 4 (Outer Gap): Static Pressure

Esempio condotto 4



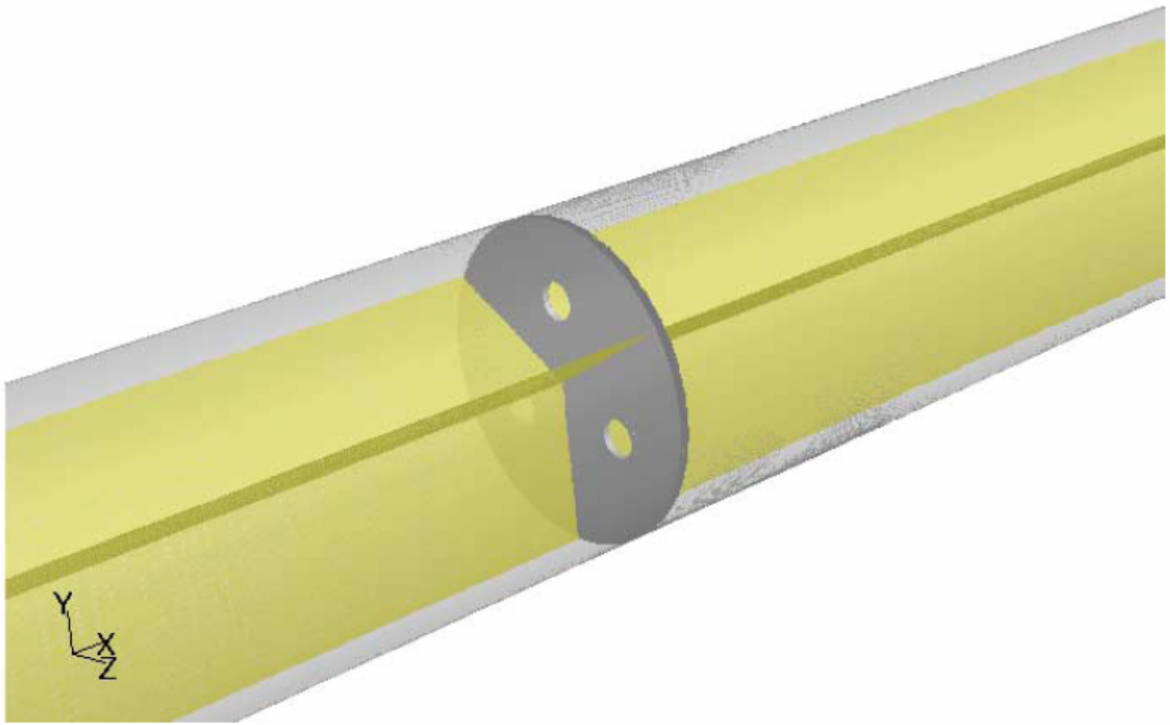
Case 4 (Outer Gap): Velocity Magnitude

Esempio condotto 5

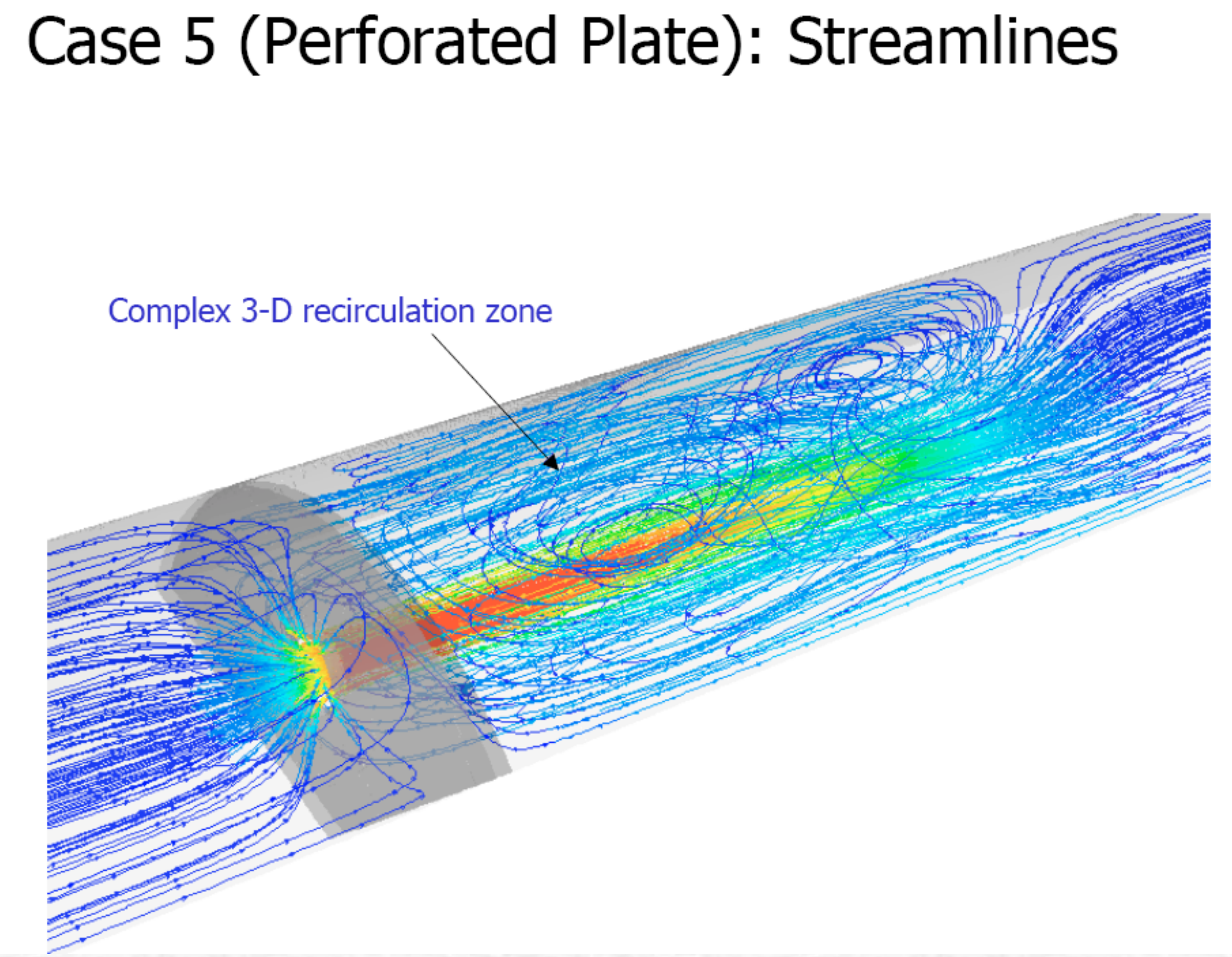

Case 5 (Perforated Plate)
Y
X
Z

Esempio condotto 5

Case 5 (Perforated Plate)



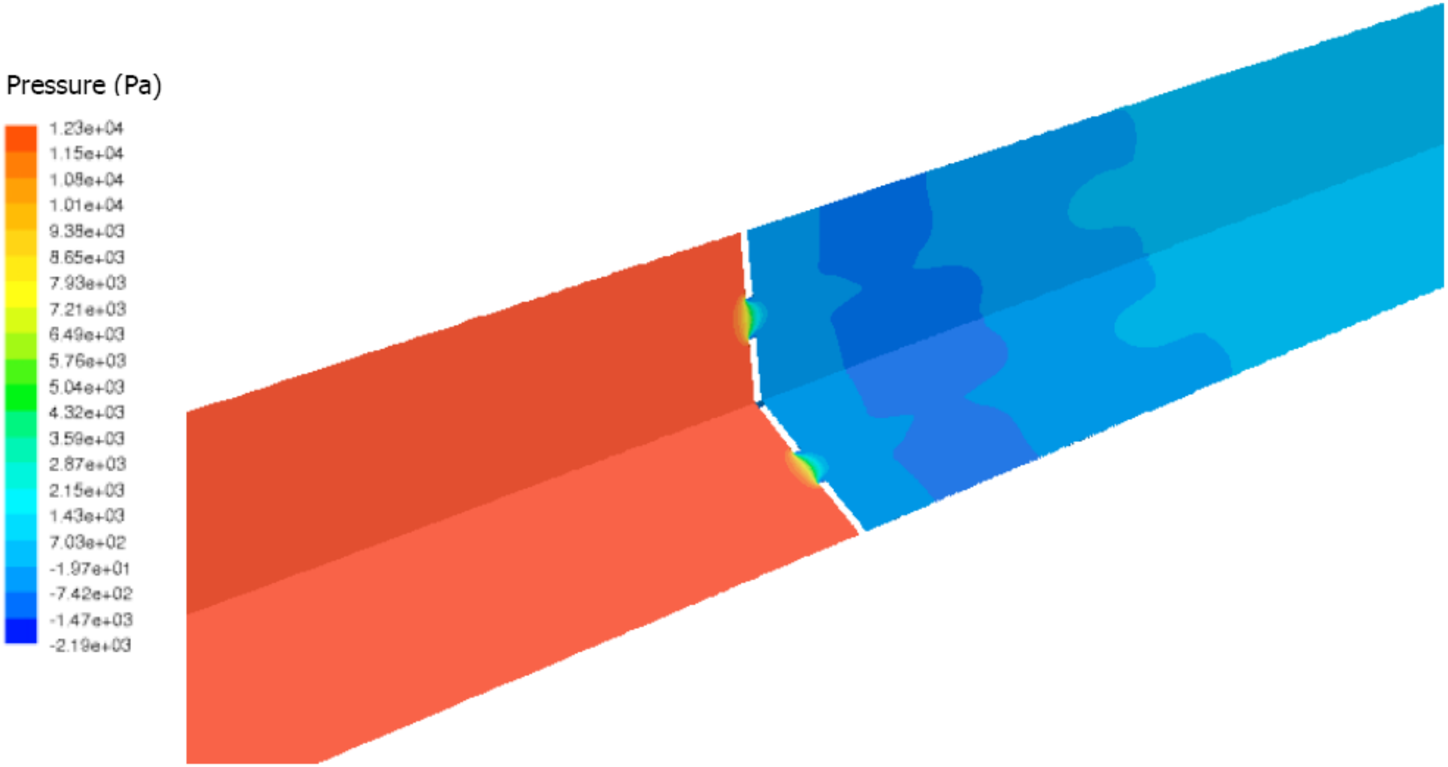
Esempio condotto 5



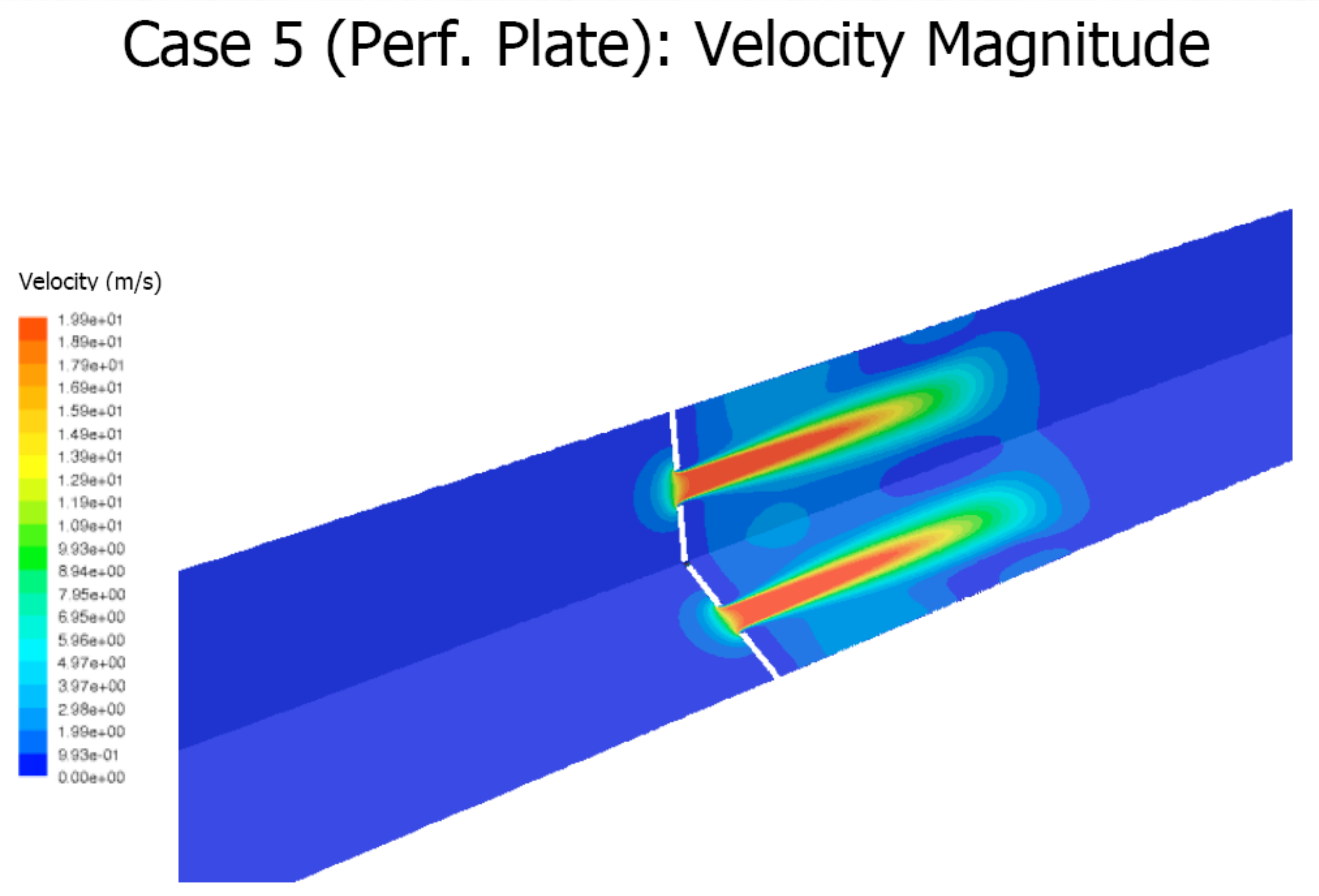
Case 5 (Perforated Plate): Streamlines

Complex 3-D recirculation zone

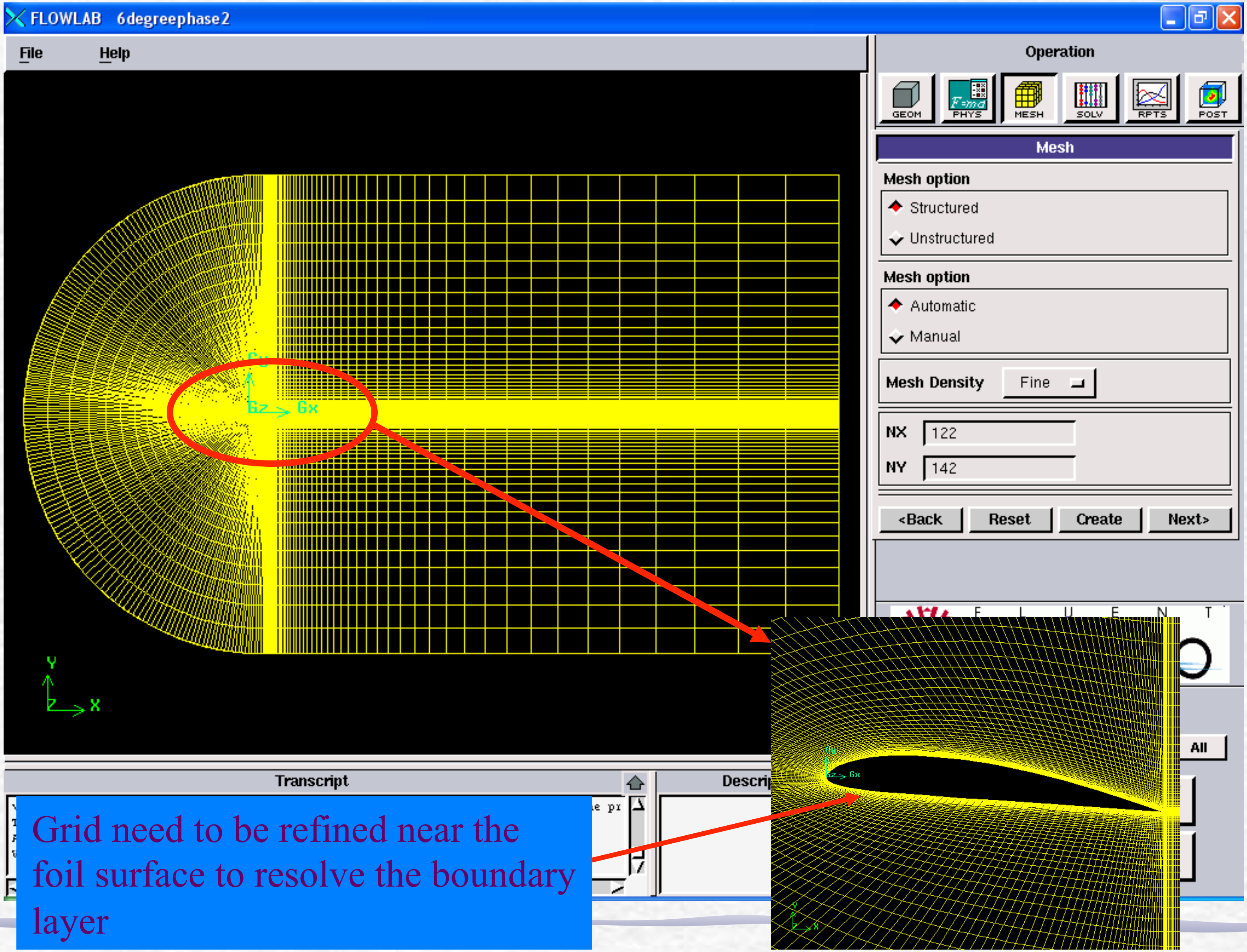
Esempio condotto 5


Case 5 (Perforated Plate): Static Pressure

Esempio condotto 5



Esempio wing (Mesh)


FLOWLAB 6degreephase2
File
Help
Operation
GEOM
PHYS
MESH
SOLV
RPTS
POST
Mesh
<Back
Reset
Create
Next>
Transcript
Descri
All
Y
X
Gx
Gz

Mesh option
- ● Structured
- ○ Unstructured

Mesh option
- ● Automatic
- ○ Manual

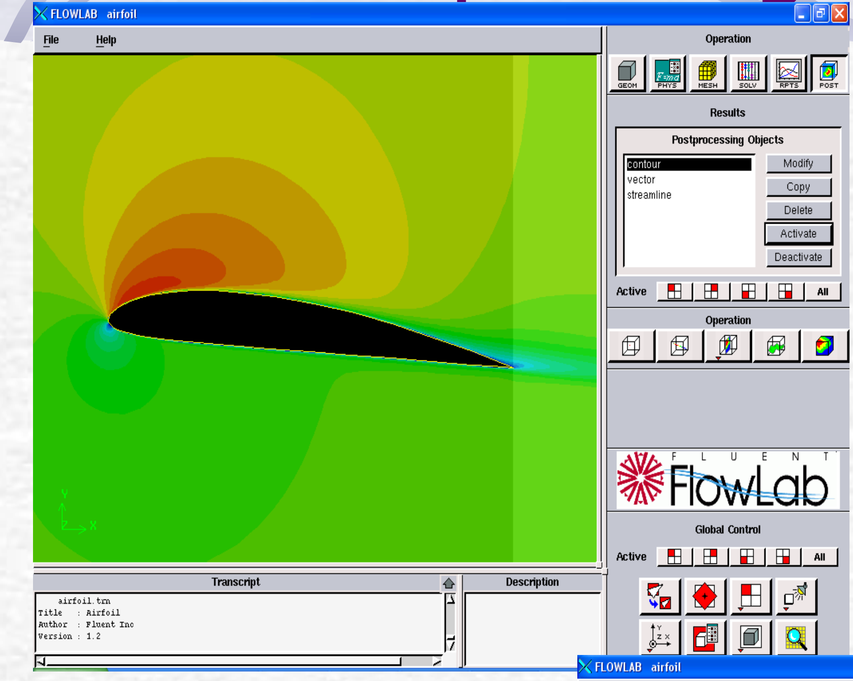
Mesh Density: Fine

NX 122

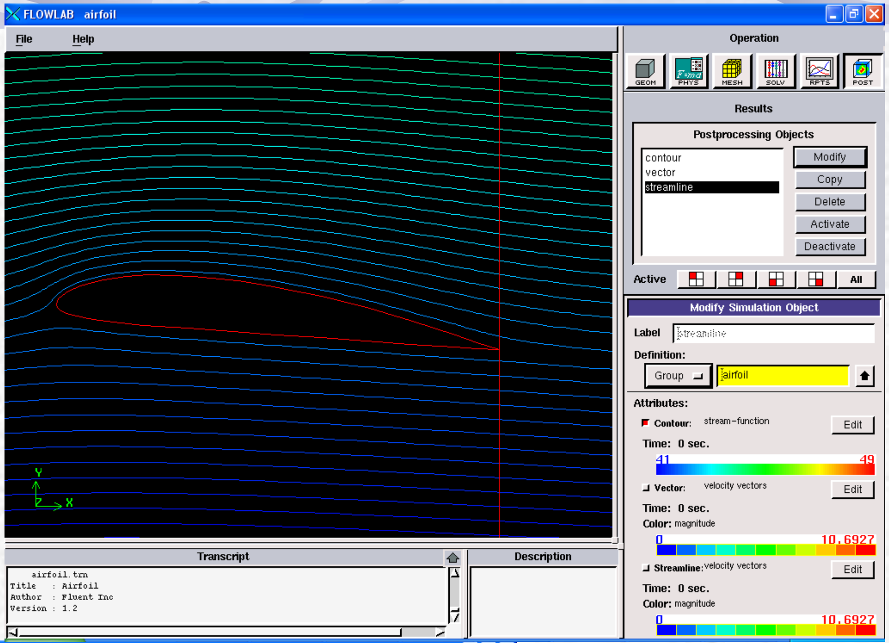
NY 142

Grid need to be refined near the foil surface to resolve the boundary layer

Esempio wing (Post-processing)


FLOWLAB airfoil
File
Help
Operation
GEOM
PHYS
MESH
SOLV
RPTS
POST
Results
Postprocessing Objects
contour
vector
streamline
Modify
Copy
Delete
Activate
Deactivate
Active
All
Operation
F L U E N T
FlowLab
Global Control
Active
All
Y
X
Transcript
Description

```
airfoil.trn
Title   : Airfoil
Author  : Fluent Inc
Version : 1.2
```


FLOWLAB airfoil
File
Help
Operation
GEOM
PHYS
MESH
SOLV
RPTS
POST
Results
Postprocessing Objects
contour
vector
streamline
Modify
Copy
Delete
Activate
Deactivate
Active
All
Modify Simulation Object
Edit
Edit
Edit
Y
X
Transcript
Description

```
airfoil.trn
Title   : Airfoil
Author  : Fluent Inc
Version : 1.2
```

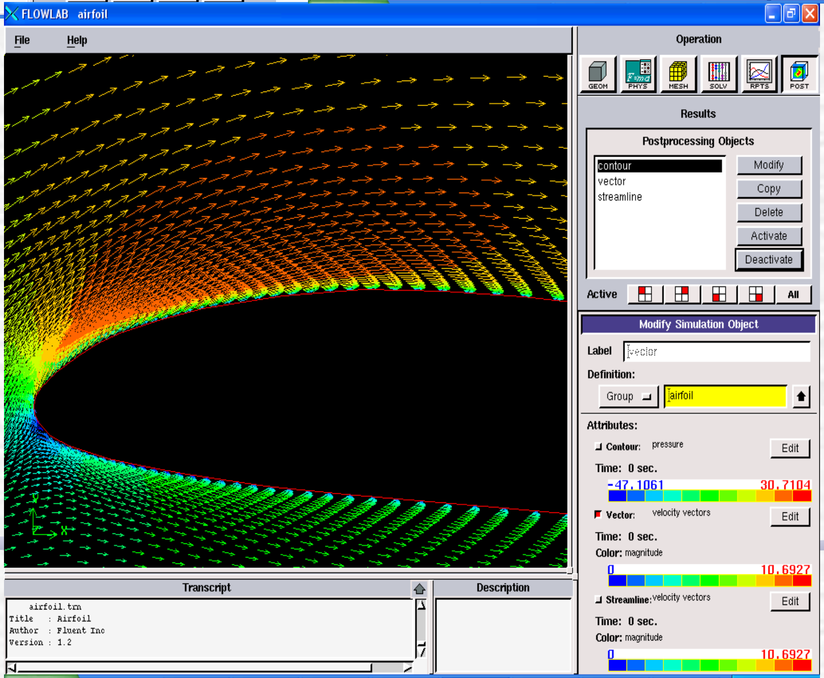
Label: streamline
Definition: Group airfoil

Attributes:

Contour: stream-function
Time: 0 sec.
41 49

Vector: velocity vectors
Time: 0 sec.
Color: magnitude
0 10.6927

Streamline: velocity vectors
Time: 0 sec.
Color: magnitude
0 10.6927


FLOWLAB airfoil
File
Help
Operation
GEOM
PHYS
MESH
SOLV
RPTS
POST
Results
Postprocessing Objects
contour
vector
streamline
Modify
Copy
Delete
Activate
Deactivate
Active
All
Modify Simulation Object
Edit
Edit
Edit
Y
X
Transcript
Description

```
airfoil.trn
Title   : Airfoil
Author  : Fluent Inc
Version : 1.2
```

Label: vector
Definition: Group airfoil

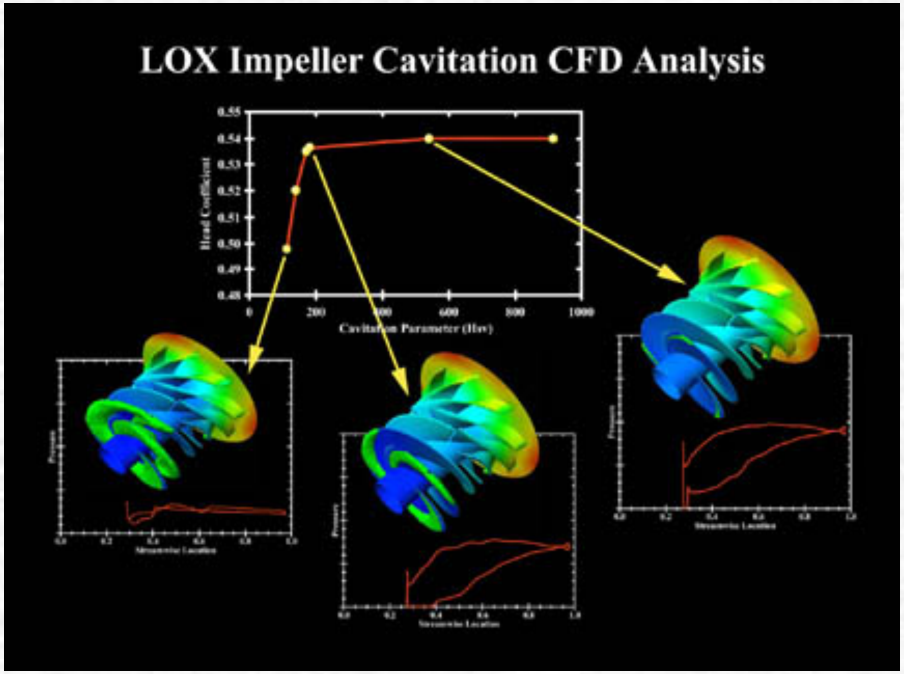
Attributes:

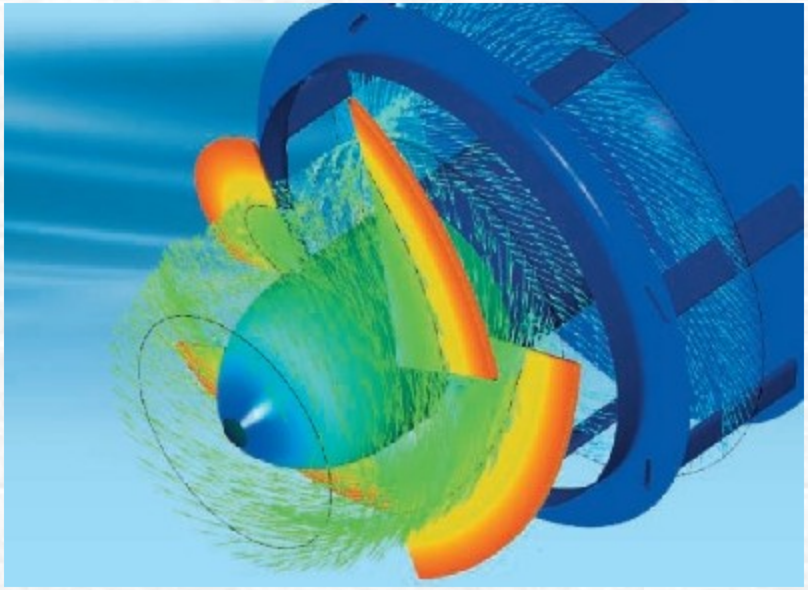
Contour: pressure
Time: 0 sec.
-47.1061 30.7104

Vector: velocity vectors
Time: 0 sec.
Color: magnitude
0 10.6927

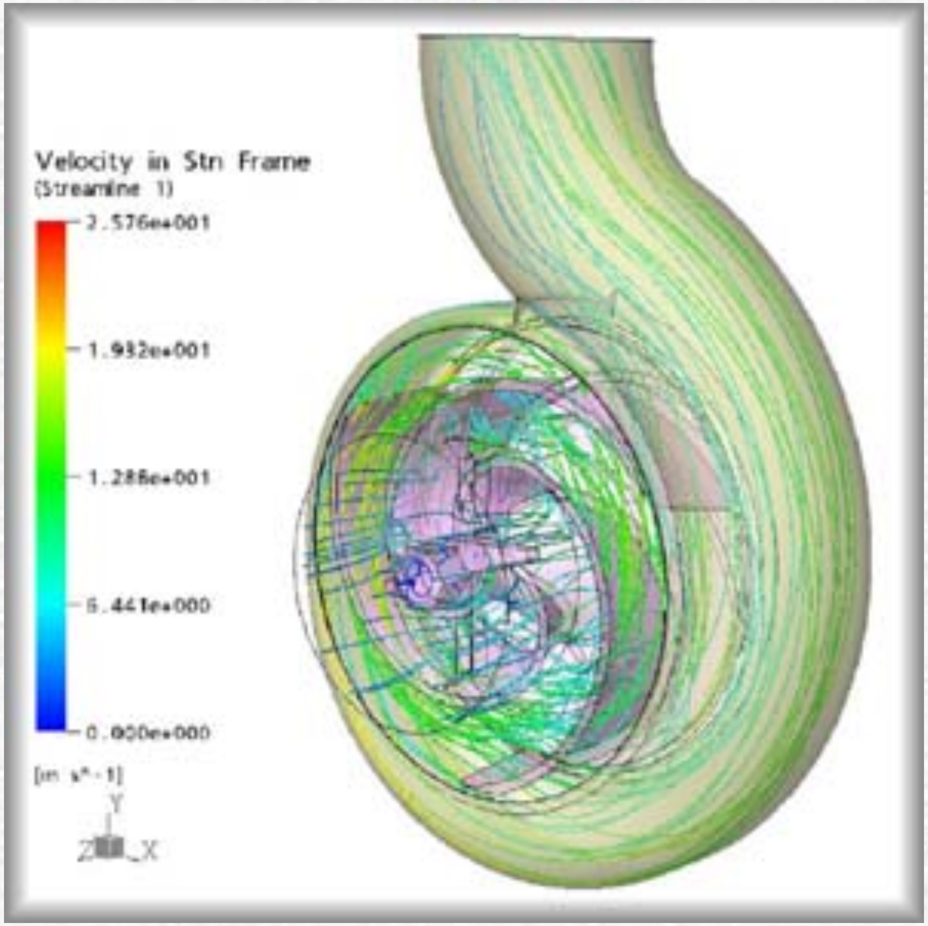
Streamline: velocity vectors
Time: 0 sec.
Color: magnitude
0 10.6927

Esempi su geometrie complesse

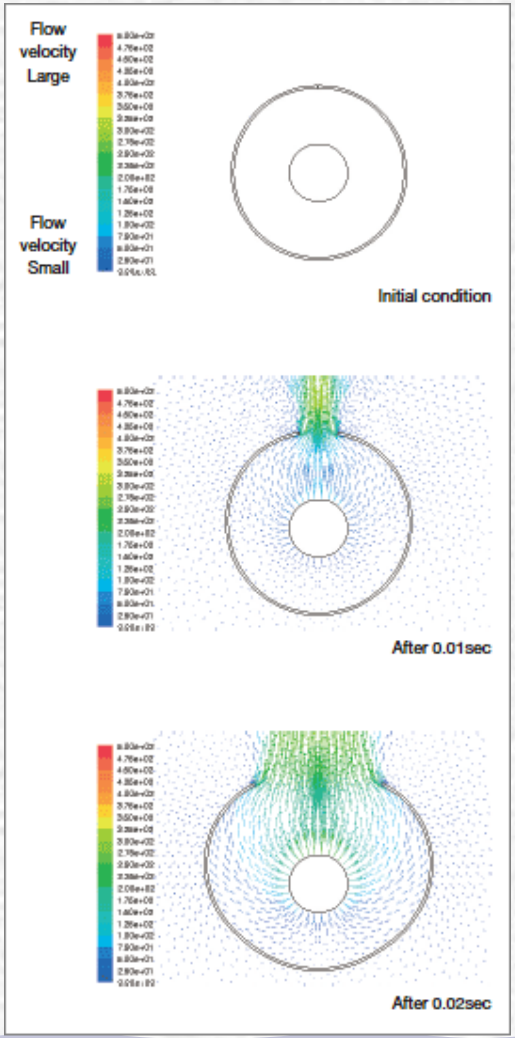




Esempi su geometrie complesse


Velocity in Stn Frame
(Streamline 1)
2.576e+001
1.932e+001
1.288e+001
6.441e+000
0.000e+000
[m s^-1]
Y
Z
X

Esempio non stazionario



Conclusioni:Cose da ricordare…

1- CFD = FEM “generalizzato”

2- Numero di equazioni impegnativo per hardware e software

3- Preparazione teorica dell’utente

4- Analisi di elevato onere computazionale

5- Analisi che richiede “tempo”

6- Costi tendenzialmente elevati