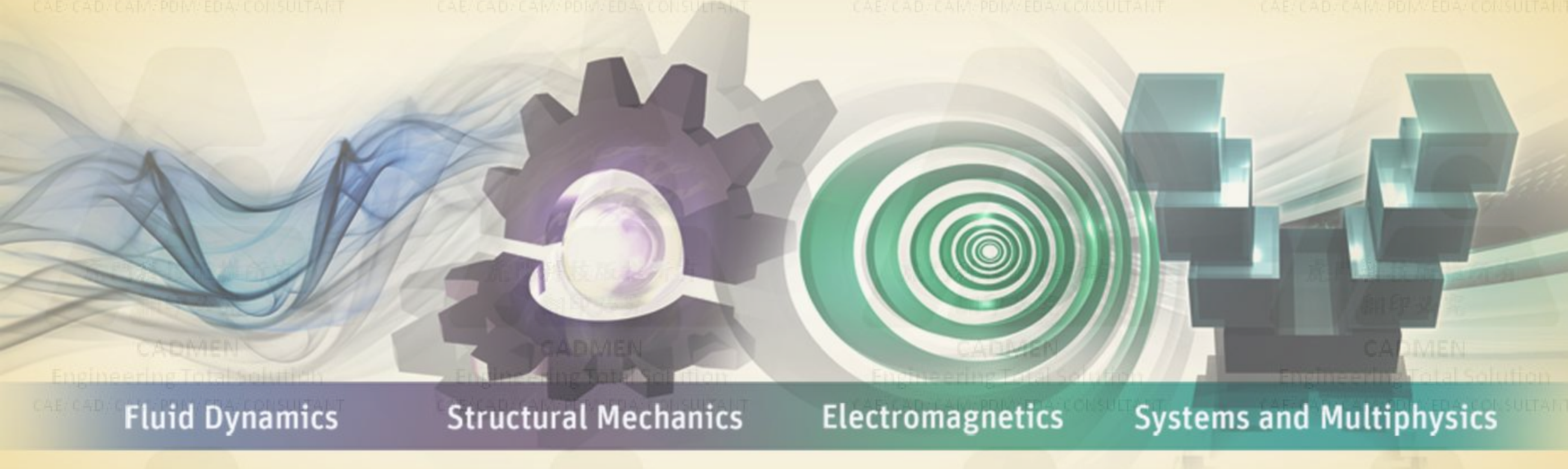
ANSYS

Realize Your Product Promise™

# 化工製程單元電腦模擬分析



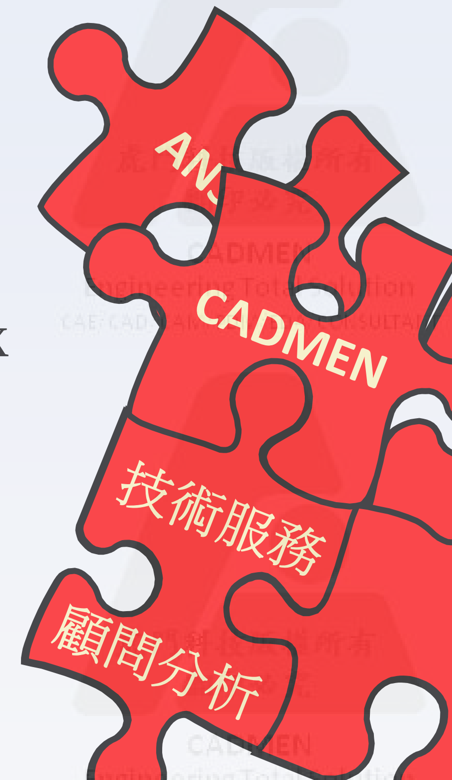

李龍育 Dragon
CFD技術副理
虎門科技

# 虎門科技 CADMEN

虎門科技股份有限公司，創立於1980年，提供客戶全球最優質的工程分析軟體ANSYS與技術服務


- 總公司 : 新北市板橋區
- 分公司 : 台中市文心路

- 結構強度分析
  ANSYS Mechanical
- 落摔分析
  ANSYS LS-DYNA
- 散熱與熱流場分析
  ANSYS FLUENT、 ICEPAK、CFX
- 電磁場分析
  ANSYS Emag、 Maxwell
- 多物理耦合分析



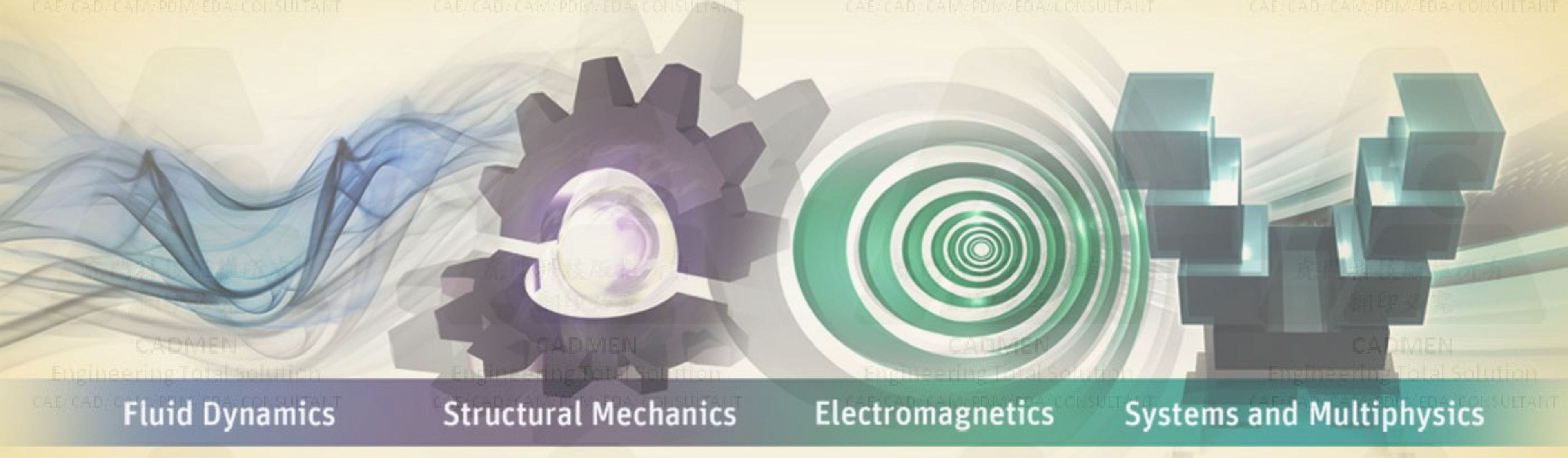

**Provider of Engineering Solutions and Methodology**

# 簡報大綱

- ANSYS FLUENT於化工產業應用簡介
- 氣體擴散反應範例介紹
- Heat Exchanger 熱交換器CFD分析（操作）
- ANSYS FLUENT 功能延伸討論與應用範圍
- Mixing Tank 攪拌槽CFD分析（操作）

Realize Your Product Promise™

# ANSYS FLUENT於化工產業應用簡介



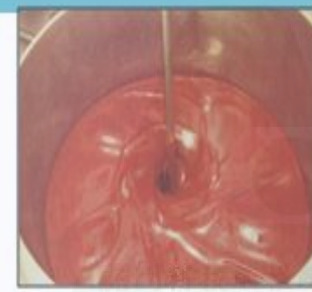

李龍育 Dragon
CFD技術副理
虎門科技

# *Why to model CFD & reacting flows?*

- **Devices are very complex**
  - **Complex geometry, complex BCs, complex physics (turbulence, multi-phase, chemistry, radiation,…), complex systems, …**
- **Tool to gain insight and understanding**
- **Reduce expensive experiments**
- **Eventually design!**

**Reactor Design**

- Blenders
- Polymerizers
- Hydrogenerators
- Crystallizers
- Fermentators
- Fluidized bed

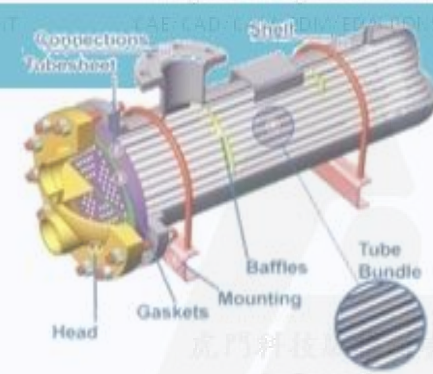


**Filtration**

- Centrifugal
- Granular bed
- Pressure
- Vacuum
- Ulterafine
- Chromatography

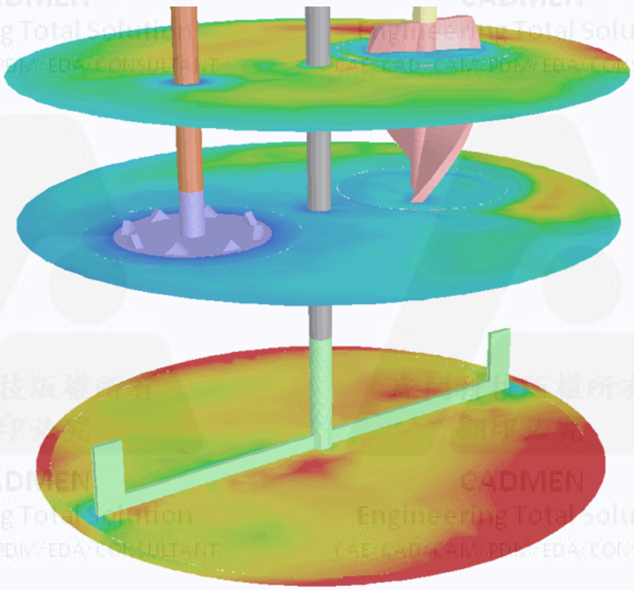
**Heat exchangers**

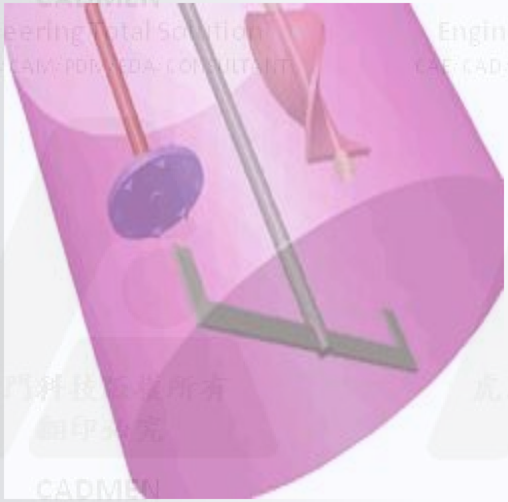
- Plate
- Tube and shell
- Jacket vessels



# *Why to model CFD & reacting flows?*

- **Devices are very complex**
  - **Complex geometry, complex BCs, complex physics (turbulence, multi-phase, chemistry, radiation,…), complex systems, …**
- **Tool to gain insight and understanding**
- **Reduce expensive experiments**
- **Eventually design!**



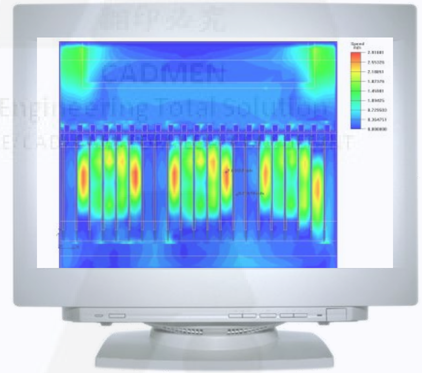


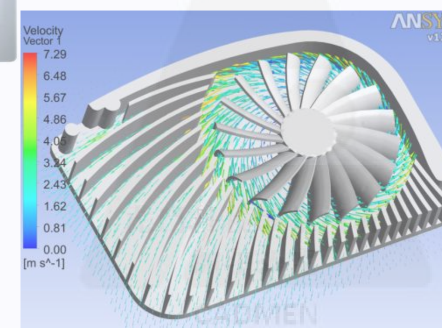


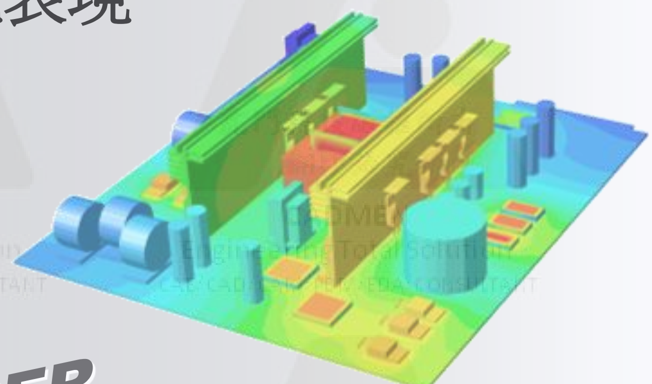
  

# CFD Simulation Driven Product Development

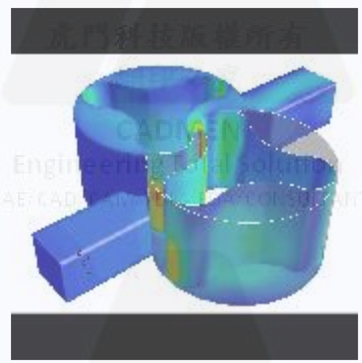
- 計算流體力學
- 數位實驗
- 設計與偵錯工具
- 深入了解產品問題改良產品性能表現

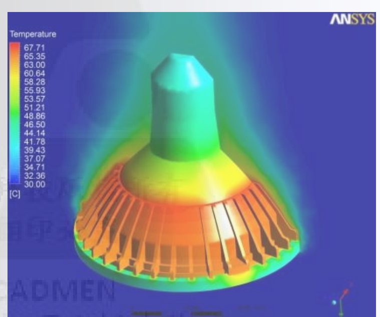


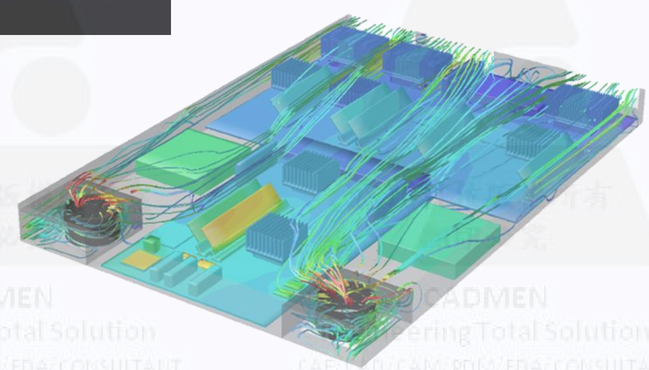


**BETTER**
**FASTER**
**CHEAPER**





# Engineering Challenges

ANSYS

| Engineering Challenges | How can Simulation help |
|---|---|
| | |

**Engineering Challenges**

- Increase plant yield
- Optimize processes
- Reduce cost
- Control product quality
- Emission and pollution
  -Sustainable and green practices
- Safety

**How can Simulation help**

- Reactor scale-up
- Design new equipment and evaluate vendor designs
- Process performance on operating scenarios
  -Increase plant yield
- Provide plant support by troubleshooting unit operations
  -Root-cause failure analysis
- Address safety concerns and emission norms

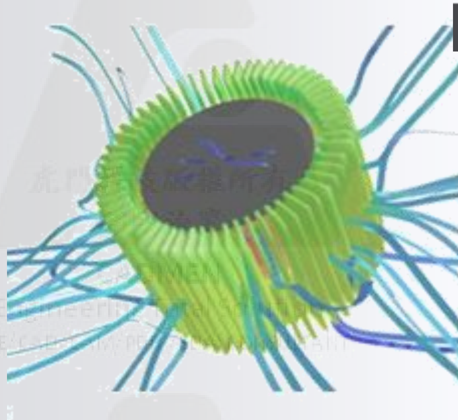
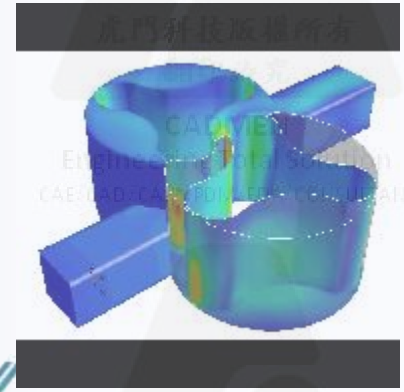
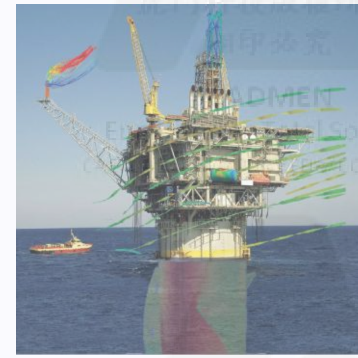
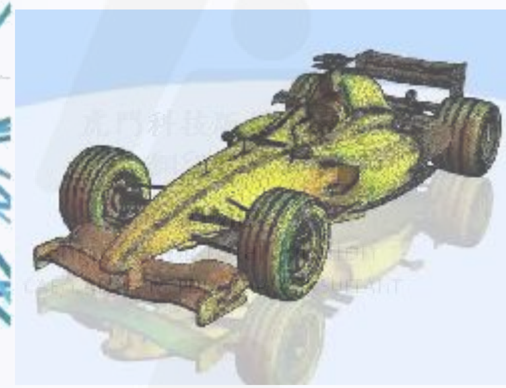
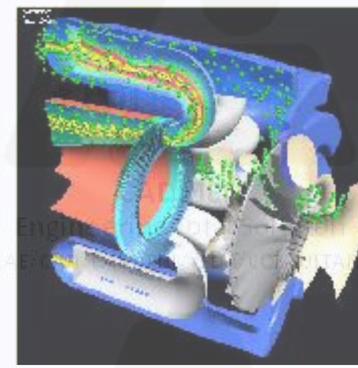
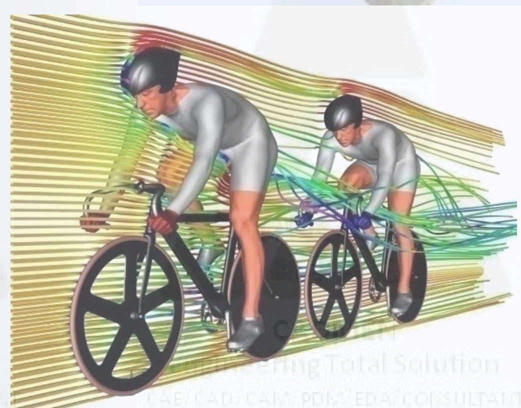
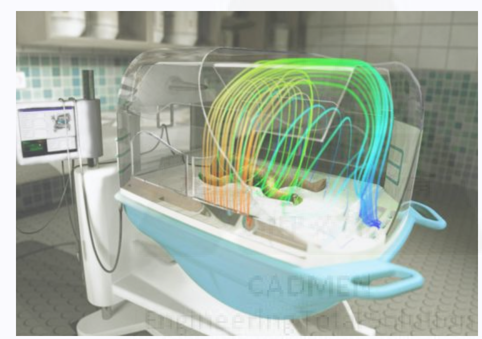
  

# ANSYS CFD模擬軟體介紹

**ANSYS CFD爲功能強大、模組廣泛的旗艦型 CFD 產品**

核心技術
- 移動/變形網格
- 多相流
- 化學反應流
- 紊流
- 熱輻射
- 氣動噪音

擴展分析
- 磁流模組 (MHD)
- 燃料電池模組
- 流固耦合 (FSI)
- 最佳化分析

ANSYS

# Physics Model

### Turbulence Model
- Spalart - Allmaras
- Standard,RNG k -ε
- Realizable k -ε
- Standard,SST k - ω
- $V^2F$ (option)
- RSM
- Transition(SST,k-kl-omega)
- SAS
- DES,LES,E-LES

### Physical Property
- Non Newtonian
- Real Gas
- NIST

### Module
- Fiber & Acoustic
- PBM &
- Adjoint Solver
- Battery Module

### Module (option)
- MHD
- PEM&SOFC

### Reaction& Combustion
- Eddy Dissipation
- Premixed Combustion
- Non-Premixed Equilibrium
- Partially-Premixed
- Laminar Flamelet
- Laminar Finite-Rate
- EDC
- Composition PDF Transport
- Wall Surface & CVD
- Particle Surface
- Spark & Autoignition
- NOx&SOxSoot

### Governing Equation
Mass
Momentum
Energy
Species

### Multiphase Model
- DPM
- VOF
- Mixture
- Euler- Euler
- Euler-Granular
- Dense Discrete Phase
- Discrete Element Method
- Cavitation

### Solidification &Melting
### Wall Boiling

### Porous Model
- Porous Jump
- Porous Media

### Radiation Model
- P1, Roseland
- DTRM, DO
- S2S
- Solar Load



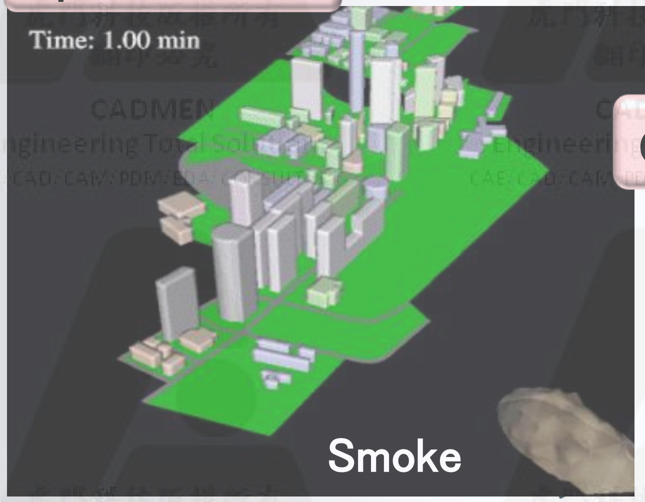
  

# Physics Model

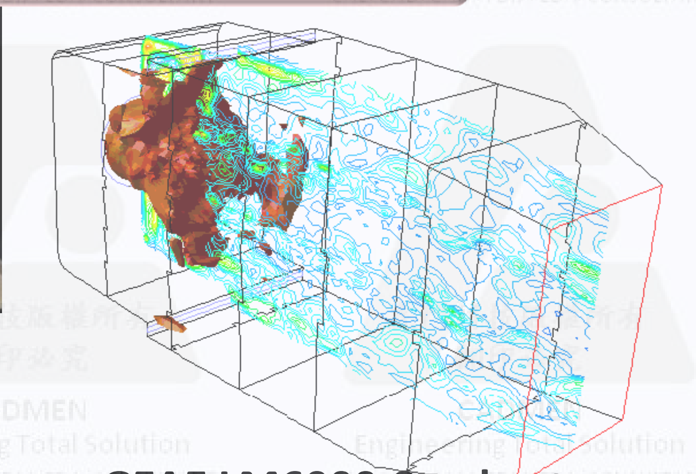
Reaction& Combustion

- Eddy Dissipation
- Premixed Combustion
- Non-Premixed Equilibium
- Partially-Premixed
- Laminar Flamelet
- Laminar Finite-Rate
- EDC
- Composition PDF Transport
- Wall Surface & CVD
- Particle Surface
- Spark & Autoignition
- NOx&SOxSoot

## Species Model

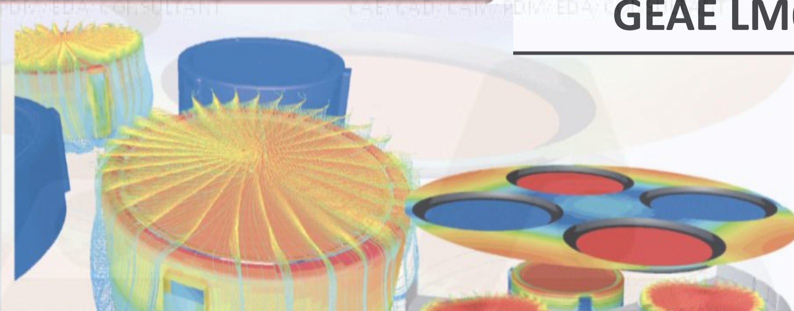



## Combustion Model



GEAE LM6000 Combustor

## Chemical Vapor Deposition



- パスライン(Color SiH4 concentration)
- Deposition speed of Tungsten on the wafer

300mm CVD Cahmber、Novellus Systems, Inc.

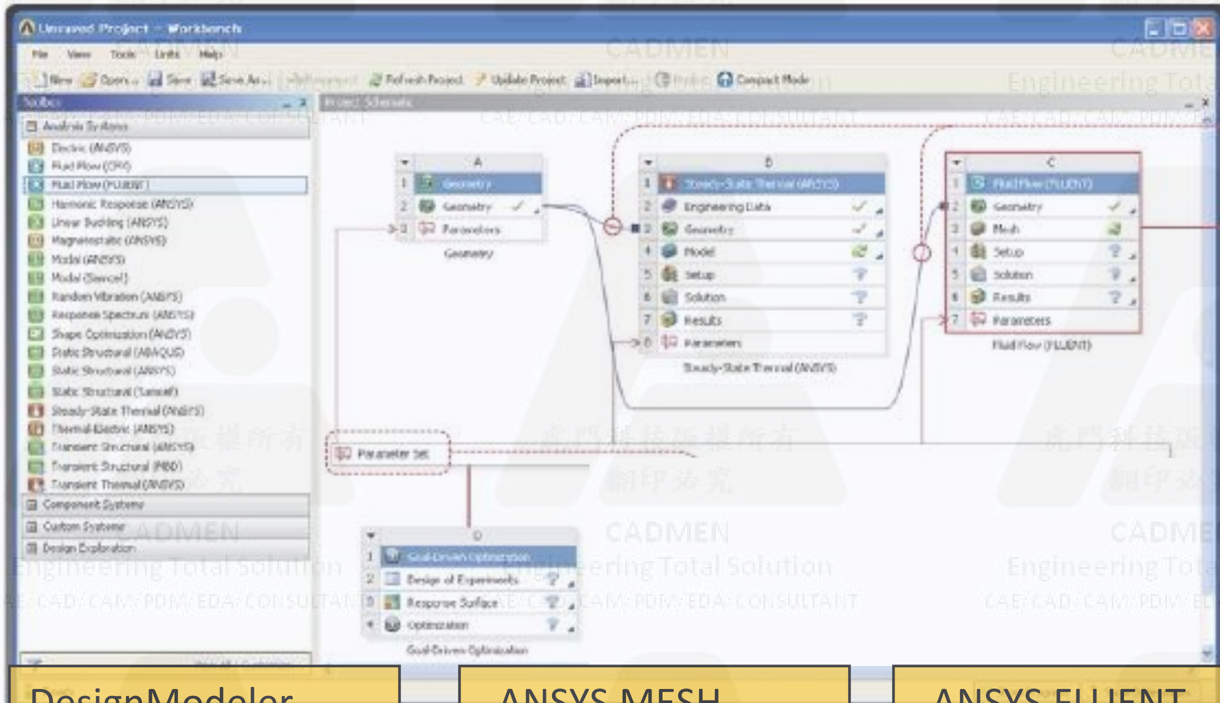
  

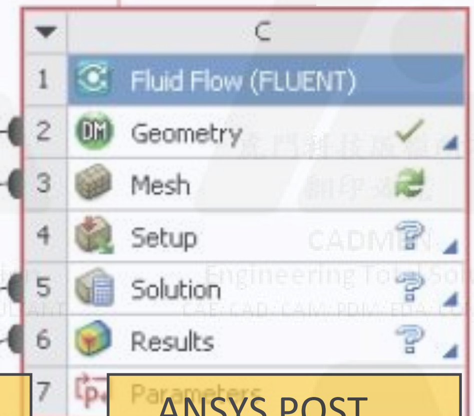
# *About ANSYS Workbench*

ANSYS

## CAE Platform

- 分析專案管理
- 重複應用
- 參數分析
- 最佳化使用
- 耦合運算






DesignModeler
建構分析模型

ANSYS MESH
網格建構

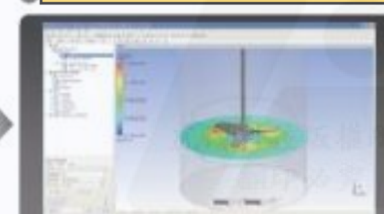
ANSYS FLUENT
設定&計算

ANSYS POST
後處理可視化

ANSYS
DesignModeler

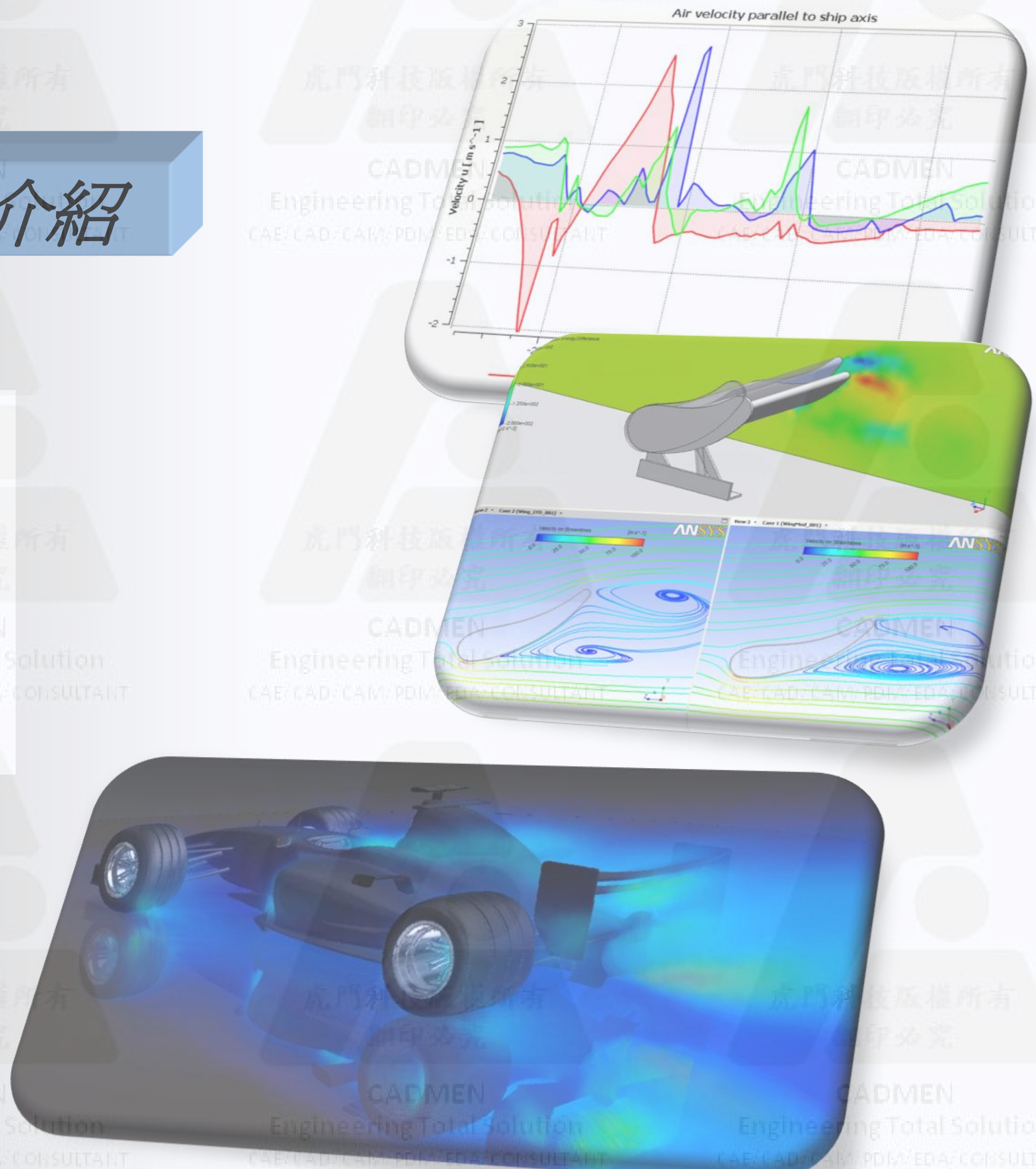
ANSYS
Meshing Platform

ANSYS FLUENT
ANSYS CFX など



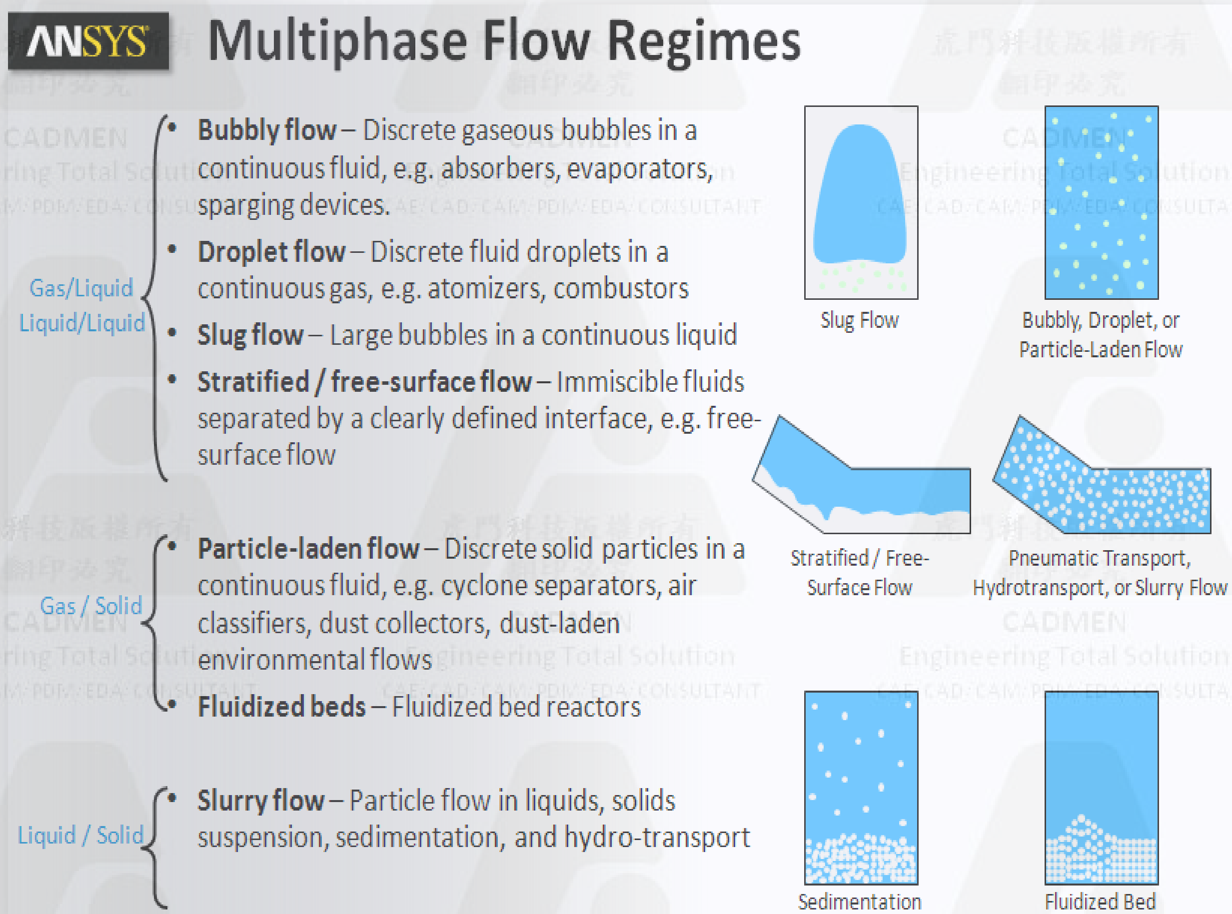
ANSYS
CFD-Post

# CFD相關應用範例介紹



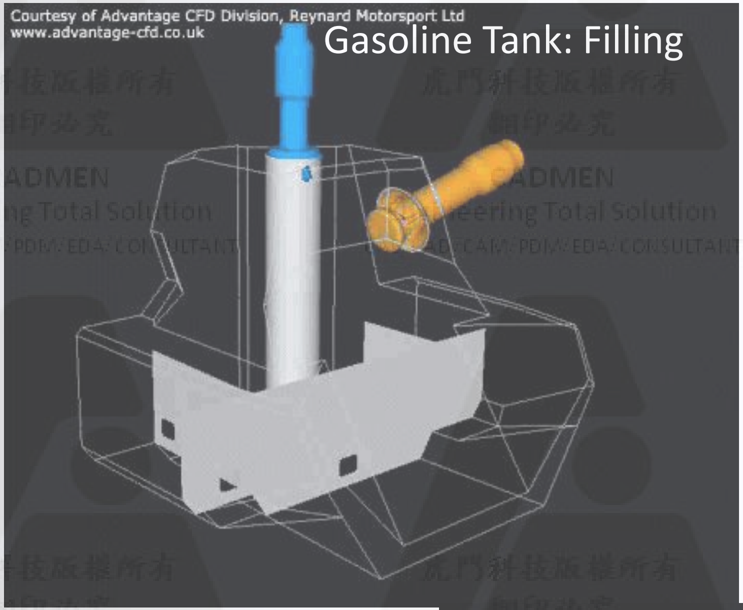
# Physics Model

Multiphase Model
- DPM
- VOF
- Mixture
- Euler- Euler
- Euler-Granular
- Dense Discrete Phase
- Discrete Element Method
- Cavitation

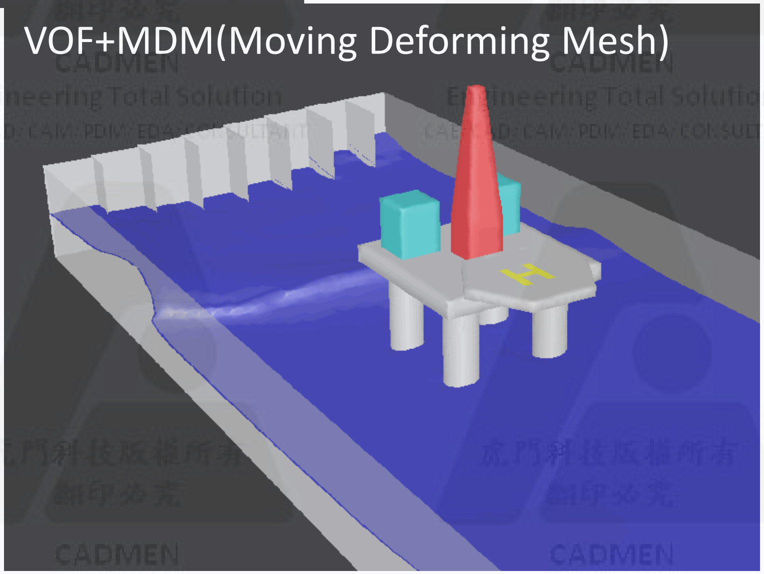


# Physics Model

## VOF Model




Multiphase Model
- DPM
- VOF
- Mixture
- Euler- Euler
- Euler-Granular
- Dense Discrete Phase
- Discrete Element Method
- Cavitation

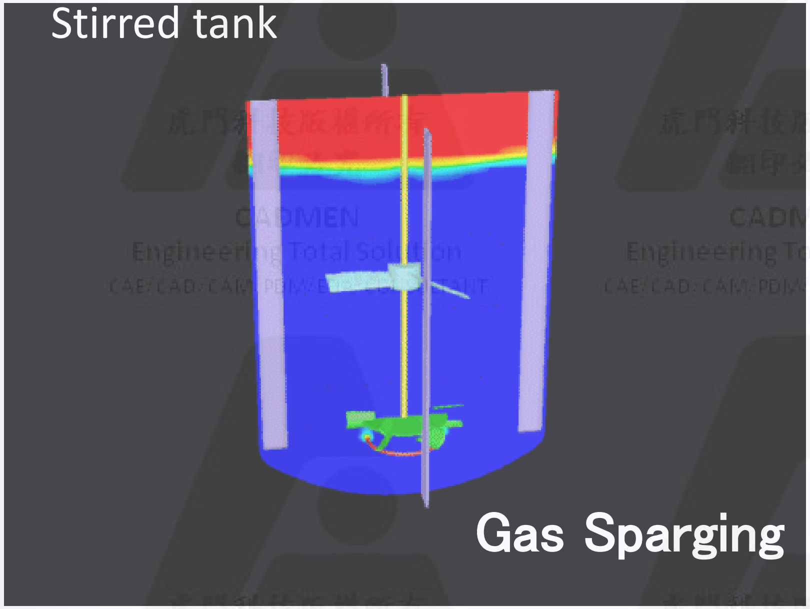


ANSYS

# Physics Model

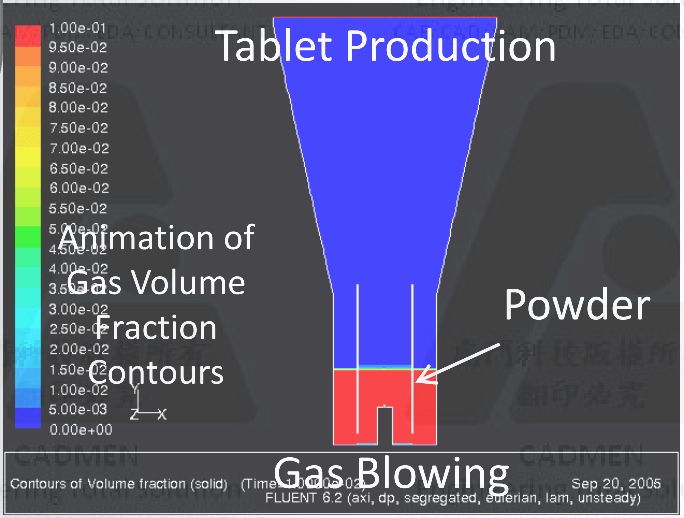
## Euler–Euler Model

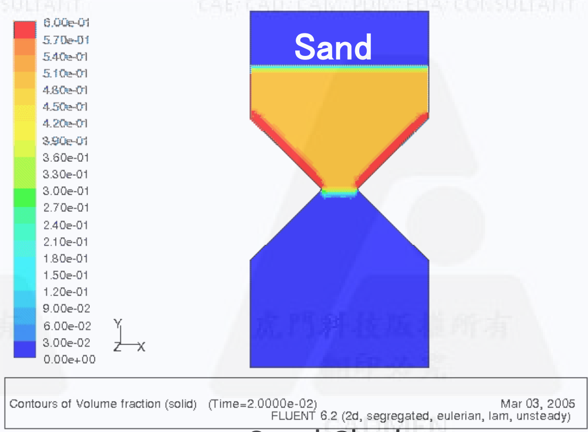



Multiphase Model

- DPM
- VOF
- Mixture
- Euler- Euler
- Euler-Granular
- Dense Discrete Phase
- Discrete Element Method
- Cavitation

## Euler–Granular Model



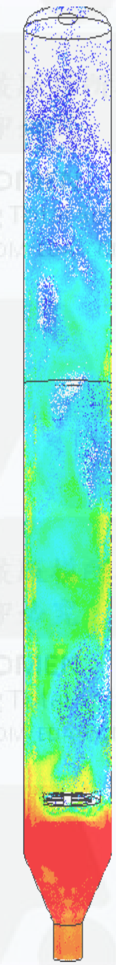

Sand Clock

ANSYS

# Physics Model

## Dense Discrete Phase Model

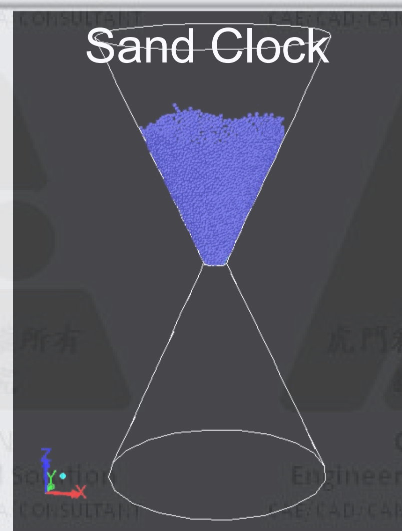


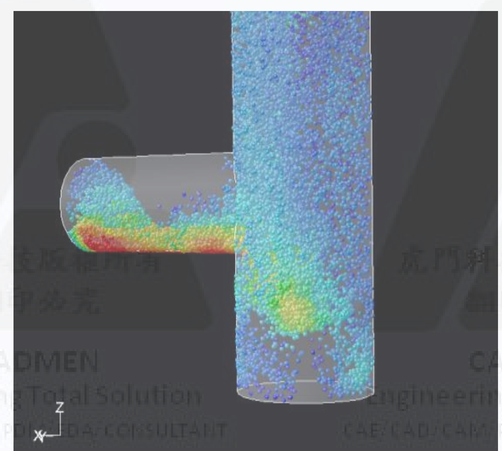
## Discrete Phase Model


Multiphase Model

- DPM
- VOF
- Mixture
- Euler- Euler
- Euler-Granular
- Dense Discrete Phase
- Discrete Element Method
- Cavitation

## Discrete Element Method

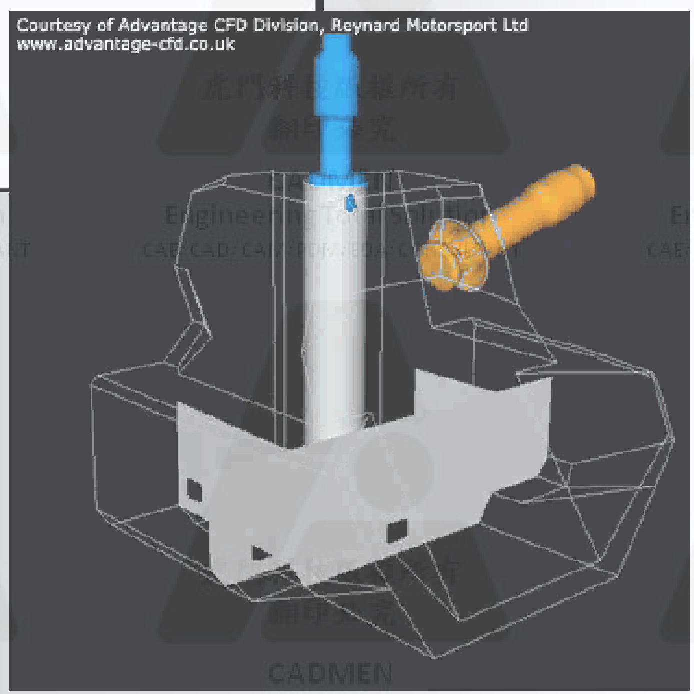


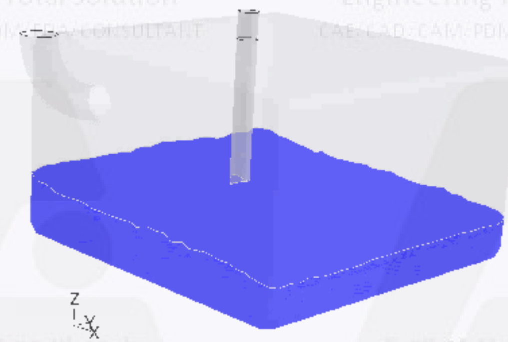
Contours of Temperature

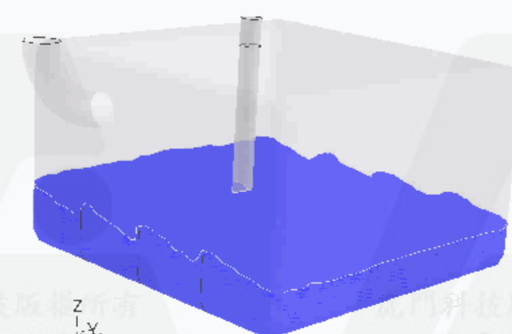
  

# Filling, Tank Sloshing



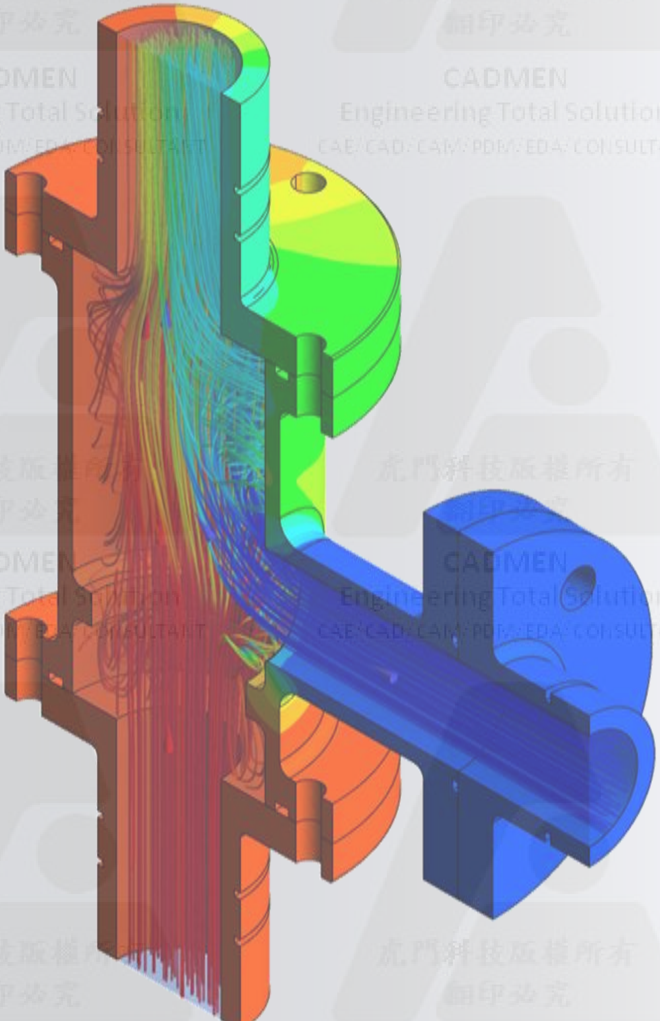

No Baffles



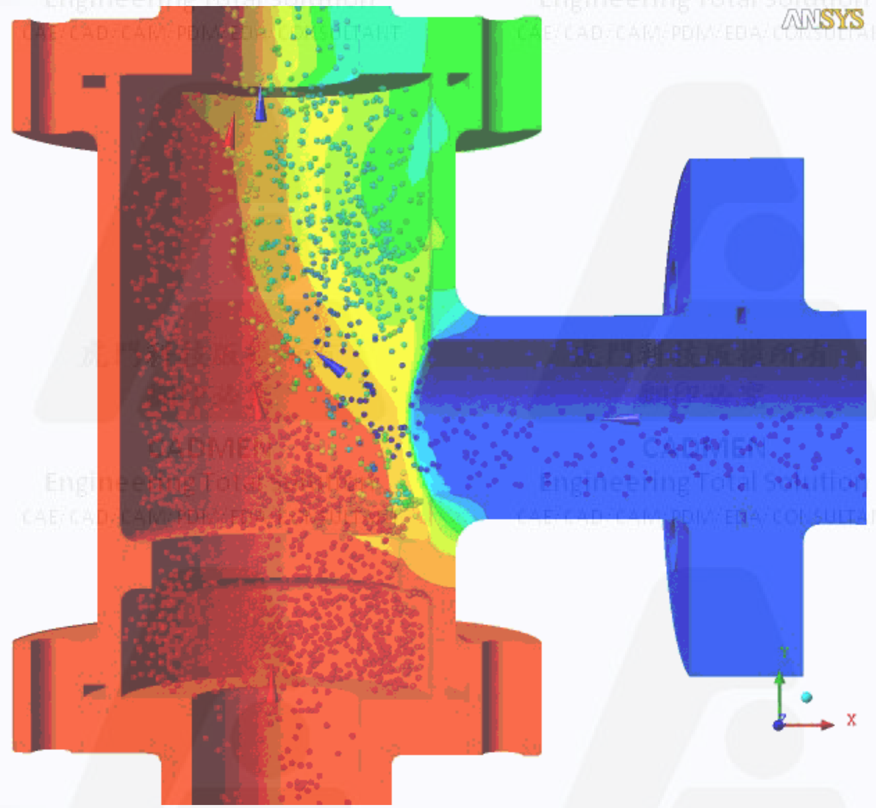


With Baffles

# 管路流場分析



顆粒流動分析



# Sand/Particulate Transport

- Sand is often produced in both onshore and offshore production systems,
- Sand production may be continuous, or sudden
- The sediment consists mud, sand and scale picked up during the transport of the oil
- Sand deposition could lead to corrosion of the pipeline
- Problem of sand deposition and re-entrainment can be addressed by Particulate modeling in ANSYS CFD.

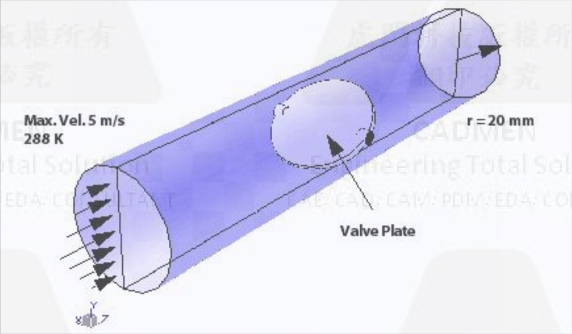
*Internal flow of natural gas containing sand particles.*

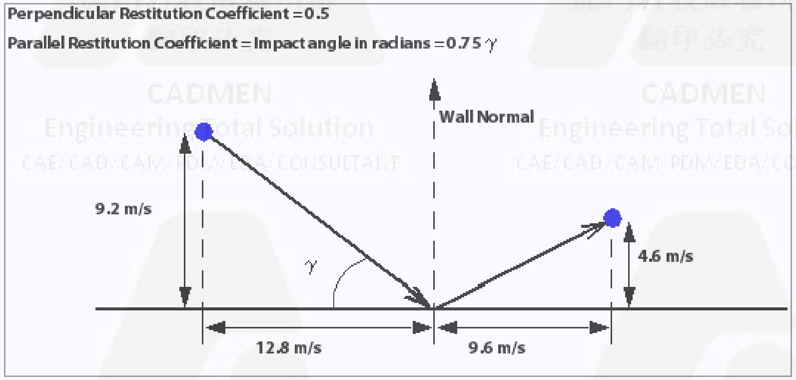
*Selected particle trajectories are colored in grey*

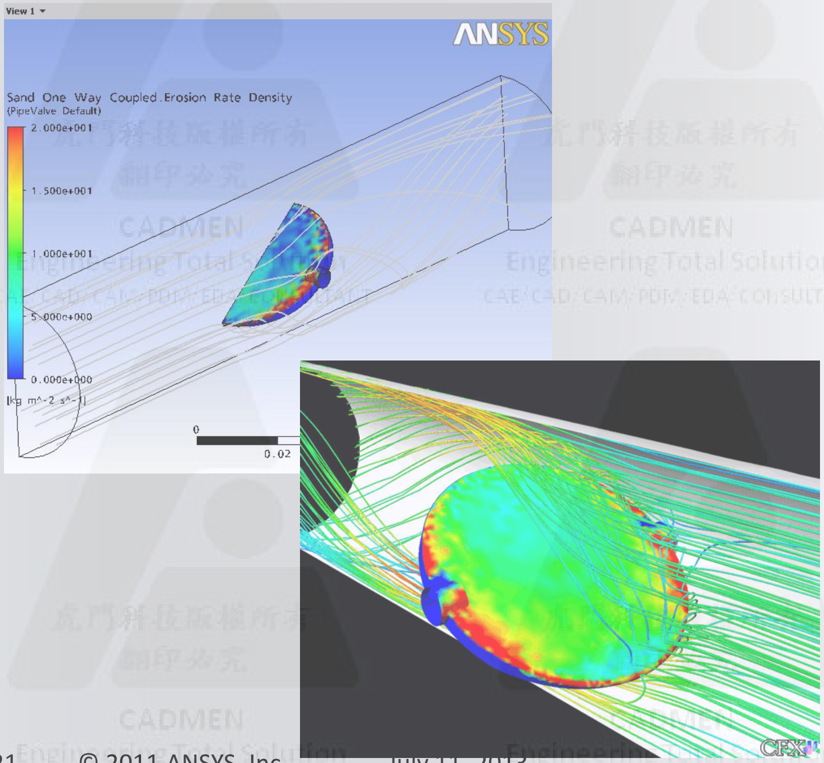
*The erosive wear hotspots on the piping is colored out in red.*

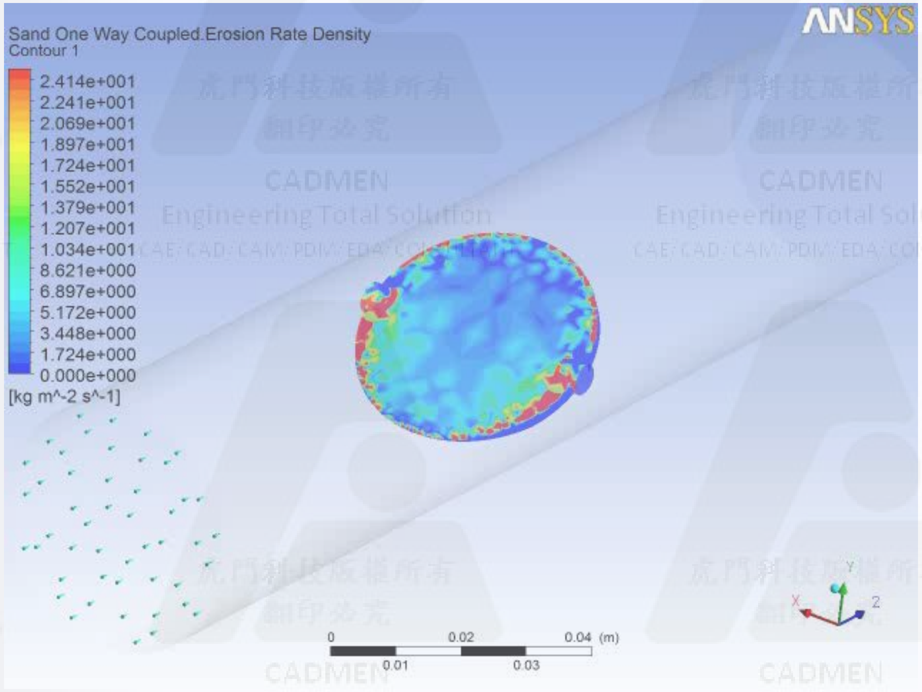
  July 11, 2013

# 泥沙沖刷 – 考慮顆粒侵蝕效應



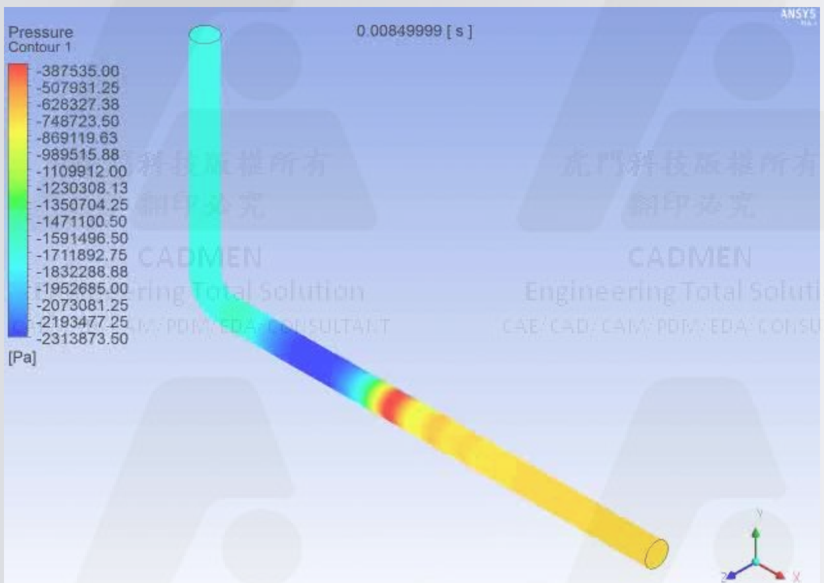

  

# *Water Hammer*

ANSYS

考慮流固耦合，可分析因
水槌現象所造成之管路偏移與應力

管路出口突然關閉
所造成之水槌現象



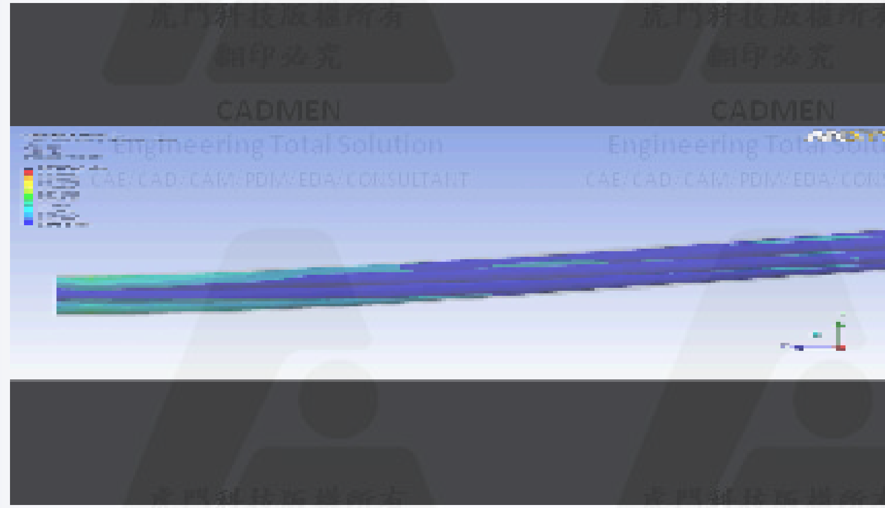



  July 11, 2013

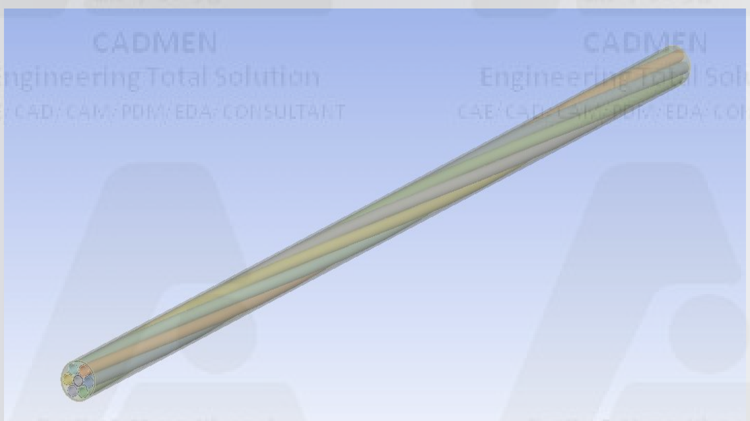
# *Oil and Gas Transport Client Case: Umbilical, Risers, Flexible Piping*

## Challenges

- Very high reliability requirements
- Transmitting signals and transporting flows over much longer distances
- Tubing exposed to harsh conditions
- Develop reliable piping for variety of offshore applications
- Requires structural reliability while handling thermal stresses and fatigue
- Manufacturing of multi-tubes including the steel wrapped around
- Complex multi-layer, varying material manufacturing channels



Equivalent stress response of the umbilical to a 36 degrees bend



Complex multi-layer umbilical ….

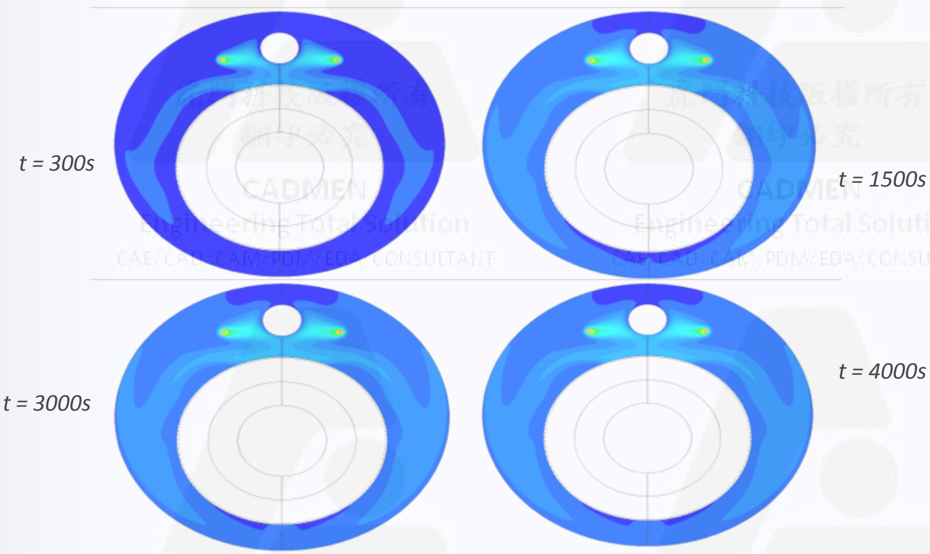
## Sample Client Case—Used CAE to:

- Build the core tubing and additional helical tube models in an easy to use environment
- Account for different material proprieties
- Evaluate thermal, flow loading and bending stresses. Account for all applied loads including
  - High bending angles
  - Hydrostatic loads
  - End tension
  - Gravity

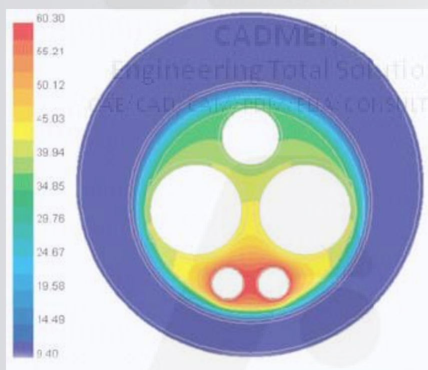
# Oil and Gas Transport Flow Assurance

## Challenges

- **Deeper water depths leads to complex tubing design and manufacturing concerns including**
  - **Depositions**
  - **Wax formation**
  - **Thermal management**
  - **Erosion inhibitors**
- **Active heating, insulation and bundle design in harsh environment and deep waters**




Contours of corrosion inhibitor



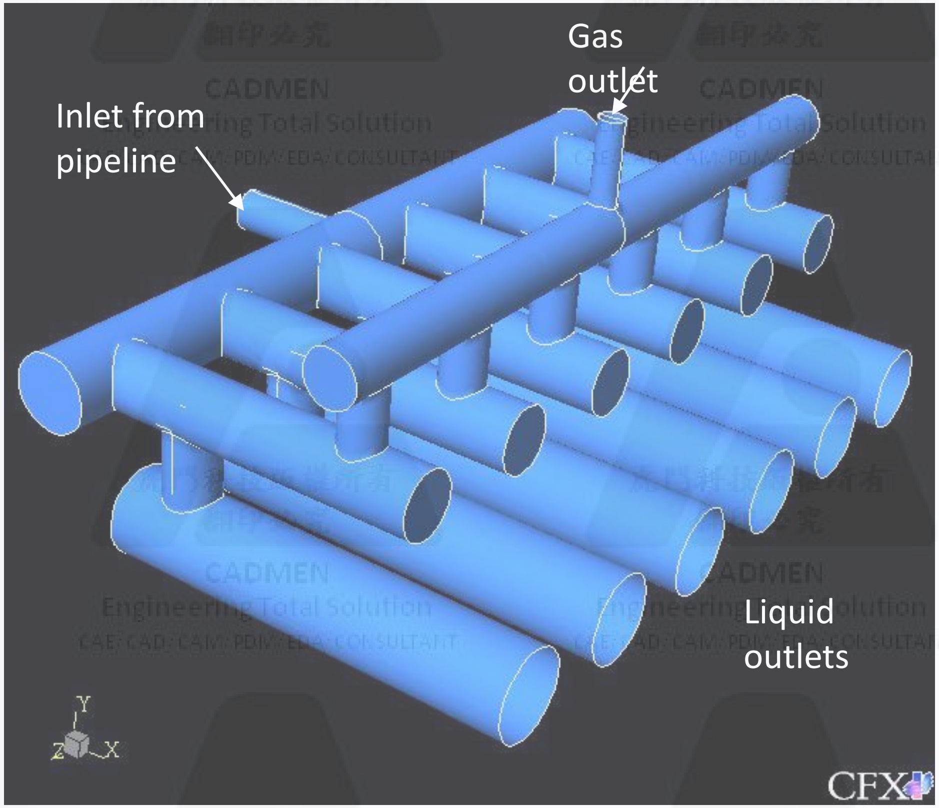

## ANSYS CAE Solutions

- **Ability to optimize thermal and structural behavior of tube bundles used for deep and ultra deep waters**
- **Can develop technology for fouling, wax and hydrate formation by obtaining detailed thermal management information, including insulation in the analysis**
- **Thermal uniformity for start up and shut downs**
- **Thermal stress and fatigue**

Some images courtesy of Subsea 7

# Flow Assurance
# Example - Slug-Catcher

- **Gas pipeline from off-shore field to land-based Hannibal terminal**
- **Slug catcher separates residual liquid from gas at end of pipeline**
- **Plan to increase pipeline capacity to supply new power station**
- **Question: Does capacity of slug catcher also have to be increased?**



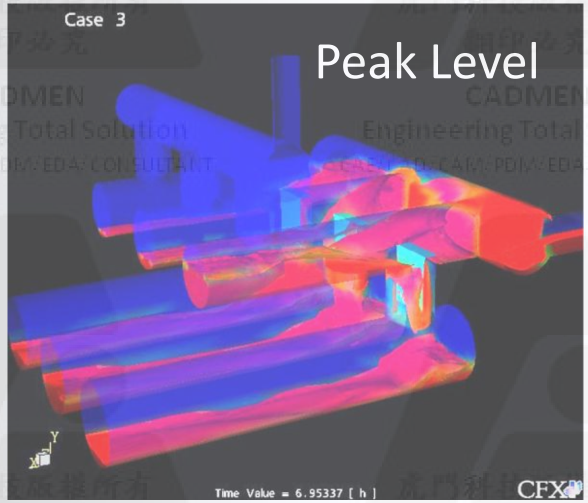

**Estimated cost of modifying slug catcher $25M**

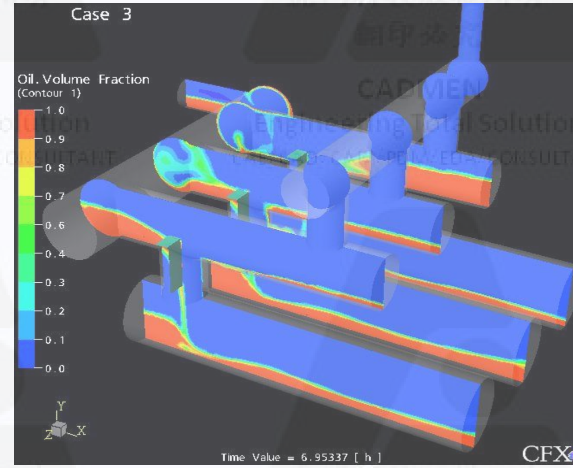
Courtesy of Genesis Oil and Gas

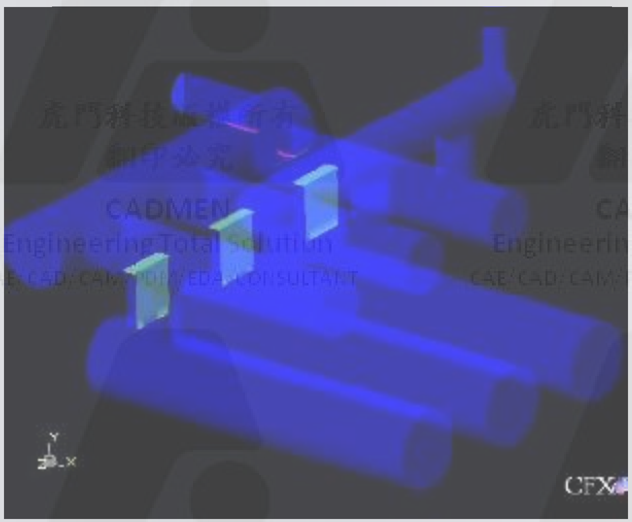
  

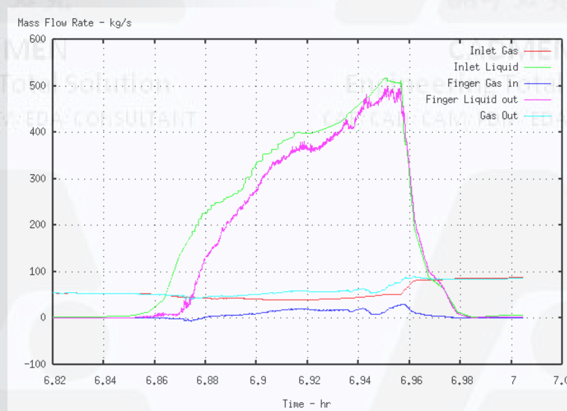
# *Slug Catcher High Flow Operation*

- **Can slug catcher cope with increase in capacity of pipeline? – Yes!**
- **Liquid carry-over limited to a fine aerosol**

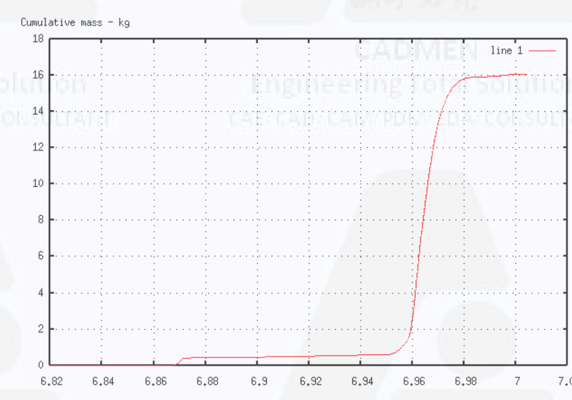



**Flow rates**



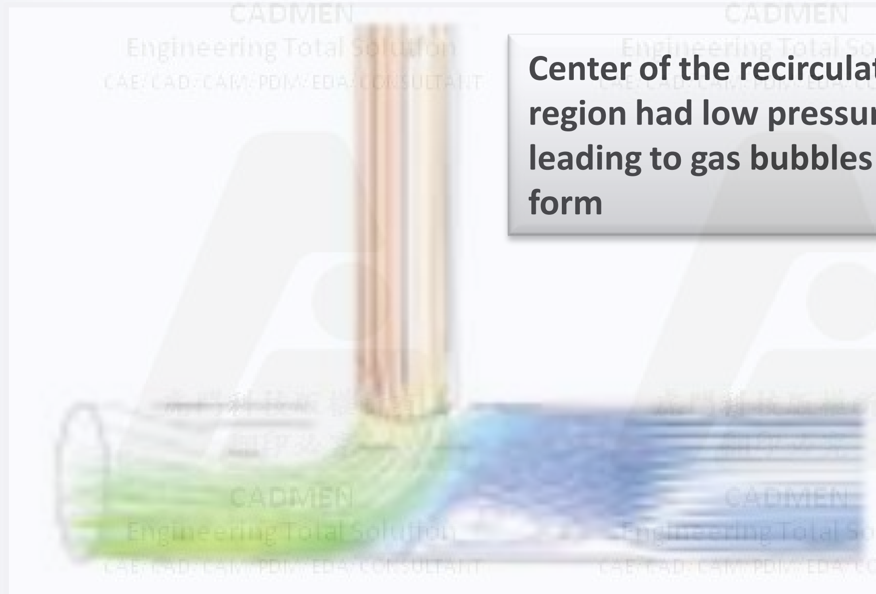

**Liquid carry-over**

Courtesy of Genesis Oil and Gas



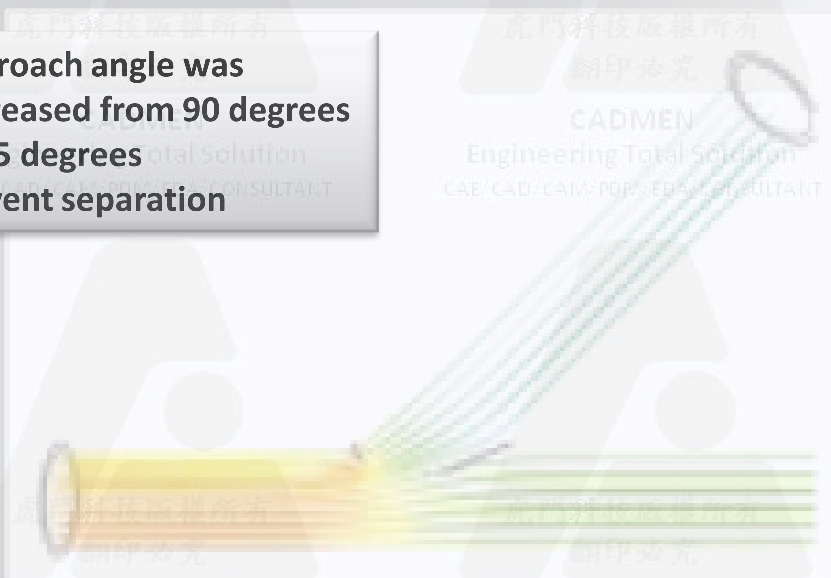
  

# *Case Study: Analyzing Cavitation in the Pipe*

**Center of the recirculation region had low pressure leading to gas bubbles to form**

**Approach angle was decreased from 90 degrees to 45 degrees prevent separation**

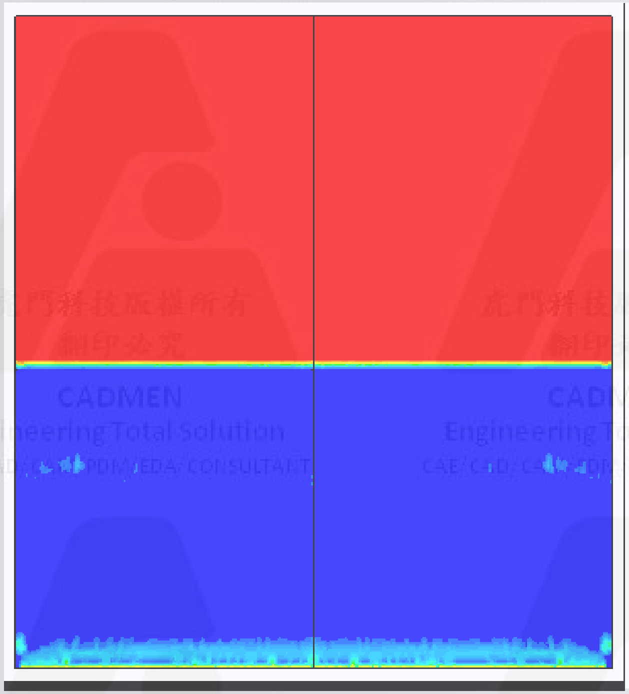


Courtesy: **Saudi Aramco**

# *Fluidized bed and Mixing reactor*

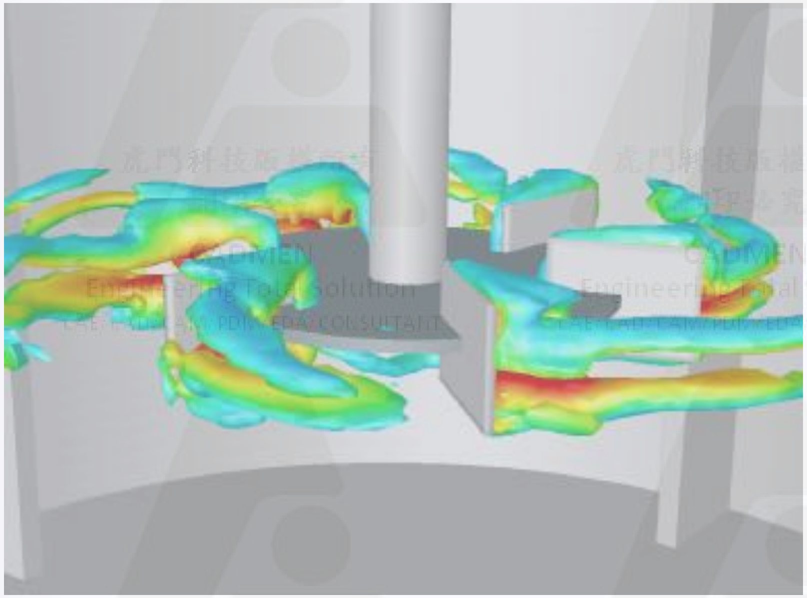
- **Fluidized bed**

Rising bubbles of gas are predicted in a fluidized bed.
Bubbles pass through bed surface and enter the gas space above



固體顆粒處於懸浮狀態時所造成的流體化現象，藉由固體顆粒與流體分子充分接觸碰撞，以促進熱傳、質傳或化學反應之效率。

- **Mixing reactor**



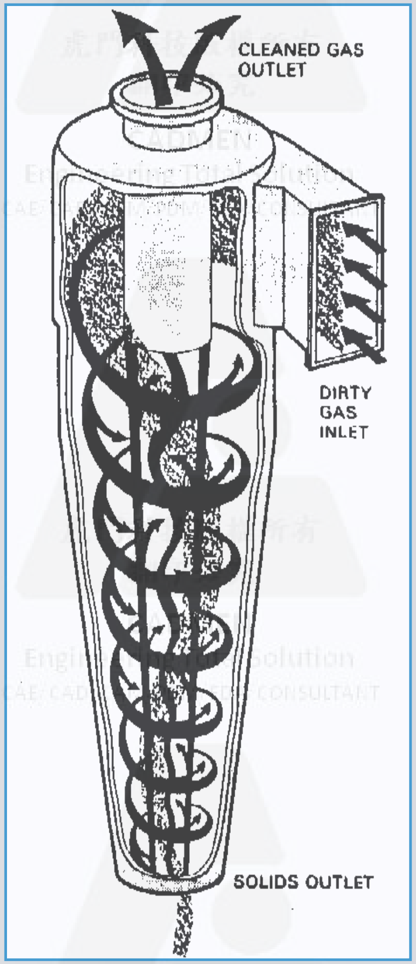
LES turbulence model predicts the occurrence of vortices behind the blade.

# Equipments Design - Cyclones

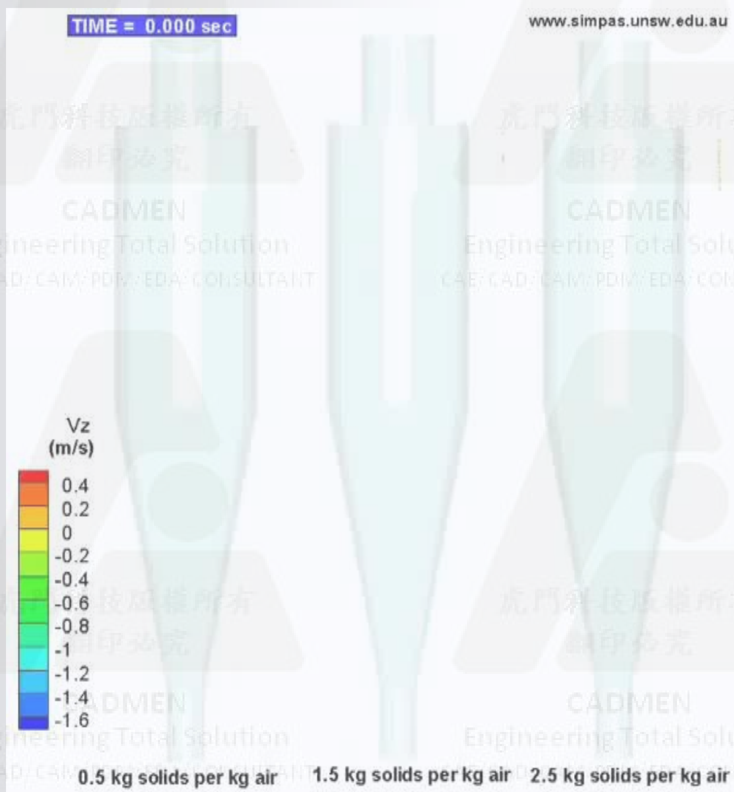
ANSYS

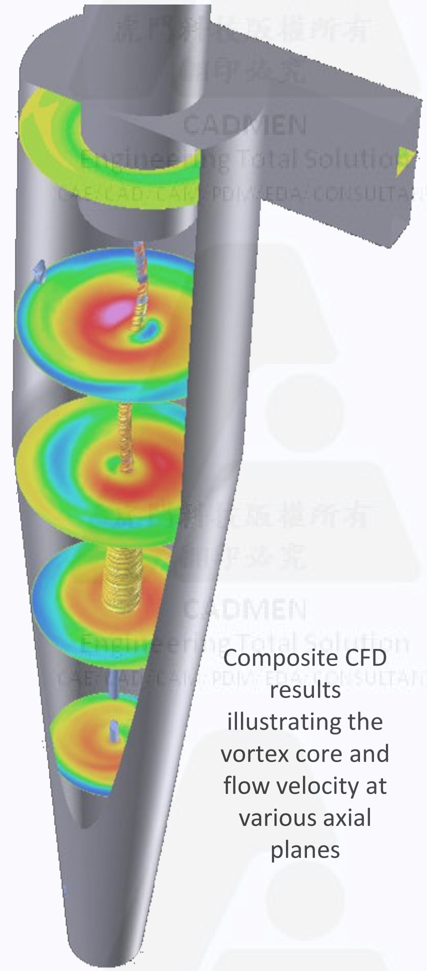
## ANSYS CAE Solutions

- **Optimize inlet design to reduce erosion, increase efficiency and find the range of device's usability**
- **Geometry and design optimization for various particle loading in 2 phase and 3 phase applications**




Schematic of complex flow motion in a cyclone separator



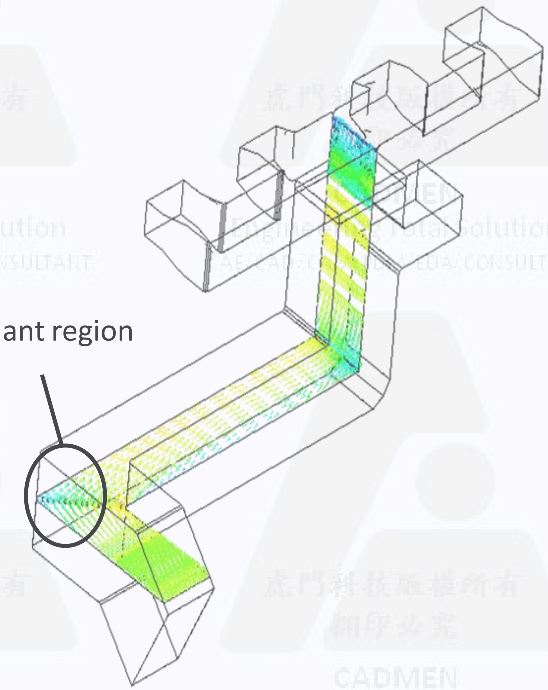



Composite CFD results illustrating the vortex core and flow velocity at various axial planes

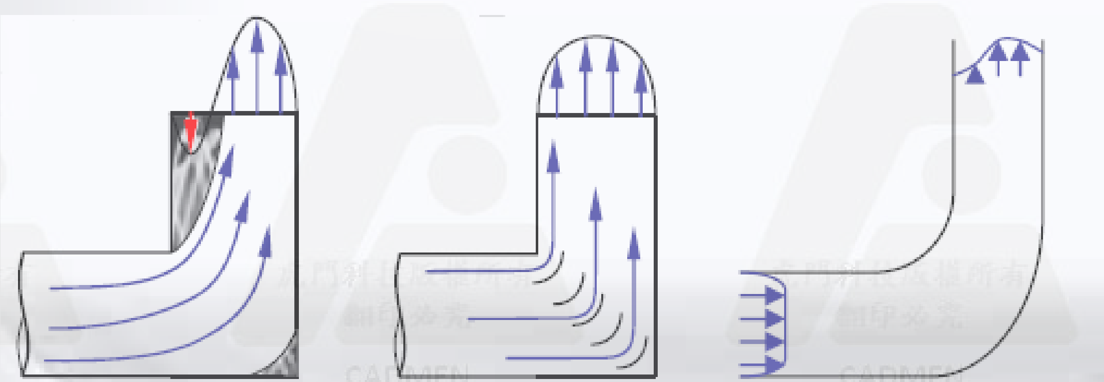
  

# 汙染物清理設備優化

- □工業燃燒主要汙染物– NOx, SOx
- □燃燒產物中汙染物的控制

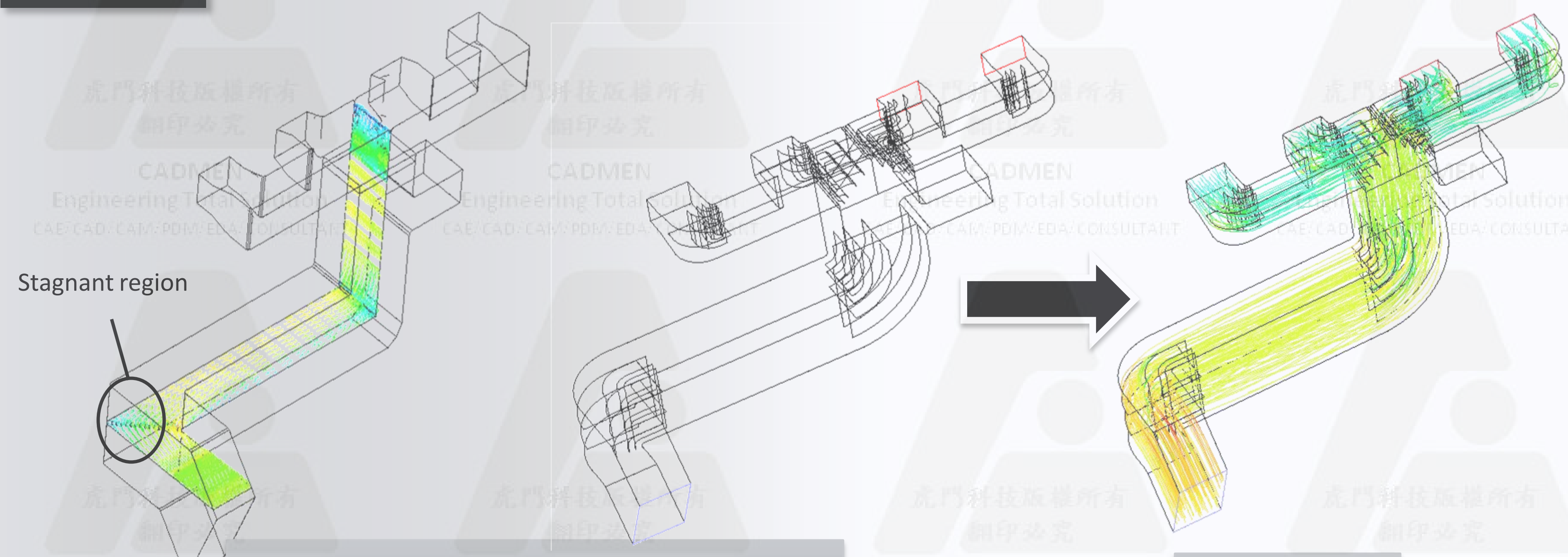



- 燃燒器的設計– 火焰形狀及溫度分佈的控制
- 燃燒後污染物的清理
  - -噴淋注流催化/非催化還原– SCR, SNCR, FGD
  - -污染物清理效率很大程度取決於流場的均勻性
- 清理設備的優化– 流道設計的優化
  - -減小流道內邊界層分離



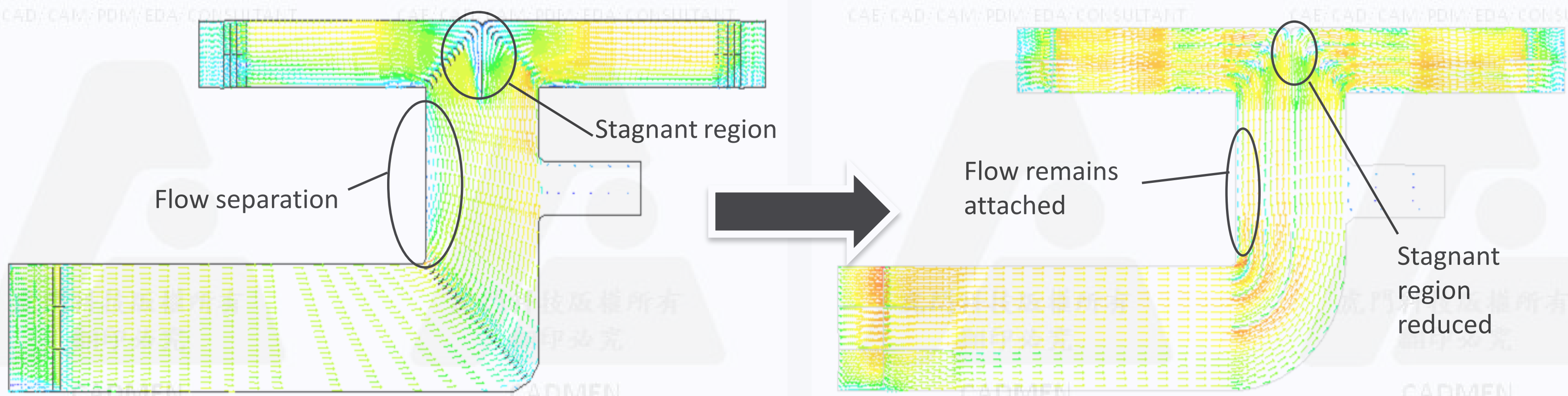
# *Case Study: Gas Ducting*

ANSYS




**Flow biased towards right hand branch**

**No flow bias**





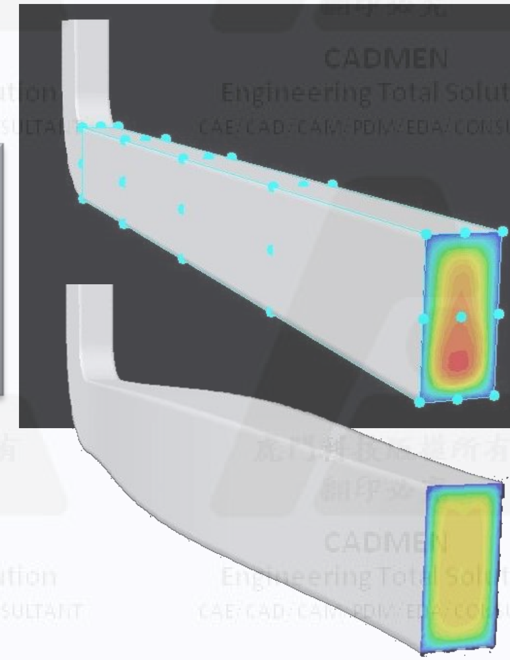
# *Shape Flow Optimization*

ANSYS

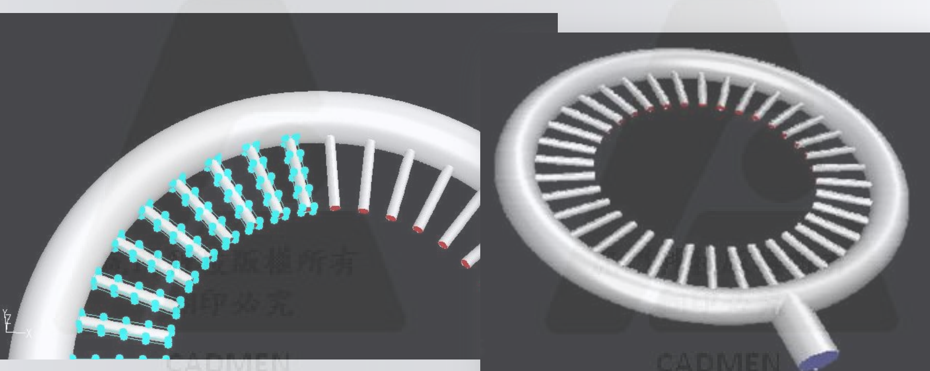
## Design requirement:

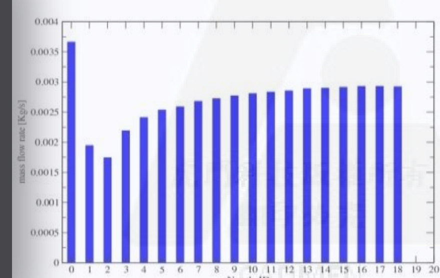
- **How to get uniform flow across 18 outlet nozzles? Or out of a duct?**
- **Use mesh morphing and optimizer, both integrated into the FLUENT solver**

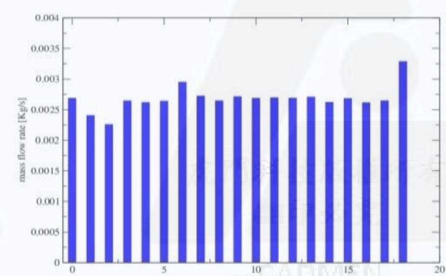
**Design criterion: uniform flow at outflow of duct. Optimize shape**



**Design criterion: uniform flow out of nozzles. Optimize nozzle length**





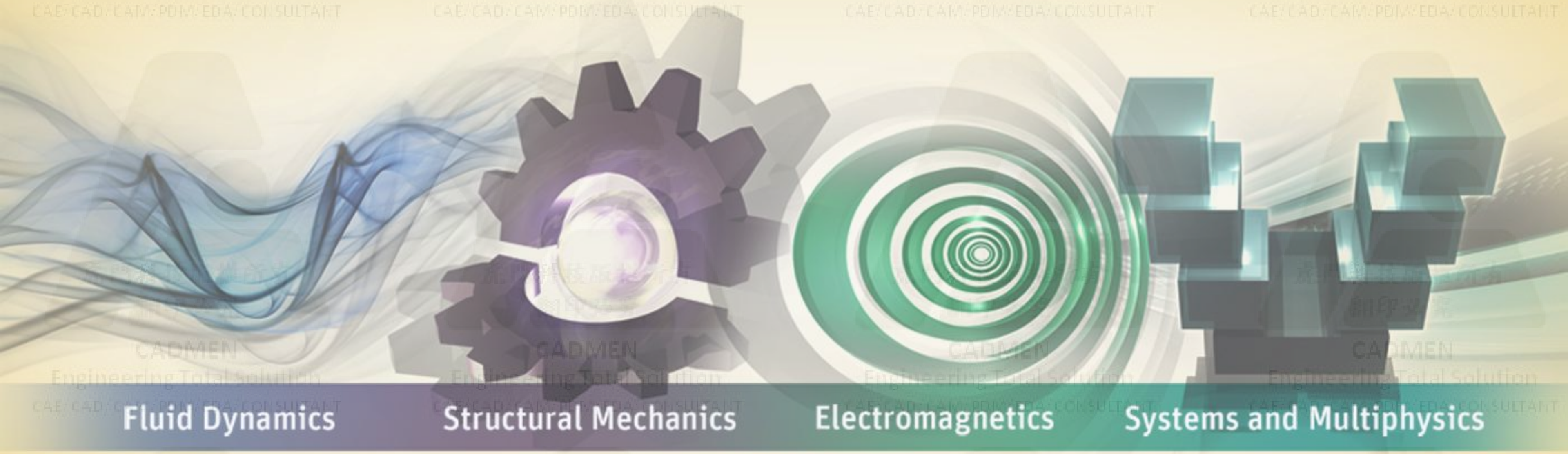


ANSYS

Realize Your Product Promise™

# 氣體擴散反應範例介紹




李龍育 Dragon
CFD技術副理
虎門科技

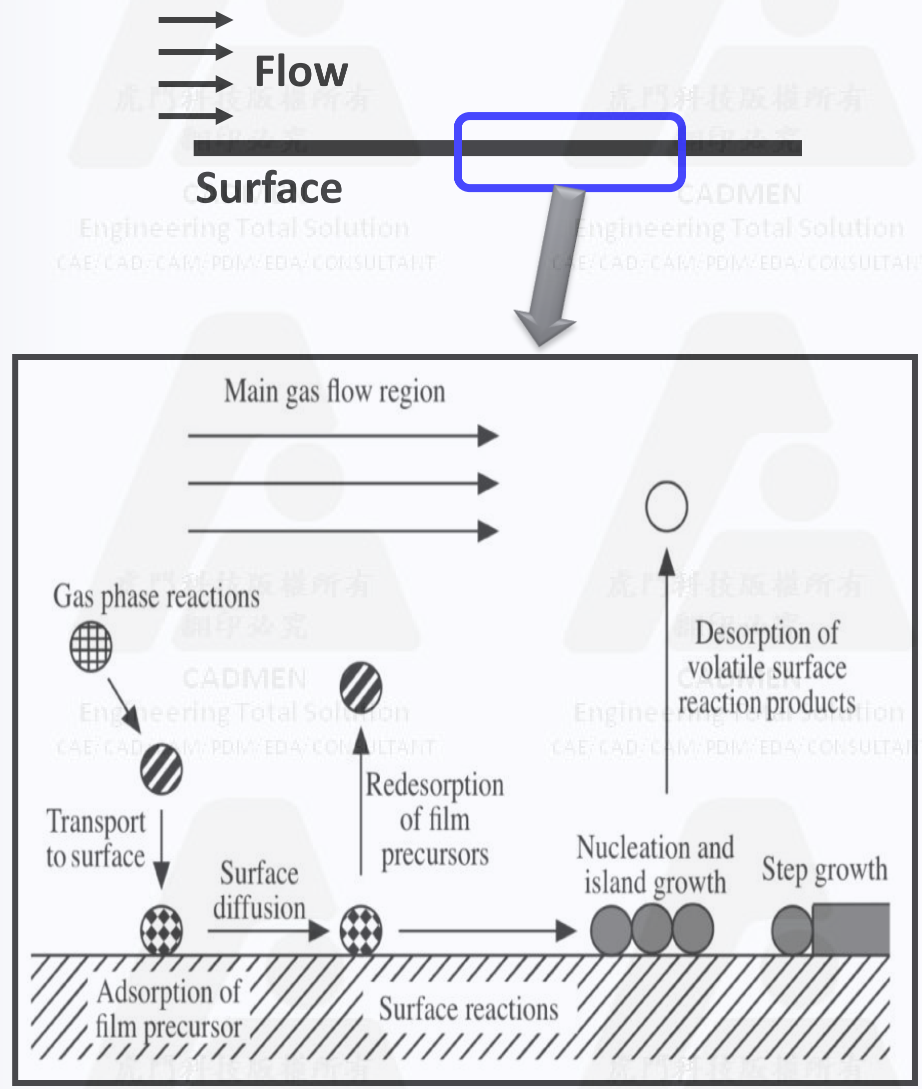
# ANSYS FLUENT Species Reaction Model

ANSYS

## Surface reaction mechanism

- Decomposition reactions of the precursors in the gaseous phase
  (氣態體分解反應)
- Adsorption reactions of the reactive products by the surface
  (表面反應產物的吸附反應)
- Desorption reactions of the adsorbed species
  (吸附物種的脫附反應)
- Deposition of the required material
  (沉積所需的材料)

**Heat required for the reactions may be supplied either by heating the reaction chamber wall or the substrate itself**

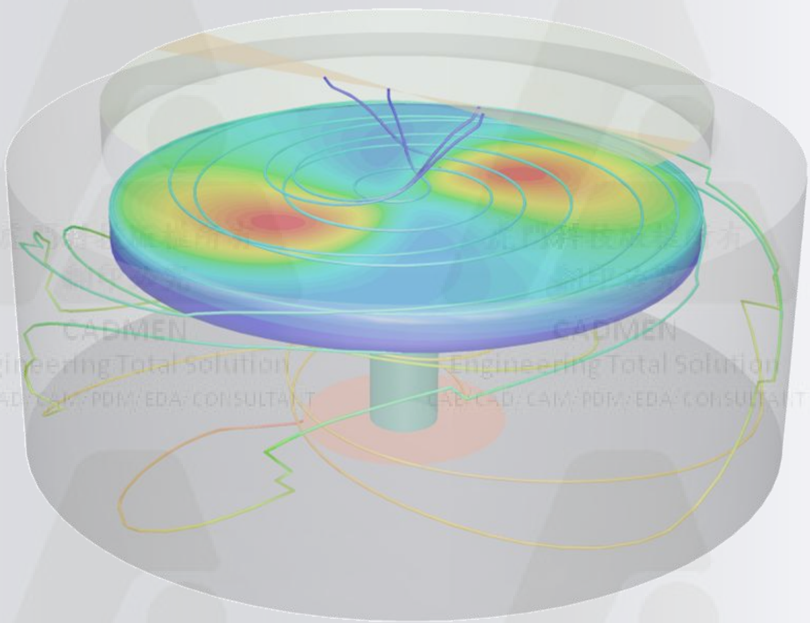




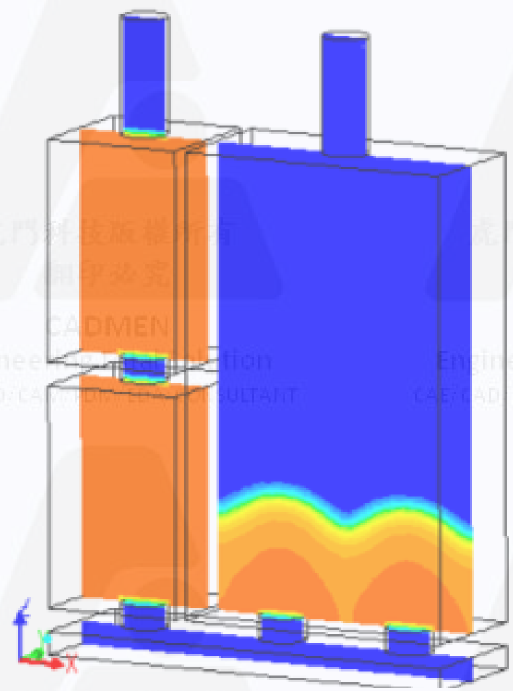
# *Surface Reactions Models*

- **Resolved surfaces model**
  - **Surface reaction on resolved wall surfaces**



**Growth Rate of Gallium Arsenide in a vertical rotating disc reactor**

- **Un-resolved surfaces model**
  - Surface reactions in porous media

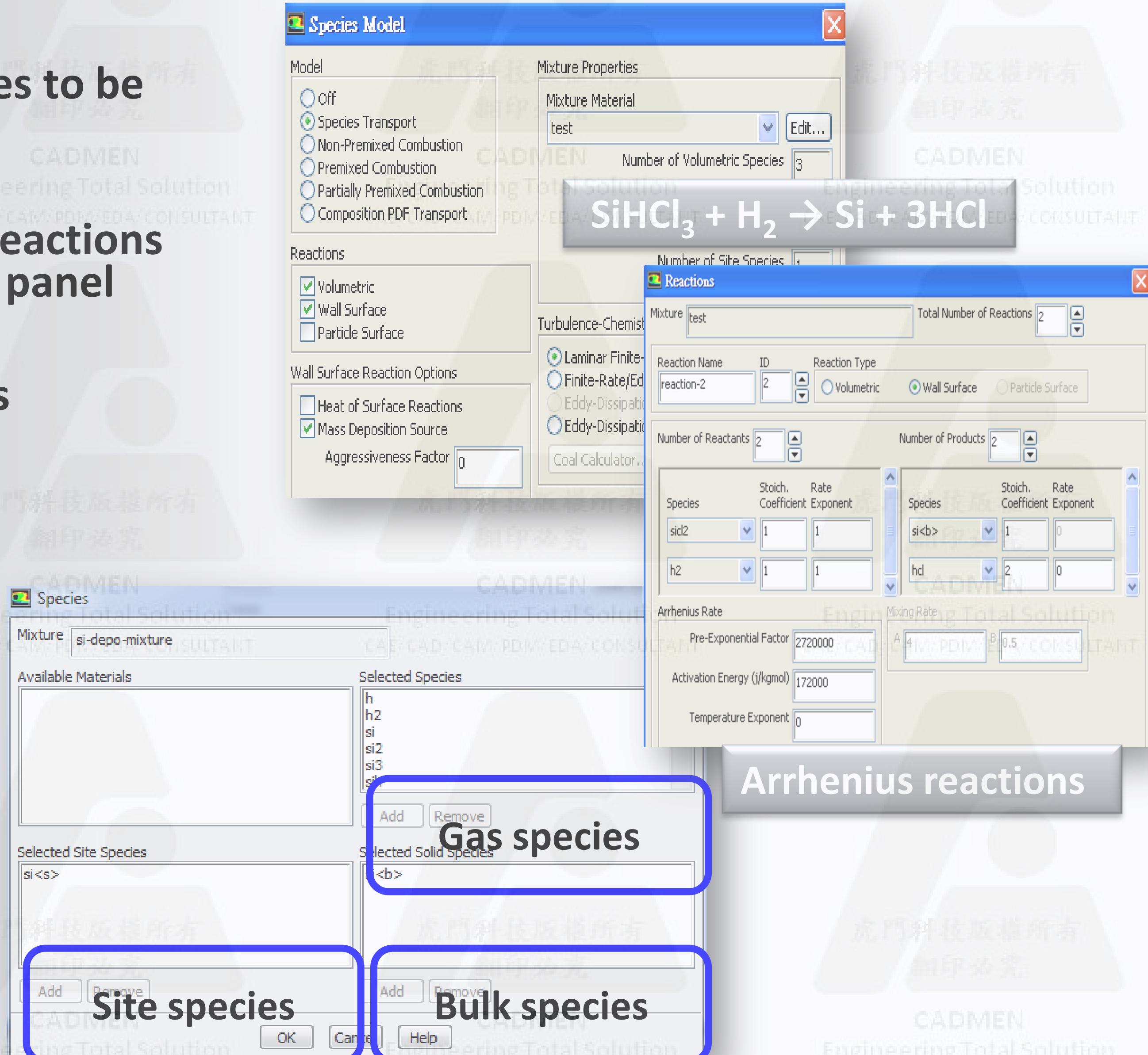


**Hydrocarbon capture in carbon canister**

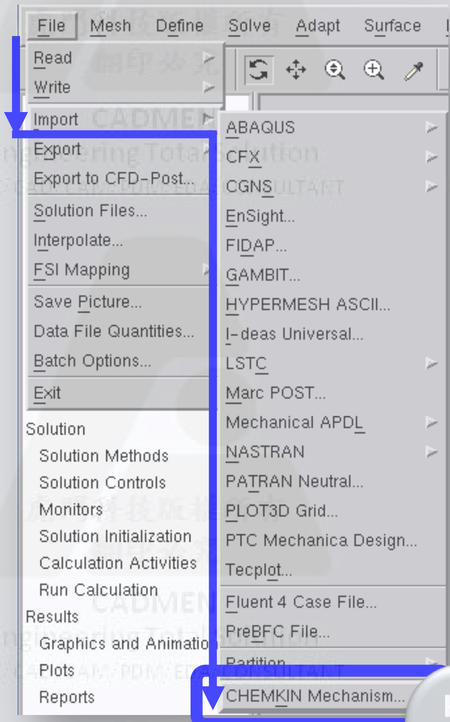
  

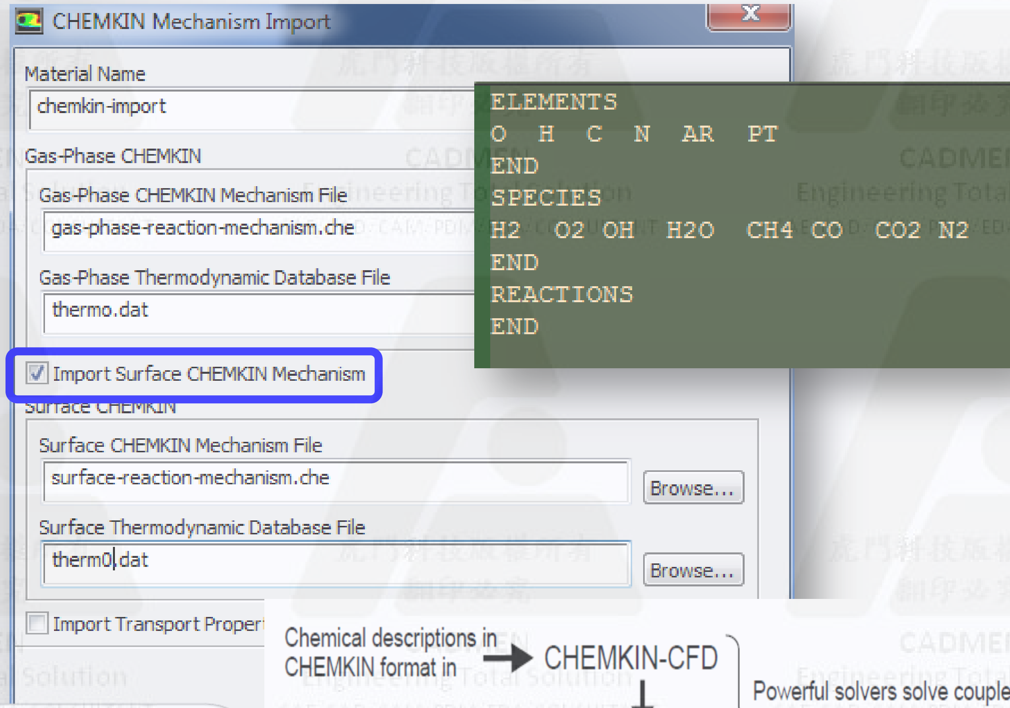
# Material & Reaction Set Up

- Gas, site and bulk species to be defined as type **fluid**
- Switch on **wall surface** reactions in the species transport panel
- Include required species appropriately
- Define reactions
  - Volumetric
  - Wall surface



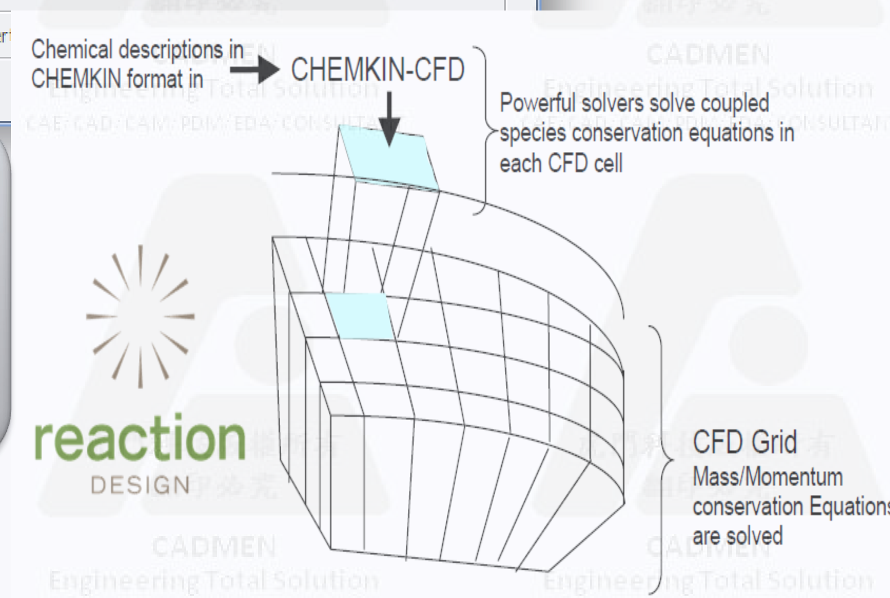
# Surface Reaction Import




**KINetics**

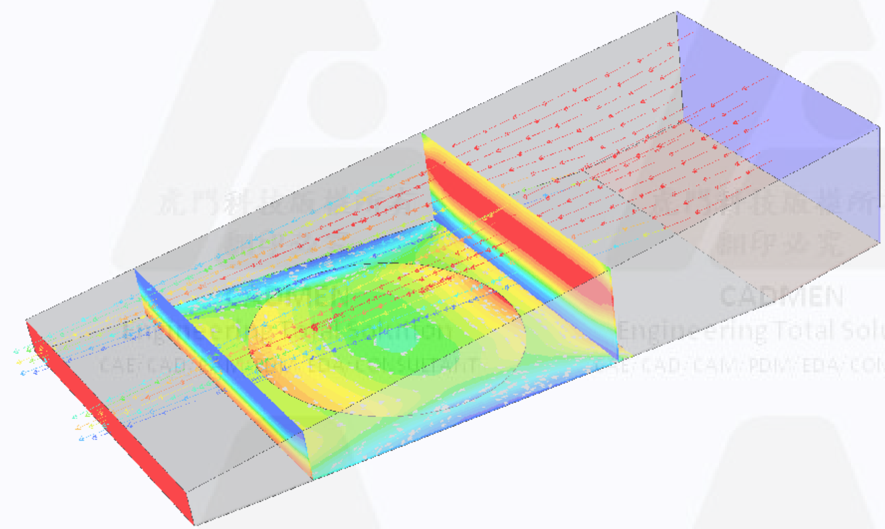
**CHEMKIN for CFD**
**Stiff Equation Solvers**
**Gas and Surface Chemistry**
**Property databases**
**Multicomponent Transport**



# GaAs Deposition

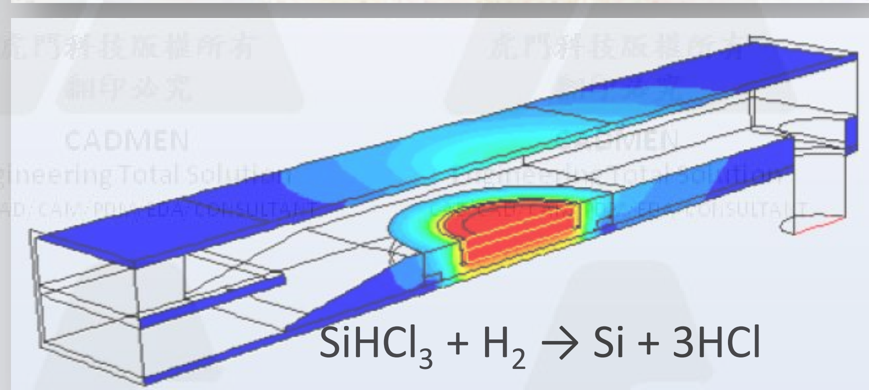
Deposition in a horizontal reactor with tilted susceptor and rotating substrate

- Boundary Conditions-
- Operating pressure = 70 Torr
- Inlet;  V=0.4m/s, T=298K
- Outlet; pressure = 70 Torr
- Substrate rotational speed = 50 RPM
- Susceptor and substrate at T= 913 K
- and surface reactions
- 11 gas phase + 25 surface reactions

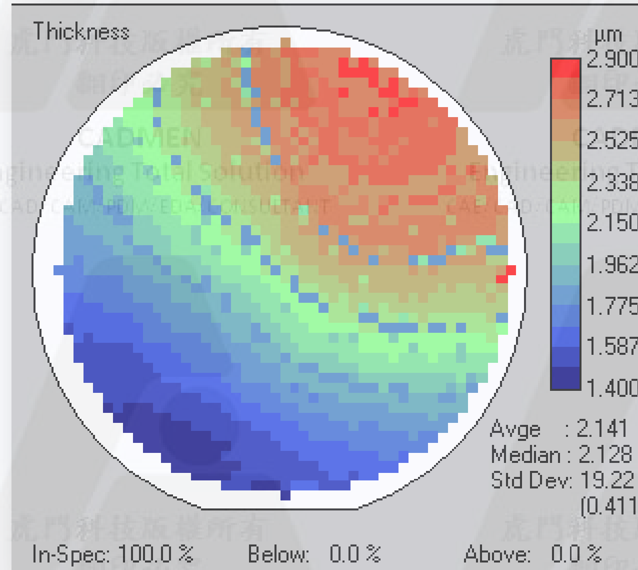


*GaAs growth on substrate and parasitic deposition on reactor walls*

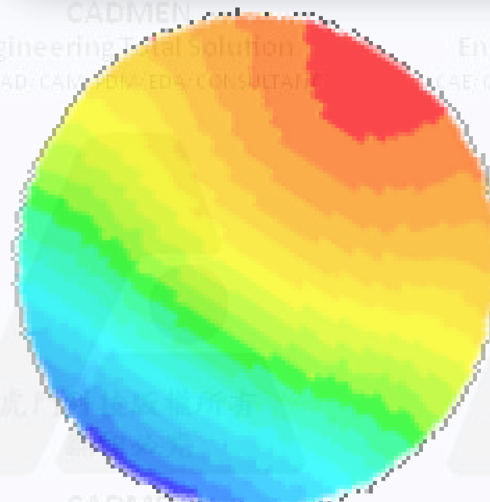
# Example: Aixtron 200 Horizontal Reactor




Experiment



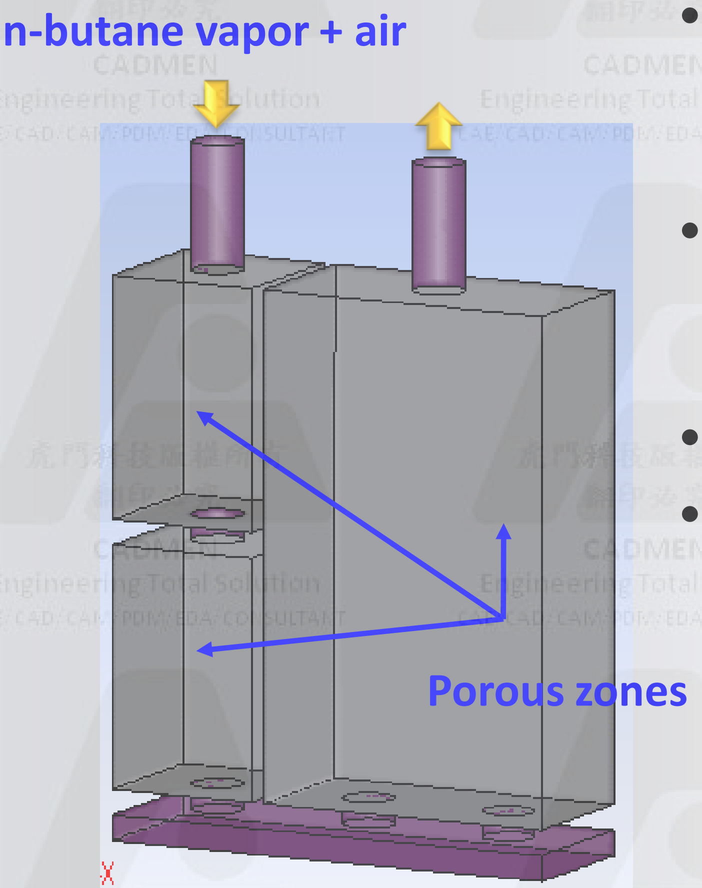

Simulation



- **Growth profile predicted by CFD is in excellent comparison with that of experiment**
- **Bending of iso-thickness lines is nicely captured**

# Example: Carbon Canister Modeling




- n-butane vapor and air enter at the inlet
  - Mass flow rate = 1e-5 kg/s
  - n-butane mass fraction = 0.675
- n-butane is captured at three porous zones using surface reactions
  - Made up reaction rates
- Outer walls: adiabatic
- Fluid and solid temperatures in the porous zones are monitored using non equilibrium thermal model

# Non Equilibrium Thermal Model

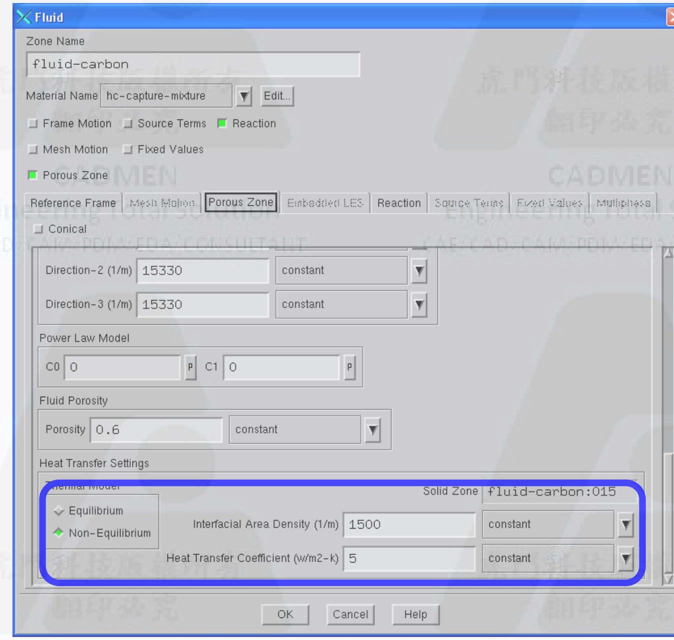
ANSYS

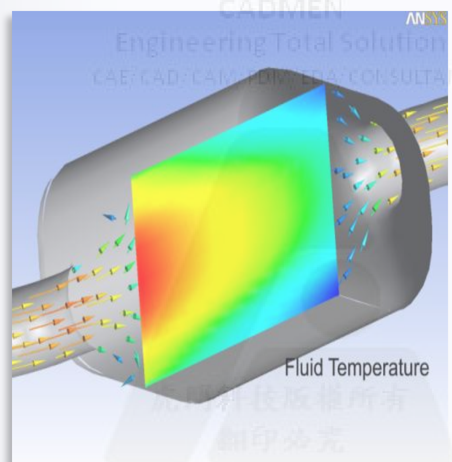
## Inputs

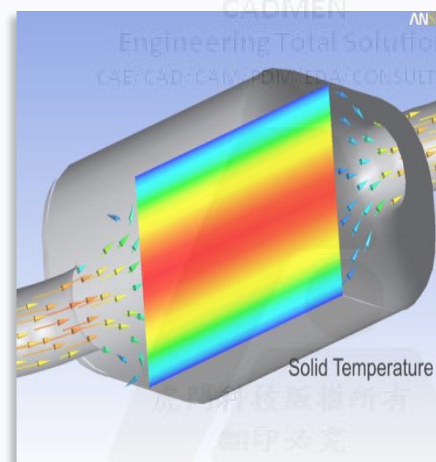
- Interfacial area density
  - Surface area to volume ratio
- Heat transfer coefficient
- *Note*: Set surface area to volume ratio in reaction tab as well

## Solver settings

- Momentum, species and energy second order
- Time step size: 10s
- Total flow time: ~1500s

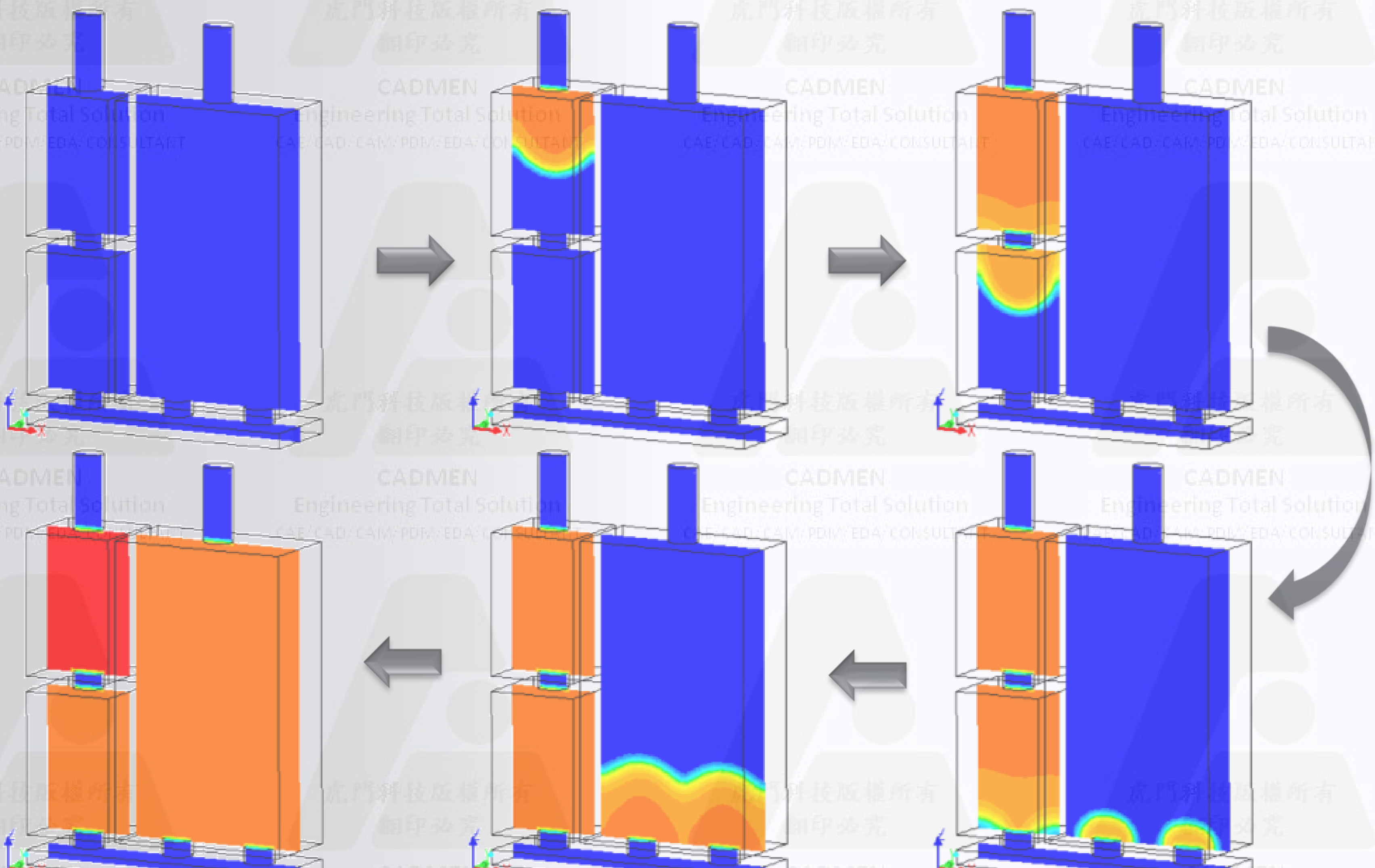


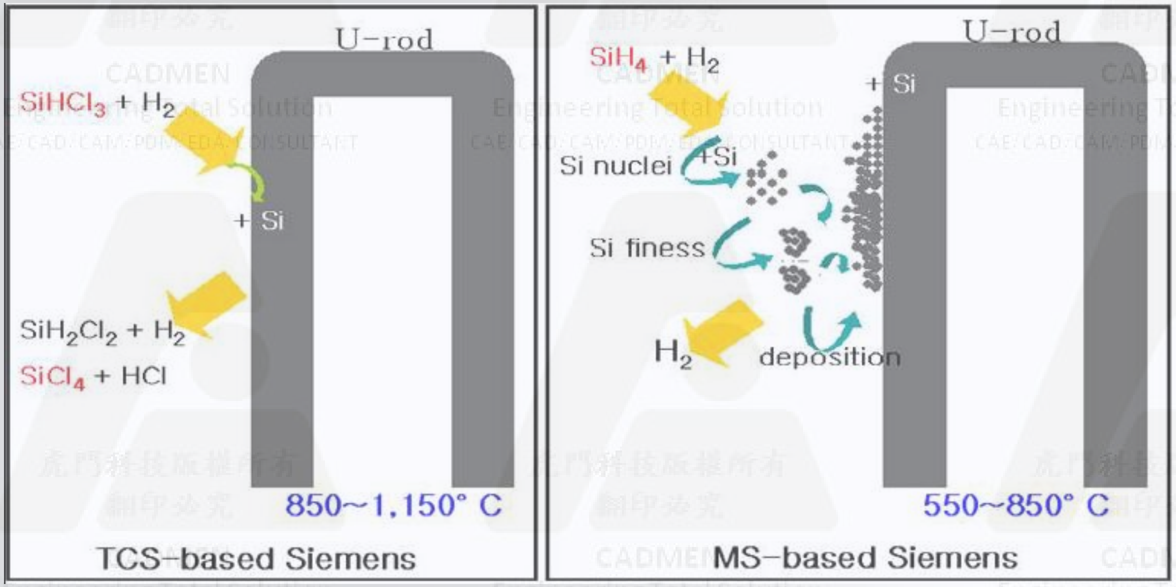

**Solid and fluid temperature fields in a catalytic converter**

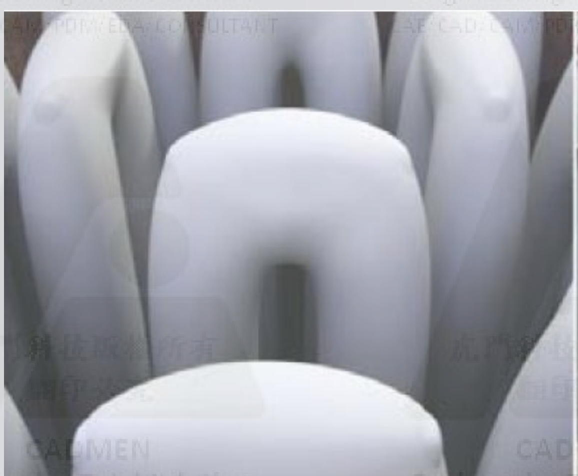
# Hydrocarbon (N-butane) Capture



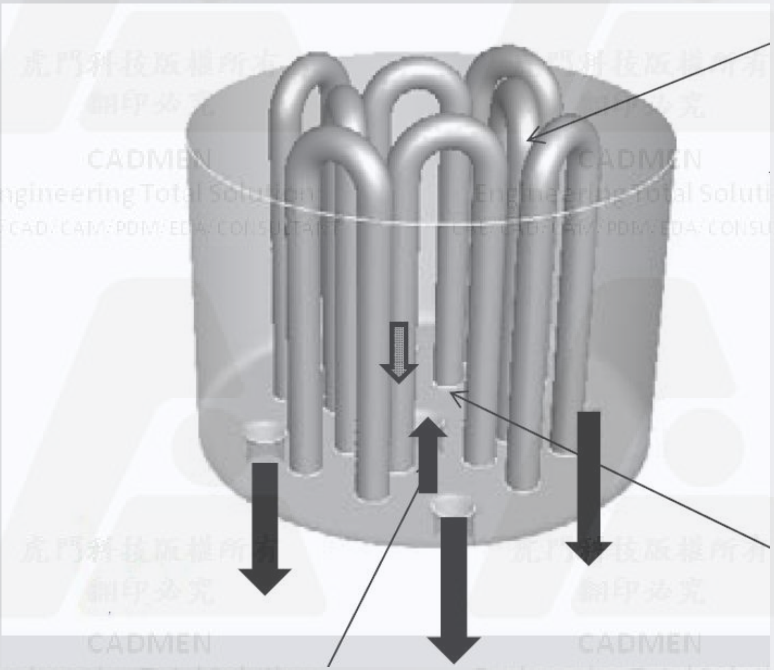
  

# 太陽能多晶矽反應器熱流分析（Siemens 製法）



# CVD和表面反應（Siemens 製法）



V:5[m/s]
T:300[k]
$AsH_3$:0.4
$Ga(CH_3)_3$:0.15

**Deposition species at wall surface : As, Ga**

*reaction-1*
AsH3+Ga(solid) →Ga + As(solid) + H2

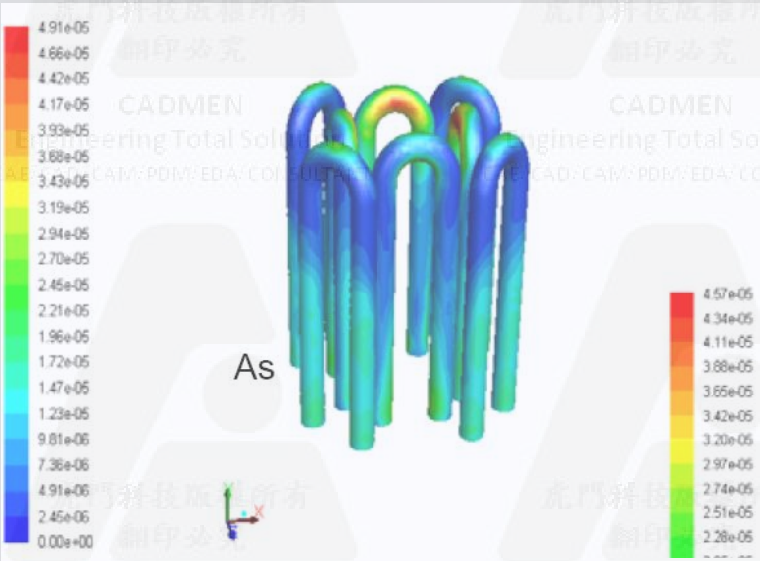
*reaction-2*
Ga(CH3)3 + As(solid) →As + Ga(solid) + CH3

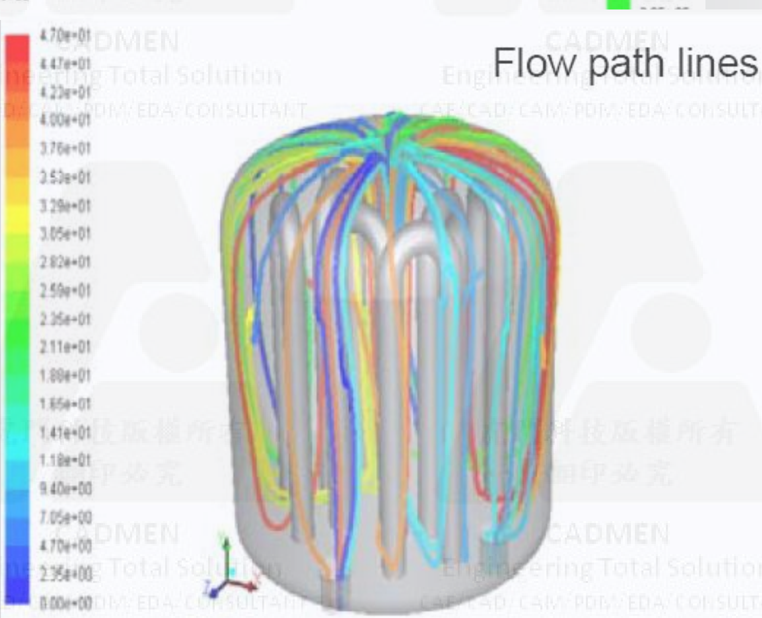
Reaction-1 : 1e6*T0.5
Reaction-2 : 1e12*T0.5

**Operating pressure : 10,000[Pa]**

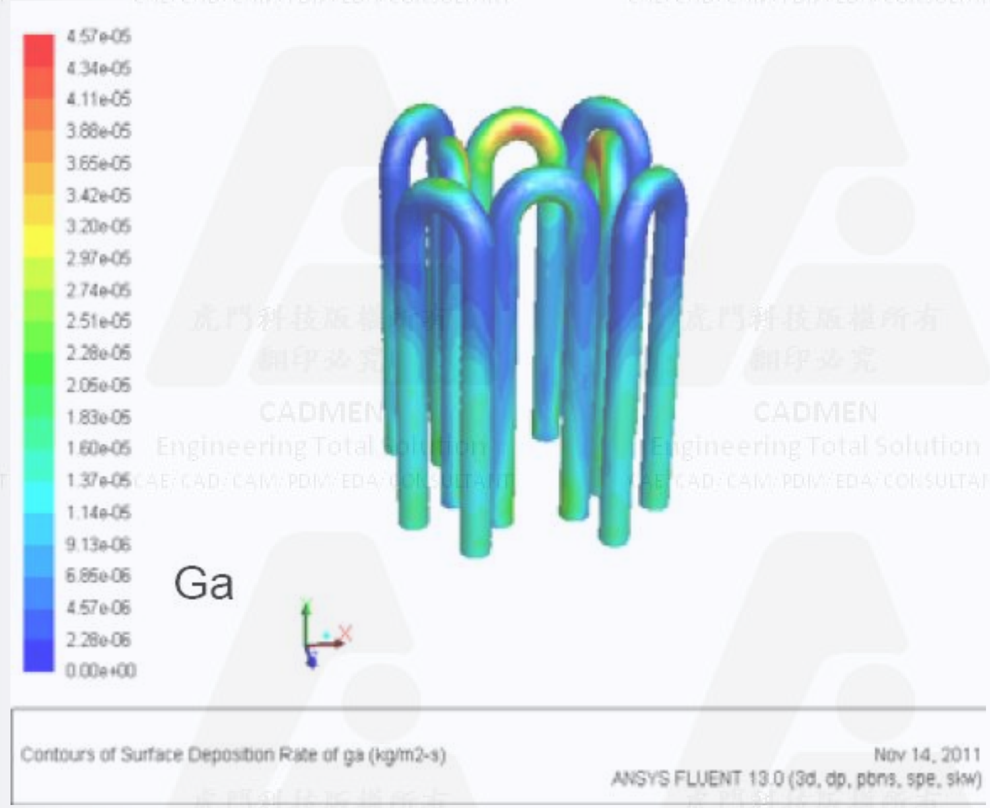
Mixture Density: ideal gas law
Cp : mixing Las
Thermal Conductivity, Viscosity: Ideal gas mixing law
Mass diffusivity, Thermal Diffusion : Kinetic theory

# CVD和表面反應（Siemens 製法）



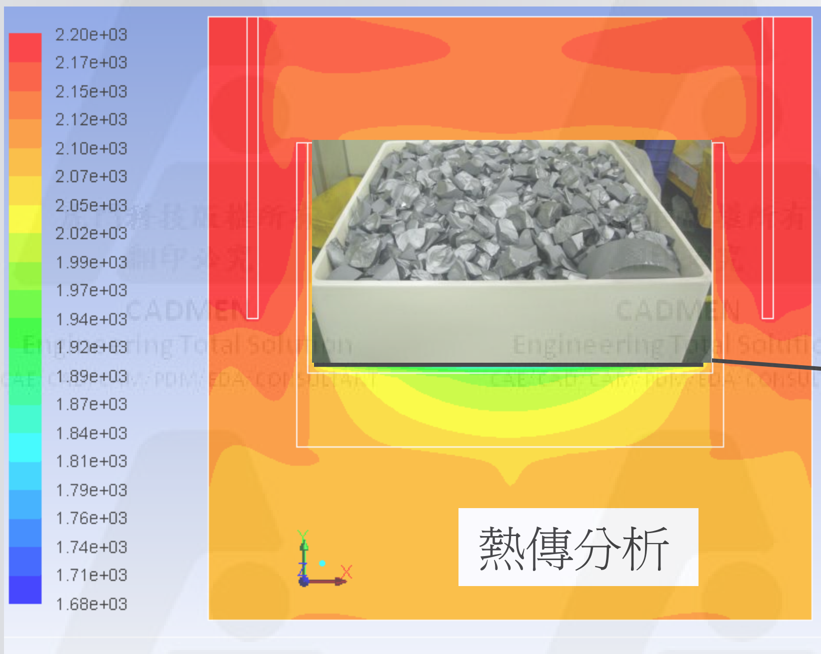

Deposition species rate[kg/m2/s]



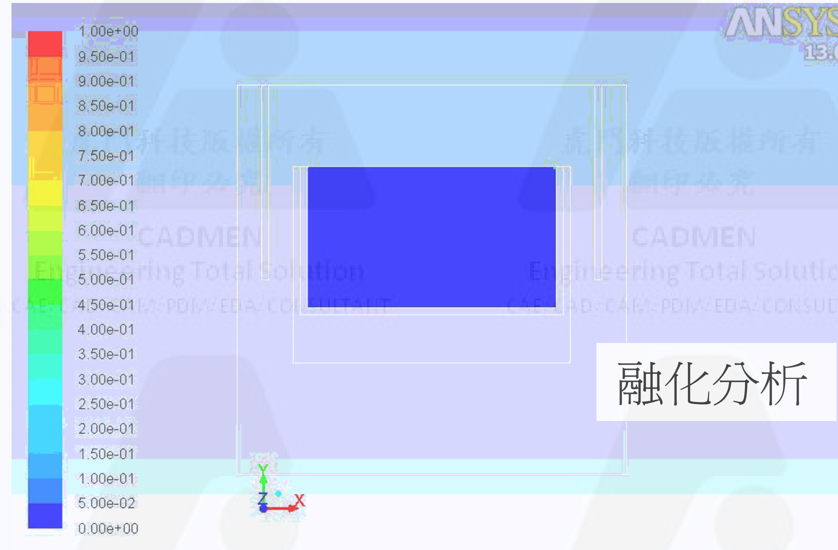
# 固液相變化分析

## 定義加熱器溫度爲2200K

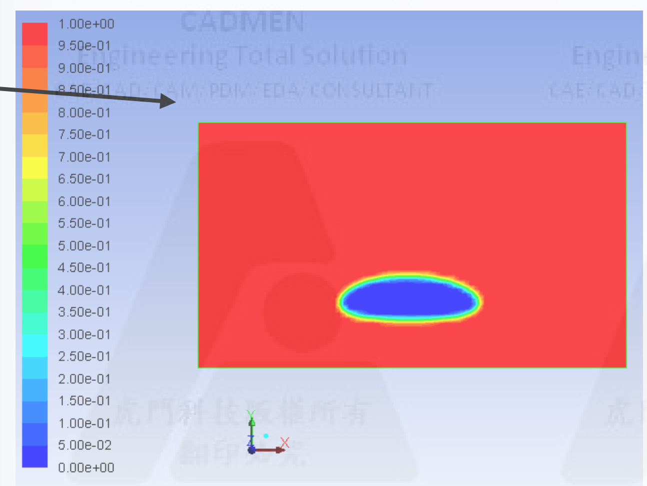
## Temperature Field and Liquid Fraction



Contours of Static Temperature (mixture) (k) (Time=8.1000e+03)



Contours of Liquid Fraction (mixture) (Time=1.5000e+01)
Feb 17, 2011
ANSYS FLUENT 13.0 (2d, pbns, vof, ske, transient)



Contours of Liquid Fraction (mixture) (Time=8.1000e+03)

# 選擇性催化還原脫氮 (SCR)

- 通過NO與氨(NH3)的還原反應

  $$4NO + 4NH_3 + O_2 \rightarrow 6H_2O + 4N_2$$

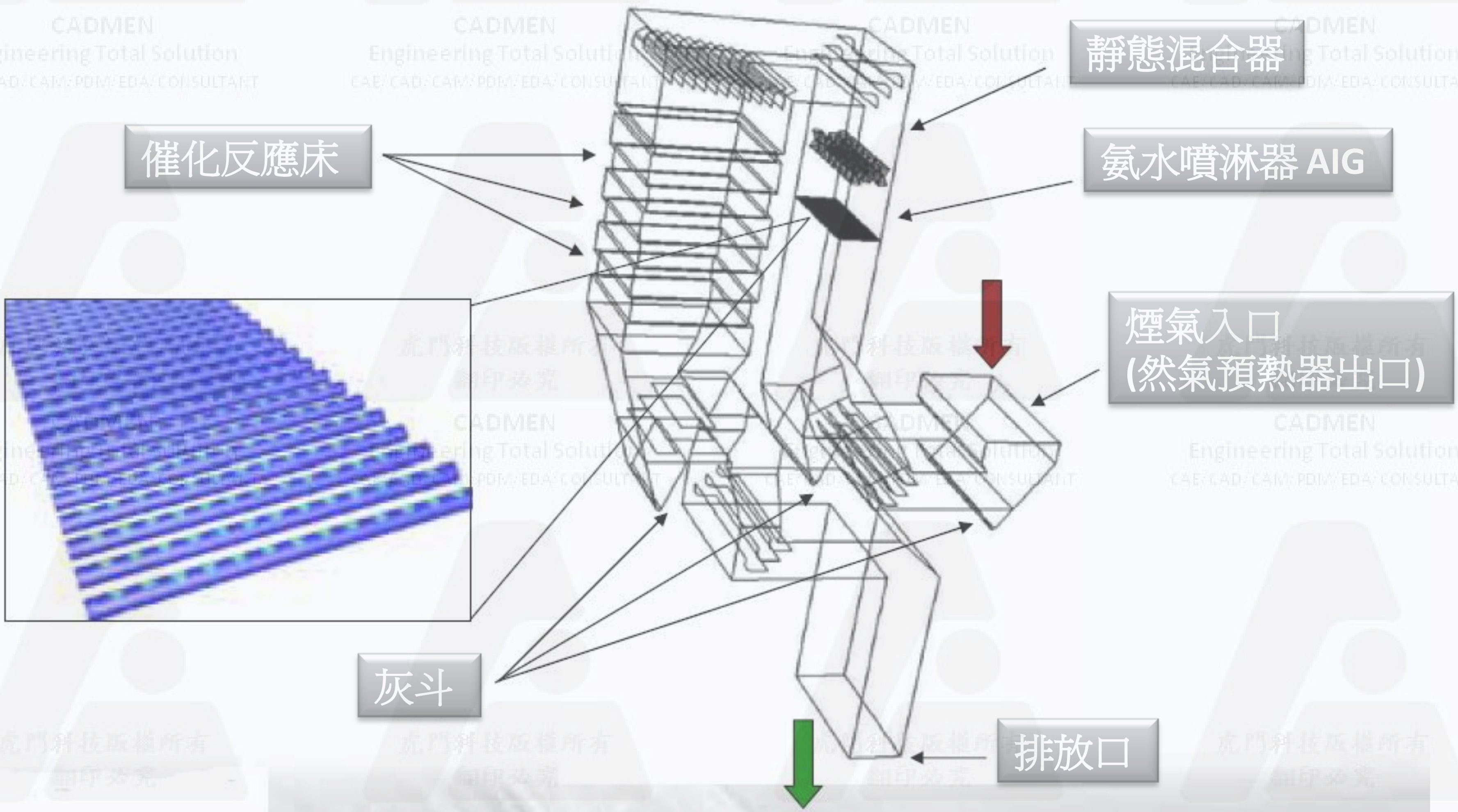
- 通常煙氣溫度(200~400℃)下還原反應緩慢
  -通過催化劑加速反應

- NH3噴淋– 蒸發– 混合– 反應
  - 脫氮效率70% ~ 90%

# 選擇性催化還原脫氮 (SCR)

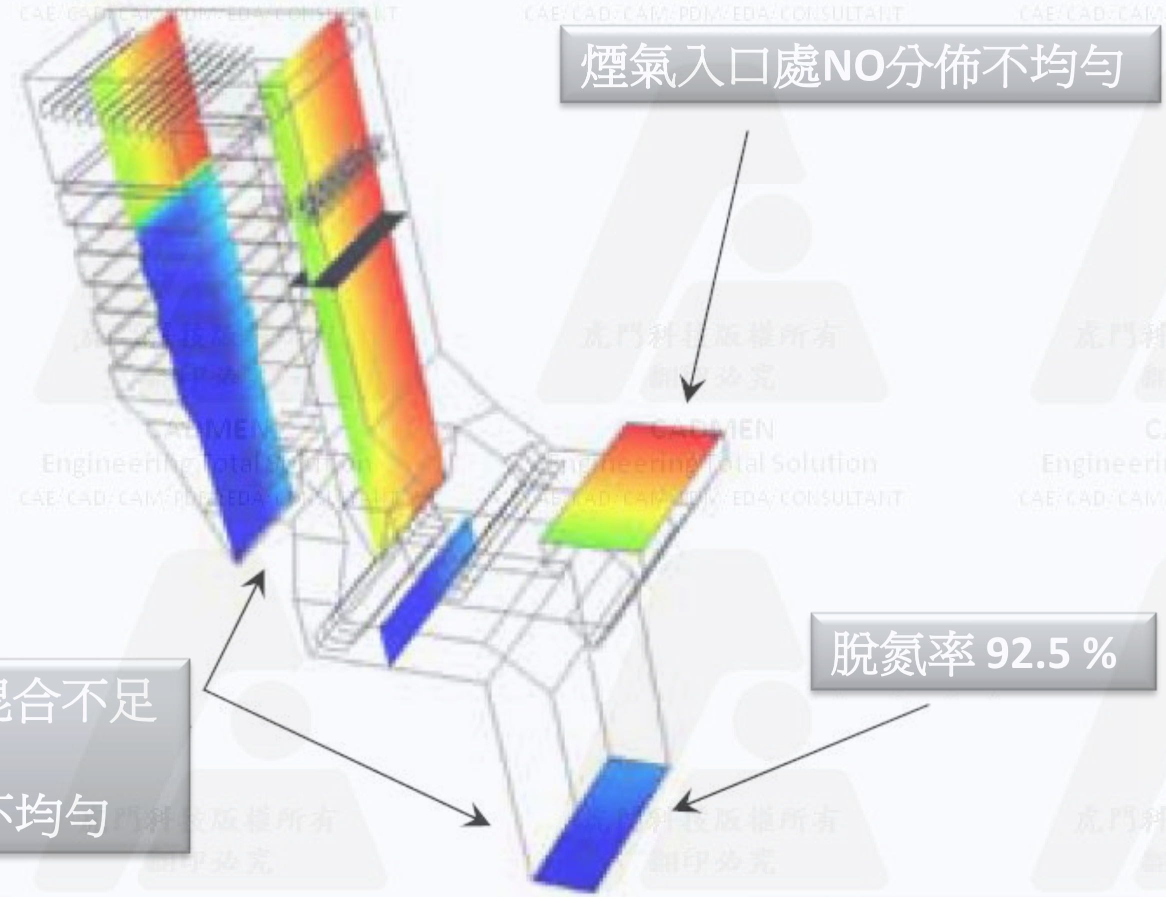
典型燃煤鍋爐SCR裝置



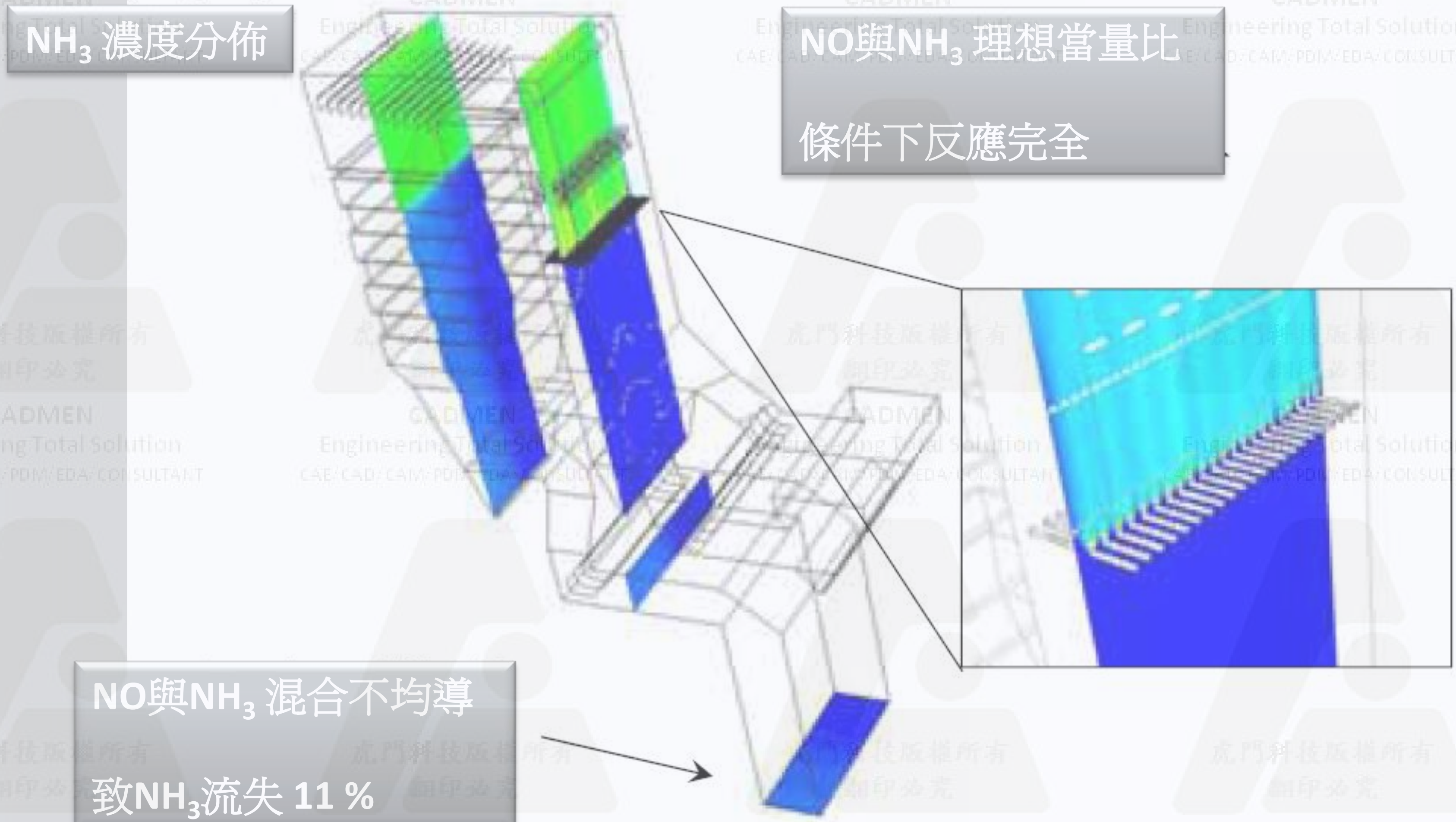
# 選擇性催化還原脫氮 (SCR)

典型燃煤鍋爐SCR裝置



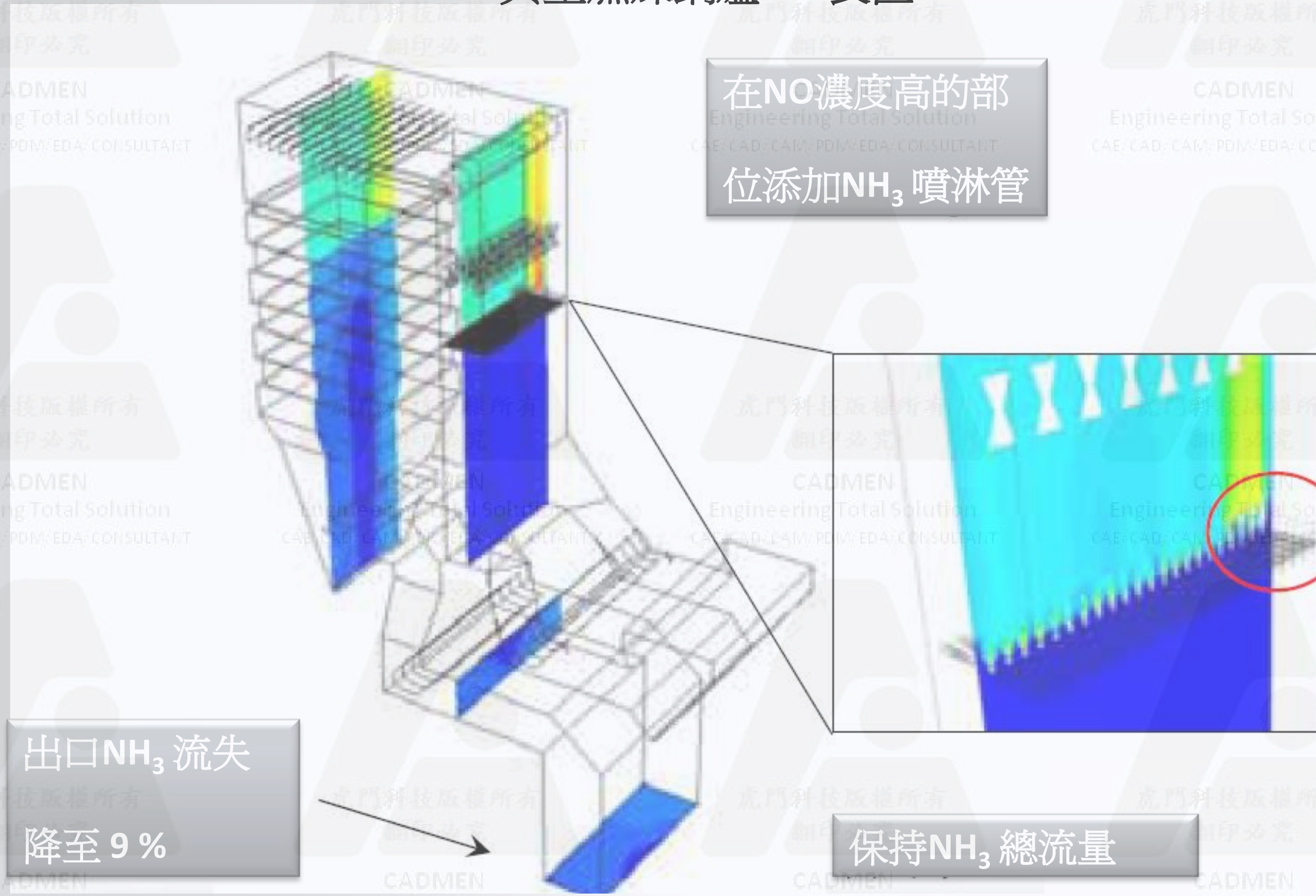
# 選擇性催化還原脫氮 (SCR)

## 典型燃煤鍋爐SCR裝置



# 選擇性催化還原脫氮 (SCR)

## 典型燃煤鍋爐SCR裝置



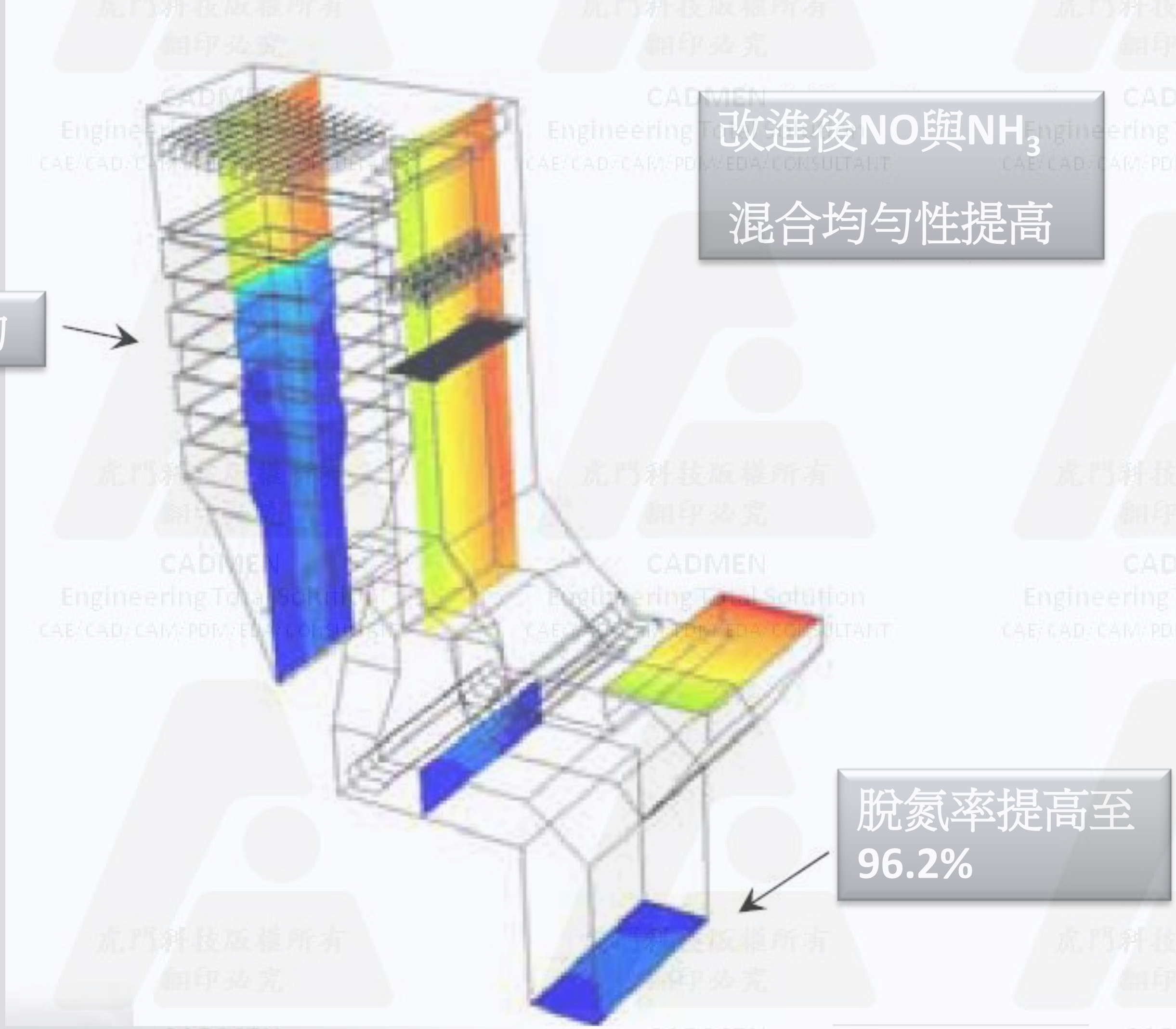
在NO濃度高的部位添加$NH_3$ 噴淋管

出口$NH_3$ 流失降至 9 %

保持$NH_3$ 總流量

# 選擇性催化還原脫氮 (SCR)

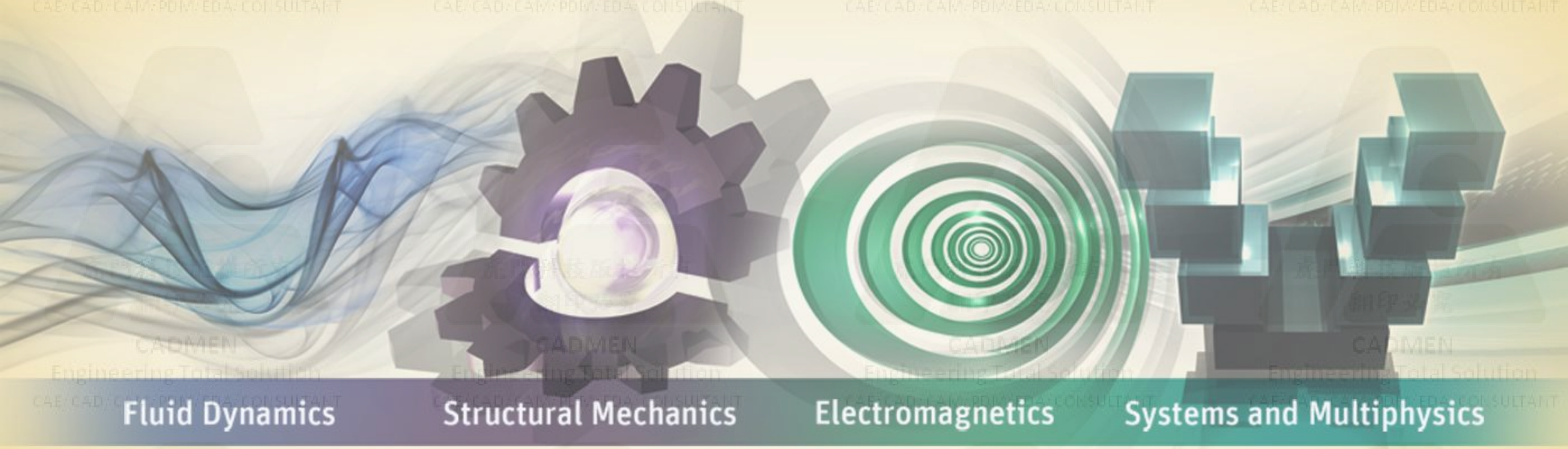
典型燃煤鍋爐SCR裝置

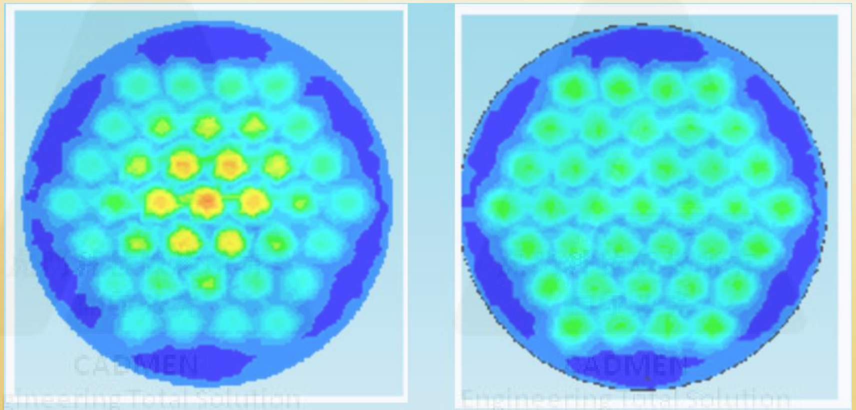


ANSYS

Realize Your Product Promise™

# Heat Exchanger 熱交換器CFD分析(操作)



Maldistributed

Improved

李龍育 Dragon
CFD技術副理
虎門科技

# 問題描述

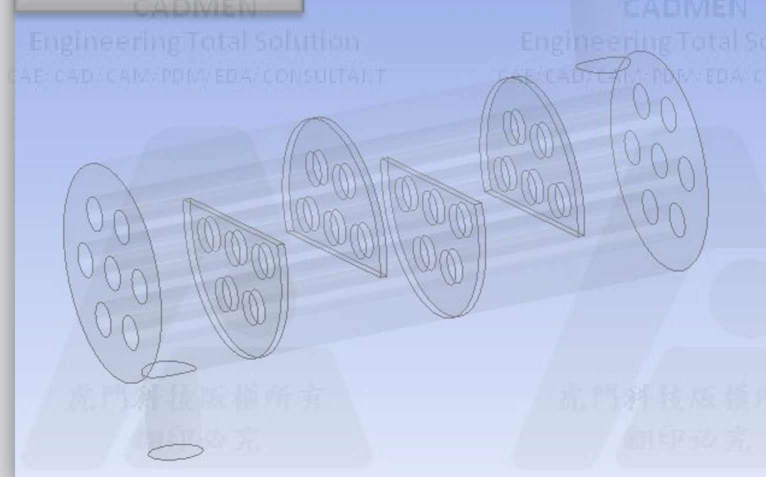
分析一熱交換器將熱空氣透過較冷之水做散熱，以降低出口的空氣溫度

## 幾何處理

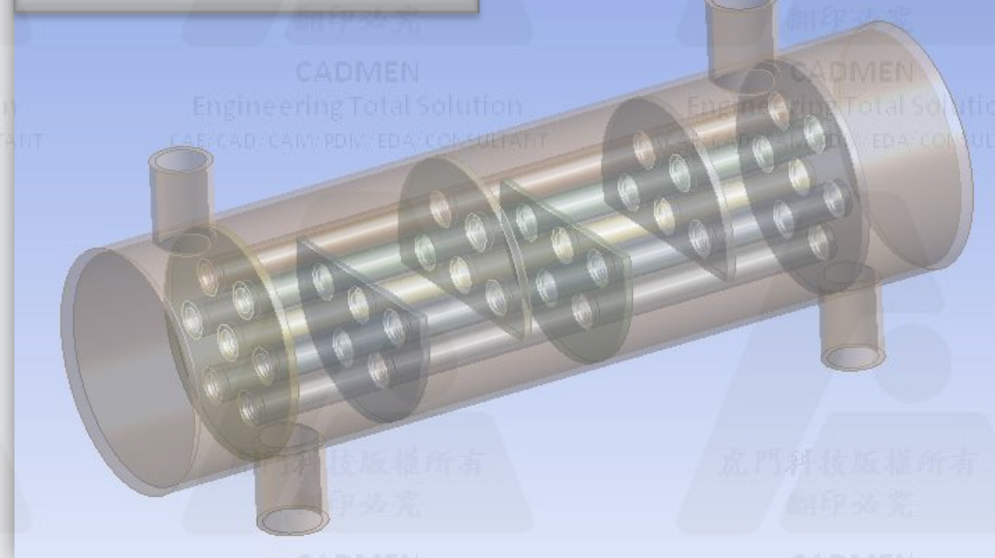
利用**Designmodeler (DM)**建構出熱交換器分析模型，並同時將流體區萃取建立

熱空氣區域

冷水區域

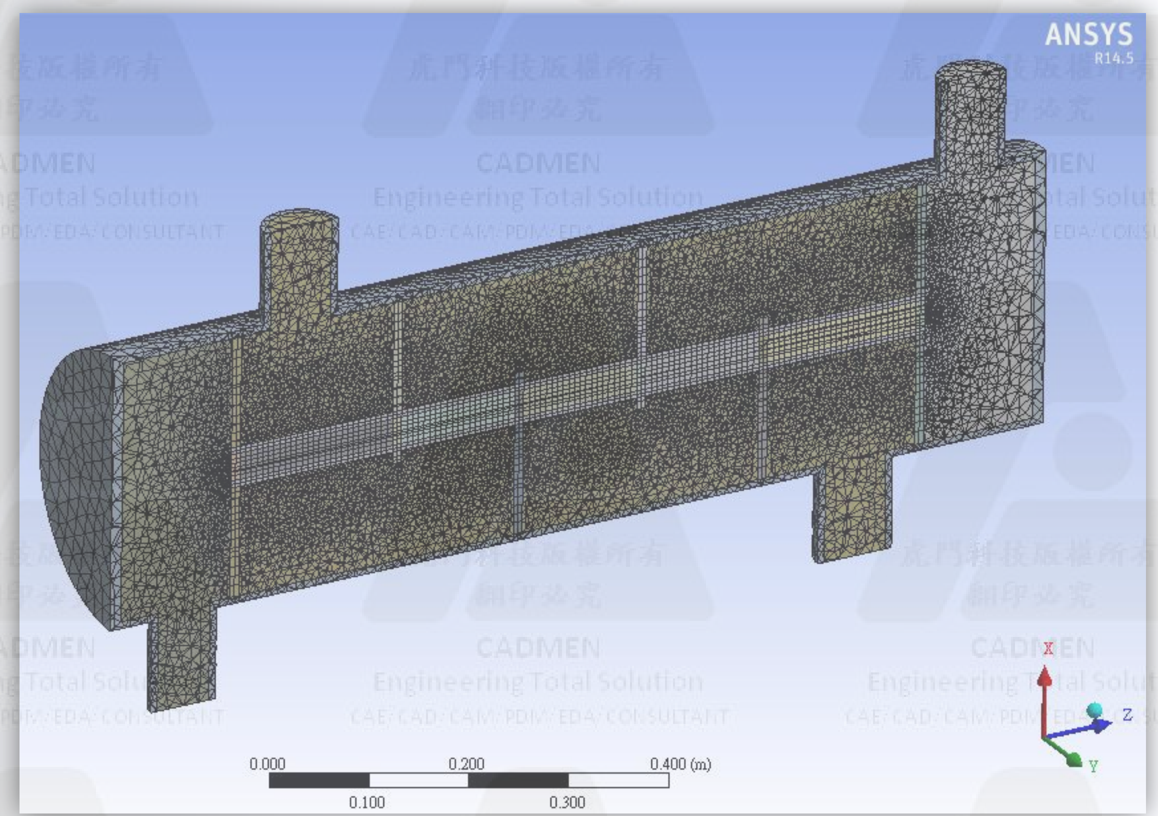


熱交換器分析模型



# 網格建構

利用ANSYS Meshing (AM)執行熱交換器分析模型之網格建構，設定全域與局部網格參數控制，即可執行網格產生程序，總網格數約100萬，品質亦在標準內




Details of "Mesh"

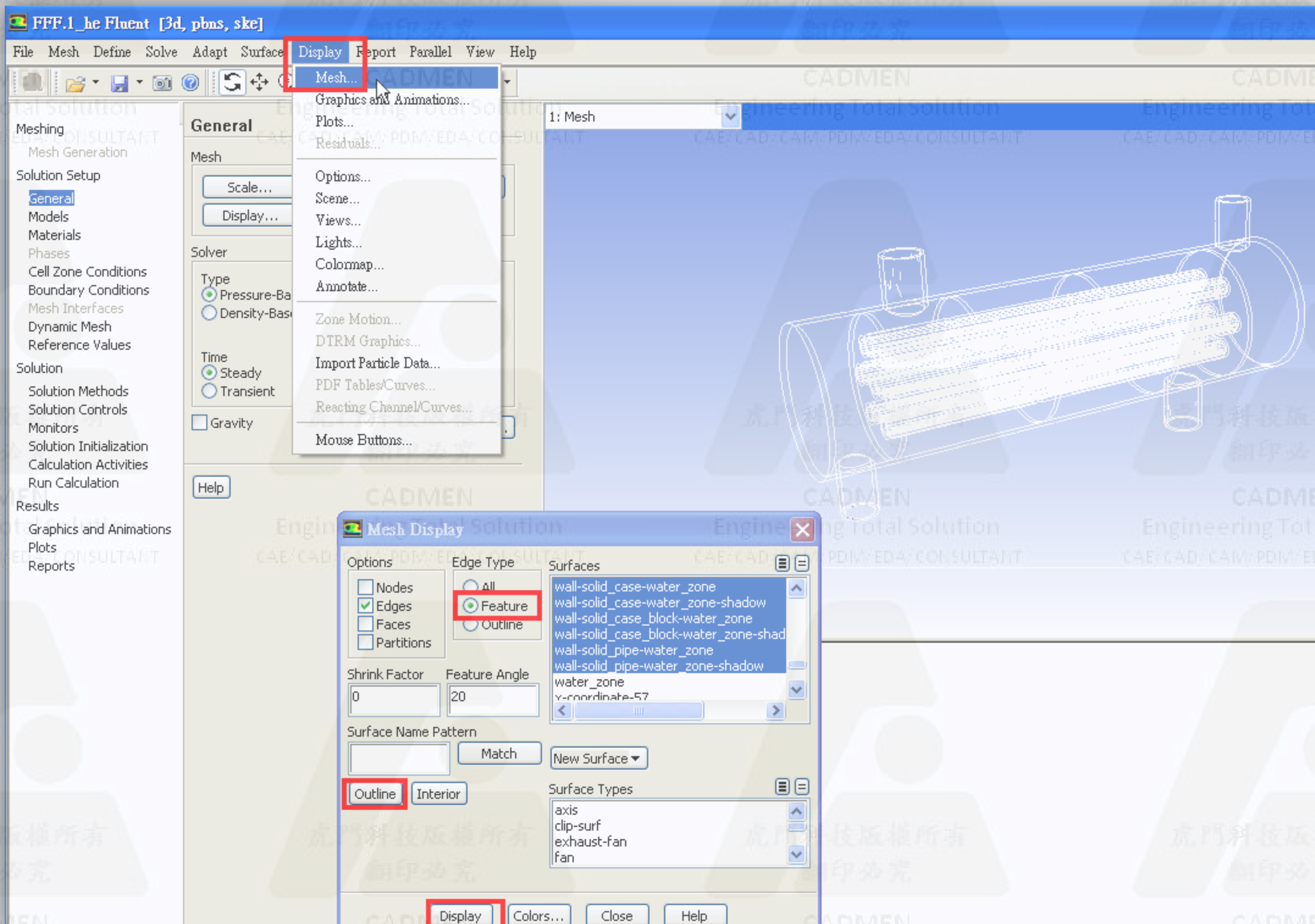
| | |
|---|---|
| **Defaults** | |
| Physics Preference | CFD |
| Solver Preference | Fluent |
| Relevance | 80 |
| **Sizing** | |
| Use Advanced Size Function | On: Curvature |
| Relevance Center | Medium |
| Initial Size Seed | Active Assembly |
| Smoothing | Medium |
| Transition | Slow |
| Span Angle Center | Fine |
| Curvature Normal Angle | 24.0° |
| Min Size | Default (2.0559e-004 m) |
| Max Face Size | Default (2.0559e-002 m) |
| Max Size | Default (4.1119e-002 m) |
| Growth Rate | Default (1.1280 ) |
| Minimum Edge Length | 1.e-002 m |

| **Statistics** | |
|---|---|
| Nodes | 243269 |
| Elements | 1036390 |
| Mesh Metric | Skewness |
| Min | 9.52724814684558E-06 |
| Max | 0.899069722316064 |
| Average | 0.220443462868575 |
| Standard Deviation | 0.122558210453467 |

# FLUENT 設定

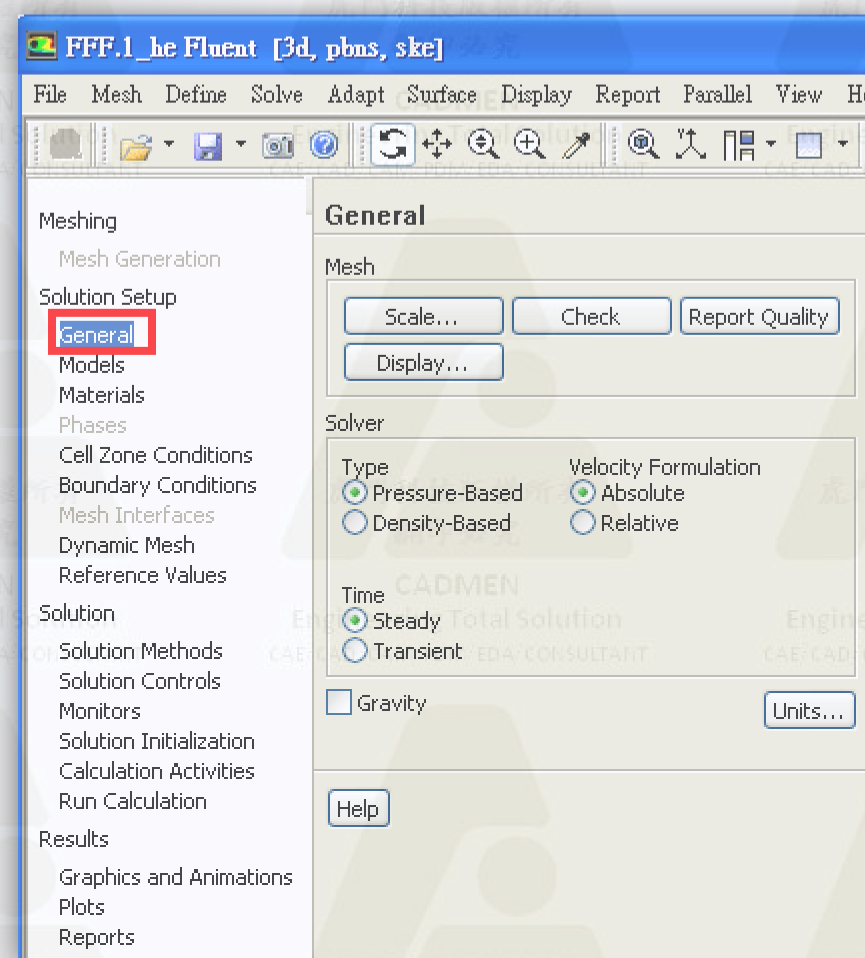
## 顯示調整



# FLUENT 設定

## 基本設定(此案例保持預設)

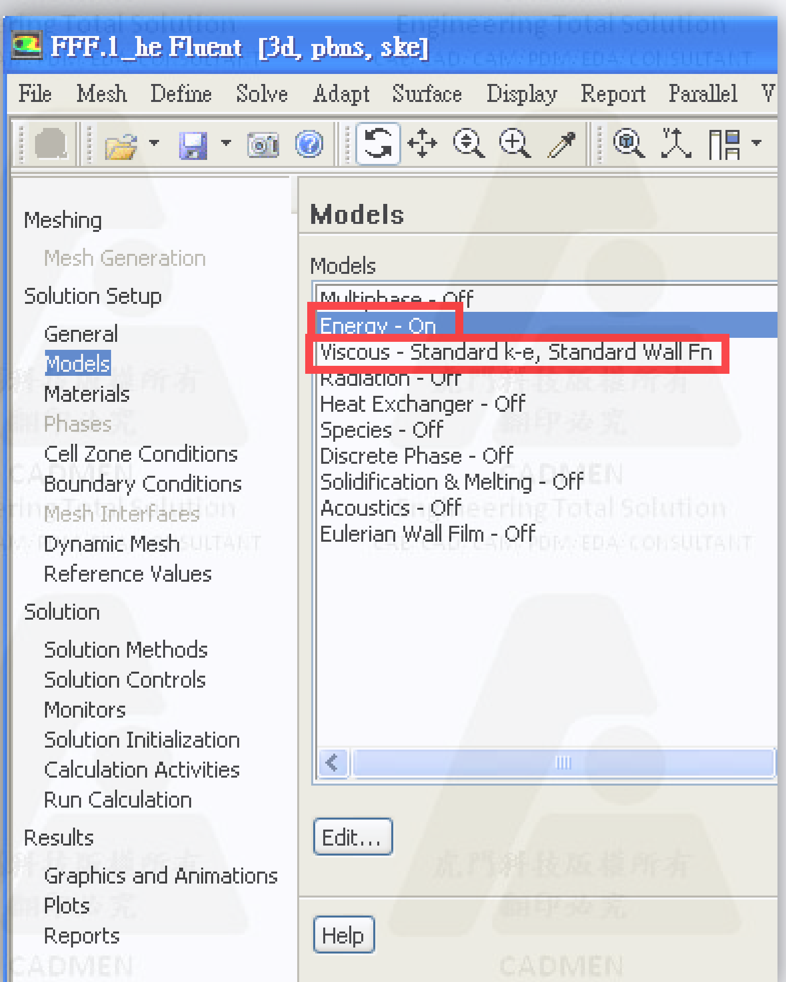
1. 單位改變
2. 求解器型式
3. 穩態或暫態求解
4. 重力考量

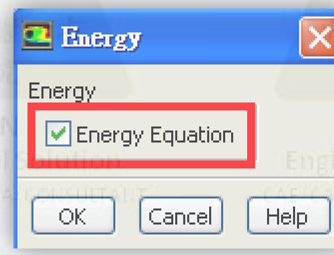


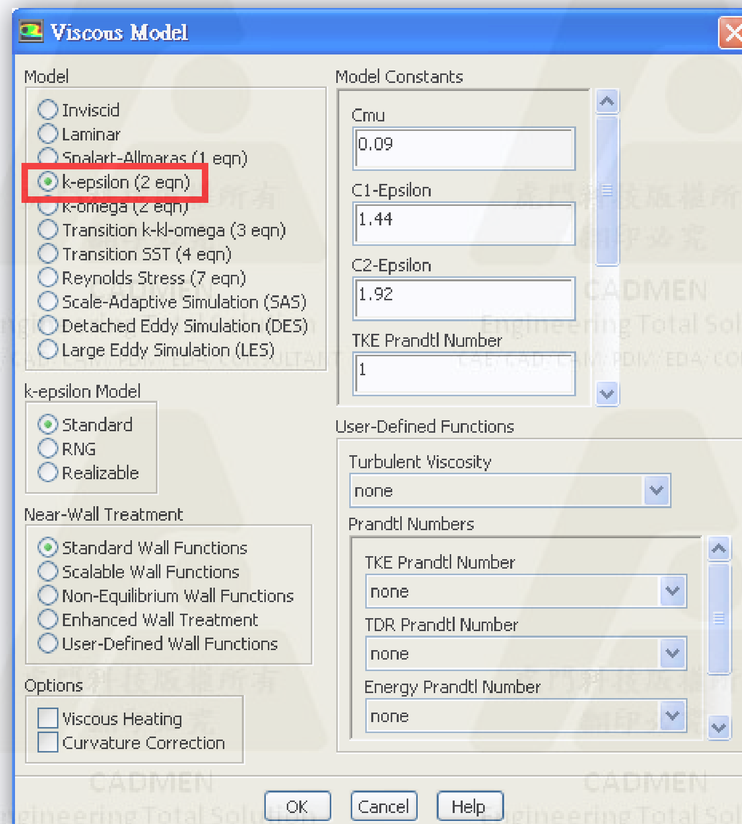
# FLUENT 設定

## 模組使用設定

### 啓動能量與紊流模組



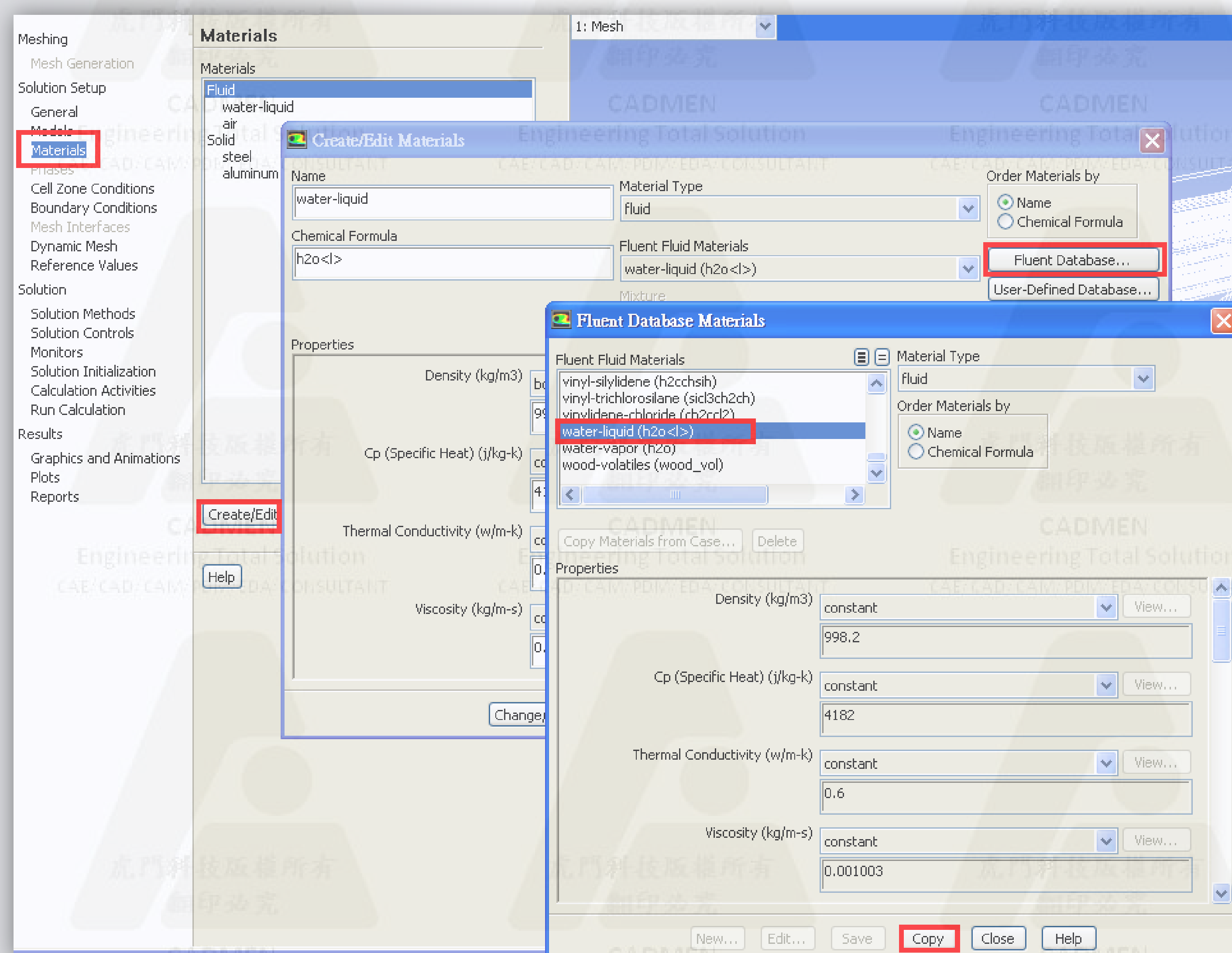



# ANSYS FLUENT 設定

## 材料設定

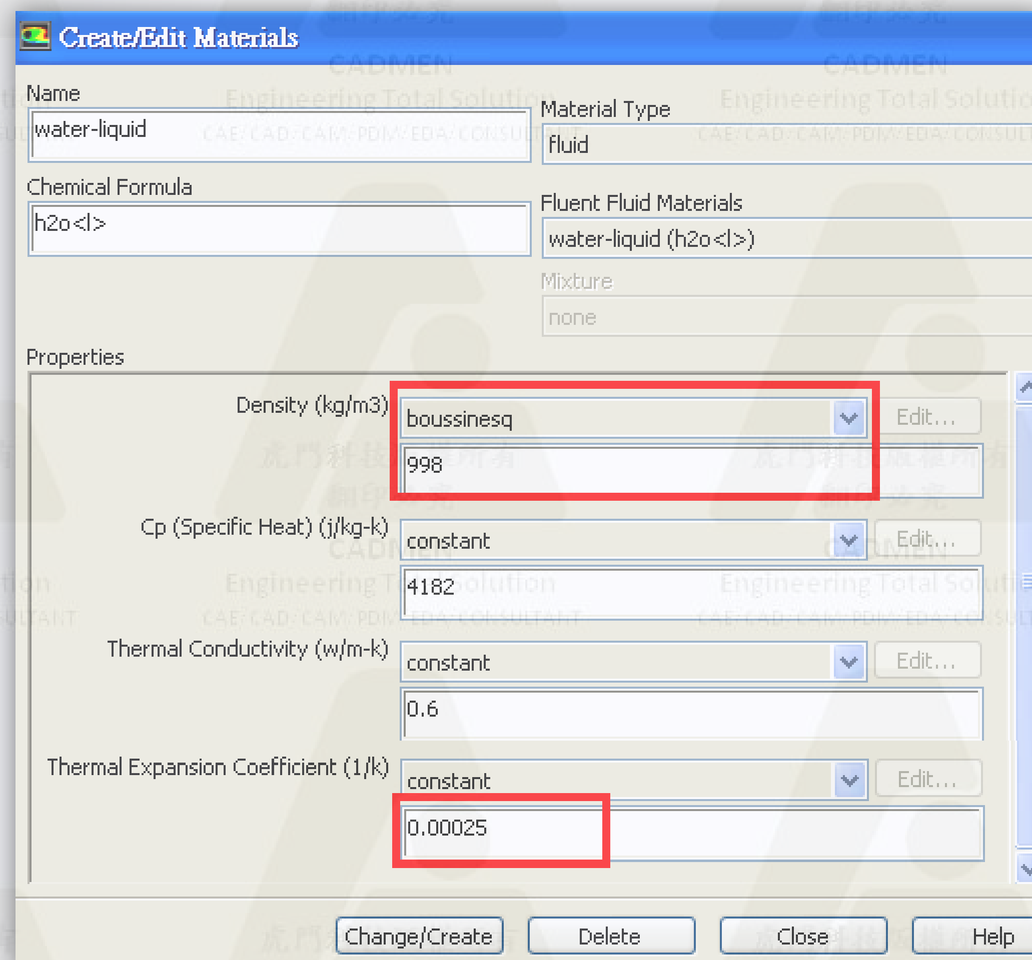
將database裡的水複製



# FLUENT 設定

## 材料設定

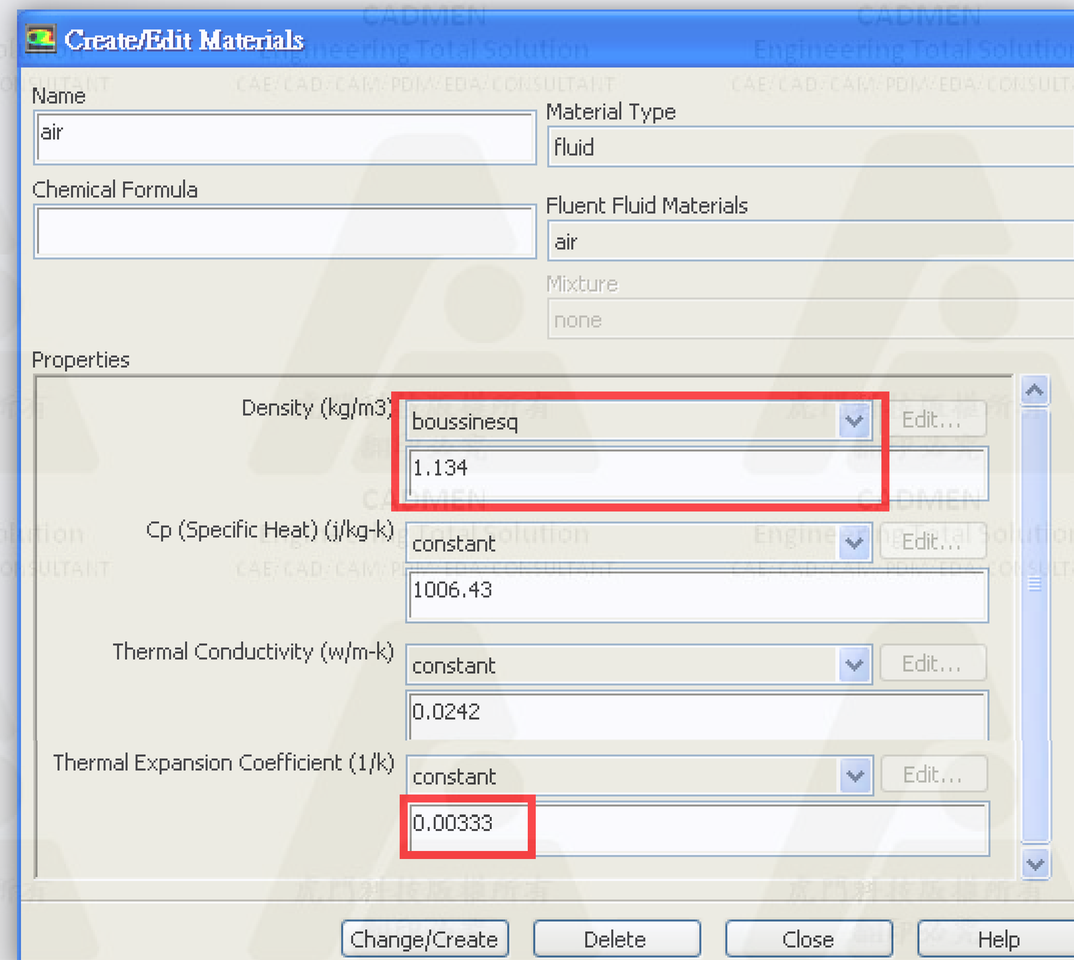
將材質水的密度選爲Boussinesq，
並設定參考密度與熱膨脹係數



# FLUENT 設定

## 材料設定

將材質空氣的密度選為 **Boussinesq**，並設定參考密度與熱膨脹係數



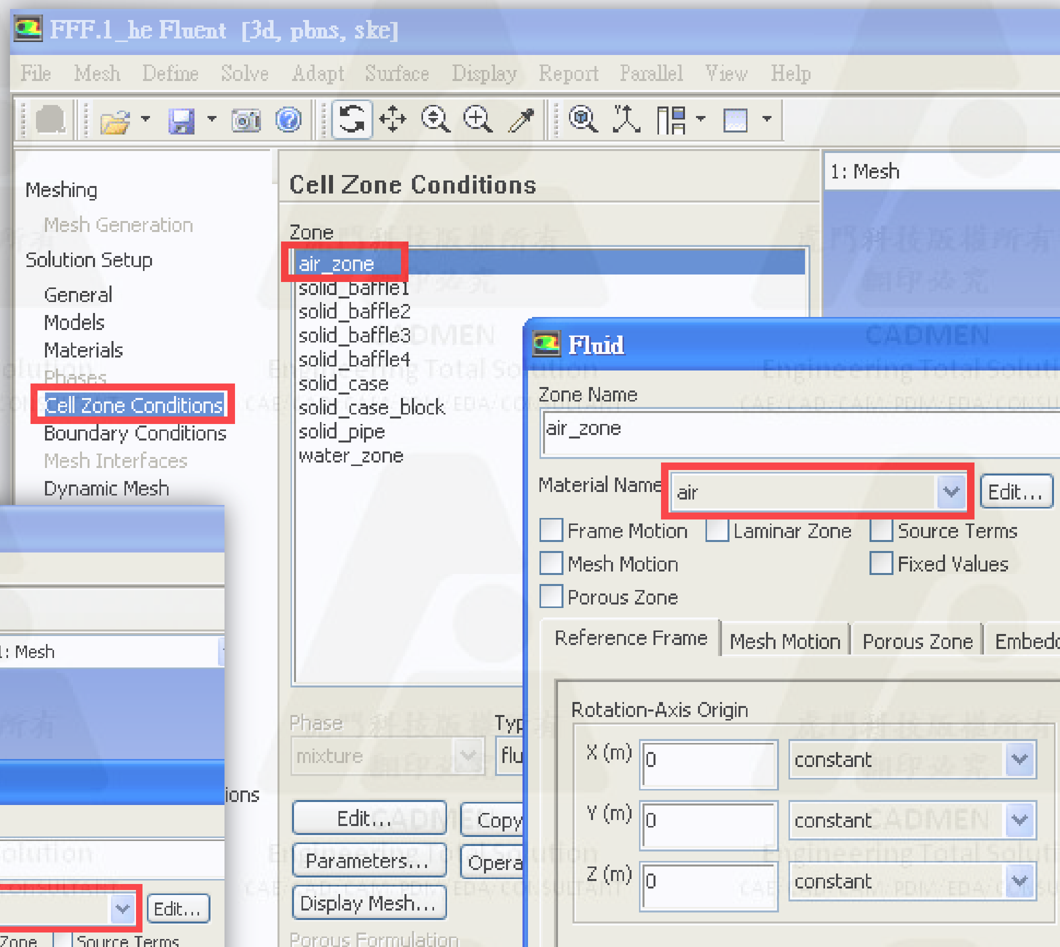

  

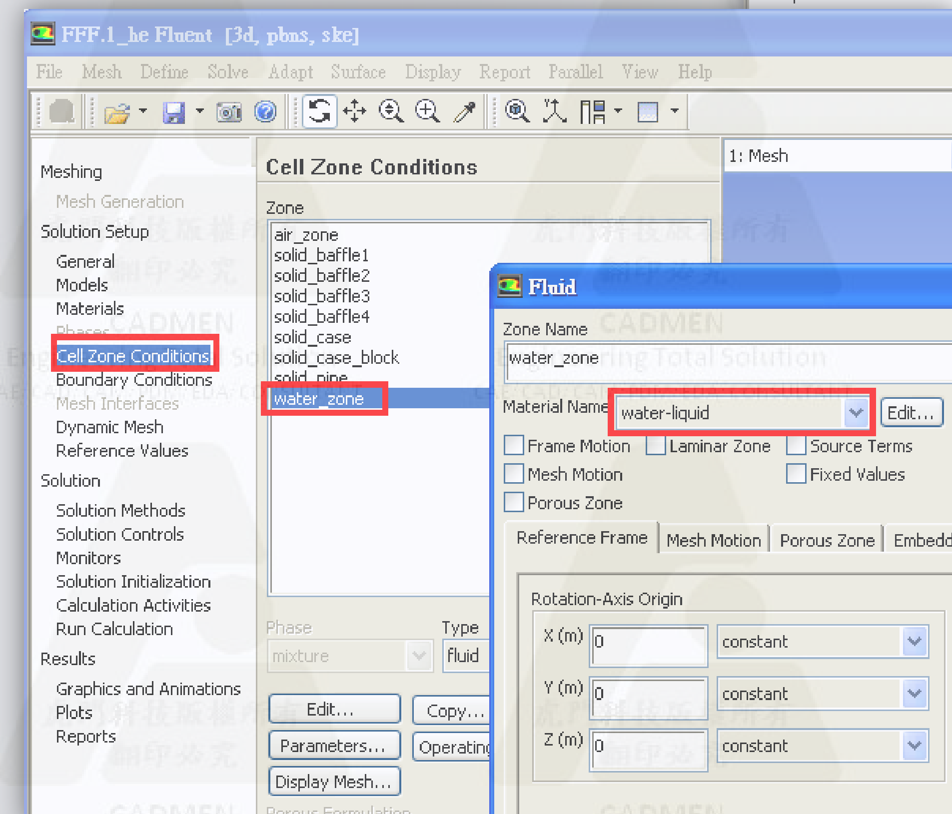
ANSYS

# FLUENT 設定

## 材料指定

將對應的材料選擇到對應的計算域中



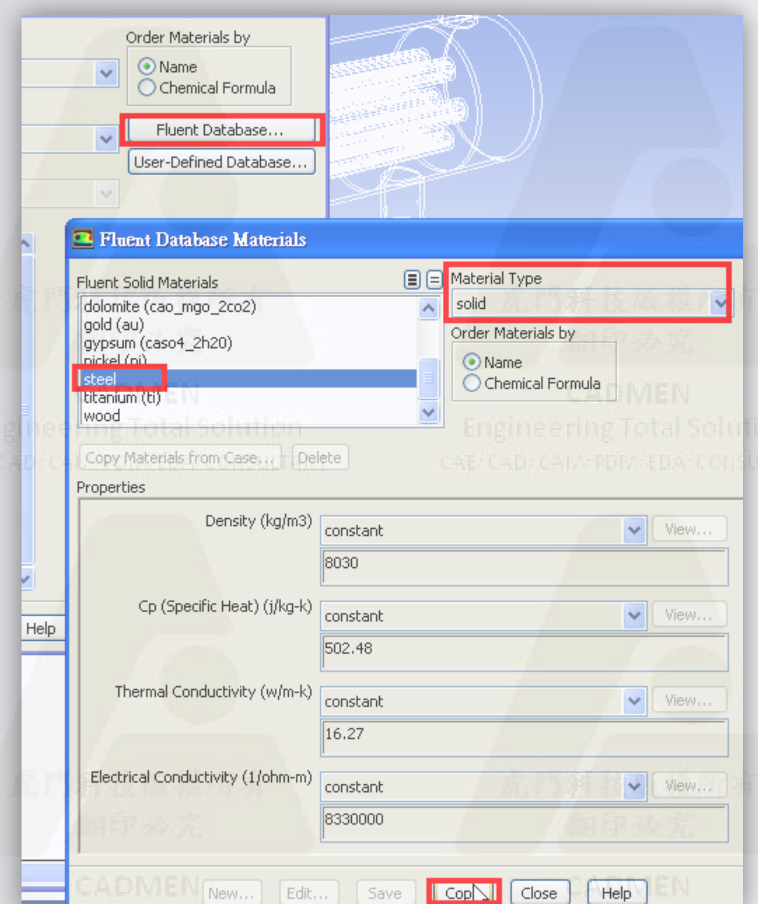



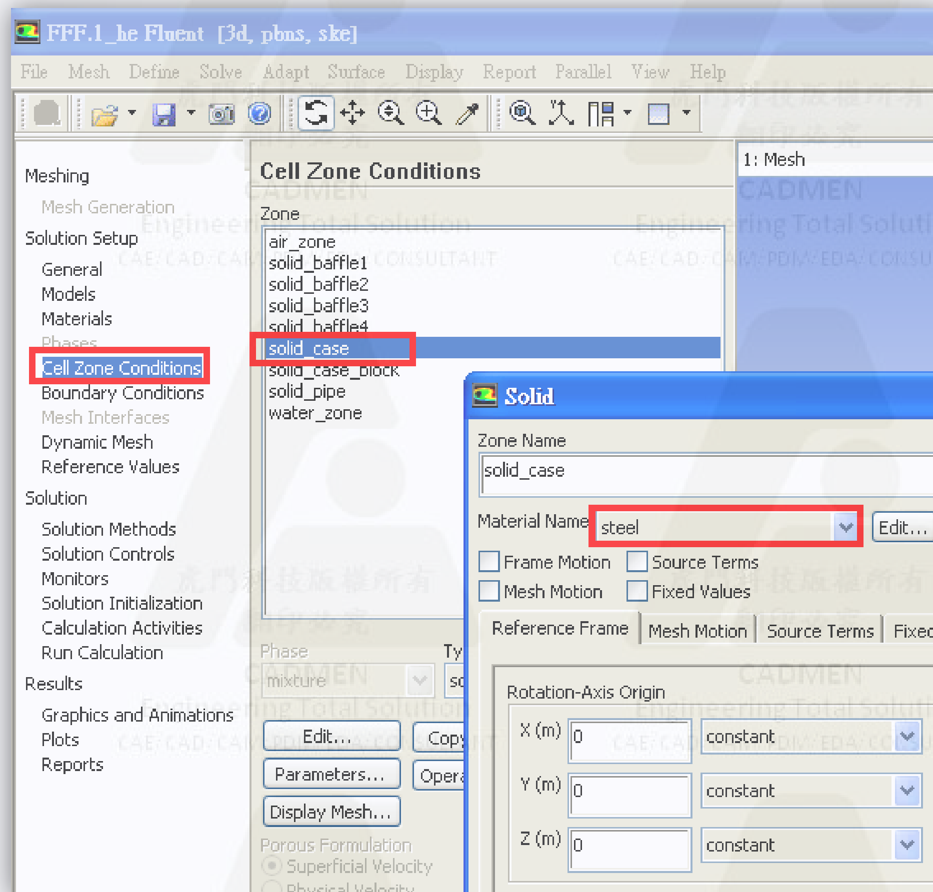

  

# ANSYS FLUENT 設定

## 材料指定

新增加鋼材，並指定到外殼的固體計算域

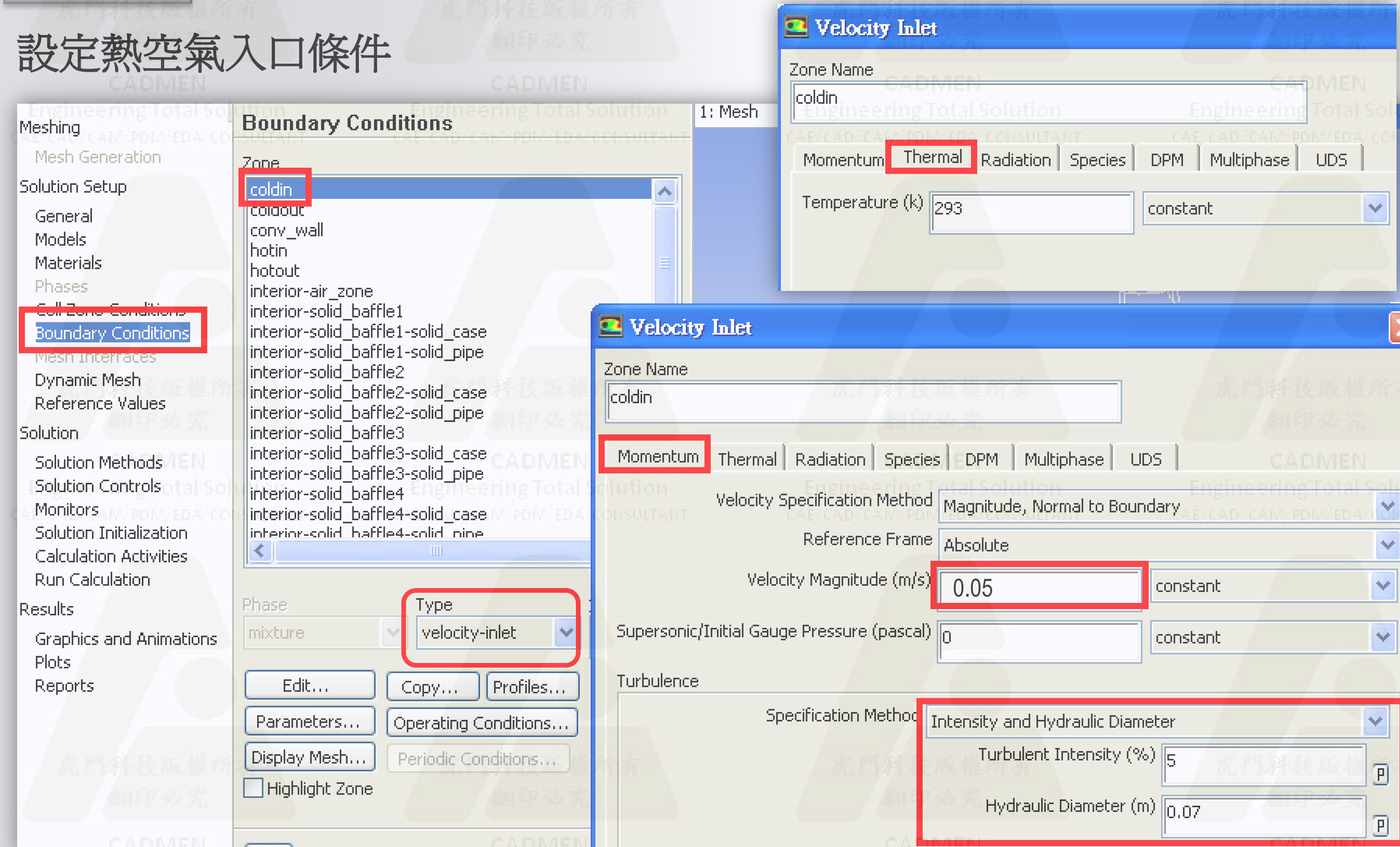


# FLUENT 設定

## 邊界設定

設定熱空氣入口條件





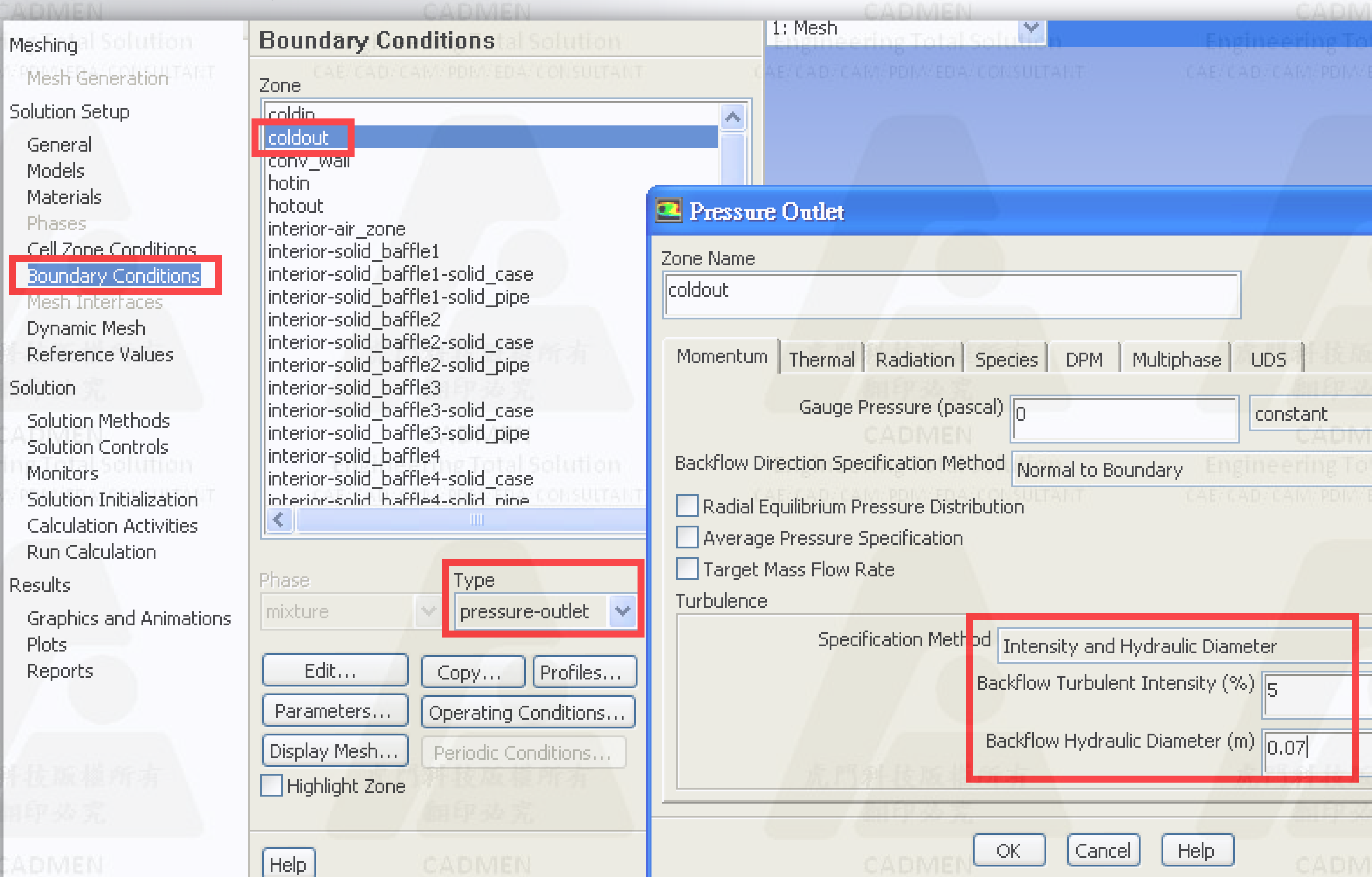

  

ANSYS

# FLUENT 設定

## 邊界設定

### 設定熱空氣出口條件

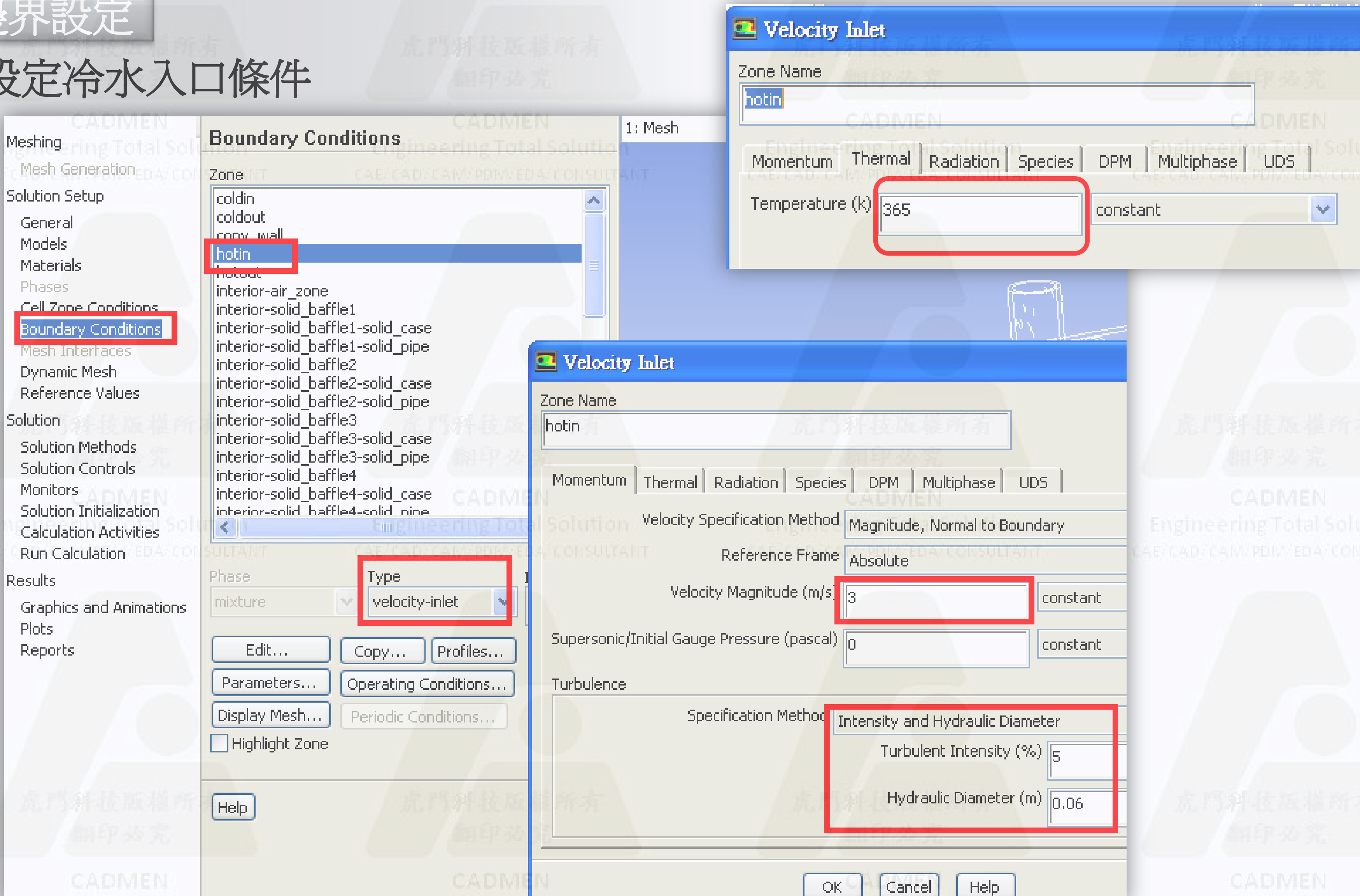


ANSYS

# FLUENT 設定

## 邊界設定

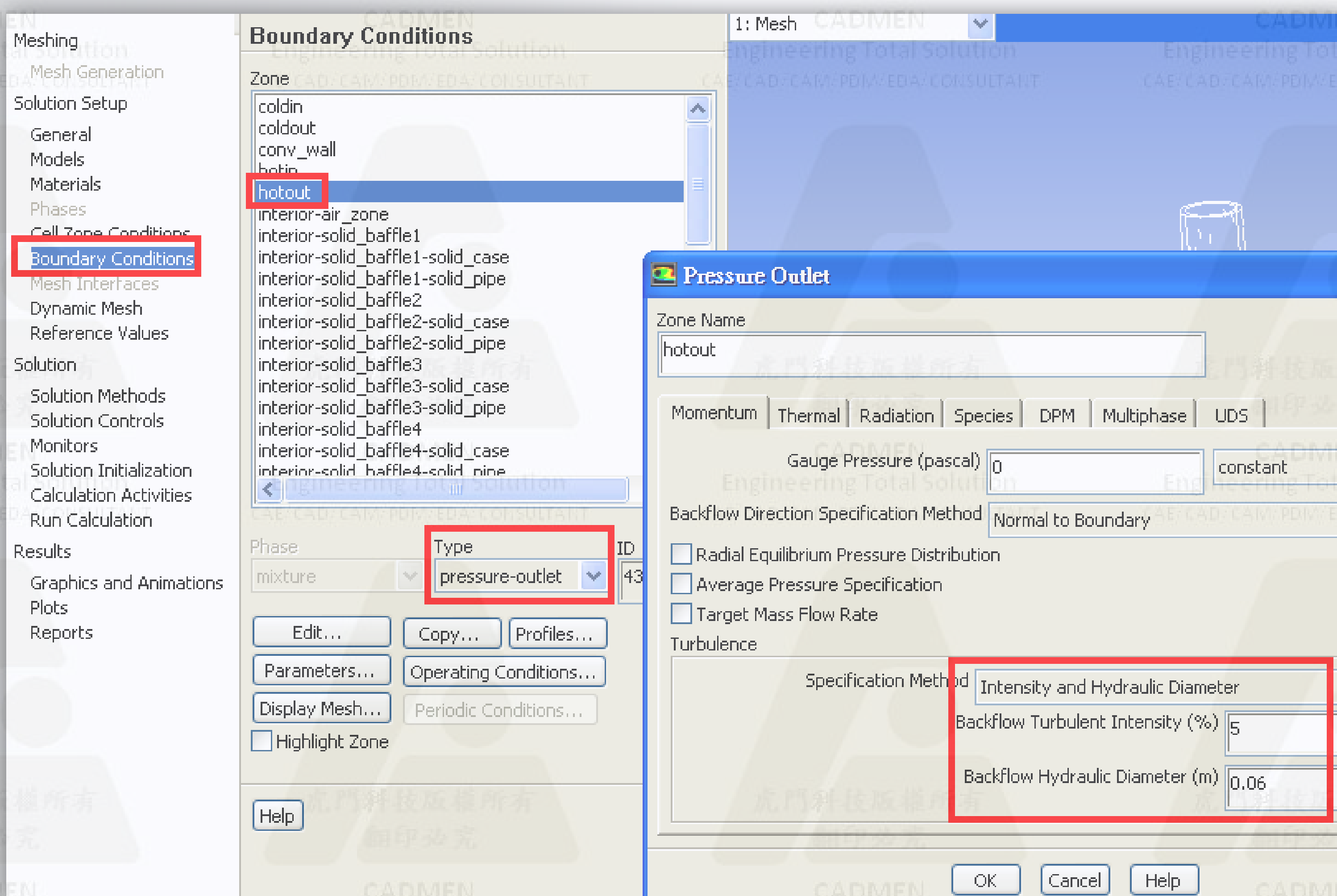
設定冷水入口條件



# FLUENT 設定

## 邊界設定

設定冷水出口條件



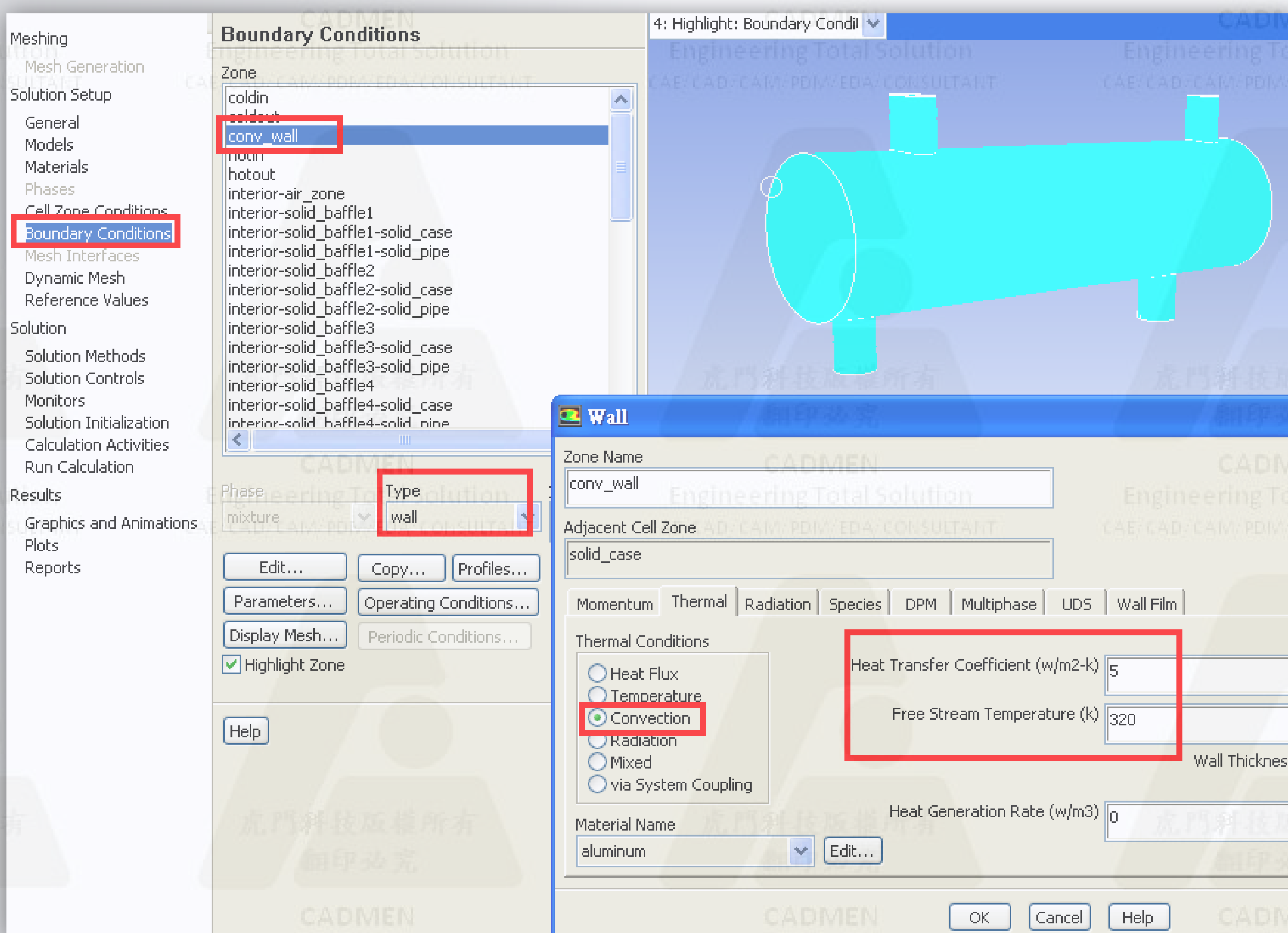


# FLUENT 設定

## 邊界設定

設定外殼壁面對外熱傳特性

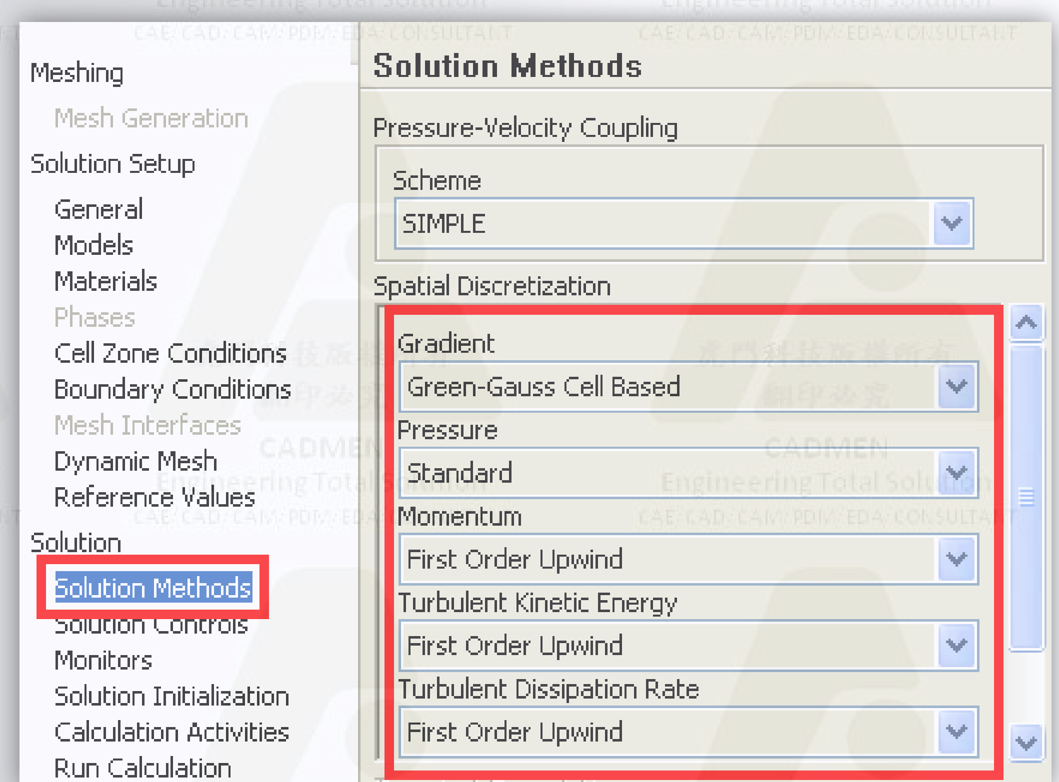


ANSYS

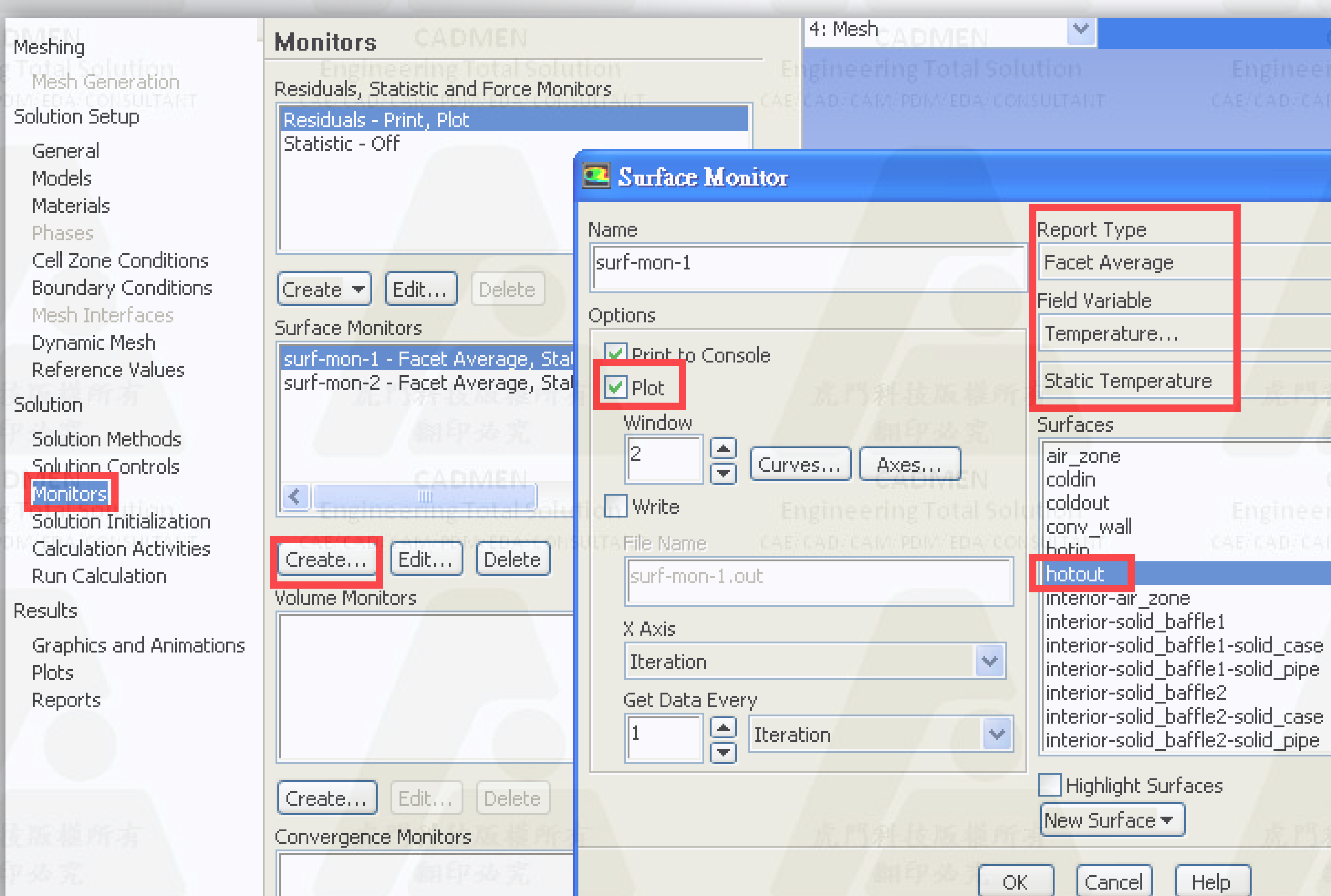
# FLUENT 設定

## 求解器設定



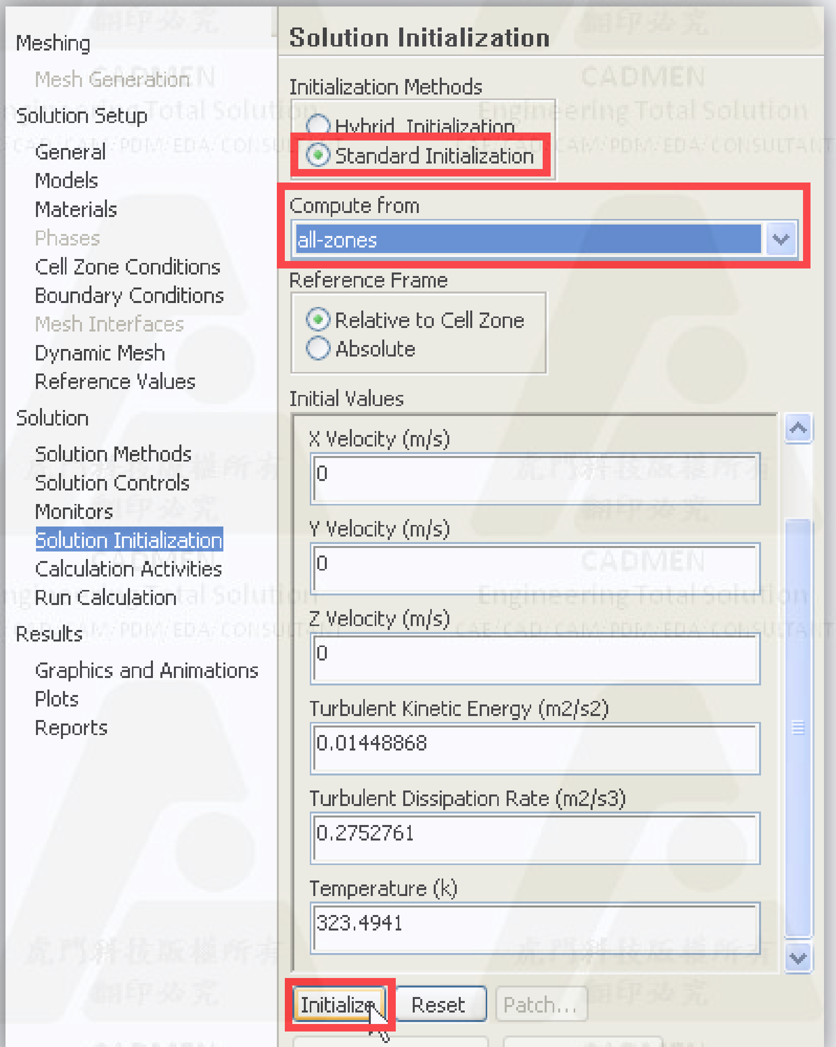
# FLUENT 設定

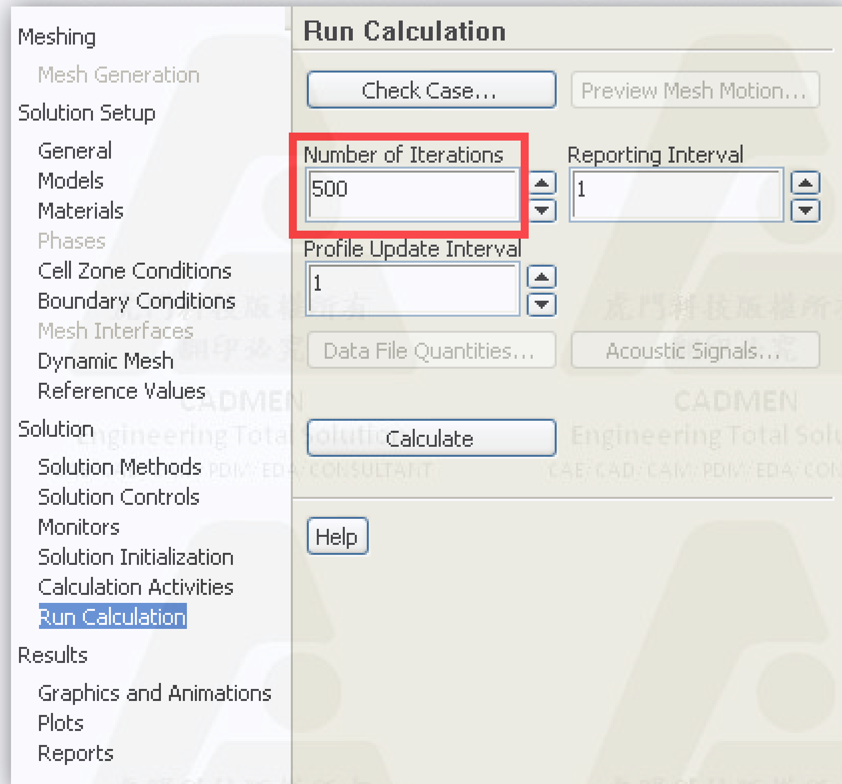
## 設定熱空氣出口平均溫度監控



# ANSYS FLUENT 設定

## 初始化與疊代求解設定



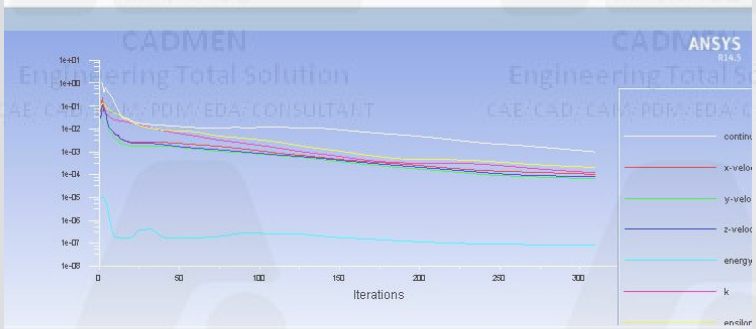



  July 11, 2013

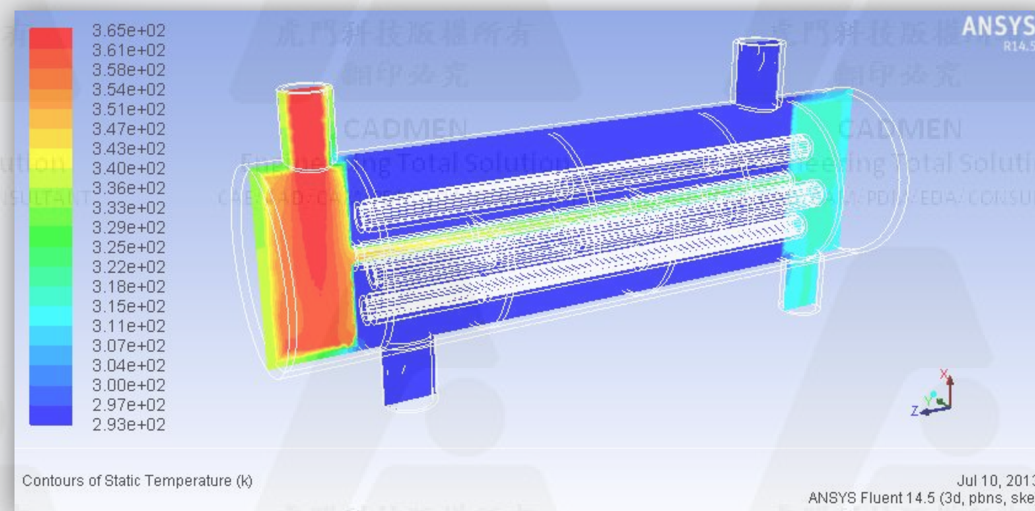
# FLUENT 設定

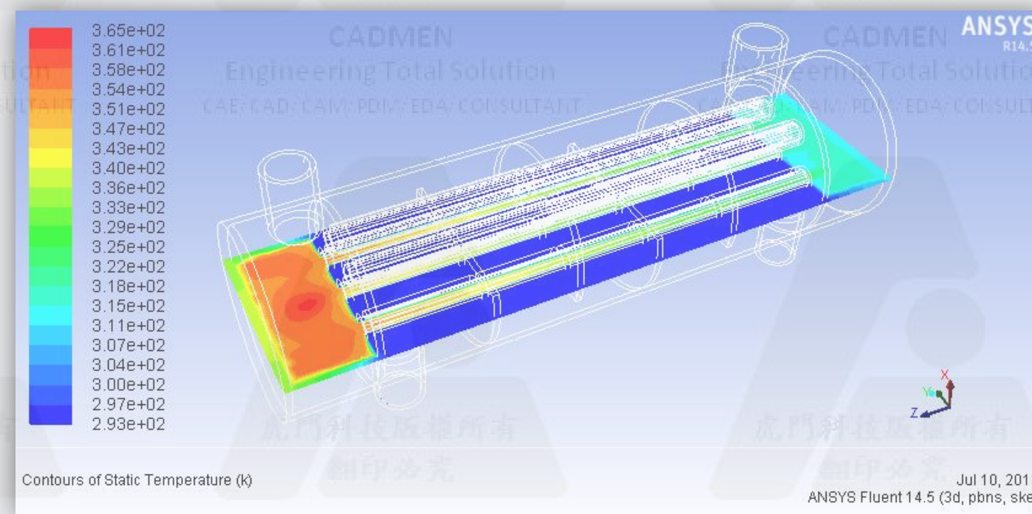
## 收斂性&後處理溫度分佈

### 出口均溫定量監控

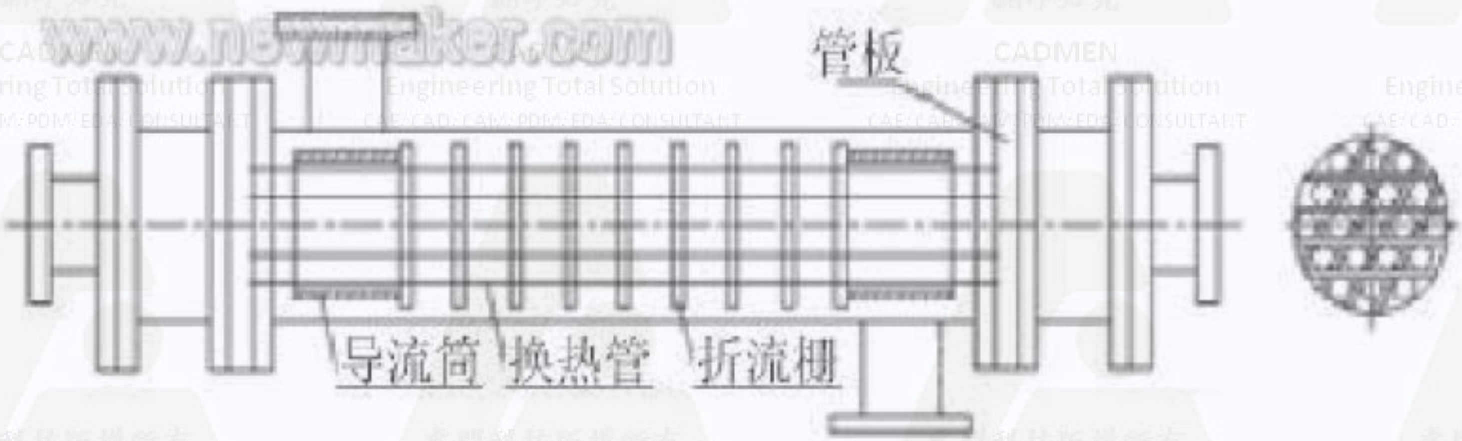




### 殘值收斂監控

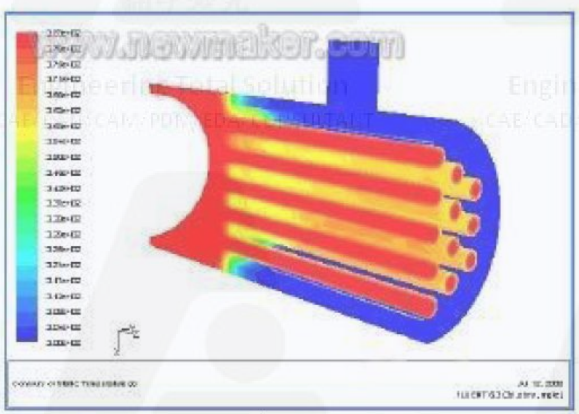


# 案例-管殼式熱交換器 CFD與結構耦合分析

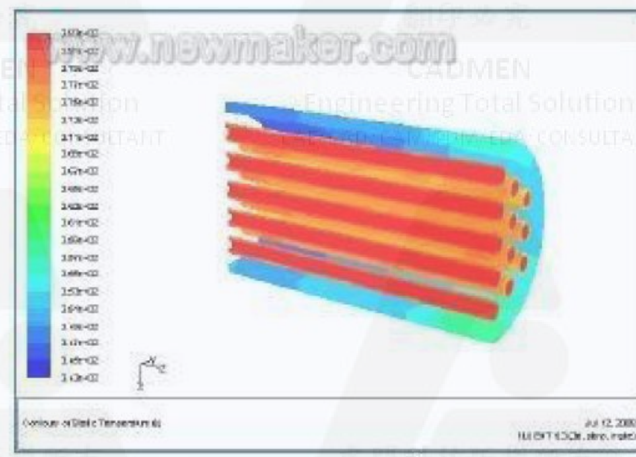



换热器结构示意图



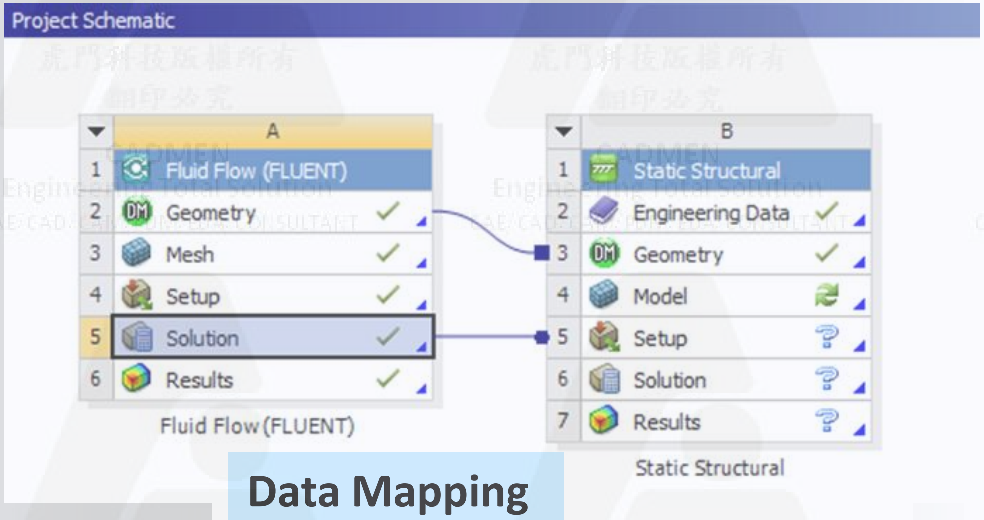
入口段 CFD 模拟温度场分布

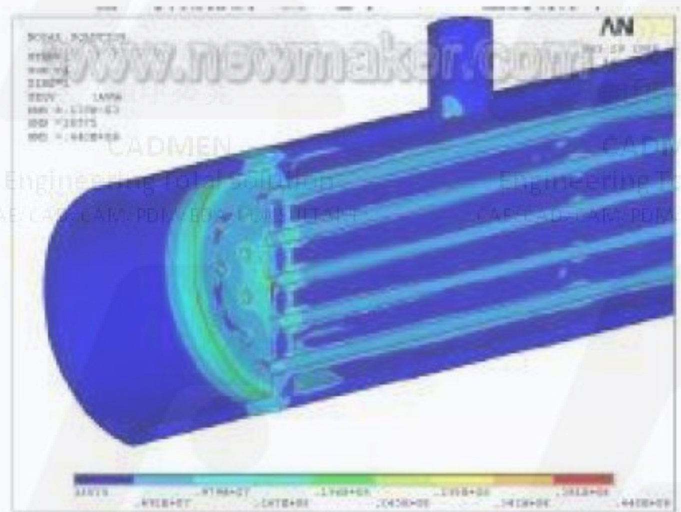
入口段管子及管板 CFD 模拟温度场分布



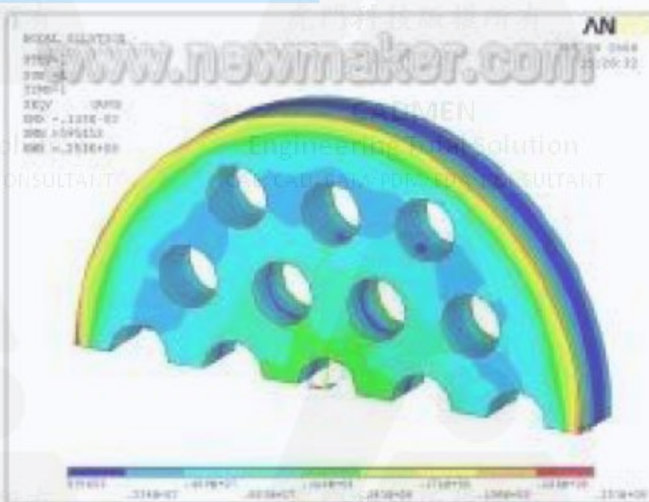
折流栅段（四）CFD 模拟温度场分布

# 案例-管殼式熱交換器 CFD與結構耦合分析



管束、壳体与管板的应力分布



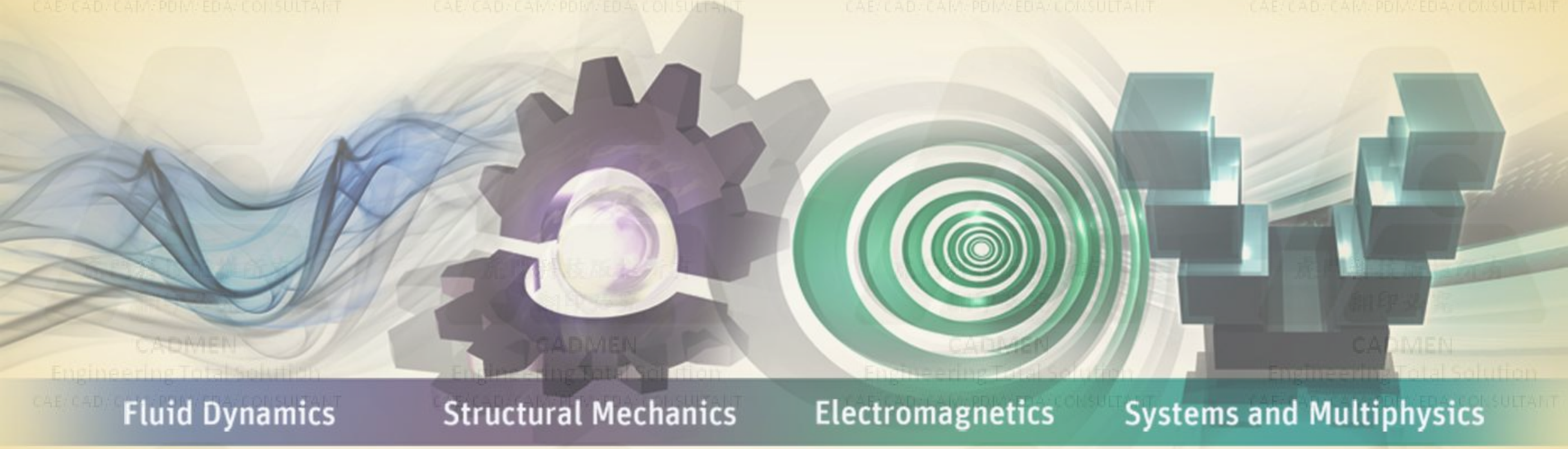
管板应力分布（管程侧）

壳体轴向（Z）应力分布

ANSYS

Realize Your Product Promise™

# ANSYS FLUENT
# 功能延伸討論與應用範圍




李龍育 Dragon
CFD技術副理
虎門科技

# 攪拌器應用介紹

虎門科技/李龍育 CFD技術副理

# 攪拌器分析之需求

ANSYS

攪拌設備歷史悠久，應用範圍廣泛。但是針對攪拌操作的研究卻遠遠不夠。攪拌操作所涉及的因素極爲複雜：攪拌的物料的物性千差萬別，攪拌的目的也不盡相同，攪拌設備形式多種多樣，再加上物料在攪拌設備內部流動極其複雜，如何合理正確的設計以及選擇攪拌器都沒有一個嚴密的理論指導，仍存在很大程度上依賴於經驗設計。

攪拌槽內的流動是三維和高度不穩定的湍流，脈動和隨機紊流給流速測定帶來了很大的困難。

實驗法：
激光多普勒測速儀（Laser Doppler Velocimetry，LDV）
粒子成像測速儀（Particle Image Velocimetry，PIV）
無論是LDV還是PIV技術，都需要花費大量的時間來進行測量。

- **實機實驗資訊取得不易**
- **實驗所能獲取的資訊有限**
- **實驗與研發成本高昂**
- **製程參數改變，優化時程緩慢**

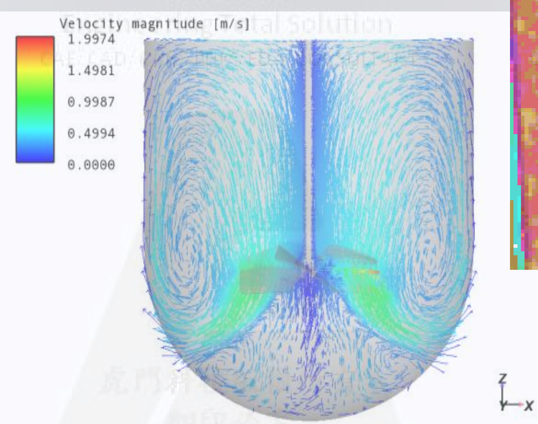
- 製程參數變更
- 幾何設計變更
- 材料改變
- 複雜流場現象掌握

# 攪拌器分析之需求

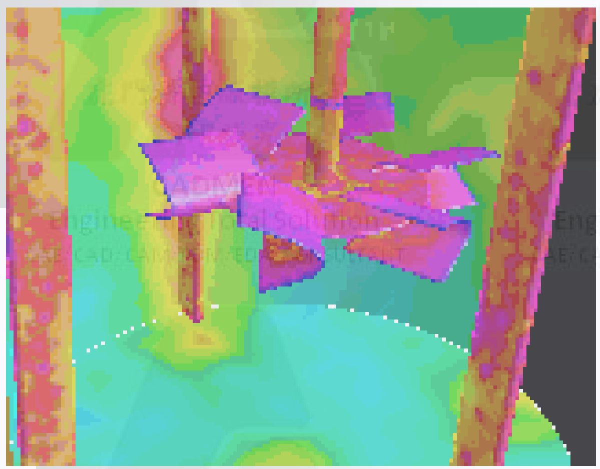
ANSYS

攪拌設備歷史悠久，應用範圍廣泛。但是針對攪拌操作的研究卻遠遠不夠。攪拌操作所涉及的因素極爲複雜：攪拌的物料的物性千差萬別，攪拌的目的也不盡相同，攪拌設備形式多種多樣，再加上物料在攪拌設備內部流動極其複雜，如何合理正確的設計以及選擇攪拌器都沒有一個嚴密的理論指導，仍存在很大程度上依賴於經驗設計。

攪拌槽內的流動是三維和高度不穩定的湍流，脈動和隨機紊流給流速測定帶來了很大的困難。




Velocity vectors in an unbaffled reactor



- 製程參數變更
- 幾何設計變更
- 材料改變
- 複雜流場現象掌握

# Mixing And Agitated Vessels

ANSYS

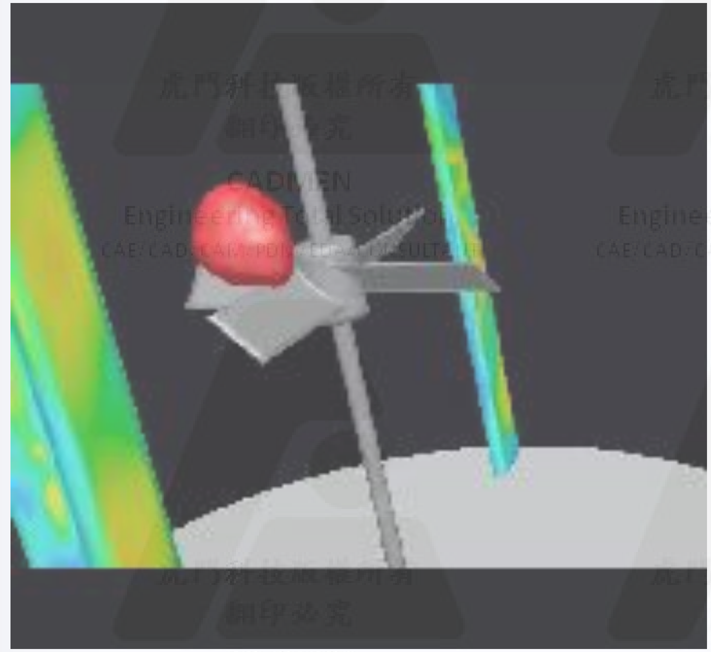
**Single phase**
- Velocity field prediction
- Turbulence prediction
- Turbulence

**Gas liquid flows**
- Bubble size distribution
- Mass transfer

**Liquid solid flows**
- Solid suspension

**Reacting flows**
- Product selectivity
- Heat transfer
- Mass transfer
- Crystallization

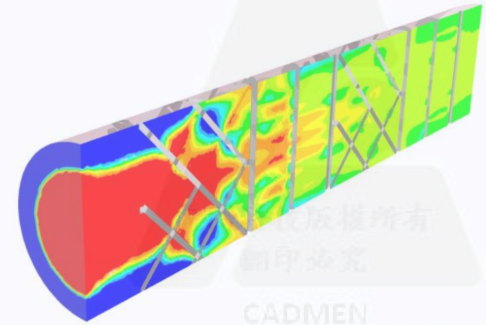
**ANSYS tools can model all above processes individually or in combination**



分析需求模組
- Multiphase : VOF , Mixture , Eulerian
- DPM
- Newtonian or Non-Newtonian
- Moving Substrate
- Unsteady State
- Dynamic Mesh
- Reaction

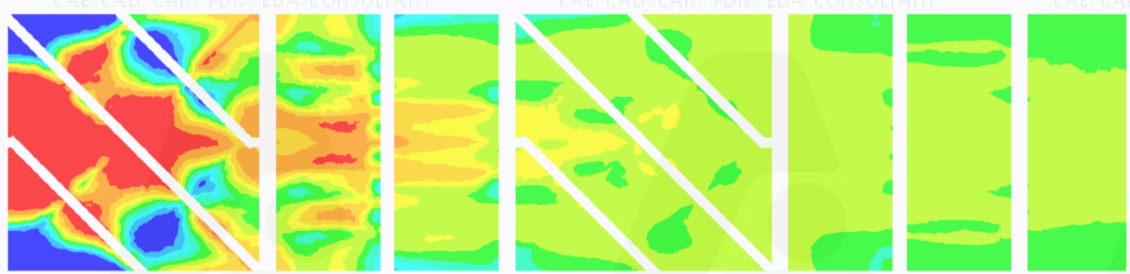
Increasing complexity

# Analysis Case Example Static Mixer



(a)

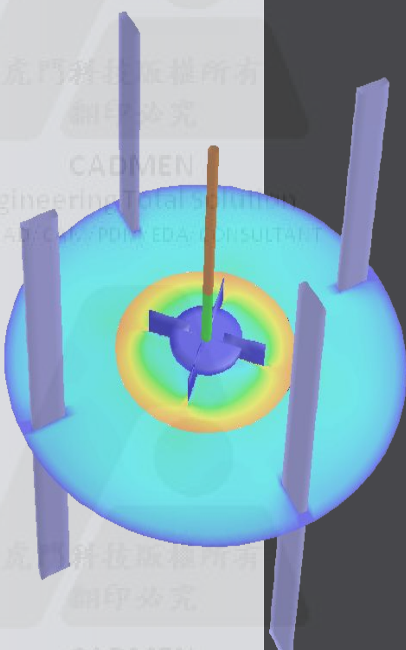
(b)

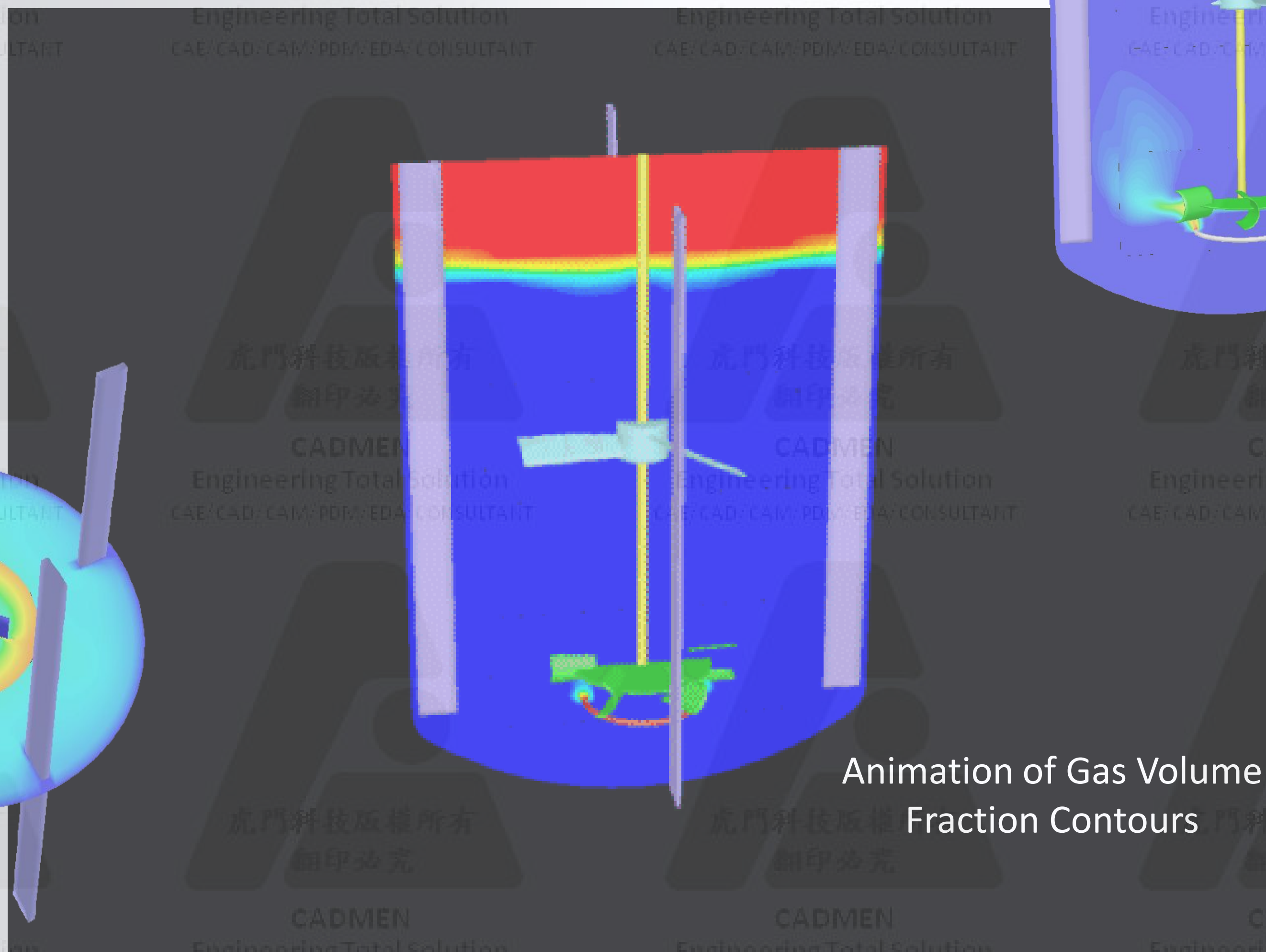


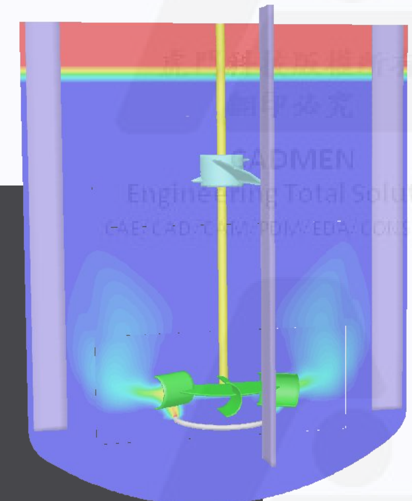
a) Experiment (Sultzer), b) Fluent results

# Agitation Analysis

## Gas Separating *Stirred tank*





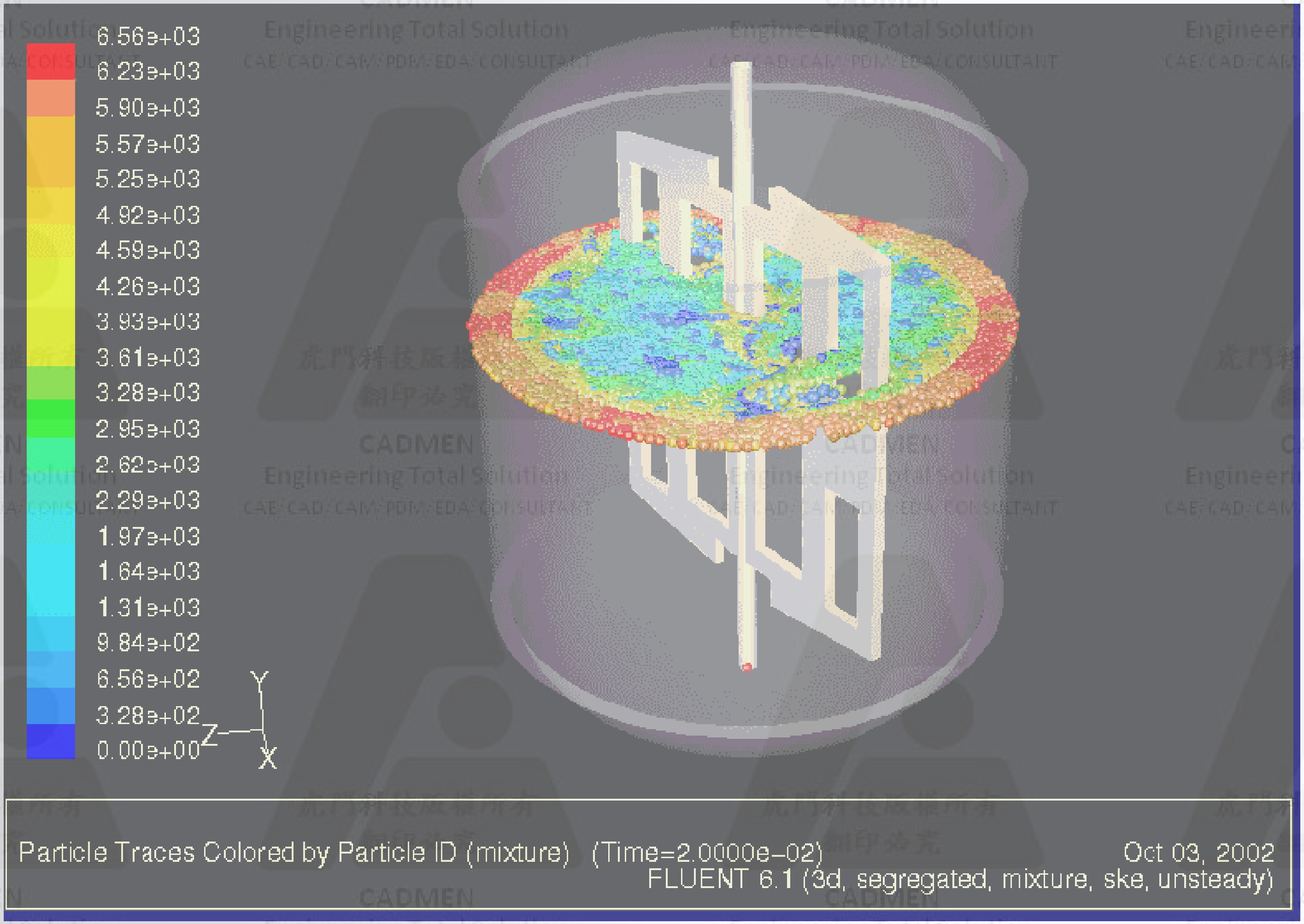


# Agitation Analysis

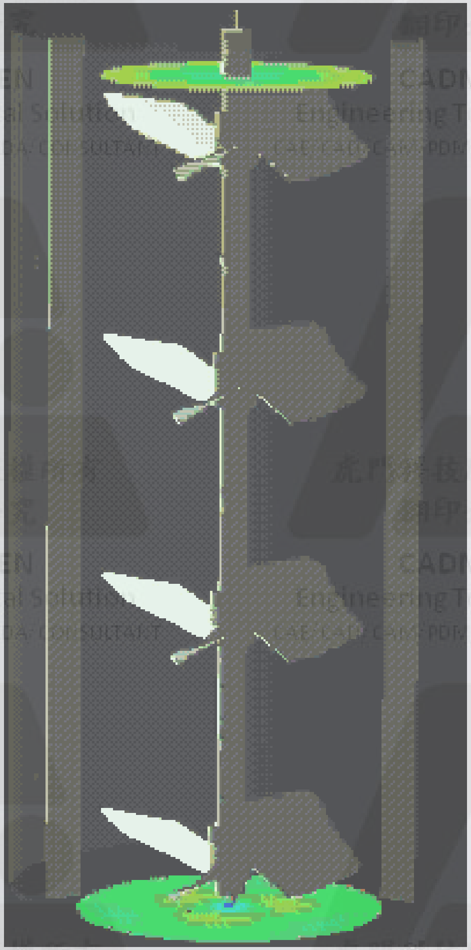
ANSYS

## Particle Traces Analysis

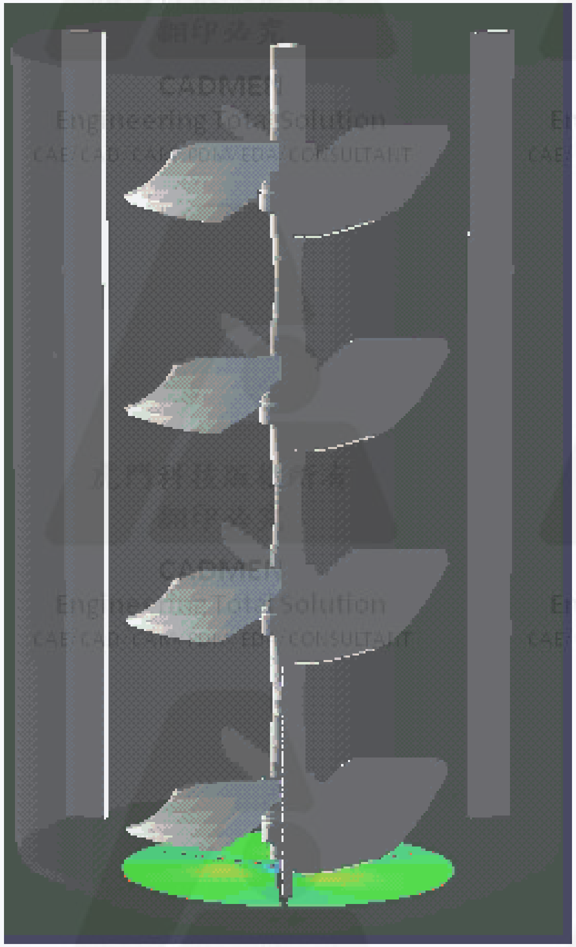


# Agitation Analysis
# Optimization of Impeller Position



Before



After

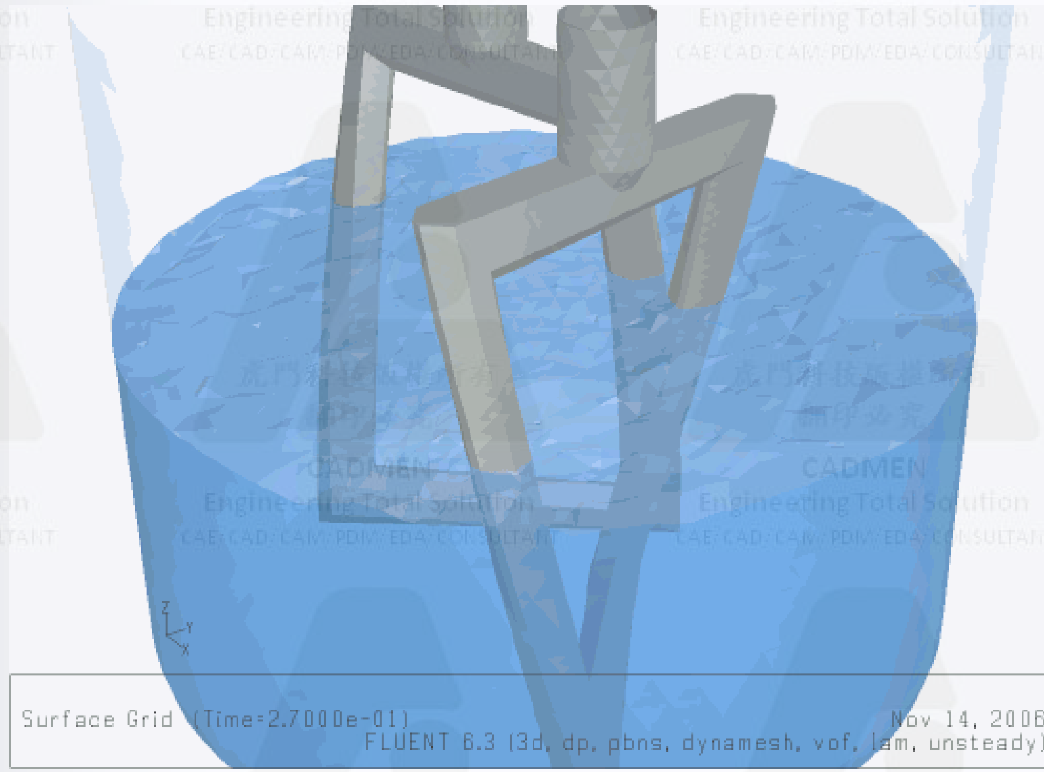
  

# Agitation Analysis

## High viscosity mixing

## 控制攪拌刀具 公轉與自轉

```
DEFINE_CG_MOTION(fast, dt, vel, omega, time, dtime)
{
        NV_S(vel, =, 0.0);
        NV_S(omega, =, 0.0);
 omega[0] = -12.77581011;
 beta1 = 2.0943951;
 vel[1] = -beta1*0.0665*cos((0.01745329*0)+beta1*time);
 vel[2] = beta1*0.0665*sin((0.01745329*0)+beta1*time);
 vel[0]= 0;
 }
```

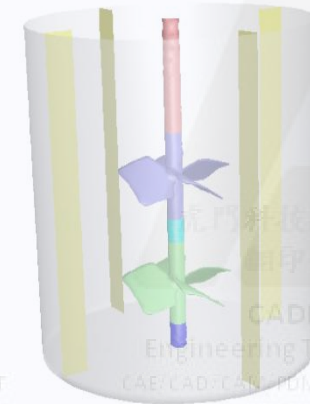



Two-axis Rotation Blade

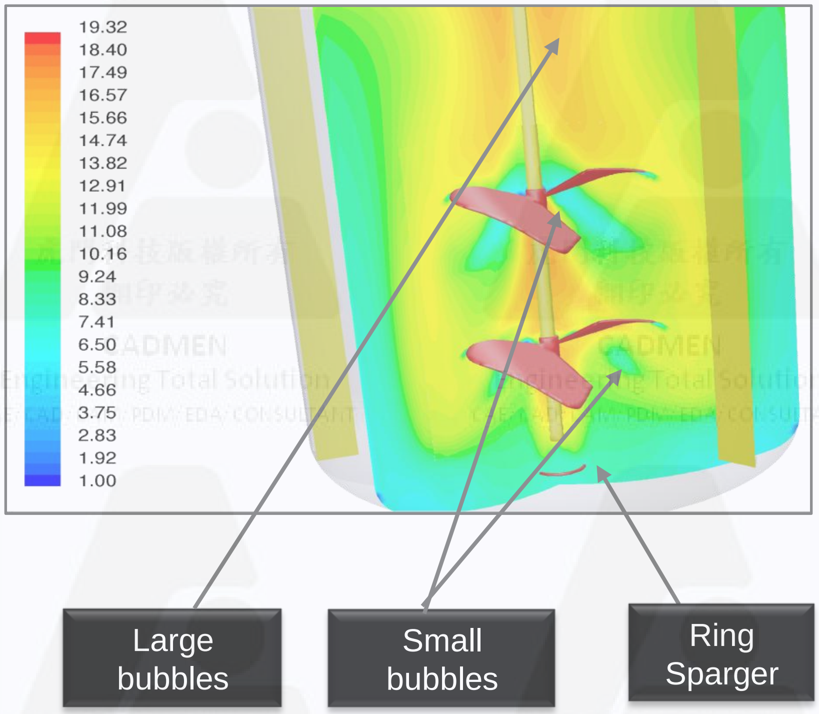
# Agitation Analysis

ANSYS

## Particle Size distribution



- Detailed information about the PSD at different operating conditions is crucial for design and scale up
  - Product quality
  - Downstream processing




- Chemical reactions and mass/heat transfer depend on the local particle size distribution (PSD)

Bubble Dia. in mm



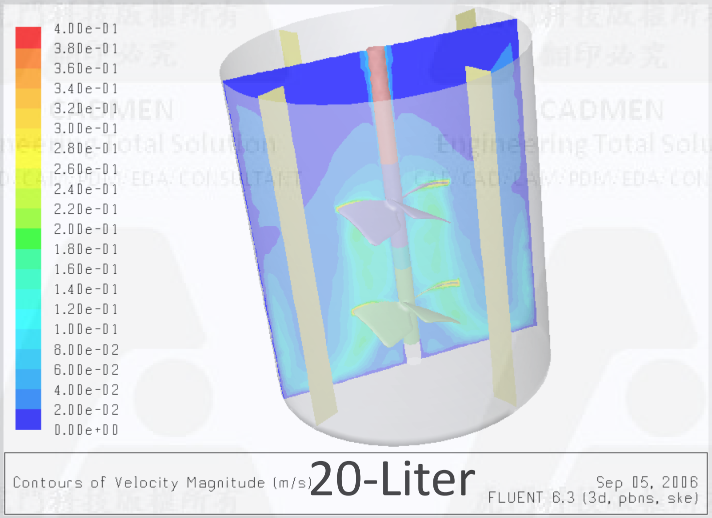
  

# *Agitation Analysis Scale-Up Example*

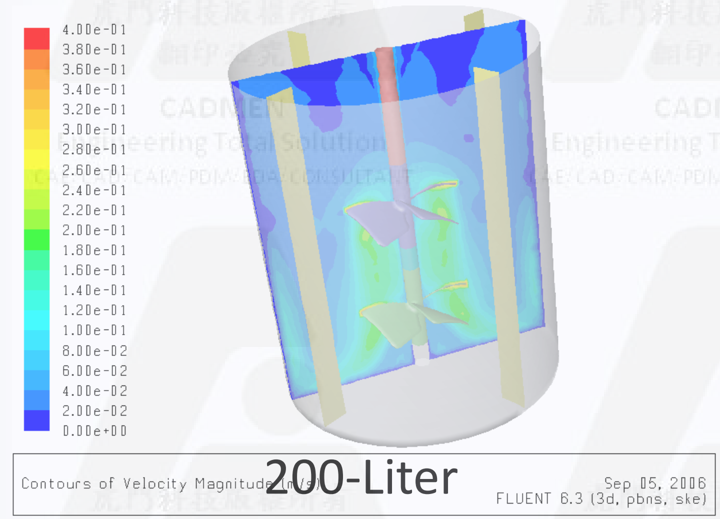
Agitation power per unit volume

## Result Velocity Magnitude

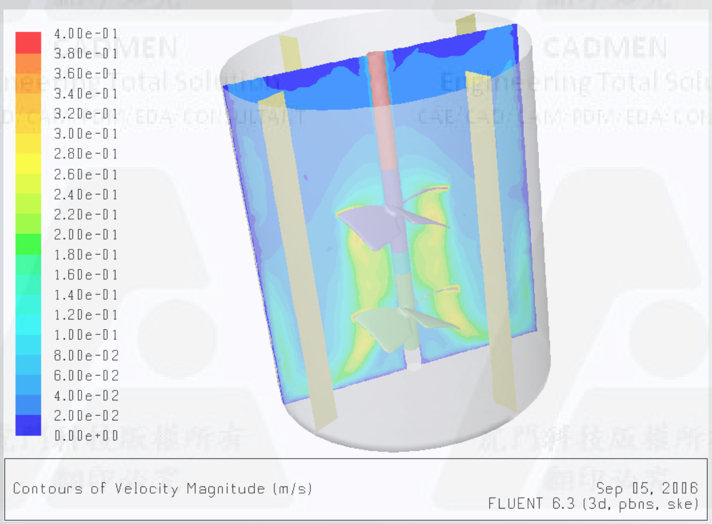
$Pv = Nd^{2/3} = const.$



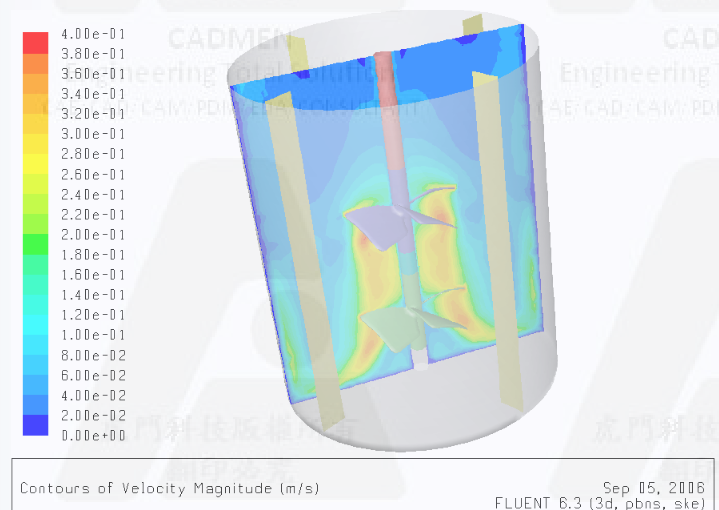

20-Liter




200-Liter




2000-Liter




20000-Liter

# Agitation Analysis Scale-Up Example

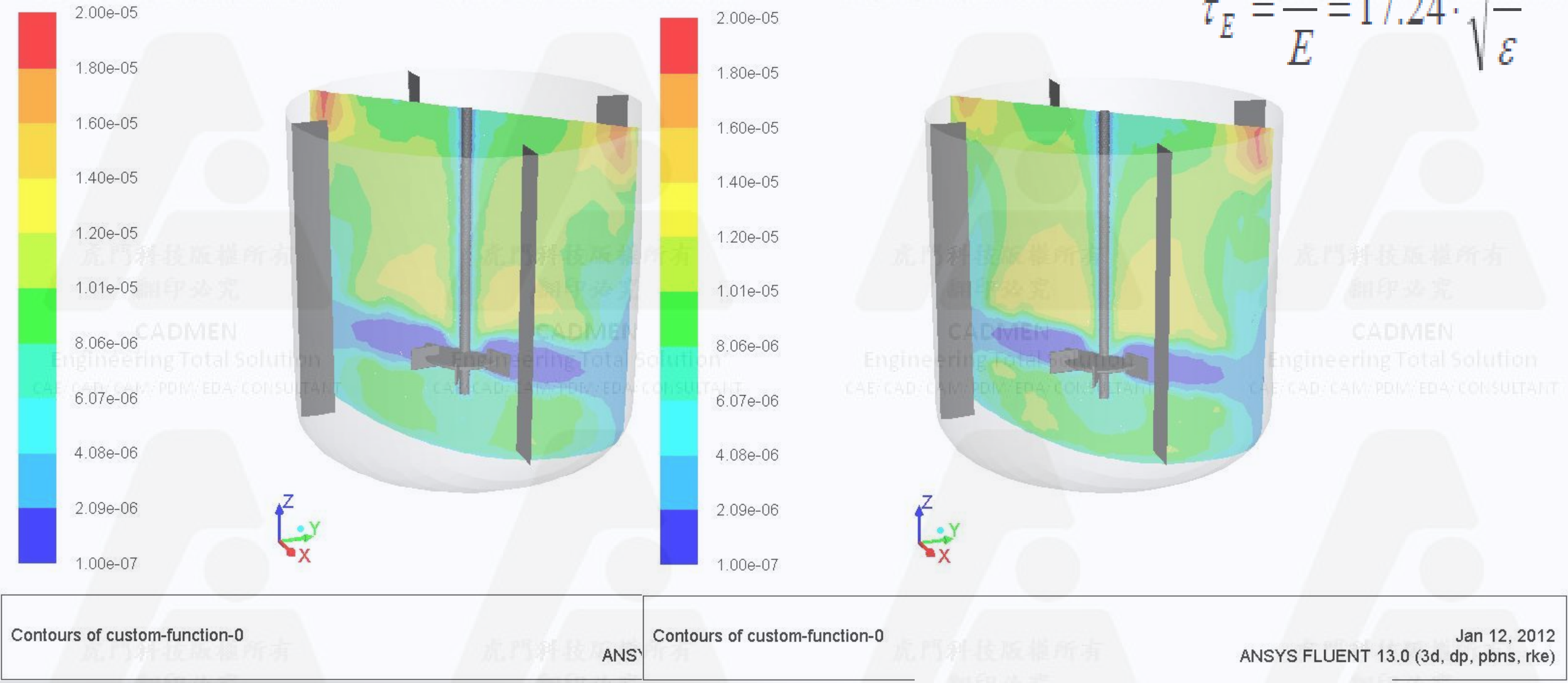
ANSYS

Result : Microscale Mixing Time

Almost same resut : Same reaction time

$$\tau_G = \tau_E \left( 0.0303 + \frac{17050}{Sc} \right)^{-1}$$

$$\tau_E = \frac{1}{E} = 17.24 \cdot \sqrt{\frac{\nu}{\varepsilon}}$$




3L-Tank

5000L-Tank

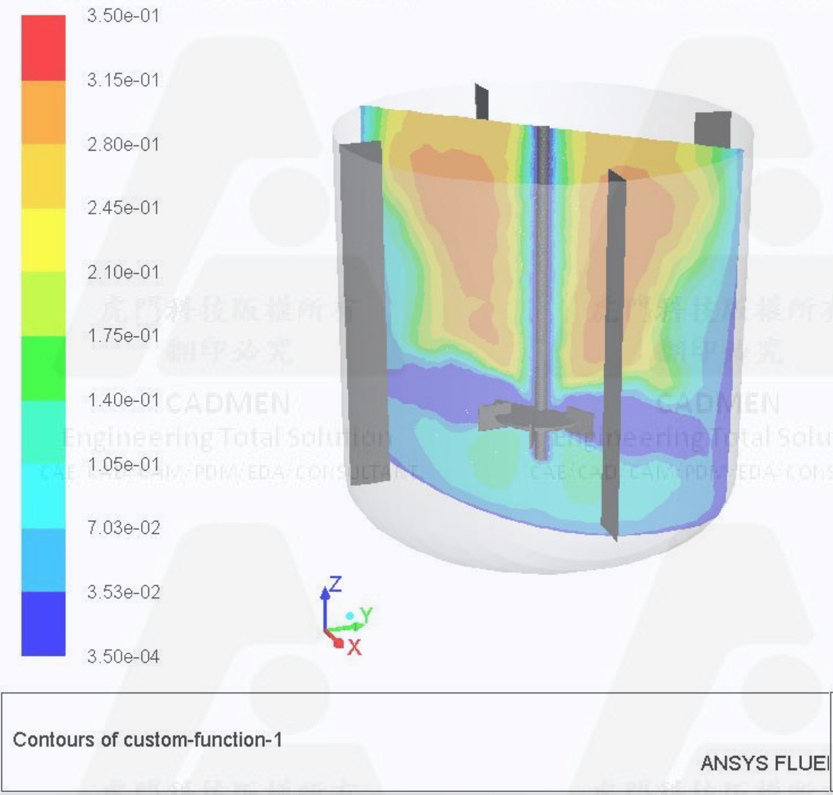
# *Agitation Analysis Scale-Up Example*

ANSYS

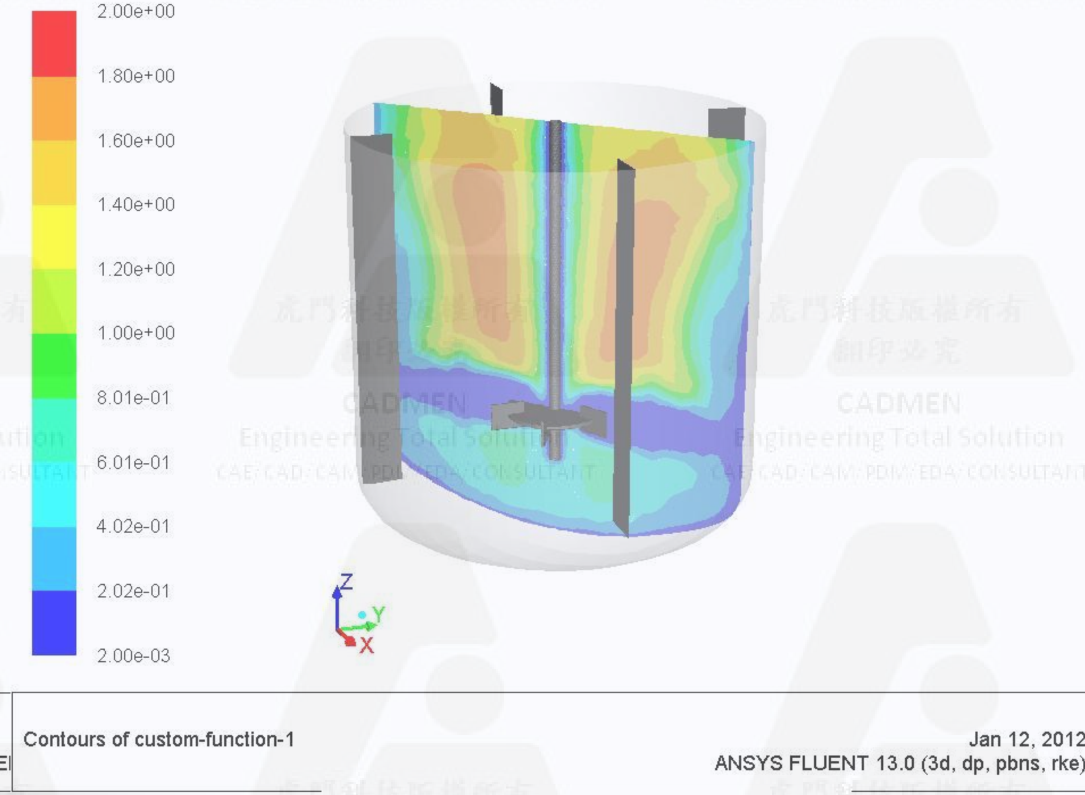
$$\tau_s = \frac{3L^{2/3}}{4\varepsilon^{1/3}} = \frac{k}{2\varepsilon}$$

## Result : Mesoscale Mixing Time

Different resuts : 5000L-tank needs more mixing time



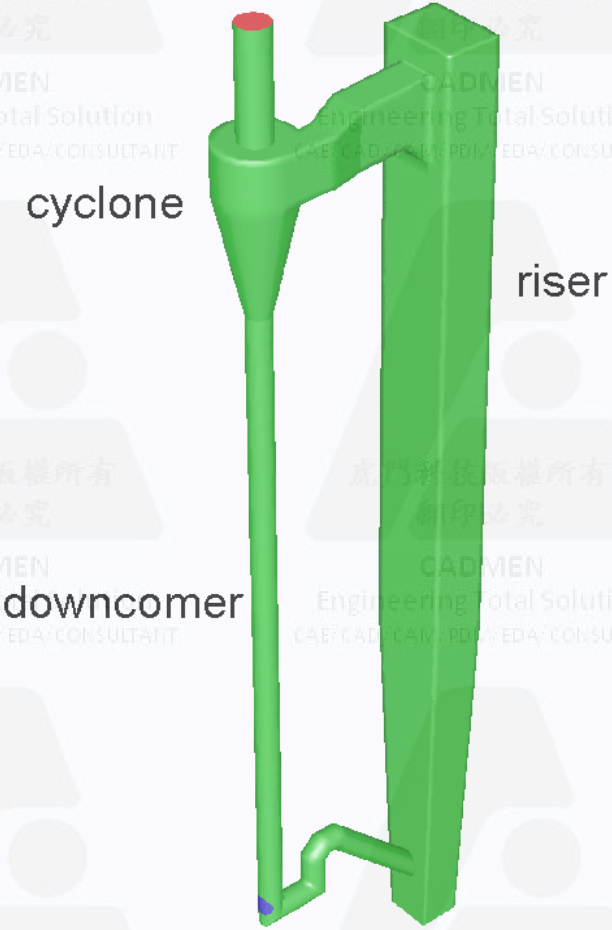

**3L-Tank**




**5000L-Tank**

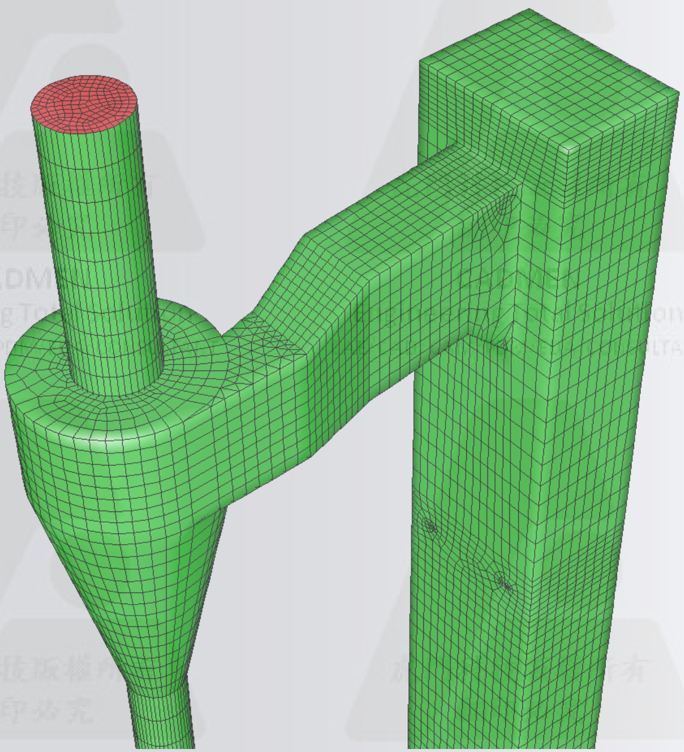
# Circulating Fluidized Bed

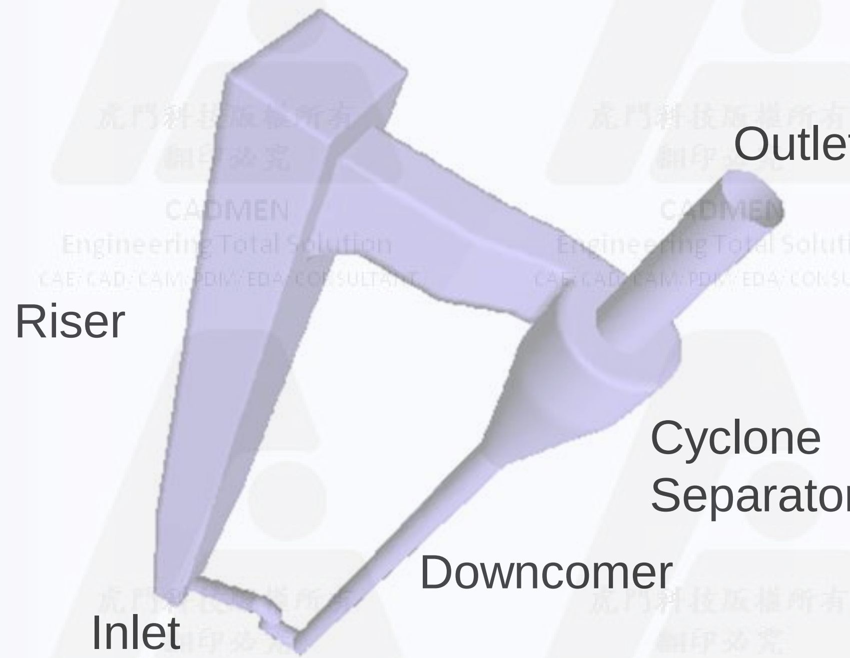
- **Circulating fluidized beds (CFBs) consist of a cyclone, downcomer, riser, numerous inlets, and a single outlet (red)**
- **They contain a circulating mixture of gas and solids**
  - **Different processes clean or burn the solids**
- **The Eulerian granular multiphase model in FLUENT is used to simulate the multiphase flow in a typical unit**



# Circulating Fluidized Bed

- **A fully 3D circulating fluidized bed is modeled.**
- **74,000 cell hybrid mesh**
- **Gas/Solids dilute flow (average solids volume fraction around 7%)**
- **Have a diameter of 85 µ, and density of 2200 kg/m$^3$**





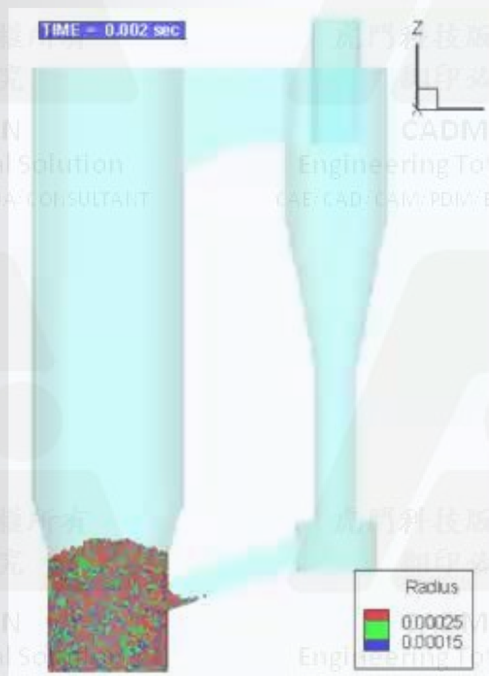

  

# Circulating Fluidized Bed

**Path lines (colored by air velocity magnitude) show the motion of air.**

- **Up the riser**
- **Through the connecting channel**
- **Spinning in the cyclone**
- **Exiting through the outlet**

**Some air falls through the downcomer and is recirculated with the solids**



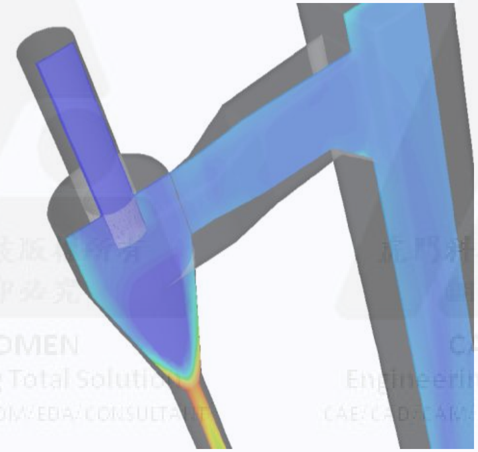

# Circulating Fluidized Bed

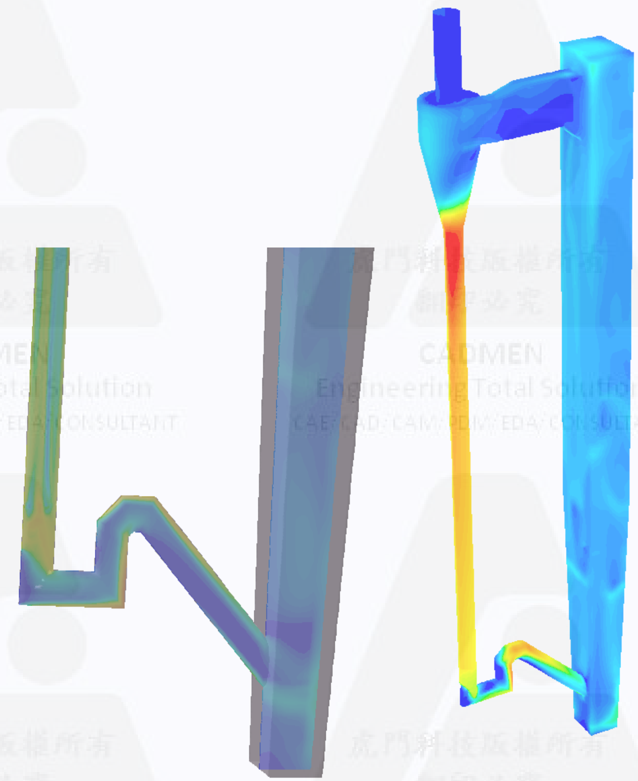
**Solids volume fraction on the walls of the unit show**

- Maximum concentration at the base of the cyclone, after separation
- High concentration throughout the downcomer
- Weak but uniform concentration in the riser due to upward-angled inlets positioned along the riser walls

**Flow field is consistent with expect- ations and reports in the literature**

**Results suggest the FLUENT Eulerian multiphase model is well suited for this application**





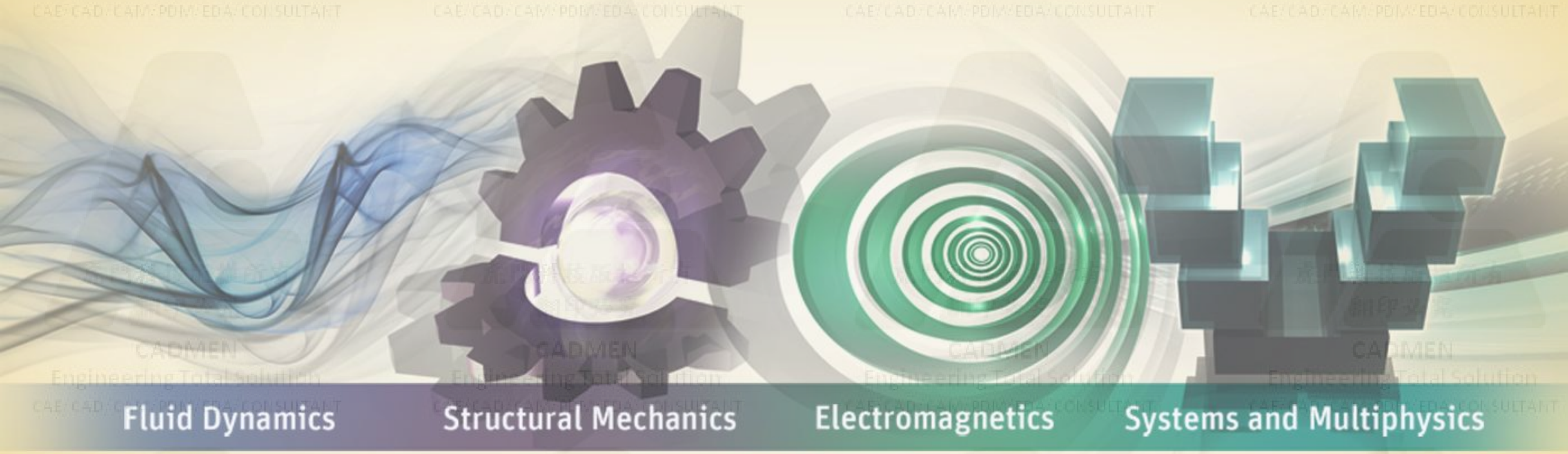
Contours of Solid volume fraction

*Courtesy of RWE Energie AG, Niederauben, Germany*

ANSYS

Realize Your Product Promise™

# Mixing Tank 攪拌槽CFD分析(操作)




## 雙段攪拌翼＆通氣環分析設定

**CFD技術團隊**

**虎門科技**

# 範例簡介

本範例是使用HE-3/CD-6兩種攪拌翼，爲常見的雙段型攪拌槽，經由下方通氣環來注入氣體(Bakker A, Smith J.M. and Myers K.J, 1994)。

此雙段型通氣攪拌槽爲一般常用於工業界的反應槽，包含生化反應、發酵現象及需要氣體參與反應的情況。

本ANSYS FLUENT教程中有：

尤拉的兩相設定。

multiple reference frame (MRF)&通氣環系統的設定教學。

如何設定和求解。

輸出計算數值及後處理。

# 攪拌槽機械構造

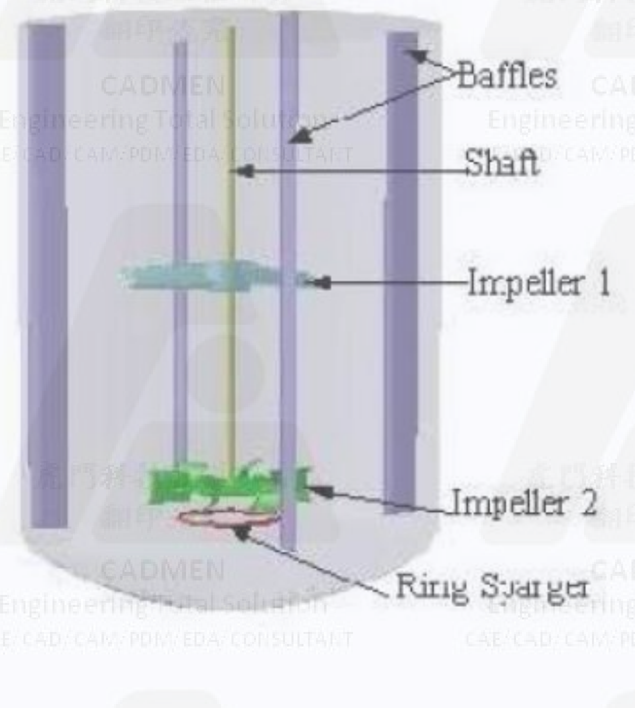


Figure 1: Schematic Figure

Figure.1 爲攪拌翼示意圖

攪拌槽直徑2m，高2.8m，底部圓弧深0.3394m

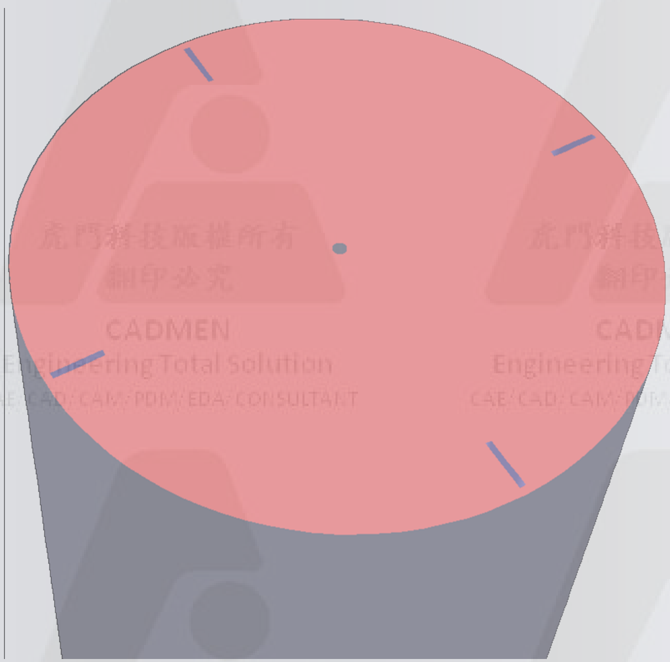
液面高2.4m.

四擋板，雙段攪拌翼(CD-6 and HE-3)，和通氣環。

# 攪拌槽機械構造

ANSYS

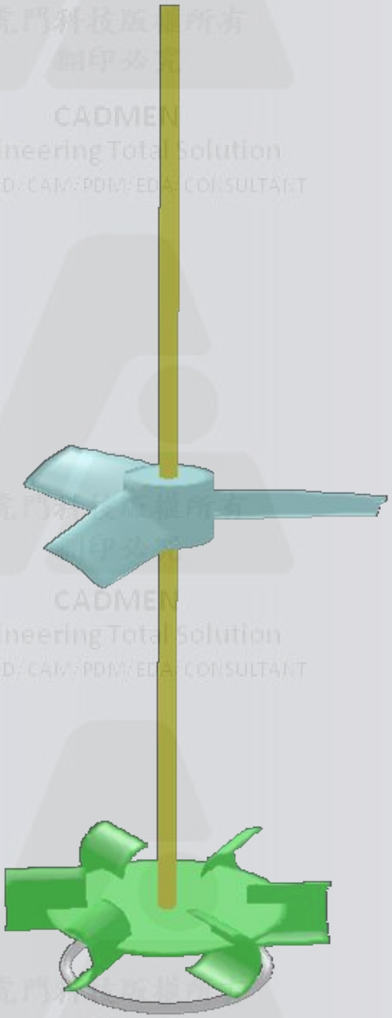


各項機械參數：

1. 轉軸:
   轉軸直徑= 0.045 m
   轉軸長= 2.2 m
   轉速=84rpm，順時針旋轉

2. 擋板:
   擋板數= 4
   擋板寬= 0.1667 m
   與槽壁距離= 0.0278 m
   與底部距離= 0.0833 m

# 攪拌槽機械構造




3.攪拌翼HE-3:
攪拌葉片數= 3
直徑= 1.04 m
葉片寬= 0.1664 m
葉片傾斜角度= 30 degrees

4.攪拌翼CD-6：
攪拌葉片數= 6
葉片直徑= 0.8 m
圓盤直徑= 0.6 m
葉片高度= 0.16 m
葉片長度= 0.2 m ，葉片以180°向前彎曲
攪拌翼距底部= 0.6 m
圓盤厚度= 0.005 m

5.通氣環:
通氣環直徑= 0.56 m
距離底部= 0.44 m
通氣速度= 0.1 $m^3/s$

# 教程簡介

本攪拌槽設計爲增加氣體與液體接觸情況。

攪拌槽中HE-3及CD-6分別爲軸向型及徑向型攪拌翼與擋板及攪拌槽形成一個複雜的流場，使流場主要在攪拌翼及擋板之間流動。

當氣體經由通氣環注入時，氣泡會被液體包覆並隨著流場流動，但由於浮力的關係，氣泡會由上方逸散，因此整個攪拌槽在穩態下的氣含率(gas holdup)，就是我們這次所要觀察的重點。

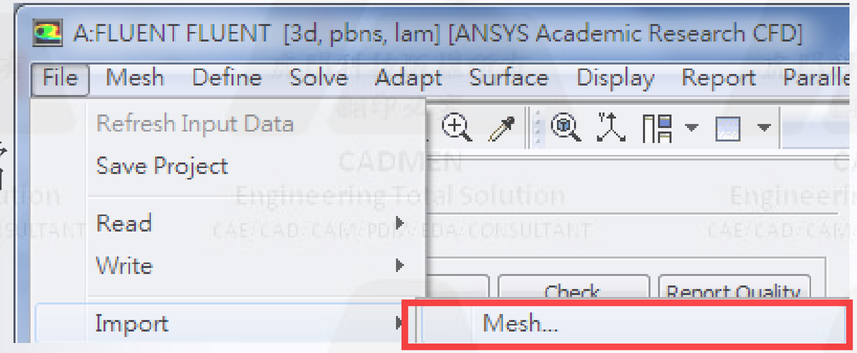
本次模擬教學主要爲設定一個CFD的氣液兩相流模式，以及觀察它的混合效益及氣含率，和攪拌翼在旋轉流體部份的MRF的設定。

# 前處理

Step.1：讀取檔案

讀取(sparger.msh.gz)的網格檔

File→Import→Mesh

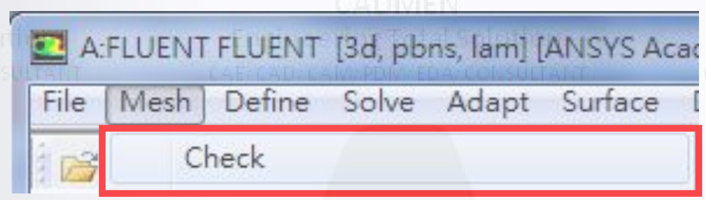



Step.2：一般設定

1.保留系統設定

2.檢查網格

Mesh→Check




ANSYS FLUENT將在網格上自動進行各種檢查，以及不連續的體積

# 前處理

3.顯示網格結構

Display→Mesh


可選擇你要顯示的攪拌槽各個面、邊界條件、網格

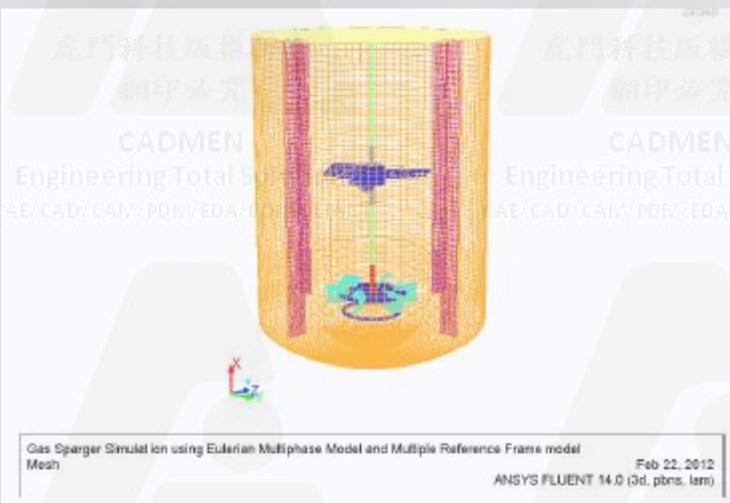


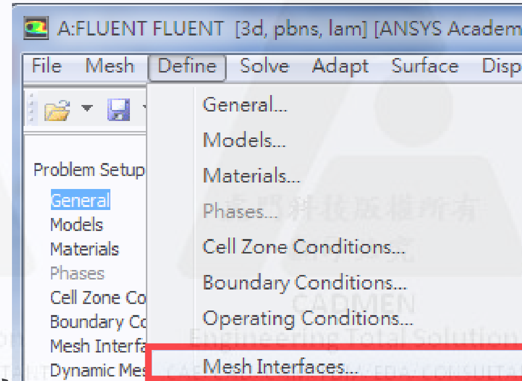

Figure 2: Mesh Display

ANSYS

# 前處理

4.定義內部連接面(Interface)

Define→Mesh Interfaces

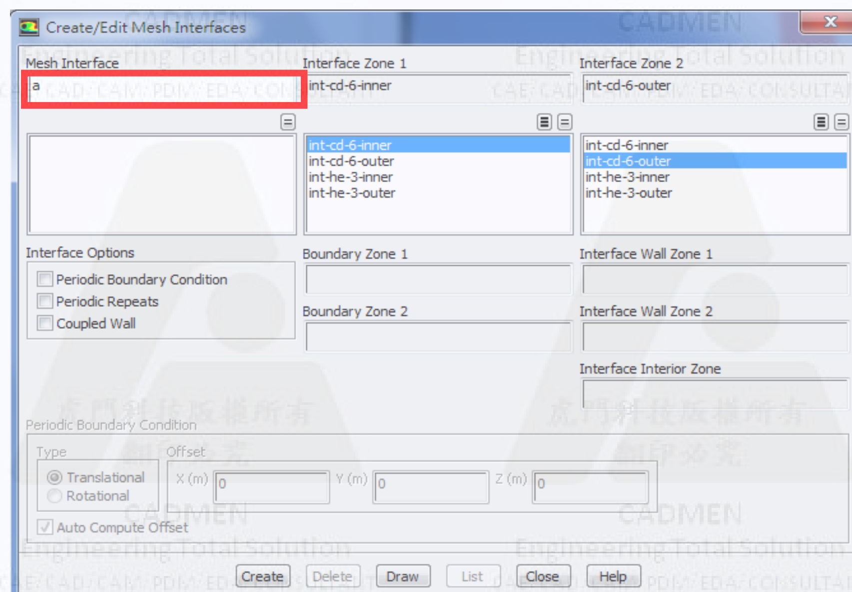



我們在建模時是設定，有一旋轉流體在攪拌槽內旋轉，在繪圖時它是與攪拌槽不連接的區域，而在Fluent中要設定連接。

分別選擇int-cd-6-inner及int-cd-6-outer，在Mesh Interface中設定a。

接著選擇int-he-3-inner及int-he-3-outer，在Mesh Interface中設定b。

點選Create完成。

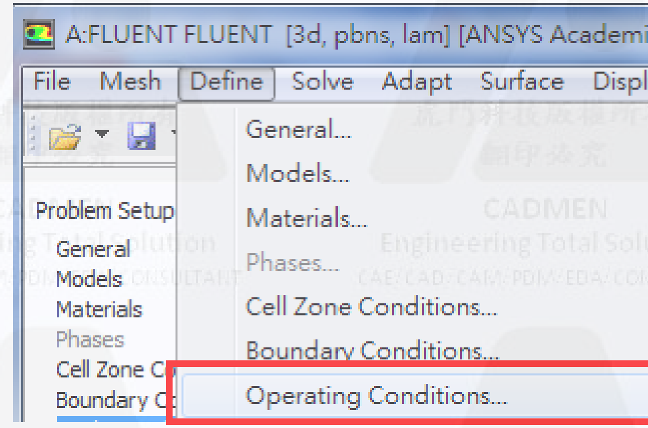


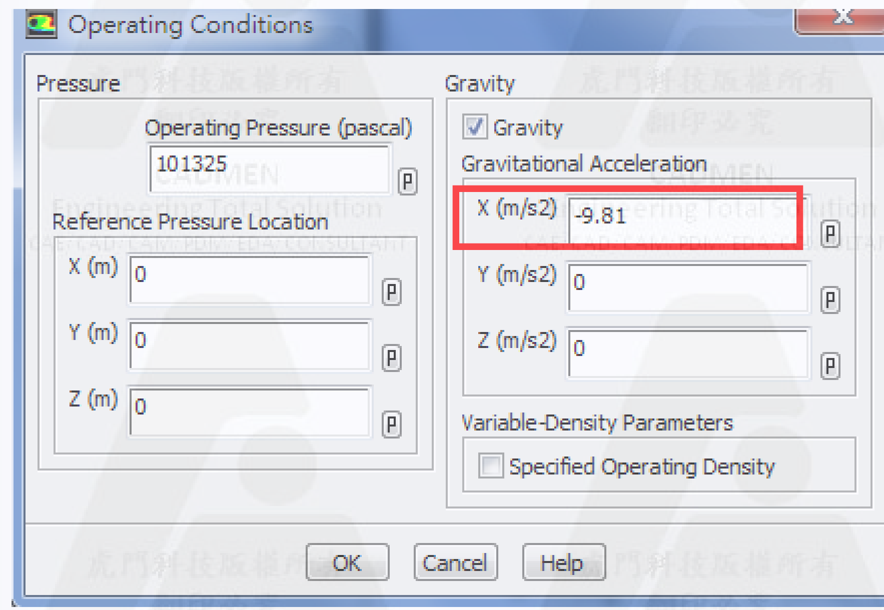
# 前處理

5.設定重力向

Define→Operating Conditions

點選Gravity→在X向中設定-9.81爲其重力向



# 模式設定

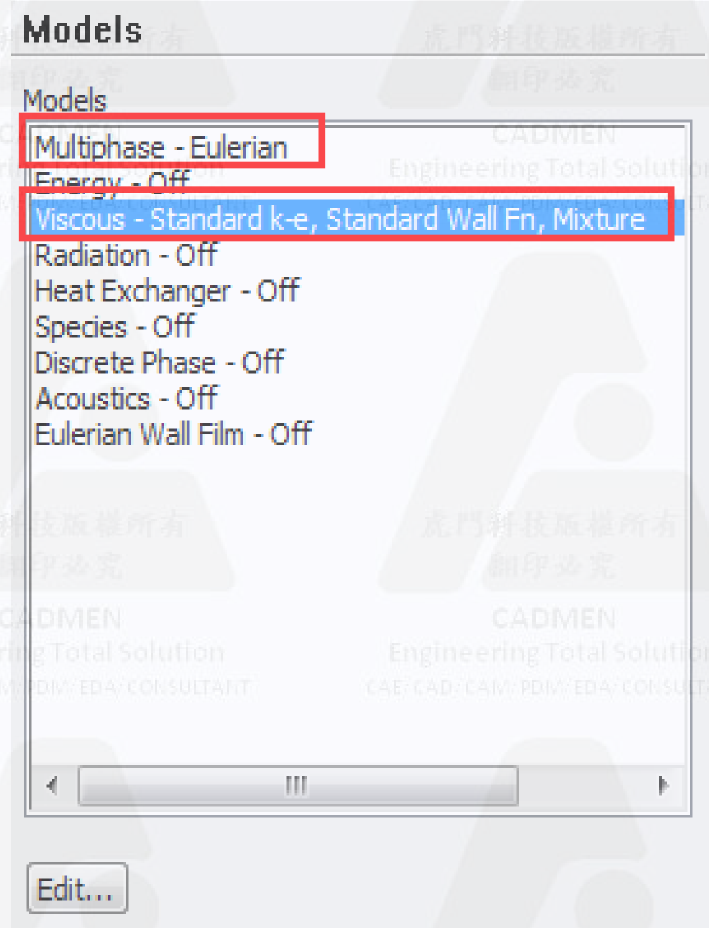
Step.3：

1.設定Eulerian 2相模組

Define→Models→Multiphase→Eulerian

2.黏度模型選擇『k-ε』

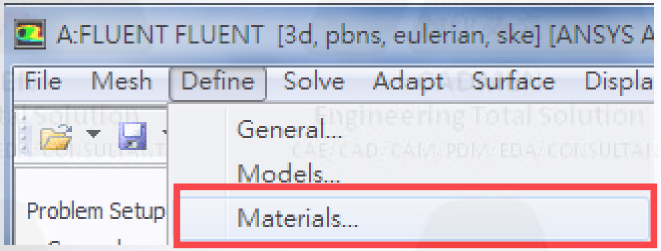
Define→Models→Viscous→k-ε



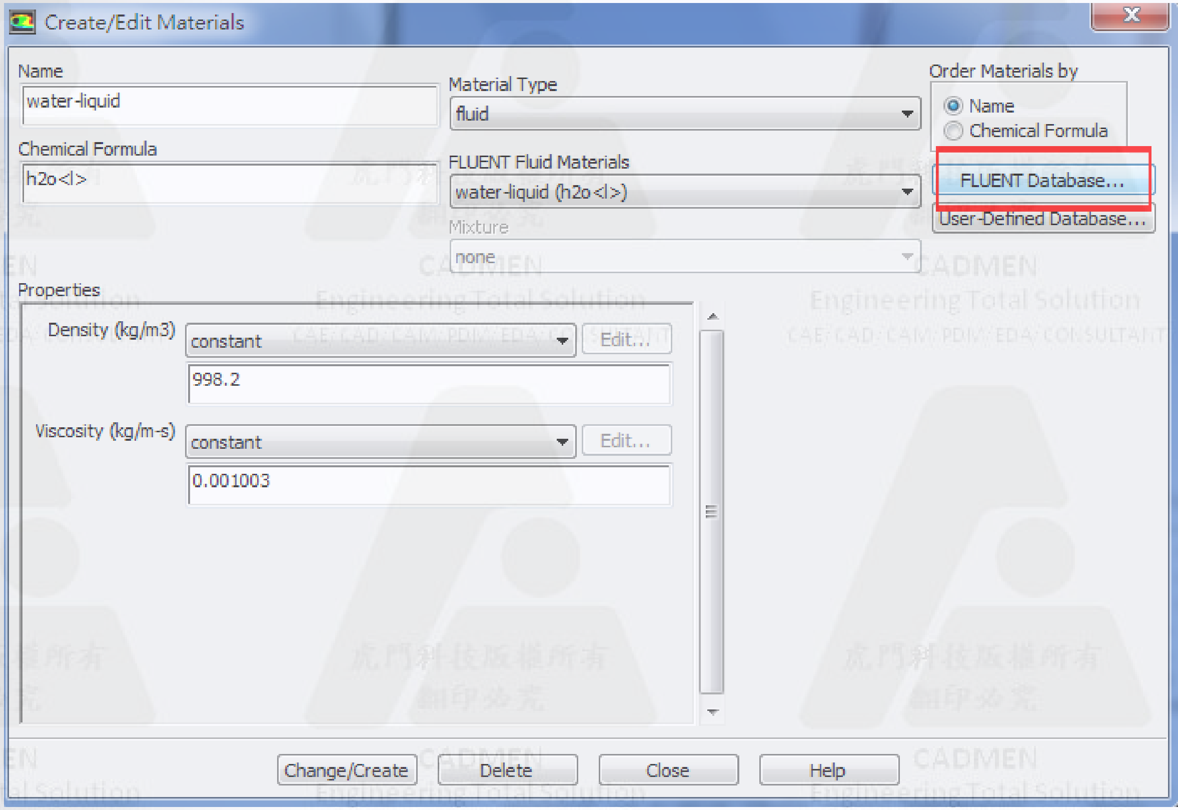
# 選擇流體兩相

Step 4:流體設定

Define→Materials→Fluid→
Fluid Datebase→Water




本模擬爲兩相模擬，包含氣體液體雙相，Fluent內含空氣，請將選單下拉，倒數第三項爲Water-liquid



# 選擇流體兩相

Step.5:流體相設定

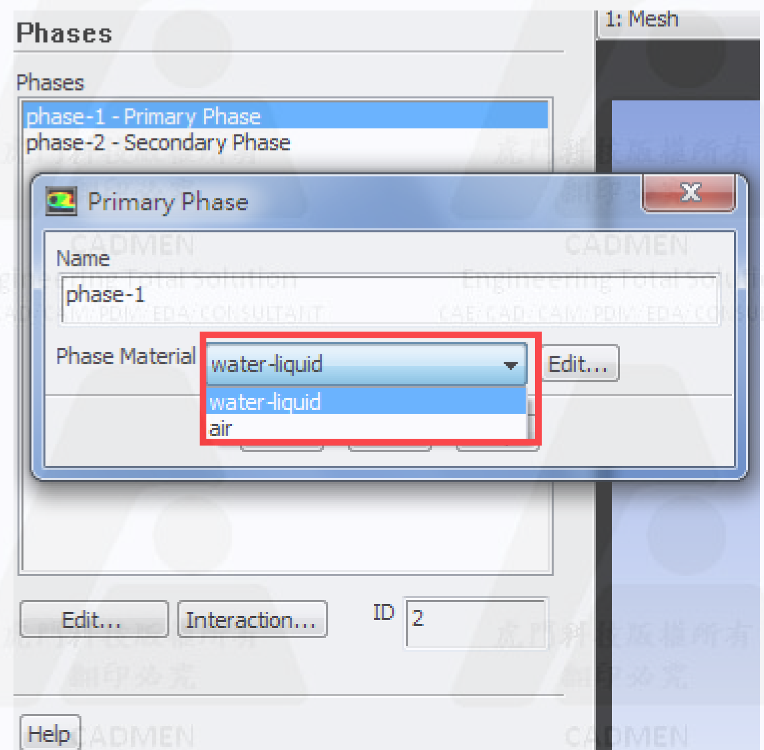
設定主要流體(Primary Phase)
Define→Phase→Primary Phase→Water liquid

將Name改成Water

設定次要流體(Secondary Phase)
Define→Phase→Secondary Phase→Air

將Name改成Air，Diameter改成0.002

結束後點選ok



# 旋轉流體設定

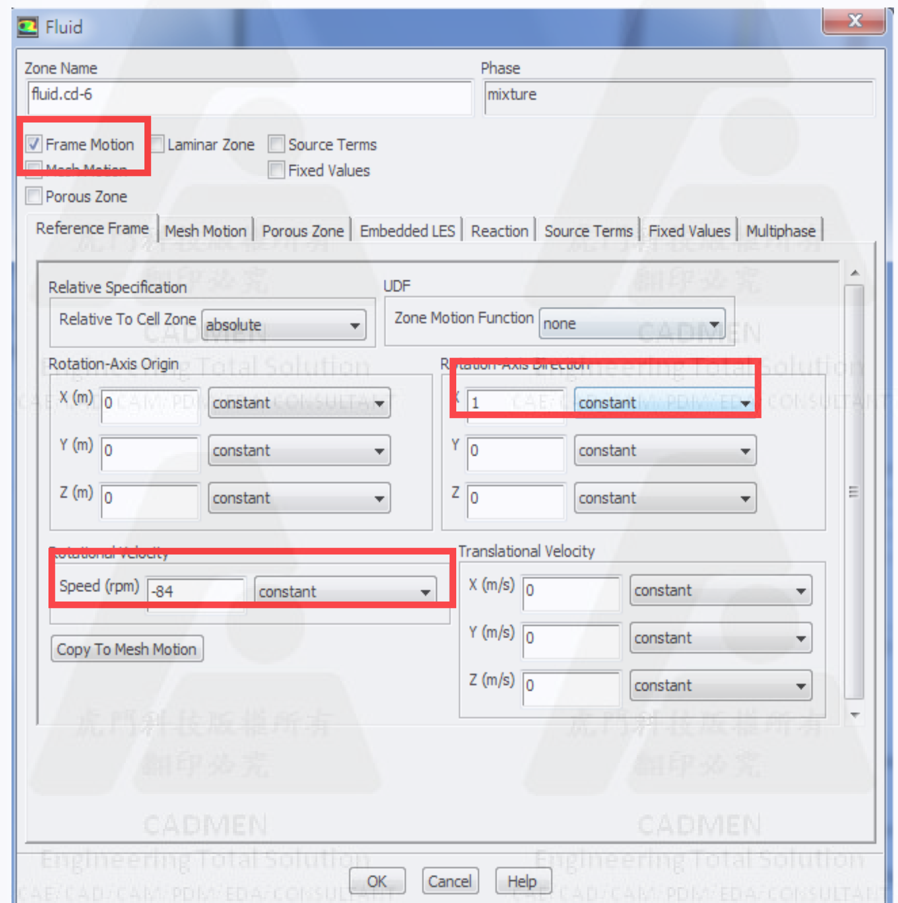
Step.5:選擇區域條件

Define→Cell Zone Conditions→Fluid.cd-6

選擇Frame Motion
X.Y.Z(1.0.0)
轉速84rpm

Fluid.he-3也依照上方設定

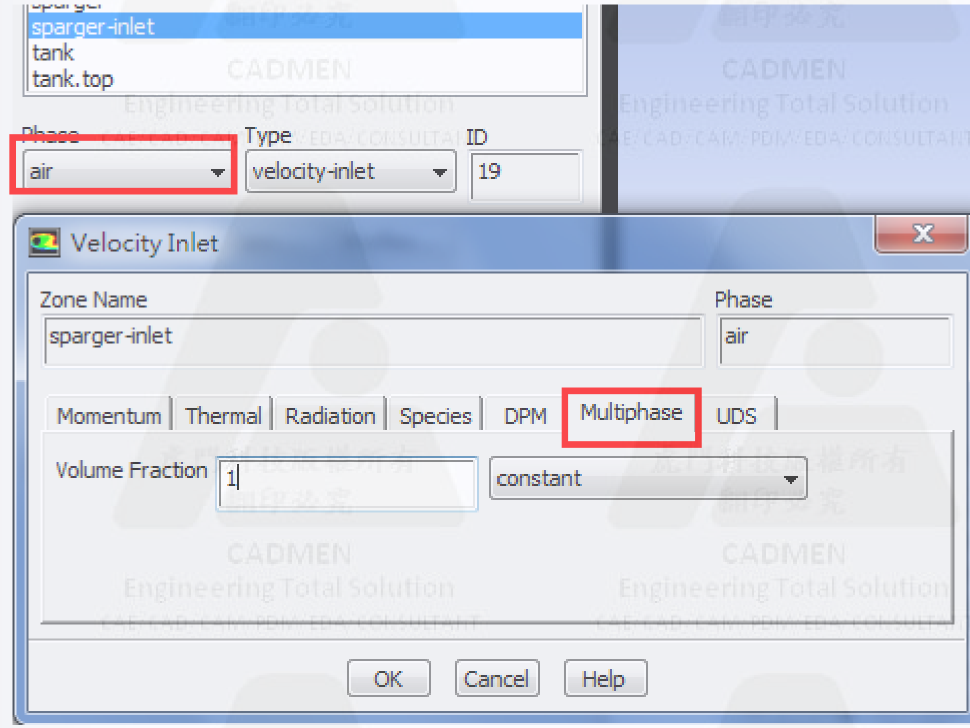
Fluid tank爲固定的流場，不必更動



# 邊界條件設定

Step 7:邊界條件設定

Define→Boundary Conditions




通氣環設定：

Sparger-inlet在下拉選單中選擇air

流速輸入0.1

Multiphase中輸入1

流速爲空氣流速，Multiphase的設定是表示從此孔進入的皆爲空氣。



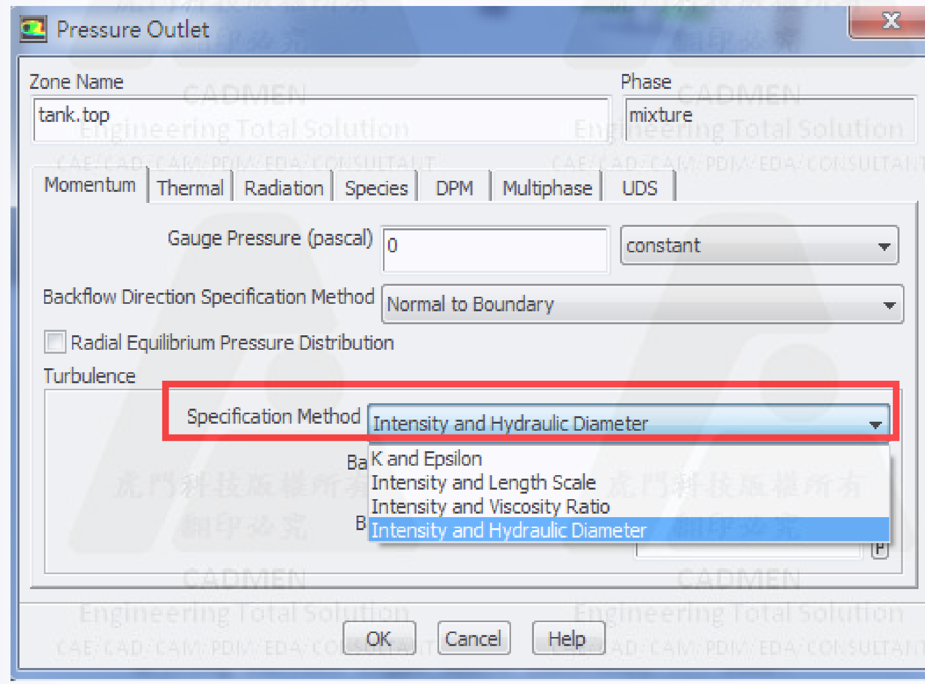
# ANSYS 邊界條件設定

Step 7:邊界條件設定

Define→Boundary Conditions

壓力出口設定：
Tank.top在下拉選單中選擇mixture

設定亂流強度及水力半徑
亂流強度：5%
水力半徑：2m
接著在下拉選單中選擇air，Multiphase一樣設定成1，使回流的液體爲空氣。



# 邊界條件設定

Step.7:邊界條件設定

Define→Boundary Conditions

轉軸設定：

shaft在下拉選單中選擇mixture

點選Moving Wall

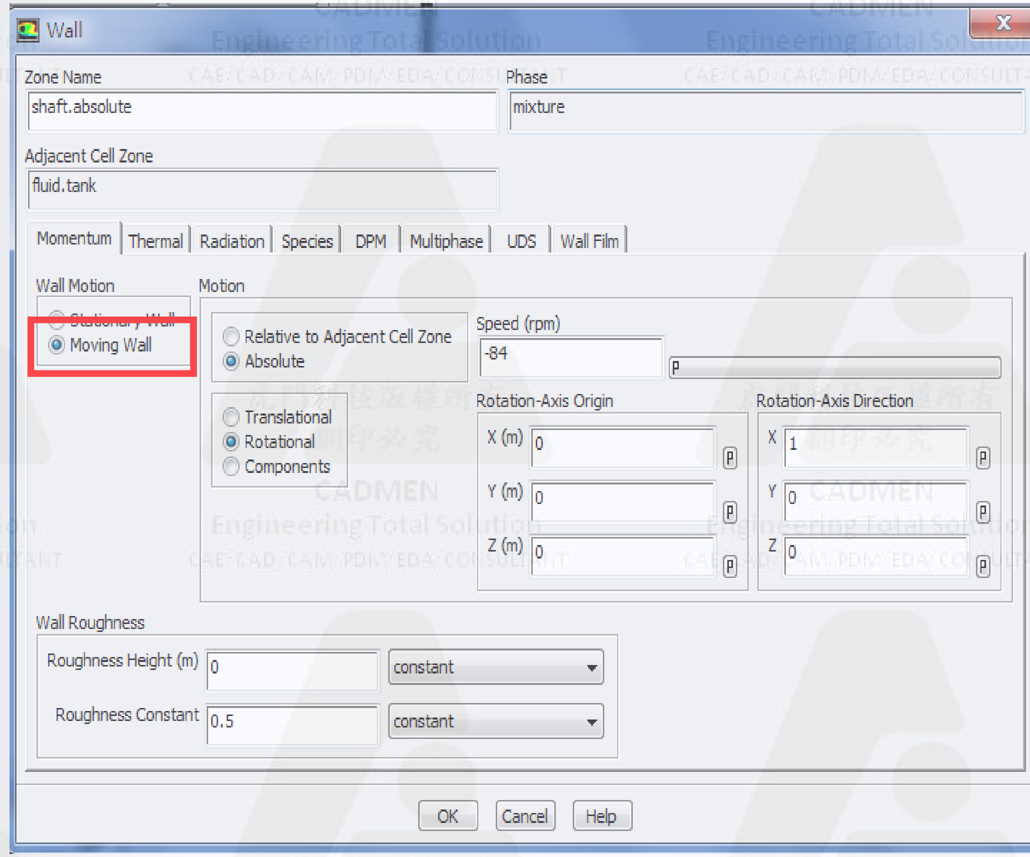
轉速設定84rpm

shaft.absolute

shaft.cd-6

shaft.he-3

皆套用以上設定



# 設定求解器

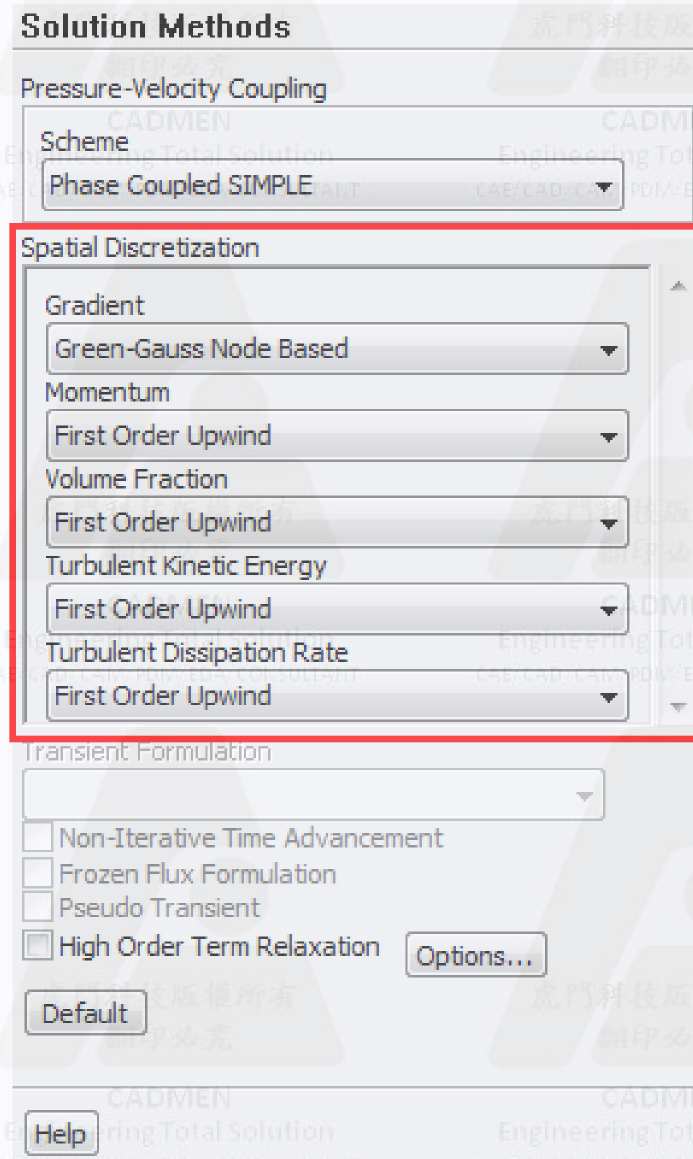
Step.8:求解器設定
Solve→methods

將梯度設定Green-Gauss Node Based
Momentum

Volume Fraction

Turbulent Kinetic Energy

Turbulent Dissipation Rate
皆設定成一階



# 設定流體區域

Solution 1:

設定流體區域

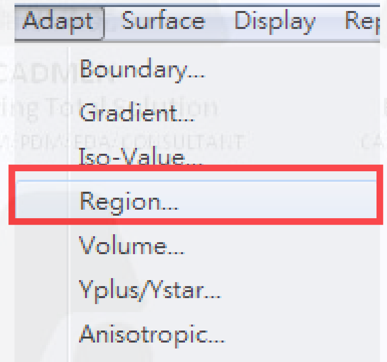
Adapt→Region

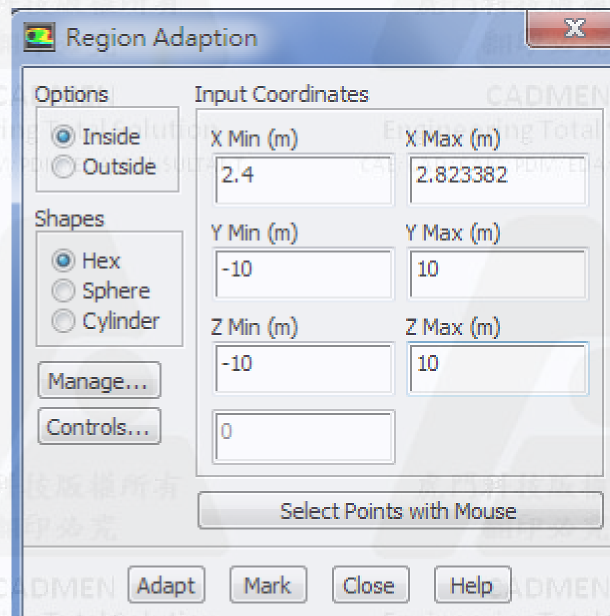
X：2.4～2.823382

Y：-10～10

Z：-10～10

點選Mark



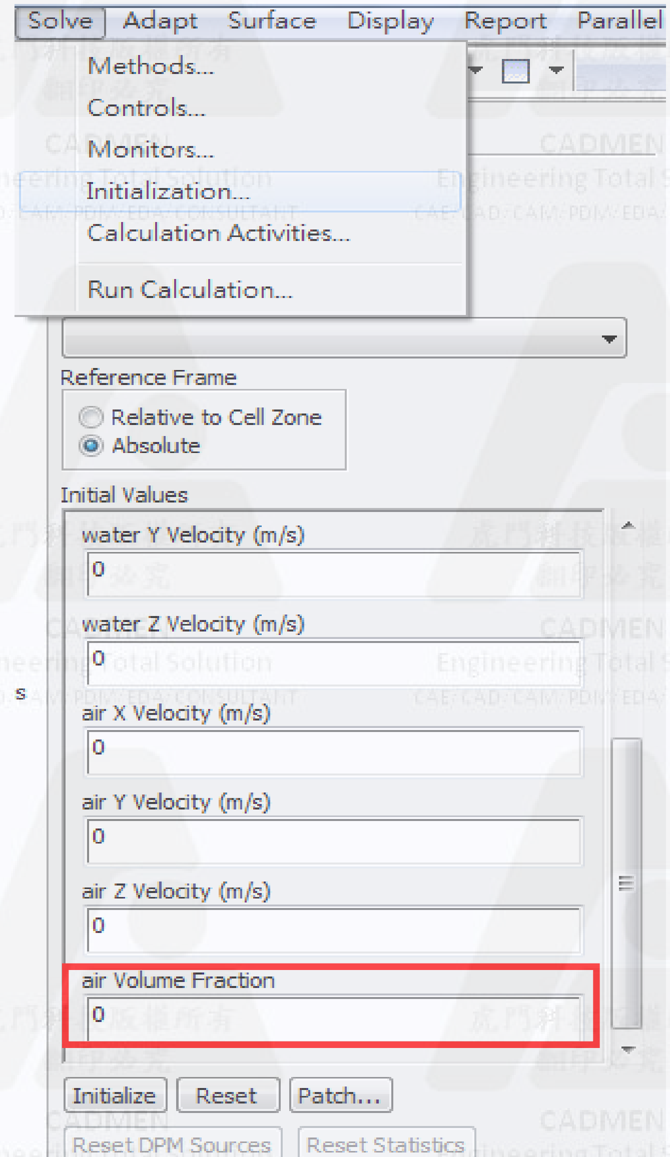
# ANSYS 初始化

Solution 1:

環境初始化
Adapt→Region

將air volume Fraction設定為0

接著選擇Patch



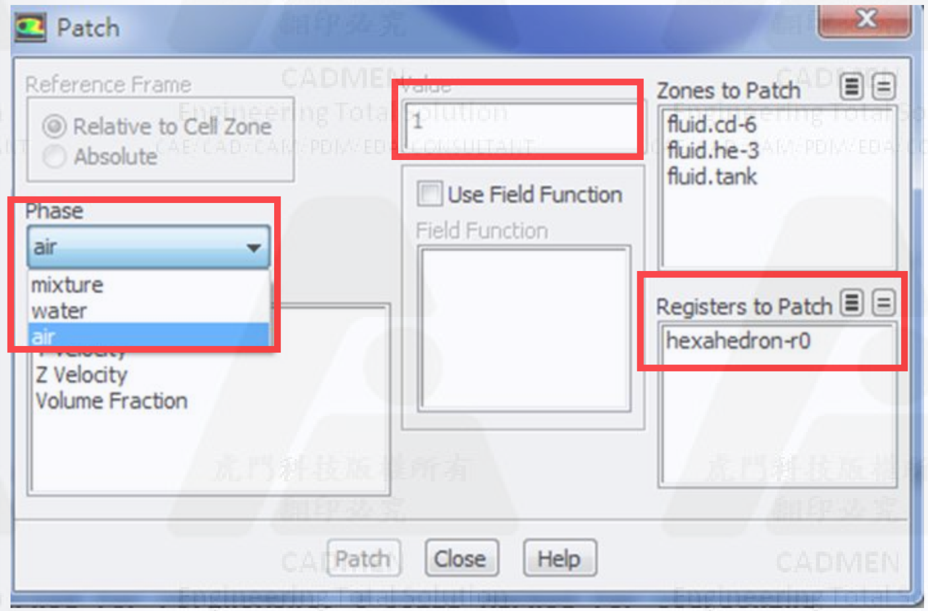

  

# 設定流體區域

Solution 1:

將區域設定爲氣相

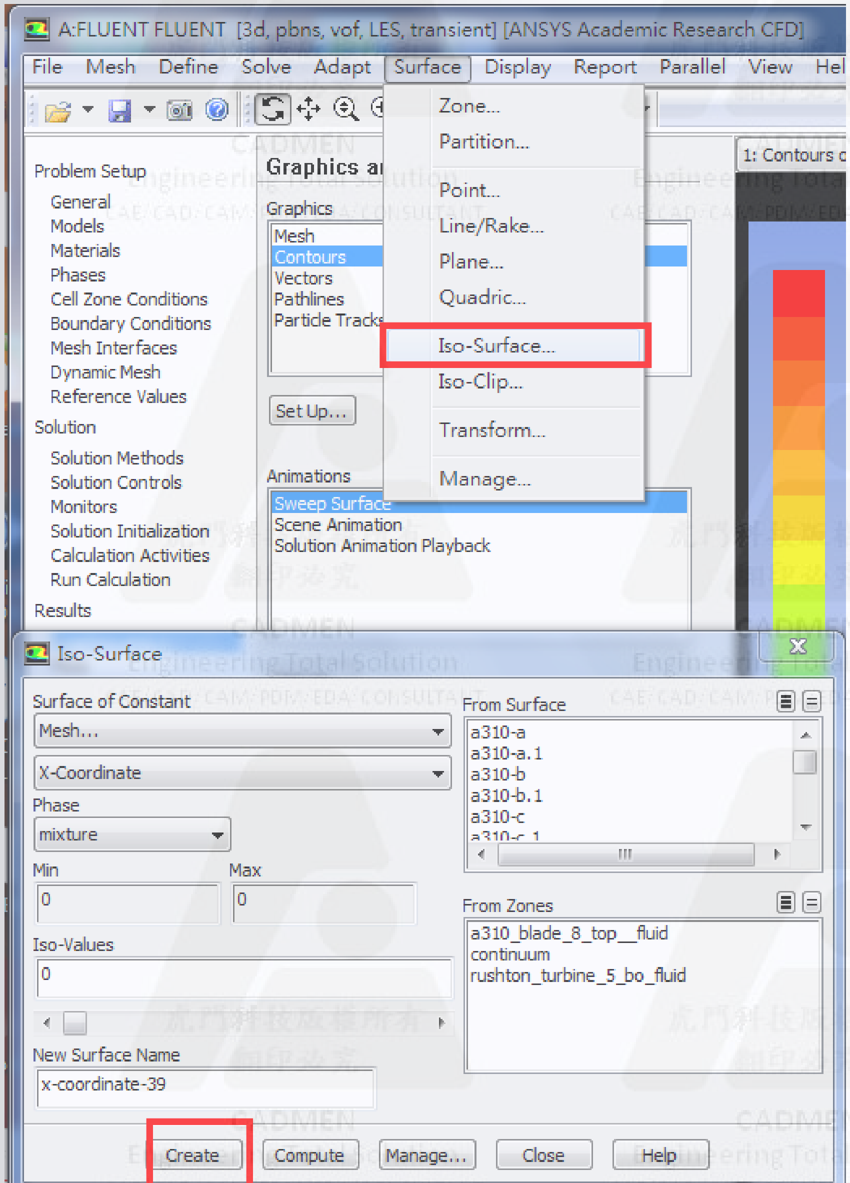
將value值設定成1
選擇Patch完成

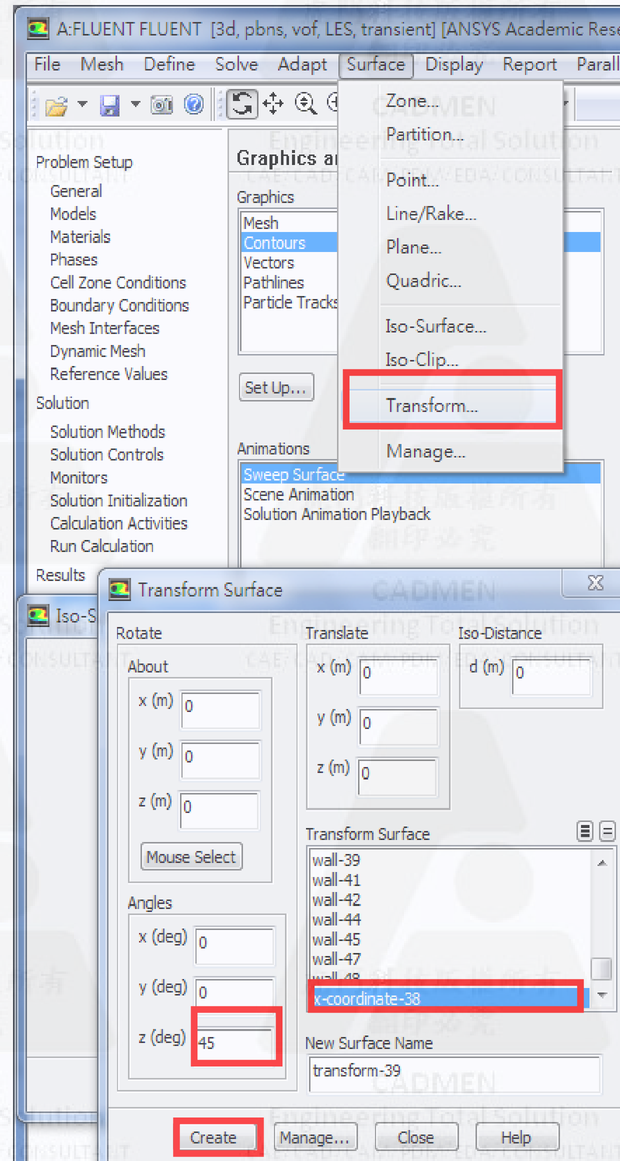


# 設定觀察切面＆旋轉切面

先設定切面，但是因為切面會在擋板上，所以將切面以Z軸旋轉45°

方便求解完之後的觀察。



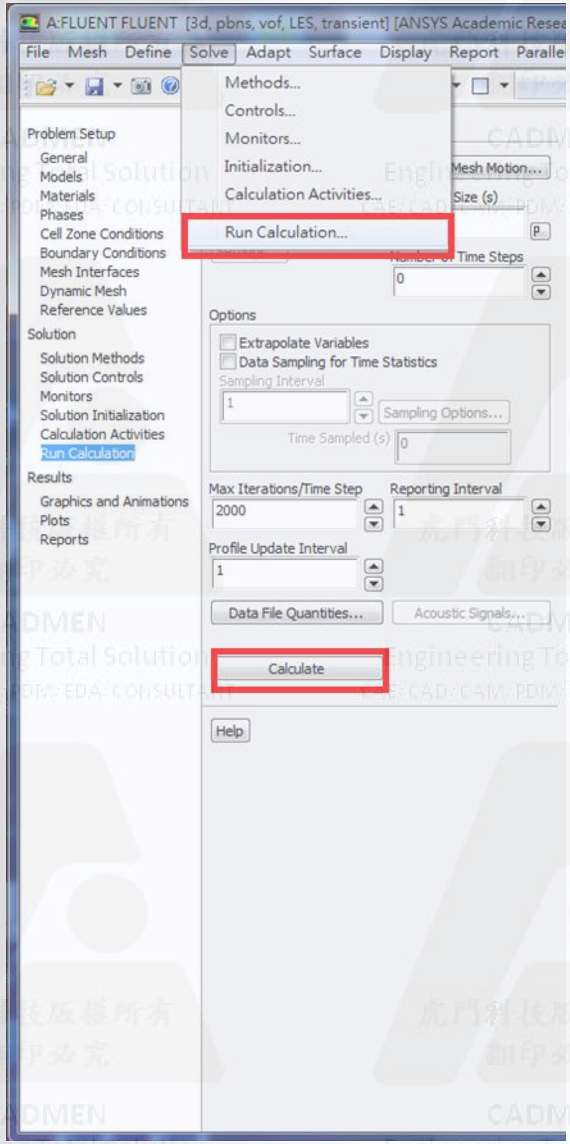





# 求解

求解


A:FLUENT FLUENT [3d, pbns, vof, LES, transient] [ANSYS Academic Resea
File Mesh Define Solve Adapt Surface Display Report Paralle
Methods...
Controls...
Monitors...
Initialization...
Calculation Activities...
Run Calculation...
Problem Setup
General
Models
Materials
Phases
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Solution Initialization
Calculation Activities
Run Calculation
Results
Graphics and Animations
Plots
Reports
Mesh Motion...
Size (s)
Number of Time Steps
0
Options
Extrapolate Variables
Data Sampling for Time Statistics
Sampling Interval
1
Sampling Options...
Time Sampled (s)
0
Max Iterations/Time Step
2000
Reporting Interval
1
Profile Update Interval
1
Data File Quantities...
Acoustic Signals...
Calculate
Help

ANSYS

# 後處理

後處理時，可以觀察

網格結構圖

流場等位面圖

流場向量圖

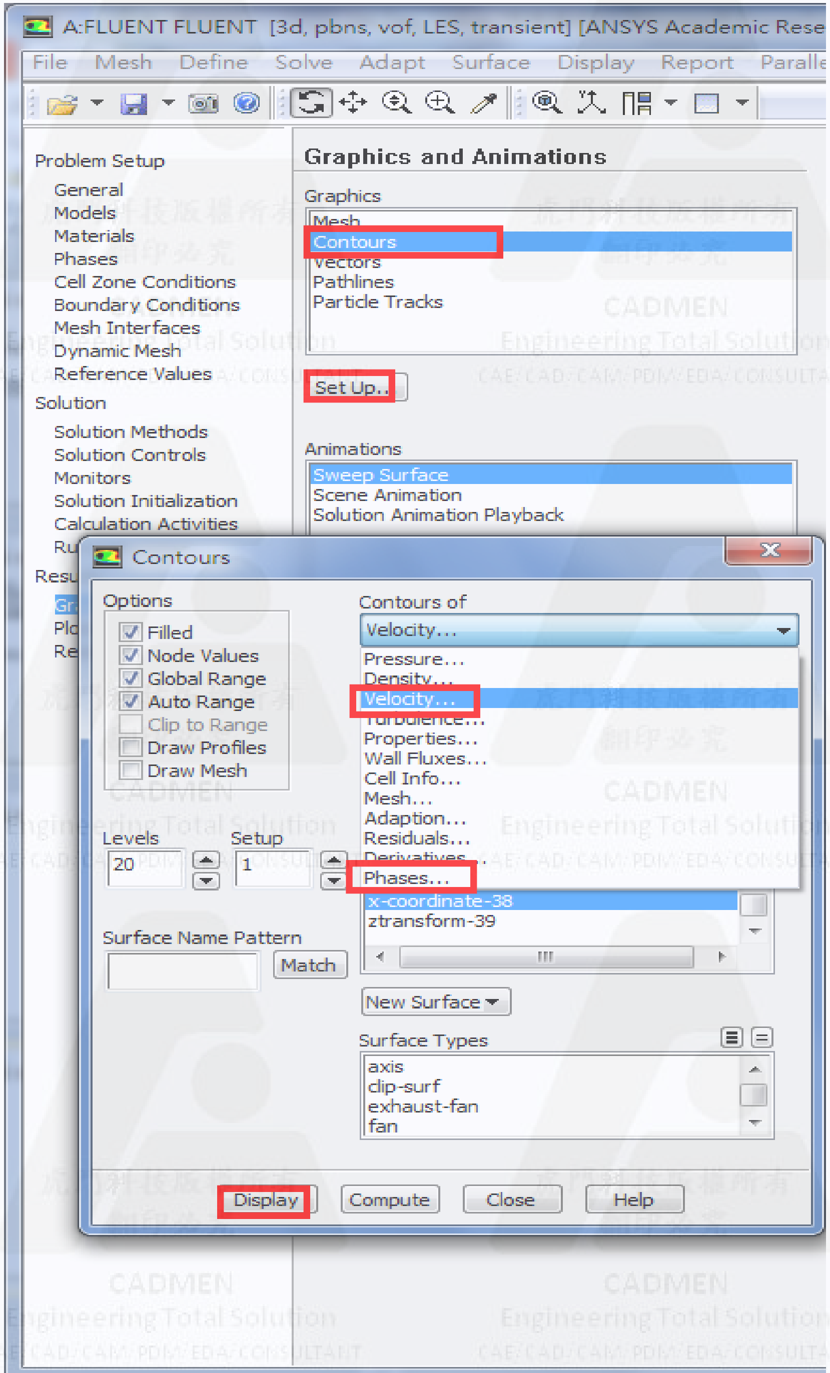
流場流線圖

粒子軌跡圖

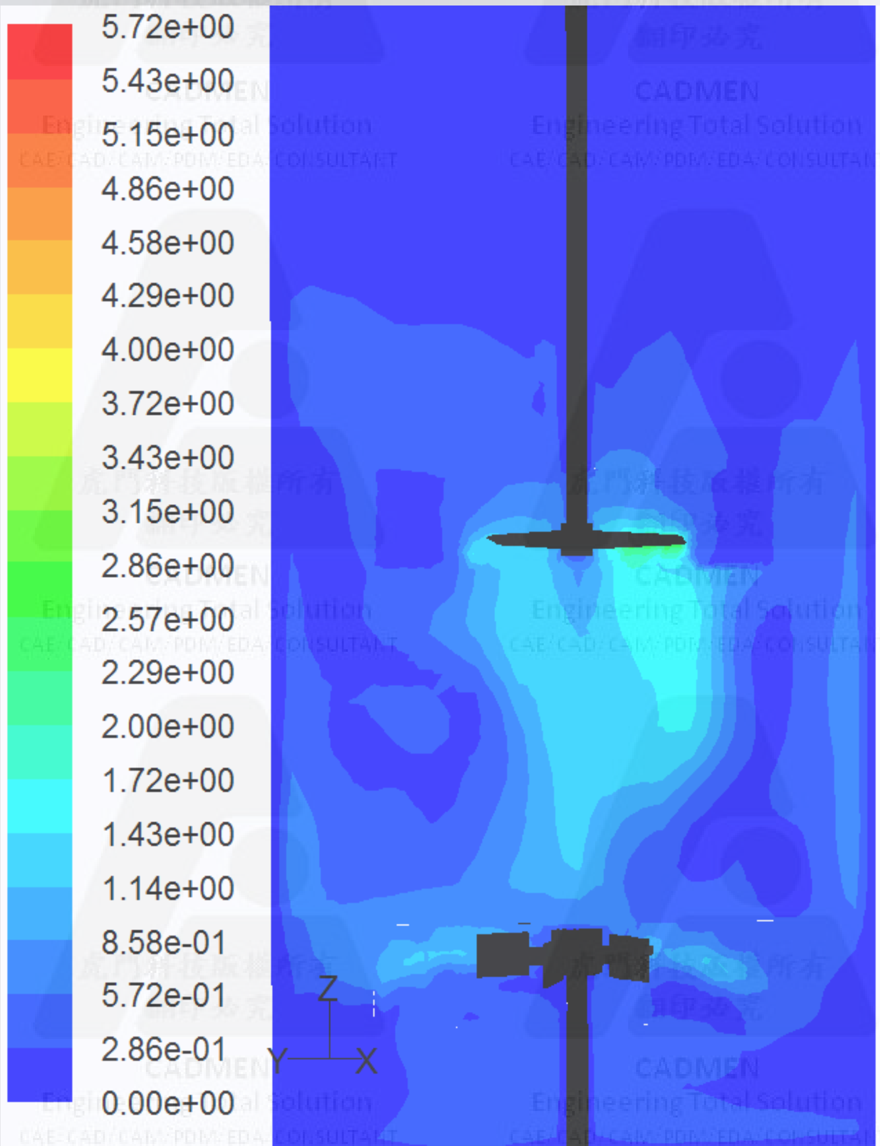
可以選擇各項觀察條件：

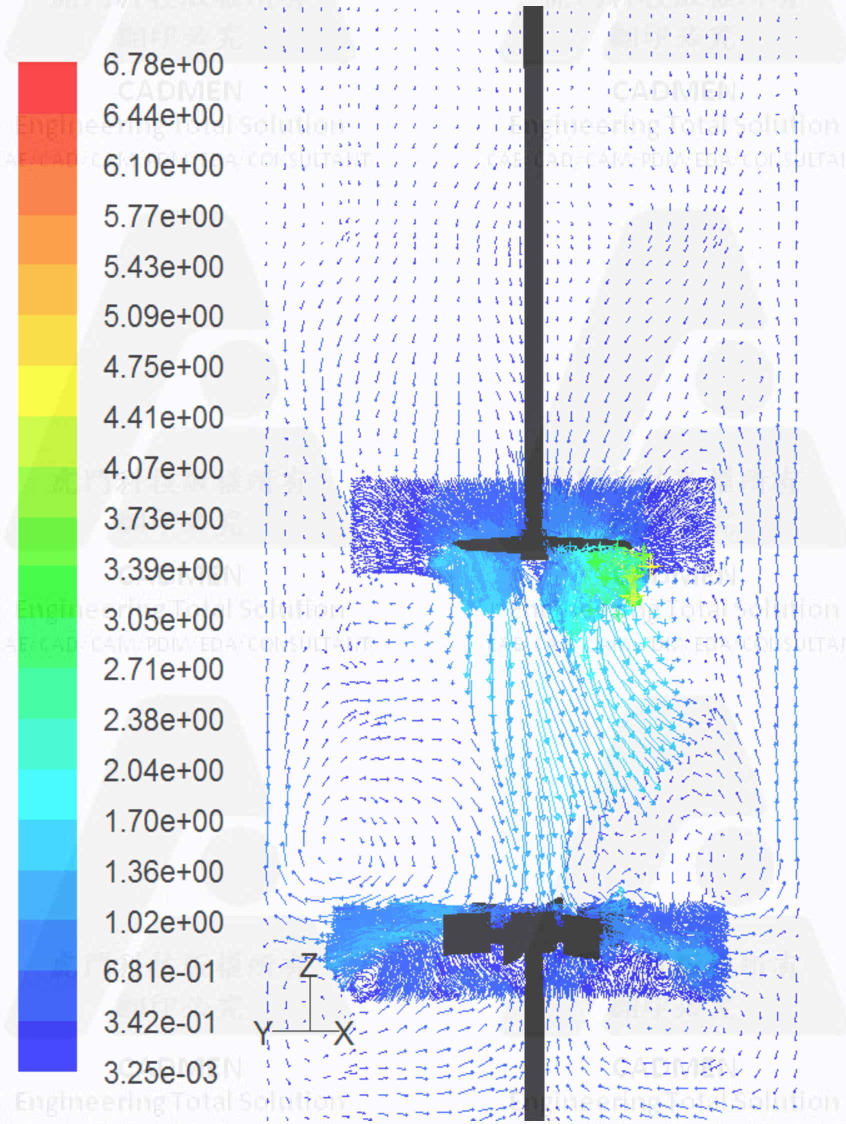
壓力、密度、速度、亂流.....等

選擇之前設定好的面，Display，完成！



# 流場等位面圖&流場向量圖



# Thank you…

