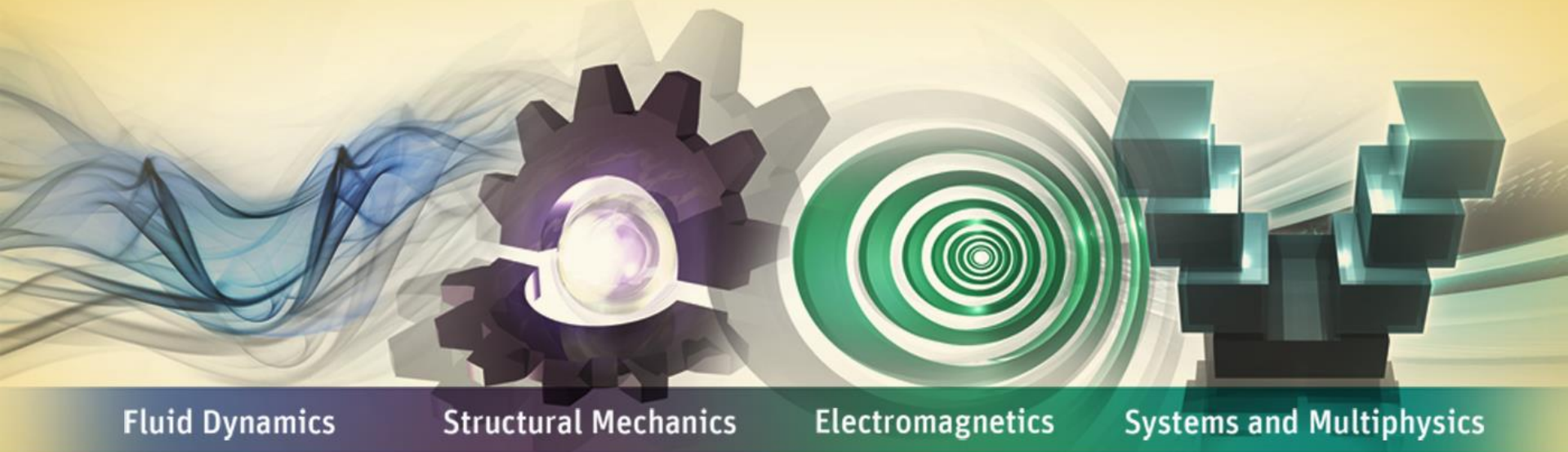


# CAE工程輔助分析 流體傳動元件上的應用



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

**李龍育 Dragon**  
**CAE事業群, 協理**  
**虎門科技**

Line: 0927520067

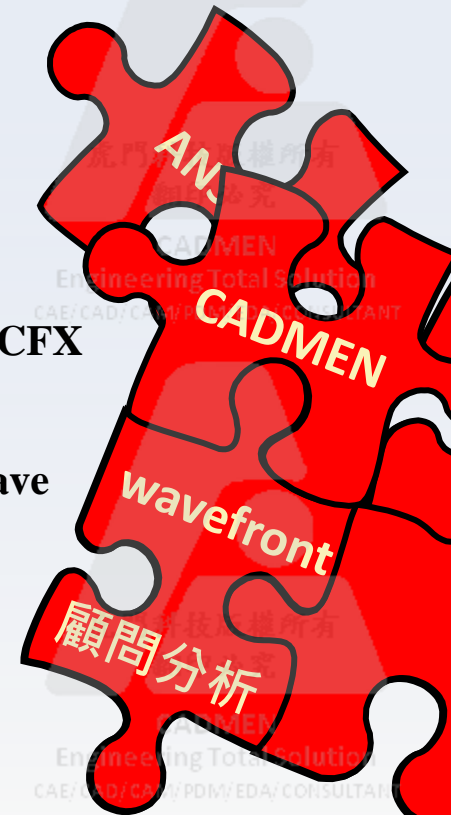
Email: [dragon.li@cadmen.com](mailto:dragon.li@cadmen.com)



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，同時榮獲ANSYS菁  
英級代理商與國家磐石獎的殊榮

- 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 Maxwell、HFSS、SIwave
- 多物理耦合分析

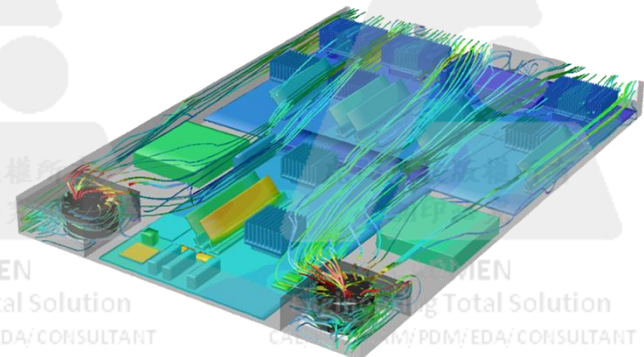
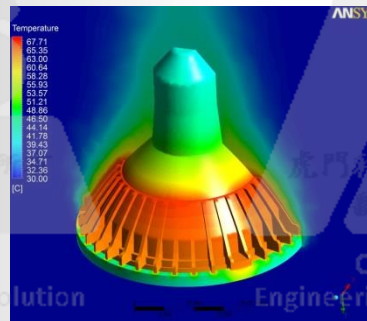
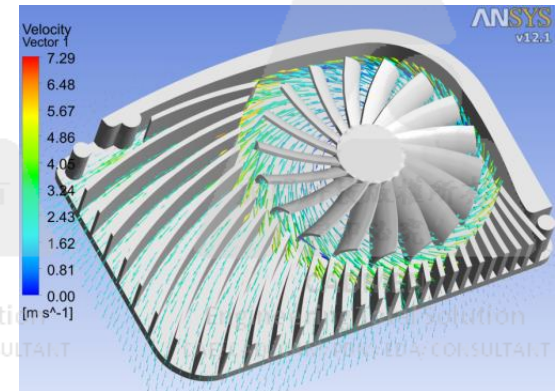
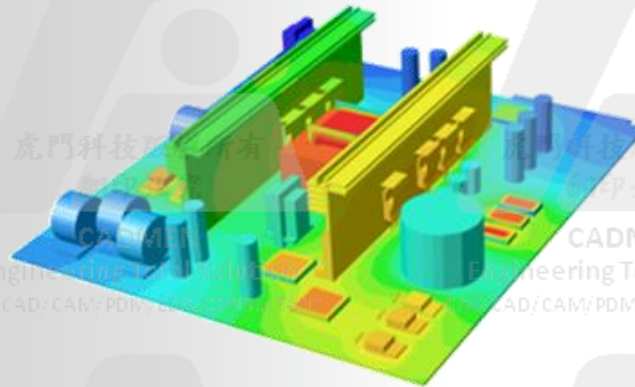
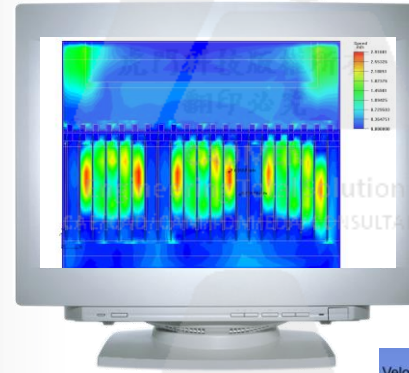
- 總公司：新北市板橋區
- 分公司：台中、新竹、台南



Provider of Engineering Solutions and Methodology

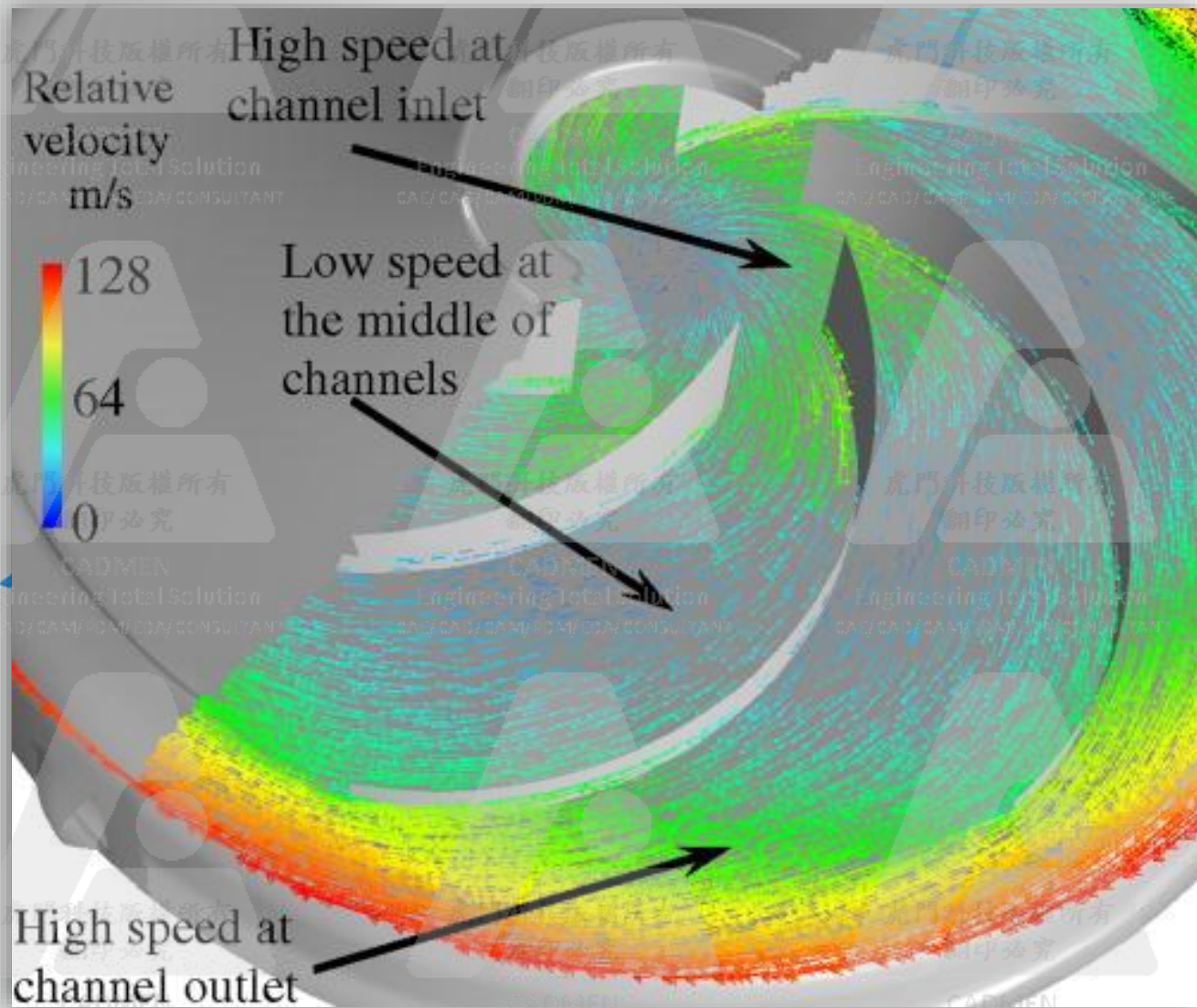
- 數位實驗
- 設計與偵錯工具
- 深入了解產品問題
- 改良產品性能表現

- 研發成本
- 開發時效
- 性能提升
- 創新研發利器



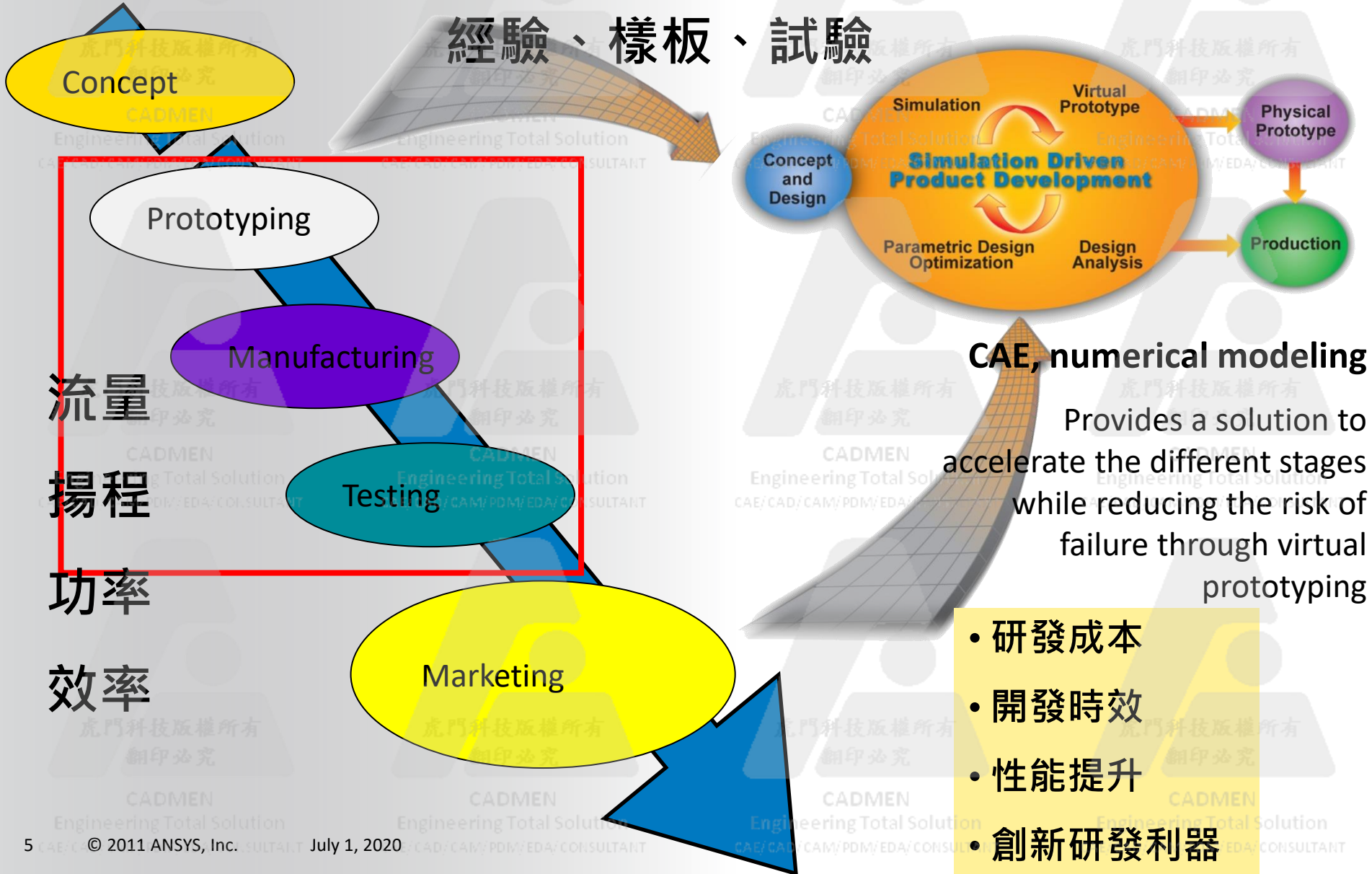
- 數位實驗
- 設計與偵錯工具
- 深入了解產品問題
- 改良產品性能表現

- 研發成本
- 開發時效
- 性能提升
- 創新研發利器



# Virtual Prototyping for PUMP Design

經驗、樣板、試驗



流量

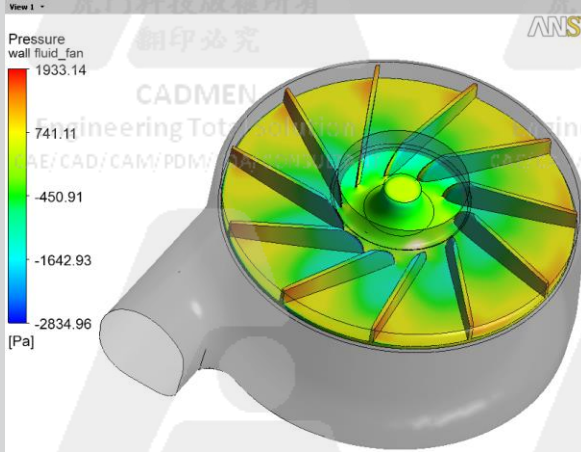
揚程

功率

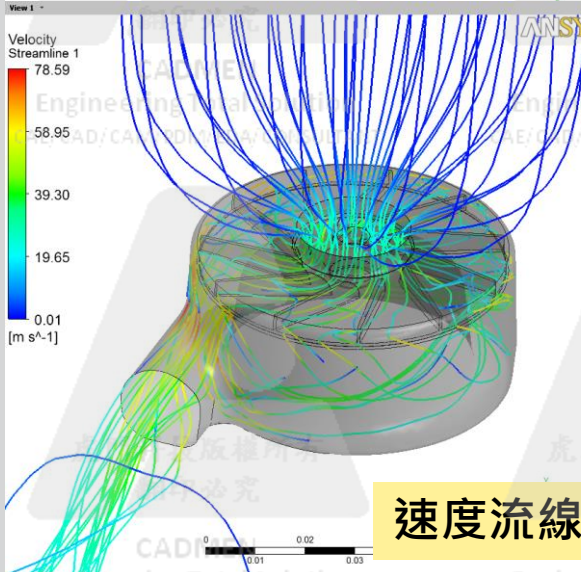
效率

# Virtual Prototyping for PUMP Design

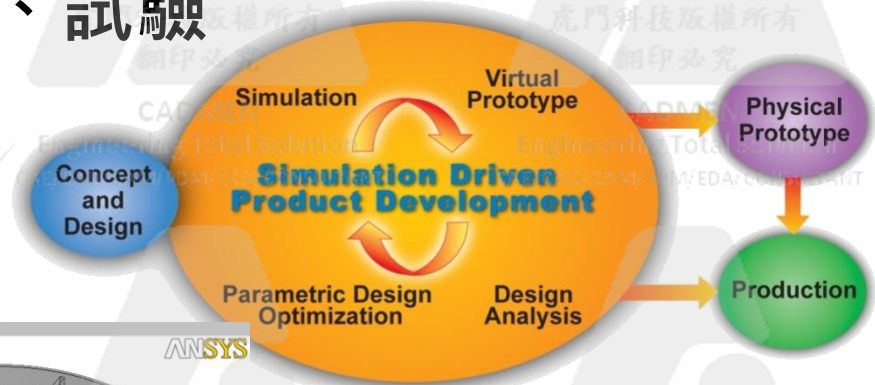
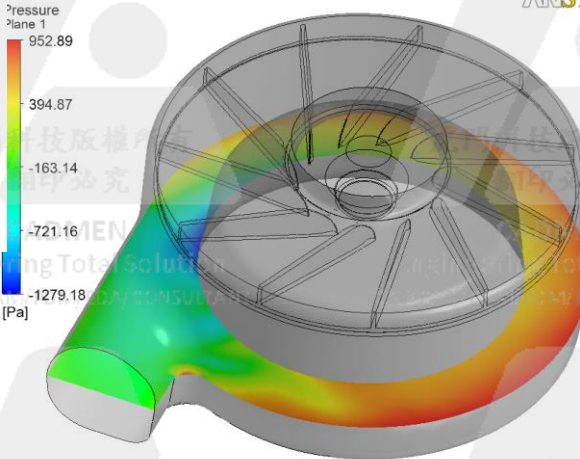
經驗、樣板、試驗



壓力分佈



速度流線



CAE, numerical modeling

Provides a solution to accelerate the different stages while reducing the risk of failure through virtual prototyping

- 研發成本
- 開發時效
- 性能提升
- 創新研發利器

# 將CAE導入研發流程具以下優點：

- 於電腦內做出虛擬原型機 (Virtual Prototypes)分析，降低實際原型機(True prototypes) 測試成本。
- 累積技術 know-how，提升研發能力。
- 抓出產品之設計規則與重要參數，更了解產品的物理特性。
- 產品不良點提前於設計階段改善，減少量產後之不良問題。
- 針對產品破壞個案進行分析，找出損壞原因 (最大應力 ...)
- 降低傳統設計流程之試誤法 (trial-and-errors) 次數。
- 做出更好更安全之產品。
- 降低成本，縮短上市時間 (time to market)。
- CAE 分析成果和動畫，可做為宣傳工具。

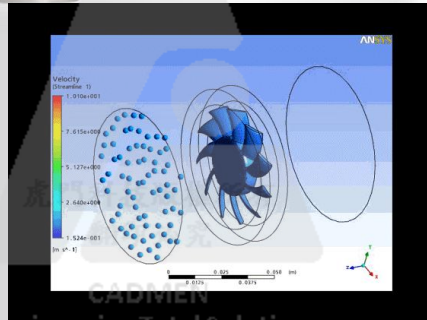
# 創新的研發 - Household Fan



1882

130 Years of Real Innovation?

Dyson

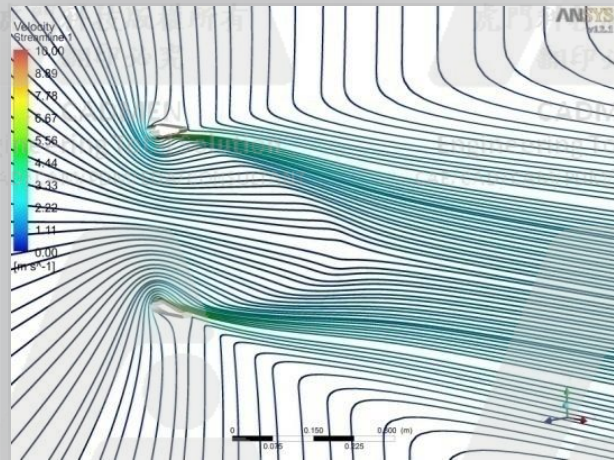


- Remove chopiness associated with all conventional fan designs...
- Maximize amplification ratio for a given size and power consumption
- More stylish design
- **Less dangerous!**

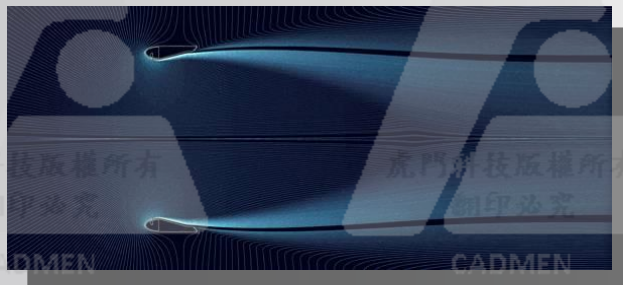
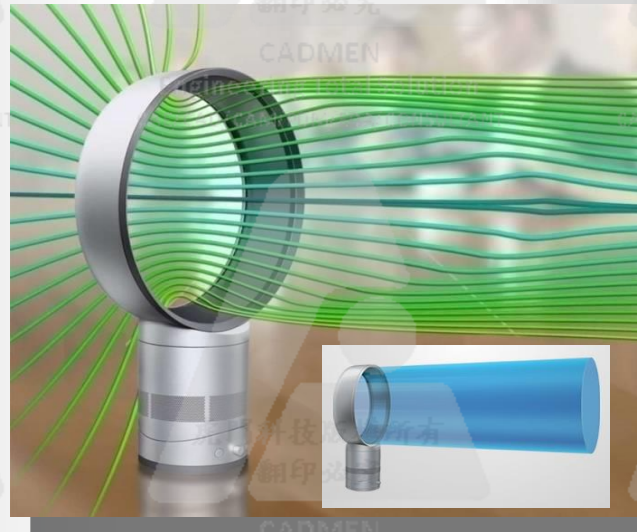
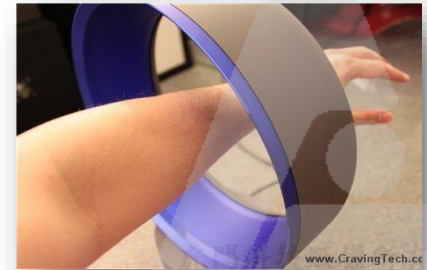




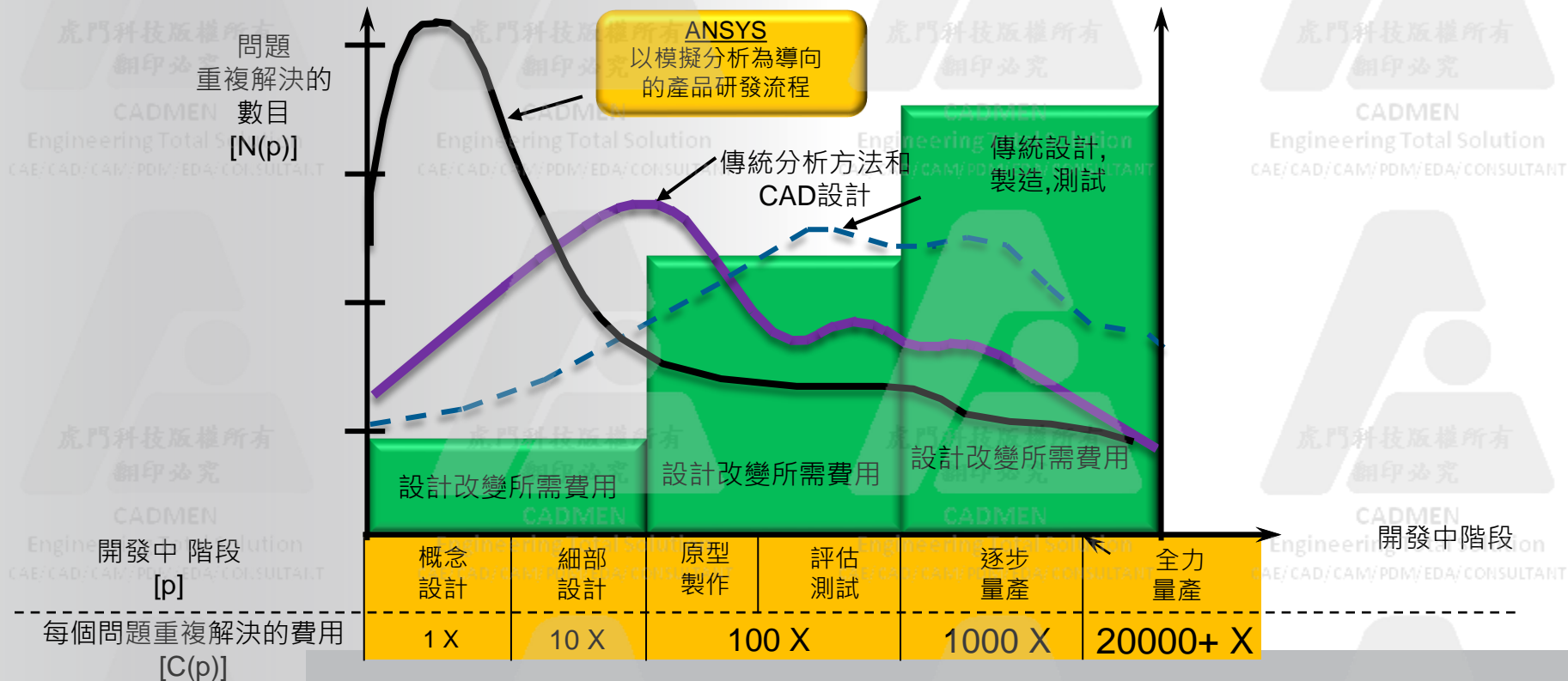
# 創新的研發 - *Dyson Reinvents the Fan*



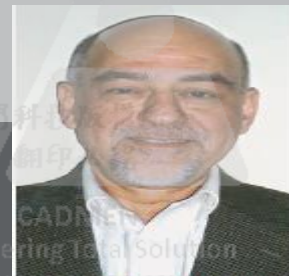
- Improved performance 250% over original design
- Continuous airflow!
- Safe device...



# 用CAE-展現創新的研發力



“大部分的高科技公司，現在已經了解到模擬分析對產品效能所帶來的潛在效益。他們甚至更清楚的了解到在整個設計周期中，提前分析所帶來的好處，比事後再來發現與解決問題來得更有效率與節省成本。”



Fereydoon Dadkhah, Delphi 电子与安全系统事业部

# ANSYS and the Electronics Design Flow

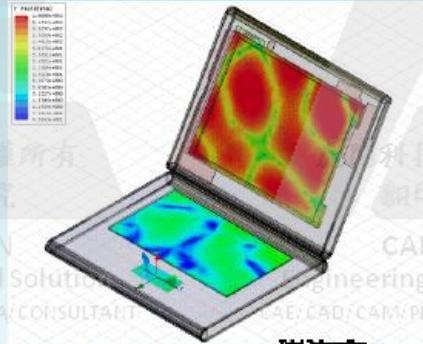
End Product

結構強度



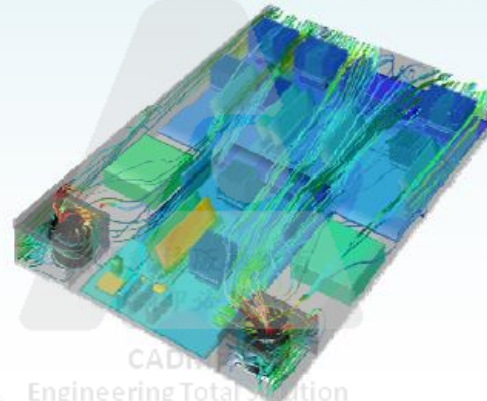
散熱設計

碰撞落摔



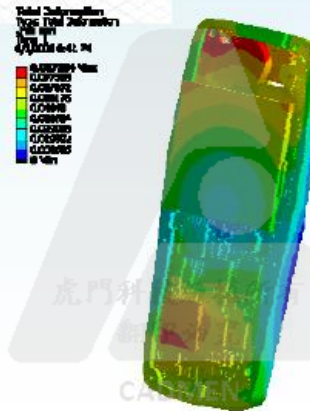
電路設計

包材設計



訊號干擾

震動噪音

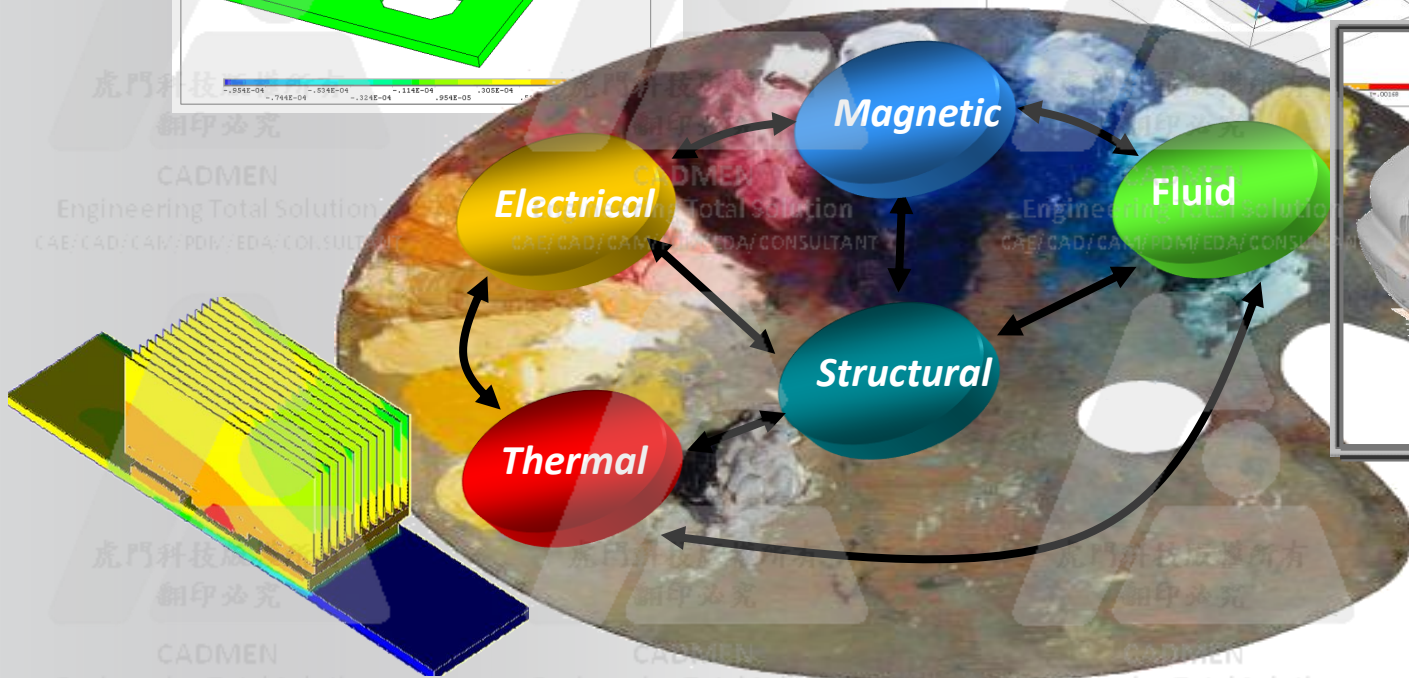
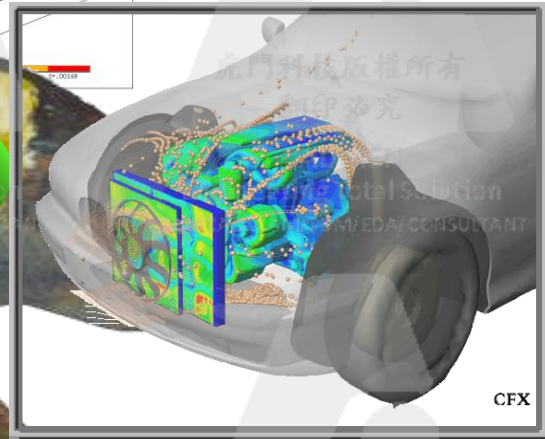
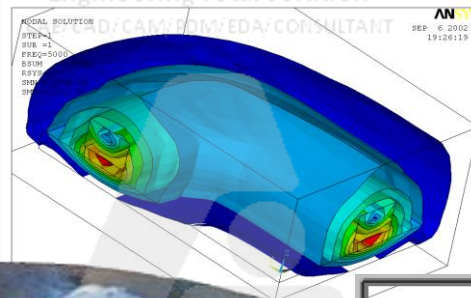
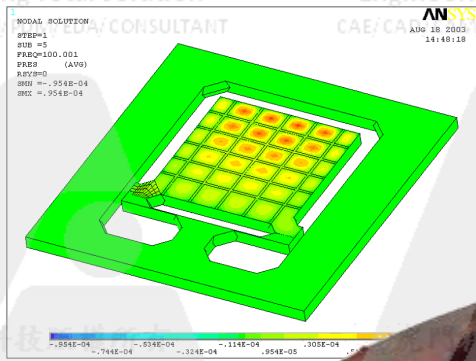


整合運算

# 全方位的CAE解決方案

## In WorkBench

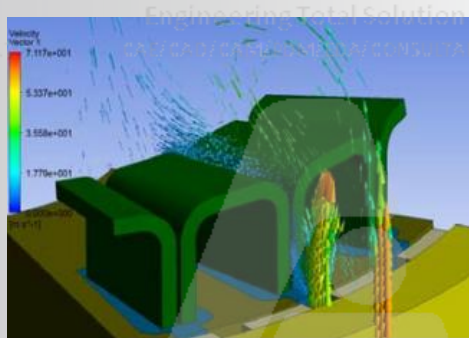
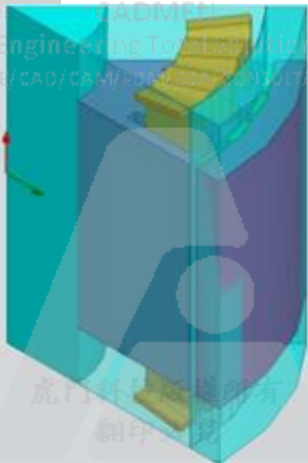
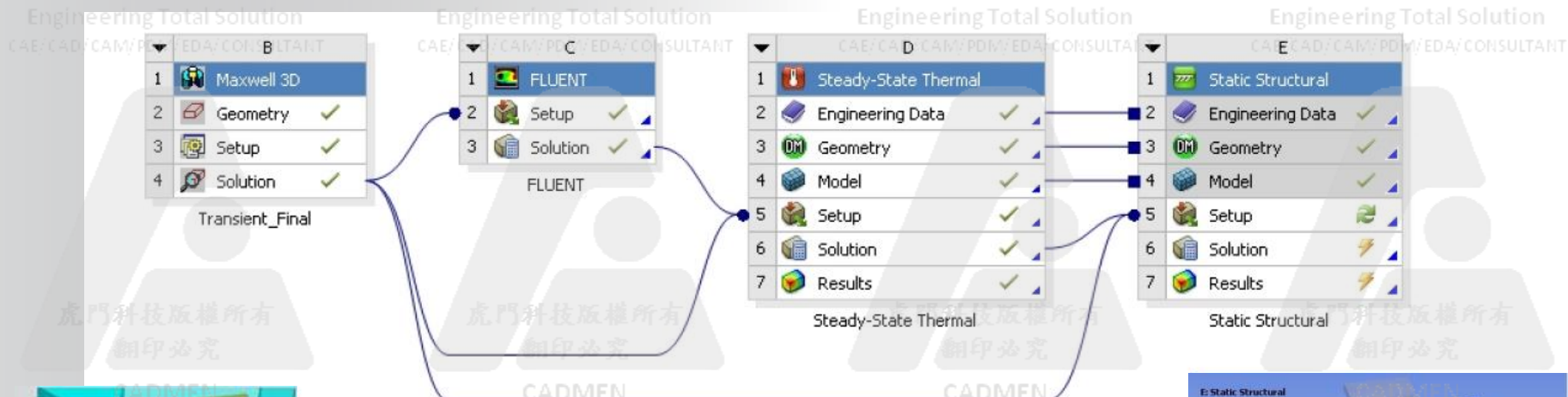
全方位解決方案—對眾多複雜的物理現象的整合解析  
From electromagnetic fields to heat, cooling, and stress



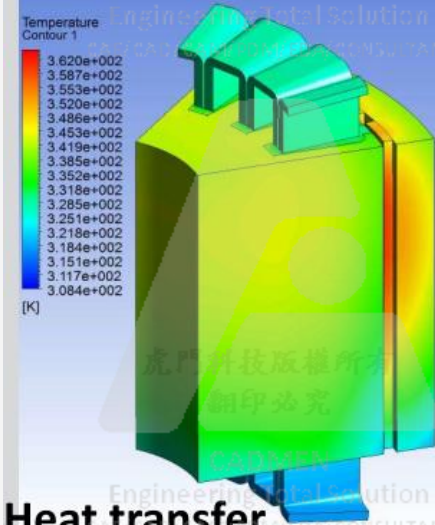
# 全方位的CAE解決方案

In WorkBench

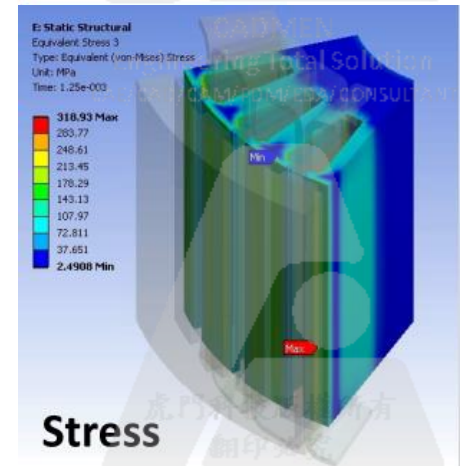
全方位解決方案—對眾多複雜的物理現象的整合解析  
From electromagnetic fields to heat, cooling, and stress



Thermo-fluid



Heat transfer



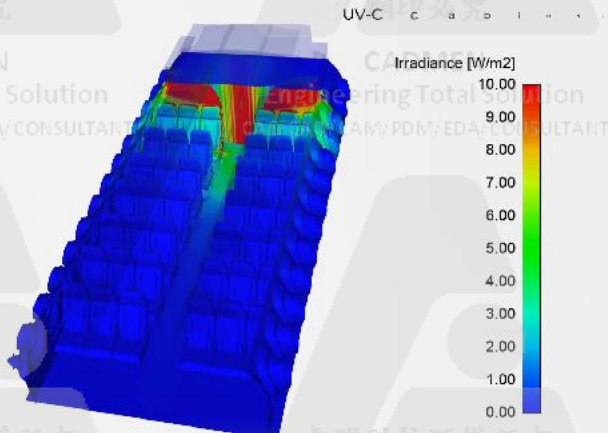
Stress

Electromagnetic field

# CFD for COVID-19

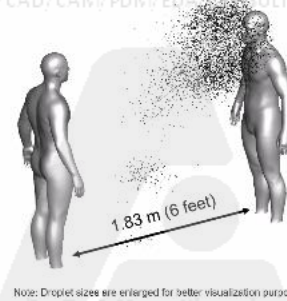
## Simulation-Based Insights to Limit the Spread

Disinfect airline cabins with UV light

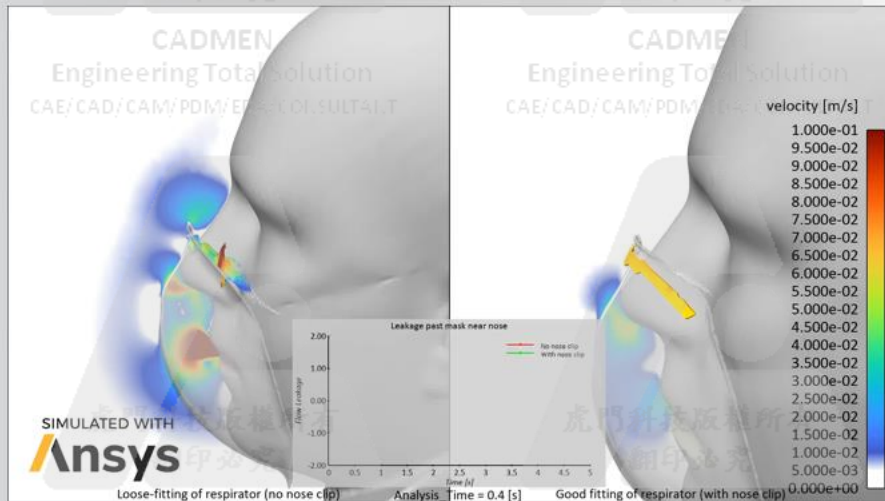
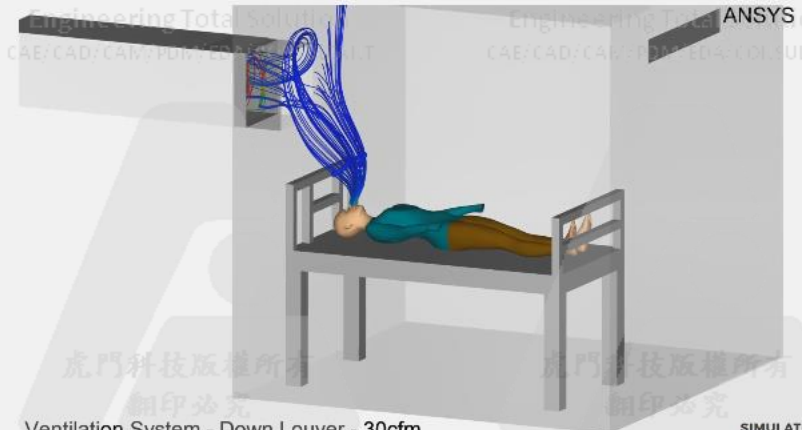


Consider wind when social distancing

Static Air with 1.83 m (6 Feet) Social Distance

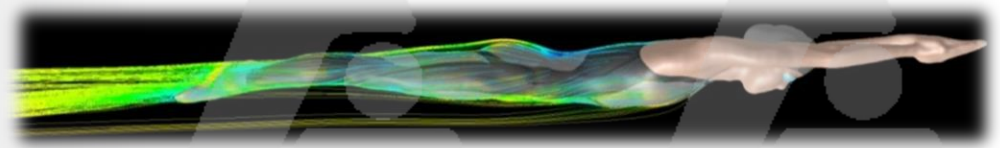


Designing a Negative Pressure Room to Minimize Contamination



Wear a mask

# What is CFD?



**Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena.**

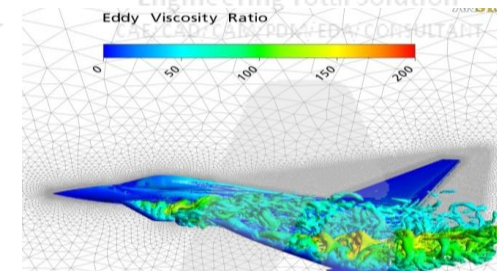
**To predict these phenomena, CFD solves equations for conservation of mass, Newton's second law ( $F=ma$ ), conservation of energy etc.**

**CFD can provide detailed information on the fluid flow behavior:**

- Distribution of pressure, velocity, temperature, etc.
- Forces like Lift, Drag... (external flows, Aero, Auto...)
- Distribution of multiple phases (gas-liquid, gas-solid...)
- Species composition (reactions, combustion, pollutants...)
- Much more...

**CFD is used in all stages of the engineering process:**

- Conceptual studies of new designs
- Detailed product development
- Optimization
- Troubleshooting
- Redesign

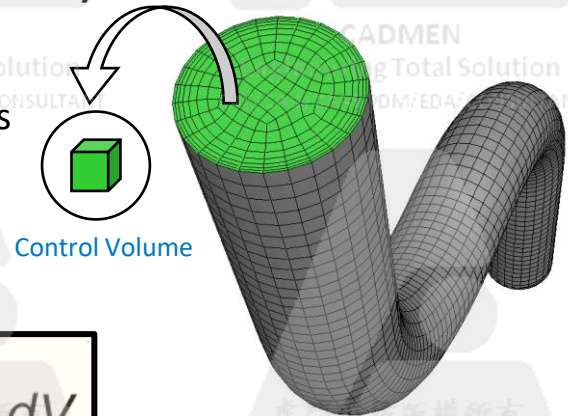


**CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation**

# How Does CFD Work ?

## ANSYS Fluent solvers are based on the finite volume method (FVM)

- Domain is discretized into a finite set of control volumes
- General transport equations are solved on this set of control volumes
  - Continuity equation (conservation of mass)
  - Momentum equations (Newton's second law)
  - Energy equation (conservation of energy)



Eulerian description

$$\frac{\partial}{\partial t} \int_V \rho \phi dV + \oint_A \rho \phi \mathbf{V} \cdot d\mathbf{A} = \oint_A \Gamma_\phi \nabla \phi \cdot d\mathbf{A} + \int_V S_\phi dV$$

Unsteady

Convection

Diffusion

Generation

- Partial differential equations are discretized into a system of algebraic equations
- All algebraic equations are then solved numerically to render the solution field

| Equation   | $\phi$ |
|------------|--------|
| Continuity | 1      |
| X momentum | $u$    |
| Y momentum | $v$    |
| Z momentum | $w$    |
| Energy     | $h$    |

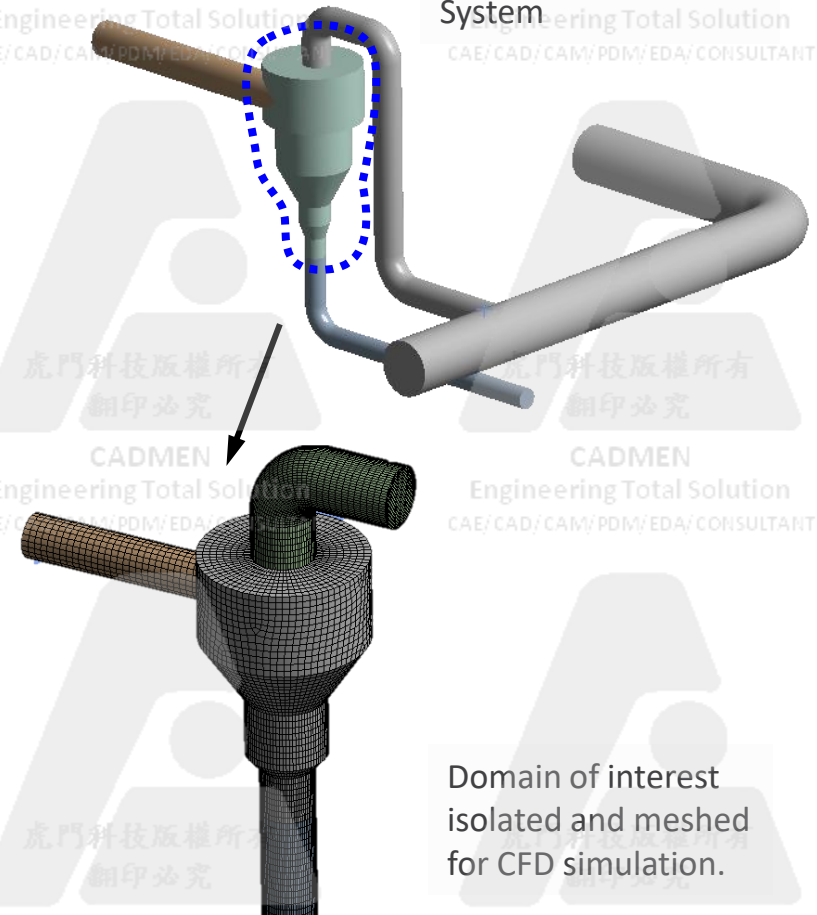


# Step 1. Define Your Modeling Goals

- What results are you looking for (i.e. pressure drop, mass flow rate), and how will they be used?
- What are your modeling options?
  - What simplifying assumptions **can you make** (i.e. symmetry, periodicity)?
  - What simplifying assumptions do you **have to make**?
  - What physical models will need to be included in your analysis
- What degree of accuracy is required?
- How quickly do you need the results?
- Is CFD an appropriate tool?

# Step 2. Identify the Domain You Will Model

- **How will you isolate a piece of the complete physical system?**
- **Where will the computational domain begin and end?**
  - Do you have boundary condition information at these boundaries?
  - Can the boundary condition types accommodate that information?
  - Can you extend the domain to a point where reasonable data exists?
- **Can it be simplified or approximated as an axis-symmetric or 2D problem?**

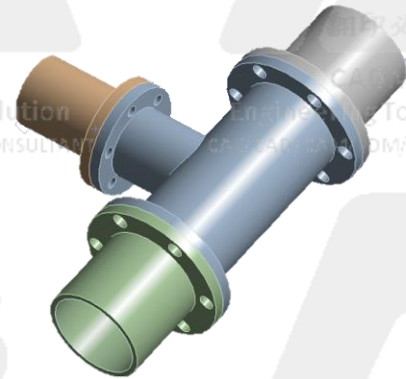


Domain of Interest  
as Part of a Larger  
System

Domain of interest  
isolated and meshed  
for CFD simulation.

# Step 3. Create a Solid Model of the Domain

- **How will you obtain a model of the *fluid* region?**
  - Make use of existing CAD models?
  - Extract the fluid region from a solid part?
  - Create from scratch?
- **Can you simplify the geometry?**
  - Remove unnecessary features that would complicate meshing (fillets, bolts...)?
  - Make use of symmetry or periodicity?
  - Are both the flow and boundary conditions symmetric / periodic?
- **Do you need to split the model so that boundary conditions or domains can be created?**



Original CAD Part



Extracted Fluid Region

# Step 4. Design and Create the Mesh

- **What is the required mesh resolution?**

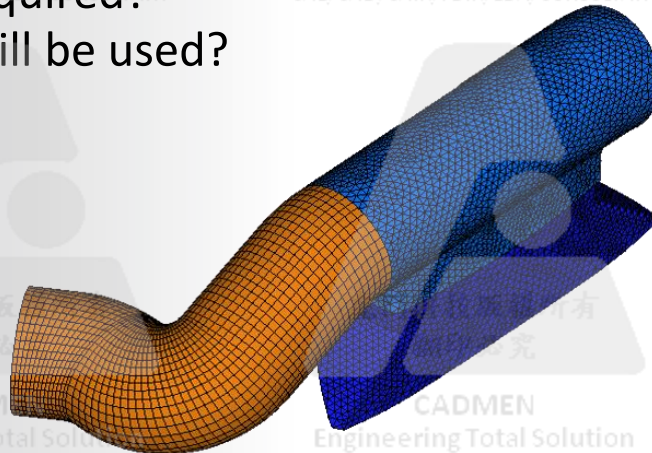
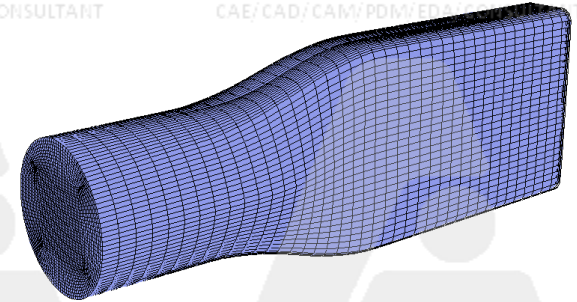
- Resolves geometric features of interest and captures gradients of concern, e.g. velocity, pressure, temperature gradients

- **What type of mesh is most appropriate?**

- Can you use a hexahedral mesh?
- Are non-conformal interfaces needed?

- **Do you have sufficient computer resources?**

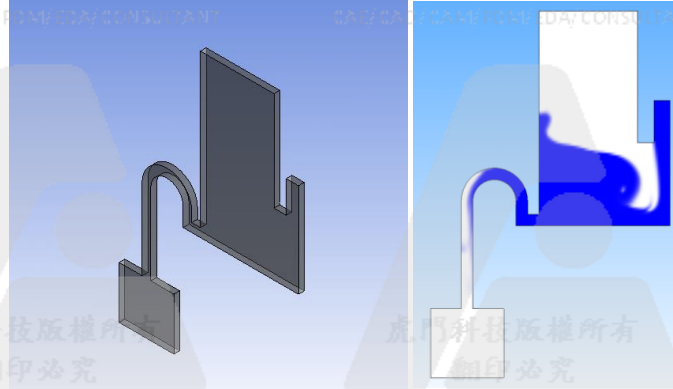
- How many cells/nodes are required?
- How many physical models will be used?



# Step 5. Set Up the Solver

- **For a given problem, you will need to:**

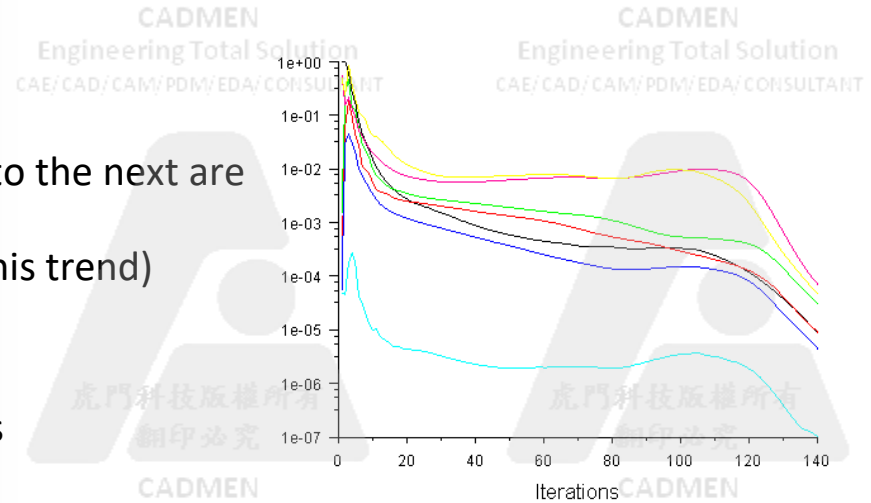
- Define material properties
  - Fluid
  - Solid
  - Mixture
- Select appropriate physical models
  - Turbulence, combustion, multiphase, etc.
- Prescribe boundary conditions at all boundaries
- Provide initial values or a previous solution
- Set up solver controls
- Set up convergence monitors



*For complex problems solving a simplified or 2D problem will provide valuable experience with the models and solver settings for your problem in a short amount of time*

# Step 6. Compute the Solution

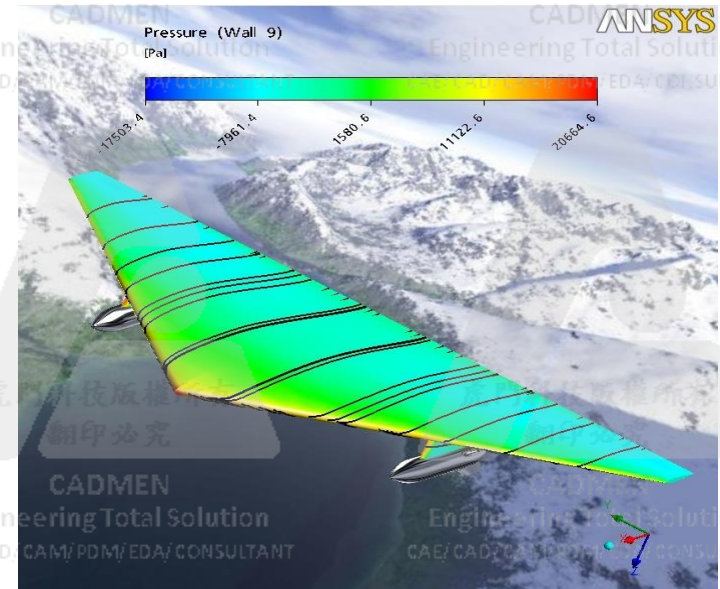
- The discretized conservation equations are solved iteratively until convergence
- Convergence is reached when:
  - Changes in solution variables from one iteration to the next are negligible  
(Residuals provide a mechanism to help monitor this trend)
  - Overall property conservation is achieved  
(Imbalances measure global conservation)
  - Quantities of interest have reached steady values  
(Monitor points track quantities of interest)
- The accuracy of a converged solution is dependent upon:
  - Appropriateness and accuracy of physical models
  - Assumptions made
  - Mesh resolution and independence
  - Numerical errors



*A converged and mesh-independent solution on a well-posed problem will provide useful engineering results!*

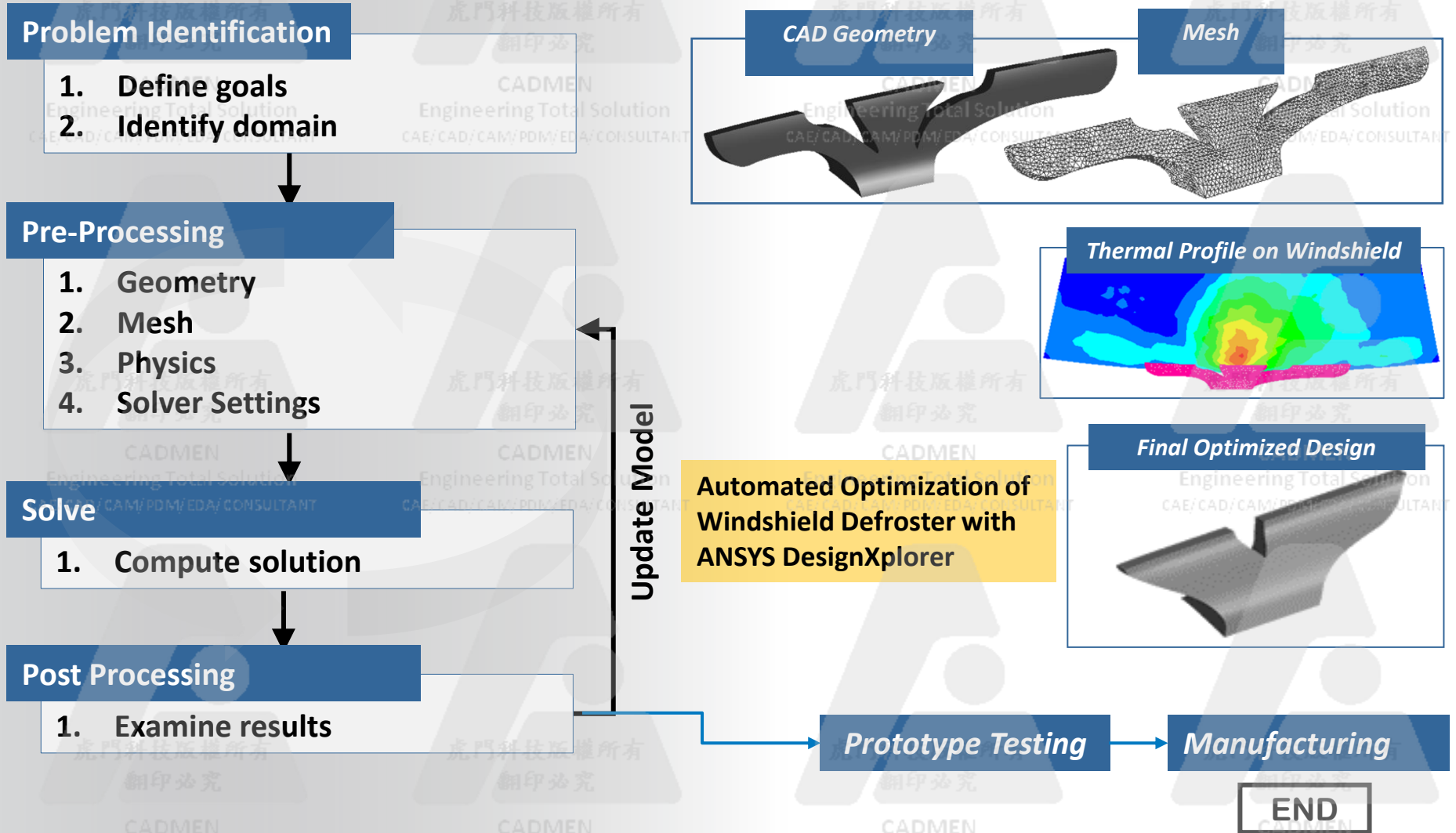
# Step 7. Examine the Results

- **Examine the results to review solution and extract useful data**
  - Visualization Tools can be used to answer such questions as:
    - What is the overall flow pattern?
    - Is there separation?
    - Where do shocks, shear layers, etc. form?
    - Are key flow features being resolved?
  - Numerical Reporting Tools can be used to calculate quantitative results:
    - Forces and Moments
    - Average heat transfer coefficients
    - Surface and Volume integrated quantities



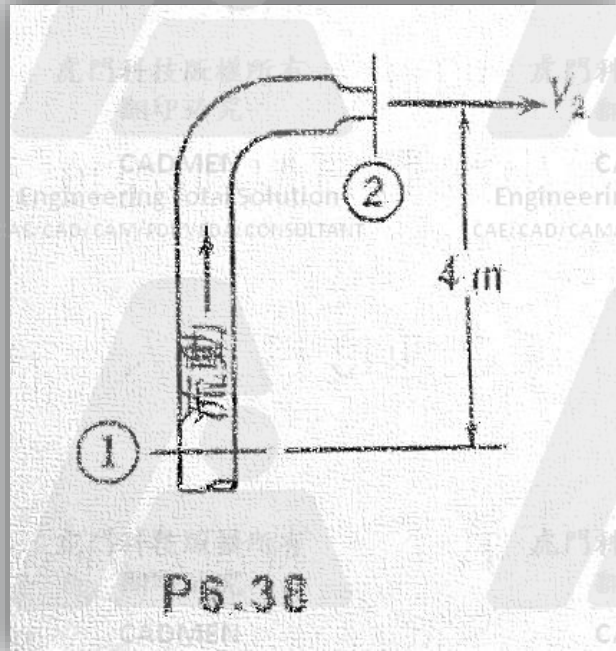
Examine results to ensure correct physical behavior and conservation of mass energy and other conserved quantities. High residuals may be caused by just a few poor quality cells.

# ANSYS CFD Workflow





# 問題描述



- 水穩定地沿著直徑為 0.1m 的垂直圓管往上流，並由直徑 0.05m 的噴嘴放流至大氣壓中。
- 噴嘴出口處的流速必須為 20 m/sec。
- 試求在截面 ① 處所需之錶壓。

$$Q_1 = Q_2$$

$$A_1 V_1 = A_2 V_2$$

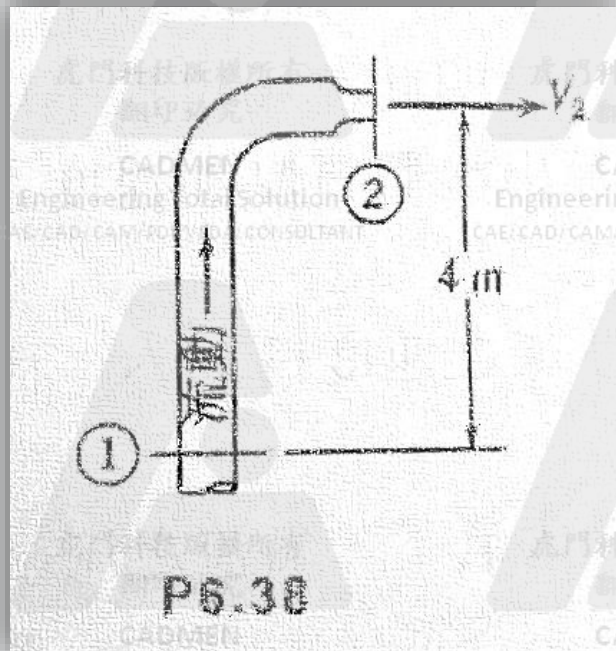
$$\frac{\pi \times 0.1^2}{4} V_1 = \frac{\pi \times 0.05^2}{4} \times 20$$

$$V_1 = 5 \frac{m}{s}$$

$$\frac{P_1}{\gamma} + Z_1 + \frac{V_1^2}{2g} = \frac{P_2}{\gamma} + Z_2 + \frac{V_2^2}{2g}$$

$$\frac{P_1}{\gamma} = 4 + \frac{20^2}{2g} - \frac{5^2}{2g} \quad \gamma = \rho g$$

$$P_1 = 226.318 \text{ kpa}$$



$$\frac{P_1}{\gamma} + Z_1 + \frac{V_1^2}{2g} = \frac{P_2}{\gamma} + Z_2 + \frac{V_2^2}{2g} + h_L$$

$$h_{lm} = K_m \times \frac{V^2}{2g}$$

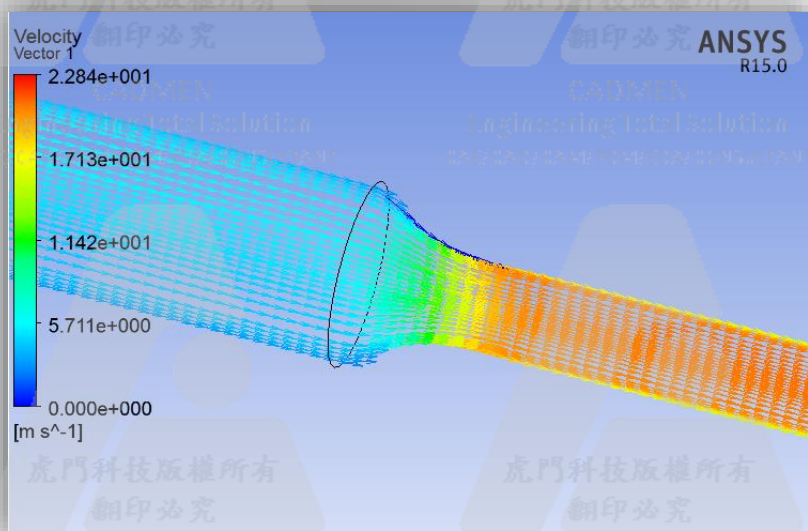
$$h_{lm} = 0.4 \times \frac{5^2}{2g} + 0.05 \times \frac{20^2}{2g} = 0.51 + 1.02 = 1.53$$

$$\frac{P_1}{\gamma} = 4 + \frac{20^2}{2g} - \frac{5^2}{2g} + 1.53 \quad \gamma = \rho g$$

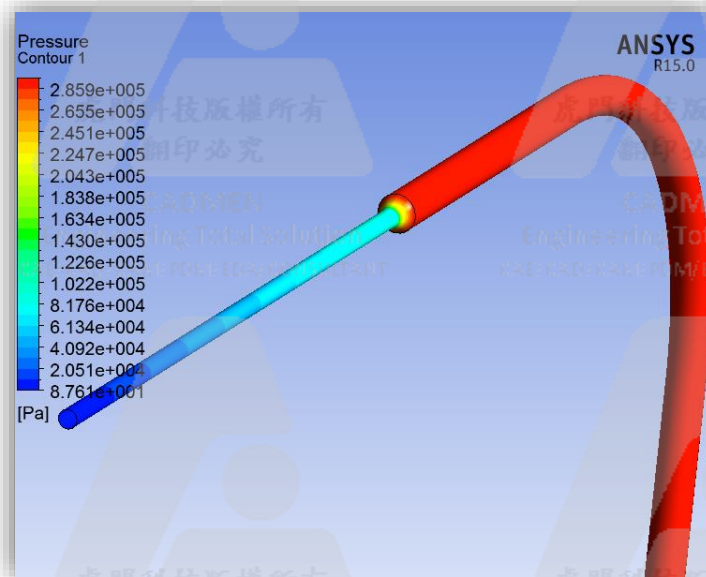
$$P_1 = 241.8 \text{ kpa}$$

- 水穩定地沿著直徑為 0.1m 的垂直圓管往上流，並由直徑 0.05m 的噴嘴放流至大氣壓中。
- 噴嘴出口處的流速必須為 20 m/sec。
- 試求在截面 ① 處所需之錶壓。

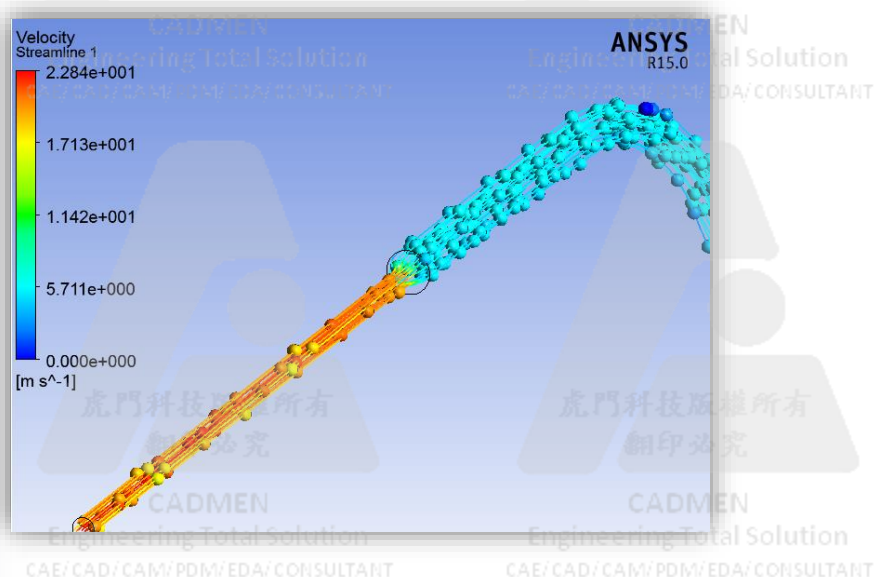
## 速度向量



## 壓力分佈



- 管入損失計算、均勻
- 流場、溫度場、壓力
- 顆粒、液體、氣體
- 材質選用探究
- 水槌現象、結構強度
- 侵蝕、空蝕

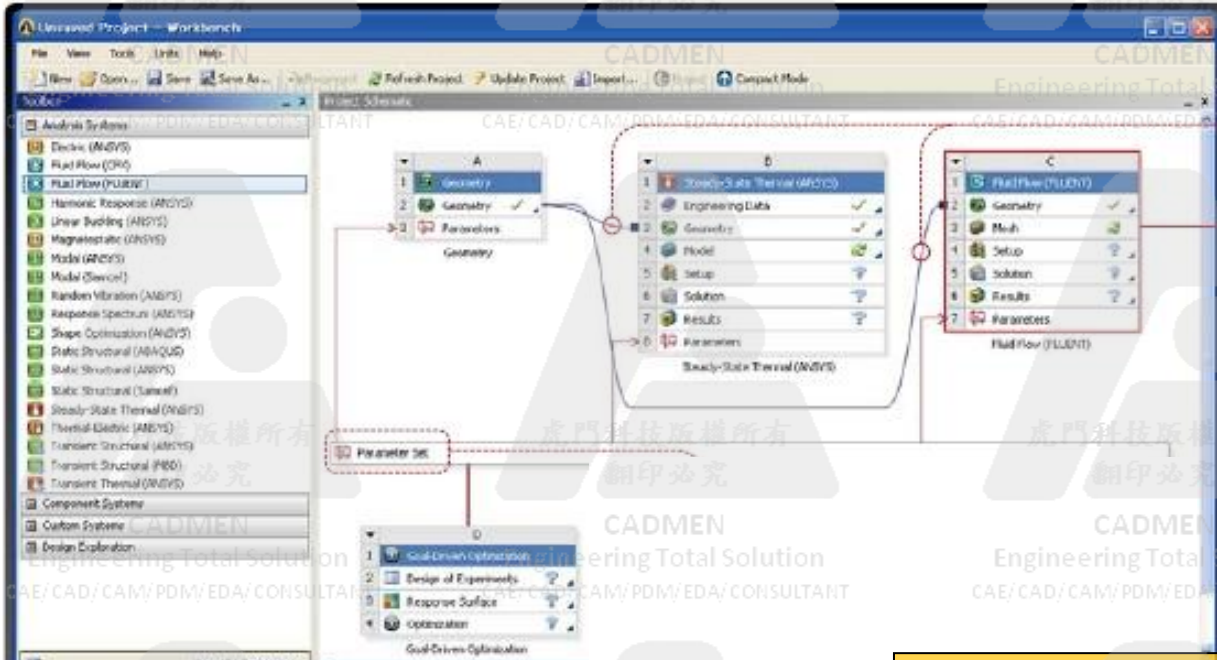




# About ANSYS Workbench

## CAE Platform

- 分析專案管理
- 重複應用
- 參數分析
- 最佳化使用
- 共通使用介面
- 耦合運算



**DesignModeler**  
**SCDM**  
**建構分析模型**

**ANSYS MESH**  
**網格建構**

**ANSYS FLUENT**  
**設定&計算**

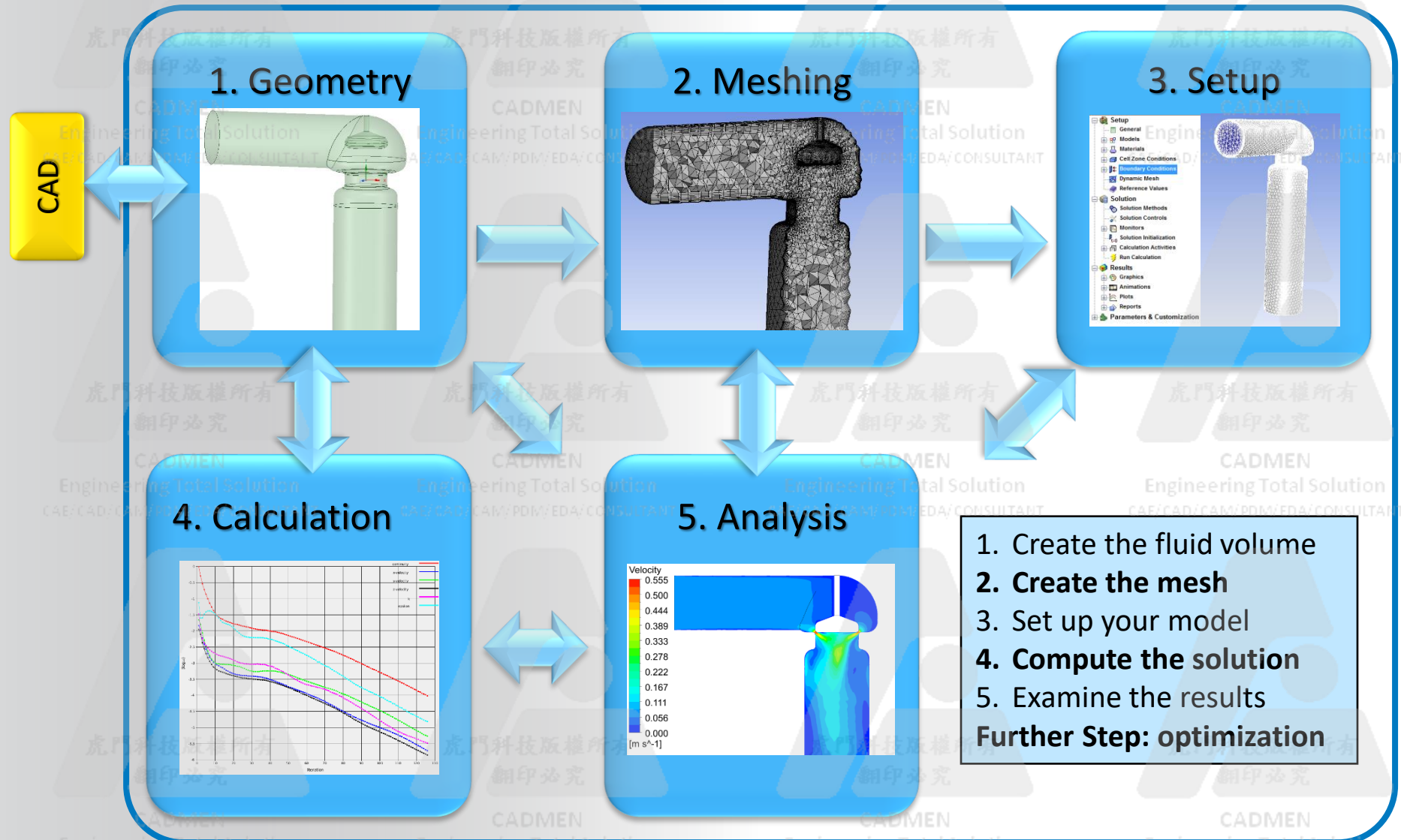
**CFD POST**  
**後處理可視化**



# The workflow



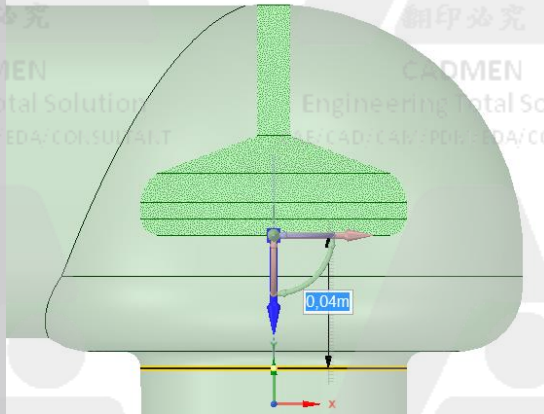
## ANSYS Workbench



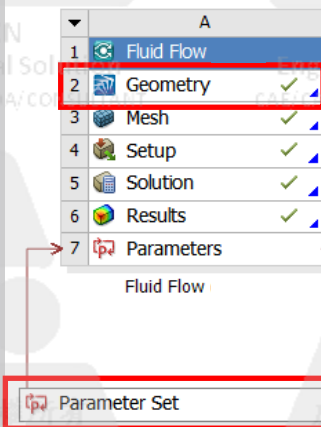
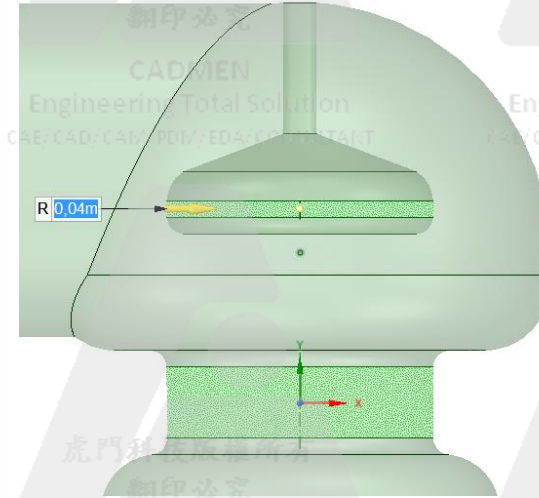
# Influence of valve position and radius



Valve position:  
0.5 cm – 5 cm



Valve radius:  
35 cm, 40 cm, 45 cm



| Name            | P3 - Ventilloeffnung | P5 - Ventildurchmesser | P6 - Durchlass | P2 - KvWert                    | P7 - deltaP |
|-----------------|----------------------|------------------------|----------------|--------------------------------|-------------|
| Units           |                      |                        |                | m <sup>3</sup> s <sup>-1</sup> | Pa          |
| DP 0            | 0,04                 | 0,04                   | 0,04           | 0,067404                       | 5,5125      |
| DP 1            | 0,05                 | 0,04                   | 0,04           | 0,069236                       | 5,2246      |
| DP 2            | 0,03                 | 0,04                   | 0,04           | 0,064429                       | 6,0333      |
| DP 3            | 0,01                 | 0,04                   | 0,04           | 0,029555                       | 28,672      |
| DP 4            | 0,02                 | 0,04                   | 0,04           | 0,054708                       | 8,3681      |
| DP 5            | 0,005                | 0,04                   | 0,04           | 0,015547                       | 103,62      |
| DP 6            | 0,05                 | 0,035                  | 0,035          | 0,055103                       | 8,2486      |
| DP 7            | 0,05                 | 0,045                  | 0,045          | 0,084377                       | 3,5178      |
| DP 8            | 0,04                 | 0,045                  | 0,045          | 0,081305                       | 3,7887      |
| DP 9            | 0,03                 | 0,045                  | 0,045          | 0,07645                        | 4,2852      |
| DP 10           | 0,02                 | 0,045                  | 0,045          | 0,063065                       | 6,2972      |
| DP 11           | 0,01                 | 0,045                  | 0,045          | 0,033727                       | 22,017      |
| DP 12           | 0,005                | 0,045                  | 0,045          | 0,018157                       | 75,972      |
| DP 13           | 0,005                | 0,035                  | 0,035          | 0,013959                       | 128,54      |
| DP 14           | 0,01                 | 0,035                  | 0,035          | 0,026126                       | 36,693      |
| DP 15           | 0,02                 | 0,035                  | 0,035          | 0,04621                        | 11,729      |
| DP 16           | 0,03                 | 0,035                  | 0,035          | 0,052499                       | 9,0871      |
| DP 17 (Current) | 0,04                 | 0,035                  | 0,035          | 0,05383                        | 8,6432      |

# Analysis – examine the results

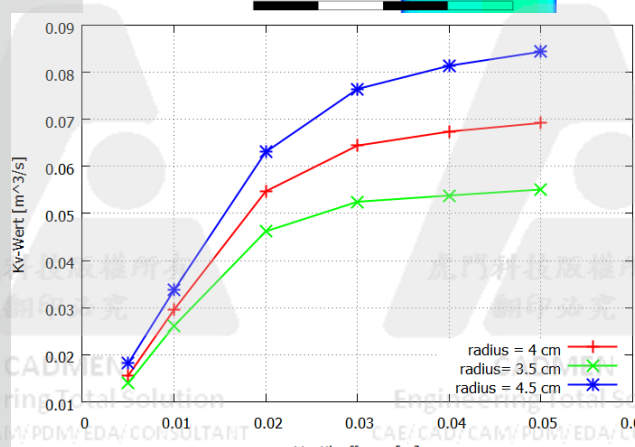
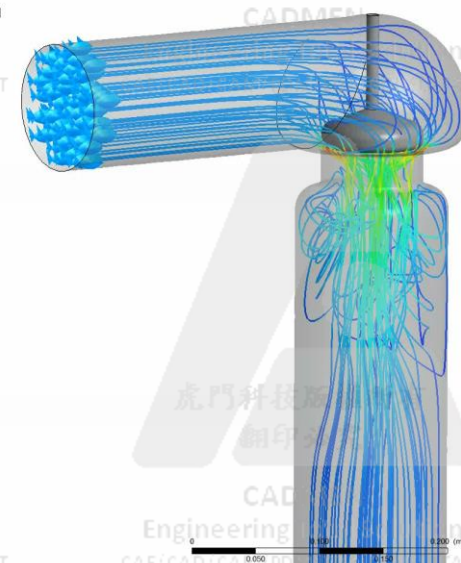
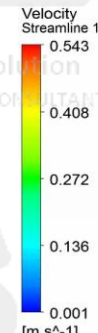
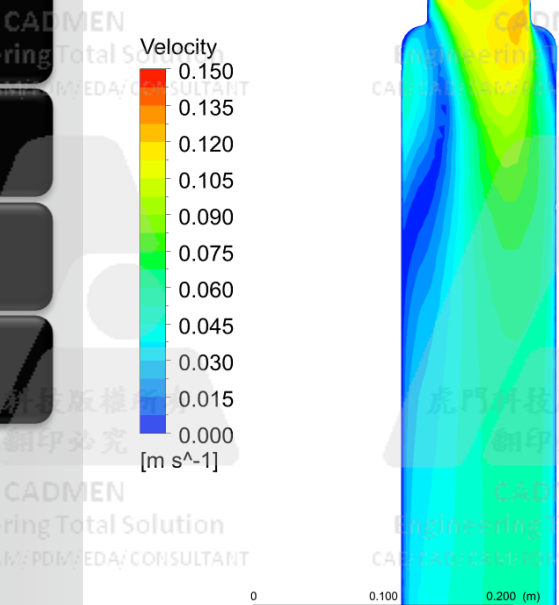
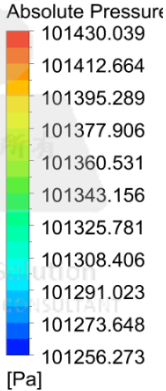
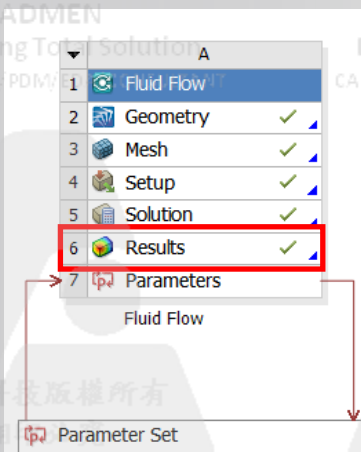
Locations & contours

Vectors & streamlines

Pictures & animations

Scripting & macros

Reports



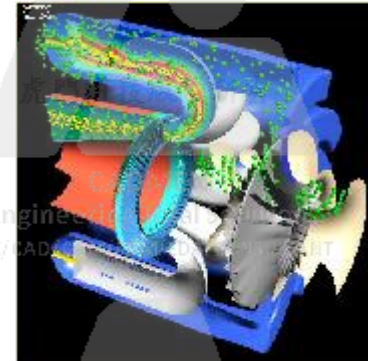
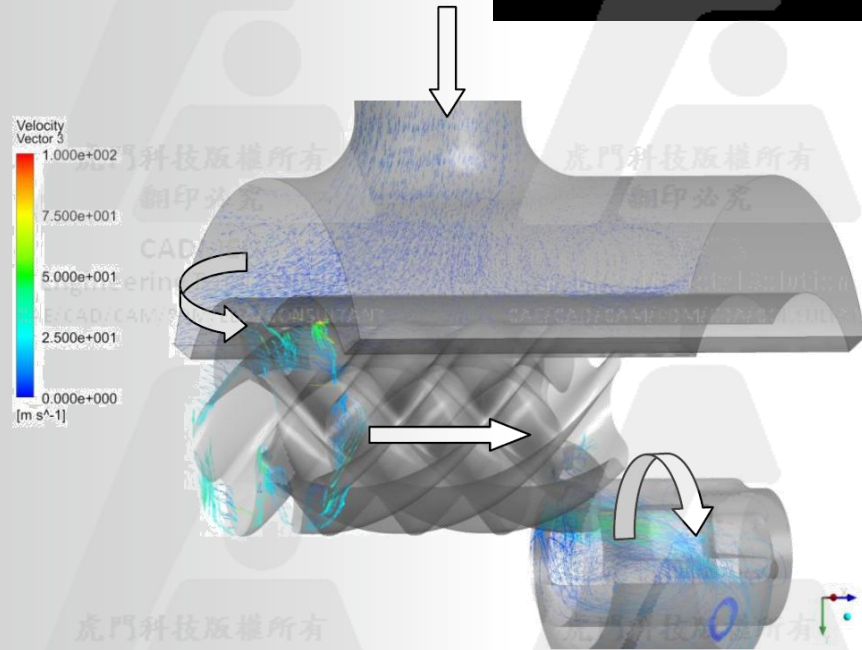
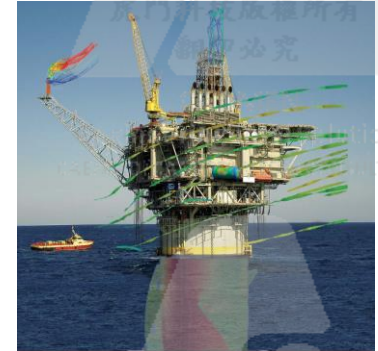
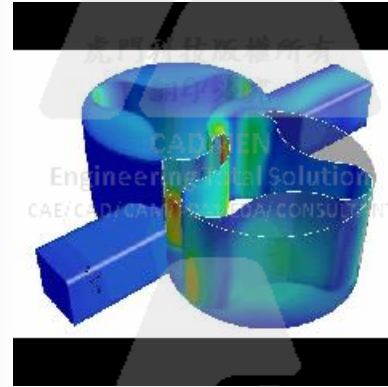
ANSYS CFD (FLUENT、CFX) 為功能強大、模組廣泛的旗艦型 CFD 產品

### 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

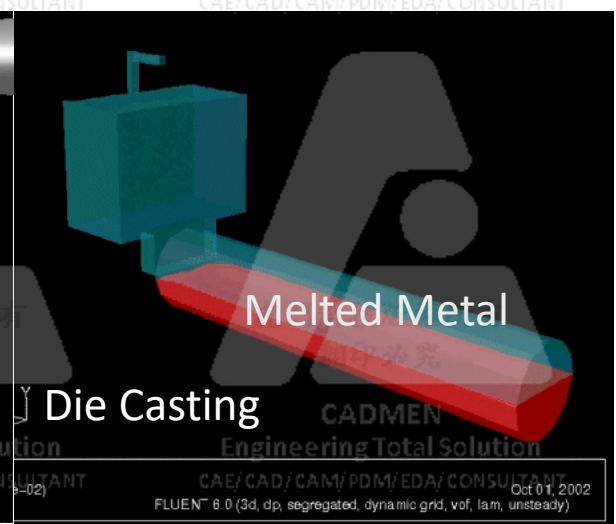
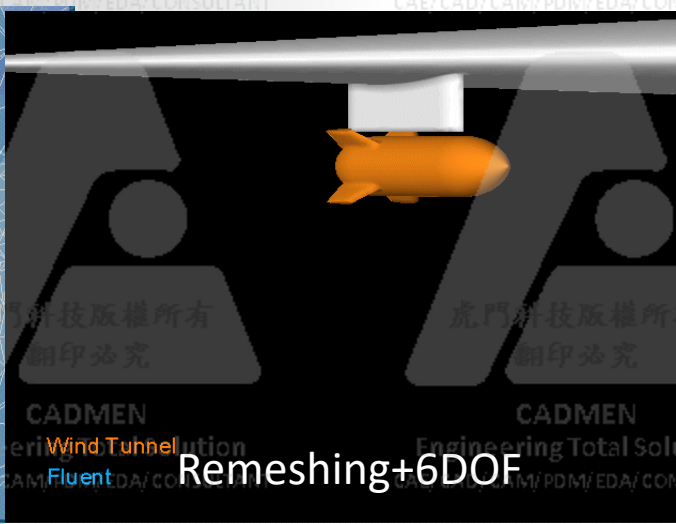
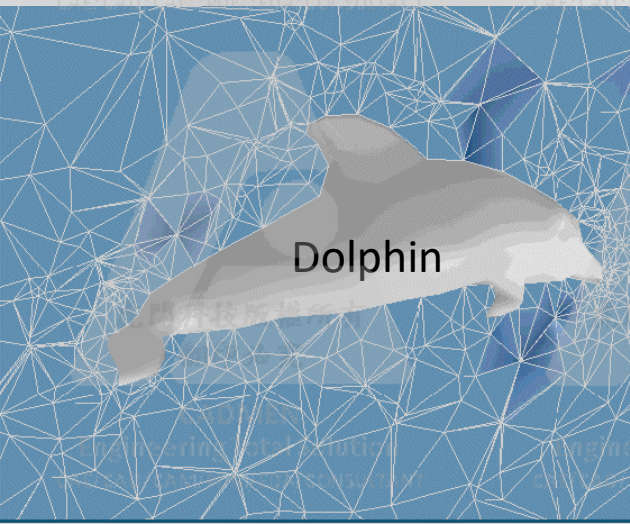
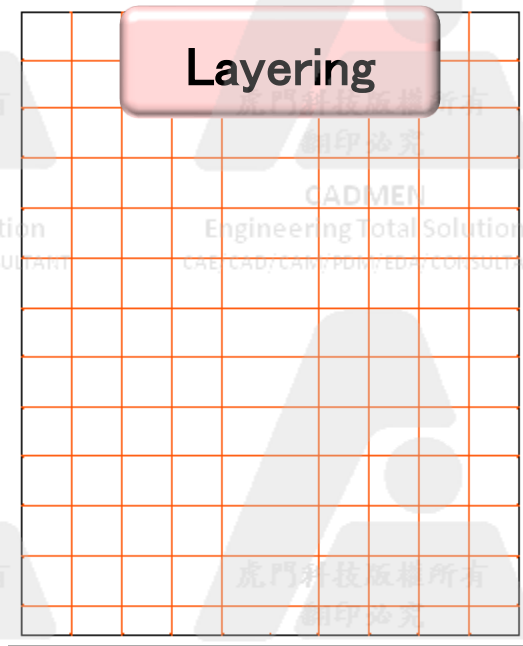
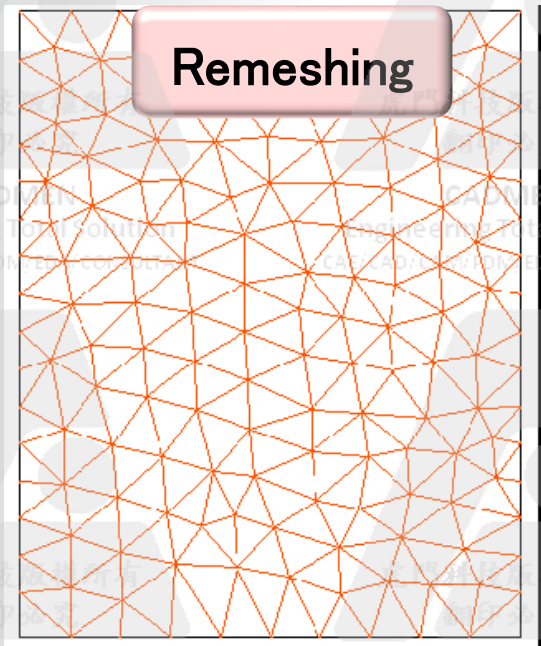
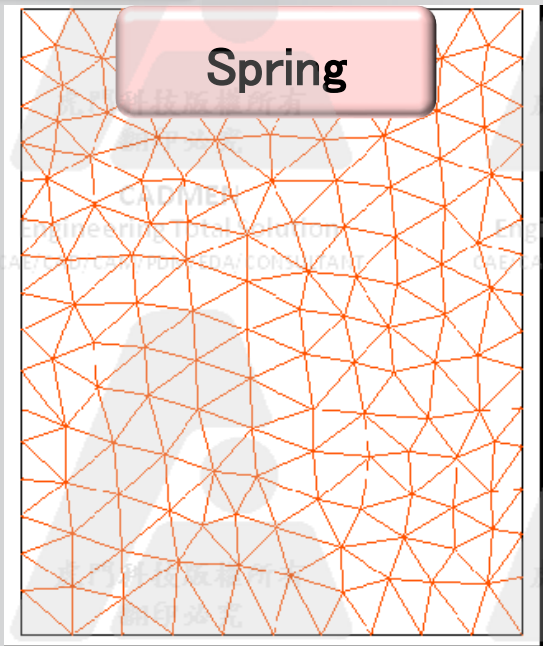
- 磁流模組 (MHD)
- 燃料電池模組
- 鋰離子電池模組
- 流固耦合 (FSI)
- 最佳化分析





# 移動/變形網格(Dynamic Mesh)

ANSYS®



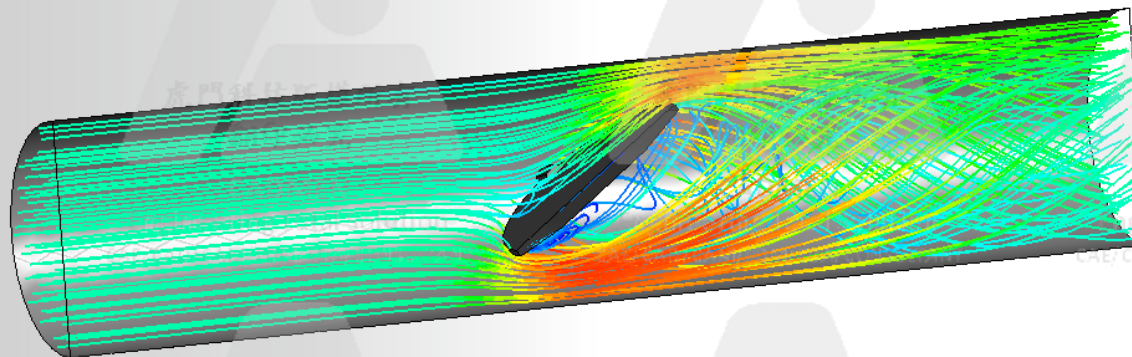
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

Remeshing+6DOF

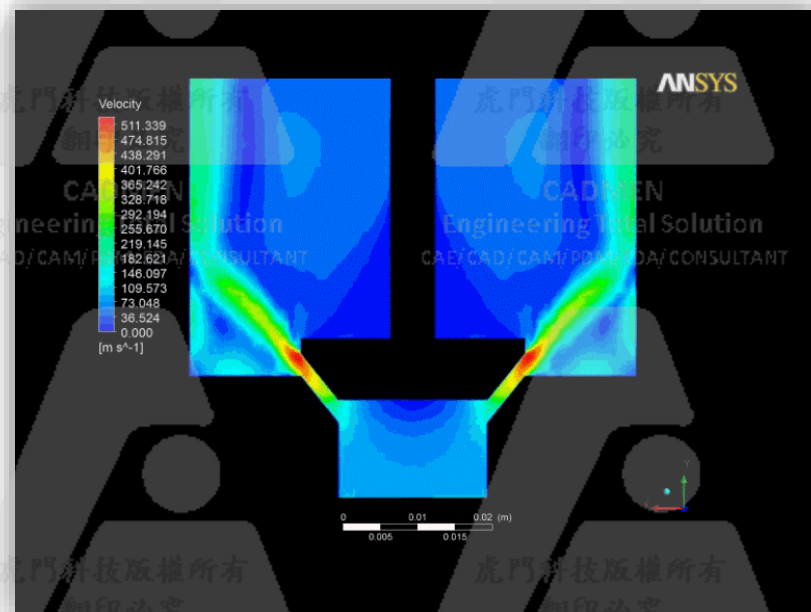
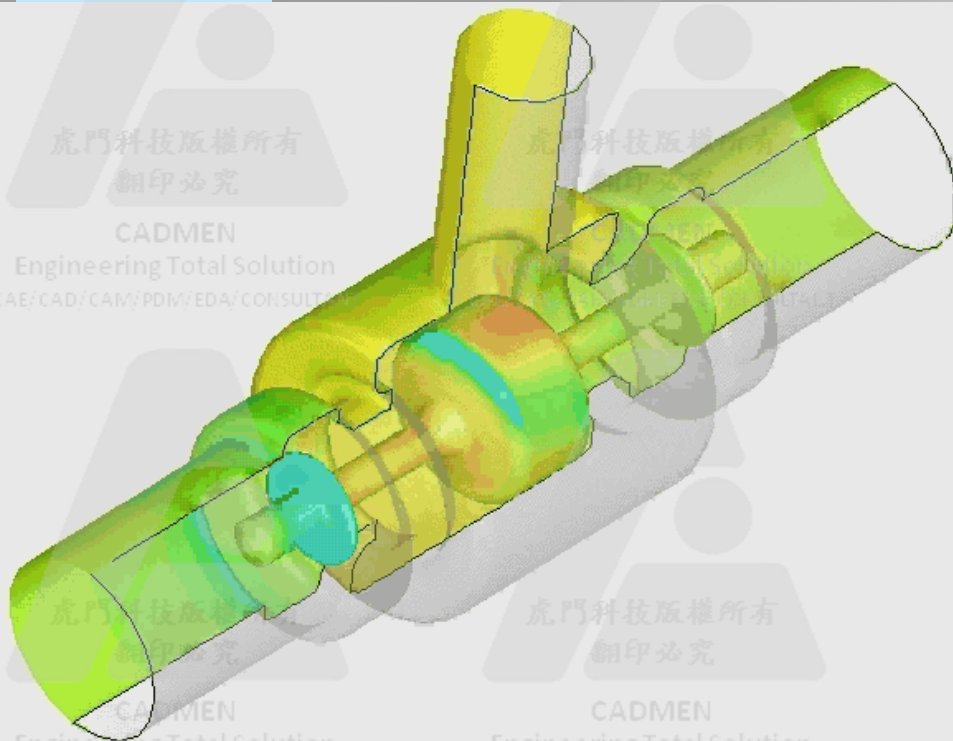
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT  
FLUENT 6.0 (3d, dp, segregated, dynamic grid, vof, lam, unsteady)  
Oct 01, 2002

# Valves 動網格案例

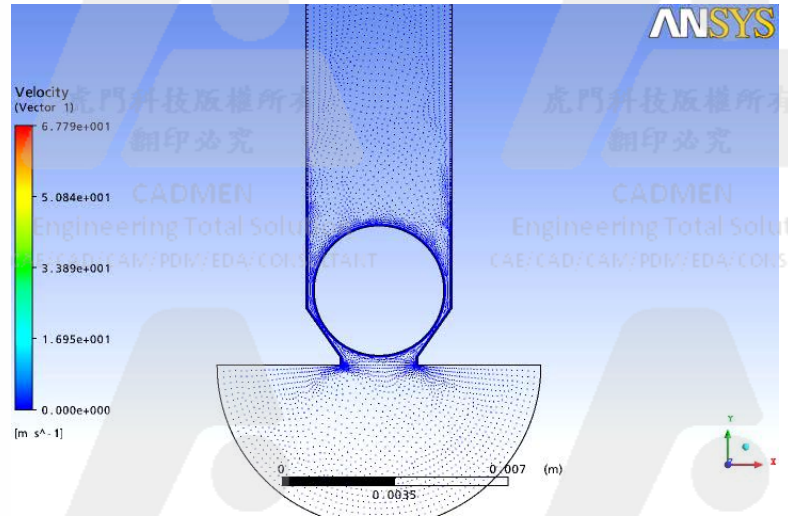
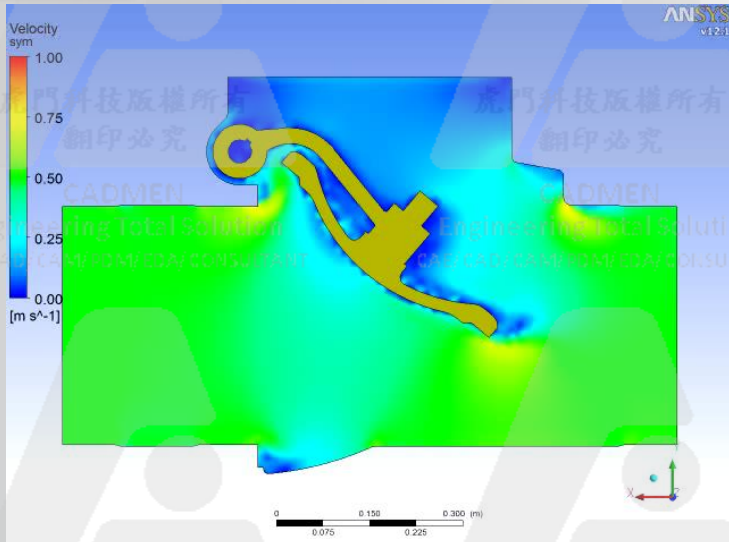


## 動態網格

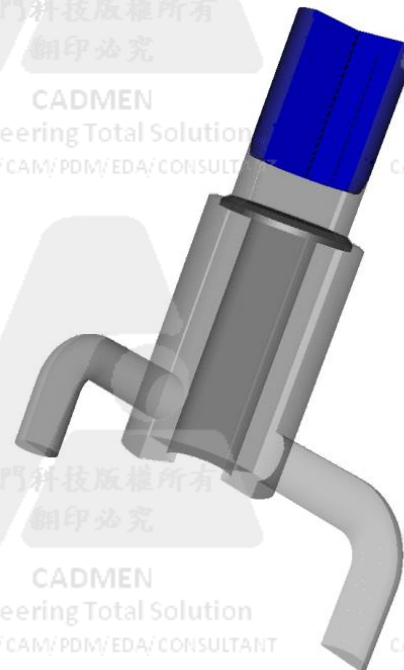
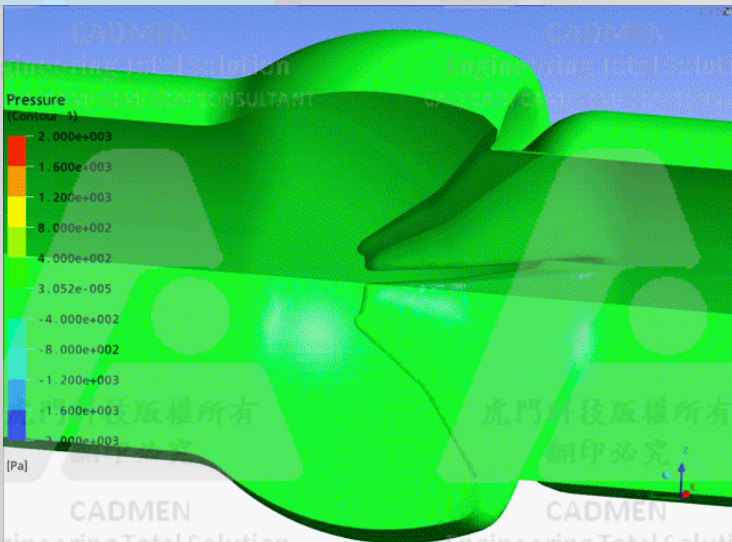


# 各類閘門案例應用

## DOF



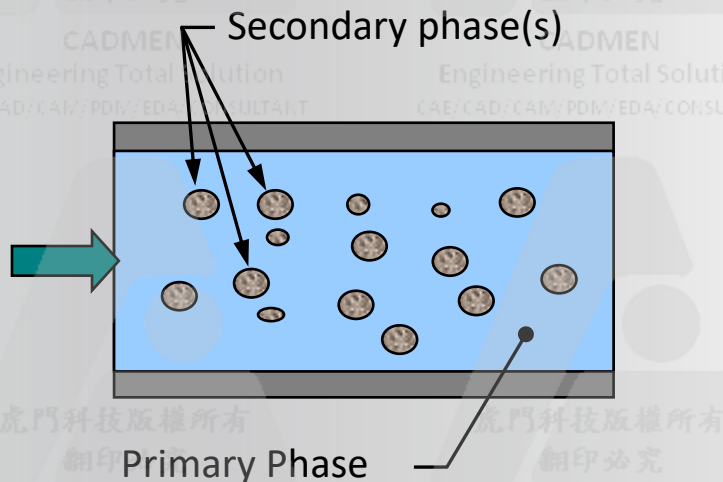
## Two Way FSI



# 多相流 Multiphase Flows

The fluid system is defined by a primary and multiple secondary phases.

- **One of the phases** is considered continuous (primary)
- The others (secondary) are considered to be dispersed within the continuous phase.
- (Note that for *free-surface flows, using the Volume of Fluid model (VOF)*, a distinct interface is defined between the phases and both could be considered continuous)



## Filling



# Multiphase Flow Regimes

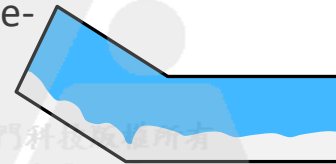
- **Bubbly flow** – Discrete gaseous bubbles in a continuous fluid, e.g. absorbers, evaporators, sparging devices.
- **Droplet flow** – Discrete fluid droplets in a continuous gas, e.g. atomizers, combustors
- **Slug flow** – Large bubbles in a continuous liquid
- **Stratified / free-surface flow** – Immiscible fluids separated by a clearly defined interface, e.g. free-surface flow



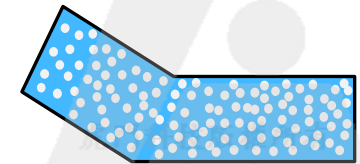
Slug Flow



Bubbly, Droplet, or Particle-Laden Flow

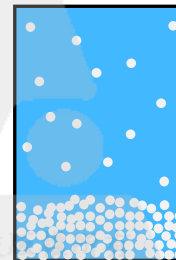


Stratified / Free-Surface Flow

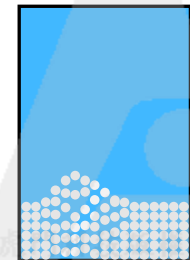


Pneumatic Transport, Hydrotransport, or Slurry Flow

- **Particle-laden flow** – Discrete solid particles in a continuous fluid, e.g. cyclone separators, air classifiers, dust collectors, dust-laden environmental flows
- **Fluidized beds** – Fluidized bed reactors



Sedimentation



Fluidized Bed

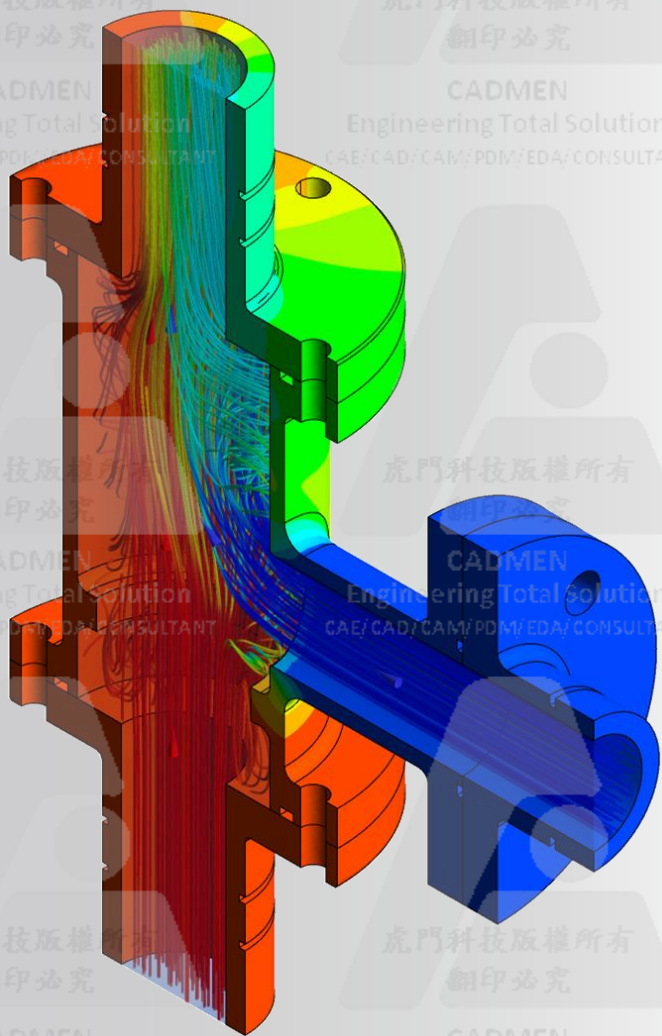
- **Slurry flow** – Particle flow in liquids, solids suspension, sedimentation, and hydro-transport

Gas/Liquid  
Liquid/Liquid

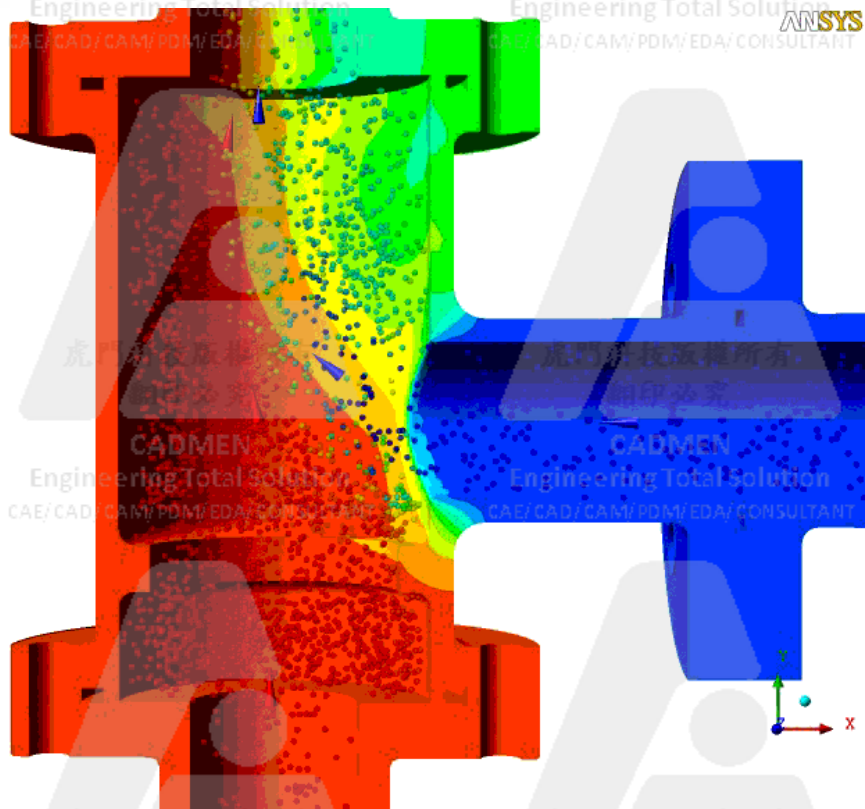
Gas / Solid

Liquid / Solid

# 管路流場分析



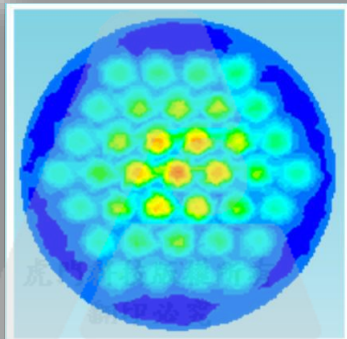
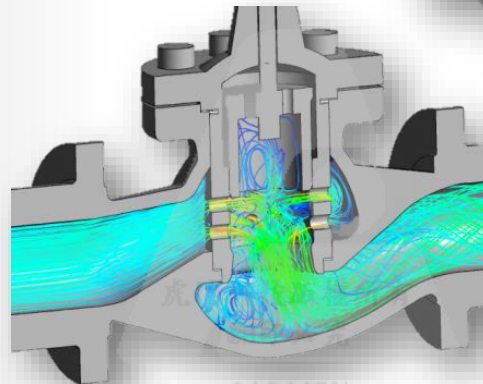
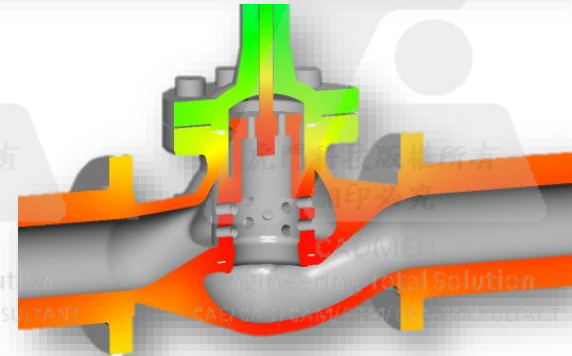
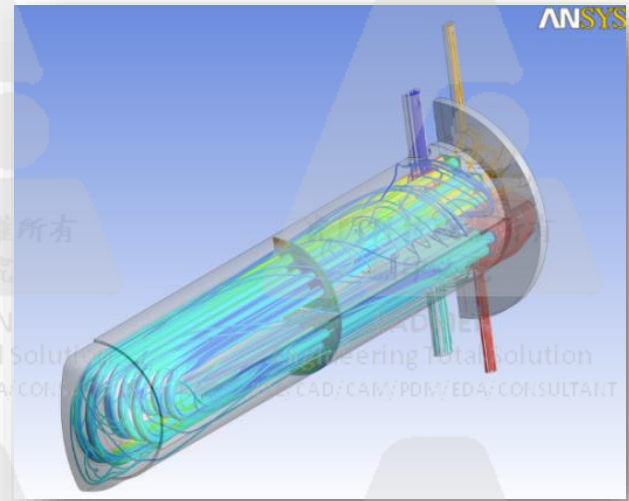
## 顆粒流動分析



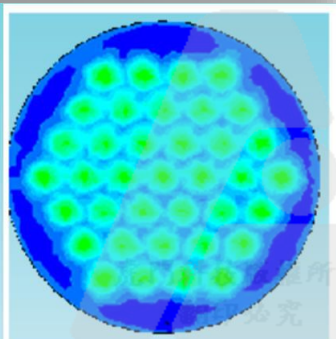
# Heat Transfer

## ANSYS Advantages

- A wide array of models for any heat transfer applications– convection, natural convection, conduction, radiation, solar radiation, conjugate heat transfer
- A large fluid library (liquid, gas, mixtures, etc.)
- The fastest solution on the market
- The most validated solution on the market
- A world-class team of technical support experts



Maldistributed



Improved

Courtesy CADFEM Russia

# 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.



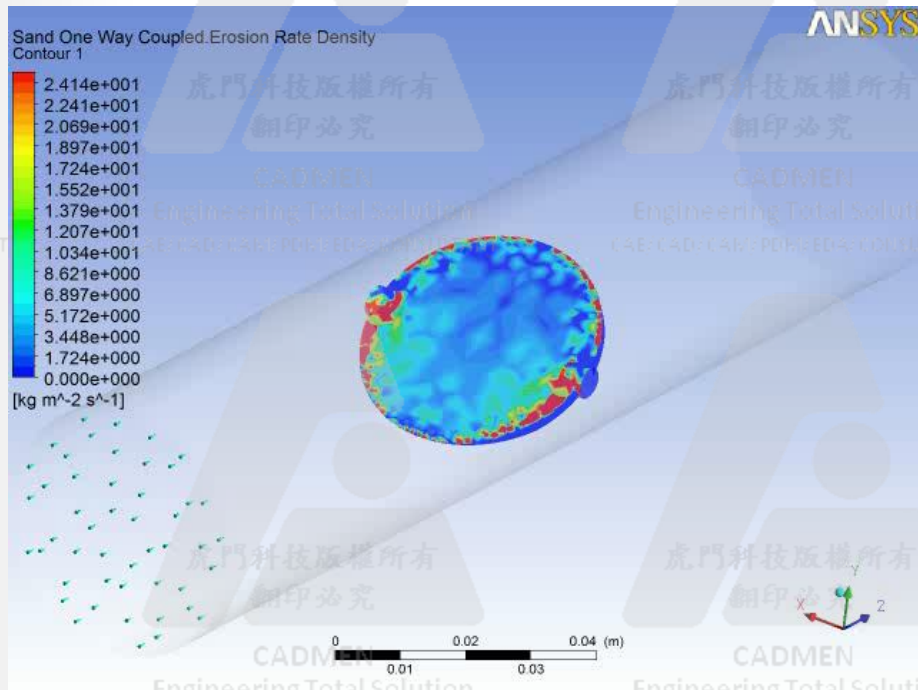
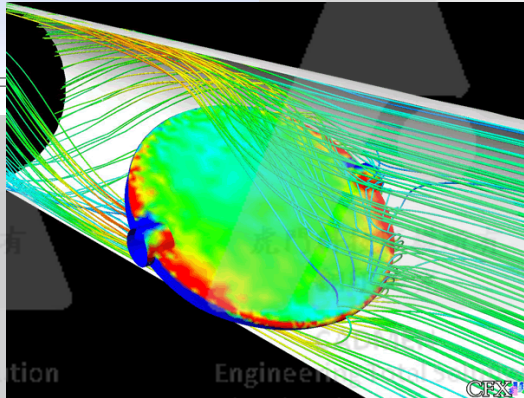
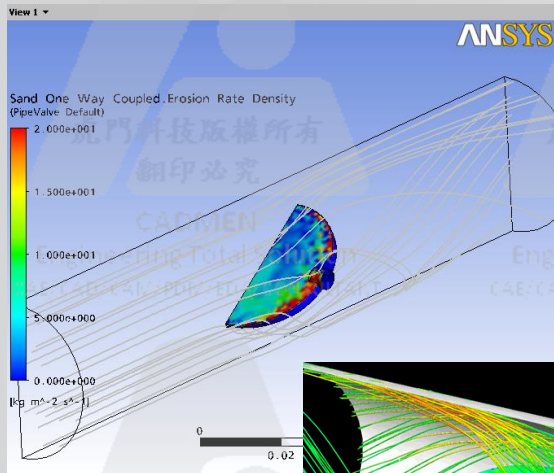
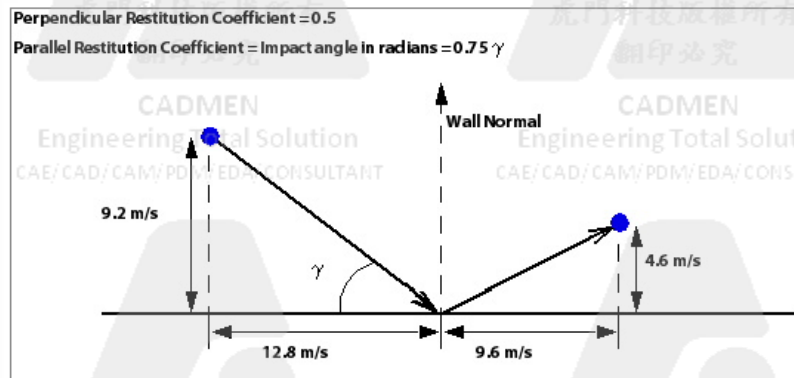
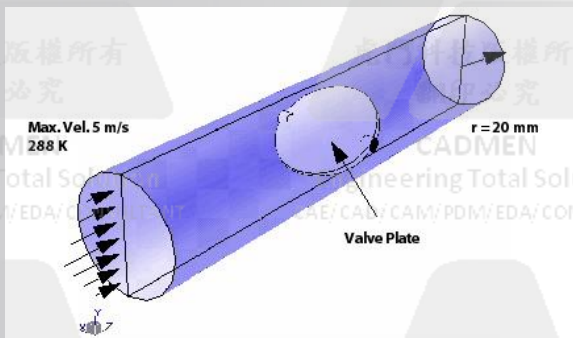
*Internal flow of natural gas containing **sand particles**.*

*Selected particle trajectories are colored in grey*

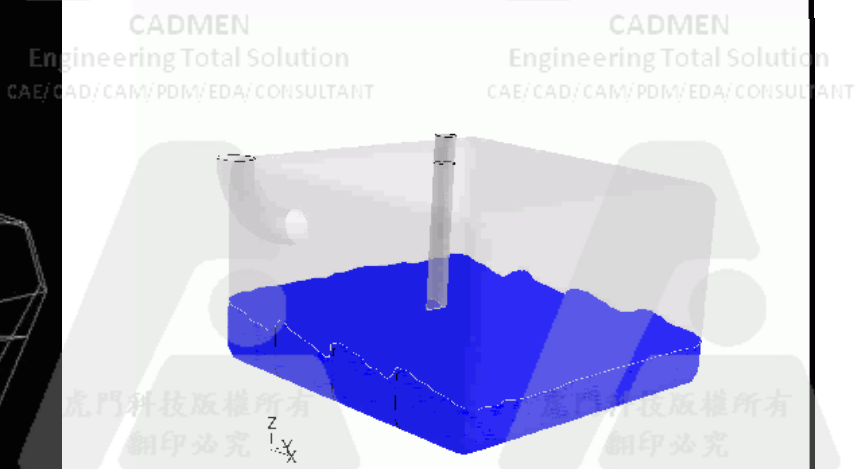
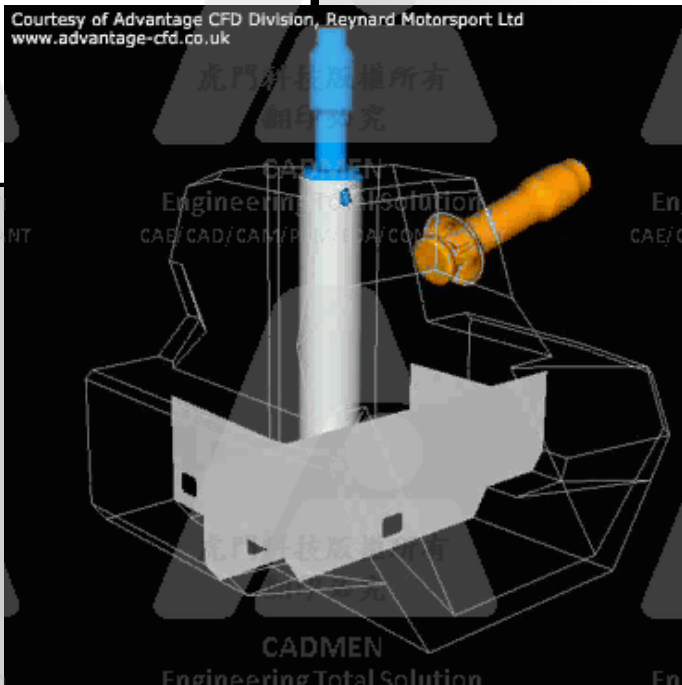
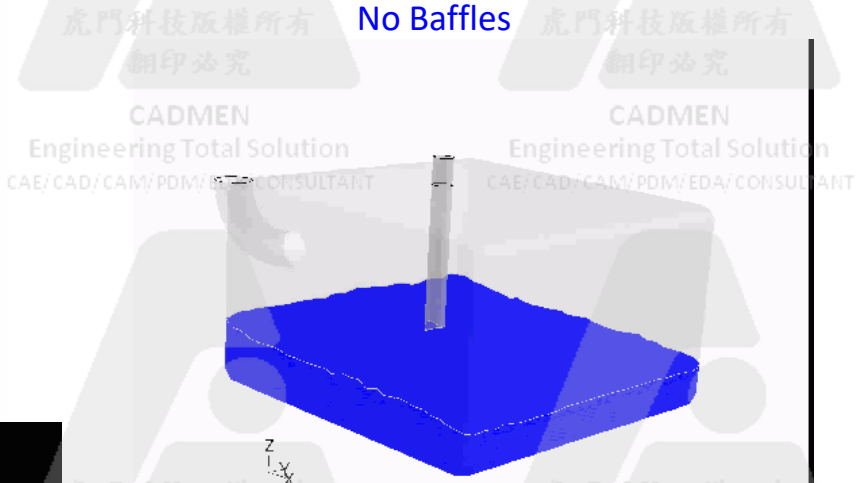
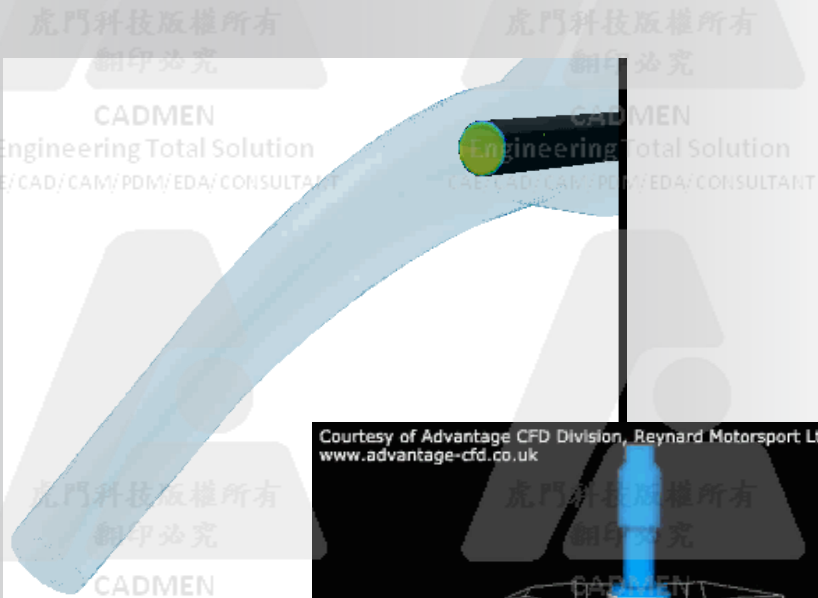
*The **erosive** wear hotspots on the piping is colored out in red.*



# 泥沙冲刷 - 考虑颗粒侵蚀效应



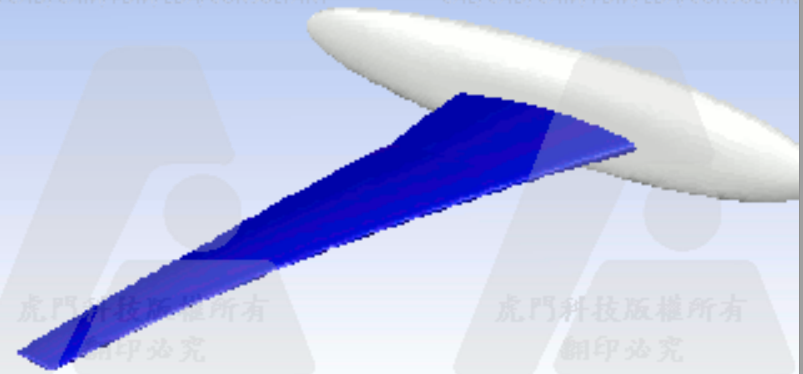
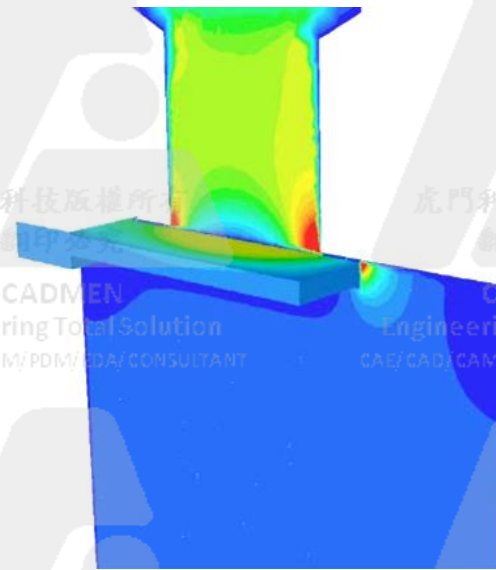
# Filling, Tank Sloshing



With Baffles

## Important in many industries

- **Construction**
  - 避免共振
- **Energy**
  - Oil & Gas: 結構面臨波動的流動
  - Power Plants: 振動棒，阻尼
- **Aerospace**
  - 航空彈性，顫振
- **Automotive**
  - 簧片閥，空氣過濾器
- **Process Equipment**
  - 管道，流量計
- **Etc.**



# Impact of Not Managing Flow Induced Motion

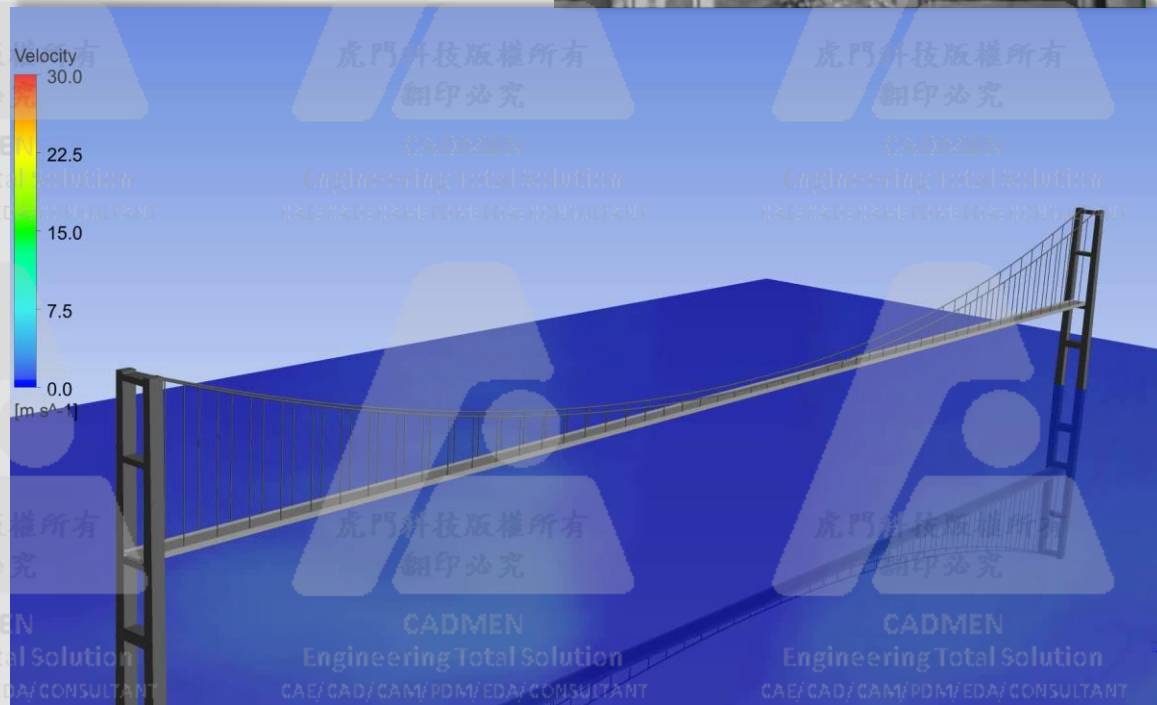
Tacoma Narrows Bridge, which collapsed 1940

## Efficiency

- Optimal performance may be inhibited by vibration and flow induced motion
- Products must perform to consumer expectations

## Safety and Reliability

- Failure due to resonance effects
- Wrong prediction of life length and maintenance needs



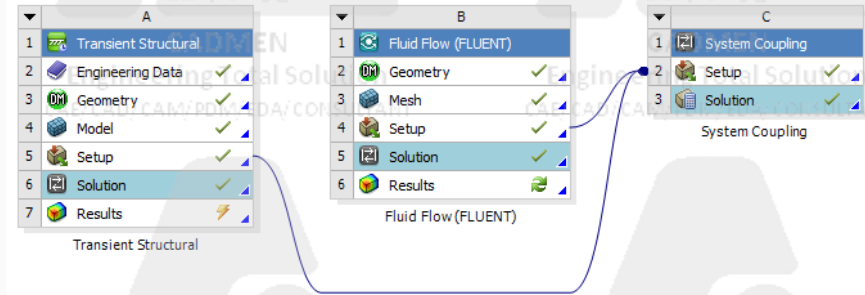
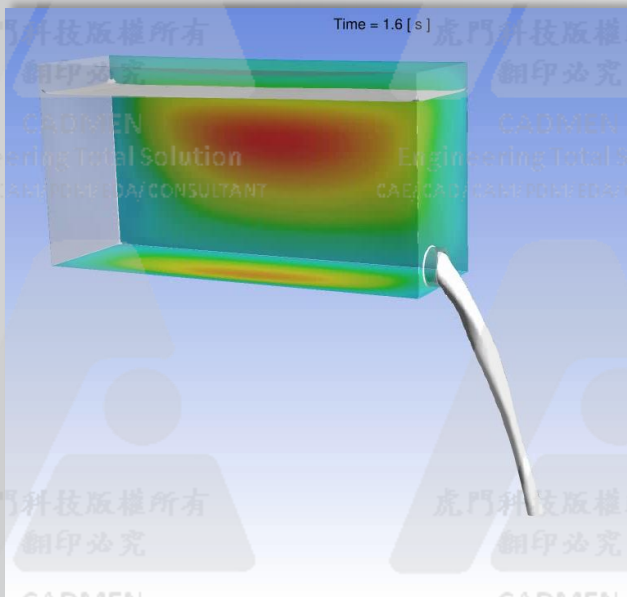
# FSI Workflows

## 2-way Load Transfers

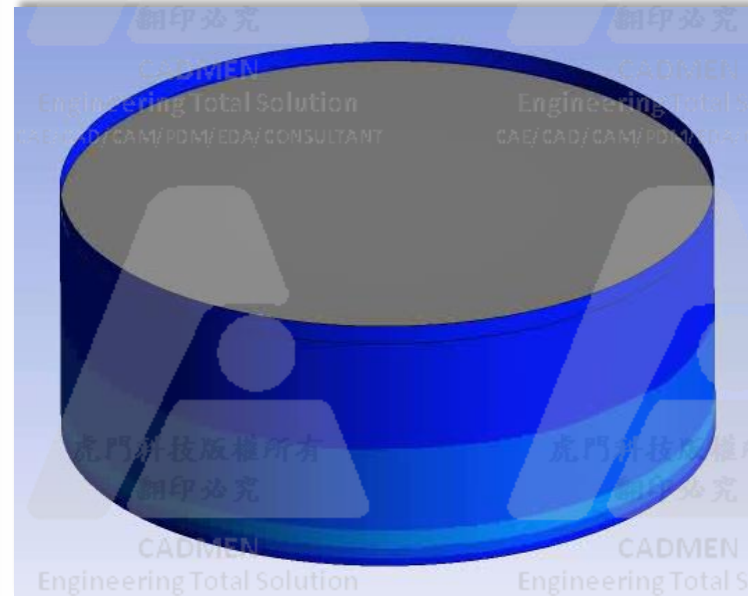
### 2-way Force/Displacement

- Available for both Fluent and CFX
  - Fluent through System Coupling (SC)
  - CFX through MFX or SC (beta in R16)

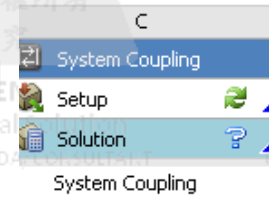
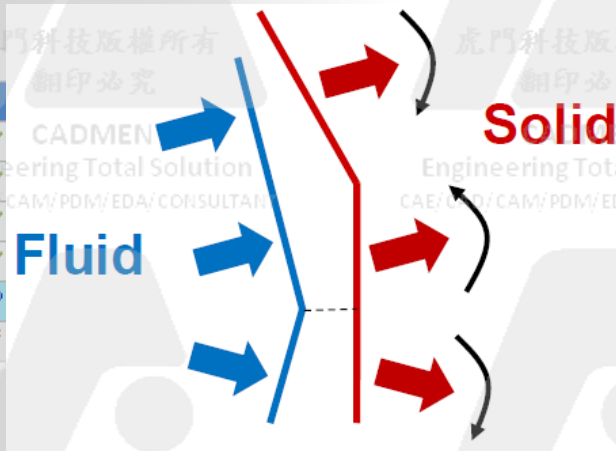
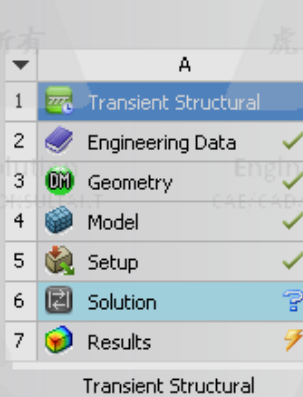
*Fluid sloshing effects in a milk package (displaying deformations and water level)*



*Earthquake structural and sloshing response for a liquid storage tank (displaying deformations and water level)*

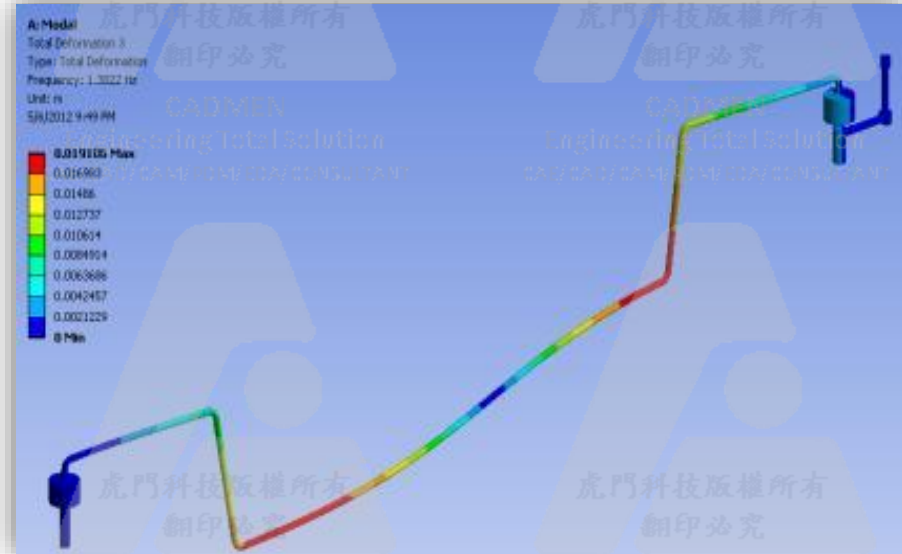
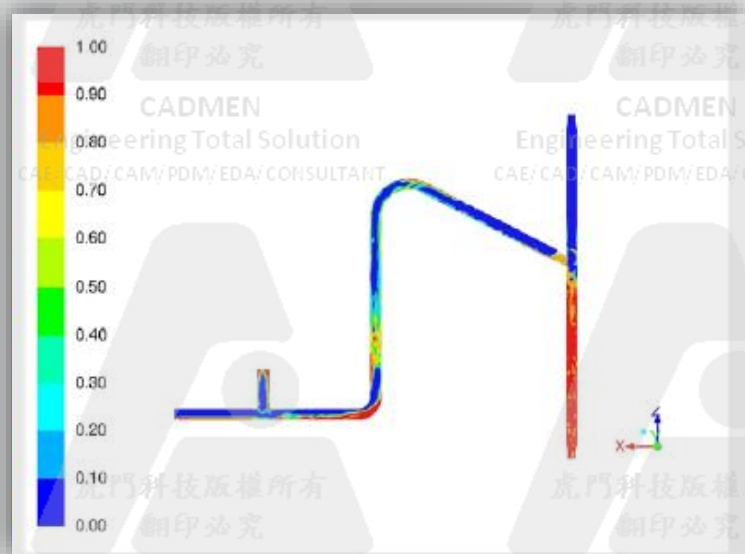


# 複雜的流動條件 -> 結構強度設計



多相流動

結構變形量



虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

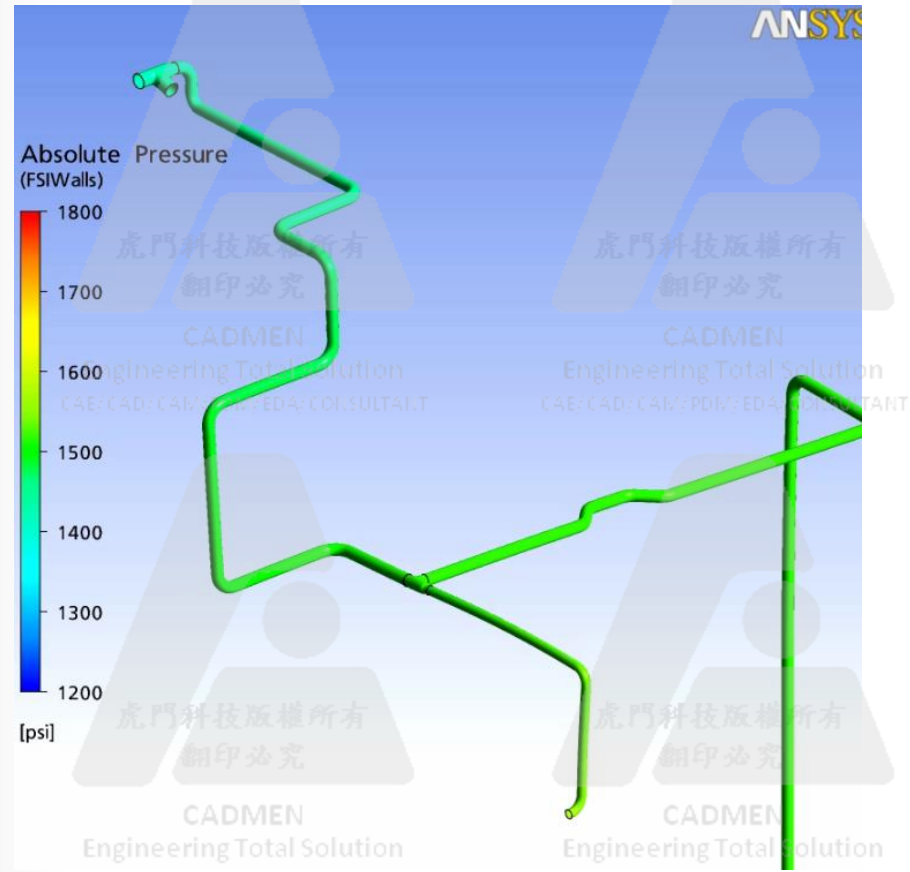
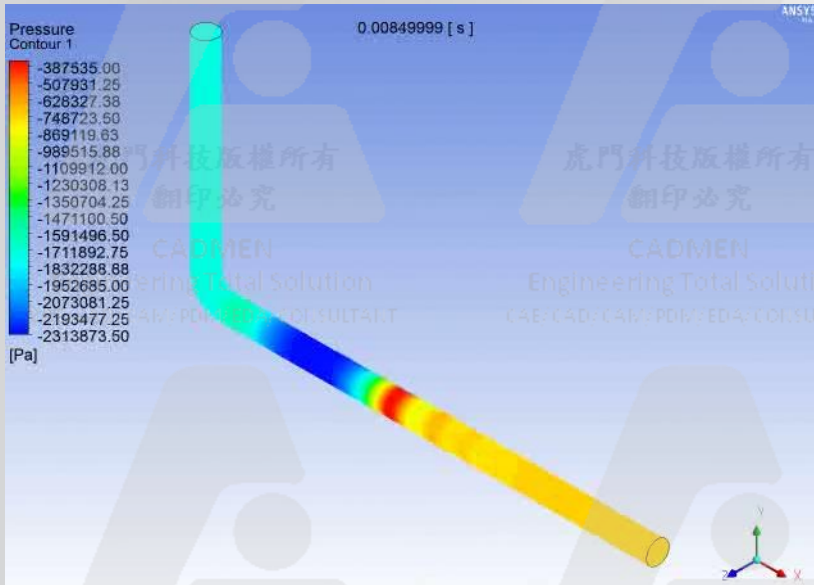
CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

管路出口突然關閉  
所造成之水槌現象

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



虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

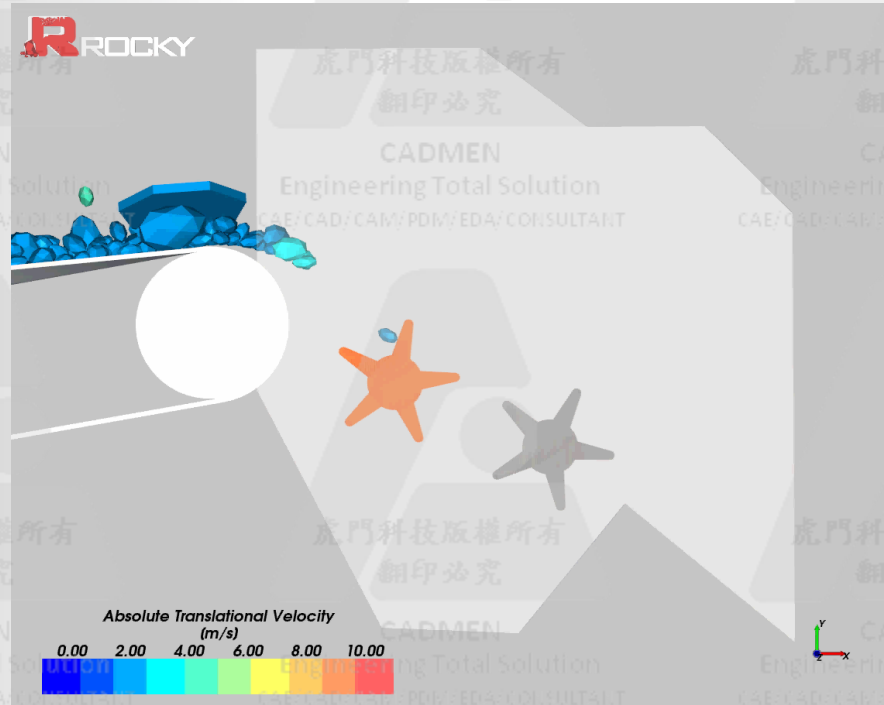
CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

# ROCKY-DEM粒子輸送與ANSYS Fluent耦合應用

唯一使用精確粒子形態和進階破損與碎片模型的商用碼  
與Fluent耦合

內建粒子、自訂粒子(STL)



One-way coupling example:  
waste separator



虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

# 流固耦合

## 工業用閥門流場與結構變形分析

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

# Valve travel effect on field flow and deformation in valve

**Project Name :**

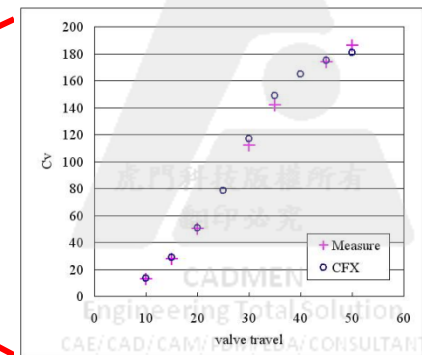
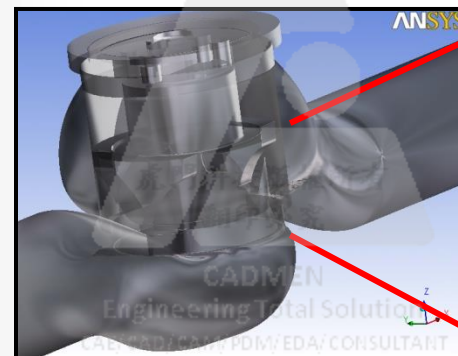
Flow Coefficient ( $C_v$ ) Study and FSI Analysis of an Industry Valve

**Problem description :**

- Evaluate the flow coefficient ( $C_v$ ) as the function of valve travel distance given a constant pressure decline.
- Verify the strength design of valve parts and check if any undesirable deformation occurs.

**Analysis tools :**

- flow simulation software :
  - pre-processor : ANSYS DesignModeler + CFX Mesh (1.6M tetra-grid)
  - solver : ANSYS CFX solver (Finite Volume CFD solver)
  - post-process : ANSYS CFX-post
- valve parts strength calculation : ANSYS (Finite Element Method)

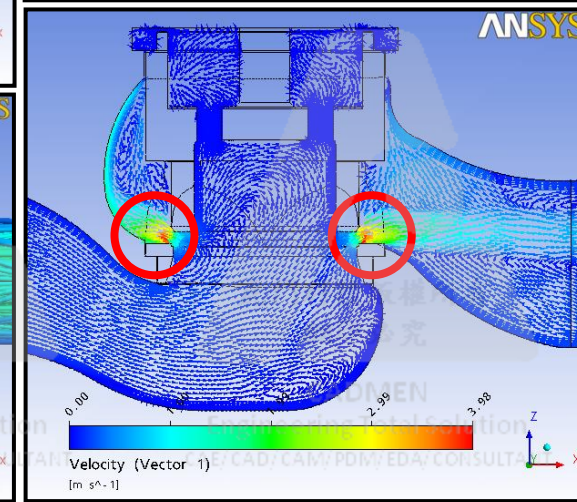
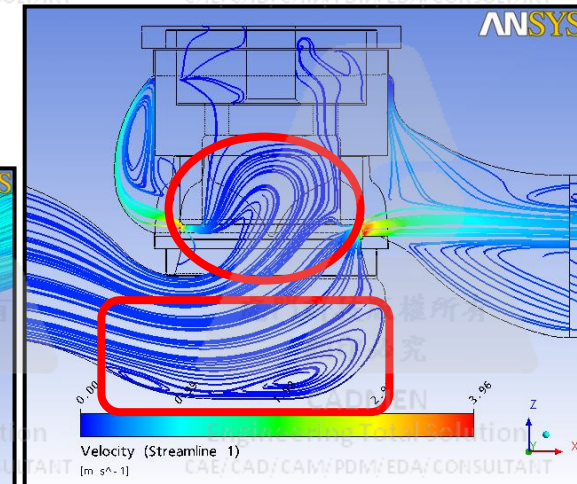
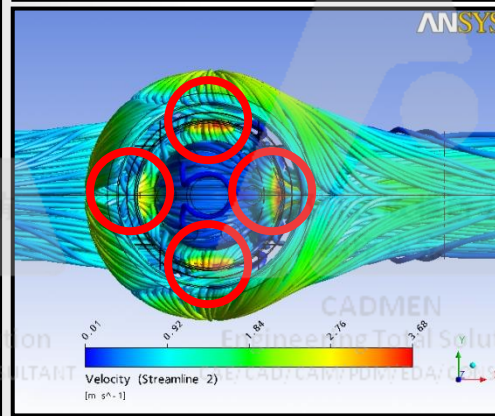
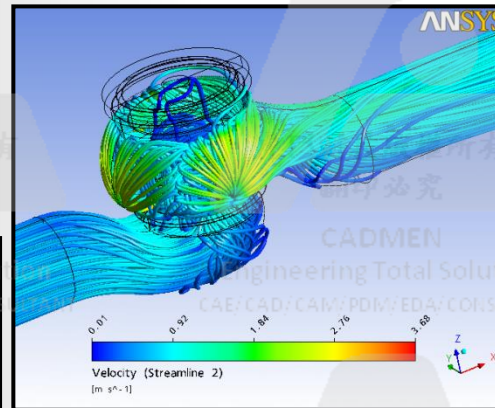
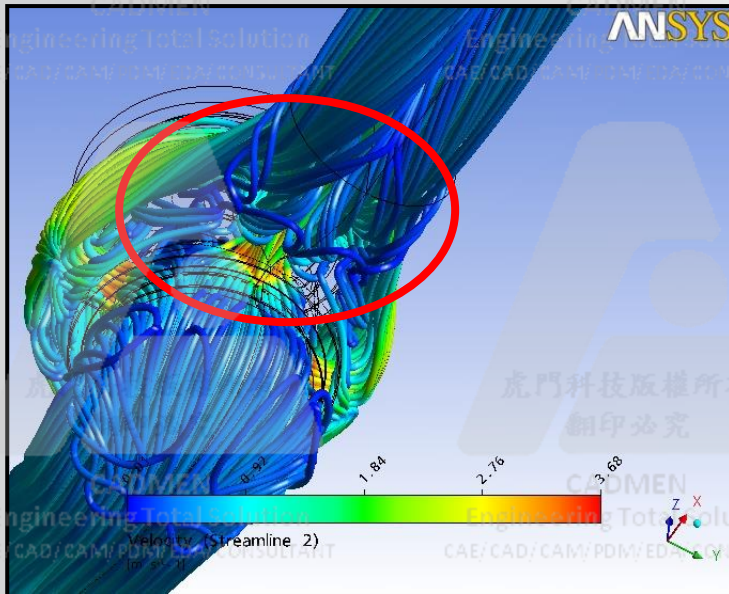
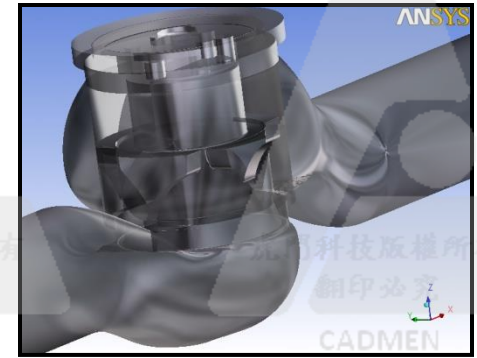


# 工業用閥門流場與結構變形分析

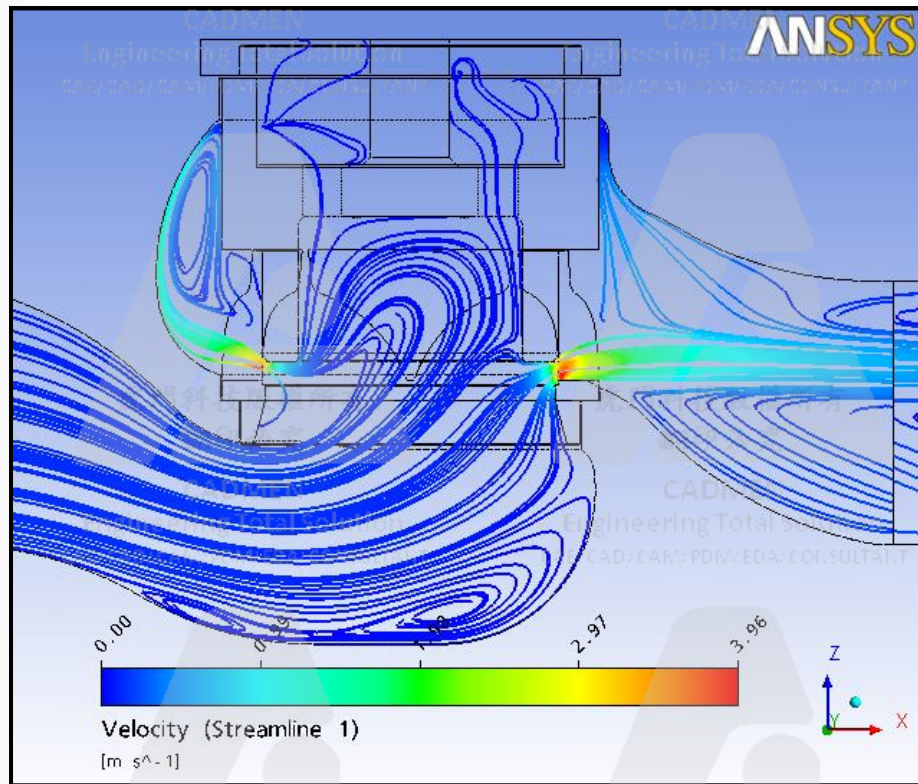
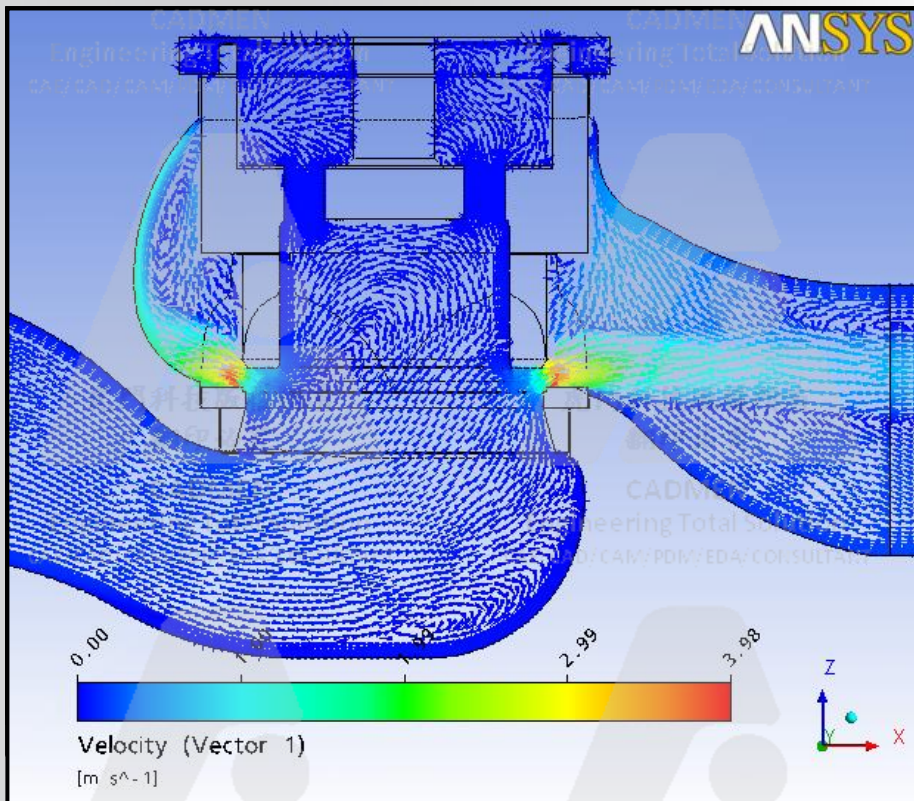
ANSYS

## Flow Simulation Result

- Strong swirling flow occurs in the regions near
  1. bottom of the valve body
  2. top of the valve gate part
  3. outlet of the valve
- High speed flow occurs near the gate opening. Cavitation risk exist in these region.



50mm 40mm 30mm 20mm 10mm

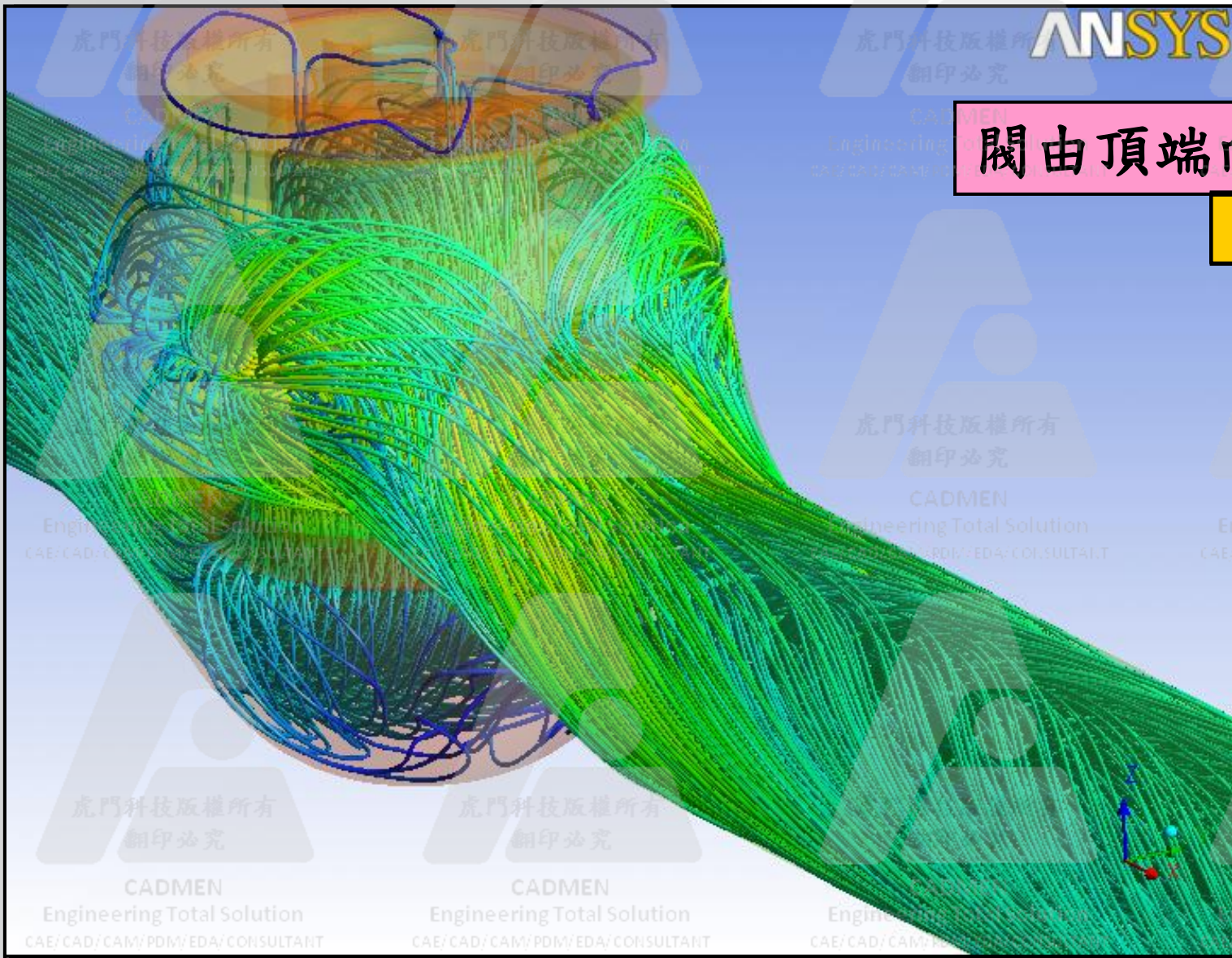


虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

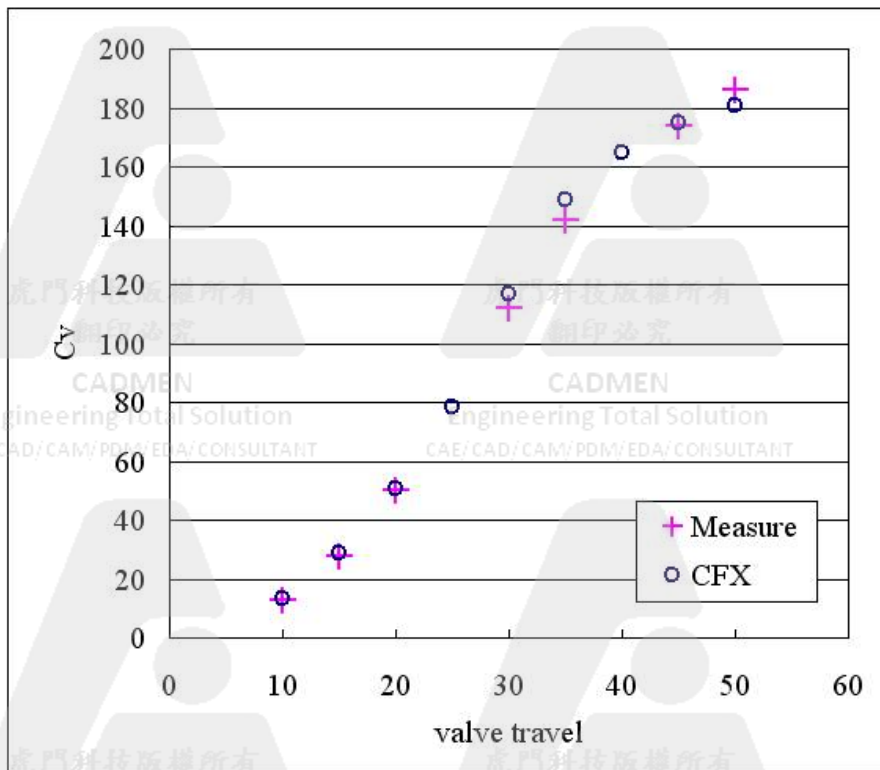


閥由頂端向下移動

Travel : 40 mm

# Valve travel effect on field flow and deformation in valve

## Flow Coefficient Study Result



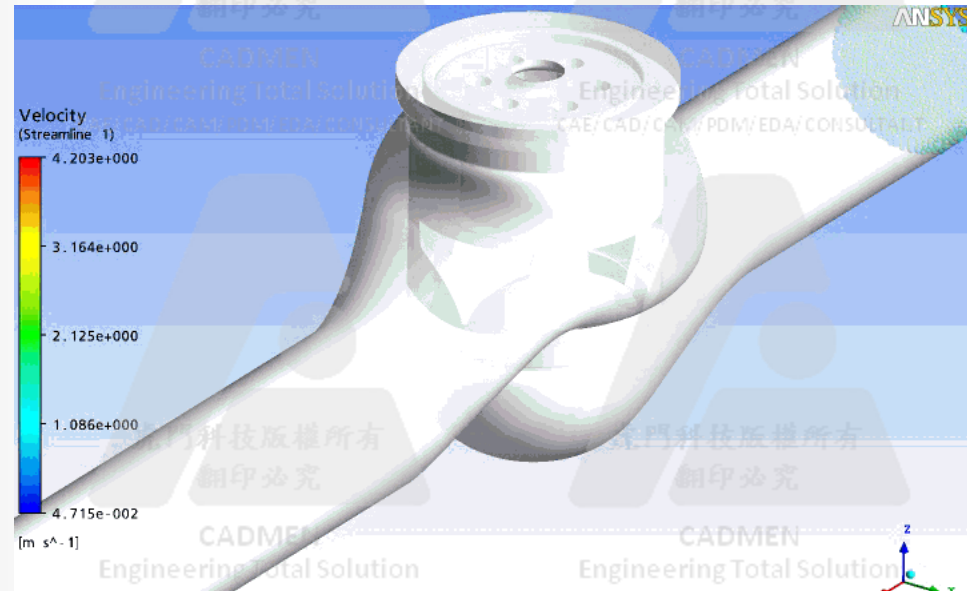
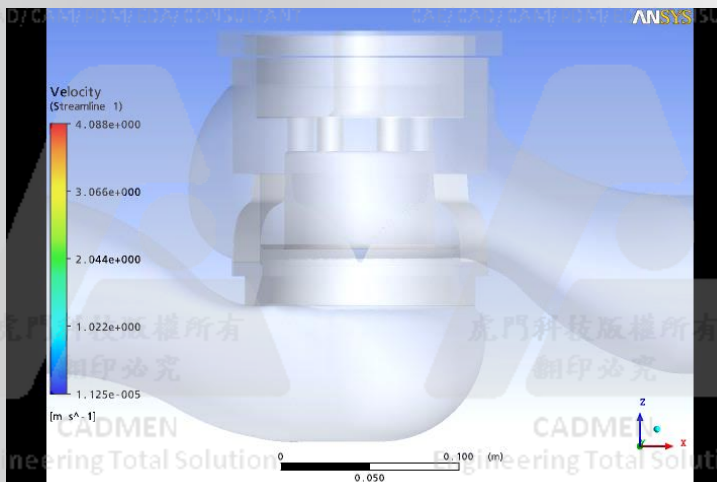
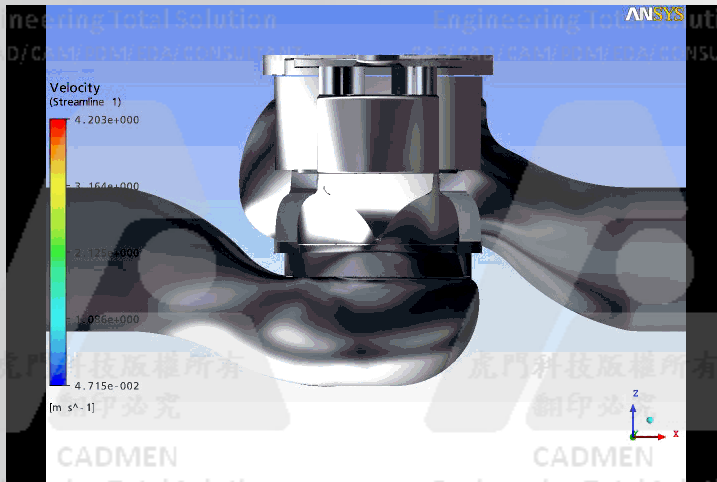
Flow Coefficient Cv as Function of Valve Travel given 0.07 pressure drop

$$C_v = \frac{1.167 \cdot Q \cdot \sqrt{G}}{\sqrt{\Delta P}}$$

| Valve travel | Cv       |
|--------------|----------|
| 10           | 13.1579  |
| 15           | 28.9473  |
| 20           | 50.8538  |
| 25           | 78.5938  |
| 30           | 116.5452 |
| 35           | 148.8508 |
| 40           | 164.5764 |
| 45           | 175.1532 |
| 50           | 180.9202 |

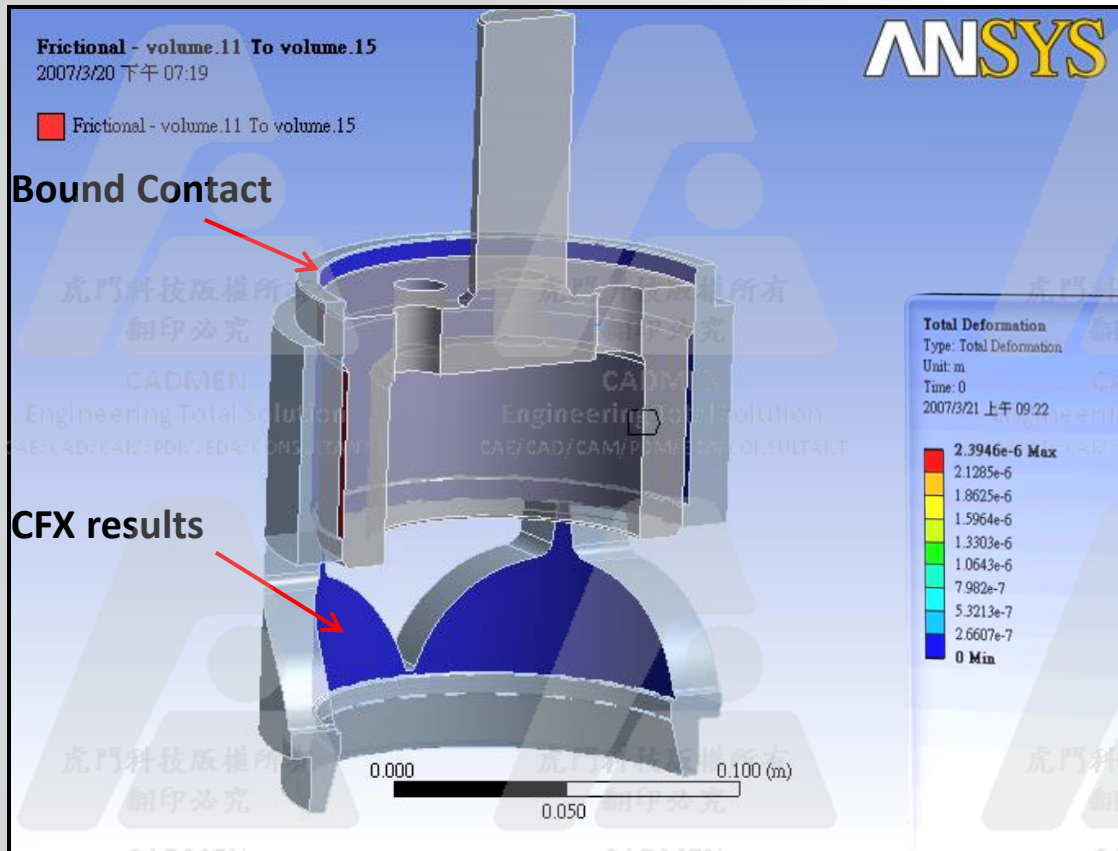
## Particle Tracking inside Valve Assembly

- Longer particle duration happens with strong swirling flow except outlet region.
- Slurry build-up may happens during the valve operation life cycle.

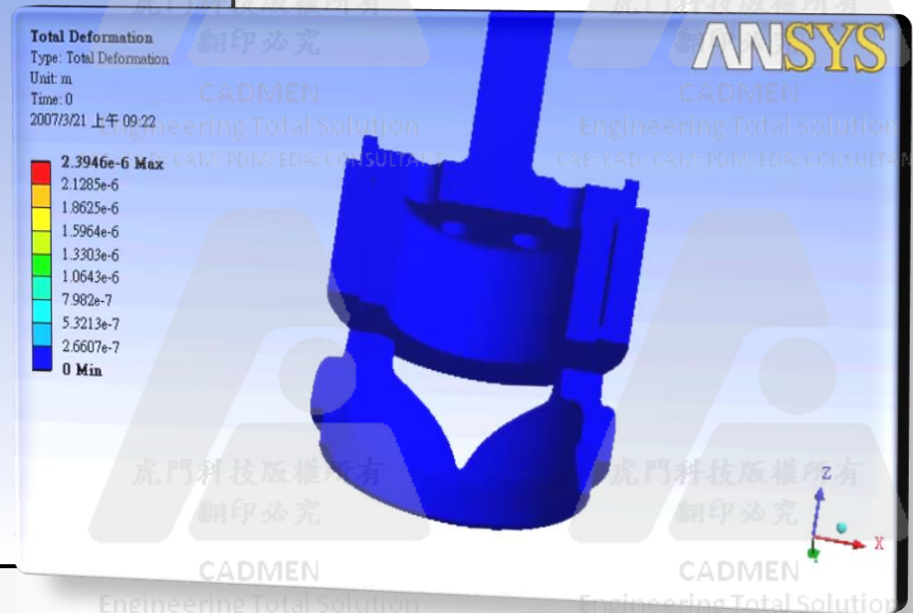


# 工業用閥門流場與結構變形分析

## Stress Analysis of the Valve Parts

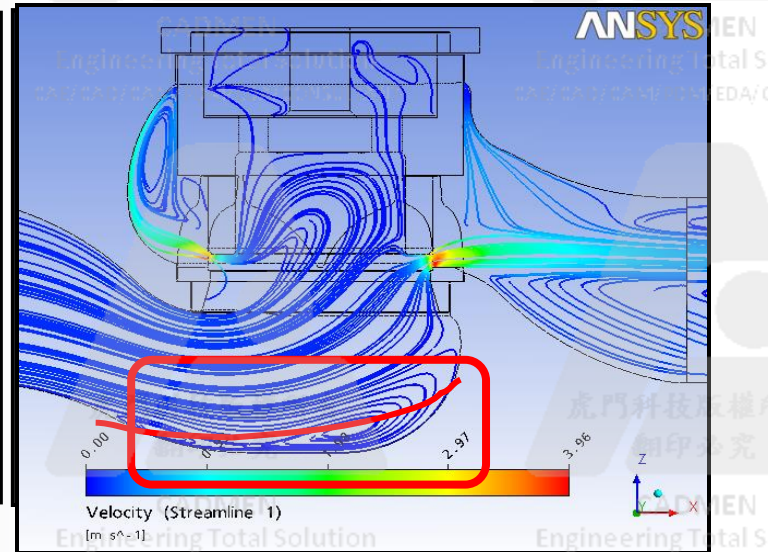
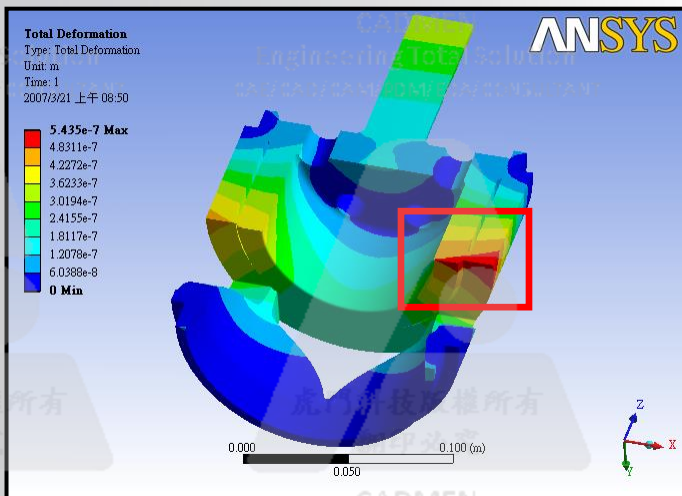
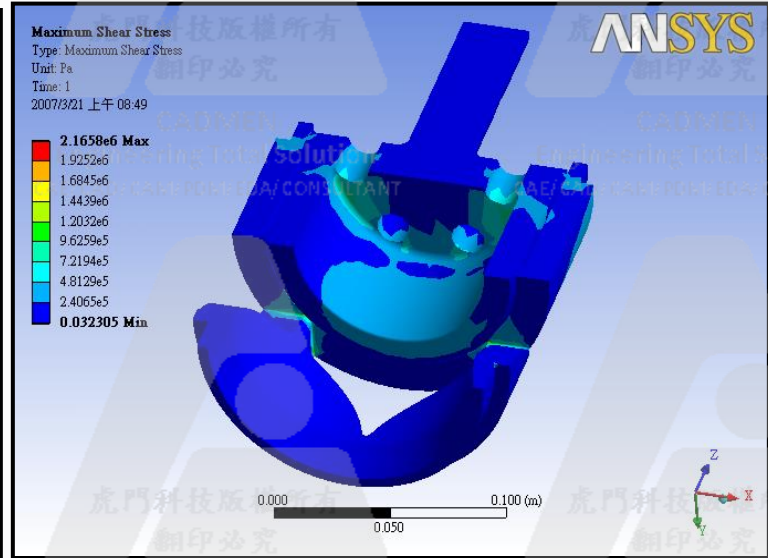
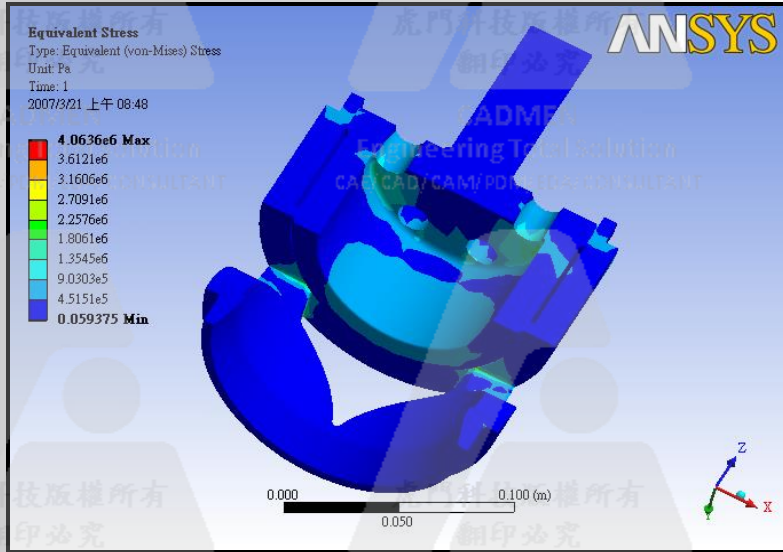


Pressure loading on the valve part surface contact with fluid can be extract directly from the flow solution.





# 工業用閥門流場與結構變形分析



虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

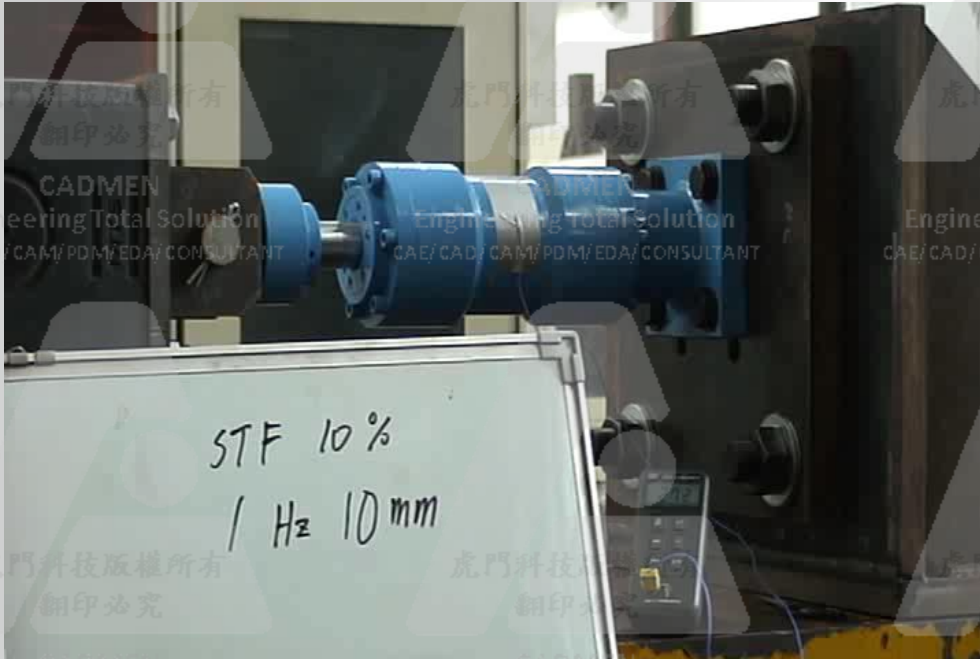
虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

# 油壓阻尼器 Hydraulic Damper – CFX Analysis



虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

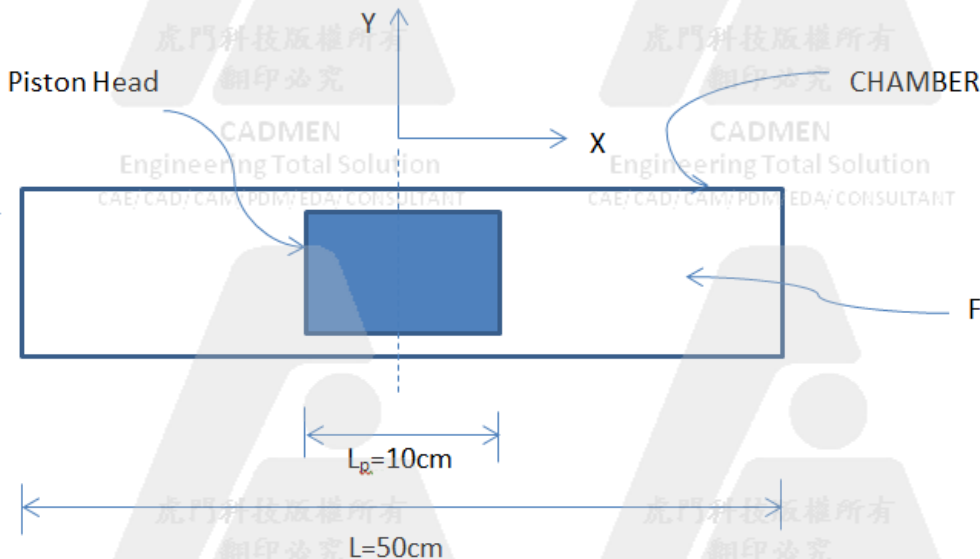
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

# 需求條件

虎門科技版權所有  
分析模型輪廓



虎門科技版權所有  
翻印必究

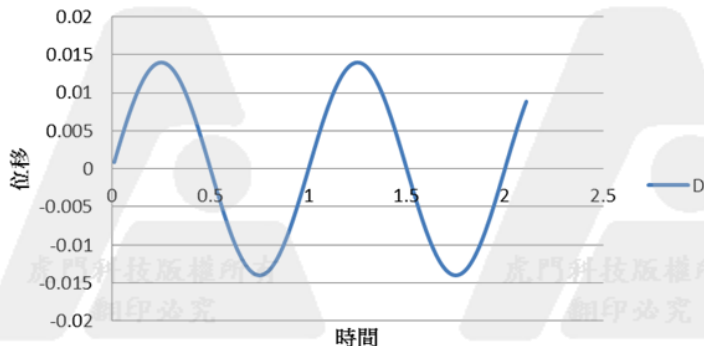
虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

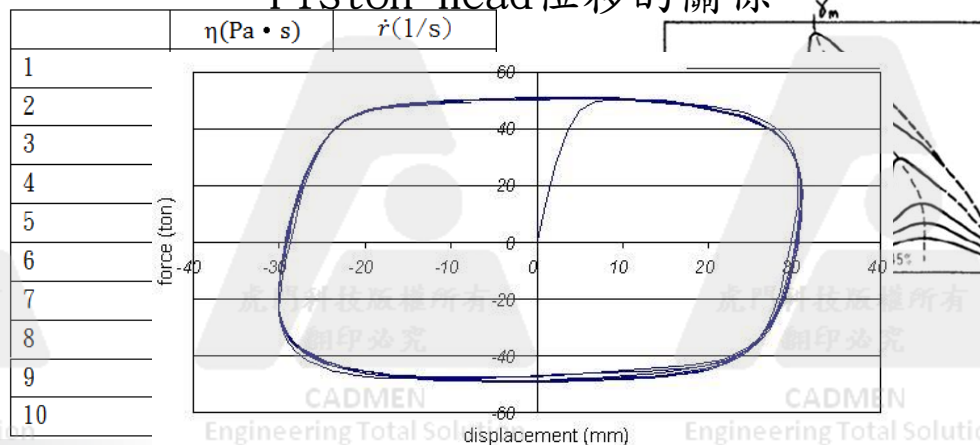
虎門科技版權所有  
翻印必究

## Piston Head運動方程

D

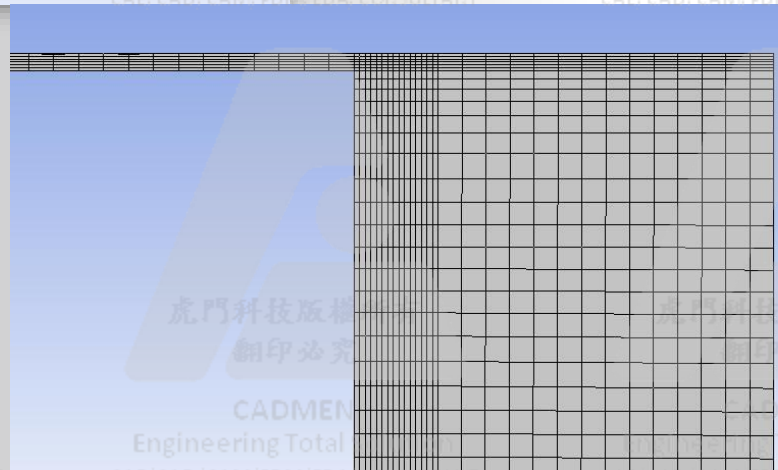
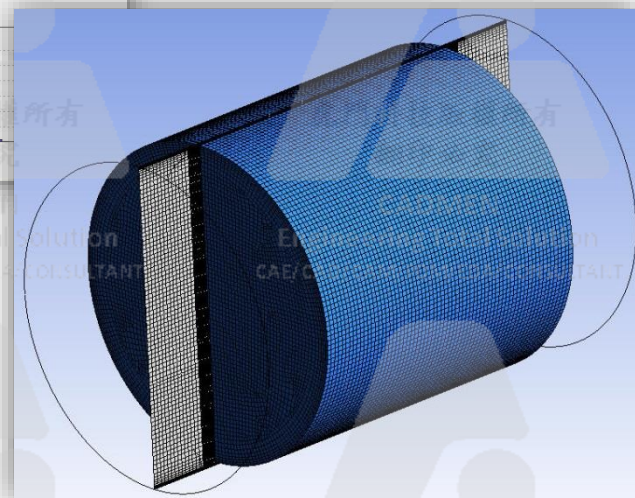
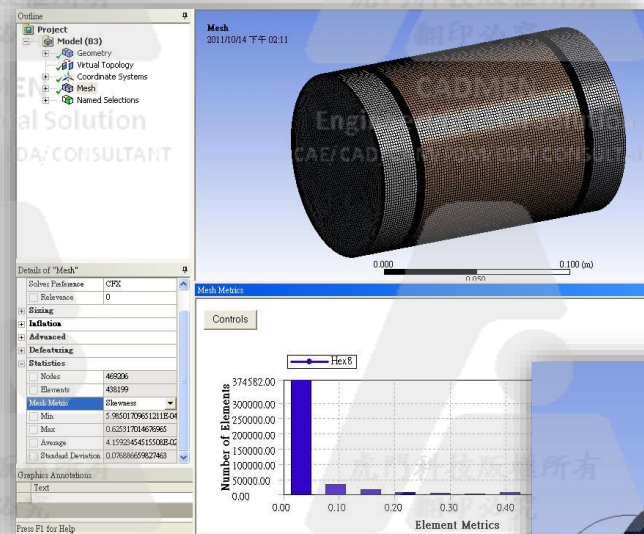
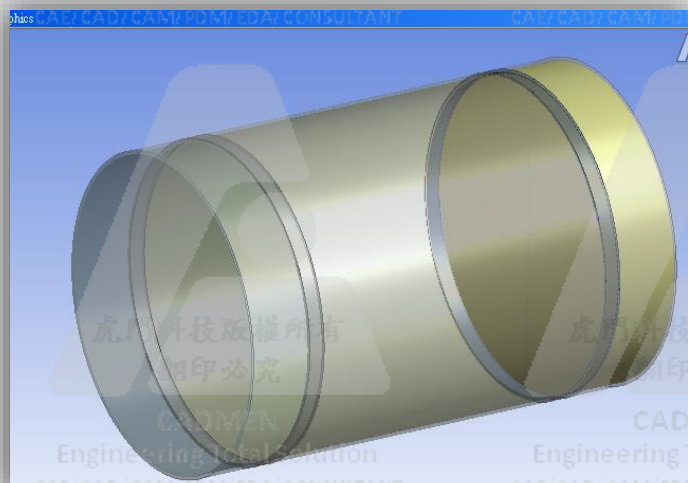


## Piston Head的阻力與 Piston Head位移的關係



## ANSYS MESHING

### Design Modeler



# 動網格作動示意

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

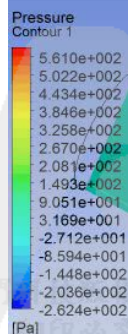
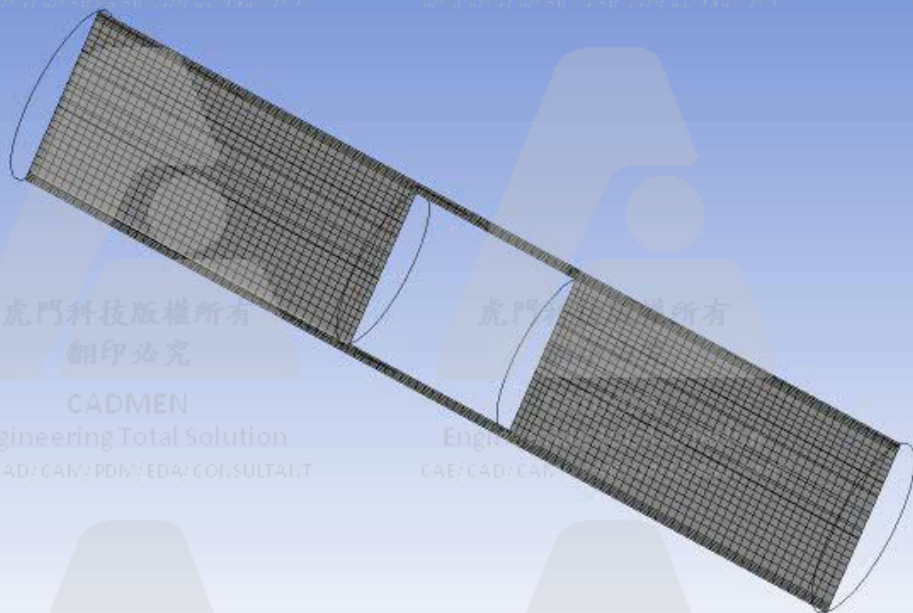
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

ANSYS  
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

ANSYS



[Pa]



虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution

CADMEN  
Engineering Total Solution

CADMEN  
Engineering Total Solution

CADMEN  
Engineering Total Solution

# CFX Setting-新增位移控制多項式

- 開啟CFX，如mesh呈閃電狀，須先按右鍵執行update
- 點擊Expression進入CEL介面，並點選右鍵新增多項式
- 新增活塞位移Expression
$$\text{deltaD} = -0.01[\text{m}] * \sin(2 * \text{pi} * 0.5 * \text{t} / 1[\text{s}])$$
- 切換為Plot並輸入預估之時間，以觀看定義是否正確

The screenshot displays the ANSYS CFX Expressions interface. On the left, the 'Expressions' panel shows a context menu with 'Expression' selected. The main window shows the 'Details of deltaD' dialog with the 'Plot' tab active. The 'Number of Points' is set to 100, and the 'End of Range' is set to 12 [s]. The 'Expression Variables' section shows 't' checked. A 'Plot Expression' button is visible at the bottom. To the right, a plot of 'deltaD [m]' versus 't' shows a sinusoidal wave oscillating between -0.01 and 0.01 m over a 12-second period.

# 新增viscosity與Shear Strain Rate的Function Table

- 在User Function處按右鍵新增，並給定此Function之名稱
- Argument Units處輸入Shear Strain Rate之單位 $[s^{-1}]$ ，Result Units處輸入viscosity的單位 $[Pa \cdot s]$ ，並勾選Extend Min與Extend Max

Function: viscosity

Details of viscosity

Basic Settings

Option: Interpolation (Data Input)

Argument Units:  $[s^{-1}]$

Result Units:  $[Pa \cdot s]$

Interpolation Data

Option: One Dimensional

|   | Coordinate | Value |
|---|------------|-------|
| 1 | 0.01009    | 1.892 |
| 2 | 0.01249    | 2.028 |
| 3 | 0.01565    | 2.109 |
| 4 | 0.02001    | 2.116 |
| 5 | 0.02509    | 2.207 |
| 6 | 0.03181    | 2.277 |

Coordinate:

Value:

Add Remove

Extend Min

Extend Max

User Functions

Insert

User Function

Edit in Command Editor

Paste

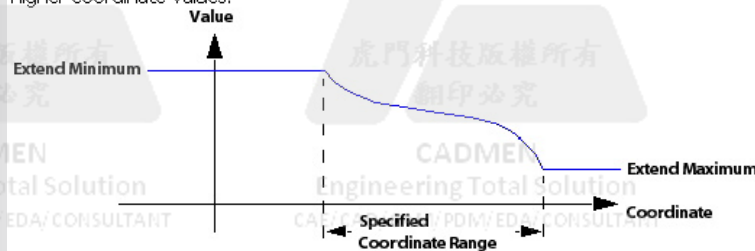
Insert Function

Name: viscosity

OK Cancel

## 28.1.1.1. Extended Minimum and Maximum

The **Extend Min** and **Extend Max** options enable you to increase the valid range of the interpolation function beyond the maximum or minimum specified coordinate values. The value the function will take at coordinate values lower than the minimum specified coordinate, which is equal to the value at the minimum specified coordinate. Similarly, the value at the maximum specified coordinate is extended for higher coordinate values.



# Import Data

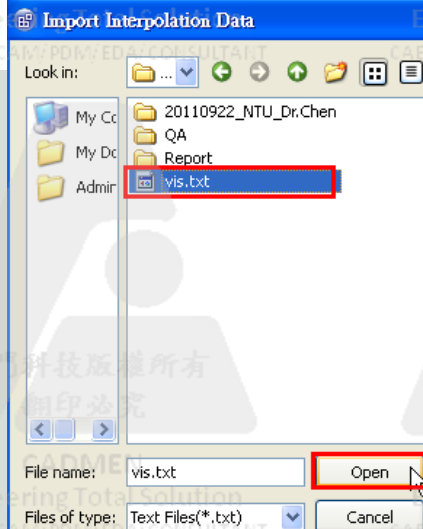
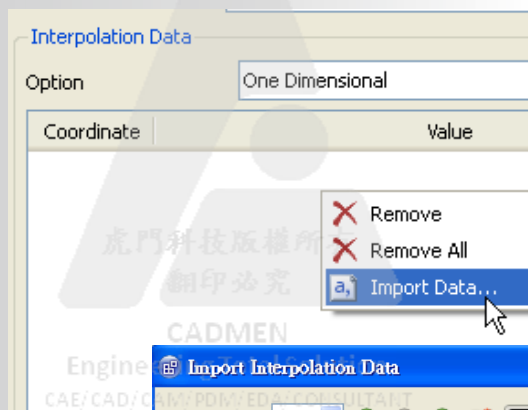
- 將viscosity與Shear Strain Rate的資料製作成txt檔，格式如下圖左為Shear Strain Rate，右為viscosity
- 按右鍵利用Import Data將所建立之txt檔匯入



vis.txt - 記事本

檔案(F) 編輯(E) 格式(O) 檢視(V)

|         |       |
|---------|-------|
| 0.01009 | 1.892 |
| 0.01249 | 2.028 |
| 0.01565 | 2.109 |
| 0.02001 | 2.116 |
| 0.02509 | 2.207 |
| 0.03181 | 2.277 |
| 0.03981 | 2.302 |
| 0.04998 | 2.266 |
| 0.06303 | 2.217 |
| 0.07966 | 2.184 |
| 0.1001  | 2.157 |
| 0.1259  | 2.136 |
| 0.1586  | 2.072 |



Interpolation Data

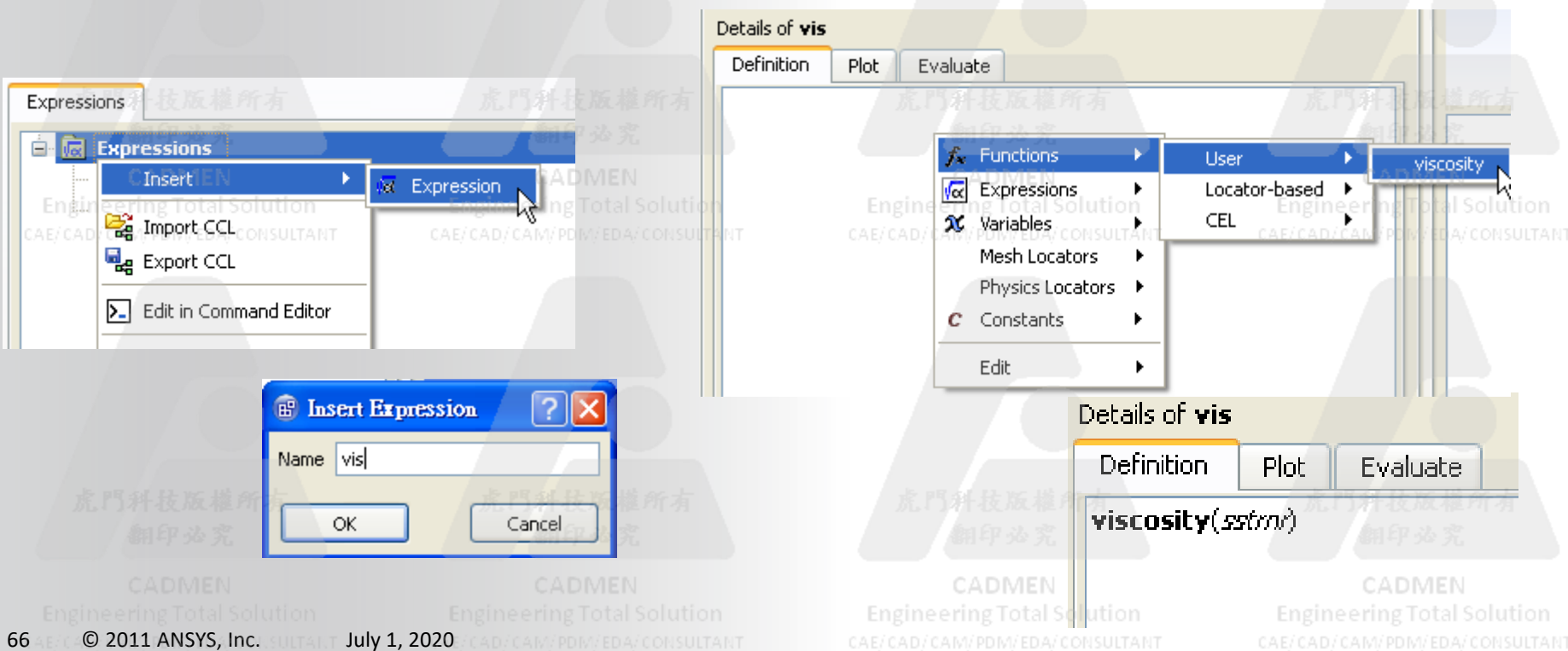
Option: One Dimensional

|   | Coordinate | Value |
|---|------------|-------|
| 1 | 0.01009    | 1.892 |
| 2 | 0.01249    | 2.028 |
| 3 | 0.01565    | 2.109 |
| 4 | 0.02001    | 2.116 |
| 5 | 0.02509    | 2.207 |
| 6 | 0.03181    | 2.277 |



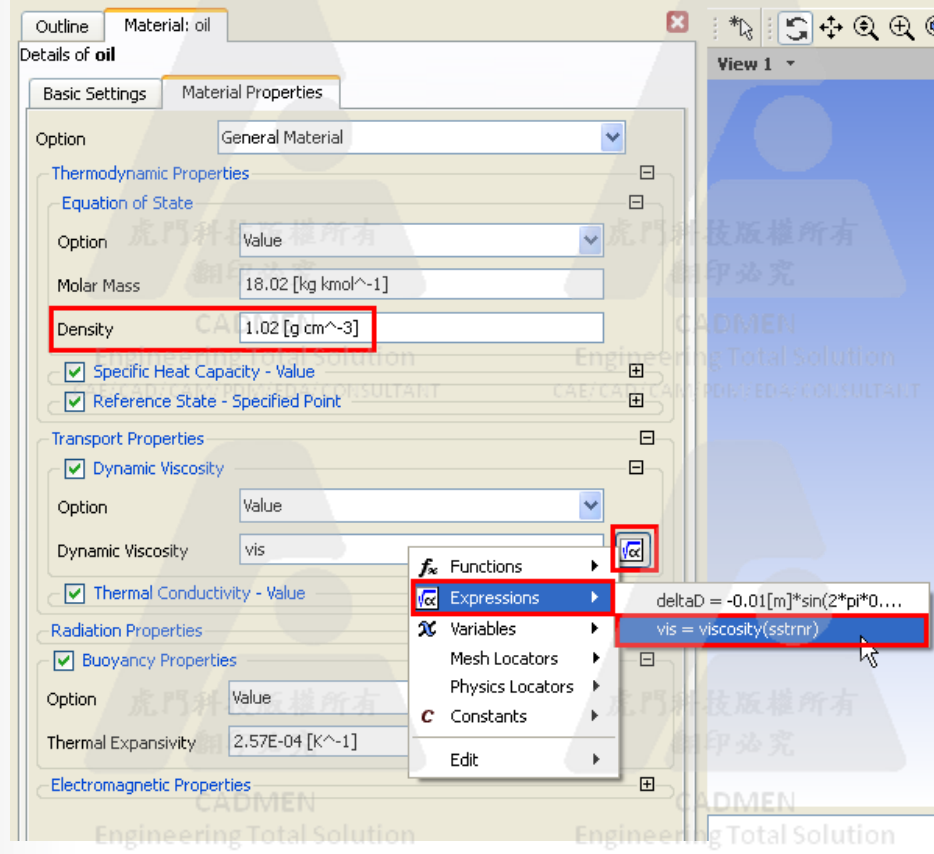
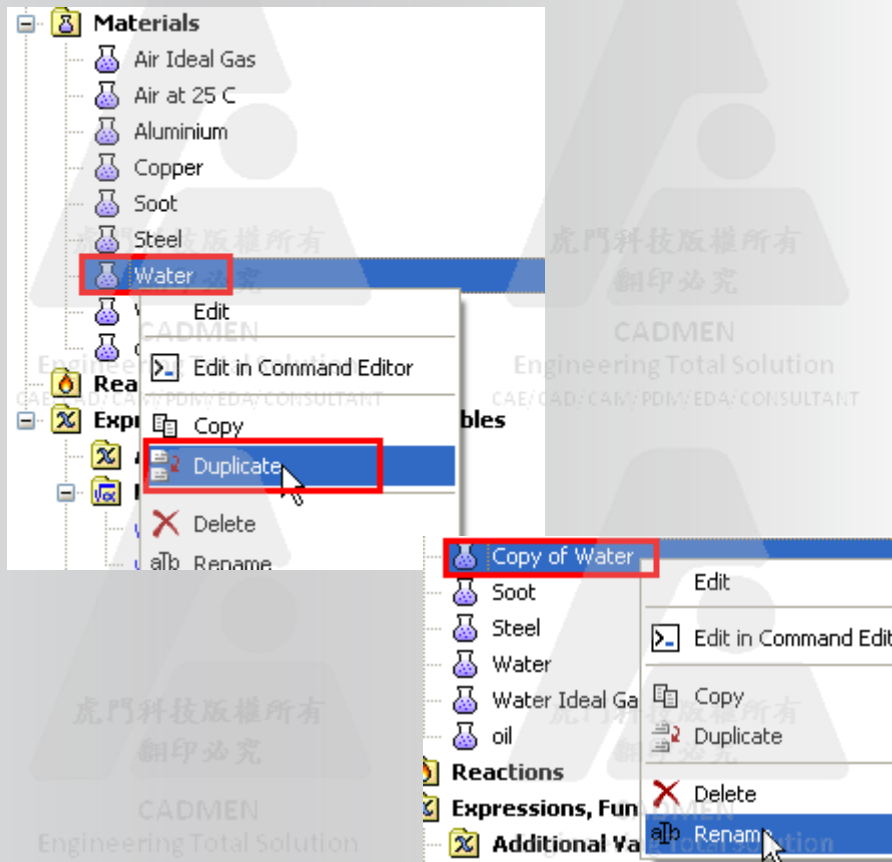
# 新增流體黏滯性之函式

- 再次進入CEL介面，並增加新的Expression名為vis
- 點選右鍵Functions->User->viscosity將所定義的Function讀入，並在viscosity()之框內給定sstrnr，如下圖，如此便能得到求解時變化之Shear Strain Rate所對應之viscosity

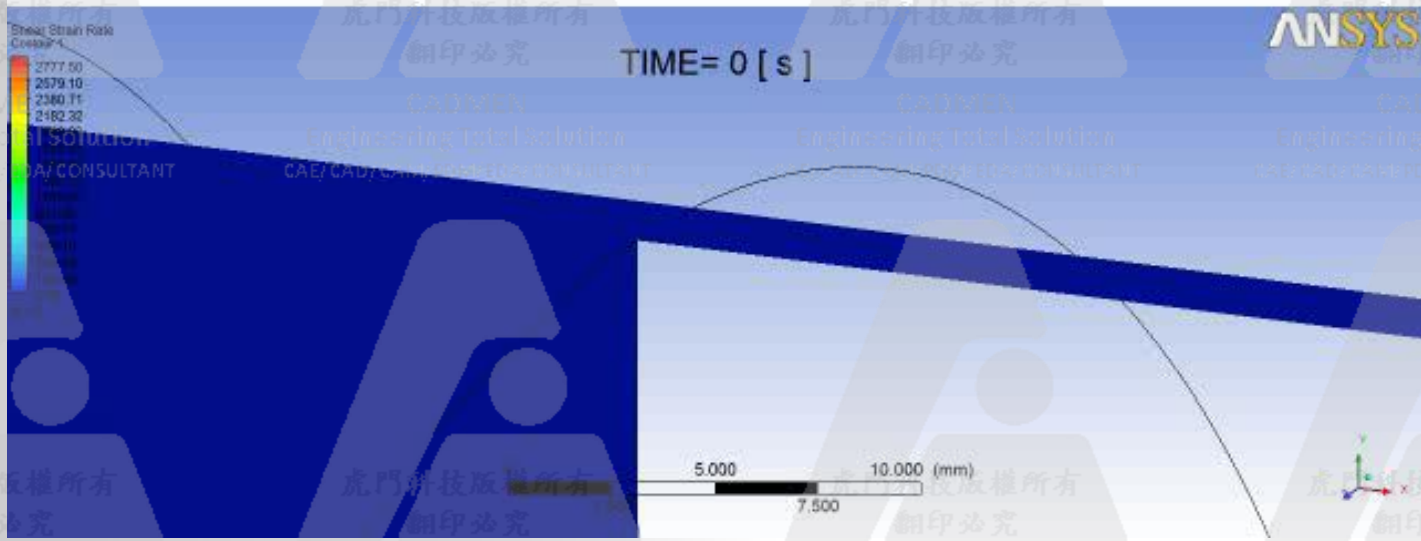
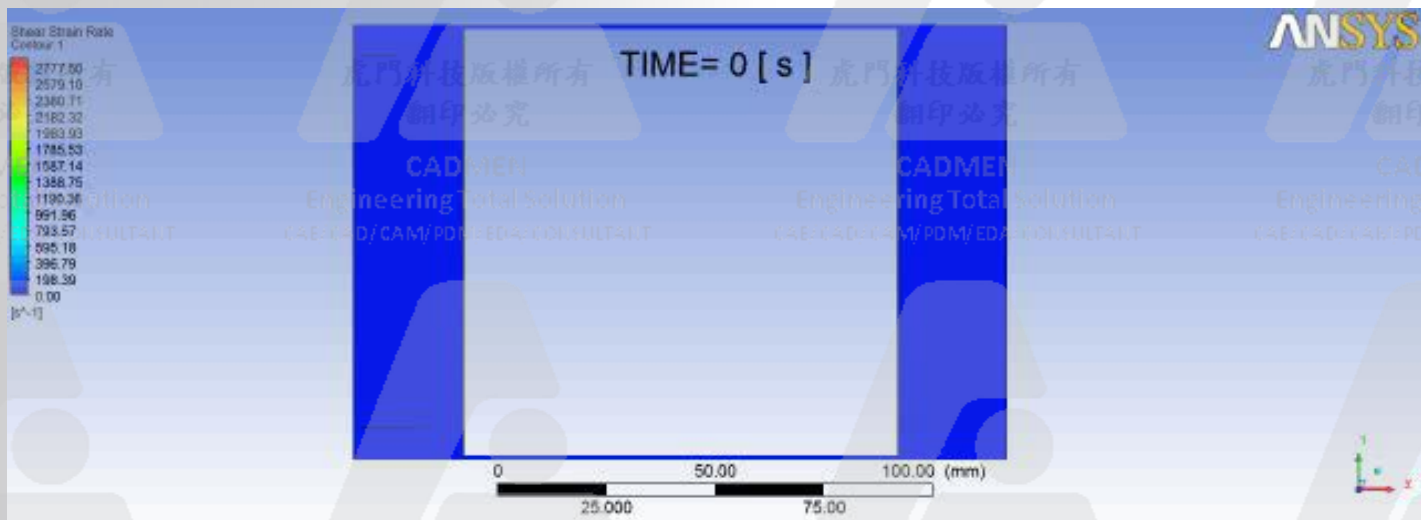


# 新增液體材料

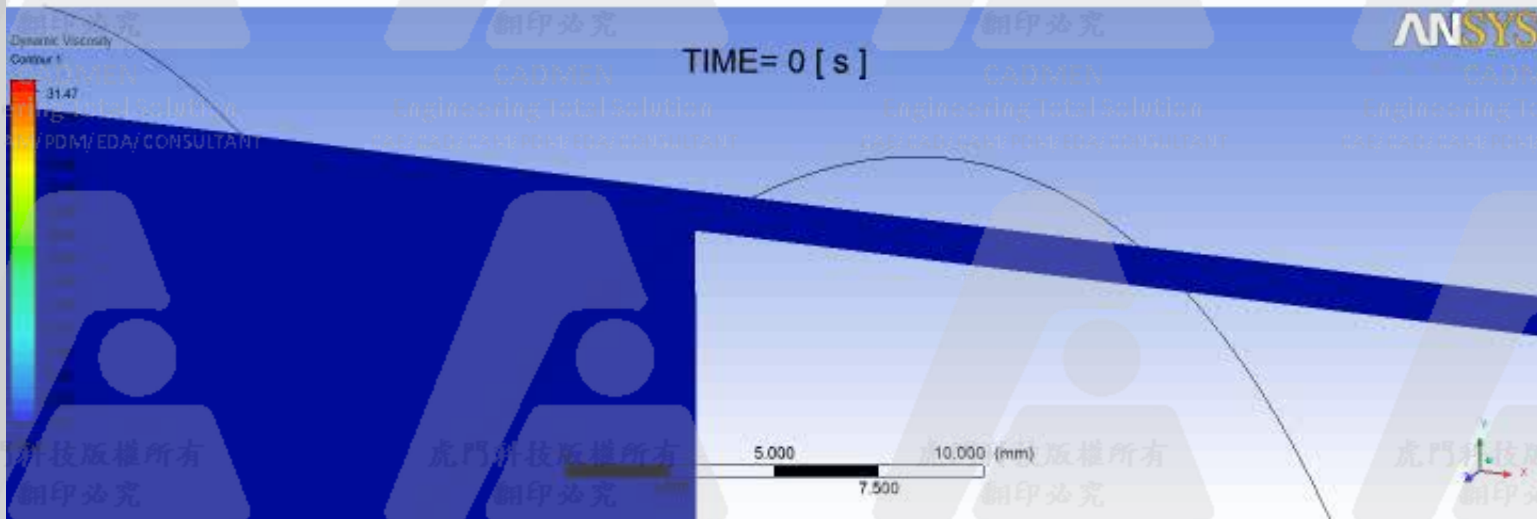
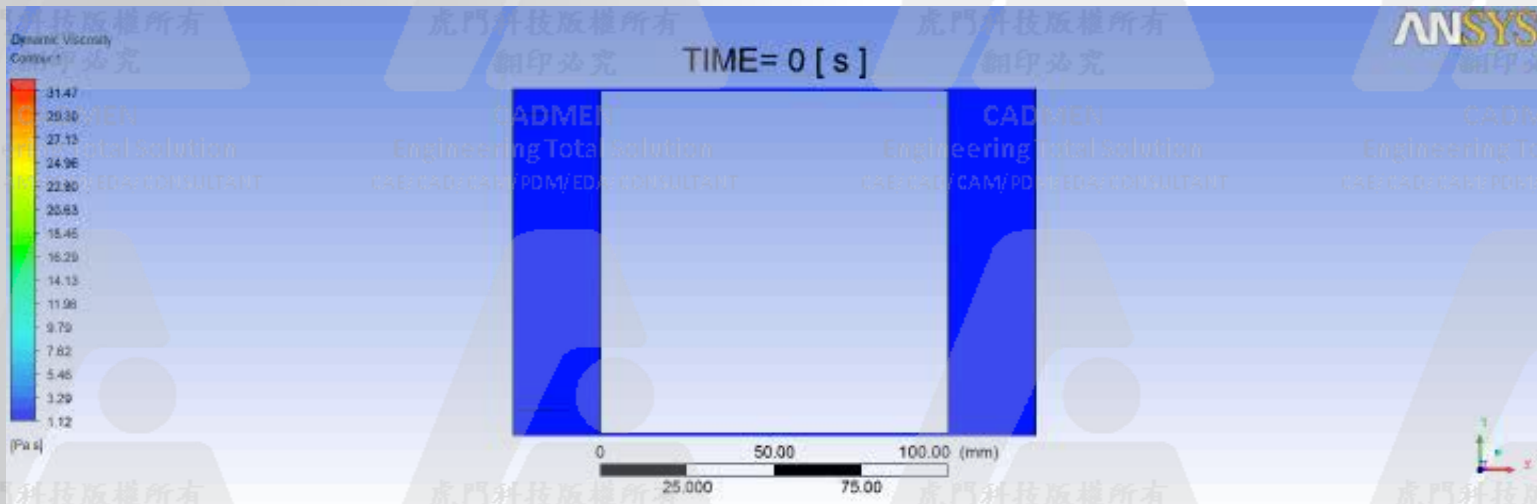
- 複製water材料，並按右鍵rename為oil
- 改變適當的參數，並將所新增的expression “vis”，選入此材料之 Dynamic Viscosity



# Results - Shear Strain Rate

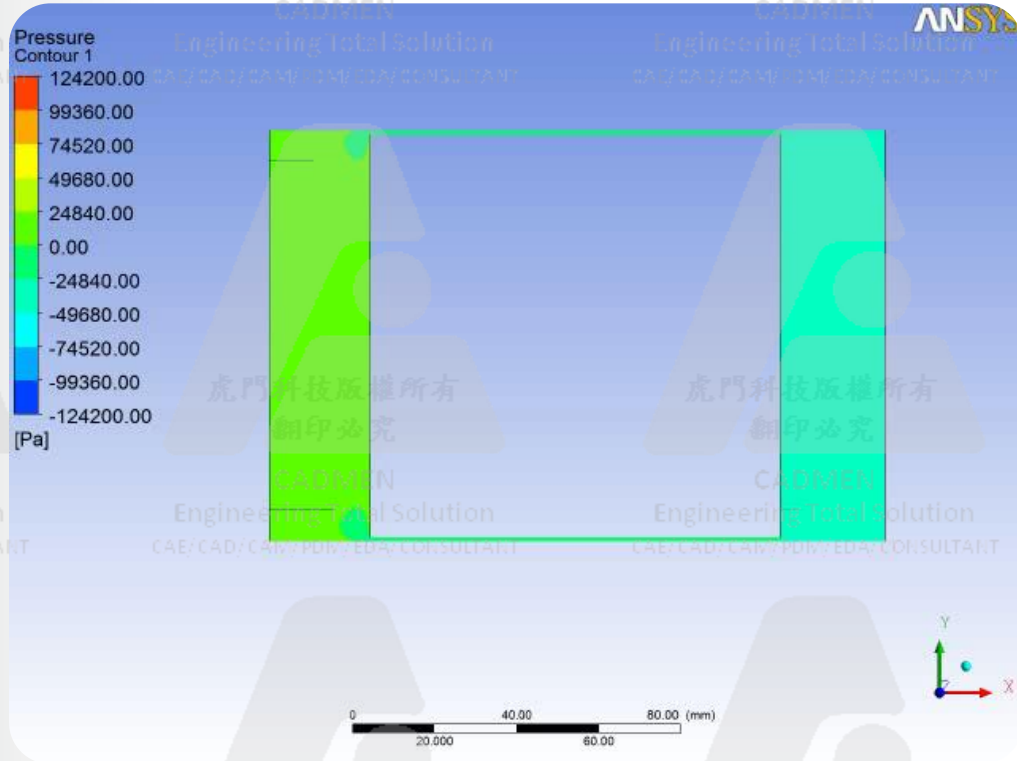
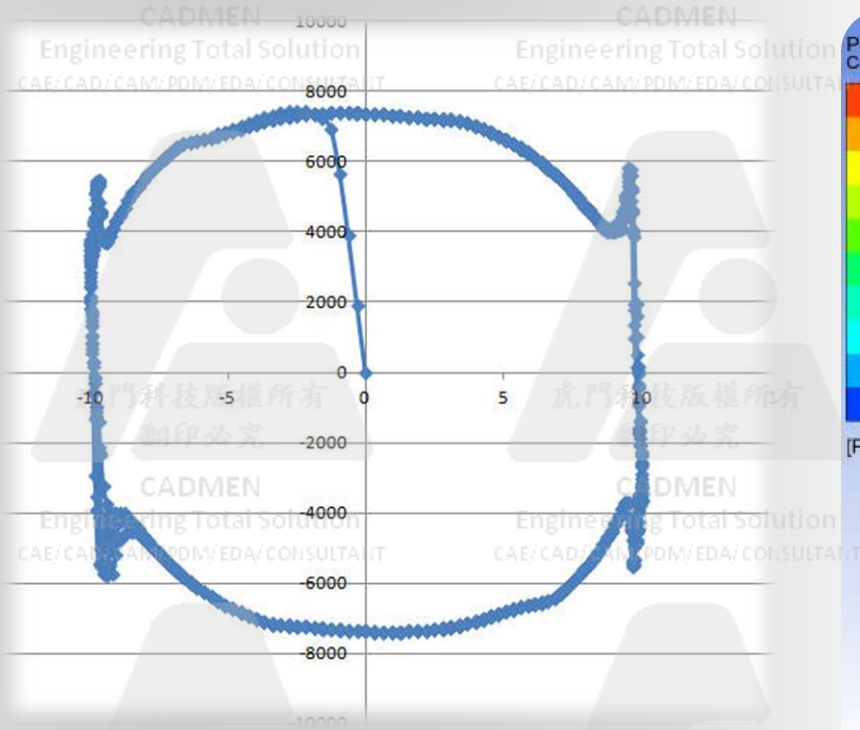


# Results - Viscosity



# Results – Pressure, Force and Displacement

Piston Head的阻力與Piston Head位移的關係



# 泵設計分析之需求

泵是一種通用機械，也是人類發明最早的將自然轉換成有用功的機械之一。曾經日本對泵的能源消耗進行了可信的統計，其結果表明泵在能源消耗方面出乎預料地佔全國總發電量的1/4。儘管如此，人們對泵內流動了解並不充分，或者說，人們還無法完全掌握其流動現象。

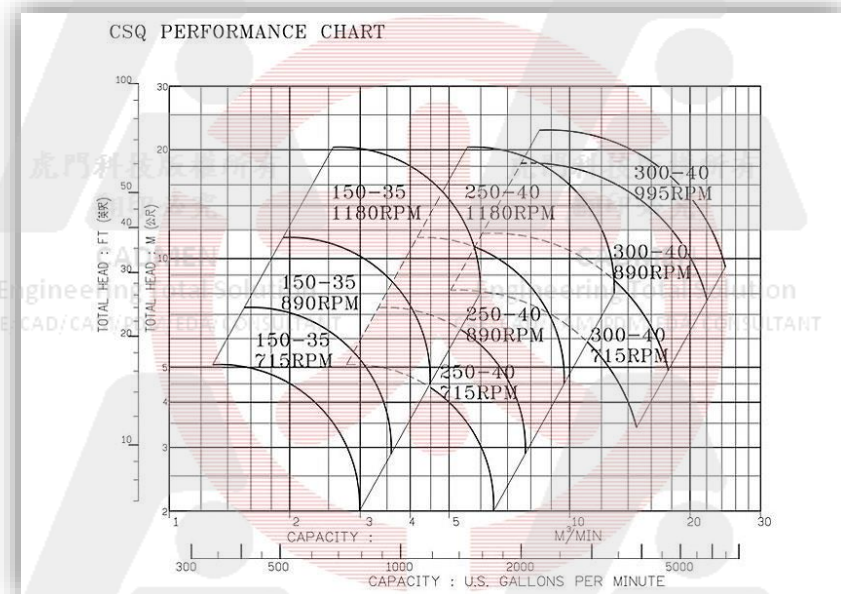
泵現有的常用設計方法，是配合理論計算模式、圖表，並輔以相似換算法或是速度係數法。這幾種方法都是以現有成功經驗為基礎的一種借鑒方式的設計方法，已難符合產業升級的設計需求。

結構強度

效能提升

震動噪音

馬達設計



# 泵設計分析之需求

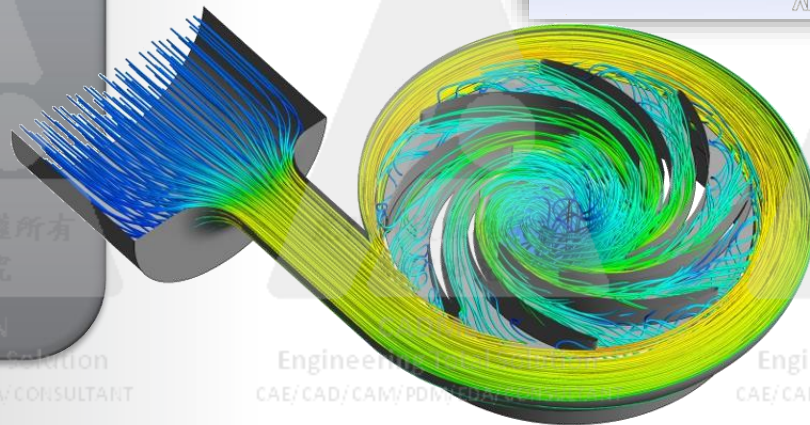
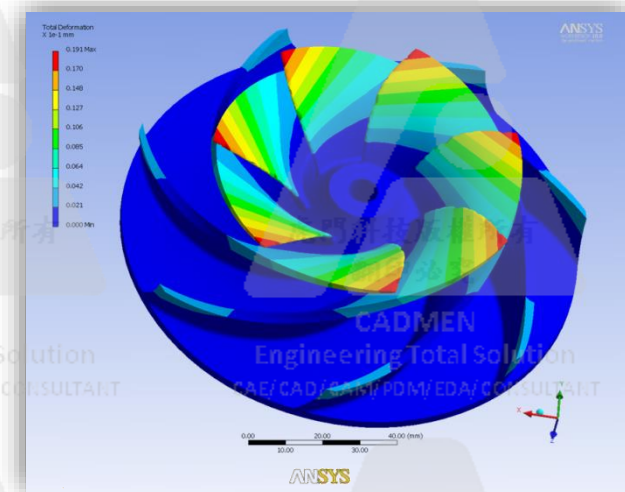
泵是一種通用機械，也是人類發明最早的將自然轉換成有用功的機械之一。曾經日本對泵的能源消耗進行了可信的統計，其結果表明泵在能源消耗方面出乎預料地佔全國總發電量的1/4。儘管如此，人們對泵內流動了解並不充分，或者說，人們還無法完全掌握其流動現象。

泵現有的常用設計方法，是配合理論計算模式、圖表，並輔以相似換算法或是速度係數法。這幾種方法都是以現有成功經驗為基礎的一種借鑒方式的設計方法，已難符合產業升級的設計需求。

## ANSYS CFD 分析

提高設計效率、縮短設計時間、優化參數組合、  
探討細部現象、提高泵的性能

- 流場分析
- 特性曲線
- 幾何設計變更
- 空蝕現象探討
- 結構耦合分析

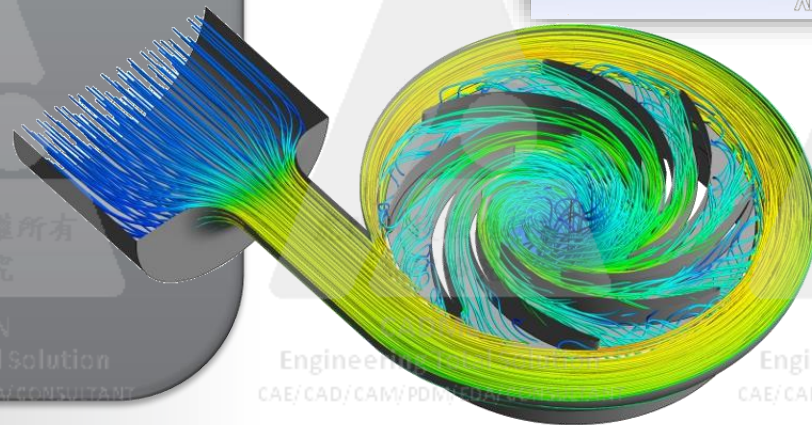
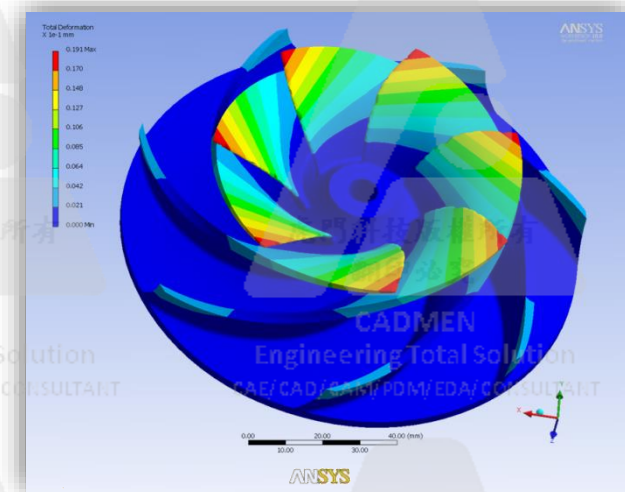


# 泵設計分析之需求

泵是一種通用機械，也是人類發明最早的將自然轉換成有用功的機械之一。曾經日本對泵的能源消耗進行了可信的統計，其結果表明泵在能源消耗方面出乎預料地佔全國總發電量的1/4。儘管如此，人們對泵內流動了解並不充分，或者說，人們還無法完全掌握其流動現象。

泵現有的常用設計方法，是配合理論計算模式、圖表，並輔以相似換算法或是速度係數法。這幾種方法都是以現有成功經驗為基礎的一種借鑒方式的設計方法，已難符合產業升級的設計需求。

- CPD(Centrifugal Pump Design)
- 豐富紊流模型
- 旋轉流體分析
- 動態網格分析
- 參數化、最佳化分析
- 多相流空蝕分析模組
- 離散項 DPM 顆粒模組
- 氣動噪音
- FSI 流固耦合分析





# Leading Companies Use ANSYS for Rotating Machinery Simulation

## ANSYS customers include.....

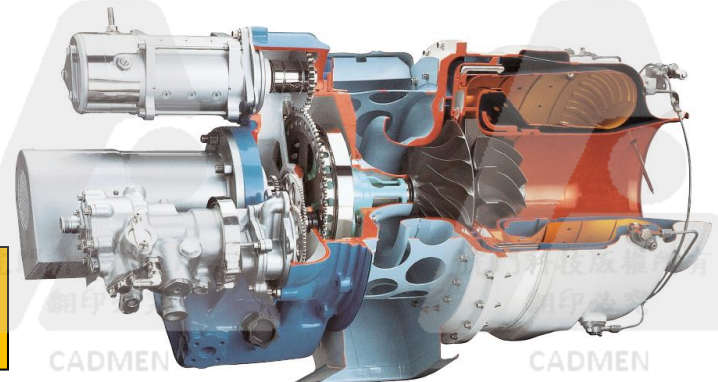
- All 5 of the top 5 largest **aircraft engine** manufacturers
- All 7 of the top 7 largest **gas turbine** manufacturers
- All 7 of the top 7 largest **steam turbine** manufacturers
- All 5 of the top 5 largest **turbocharger** manufacturers
- 8 of the top 10 largest **pump** manufacturers

.....*in the world!*

## ANSYS software is the most commonly used CAE tool for.....

- **Hydraulic (water) turbine** simulation
- **Wind turbine** simulation

**Auxiliary  
Power Unit**



**Honeywell**

# Widespread Usage of ANSYS Across Industry Sectors for Rotating Machinery Simulation

## Aircraft & Defence

- GE
- Hamilton Sundstrand
- Honeywell
- ITP
- MTU
- NASA
- Pratt & Whitney
- PW Rocketdyne
- Rolls Royce
- Snecma
- Teledyne
- Turbomeca
- Volvo Aero



## Energy

- Alstom
- Andritz
- Dresser-Rand
- GE
- Hitachi
- IMPSA
- Mitsubishi
- Rolls Royce
- Parker
- Siemens
- Solar Turbines
- Toshiba
- Vestas
- Voith

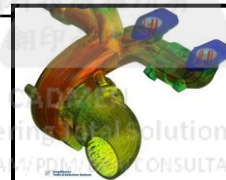
Courtesy  
Siemens AG



## Ground/Marine Transportation

- ABB Turbo
- Borg Warner
- Bosch Mahle
- Continental
- Cummins Turbo
- Ford
- GM
- Honeywell
- IHI Turbochargers
- Luk
- Napier
- Pierburg
- Rolls-Royce Marine
- Voith Turbo

Courtesy  
Borg Warner



## Chemical Process/HVAC

- Allweiler
- Cameron
- Carrier
- Clydeunion
- Dresser-Rand
- Ebara
- Emerson
- Grundfos
- Hitachi
- KSB
- MAN
- McQuay
- Praxair
- Trane
- York

Courtesy  
Dresser Rand



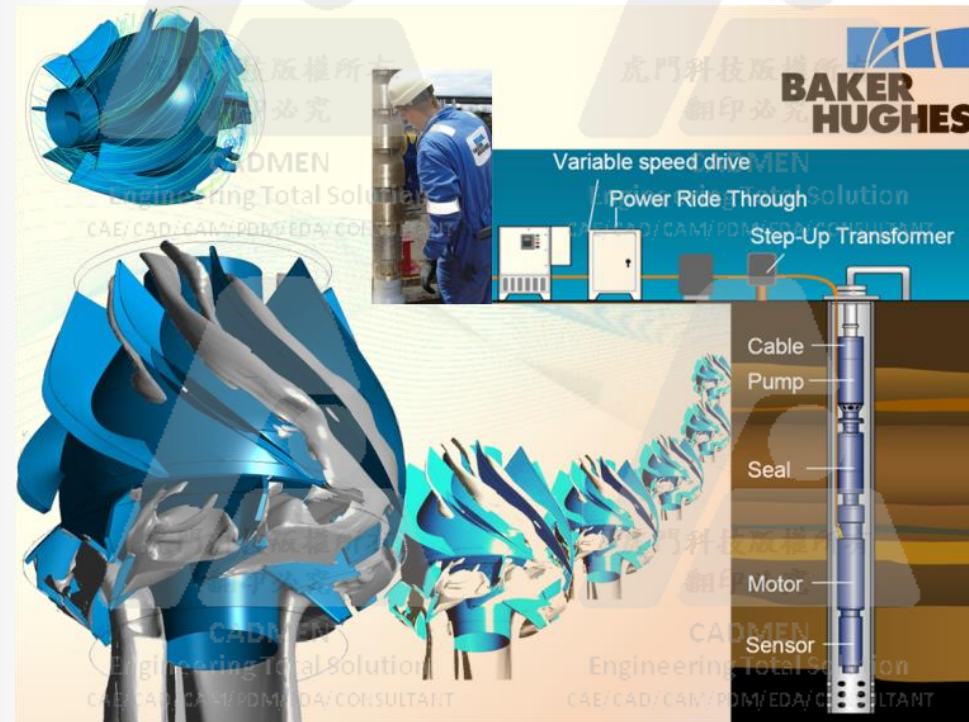
# 特殊專利沉水泵 --- Baker Hughes

## Problem

Baker Hughes produces a special patented Electrical Submersible Pump, called the MVP, that handles up to 70% gas by volume.

## Solution

Baker Hughes uses ANSYS Fluent to simulate **the two phase flow** in the MVP pump to show how the fluid behaves inside the pump and how to improve the product.



By using ANSYS Fluent, Baker Hughes received a **better understanding** of their Electrical Submersible Pump and were able to **improve it's performance**.

Baker Hughes Inc. 貝克休斯  
全球第三大油田服務公司

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

# Centrifugal Pump 設計應用

## 1D初步設計 ~ 3D流場分析

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

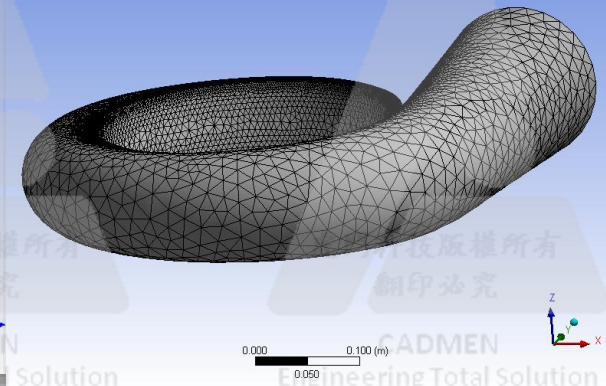
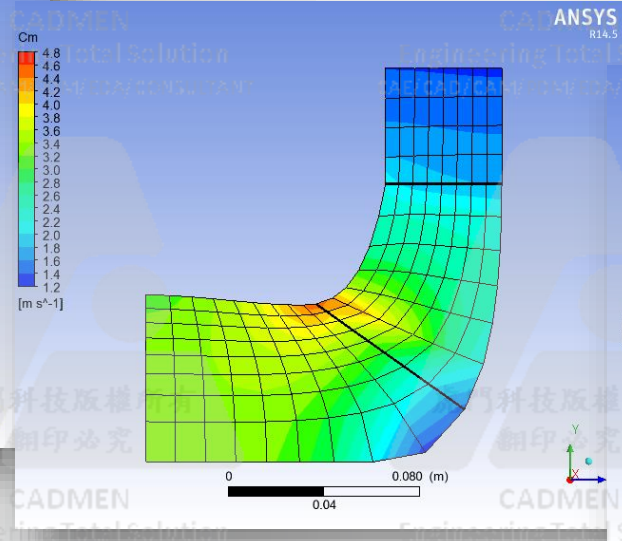
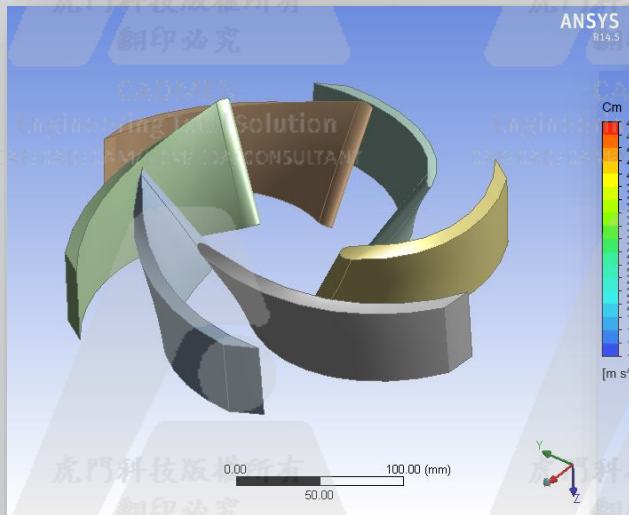
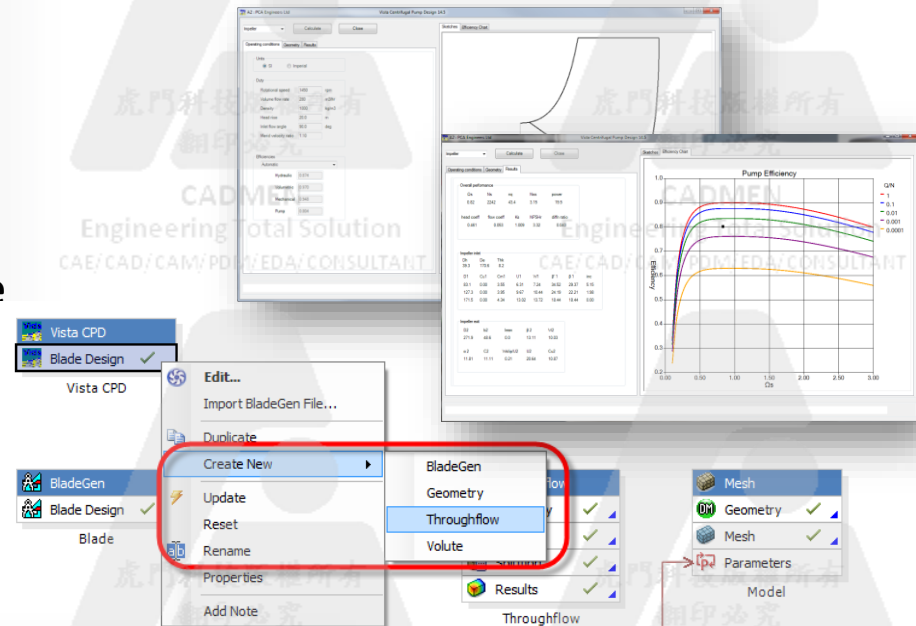
虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

# Workbench Integration of Vista CPD

## Vista CPD (Centrifugal Pump Design)

- Native Workbench application
- Ability to use Vista CPD directly create
  - Blade geometry model
  - Throughflow analysis
  - Volute geometry and mesh



# Workbench Integration of Vista CPD

Vista D2: PCA Engineers Ltd 版權所有 Vista Centrifugal Pump Design 15.0

Volume

Operating condition: **Geometry** Results

Casing rotation angle:  deg

Section Type

- Elliptical / circular
- Rectangular

Aspect ratio

Diffuser

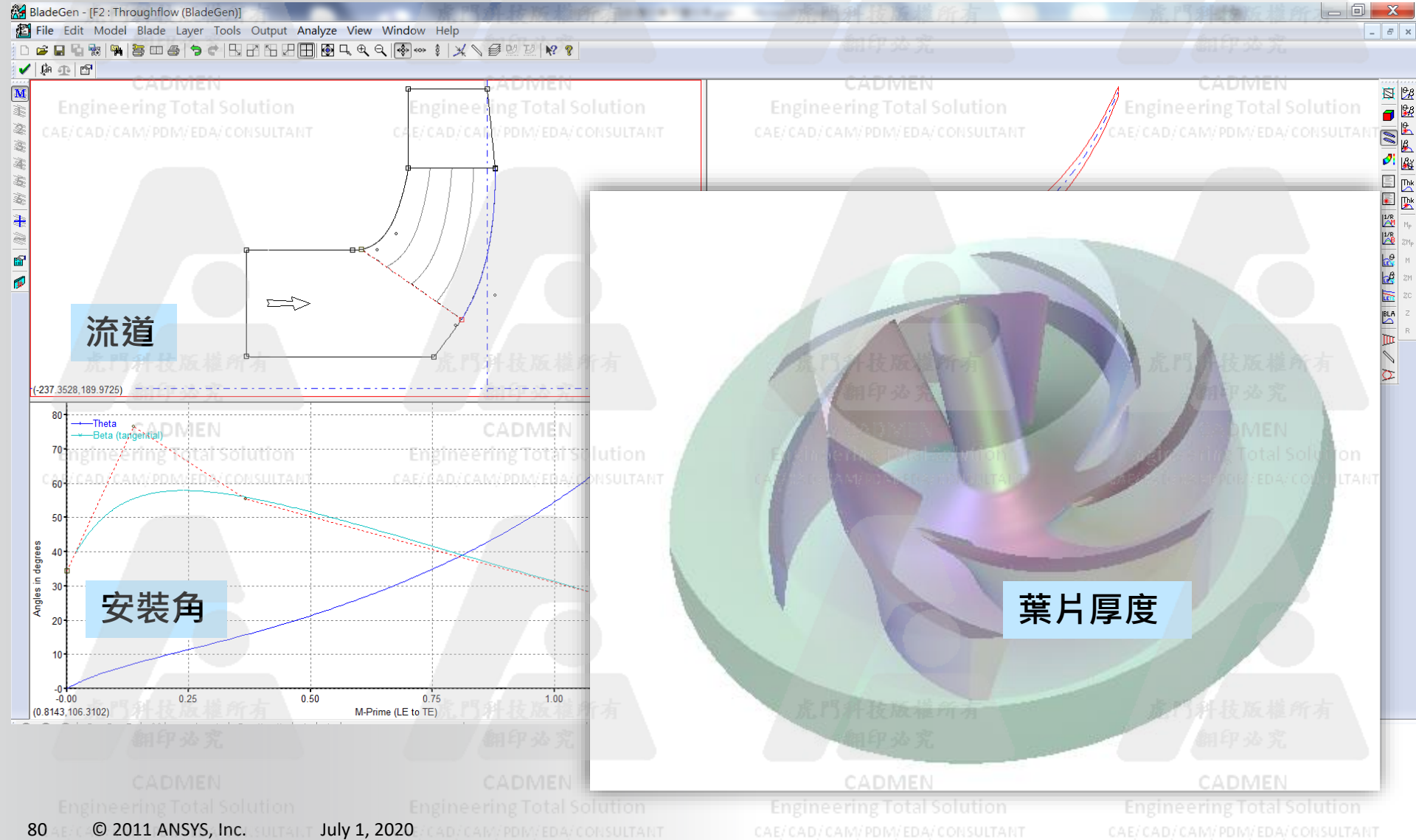
- User defined diam
- User defined leng

Sketches Efficiency Chart

Operating conditions Geometry Results

| Overall performance |            |       |           |             |
|---------------------|------------|-------|-----------|-------------|
| $\Omega s$          | Ns         | nq    | Nss       | power (kW)  |
| 0.82                | 2242       | 43.4  | 3.15      | 19.5        |
| head coeff          | flow coeff | Ks    | NPSHr (m) | diffn ratio |
| 0.461               | 0.053      | 1.009 | 3.32      | 0.040       |

# Workbench Integration of Vista CPD

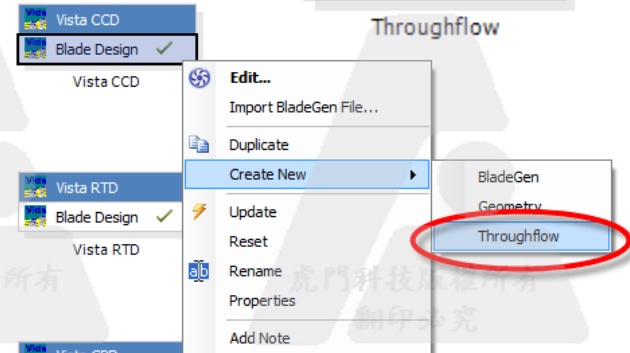
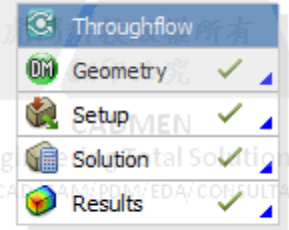


# Turbomachinery Workflows in Workbench

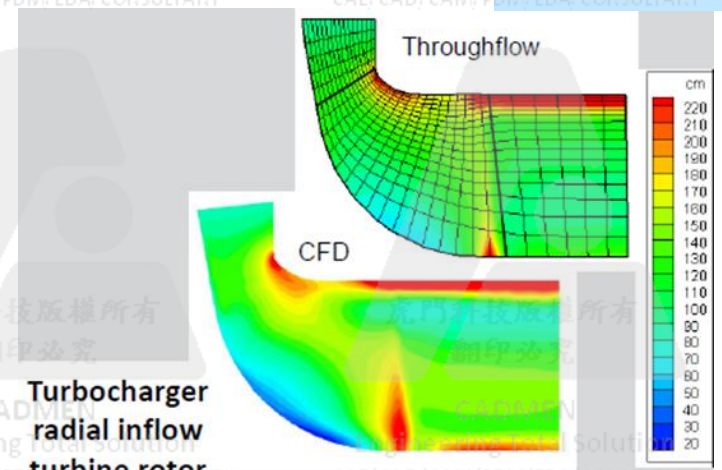
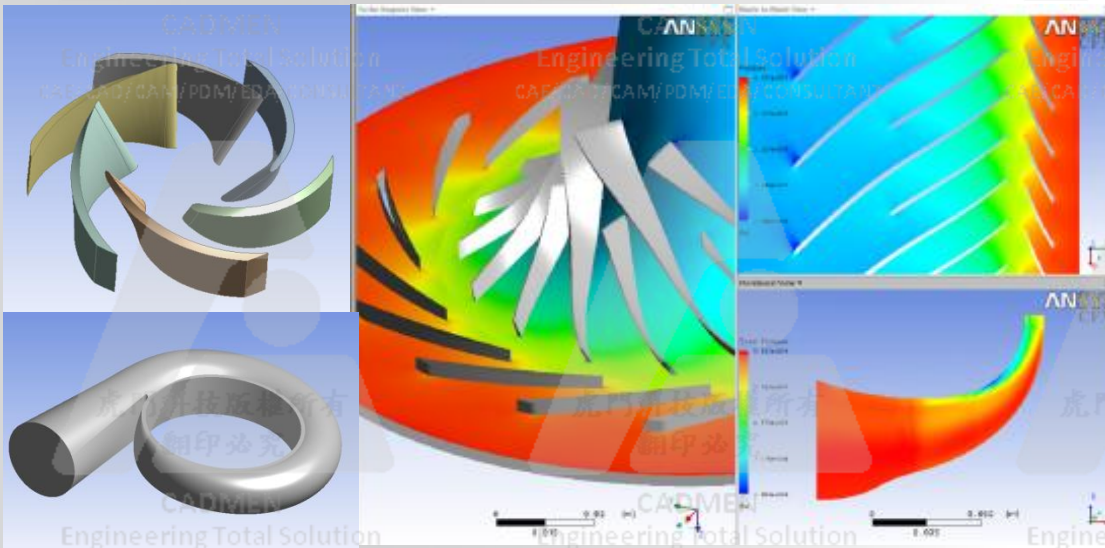
## New Throughflow System in Workbench

- Combination of geometry and Vista TF
- Ability to create directly from Vista preliminary sizing tools
  - Vista RTD/CCD/CPD
- Improves ability to explore and analyze geometry generated with Vista preliminary sizing tools, using throughflow analysis

## 3D CFD Analysis



軸向速度



Turbocharger radial inflow turbine rotor



# 性能曲線分析



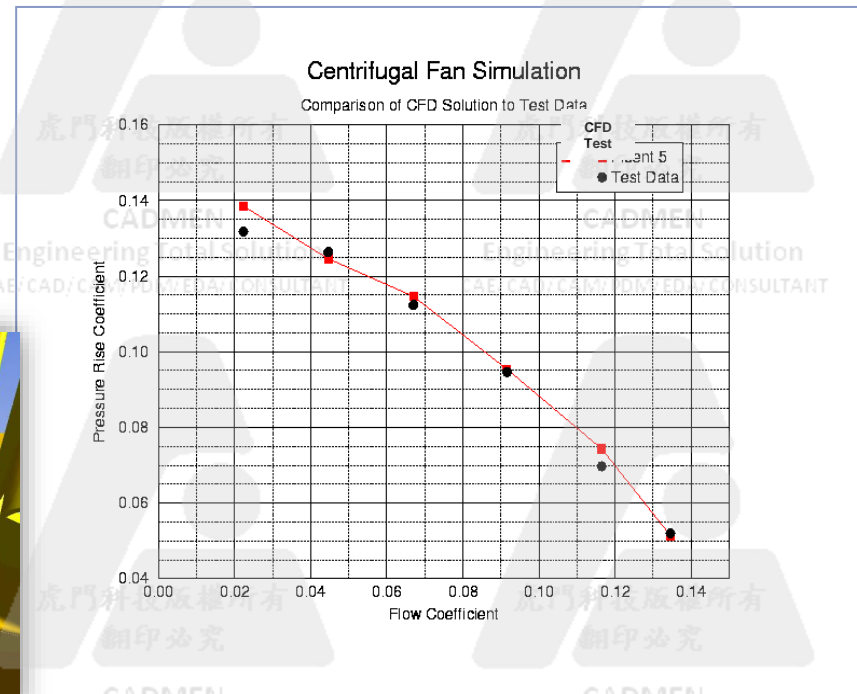
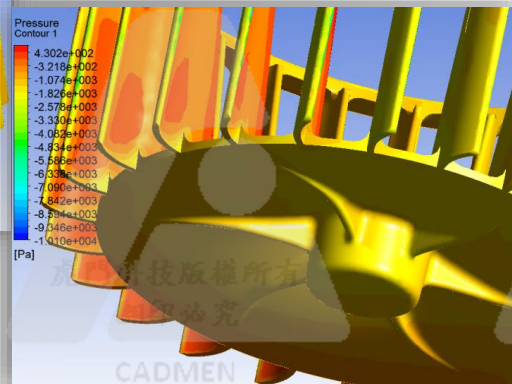
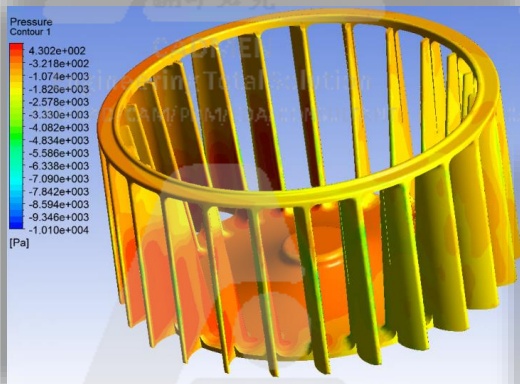
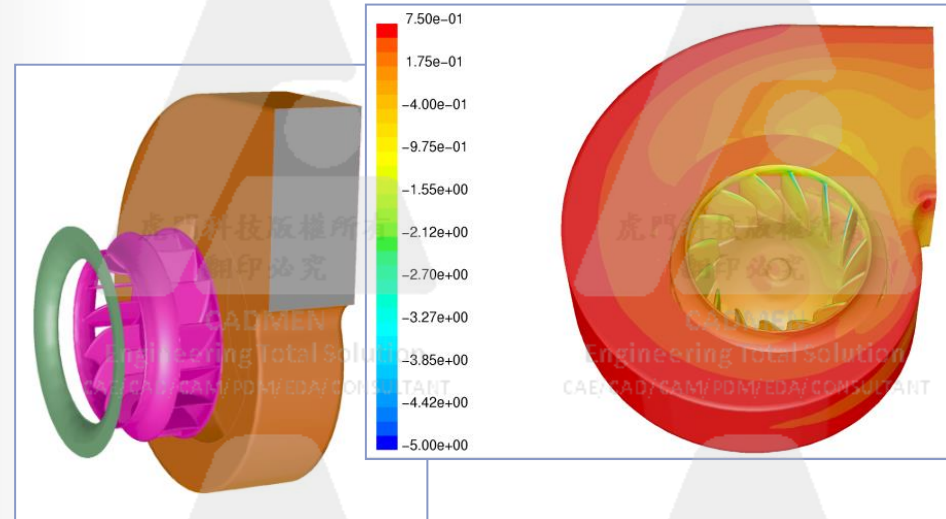
Industrial backward-inclined centrifugal fan

15 blade, shrouded wheel

Scroll volute with rectangular outlet

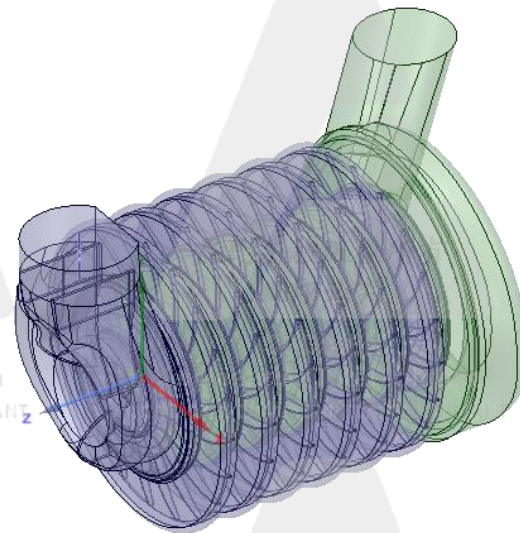
Numerical model

- ANSYS FLUENT
- Steady-state (MRF), incompressible flow (air)
- Realizable  $k-\epsilon$  turbulence model with wall functions
- Steady-state solutions obtained over range of flow rates at constant speed (fan curve)

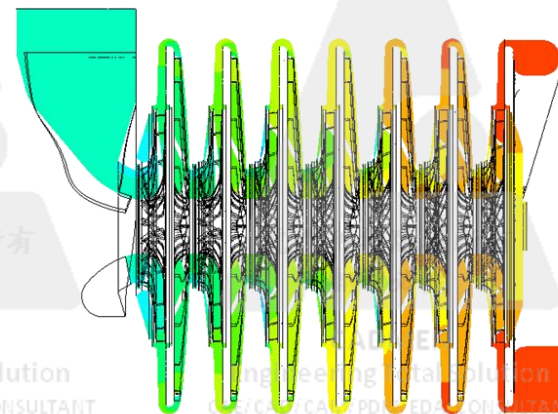
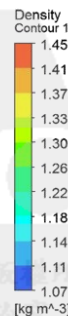
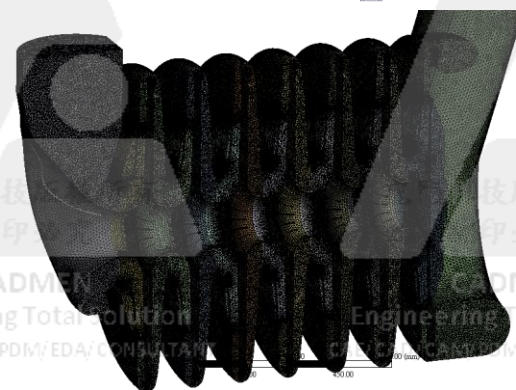
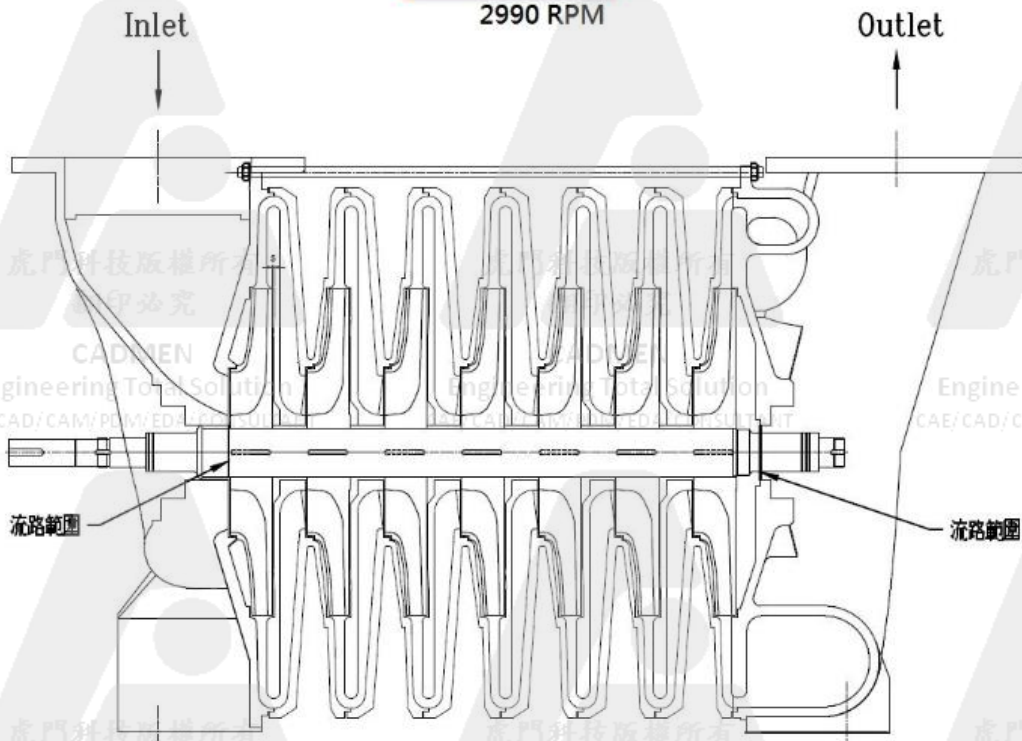


# 多級離心風機流場分析

利用ANSYS SCDM、Mesh、Fluent及CFD-Post，來分析此多段離心機之性能與流場現象。



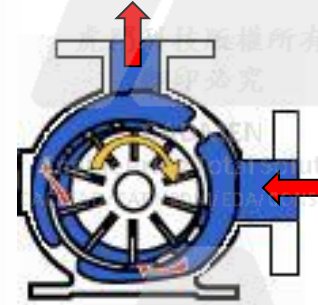
多段離心機 轉速: 3588 RPM 和 2990 RPM



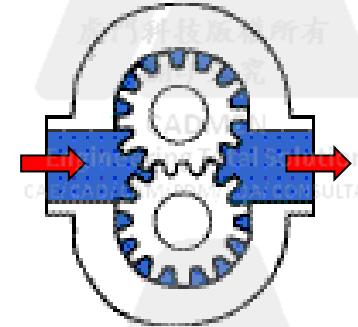
# Positive Displacement Pumps: Types

## Vane Pumps

- A rotor with sliding vanes positioned off-center in a housing. Pumping action is caused by the expanding and contracting volumes contained by the rotor, vanes and housing.



Vane pump



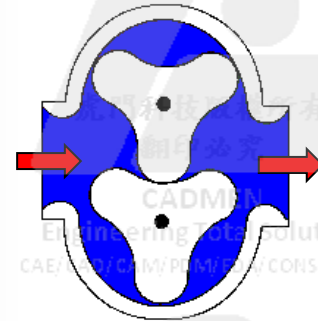
Gear pump

## Gear Pumps

- Gears rotate in opposite directions and mesh at the point in the housing which is between inlet and outlet. Liquid trapped between gear teeth and housing, and carried two separate paths.

## Lobe Pumps

- Operates like a gear pump. Lobed rotors spin in the same direction.



Lobe pump



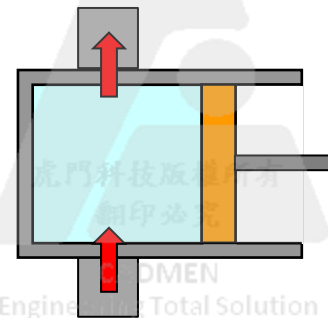
Gerotor pump

## Gerotor Pumps

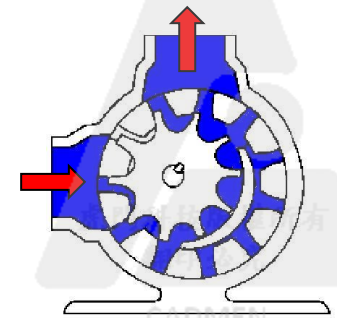
- Fluid drawn into and squeezed out of cavity formed by two rotating, intermeshing gears. Has smooth pumping action, and works well with wide range of fluid viscosities.

## Piston Pumps

- Reciprocating piston pressurizes fluid.



Piston pump



Crescent pump

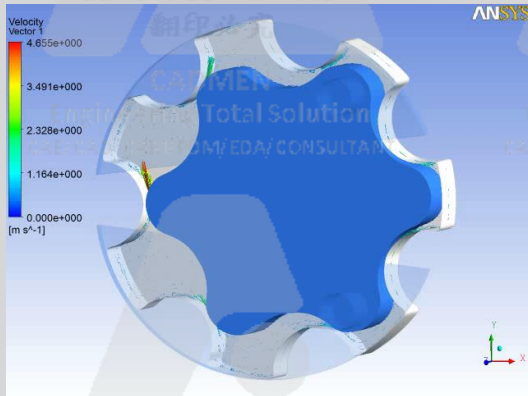
## Crescent Pumps

- Consists of two rotating gears separated by a crescent-shaped divider.

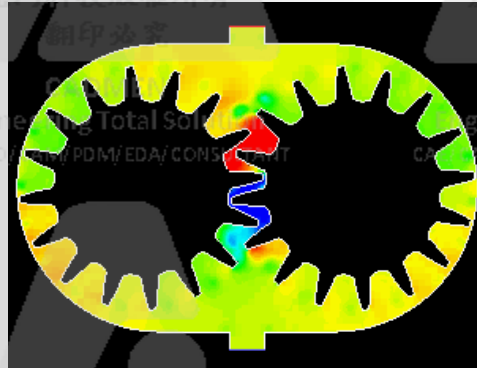
# 容積式壓縮機案例



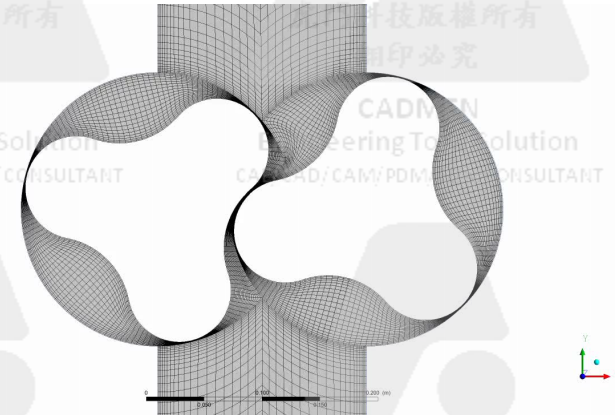
## Gerotor



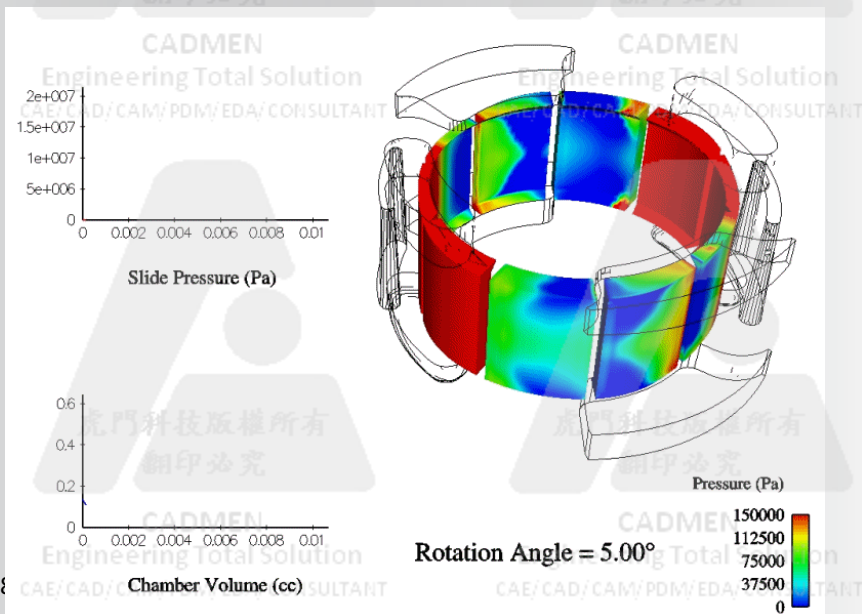
## Gear Pump



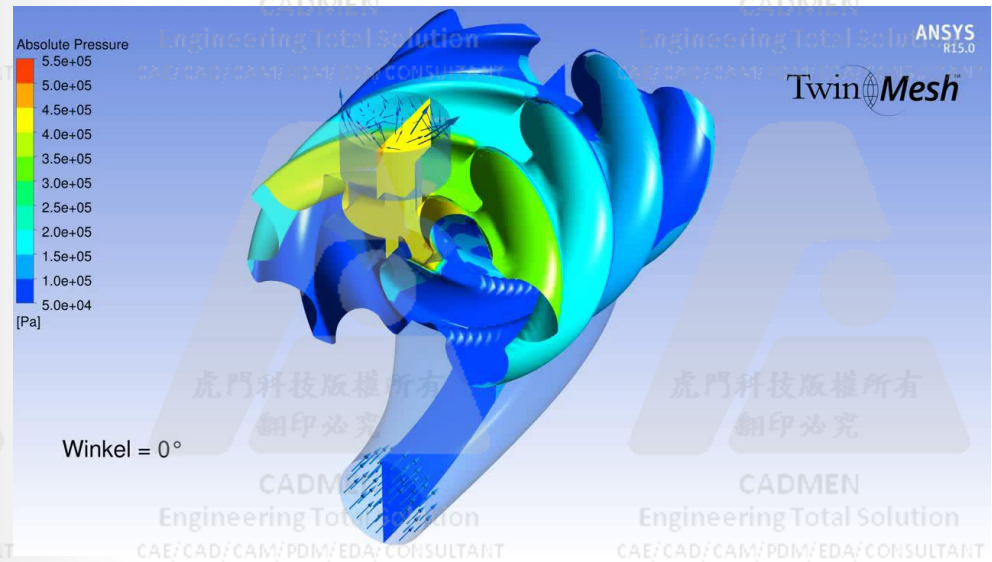
## Lobe Compressor



## Vane Pump



## Screw Compressor



# Twinmesh

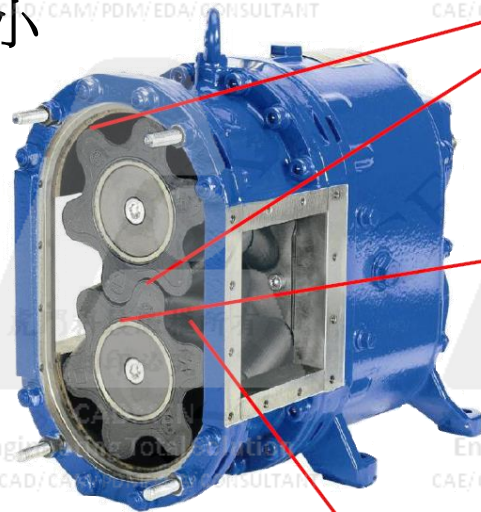
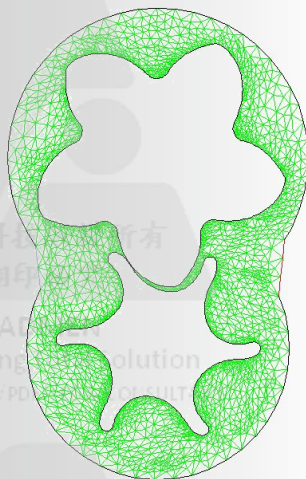
## Rotary Positive Displacement Machines

### 複雜幾何

- 連鎖轉子(通常會有螺旋狀)
- 公母轉子的間隙和轉子與外殼的間隙非常小
- 體積變化差異大

### 複雜的流體特性

- 空蝕(多相流)
- 可壓縮性
- 紊流
- 真實氣體特性
- ...



轉子間隙

出入口間隙

體積變化大

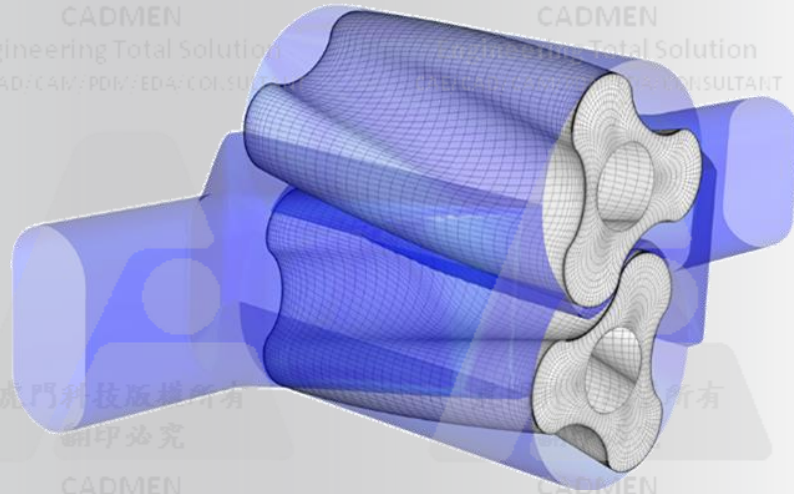
Grid (Time=8.0200e-03)

Oct 21, 2005

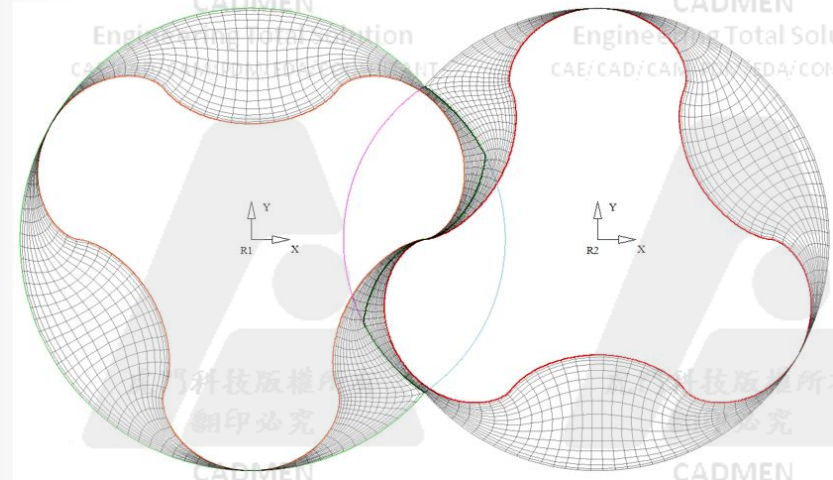
FLUENT 6.2 (2d, segregated, dynamesh, ske, unsteady)

考慮到前面的各種因素，網格的需求對於模擬相當重要，好的網格能夠使結果更趨近於真實現象

# Twinmesh Rotary Positive Displacement Machines



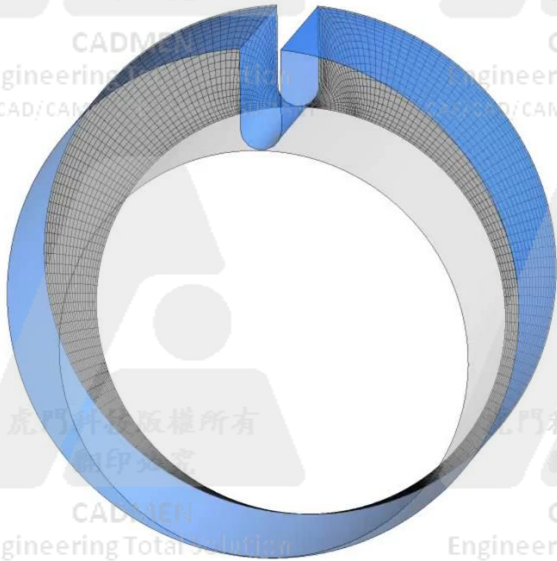
自動化生成高品質六面體網格



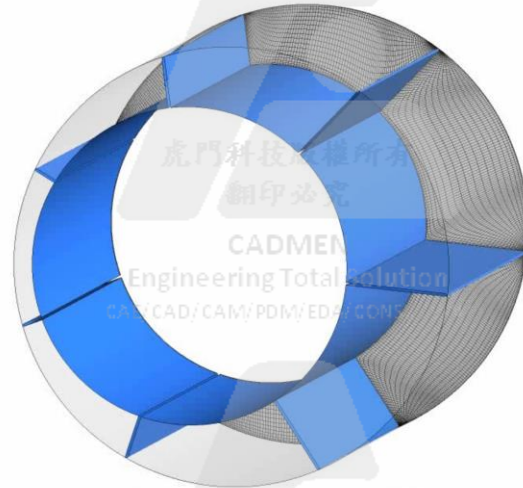
結構化的工作流程及介面



# Twinmesh Rotary Positive Displacement Machines



Rolling piston



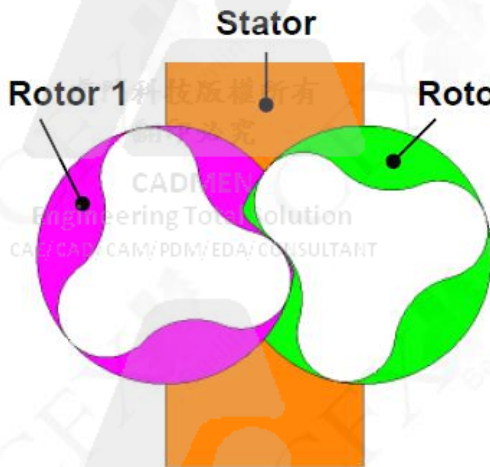
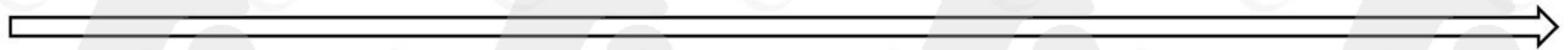
Vane pump

# TwinMesh

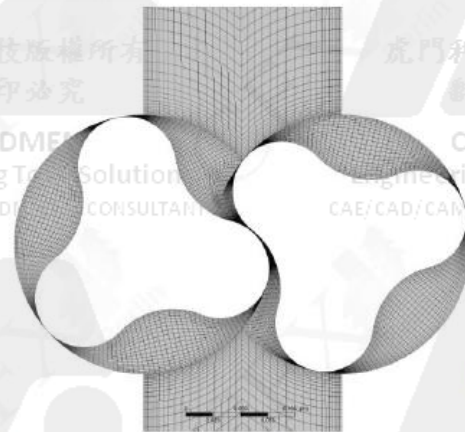
## Seven steps from CAD to Mesh

- TwinMesh is a novel software, developed by CFX Berlin Software GmbH which generates high-quality hexahedral meshes for the rotating parts of axis parallel rotary positive displacement machines.

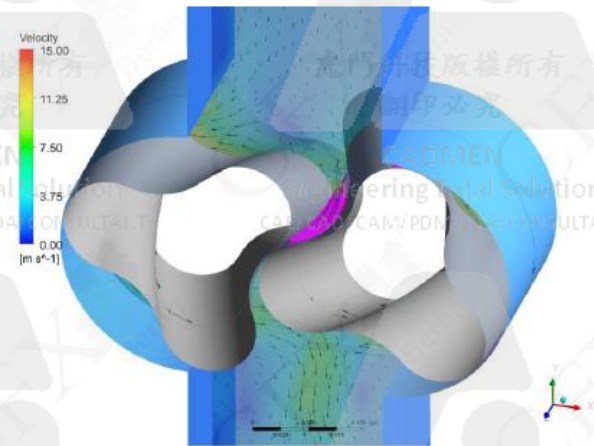
### Simulation Workflow



Split of the simulation domains into steady and rotating parts



Grid generation for the rotors with **TwinMesh** and for the Steady parts with ANSYS ICEM CFD or ANSYS Meshing



Numerical calculation with ANSYS CFX



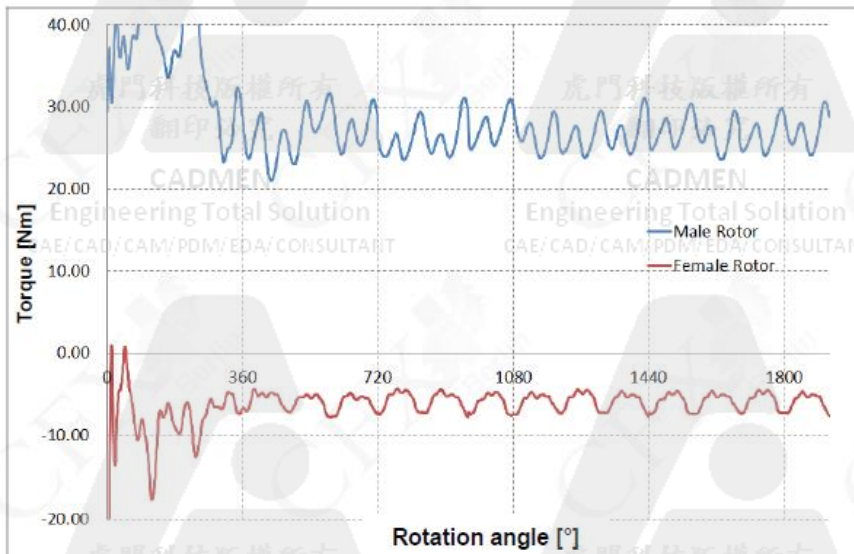
# Simulation results

## Screw compressor

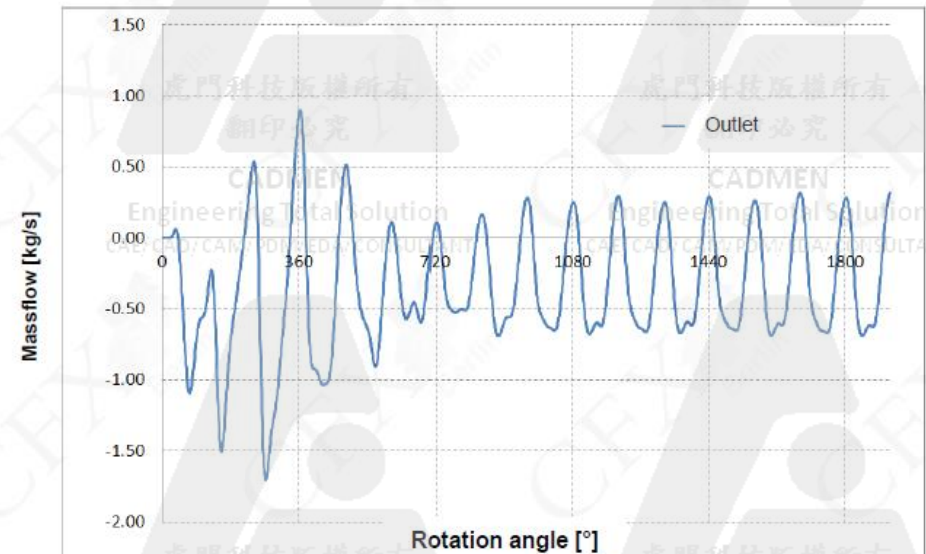
- General analyses (Torque, Power and massflow)

- Power: 41.2 kW
- Averaged volume flow: 890 m<sup>3</sup>/h

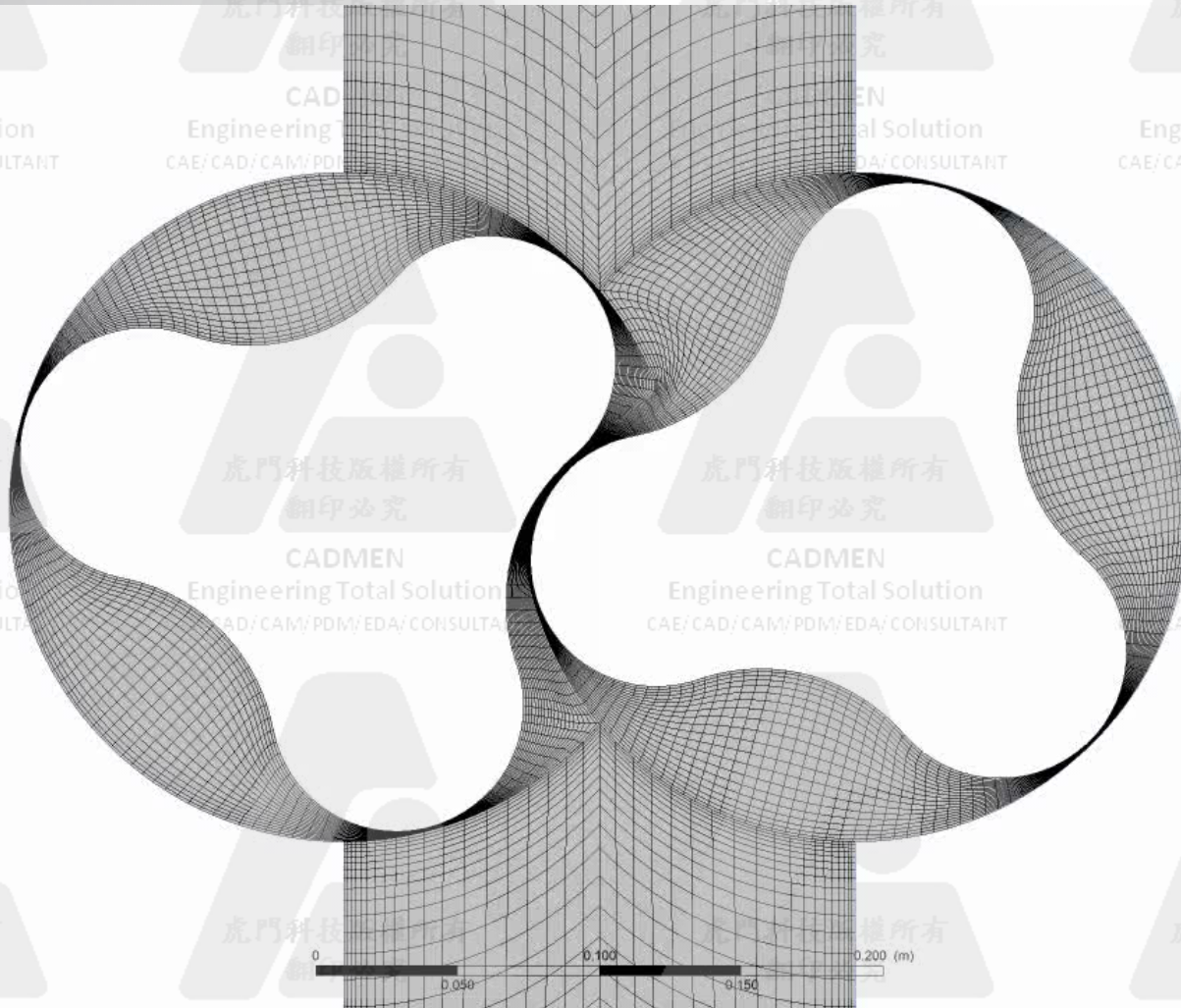
### Torque



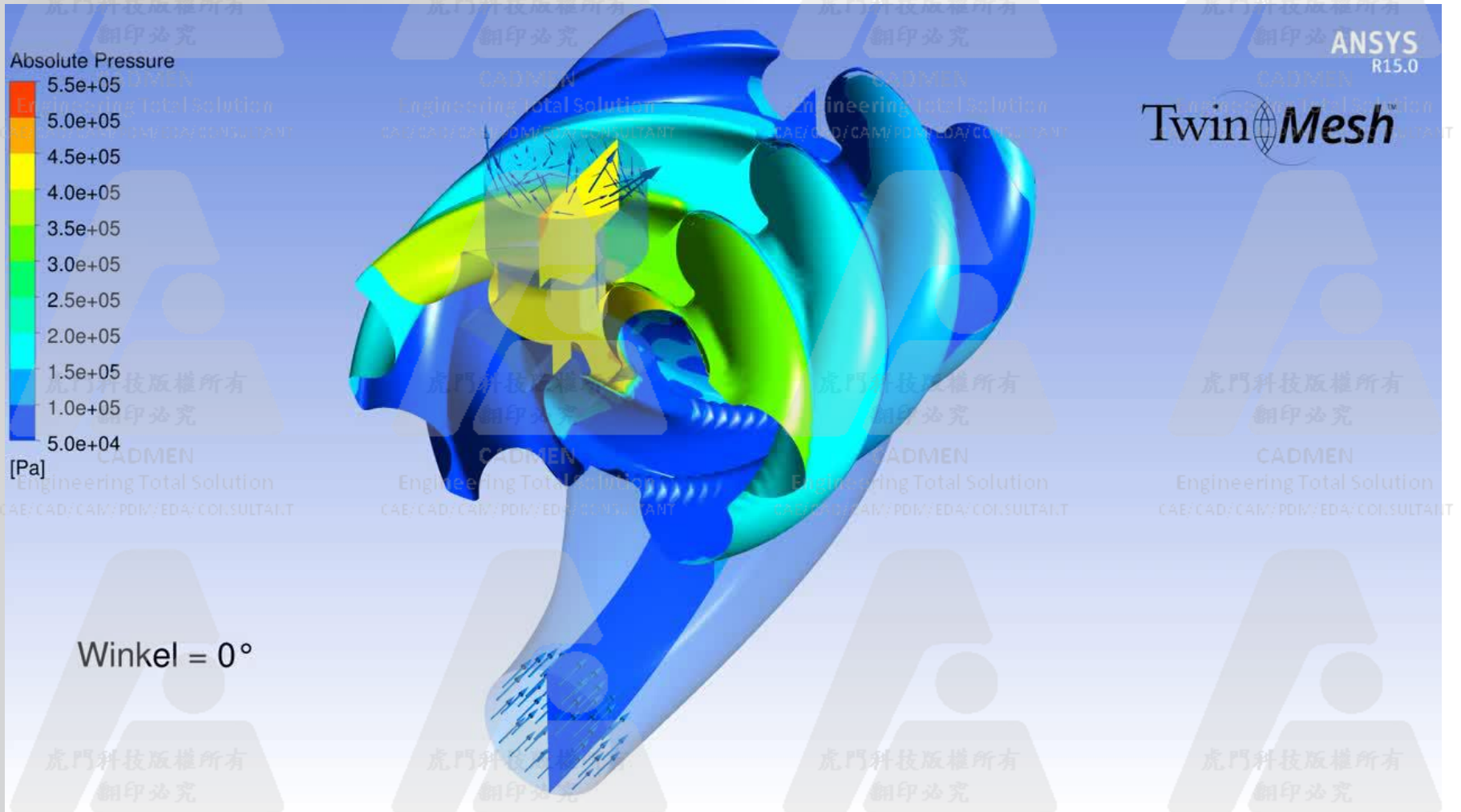
### Massflow



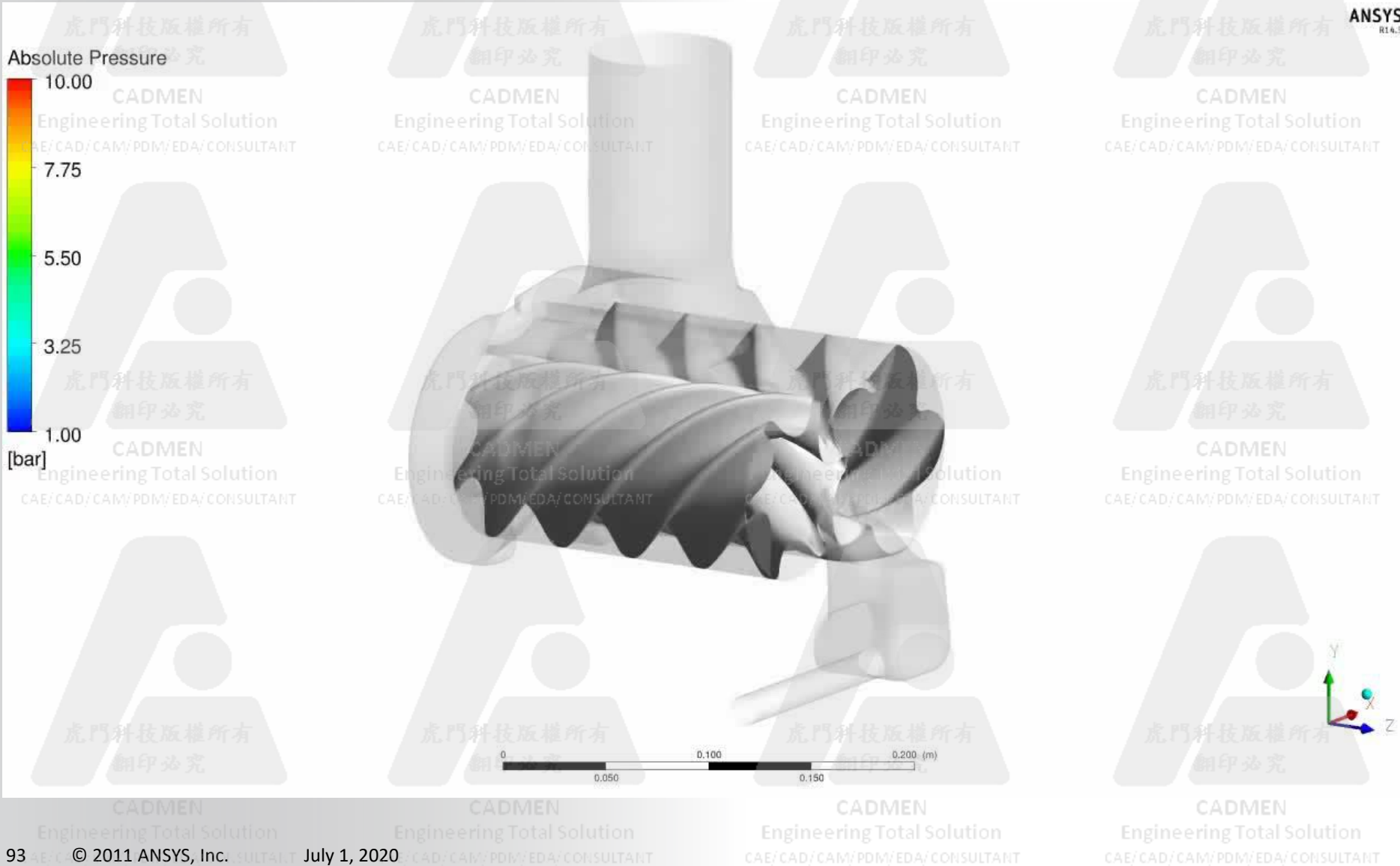
# Results-Mesh



# Results-Pressure

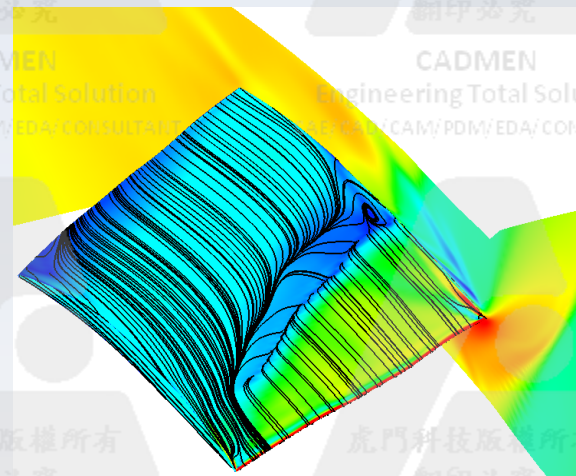


# Results-Animation



# 高效能操作平台 ANSYS Workbench

耦合分析應用

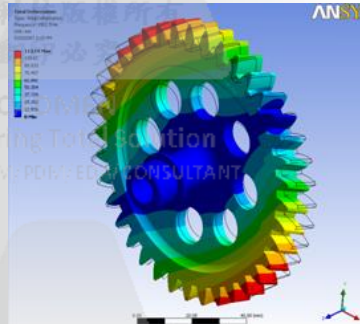


# ANSYS 結構分析 for 旋轉機械

## ANSYS Mechanical

- 結構強度、變形
- 震動噪音
- 模態、頻譜
- 疲勞壽命
- 轉子動力
- 多體動力
- 流固耦合分析

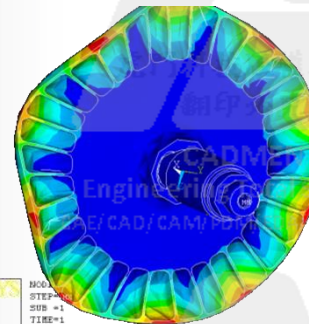
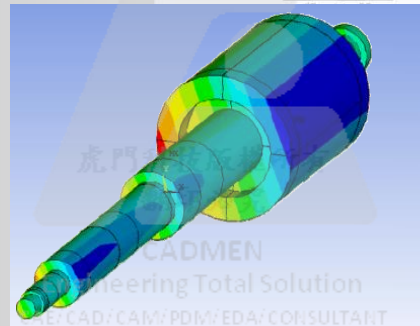
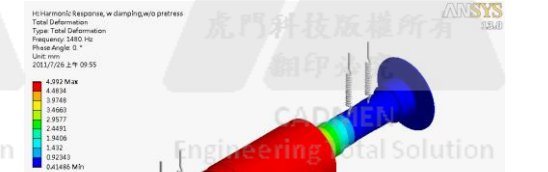
### Static Structure



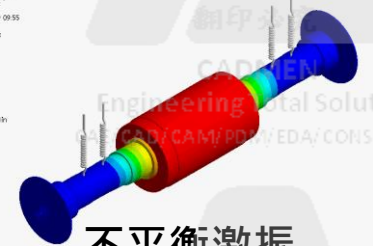
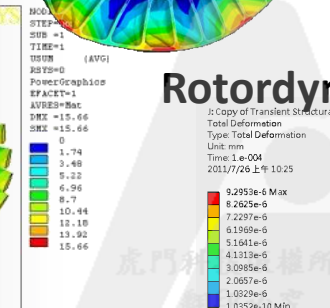
Rotational vibration deformation in a gear



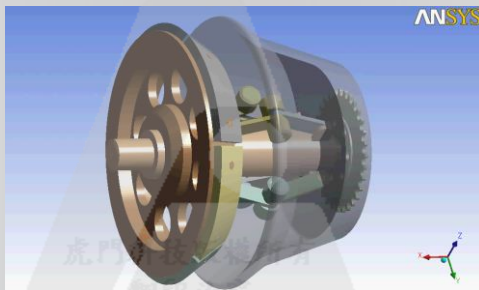
### Industrial fan (Venti Oelde)



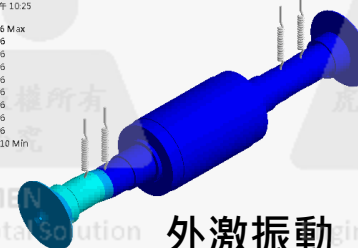
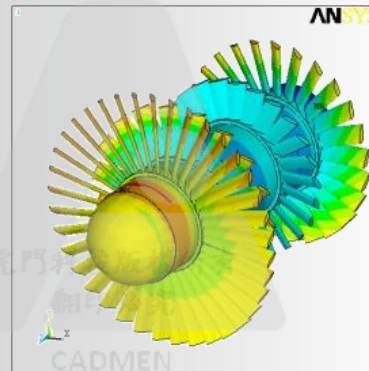
### Rotordynamics



不平衡激振



### Rigid Body Dynamic

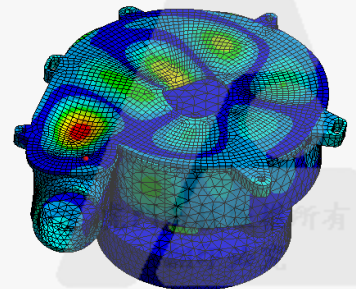
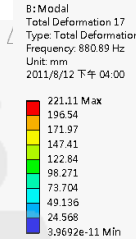
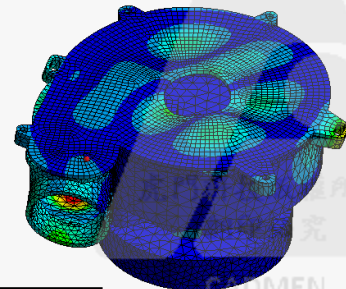
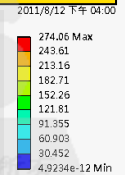
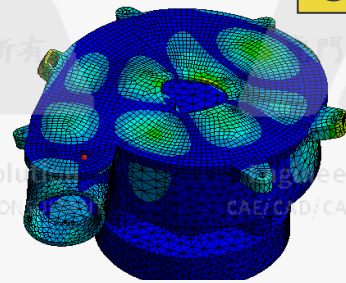
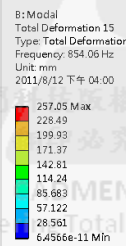
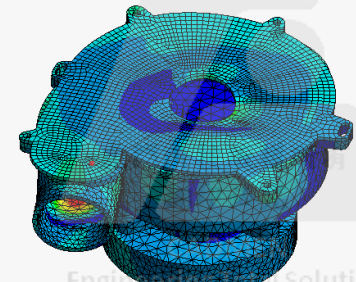
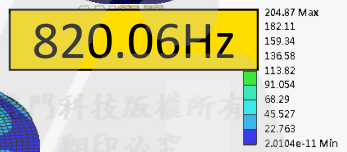
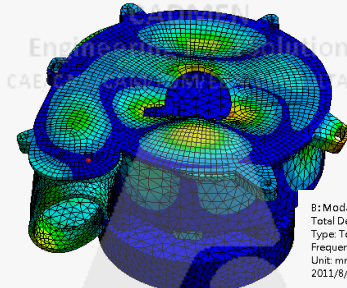
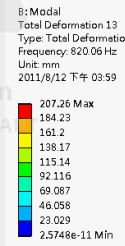
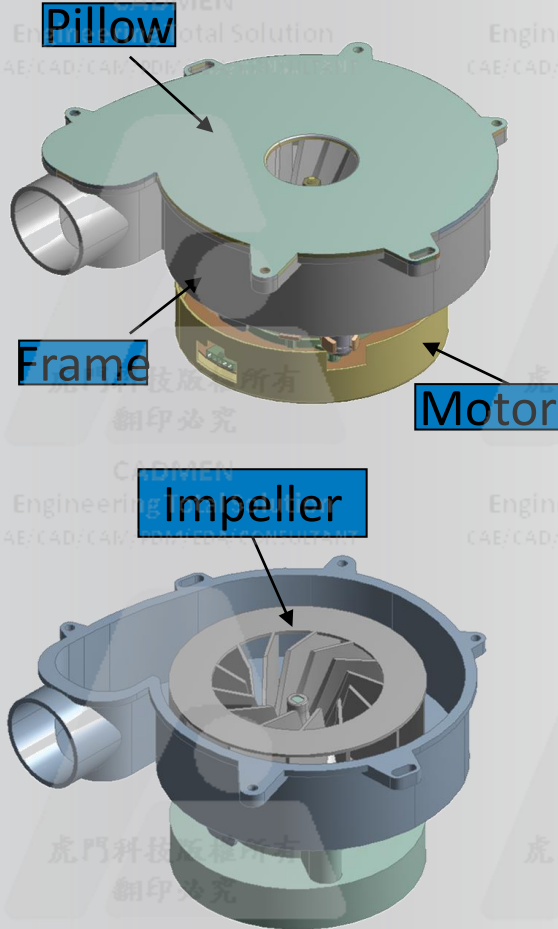


外激振動

# 振動噪音預測

風機在工作頻率(845Hz)下之振動行為。

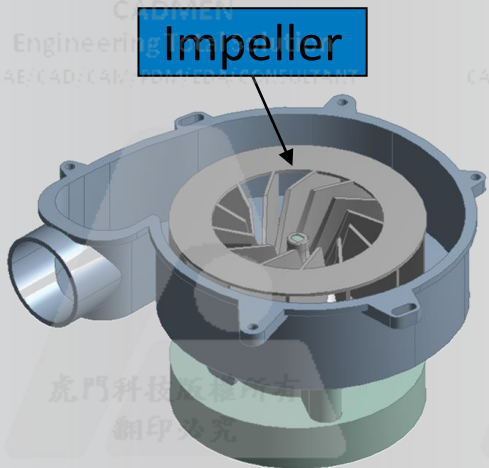
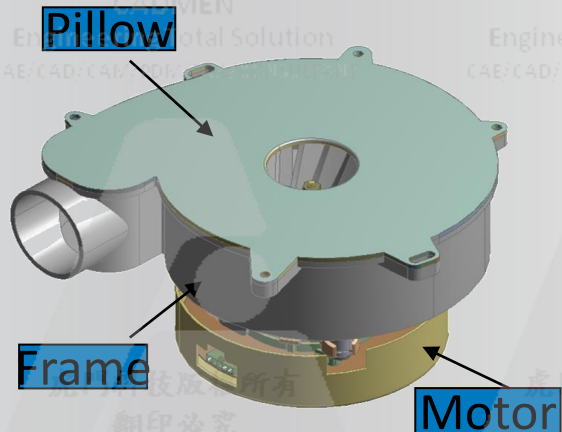
- 模態分析結果發現工作頻率(845Hz)附近之共振頻率與振型



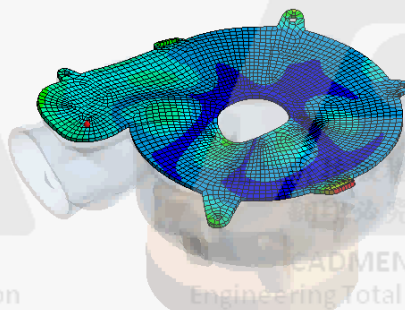
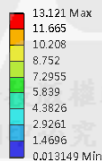
# 振動噪音預測

風機在工作頻率(845Hz)下之振動行為。

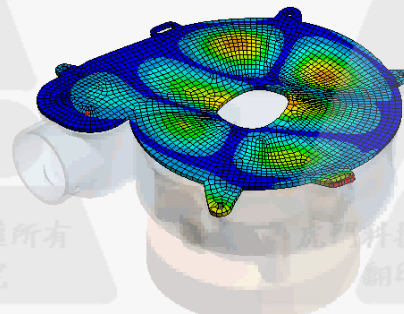
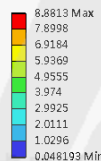
工作頻率如果在845Hz時因外力作用在模態峰值處，容易激振出周遭的共振頻率，最大為移13.121 mm，如果調整頻率為740Hz，遠離845Hz左右之共振頻率，最大位移降低到8.8813 mm。



C: Harmonic Response  
Total Deformation 2  
Type: Total Deformation  
Frequency: 845 Hz  
Phase Angle: -24.6°  
Unit: mm  
2011/8/12 下午 03:55



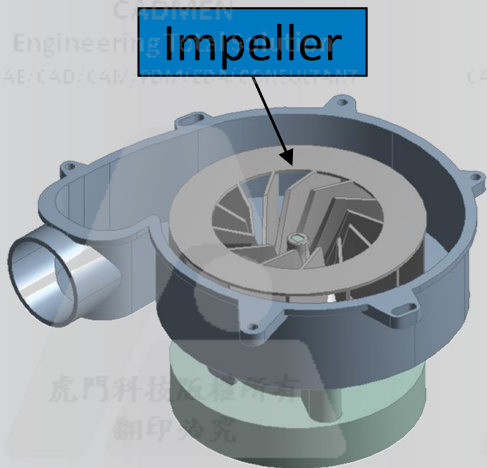
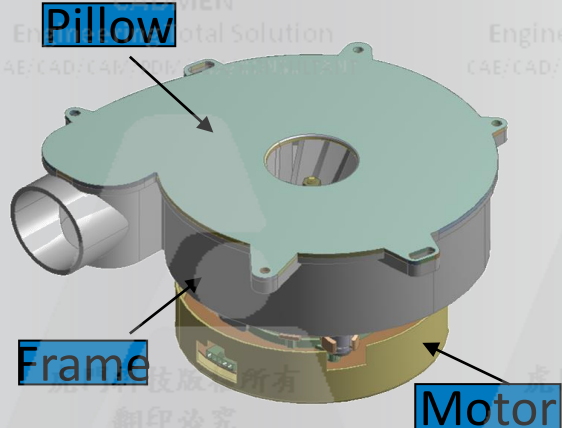
Frequency: /40 Hz  
Phase Angle: 21.9°  
Unit: mm  
2011/8/12 下午 03:56



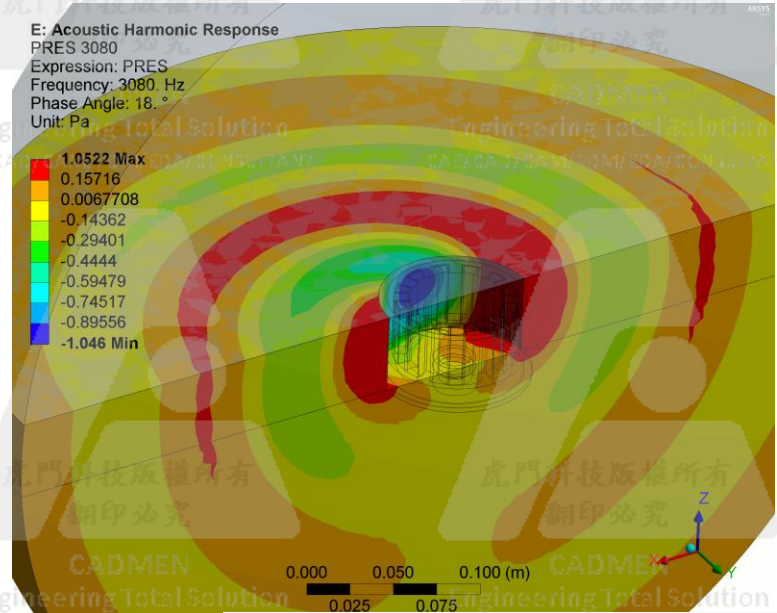


# 振動噪音預測

風機在工作頻率(845Hz)下之振動行為。



使用Harmonic進行噪音分析。



| E |                   |
|---|-------------------|
| 1 | Harmonic Response |
| 2 | Engineering Data  |
| 3 | Geometry          |
| 4 | Model             |
| 5 | Setup             |
| 6 | Solution          |
| 7 | Results           |
| 8 | Parameters        |

Acoustic Harmonic Response

D: Harmonic Response  
 Total Deformation  
 Type: Total Deformation  
 Frequency: 1540. Hz  
 Phase Angle: 360. °  
 Unit: m  
 2013/9/30 下午 03:47

Legend: 4.874e-10 Max, 4.3394e-10, 3.7926e-10, 3.2521e-10, 2.7115e-10, 2.1709e-10, 1.6303e-10, 1.0896e-10, 5.4901e-11, 8.386e-13 Min

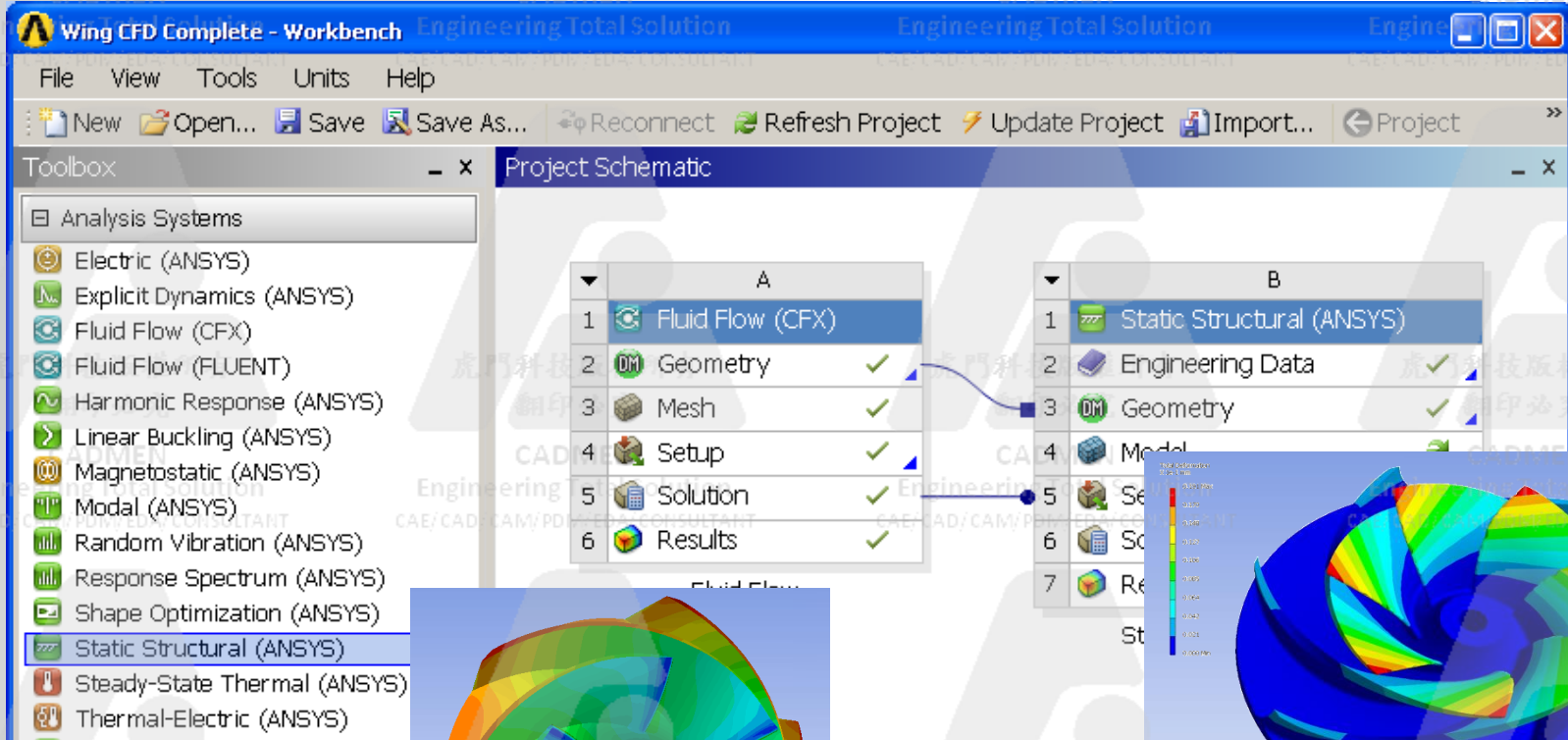
A 3D model of the impeller showing total deformation. The color scale ranges from blue (low deformation) to red (high deformation). A coordinate system (X, Y, Z) is visible at the bottom right.

# 簡易的耦合分析程序控制

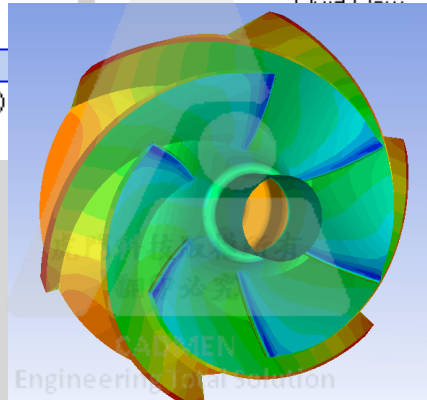


## Engineered Scalability

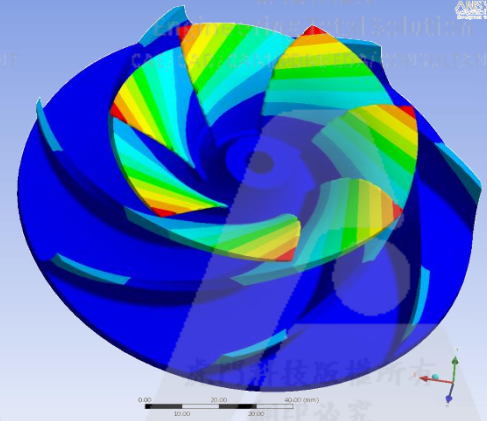
Flexible model setup enabled by the scalable, adaptive architecture of ANSYS Workbench



Pressure contours on a pump impeller

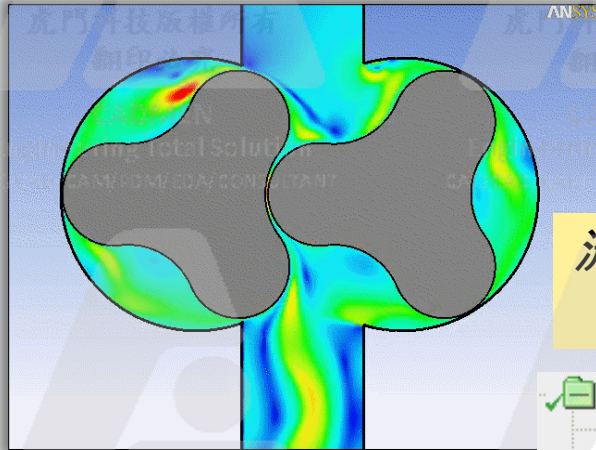


Contours of total deformation on a pump impeller in a coupled structural and flow analysis

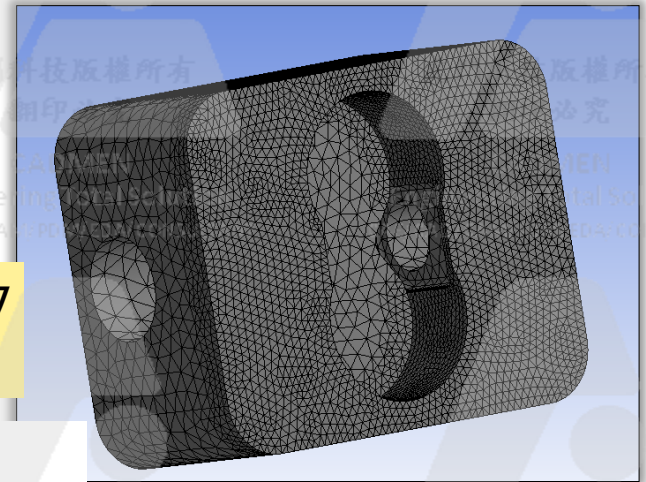


# 耦合分析 - 魯式泵面對更高操作條件時結構強度分析

## Roots Pump



流體作用於壁面之壓力  
Load Transfer



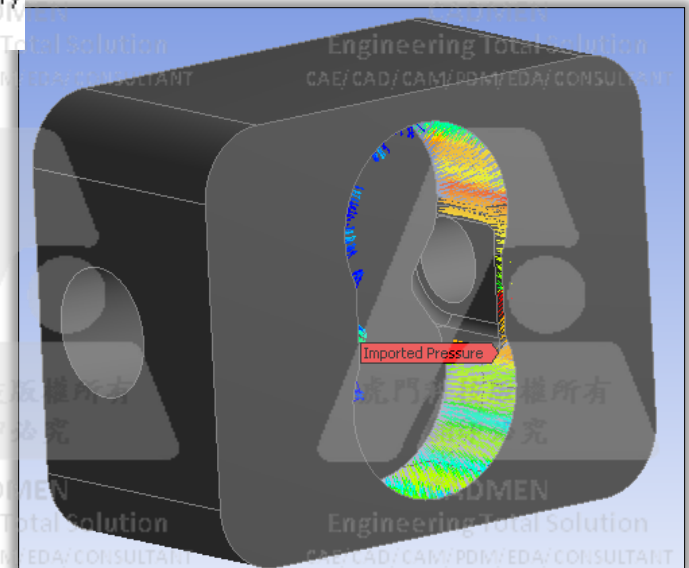
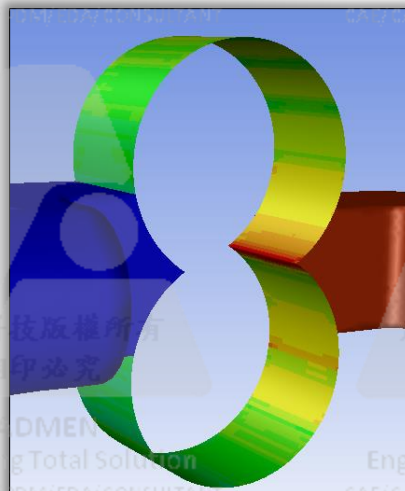
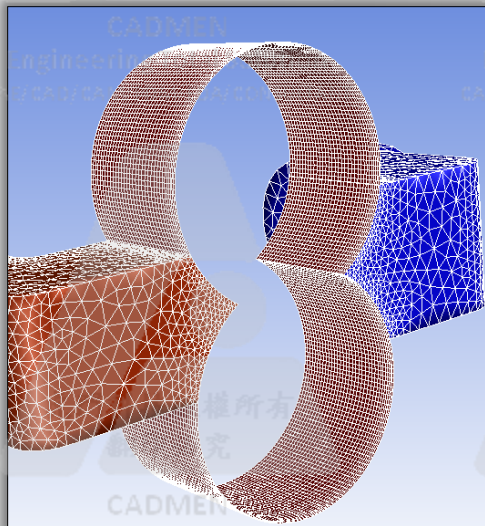
CFD



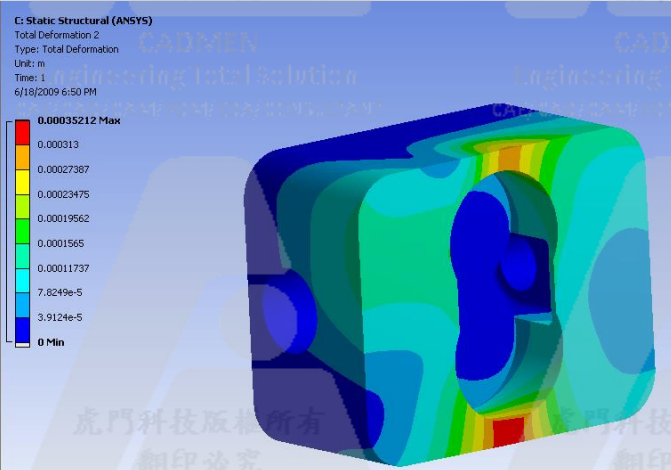
- Static Structural (C5)
  - Analysis Settings
  - Fixed Support
  - Imported Load (Solution)
  - Imported Pressure
  - Imported Load Transfer Summary
- Solution (C6)



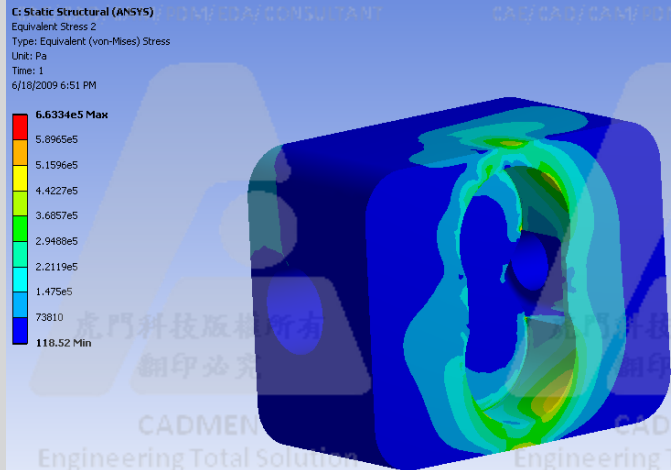
Structural



## Total Deformation

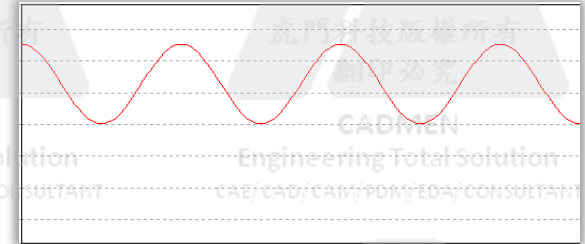


## Equivalent (Von-Mises) Stress

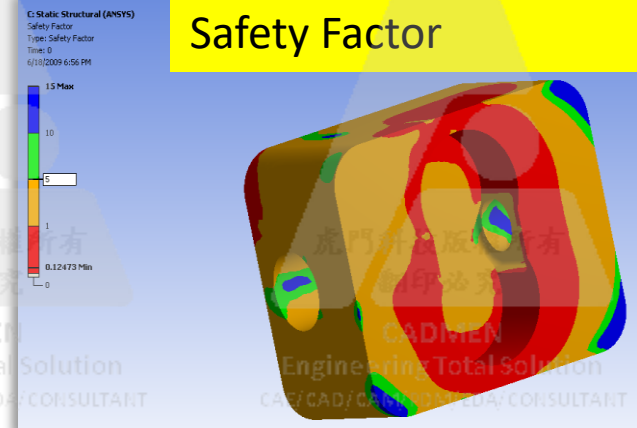


## Fatigue Analysis

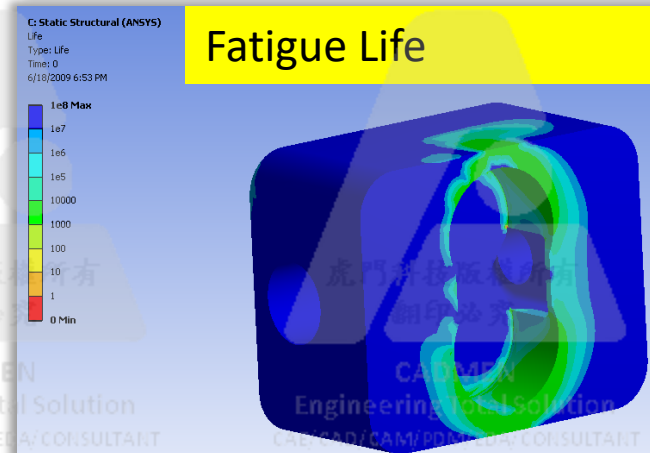
- Stress Based
- Fatigue Life
- Constant Amplitude Load – Zero Based



## Safety Factor



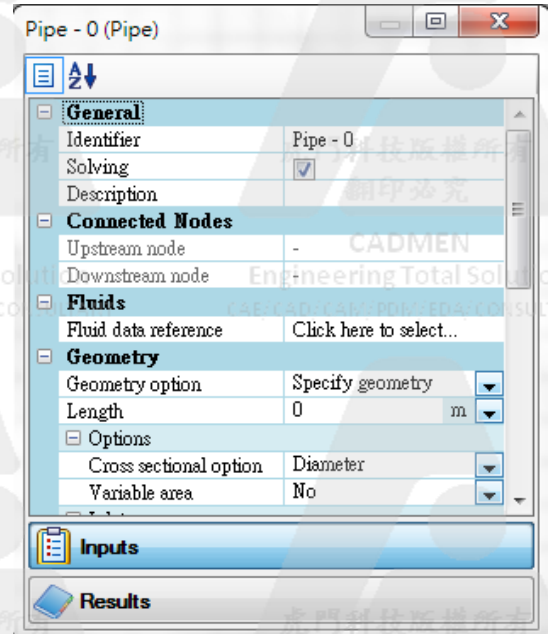
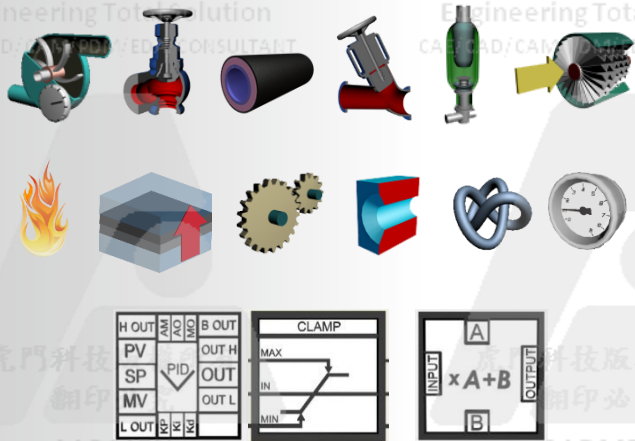
## Fatigue Life



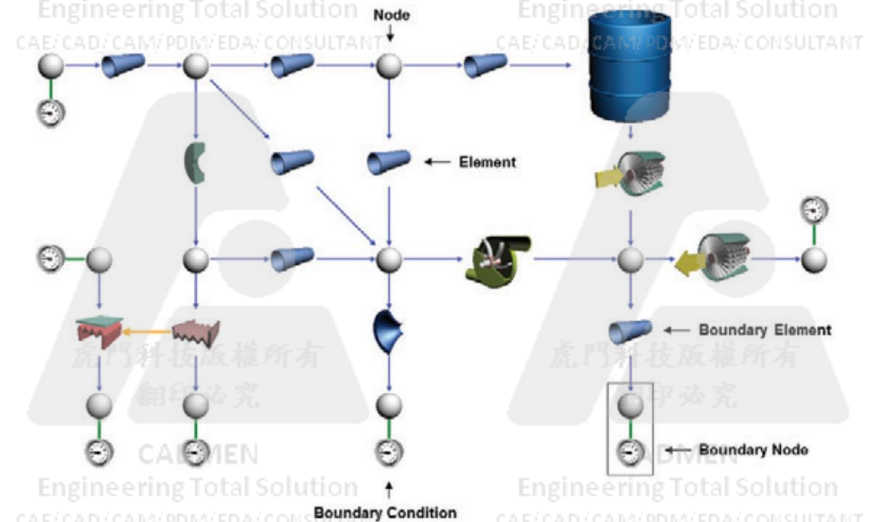
# FlowNex於冷卻散熱系統性CFD分析技術



- Library of components defined through lumped parameters
  - Pipes, Bends
  - Fans, Pumps
  - Heat exchangers
  - Nozzles
  - Valves
  - Accumulator
  - Controllers, Scripts
  - Turbines & combustor
  - Gear pumps



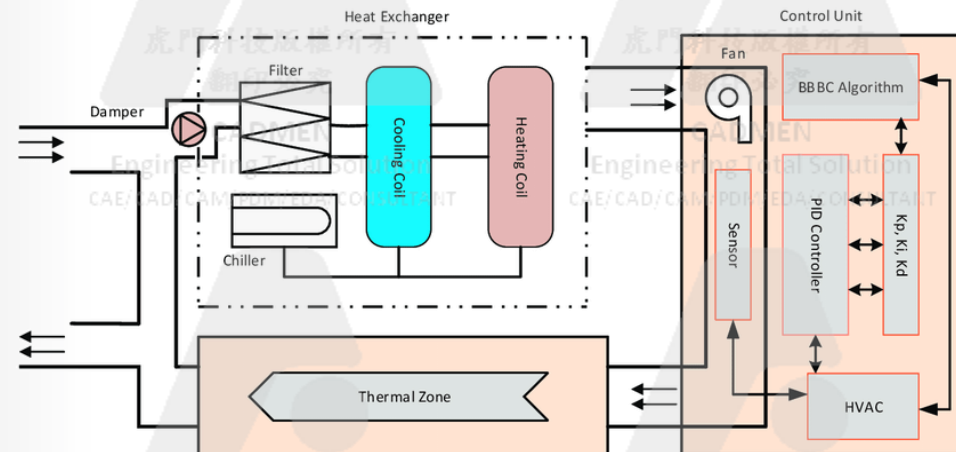
- Components are connected in a network
  - Flow distribution
  - Heat transfer
  - Masstransfer



- 若採三維CFD軟體進行全冷卻系統分析，擴日費時且極消耗成本
- 將系統內設備元件視為簡易流程方塊，進行熱力學流程分析，係為可行且合理的操作方式
- 採用一維CFD軟體(Flownex®SE)分析，可達到成本效率跟精確度的平衡



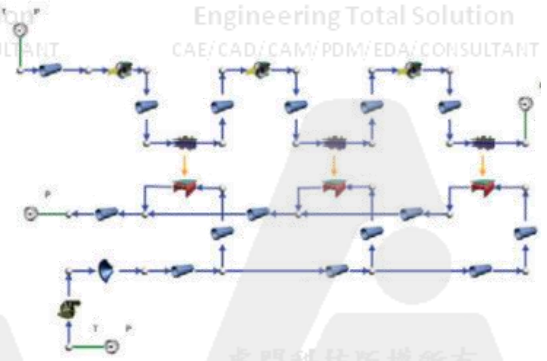
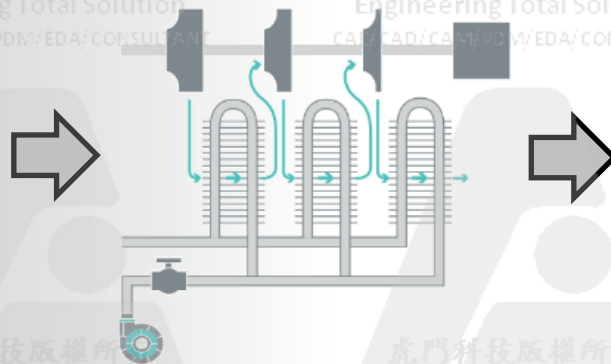
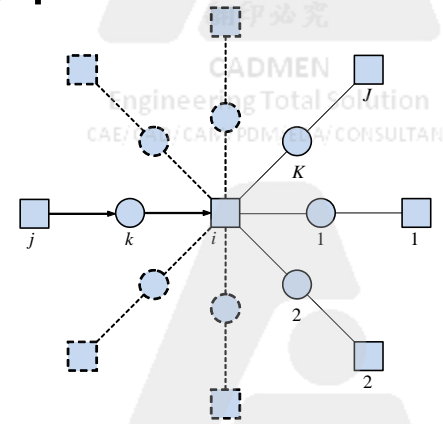
[inthebigroom.com/](http://inthebigroom.com/)



Creative Commons Attribution 4.0 International

# Flownex的功用

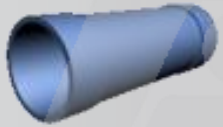
- 大規模廠務熱流系統模擬分析，節省時間與成本
- 利用流體網路模型架構，快速組成複雜系統
- 採用符合工程實況的流體資料庫，如冷媒
- 求解系統及元件壓力、溫度、流量.....的分佈
- 可執行參數設計、敏感分析及優化功能
- 可與其它軟體如ANSYS與Excel....作耦合分析或功能整合



# Flownex熱流元件庫

□ 管件、容器、閥門、旋轉機械、流阻、噴嘴、熱交換器、熱傳模型、氣液過濾、流量計機械傳動.....等

Pipe



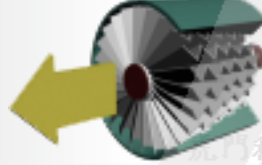
Two-phase tank



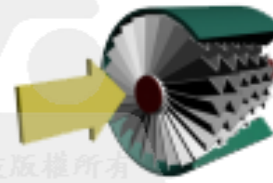
Valve



Turbine



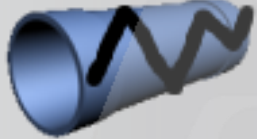
Compressor



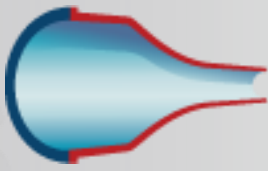
Centrifugal pump



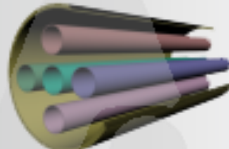
Flow resistance



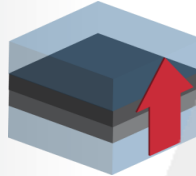
Nozzle



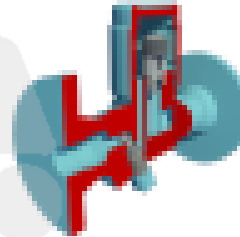
Shell-tube heat exchanger



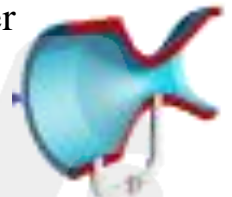
Composite heat transfer



Steam/water trap



Venturi flow meter

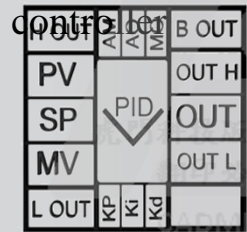




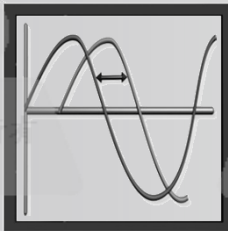
# Flownex控制元件庫

□ 控制器、延遲器、脈衝器、波型器、函數器、開關、計數器、邏輯控制、比較器、積分器和輸入器.....等

PID



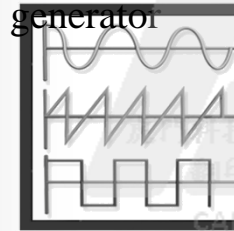
Time delay



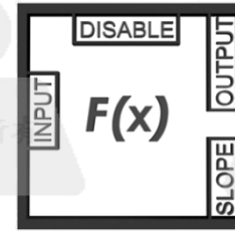
Pulse timer



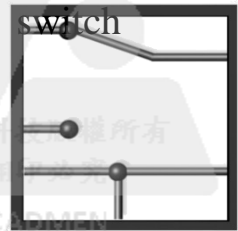
Wave generator



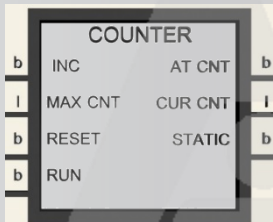
Function generator



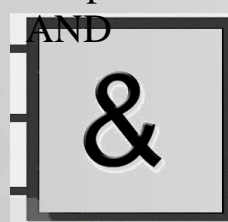
Toggle switch



Counter



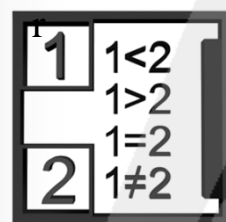
4 input AND



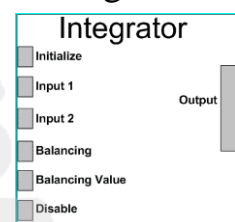
4 input OR



Comparator



Integrator



DCS



## □用以觀察參數的暫態過程或程度上的控制

Push button



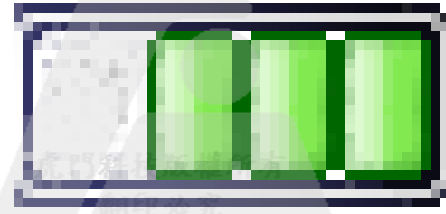
Dial



Track bar



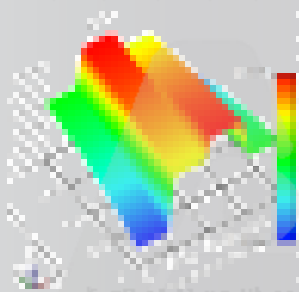
Progress bar



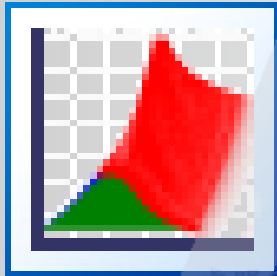
2D increment



3D increment



Fluid property graph



Scripting tool



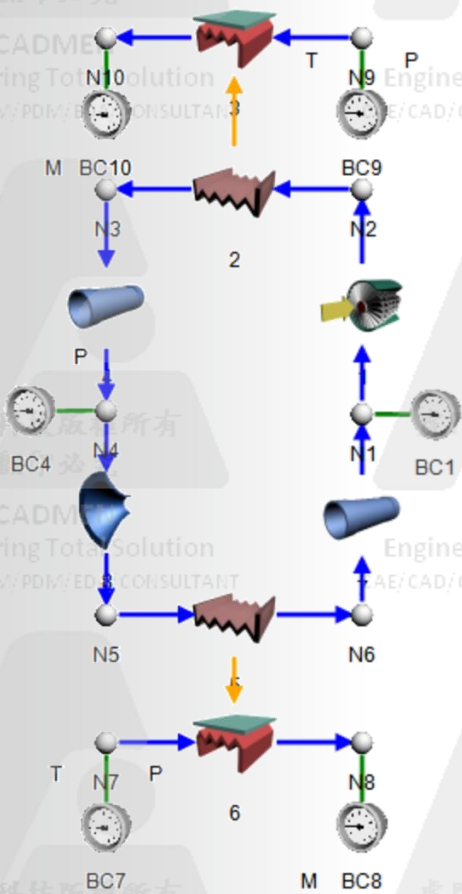
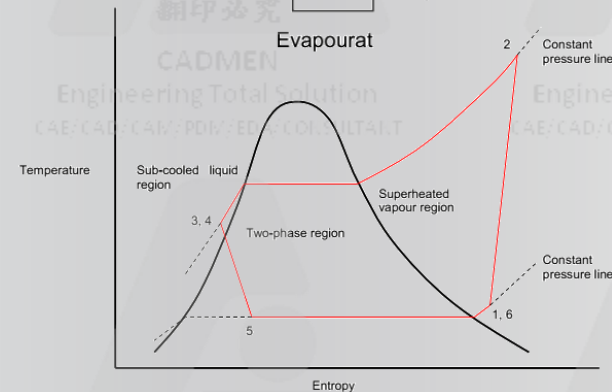
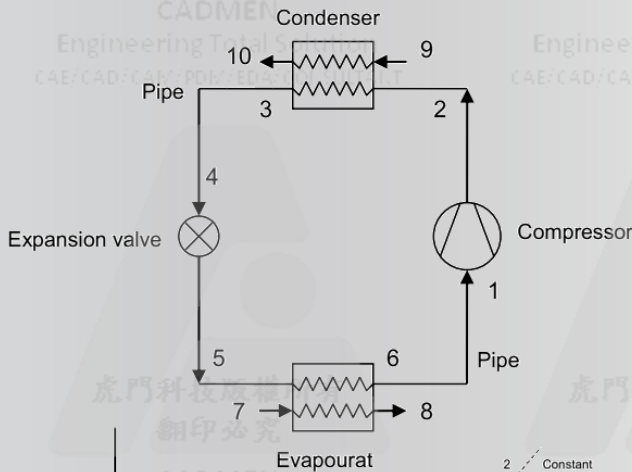
Pie graph



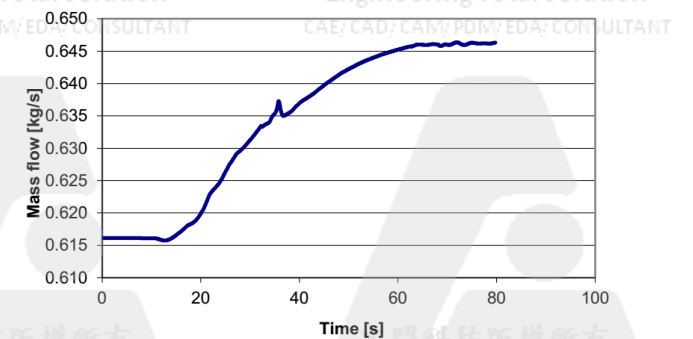
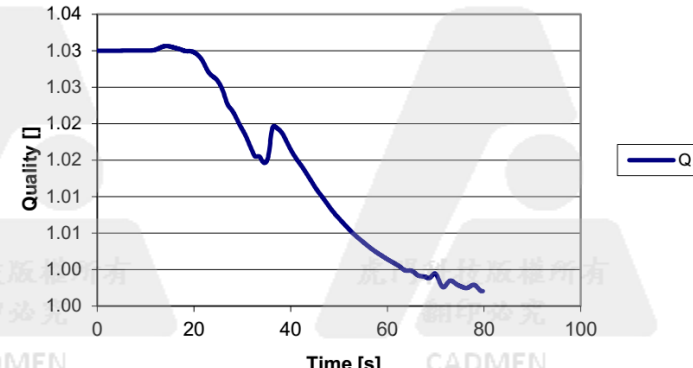
Toggle button



# 範例：蒸氣壓縮冷凍循環



不同階段時的流體的氣液比  
及狀態改變下的流量變化



虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

- 由於大陸5G雲計算的強力推動，華為將擴建計算中心規模
- 於廊坊、深圳、西安等地的計算中心將有著更多的伺服器房
- Flownex用於分析伺服器房的冷卻系統管線規劃，及一三維耦合分析

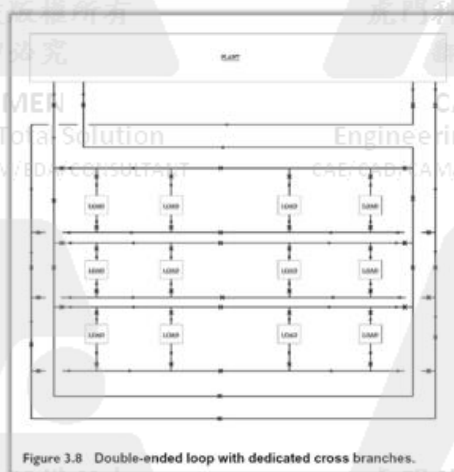
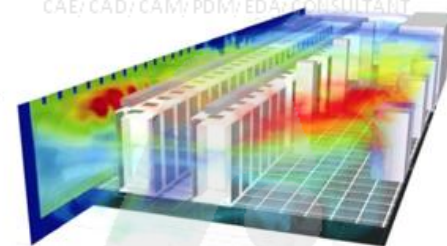
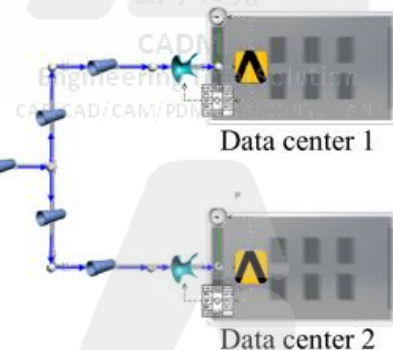
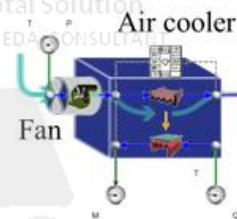


Figure 3.8 Double-ended loop with dedicated cross branches.



虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

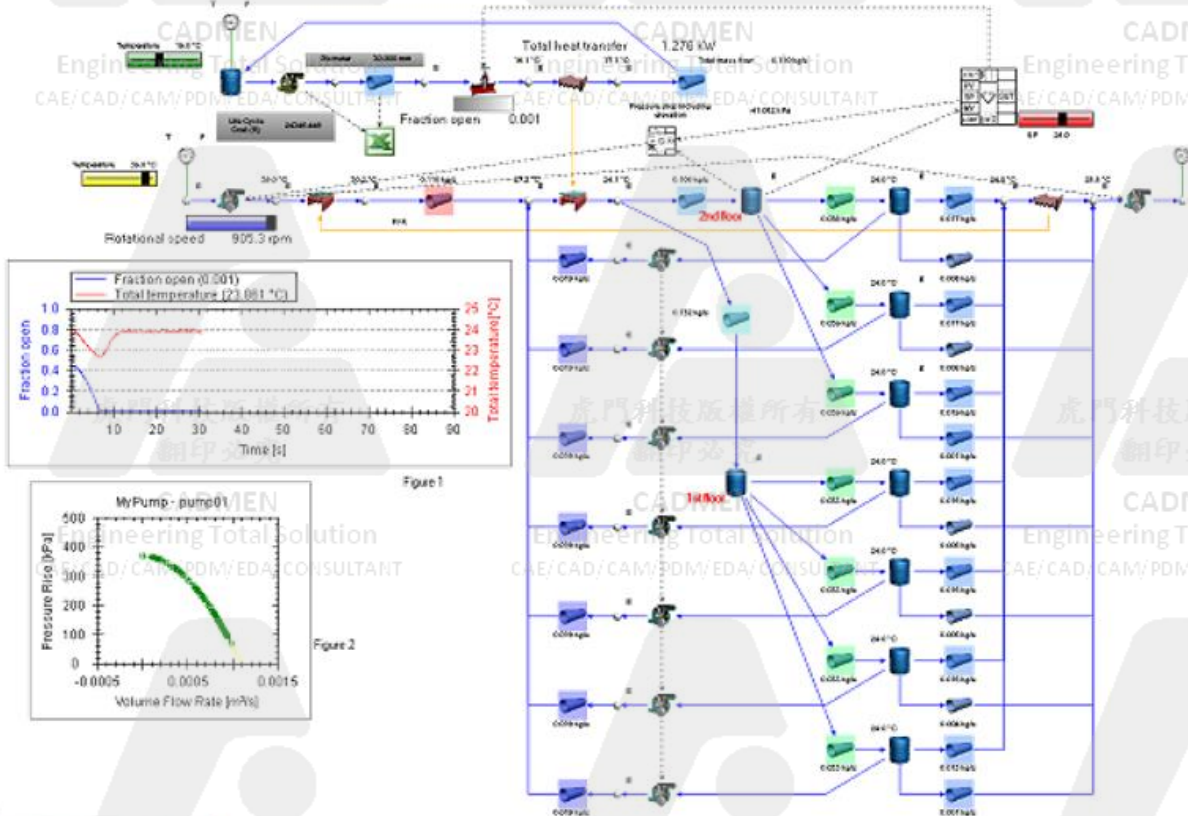
# 圖書館空調系統

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

翻印必究



- 7間房的空調系統模擬
- 冷卻水及換氣系統組成
- 熱交換器在冷卻水的中央處，另一只在換氣系統的進排氣端中間
- 冷卻水流量由pump轉速決定
- 換氣系統由PID控制器控制其運作

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

翻印必究

株式会社計算力学研究センター  
Research Center of Computational Mechanics, Inc

虎門科技版權所有  
翻印必究

翻印必究

# 電池散熱片應用

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CAD MEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

**CAD MEN**  
Taiwan Auto-Design Co.  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

200 x 150 x 1 (mm<sup>3</sup>)

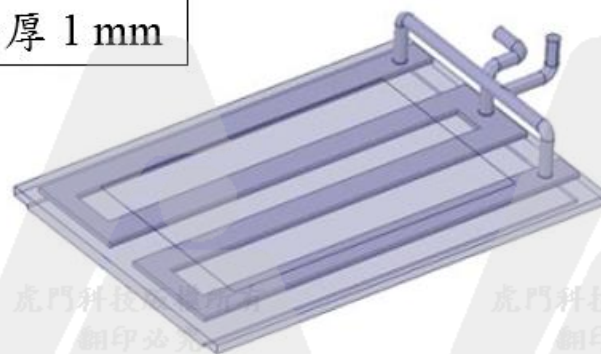
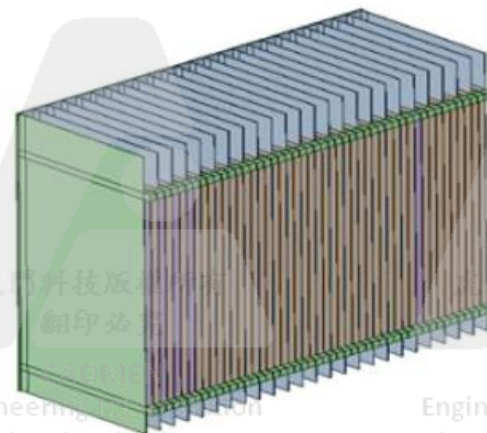


電池散熱量 0.03 kW

200 x 150 x 10 (mm<sup>3</sup>)

570 x 12 x 390 (mm<sup>3</sup>), 導熱層厚 1 mm

冷卻水  
Mass flow rate = 0.005 kg/s  
T = 18°C



# 電池散熱片應用

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

電池1

電池2

電池3

導熱片

水冷片

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution

CADMEN  
Engineering Total Solution

CADMEN  
Engineering Total Solution

CADMEN  
Engineering Total Solution

# 電池散熱片應用

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

**CADMEN**  
Taiwan Auto-Design Co.

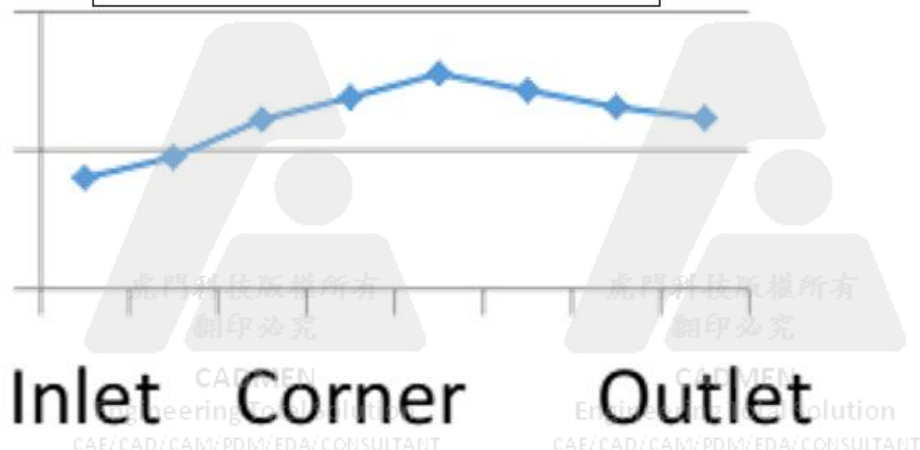
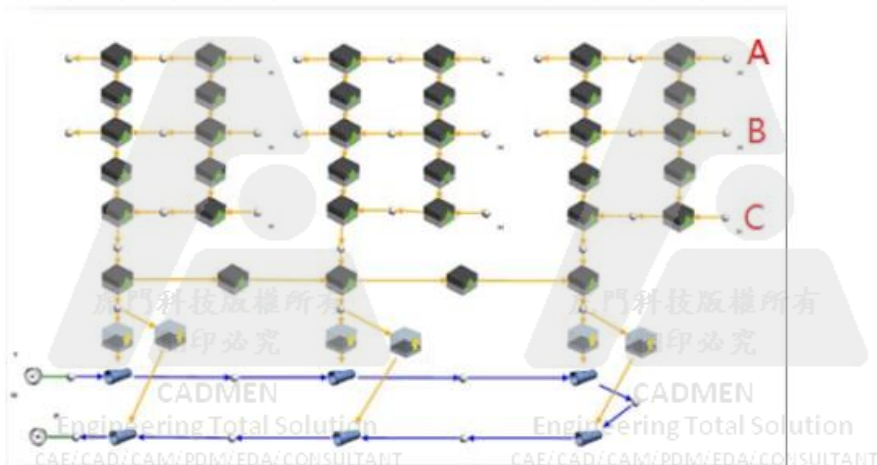
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

冷卻水管沿途溫度 (°C)



| 電池表面溫度(°C) | 電池1  | 電池2  | 電池3  |
|------------|------|------|------|
| A(頂部段)     | 27.3 | 29.2 | 30.1 |
| B(中部段)     | 25.5 | 27.4 | 28.4 |
| C(底部段)     | 22   | 23.9 | 24.8 |



虎門科技版權所有  
翻印必究

CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

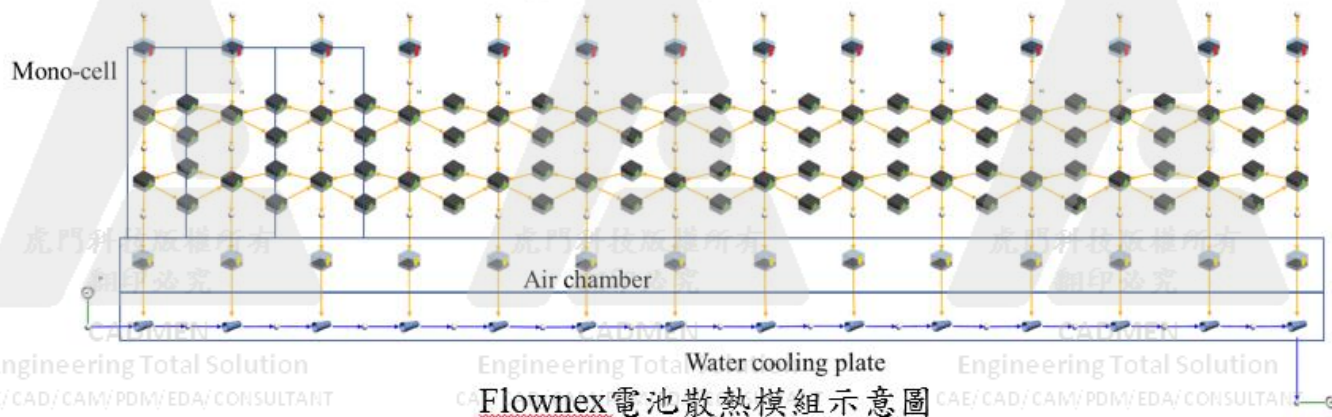
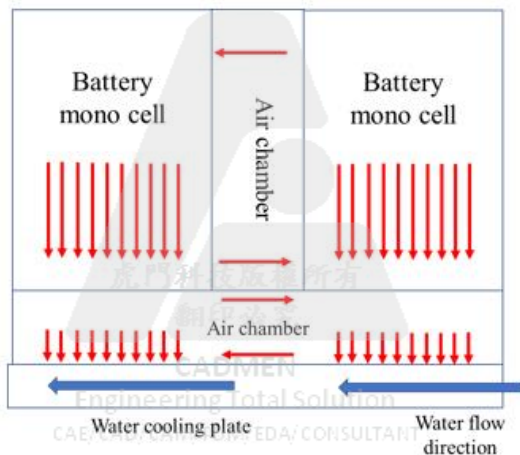
Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN  
Taiwan Auto-Design Co.

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT



- 一維的模型範例，上面有14個方芯串連形成一個模組，各方芯之間的交互熱傳導由中間的熱管進行，再傳導至下方的均溫板，均溫板與電池皆有交互熱傳，然後再經由均溫板下方的水冷板進行對流散熱
- 於0.2 kg/s條件下，最高溫度約23.31 °C

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CADMEN

Engineering Total Solution

CADMEN

Engineering Total Solution

CADMEN

Engineering Total Solution

CADMEN

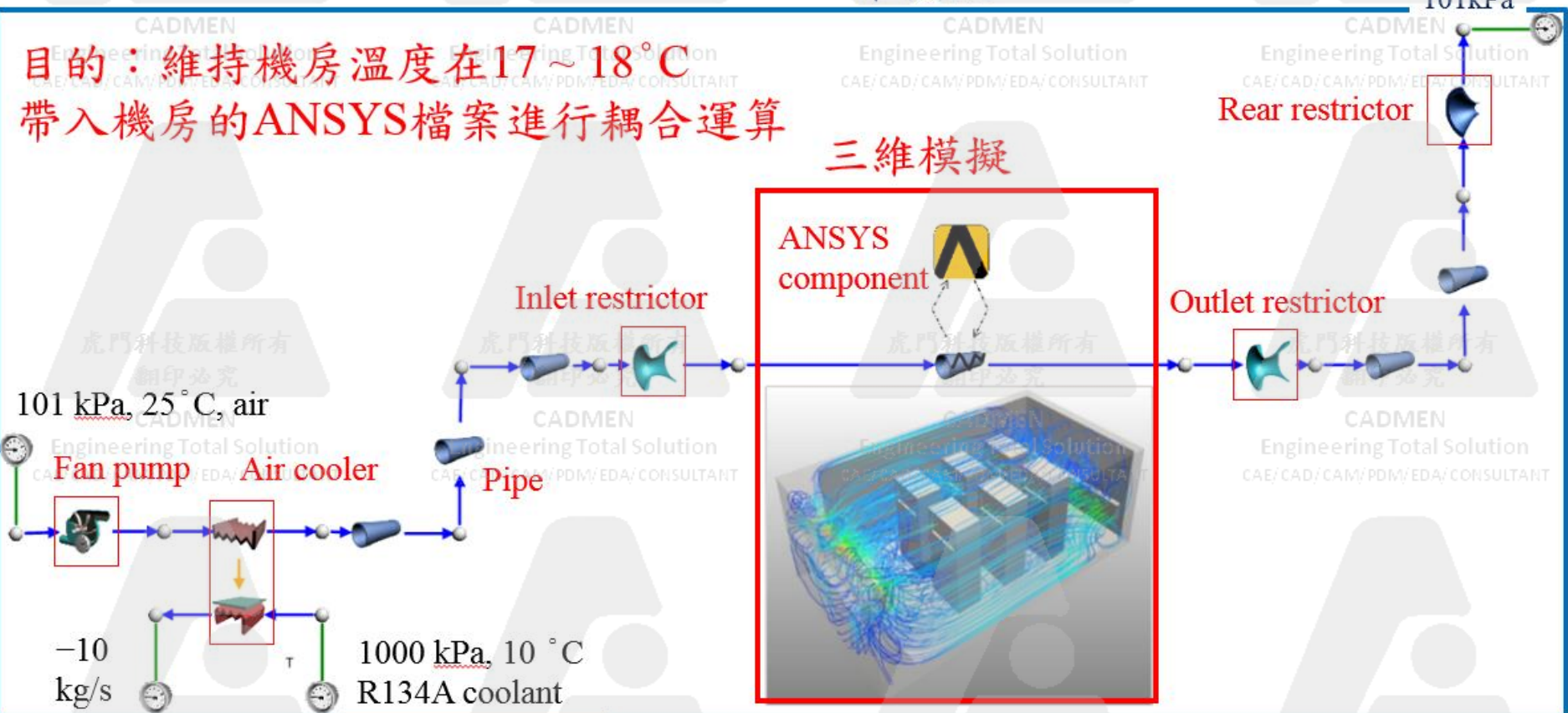
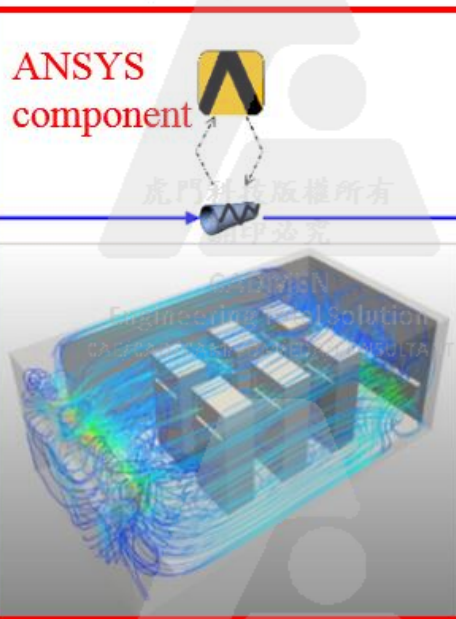
Engineering Total Solution

# 伺服機房空調系統

一維模擬

目的：維持機房溫度在 $17 \sim 18^{\circ}\text{C}$   
帶入機房的ANSYS檔案進行耦合運算

三維模擬



Flownex計算入口壓力並傳遞至ANSYS

ANSYS求解溫度、壓力與流量

求解後ANSYS得出admittance與出口溫度反饋至Flownex



輸入參數

Flow solver inlet pressure



計算結果

Inlet mass flow rate

Outlet total pressure

Inlet total pressure

Temperature distribution



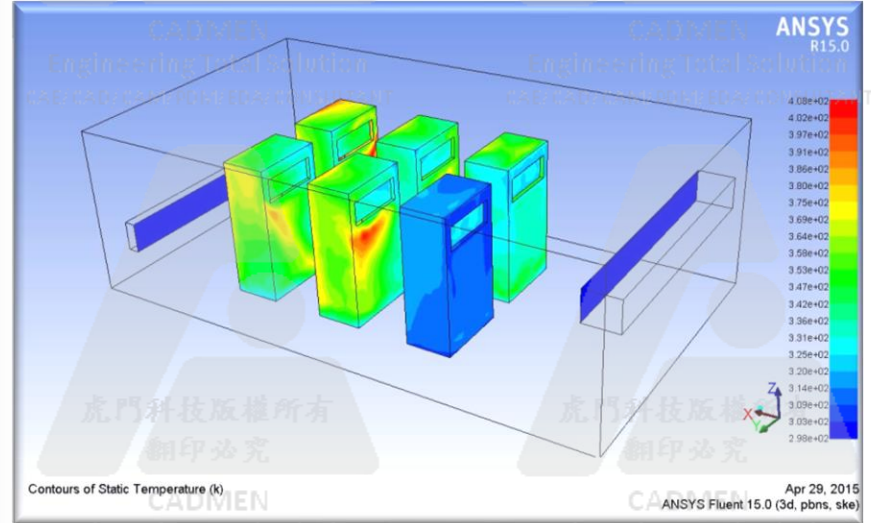
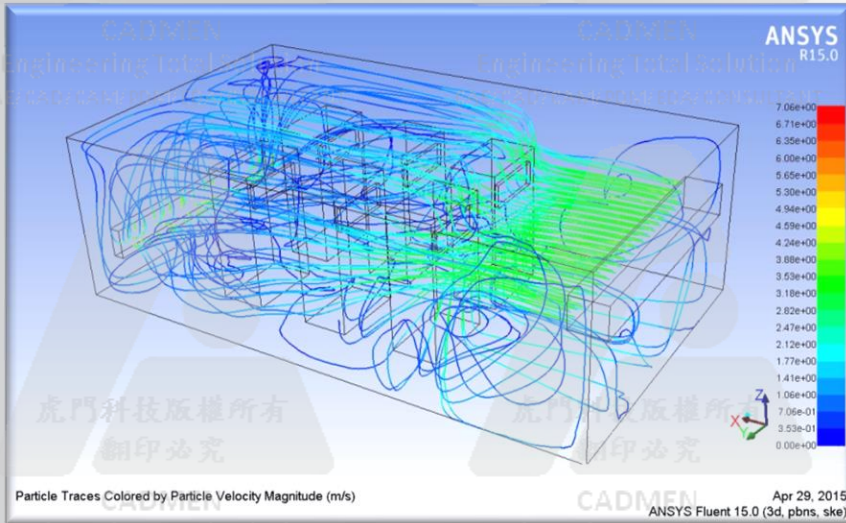
反饋結果

Flow admittance

Outlet temperature



# 伺服機房空調系統



| 計算參數                  | 結果         |
|-----------------------|------------|
| Inlet mass flow rate  | 5.734 kg/s |
| Outlet total pressure | 713.24 kPa |
| Inlet total pressure  | 737.26 kPa |

| 反饋參數               | 結果            |
|--------------------|---------------|
| Flow admittance    | 1.3704        |
| Outlet temperature | 290 K (17 °C) |

# 伺服機房空調系統

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究



CADMEN

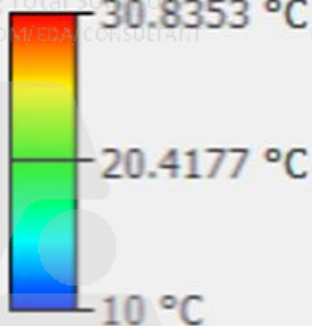
Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT



CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN

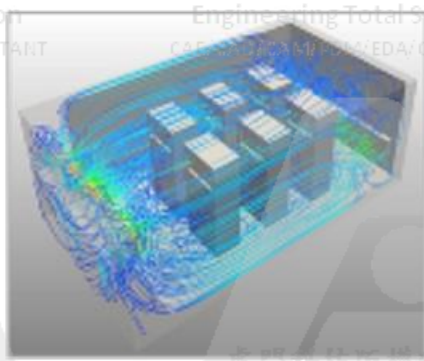
Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT



虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

虎門科技版權所有  
翻印必究

CADMEN

Engineering Total Solution

CADMEN

Engineering Total Solution

CADMEN

Engineering Total Solution

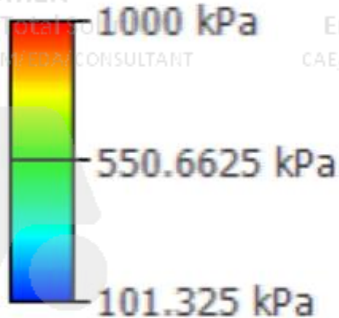
CADMEN

Engineering Total Solution

# 伺服機房空調系統

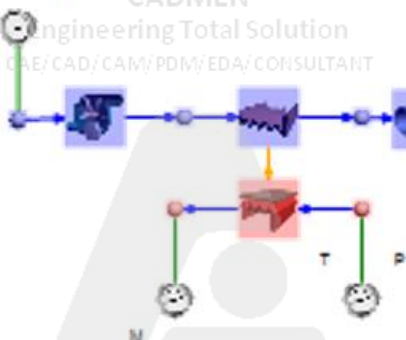
虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT



虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT



虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution

虎門科技版權所有  
翻印必究

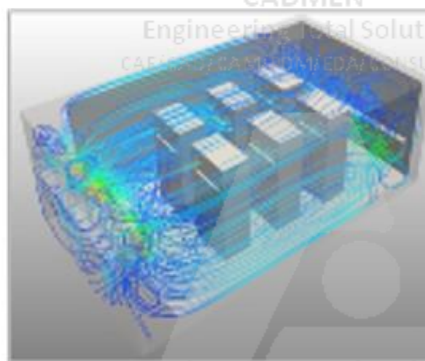
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT



CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

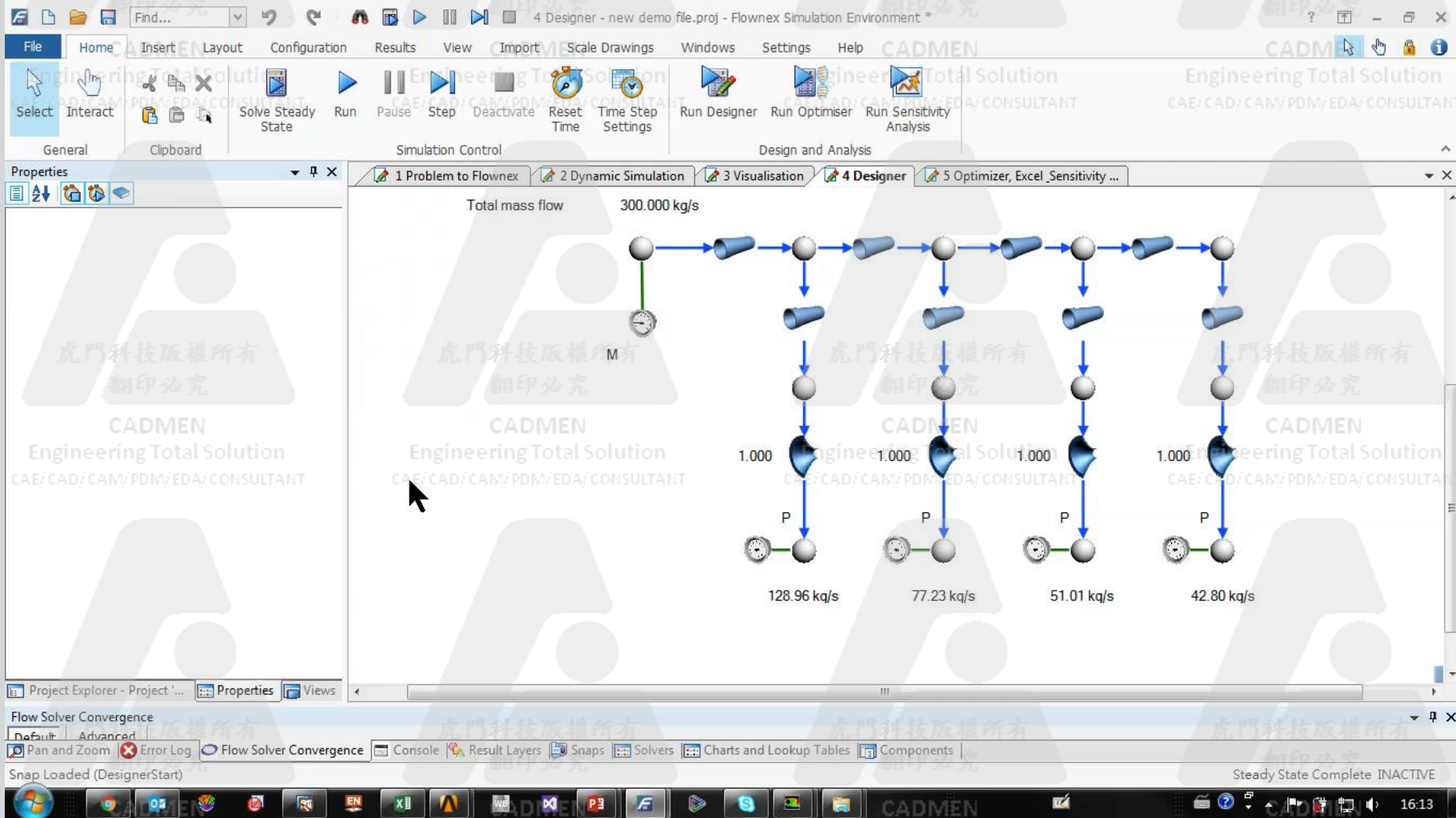
虎門科技版權所有  
翻印必究

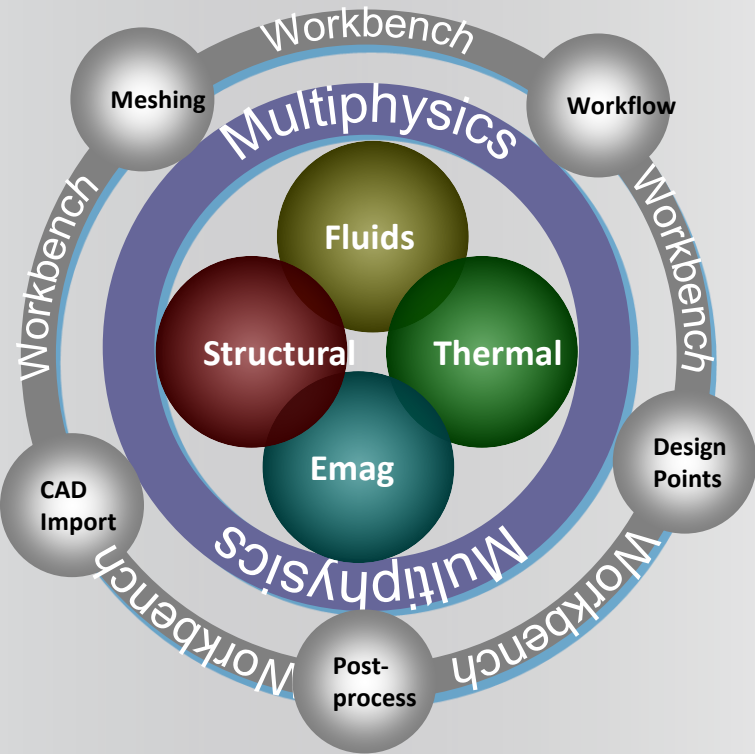
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

虎門科技版權所有  
翻印必究

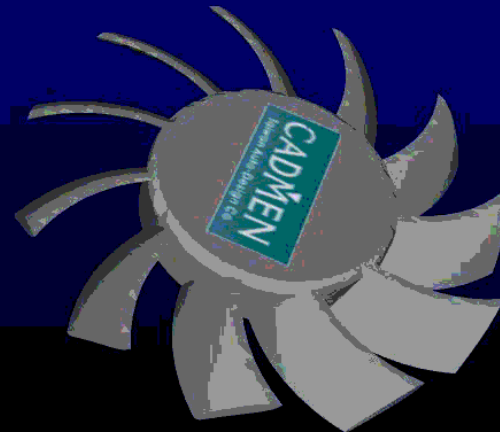
CADMEN  
Engineering Total Solution  
CAE/CAD/CAM/PDM/EDA/CONSULTANT

# 平行管流量案例





**EnSight**



**Thank you for your  
attention!**