ESSS

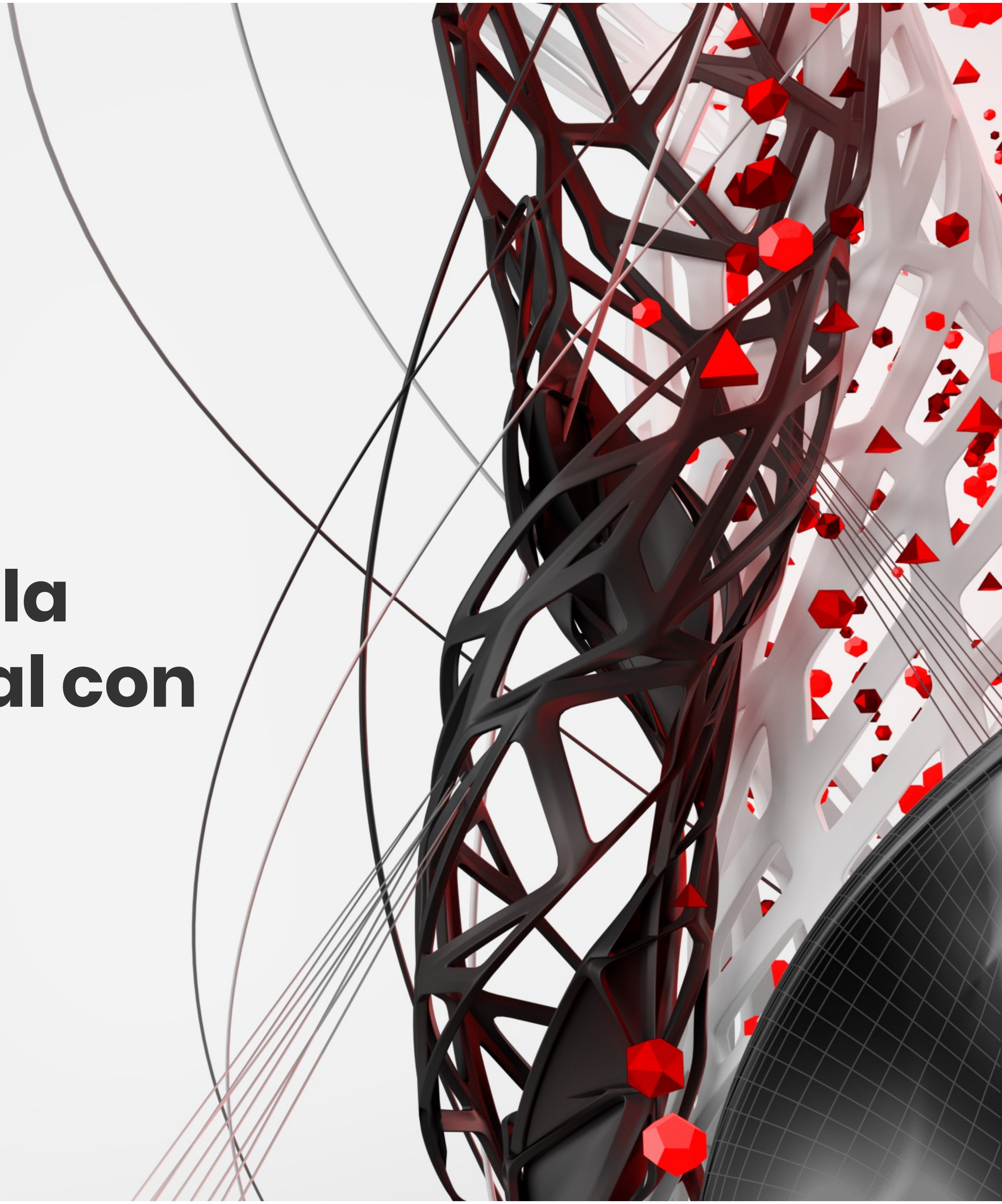
Minicurso: introducción a la simulación computacional con ANSYS

JOSÉ CARLOS ZART

CAE Applications Intern. Academic Area. Latin America

jzart@esss.co / www.esss.co

SIMULATING THE FUTURE



Programación

- 16 de noviembre: Introducción a la simulación computacional
- 19 de noviembre: Workshop CFD – Flujo laminar en tubería recta
- 22 de noviembre: Workshop FEA – Barra con cambio de sección

Agenda

1. Introducción al CAE y el proceso de simulación
2. Fundamentos de simulación fluidodinámica
3. Fundamentos de simulación estructural
4. Preguntas y respuestas

ESSS AND LATIN-AMERICAN ACADEMY

+500
ACADEMIC ACCOUNTS

Mechanical | Civil | Mining | Chemical | Electronic | Electrical | Aerospace

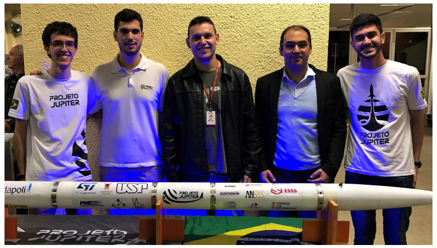
+200
ACADEMIC INSTITUTIONS

Universities
8 of the Top Ten
Latin-American Universities
2020 QS Ranking

Research Centers

Military academies

~100
ACADEMIC COMPETITION TEAMS



www.esss.co

https://mapchart.net/americas.html

https://www.topuniversities.com/university-rankings/latin-american-university-rankings/2020

ANSYS Tools, the biggest CAE education network worlwide

600+ Multiphysics Campus-Wide Installations

3,000+ Universities from 87 countries using ANSYS

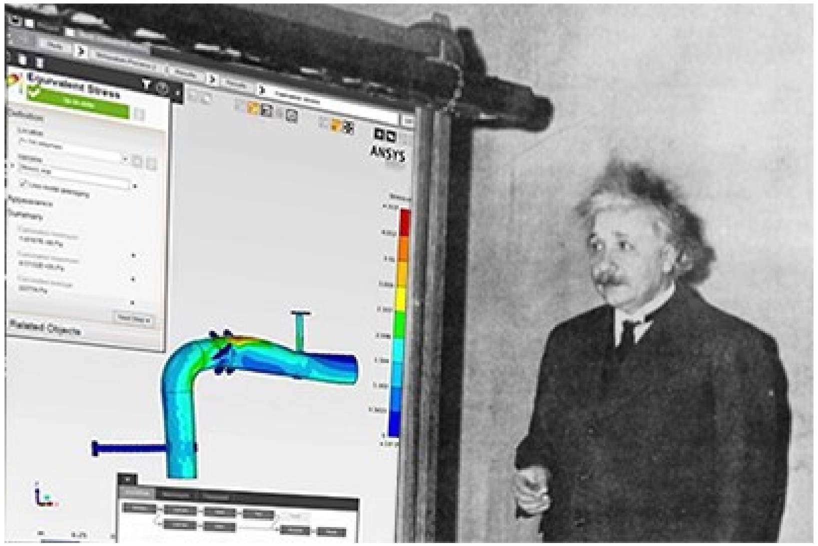
1.2 millions solver licenses installed

300K free download per year

500+ student teams sponsored by ANSYS

130,000+ users in ANSYS Academic Community

140,000+ registrations in ANSYS Cornell MOOC



ANSYS: THE ONE MULTIPHYSICS SIMULATION PLATFORM

Ansys

Fluids

Structures

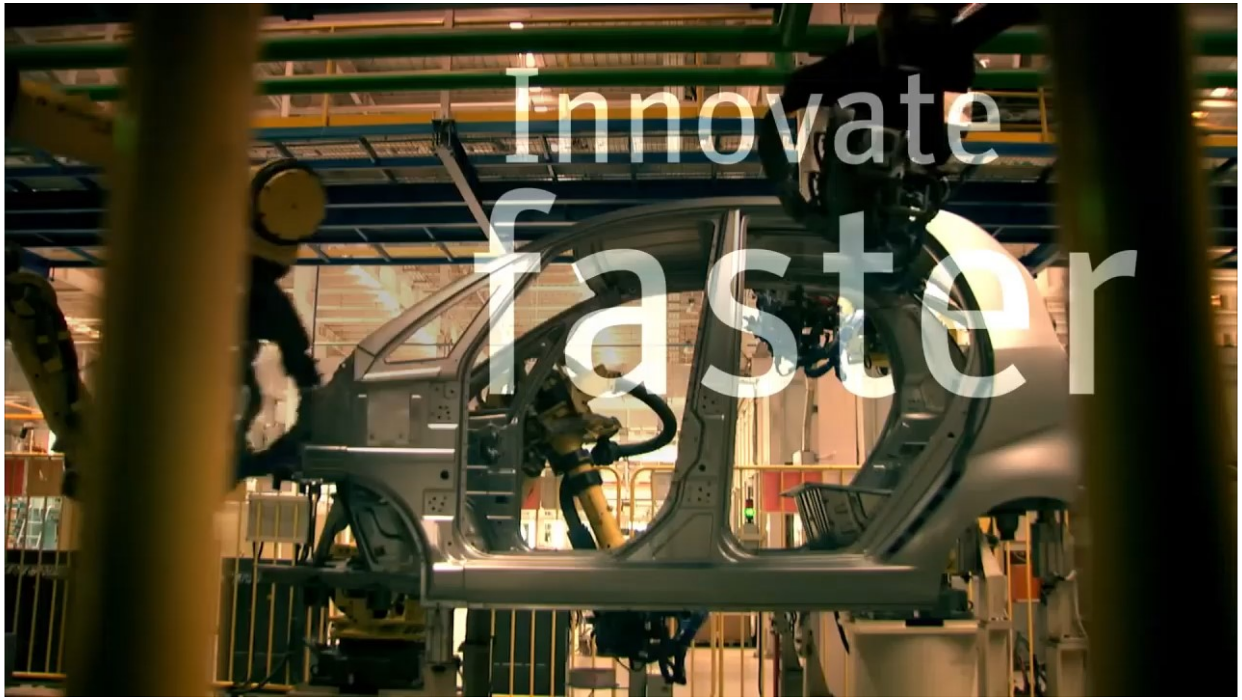
Electromagnetics

Platform

Semiconductor Power

Optical

Mission-Critical Embedded Software



U.S.ARMY CAT SAMSUNG TOYOTA BR PETROBRAS Nestlé HeartWare

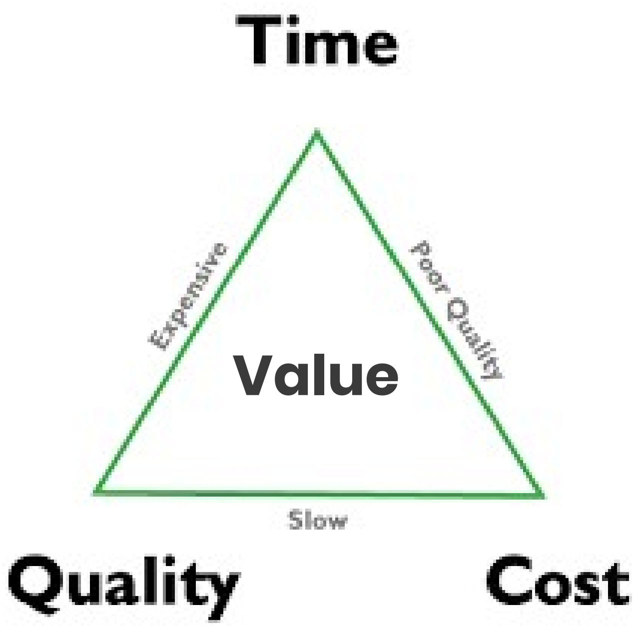
www.esss.co

Agenda

1. Introducción al CAE y el proceso de simulación
2. Fundamentos de simulación fluidodinámica
3. Fundamentos de simulación estructural
4. Preguntas y respuestas

Engineering Challenges

- Design, development or improvement of **products**, **equipments** or **processes**.
- Optimal (and difficult) balance between:



Engineering Challenges

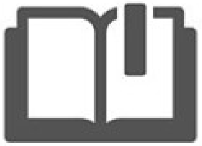
Three methods to solve engineering problems:

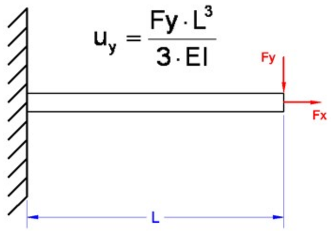
Analytic Methods

Numerical Methods

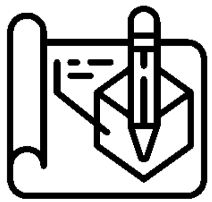
Experimental Methods

Analytic Methods





- Mathematics formula-based solutions.
- Hand developed.
- Inputs → 1 or 2 outputs
- Easy methodology, low cost, low complexity
- Provides quick & direct response to answer the equations
- *Examples: ASME, ASTM, DNV, Handbooks or Standards.*



Technical Reports

Weaknesses:

- Accuracy
- Lack of analytic methods for all physical models
- Failure behaviour?

Standards
(ASME, ASTM, DNV)

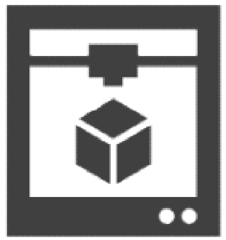
Empirical Methods

- Experimental-based solutions.
- Depend on physical prototyping, real o rescaled size.
- *Examples: Crash test on a car, wind tunnel for aerodynamics.*

Lab Tests

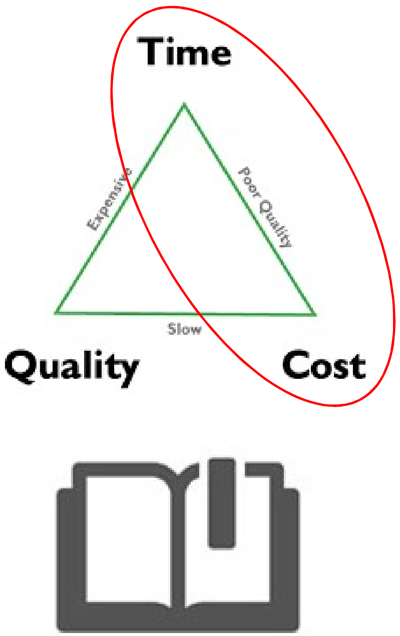
Weaknesses:

- High quality results, but expensive.
- Study effect from one variable in a response, but not the cause.
- Less predictive capabilities.
- Not flexible at all.

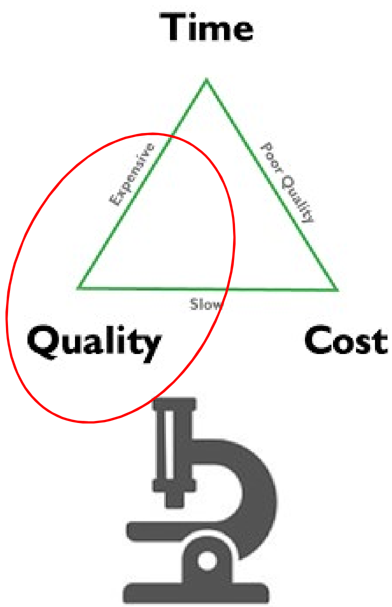


Prototypes

Numerical Methods



Analytic
Methods



Experimental
Methods

Numerical Methods

- Computational advantages
- Numerical solution of differential equations

- Virtual prototyping
- Virtual laboratory
- Validation and calibration of parameters & materials

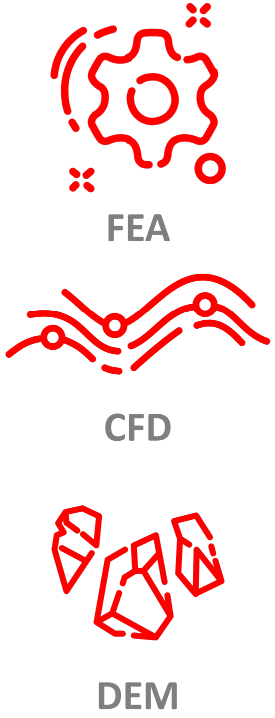
Analytic Methods

Numerical Methods

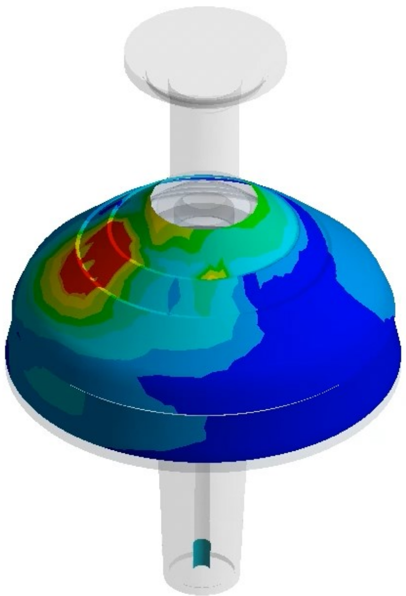
Experimental Methods

Numerical Methods

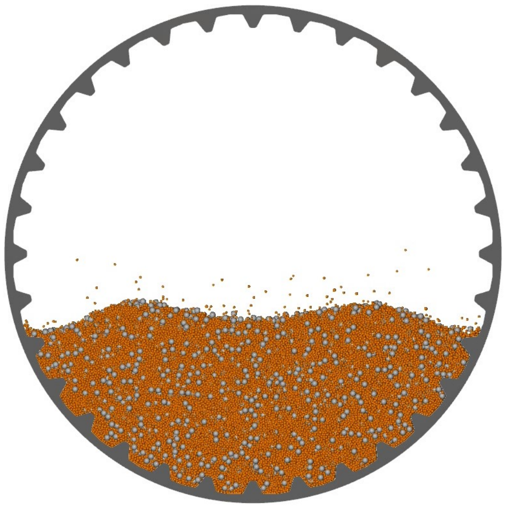
- Virtual prototyping / Virtual laboratory with 3D representation.
- Solvers capable of predicting solid behavior, fluid flow, electromagnetic field, heat transfer, acoustics, chemical/biological reactions, and related phenomena.
- Solving mathematical equations which govern these processes.
- Numerical technique solved with (large) computers:
  - Finite Element Method (FEM) for structural and termal analysis
  - Finite Volume Method (FVM) for fluid dynamics
  - Discrete Element Method (DEM) for granular systems



Numerical Methods



FEA



DEM

CFD

What are the CAE fundamentals?

CAE

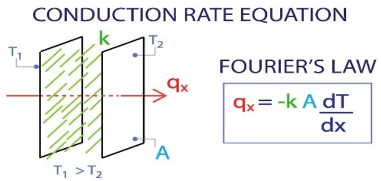
Phenomenological models

+

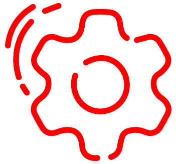
Numerical methods

+

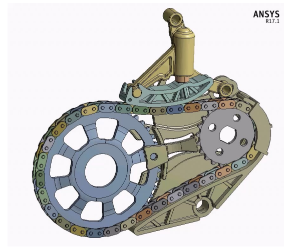
Computational resources



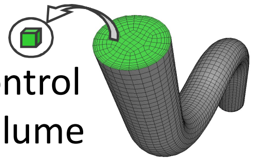
www.youtube.com: Ron Hugo



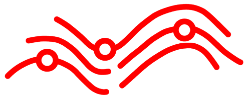
Structures



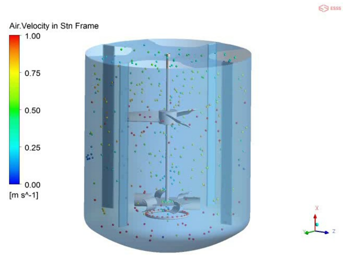
Control Volume



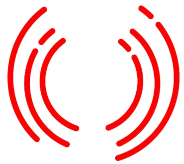
ANSYS Inc



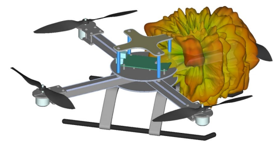
Fluids



http://rc.dartmouth.edu



EMAG



www.esss.co

Agenda

1. Introducción al CAE y el proceso de simulación
2. **Fundamentos de simulación fluidodinámica**
3. Fundamentos de simulación estructural
4. Preguntas y respuestas

Introduction to CFD

Basics & Steps

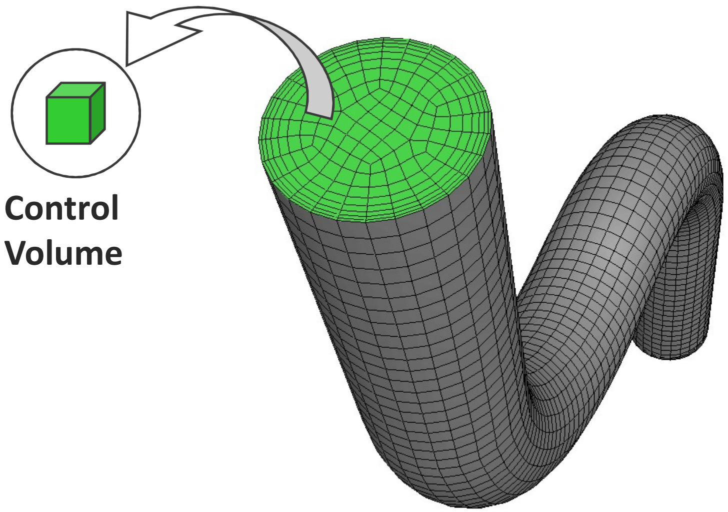
What is CFD?

Fluid Mechanics + Numerical Calculus = Computational Fluid Dynamics

How CFD works ?

ANSYS CFD → Finite Volume Method (FVM)

- **Discretized domain in finite volumes**
- **General transport equations are solved**
  - Continuity (Mass conservation)
  - Momentum (2nd Newton's Law)
  - Energy (1st Thermodynamics' Law)

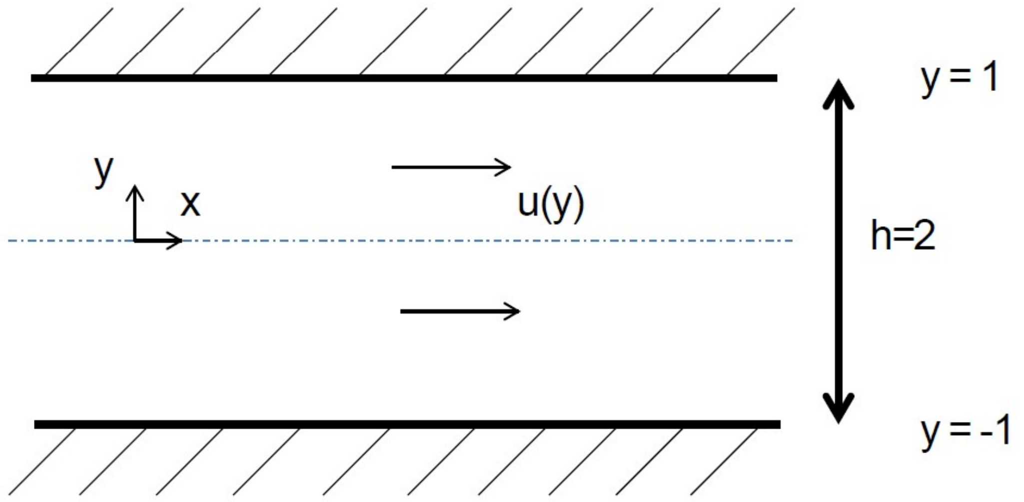


$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{Advection}} = \underbrace{\oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Generation}}$$

Equation	ϕ
Continuity	1
X Momentum	u
Y Momentum	v
Z Momentum	w
Energy	h

CFD Example

- Viscous fluid flowing in a channel under a constant applied pressure gradient dp/dx
- Fully developed, unidirectional flow, so that u = f(y)



CFD Example

1. Governing equations

Under these assumptions, the Navier-Stokes equations can be simplified to the following single governing equation (x-momentum):

$$0 = -dp/dx + \mu \frac{d^2 u(y)}{dy^2} \quad -1 \leq y \leq 1 \qquad (1)$$

with boundary conditions given by the no-slip condition at the two channel walls as follows:

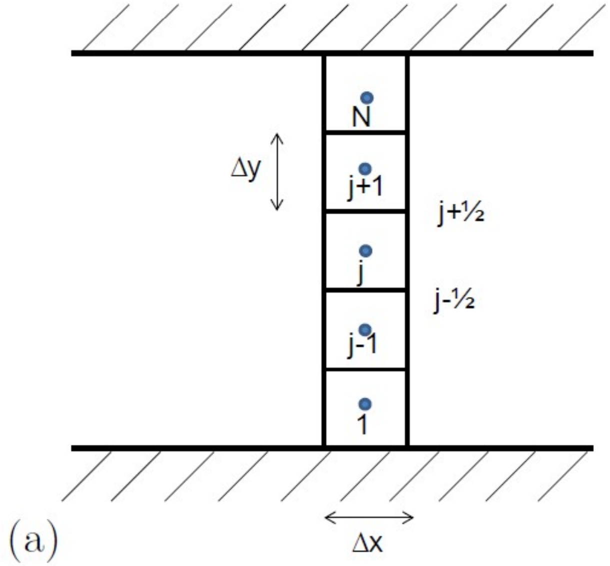
$$u(y) = 0 \;\; at \;\; y = \pm 1 \qquad (2)$$

These equations constitute a boundary value problem (BVP)! Which is to be solved numerically.

CFD Example

2. Discretization using finite-volume method

We will describe the discretization of equation (1) using the finite-volume (FV) scheme, similar to what FLUENT would do.



- Let us divide the vertical extent of the domain into *N* cells each having a height Δy and arbitrary width Δx.

CFD Example

2. Discretization using finite-volume method

The FV method involves writing out the integral form of the governing equations over a discrete control volume *j* (*CV_j*) and then using the divergence theorem to convert it into a surface integral around the control surface (*CS_j*), which we then evaluate using discrete values of the variable at the center of each cell:

$$0 = \int_{CV_j} (-dp/dx)\, d\mathcal{V}_j + \int_{CV_j} \mu \frac{d^2u(y)}{dy^2} d\mathcal{V}_j \qquad \text{(3)}$$

CFD Example

2. Discretization using finite-volume method

$$0 = \boxed{\int_{CV_j} (-dp/dx) d\mathcal{V}_j} + \boxed{\int_{CV_j} \mu \frac{d^2u(y)}{dy^2} d\mathcal{V}_j}$$

Using the divergence theorem

Since dp/dx is a constant,

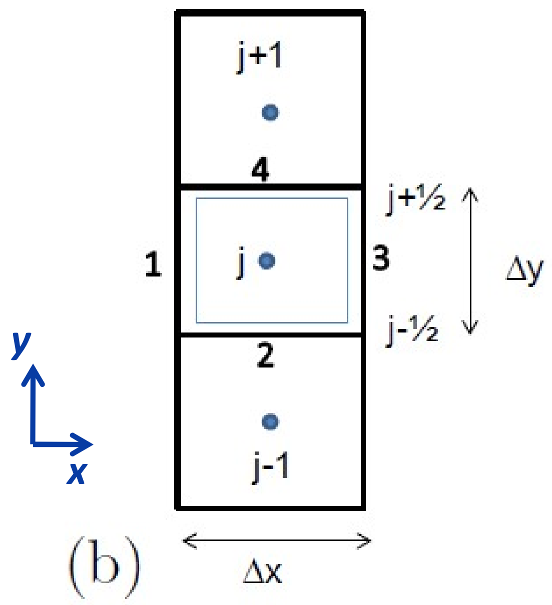
$$\begin{aligned}\int_{CV_j} (-dp/dx) d\mathcal{V}_j &= (-dp/dx)\mathcal{V}_j \\ &= (-dp/dx)\Delta x \Delta y\end{aligned}$$

$$\int_{CV_j} \frac{d^2u(y)}{dy^2} d\mathcal{V}_j = \int_{CV_j} \nabla^2 u d\mathcal{V}_j = \int_{CS_j} \nabla u \cdot \hat{n} dS_j$$

Where n is the outward normal to the control surface

CFD Example

2. Discretization using finite-volume method



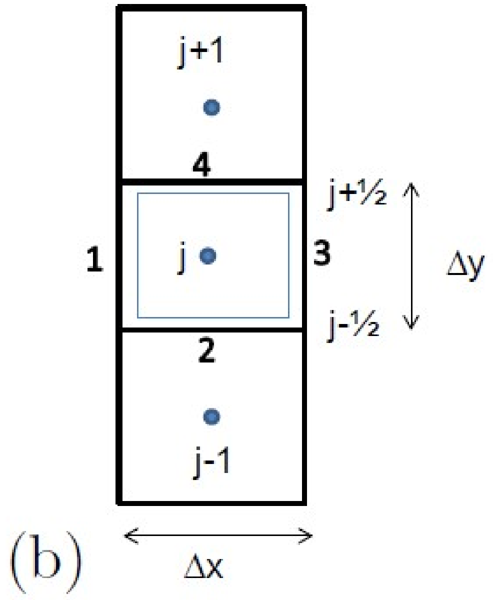
For the faces of the element ***j*** **:**

$$\int_{CS_j} \nabla u \cdot \hat{n} dS_j = \int_{CS_1} \nabla u \cdot \hat{n} dS_1 + \int_{CS_2} \nabla u \cdot \hat{n} dS_2 + \int_{CS_3} \nabla u \cdot \hat{n} dS_3 + \int_{CS_4} \nabla u \cdot \hat{n} dS_4$$

With respect to the axes xy, we see that the outward normal n to the surface is opposite to the direction of gradient for surfaces 1 and 2 and aligned with it for surfaces 3 and 4.

CFD Example

2. Discretization using finite-volume method



Also, since the flow is fully developed, there are no net fluxes in the horizontal direction, i.e., through surface 1 and 3. Therefore, we can now write for any interior cell *j* ,

$$\begin{aligned}\int_{CS_j} \nabla u \cdot \hat{n} dS_j &= 0 + \int_{CS_2} \nabla u \cdot \hat{n} dS_2 + 0 + \int_{CS_4} \nabla u \cdot \hat{n} dS_4 \\ &= -\left(\frac{du}{dy}\right)_{j-\frac{1}{2}} \Delta x + \left(\frac{du}{dy}\right)_{j+\frac{1}{2}} \Delta x\end{aligned}$$

We use a Taylor's series expansion to get an expression for the derivatives, with first-order accurate representation.

CFD Example

2. Discretization using finite-volume method

We use a Taylor's series expansion to get an expression for the derivatives, with first-order accurate representation. (Central differencing scheme):

$$
\begin{aligned}
&= -\left(\frac{du}{dy}\right)_{j-\frac{1}{2}}\Delta x + \left(\frac{du}{dy}\right)_{j+\frac{1}{2}}\Delta x \\
&= -\frac{u_j - u_{j-1}}{\Delta y}\Delta x + \frac{u_{j+1} - u_j}{\Delta y}\Delta x \\
&= \boxed{(u_{j-1} - 2u_j + u_{j+1})\frac{\Delta x}{\Delta y}}
\end{aligned}
$$

This term (multiplied by μ) is essentially a sum of the shear forces acting on the control volume *CVj*.

CFD Example

2. Discretization using finite-volume method

Therefore, putting everything together we get the discretized form of the equation (1):

$$0 = \int_{CV_j} (-dp/dx) d\mathcal{V}_j + \int_{CV_j} \mu \frac{d^2 u(y)}{dy^2} d\mathcal{V}_j$$

(Continuous)

$$\downarrow$$

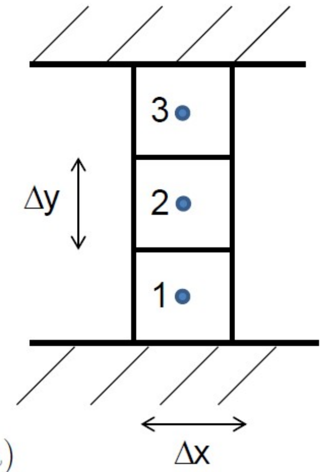
$$0 = (-dp/dx) + \mu \frac{u_{j-1} - 2u_j + u_{j+1}}{(\Delta y)^2}$$

(Discrete)

CFD Example

3. Assembly of discrete system and application of boundary conditions

In order to make our calculations simple, let us assume dp/dx = -1 corresponding to a flow in the positive *x* direction and μ = 1 , and consider a discretization involving *N = 3* cells as shown in the figure.



(a)

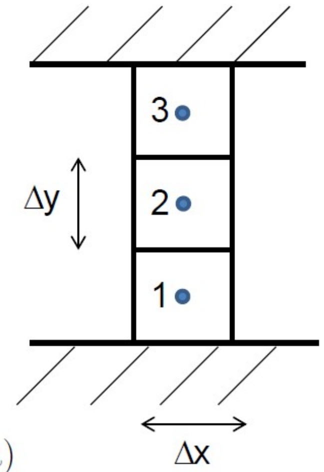
Then, our final discretized form for na interior cell becomes

$$u_{j-1} - 2u_j + u_{j+1} = -(\Delta y)^2$$

CFD Example

3. Assembly of discrete system and application of boundary conditions

$$u_{j-1} - 2u_j + u_{j+1} = -(\Delta y)^2$$



(a)

This equation is true as long as our cell (or control volume) is not adjacent to the boundary. For the boundary cells (j = 1 and j = N) , we need to slightly modify the way we calculate the fluxes in order to accomodate the boundary points N+1/2 and 1-1/2 .

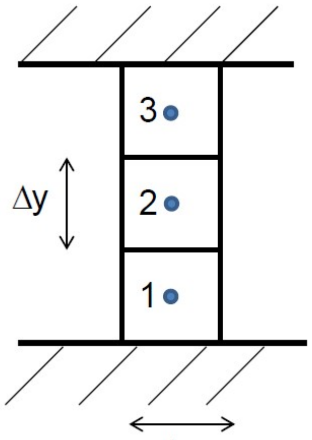
For *j = N* ,

$$\begin{aligned}\int_{CS_N} \nabla u \cdot \hat{n} dS_N &= -\frac{u_N - u_{N-1}}{\Delta y}\Delta x + \frac{u_{N+\frac{1}{2}} - u_N}{\Delta y/2}\Delta x \\ &= (u_{N-1} - 3u_N + 2u_{N+\frac{1}{2}})\frac{\Delta x}{\Delta y}\end{aligned}$$

CFD Example

3. Assembly of discrete system and application of boundary conditions

$$(u_{N-1} - 3u_N + 2u_{N+\frac{1}{2}})\frac{\Delta x}{\Delta y}$$



(a)

Since for *j = N*, there are no points corresponding to *N + 1* , we have used the boundary point *N + 1/2* to compute the flux. We can employ the same tactic for the bottom boundary cell *j = 1*. Therefore, for *N = 3* cells, our final discretized set of equations corresponding to the governing equation (1) is:

$$2u_{1-\frac{1}{2}} - 3u_1 + u_2 = -(\Delta y)^2 \quad (j = 1)$$
$$u_1 - 2u_2 + u_3 = -(\Delta y)^2 \quad (j = 2)$$
$$u_2 - 3u_3 + 2u_{3+\frac{1}{2}} = -(\Delta y)^2 \quad (j = 3)$$

CFD Example

3. Assembly of discrete system and application of boundary conditions

These equations form a system of three simultaneous algebraic equations in the three unknowns u1, u2 and u3 with specified boundary values $u_{1-\frac{1}{2}}$ and $u_{3+\frac{1}{2}}$, for which we can immediately apply the no-slip boundary conditions $u_{1-\frac{1}{2}}$ = $u_{3+\frac{1}{2}}$ = 0.

In this case, you can solve these equations by inspection, but for practical systems we need to use a large number of cells. Therefore, it is generally convenient to write this system in matrix form:

$$\begin{pmatrix} -3 & 1 & 0 \\ 1 & -2 & 1 \\ 0 & 1 & -3 \end{pmatrix} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \end{pmatrix} = -(\Delta y)^2 \begin{pmatrix} 1 \\ 1 \\ 1 \end{pmatrix}$$

CFD Example

4. Solution of discrete system

The discrete system for our 1D problem can be easily inverted to obtain the unknowns at the grid points. Using $\Delta y = 2/3$, we can solve for u1, u2 and u3 in turn and obtain:

$$u_1 = 1/3 \quad u_2 = 5/9 \quad u_3 = 1/3$$

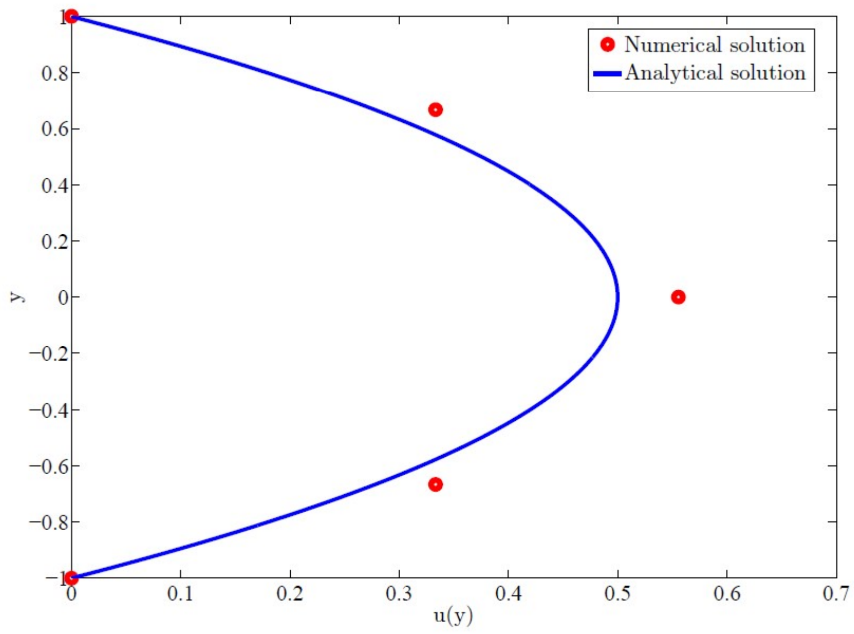
The exact or analytical solution for this problem is easily calculated to be

$$u_{exact}(y) = -y^2/2 + 1/2$$

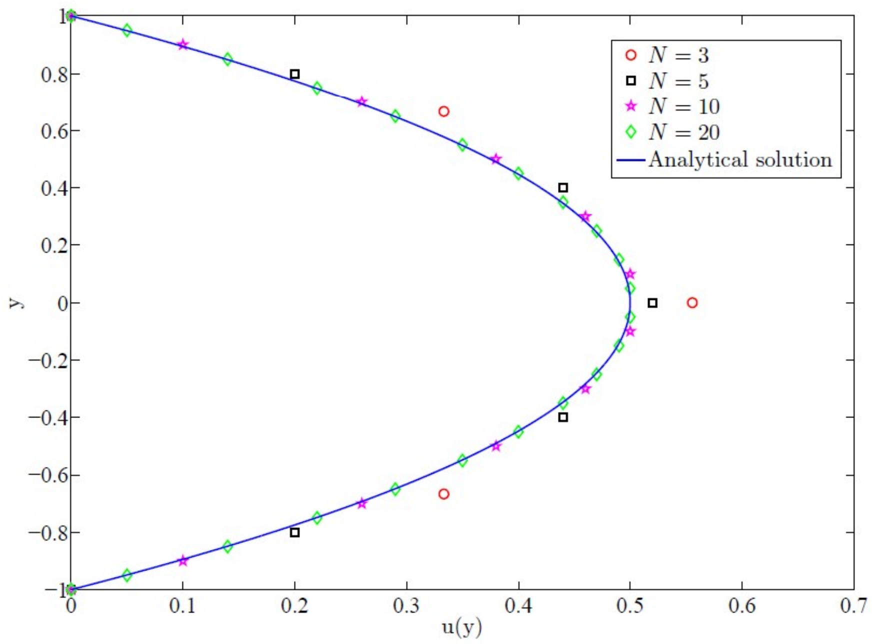
CFD Example

4. Solution of discrete system

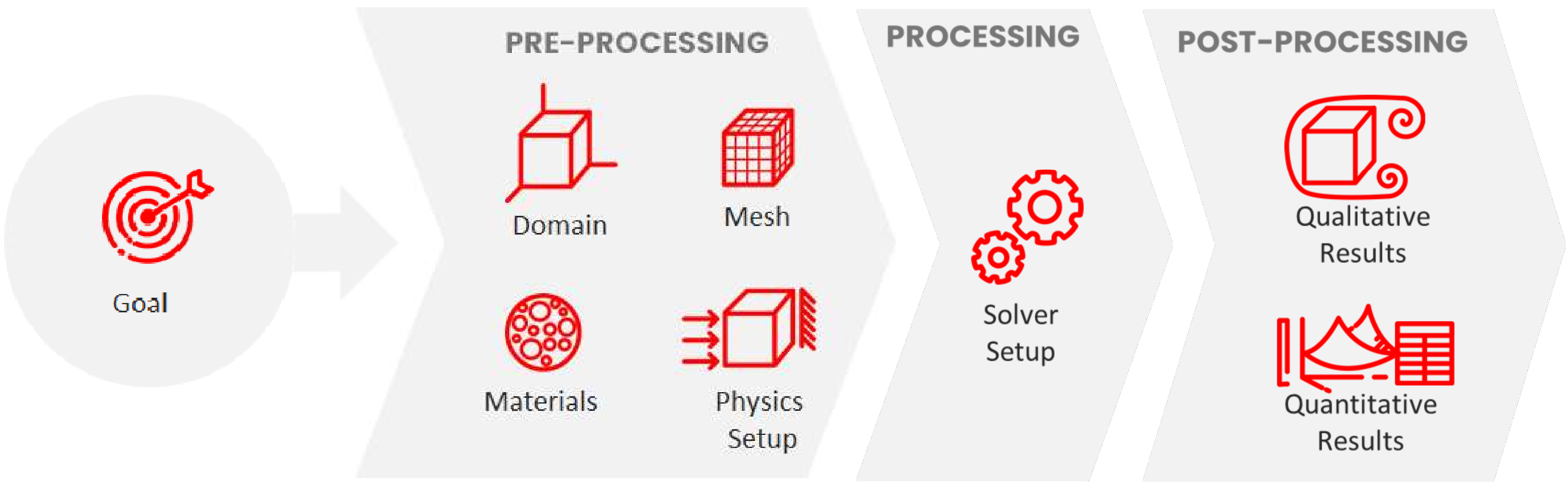
Numerical solution vs Analytical Solution



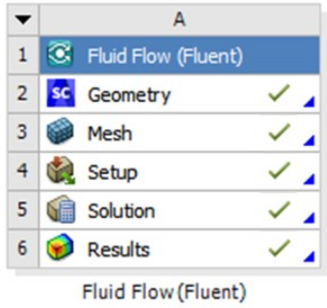
Grid Convergence Analysis

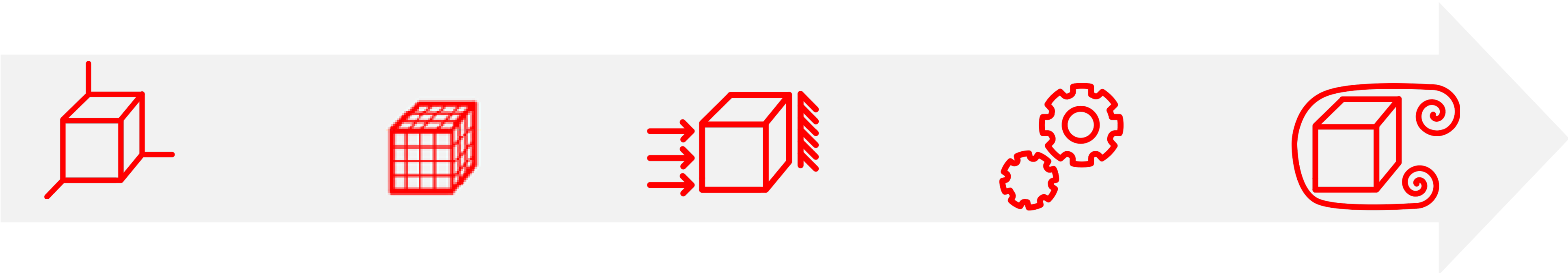


Simulation Progress



ANSYS CFD Workflow





ANSYS SpaceClaim
CAD Definition –
Domain generation

ANSYS Fluent (Meshing)
Advanced CFD
Mesh generation

ANSYS Fluent (Setup)
Advanced CFD
Configuration

ANSYS Fluent (Solver)
Advanced CFD
Solver

ANSYS CFD-Post
Advanced CFD
Post-processing

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

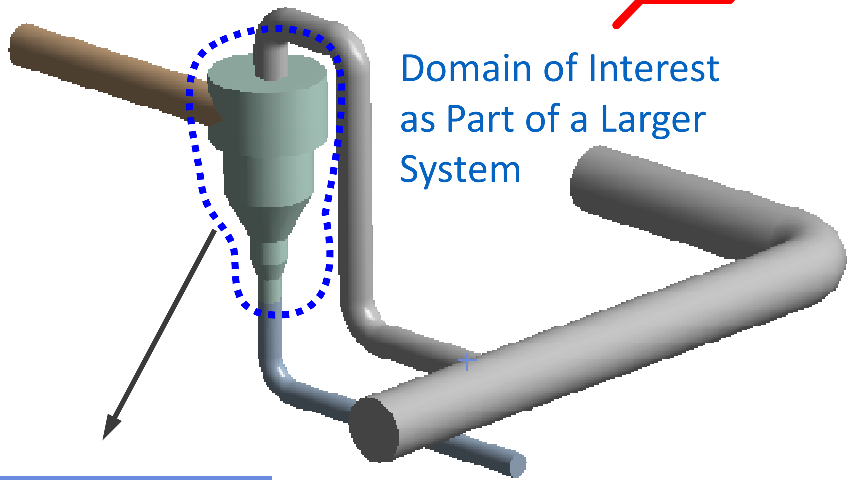
What degree of accuracy is required?

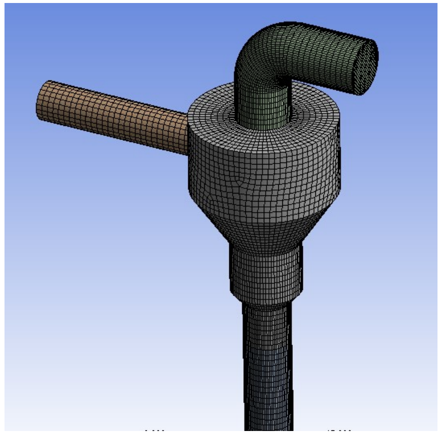
How quickly do you need the results?

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 a 2D or axi-symmetric problem?**





Step 3. Create a Solid Model of the Domain

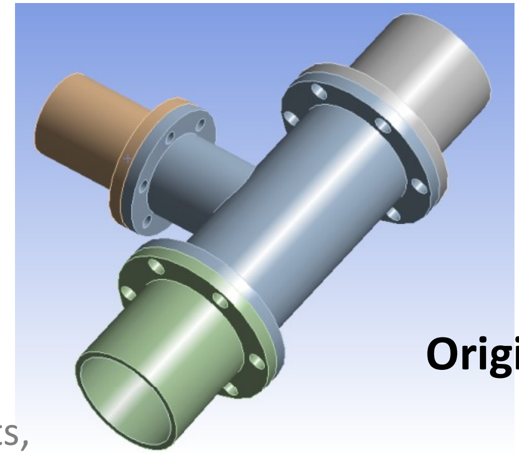
How will you obtain a model of the *fluid* region?

- Make use of existing CAD models?
- Extract the fluid region from a solid part?
- Create from scratch?

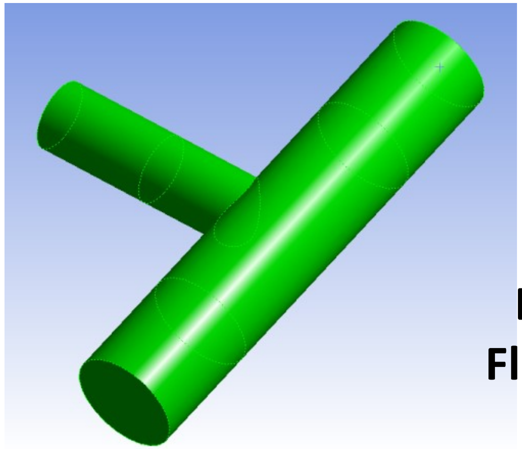
Can you simplify the geometry?

- 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

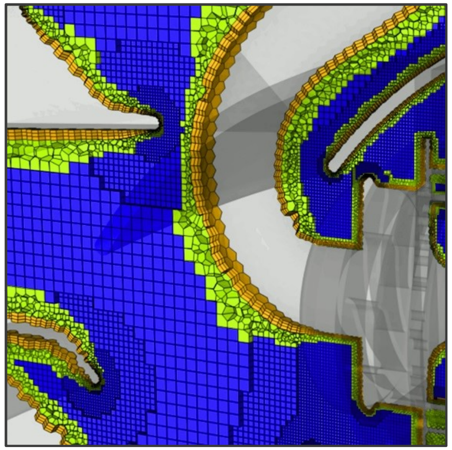


Extracted Fluid Region

Step 4. Design and Create the Mesh

What is the required mesh resolution?

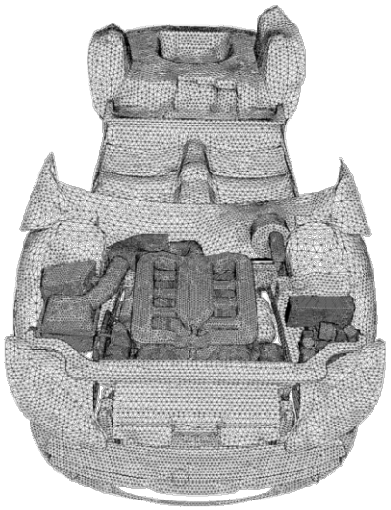
Resolves geometric features of interest and captures gradients of concern, e.g. velocity, pressure, temperature gradients



Do you have sufficient computer resources?

How many cells/nodes are required?

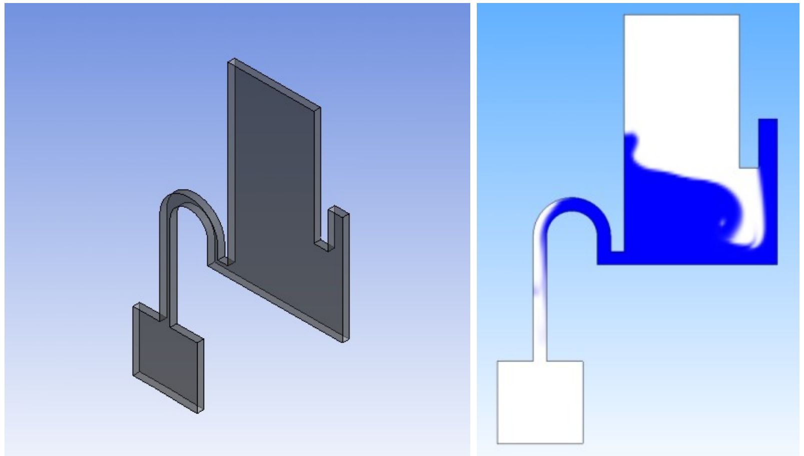
How many physical models will be used?



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

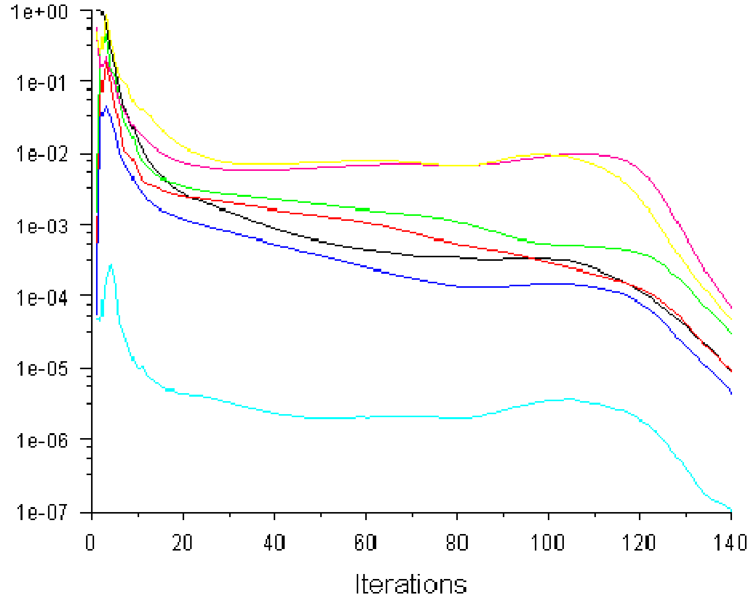
The discretized conservation equations are solved iteratively until convergence

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 (e.g. drag, pressure drop) 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

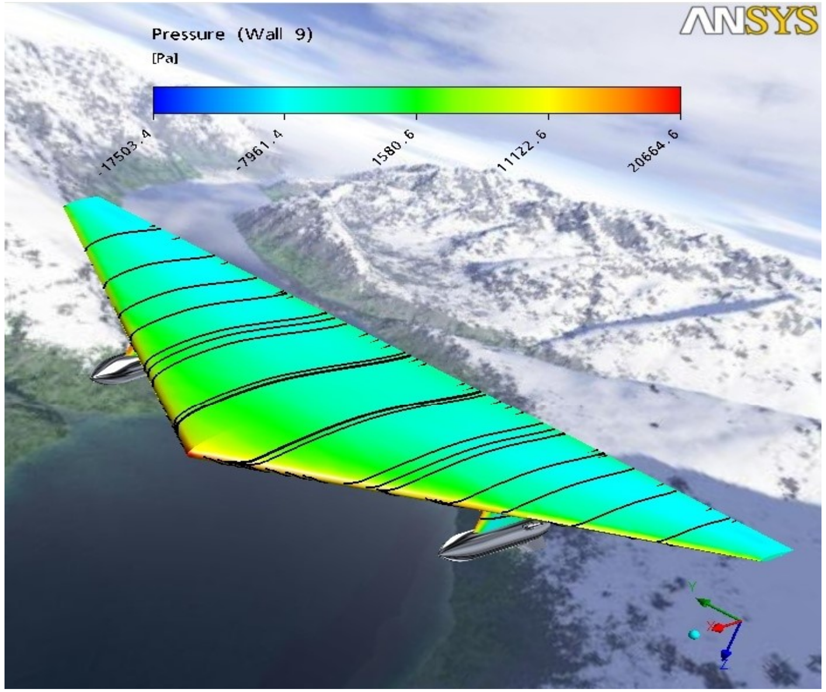


A converged and mesh-independent solution on a well-posed problem will provide useful engineering results!

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.

Step 8. Consider Revisions to the Model

Are the physical models appropriate?

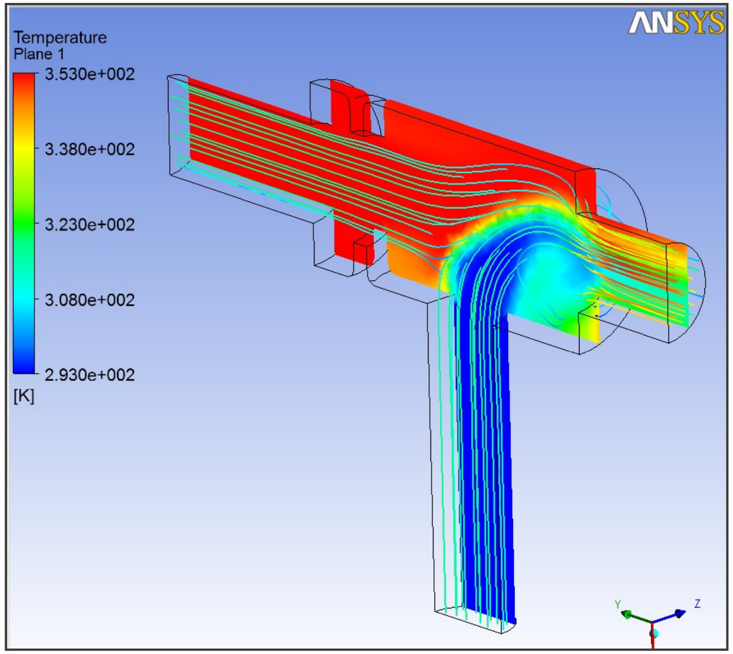
- Is the flow turbulent?
- Is the flow unsteady?
- Are there compressibility effects?

Are the boundary conditions correct?

- Is the computational domain large enough?
- Are boundary conditions appropriate?
- Are boundary values reasonable?

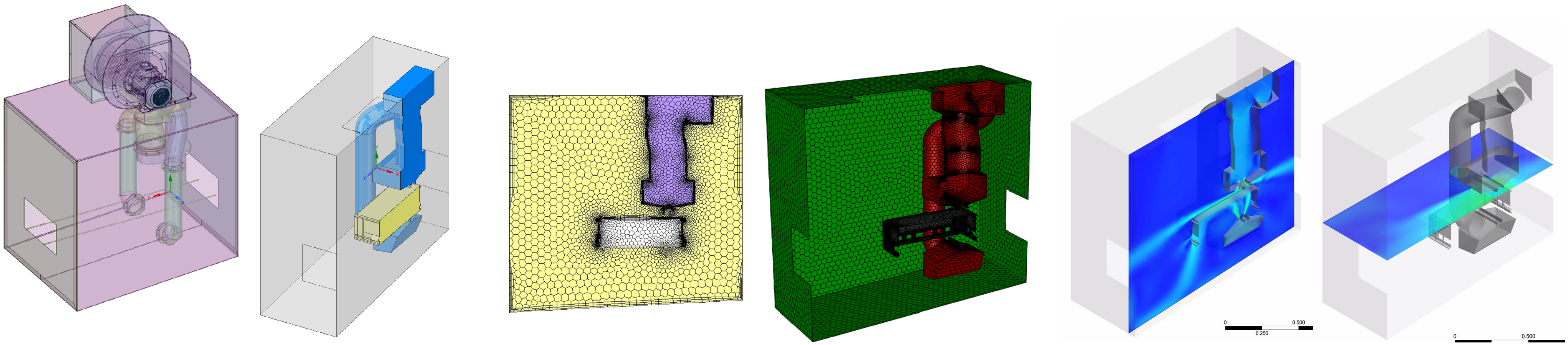
Is the mesh adequate?

- Does the solution change significantly with a refined mesh, or is the solution mesh independent?
- Does the mesh resolution of the geometry need to be improved?
- Does the model contain poor quality cells?



High residuals may be caused by just a few poor quality cells

CFD Workflow Example



ANSYS SpaceClaim
CAD Definition –
Domain generation

ANSYS Fluent
(Meshing)
Advanced CFD
Mesh generation

ANSYS Fluent
(Setup)
Advanced CFD
Configuration

ANSYS Fluent
(Solver)
Advanced CFD
Solver

ANSYS CFD-Post
Advanced CFD
Post-processing

Agenda

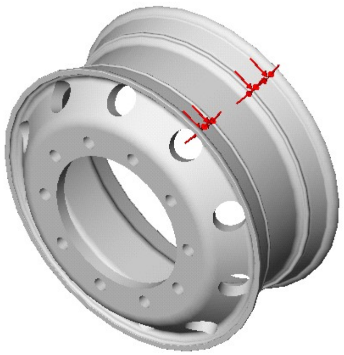
1. Introducción al CAE y el proceso de simulación
2. Fundamentos de simulación fluidodinámica
3. Fundamentos de simulación estructural
4. Preguntas y respuestas

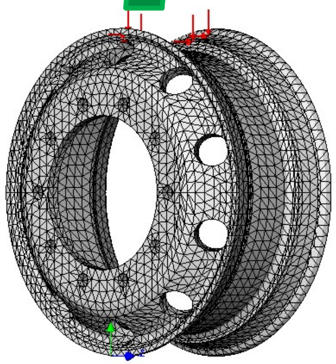
Introduction to FEM

Basics

¿Cómo es la solución por el Método de los Elementos Finitos?

Condiciones de Contorno



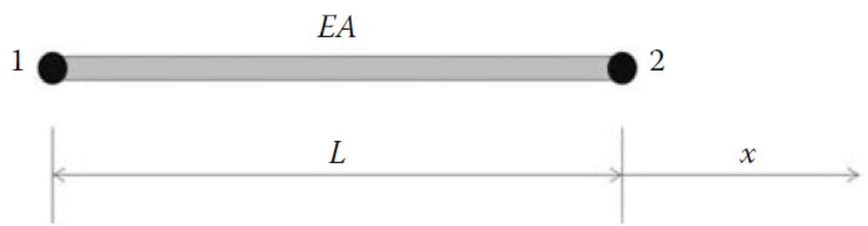


$$\{F\} = [K]\ \{U\}$$

Discretización
de la
Geometría

FEA Basics

One-dimensional truss element



Consider an element of length *L* , cross section *A* , and made of a linear elastic material having a Young's modulus *E*.

If we apply a normal force N1 at node 1 , and at the same time maintaining node 2 fixed in space, the bar shortens by an amount u1.

The force N1 is related to the displacement u1 through the spring constant:

$$N_1 = \frac{AE}{L} u_1$$

In virtue of Newton's third law, there must be a reaction force R2 at node 2:

$$R_2 = -\frac{AE}{L} u_1$$

FEA Basics

One-dimensional truss element

Similarly, if we apply a normal force N2 at node 2, and at the same time maintaining node 1 fixed in space, the bar lengthens by an amount u2.

In the same fashion, the force N2 is related to the displacement u2 through the spring constant

$$N_2 = \frac{AE}{L} u_2$$

Again, in virtue of Newton's third law, there must be a reaction force R1 at node 1:

$$R_1 = -\frac{AE}{L} u_2$$

FEA Basics

One-dimensional truss element

When the bar is subjected to both forces N1 and N2 in virtue of the principle of superposition, the total forces F1 and F2 will be:

$$F_1 = N_1 - R_1 = \frac{AE}{L}u_1 - \frac{AE}{L}u_2$$

$$F_2 = N_2 - R_2 = -\frac{AE}{L}u_1 + \frac{AE}{L}u_2$$

Rearranging these equations in a matrix form yields

$$\begin{bmatrix} AE/L & -AE/L \\ -AE/L & AE/L \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}$$

Or simply as $[K_e]\{u_e\} = \{F_e\}$

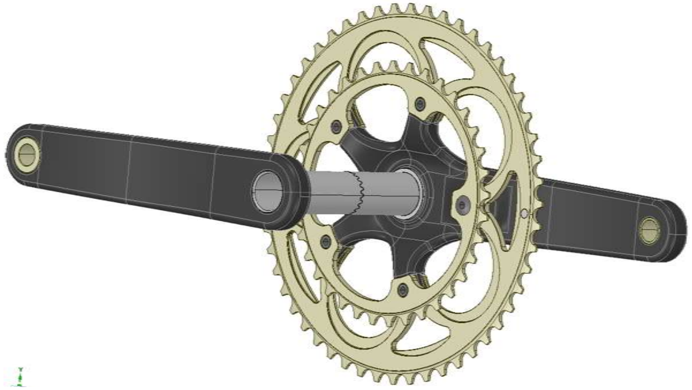
$\{u_e\}$: vector of nodal displacements

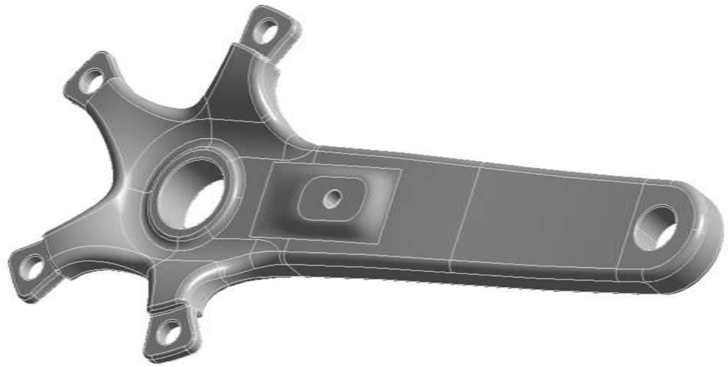
$\{F_e\}$: vector of nodal forces

$[K_e]$: stiffness matrix

*To solve this system of equations, the bar must be restrained in space against rigid body movement.

¿Cómo es el análisis estructural utilizando softwares CAE?





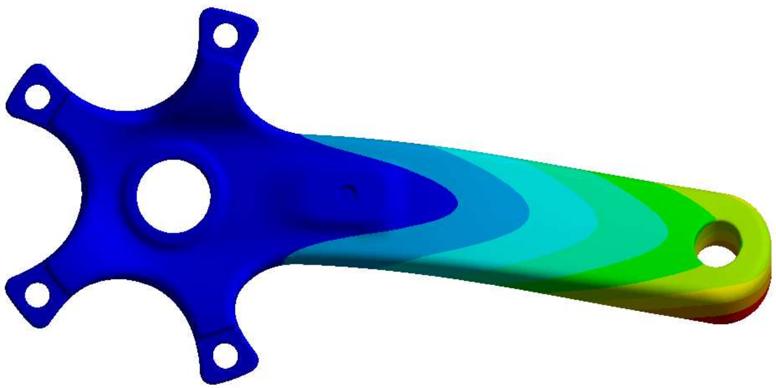
Generación de la malla de Elementos Finitos

Modelo CAE

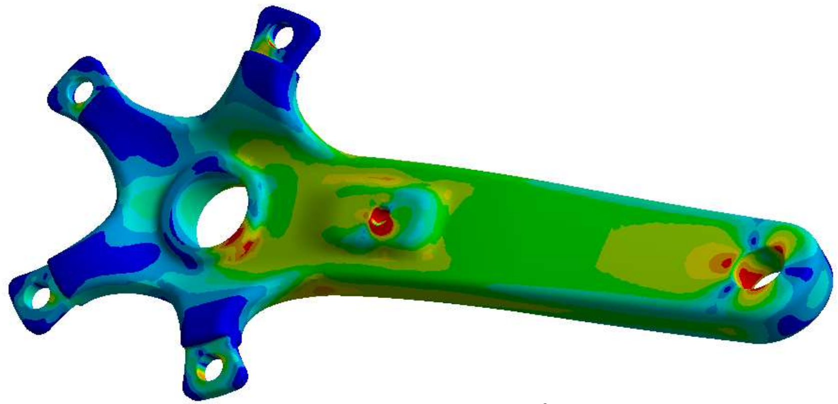
Aplicación de Cargas y Soportes

¿Cómo es el análisis estructural utilizando softwares CAE?

Respuestas del Proceso



Desplazamientos



Tensiones

X Axis	-457,24 N
Y Axis	-721,03 N
Z Axis	5,3405e-004 N
Total	853,78 N

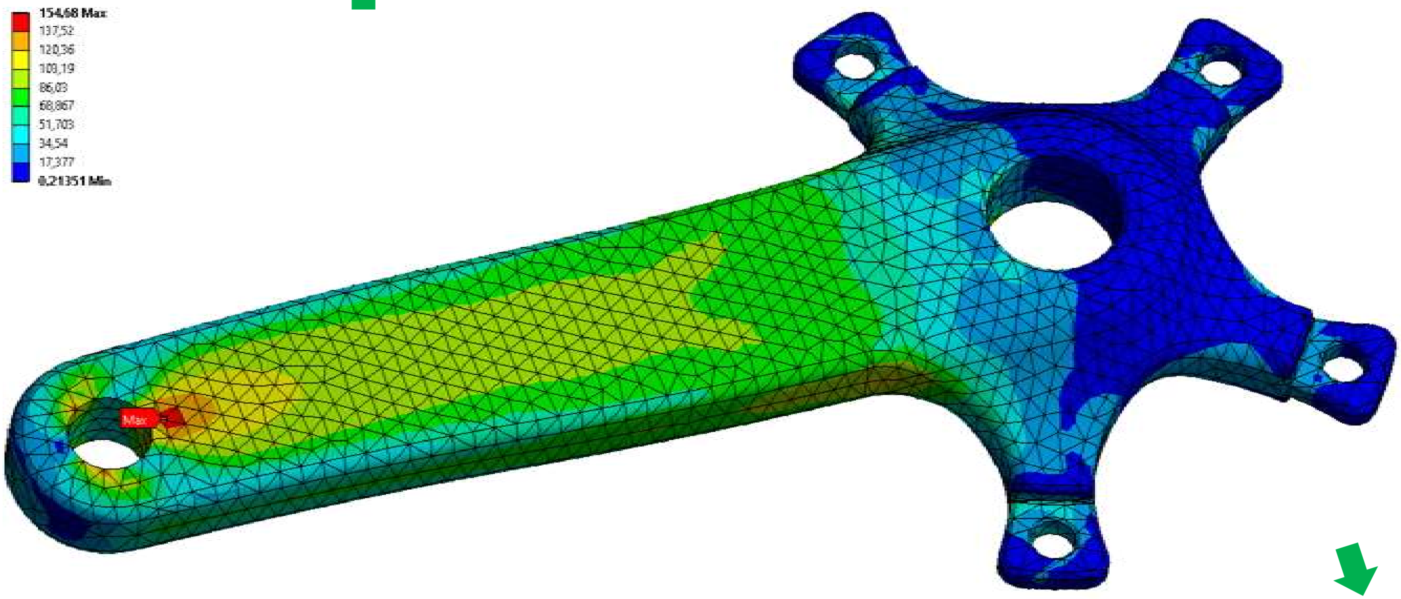
Reacciones

www.esss.co

¿Cómo es el análisis estructural utilizando softwares CAE?

Resistencia

Rigidez

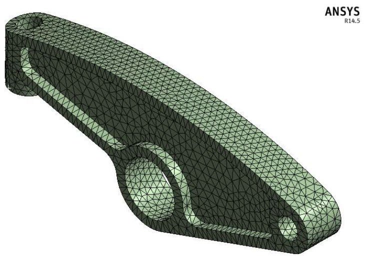


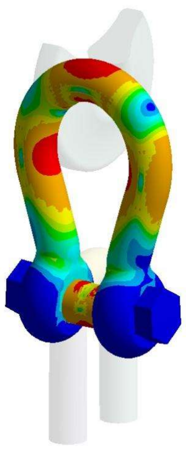
Estabilidad

Durabilidad

Análisis de las Respuestas

¿Cómo es el análisis estructural utilizando softwares CAE?





Decisiones Preliminares

↓

Preprocesamiento

↓

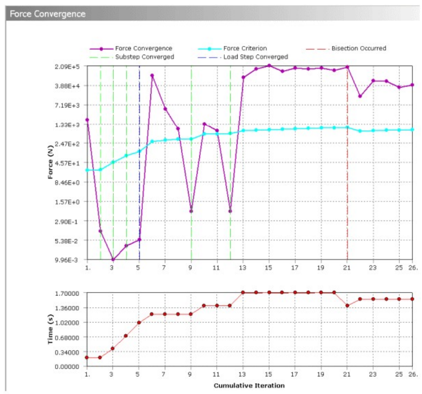
Solución

↓

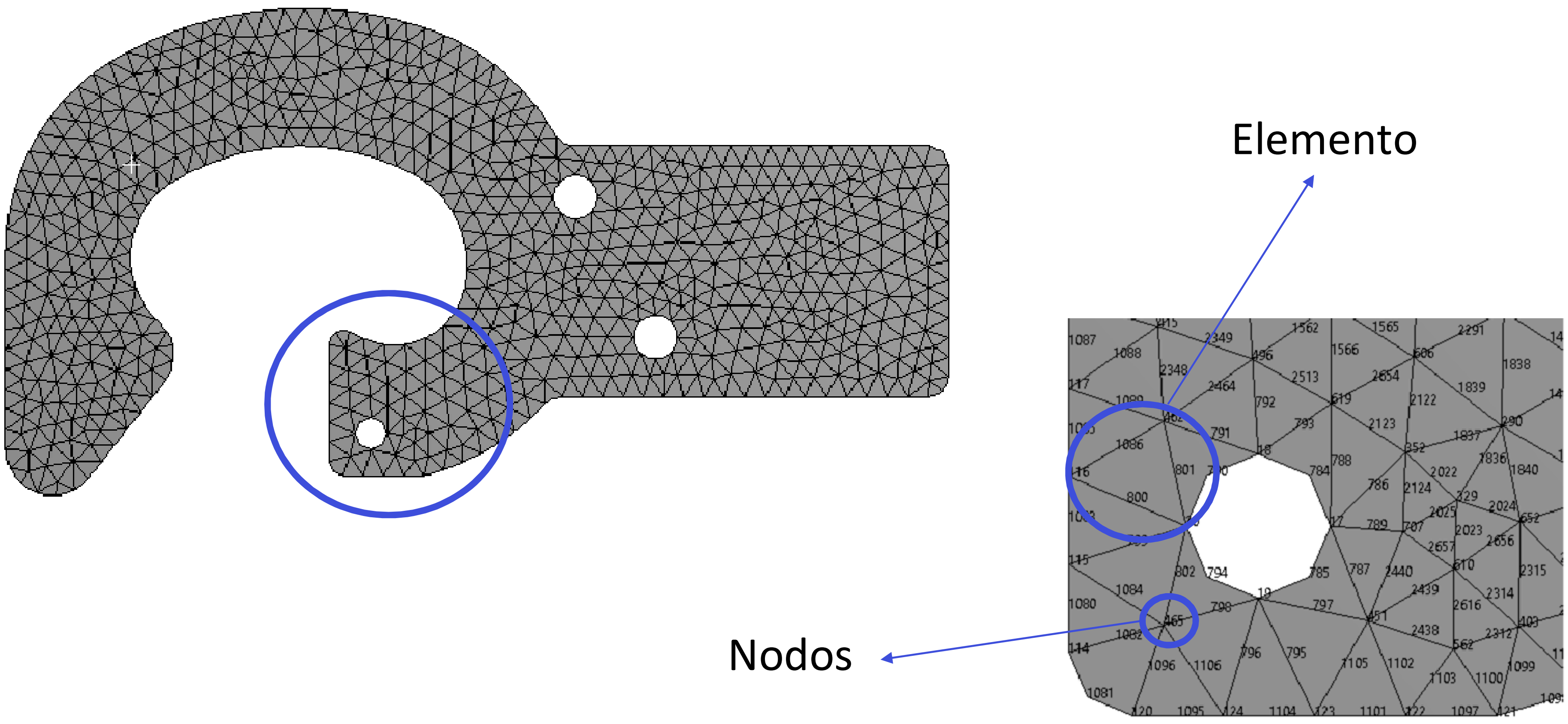
Posprocesamiento

↓

Interpretación de Resultados



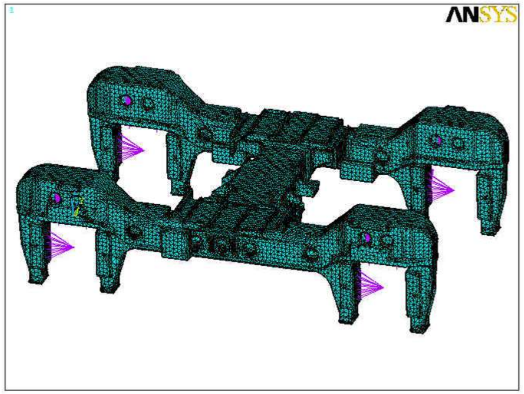
¿Cómo es la solución por el Método de los Elementos Finitos?

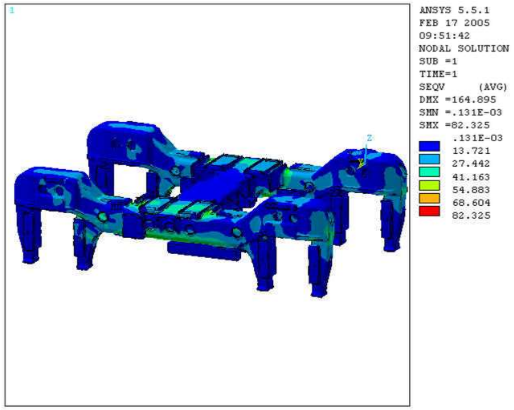


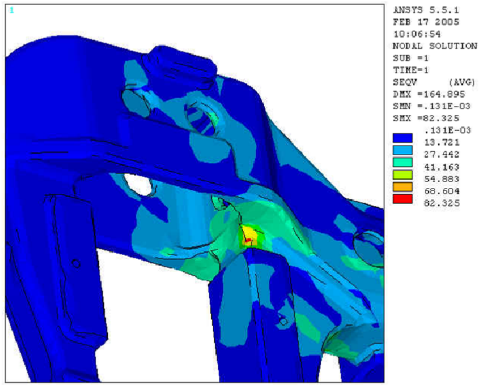
¿Cómo es la solución por el Método de los Elementos Finitos?

Tipos de Elementos

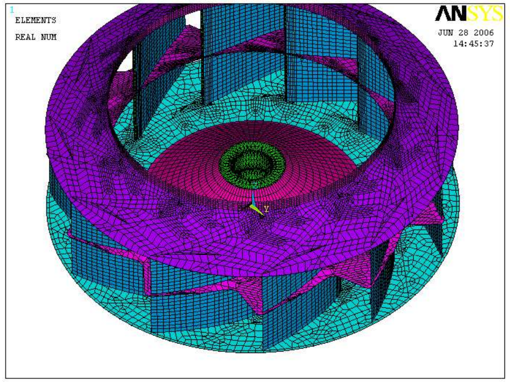
Sólidos

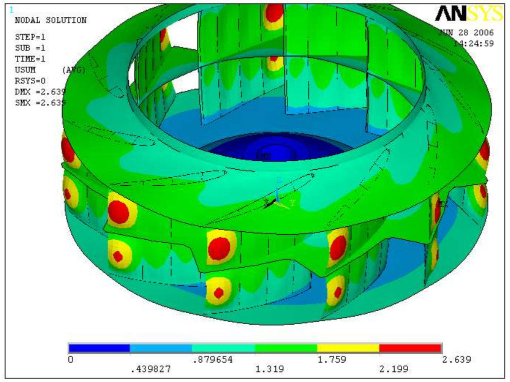


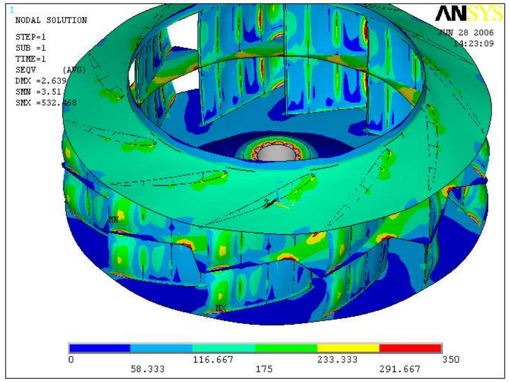




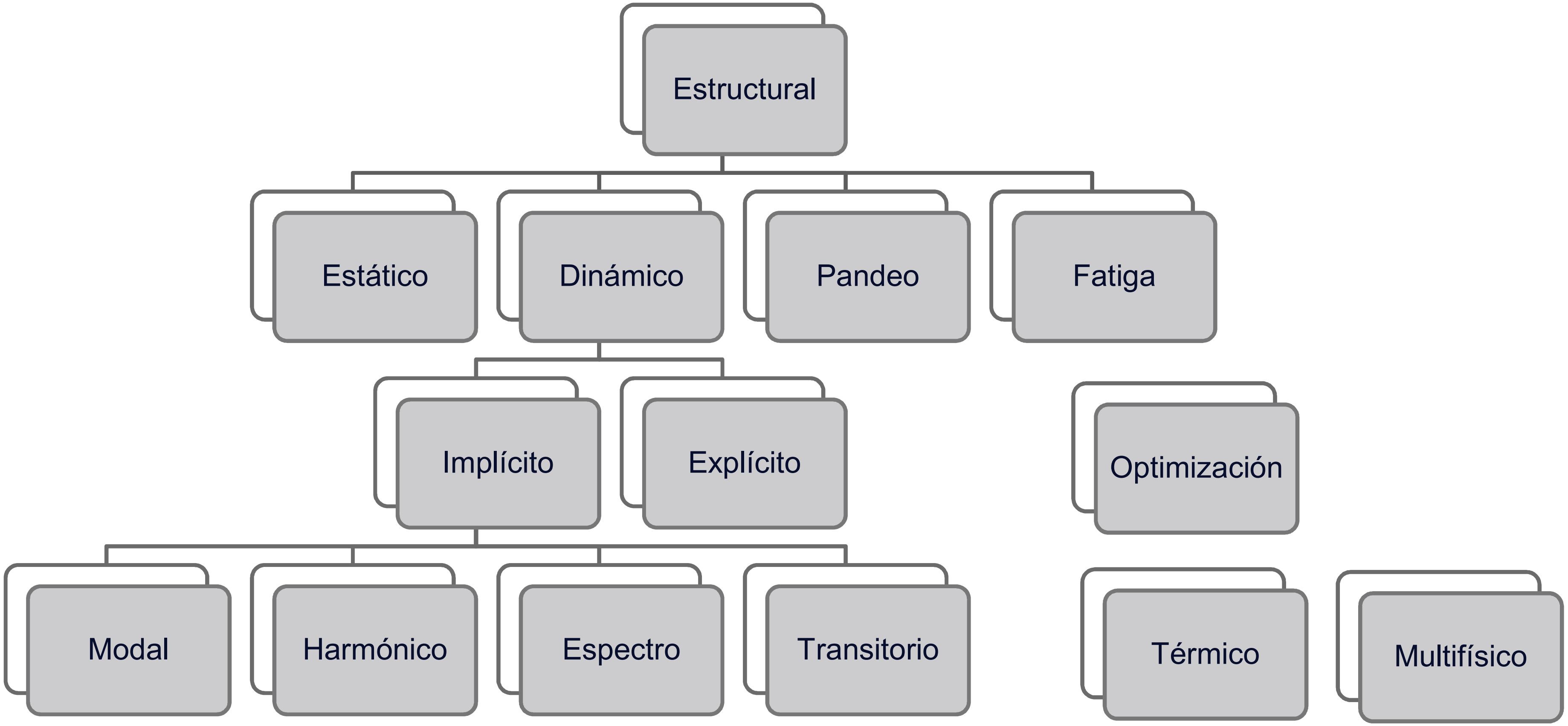
Shells







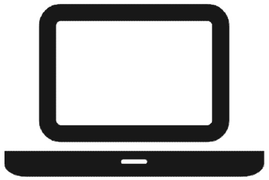
¿Cuáles son los tipos de análisis?



Conclusion – CAE advantages

- Computational advantages
- Numerical solution of differential equations

Analytic Methods



Numerical Methods

- Virtual prototyping
- Virtual laboratory
- Validation and calibration of parameters & materials

Experimental Methods

Conclusion – CAE fundamentals

CAE

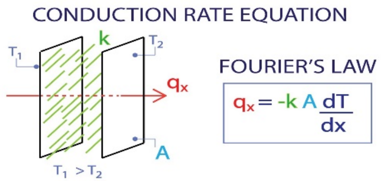
Phenomenological models

+

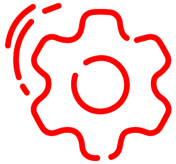
Numerical methods

+

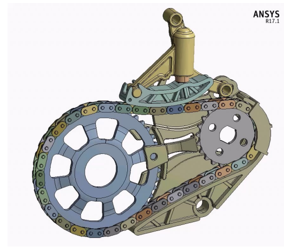
Computational resources

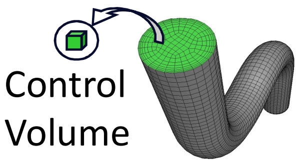


www.youtube.com: Ron Hugo

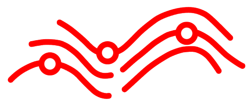


Structures

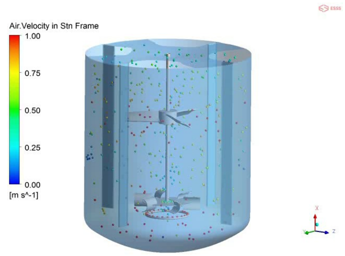




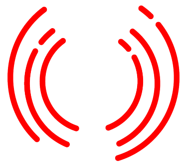
ANSYS Inc



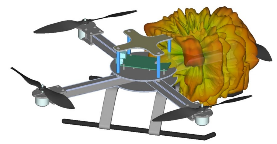
Fluids



http://rc.dartmouth.edu



EMAG



www.esss.co

Agenda

1. Introducción al CAE y el proceso de simulación
2. Fundamentos de simulación fluidodinámica
3. Fundamentos de simulación estructural
4. **Preguntas y respuestas**

WWW.ESSS.CO

Open Discussion

JOSÉ CARLOS ZART

CAE Applications Intern. Academic Area. Latin America

jzart@esss.co / www.esss.co

CONNECT WITH US:

/ESSSgroup /ESSSgroup /company/esss

Programación

- 16 de noviembre: Introducción a la simulación computacional
- 19 de noviembre: Workshop CFD – Flujo laminar en tubería recta
- 22 de noviembre: Workshop FEA – Barra con cambio de sección