EPFL

https://home.cern/science/experiments/cms

MICRO 372 - Advanced Mechanisms for Extreme Environments

Chapter 5a

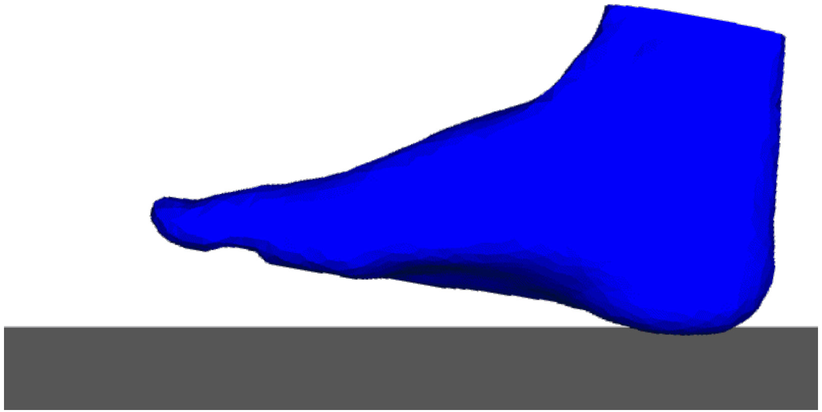
Advanced mechanisms analysis

Florent Cosandier

École polytechnique fédérale de Lausanne

Advanced mechanisms analysis: contents

- FEM theory
- FEM process flow
- Software for FEA
- Types of elements
- Meshing
- Boundary constraints
- Loads
- Types of analysis
- Postprocess



FEM versus FEA

Finite Element Method (FEM):

- Refers to the **numerical technique** used to **solve partial differential equations** (PDEs) or **boundary value problems** by dividing a complex domain into smaller, simpler elements (finite elements).
- Elements are interconnected at discrete points called **nodes**.
- FEM involves the **discretization of the problem domain**, the **formulation of element equations**, **assembly** of the global system of equations, application of **boundary conditions**, and **solution** of the resulting algebraic equations to obtain **approximate solutions**.
- FEM is a versatile method applicable to various disciplines, including **structural mechanics**, **heat transfer**, **fluid dynamics**, **electromagnetics**, and more.

Finite Element Analysis (FEA):

- Specifically refers to the **application** of the **Finite Element Method** to solve engineering problems and analyze the behavior of structures or systems under given conditions.
- FEA involves using FEM **to model and simulate the behavior of physical systems**, predict their **response to loads**, and **optimize their design**.
- Encompasses the entire process of **setting up the finite element model**, **solving it** numerically, and **interpreting the results** to make engineering decisions.
- Widely used in industries such as **aerospace**, **automotive**, **civil engineering**, **biomechanics**, and others for **design validation, optimization, and performance assessment.**

Finite Elements Analysis

- Problems in engineering are often modelled by gradient-based equations
- Solid mechanics, fluidics, electromagnetics, …

$$\frac{\partial \sigma_{xx}}{\partial x_1} + \frac{\partial \tau_{xy}}{\partial x_2} + \frac{\partial \tau_{xz}}{\partial x_3} + F_x = 0,$$
$$\frac{\partial \tau_{xy}}{\partial x_1} + \frac{\partial \sigma_{yy}}{\partial x_2} + \frac{\partial \tau_{yz}}{\partial x_3} + F_y = 0,$$
$$\frac{\partial \tau_{xz}}{\partial x_1} + \frac{\partial \tau_{xz}}{\partial x_2} + \frac{\partial \sigma_{zz}}{\partial x_3} + F_z = 0,$$

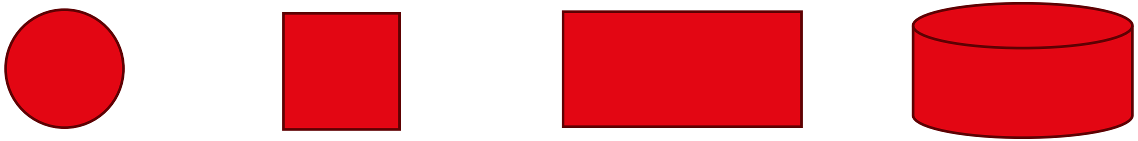
$$\nabla \cdot \overline{u} = 0$$
$$\rho \frac{D\overline{u}}{Dt} = -\nabla p + \mu \nabla^2 \overline{u} + \rho \overline{F}$$

$$\mathrm{div}\,\vec{E} = \frac{\rho}{\varepsilon_0}$$
$$\mathrm{div}\,\vec{B} = 0$$
$$\overrightarrow{\mathrm{rot}}\,\vec{E} = -\frac{\partial \vec{B}}{\partial t}$$
$$\overrightarrow{\mathrm{rot}}\,\vec{B} = \mu_0 \left(\vec{j} + \varepsilon_0 \frac{\partial \vec{E}}{\partial t} \right)$$

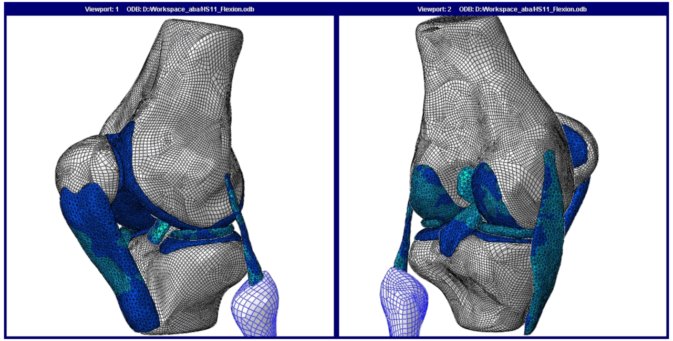
- To these **differential equations**, one add **boundary conditions**
- We then obtain **Boundary Value Problems**
- Which is defined for a specific **domain** or **geometry**

When to use FEA?

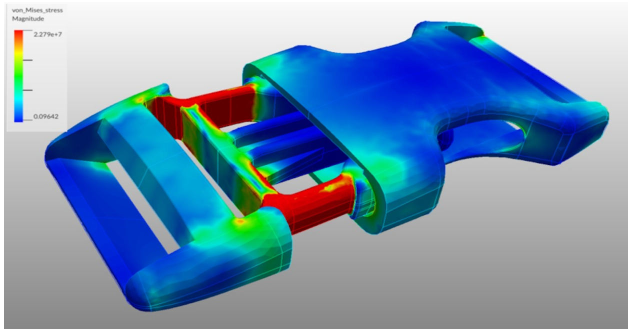
- To solve **Boundary Value Problems**, **analytical methods** like direct integration can be performed **on simple geometries only**



- **Numerical methods** are used when one need to handle **complex geometries**. FEM is one of many numerical methods.



wikipedia.org



interestingengineering

FEM theory for static solid mechanics problems

- The displacements along coordinate axes x, y and z are defined by the displacement vector u:

$$\{u\} = \{u \quad v \quad w\}$$

- Six different strain components can be placed in the strain vector:

$$\{\varepsilon\} = \{\varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx}\}$$

- For small strains, the relationship between strains and displacements is:

$$\{\varepsilon\} = [D]\{u\}$$

- where [D] is the matrix differentiation operator:

$$[D] = \begin{bmatrix} \partial/\partial x & 0 & 0 \\ 0 & \partial/\partial y & 0 \\ 0 & 0 & \partial/\partial z \\ \partial/\partial y & \partial/\partial x & 0 \\ 0 & \partial/\partial z & \partial/\partial y \\ \partial/\partial z & 0 & \partial/\partial x \end{bmatrix}$$

- Six different stress components are forming the stress vector:

$$\{\sigma\} = \{\sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx}\}$$

G. P. Nikishkov, Introduction to the Finite Element Method, 2004 Lecture Notes. University of Aizu, Aizu-Wakamatsu 965-8580, Japan

FEM theory for static solid mechanics problems

- Which are related to strains for elastic body by the Hook's law:

$$\{\sigma\} = [E]\{\varepsilon^e\} = [E](\{\varepsilon\} - \{\varepsilon^t\})$$
$$\{\varepsilon^t\} = \{\alpha T \;\; \alpha T \;\; \alpha T \;\; 0 \;\; 0 \;\; 0\}$$

- Here ε is the elastic part of strains; ε^t is the thermal part of strains; α is the coefficient of thermal expansion; T is the temperature. The elasticity matrix [E] has the following appearance:

$$[E] = \begin{bmatrix} \lambda+2\mu & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2\mu & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix}$$

- Where λ and μ are elastic Lame constants which can be expressed through the elasticity modulus E and Poisson's ratio ν:

$$\lambda = \frac{\nu E}{(1+\nu)(1-2\nu)}$$
$$\mu = \frac{E}{2(1+\nu)}$$

- The purpose of finite element solution of elastic problem is to find such displacement field which provides minimum to the functional of total potential energy Π :

$$\Pi = \int_V \frac{1}{2}\{\varepsilon^e\}^T\{\sigma\}dv - \int_V \{u\}^T\{p^V\}dV - \int_S \{u\}^T\{p^S\}dS$$

- Here $\{p^V\} = \{p_x^V \;\; p_y^V \;\; p_z^V\}$ is the vector of body force and and $\{p^S\} = \{p_x^S \;\; p_y^S \;\; p_z^S\}$ is the vector of surface force.

G. P. Nikishkov, Introduction to the Finite Element Method, 2004 Lecture Notes. University of Aizu, Aizu-Wakamatsu 965-8580, Japan

FEM theory for static solid mechanics problems

Principle of minimum potential energy

- The **potential energy of an elastic body** is defined as:

Total potential energy (V) = Strain energy (U) - Potential energy of loading (W)

$$V = U - W_z = \frac{1}{2}\int_{\Omega} \sigma_{ij}\varepsilon_{ij} d\Omega - \int_{\Omega} X_i u_i d\Omega - \int_{\Gamma} p_i u_i d\Gamma$$

Ω – domain of the elastic body, Γ – boundary, σ_{ij} – stress state tensor, ε_{ij} – strain state tensor

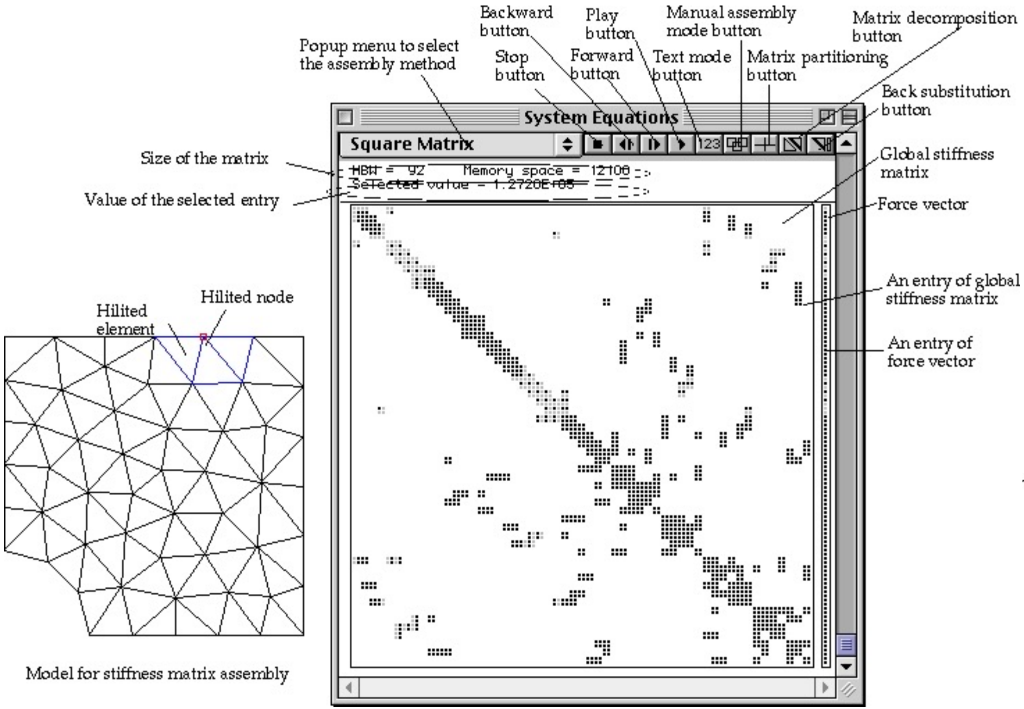
u_i – displacement vector, p_i – boundary load (traction), X_i– body loads

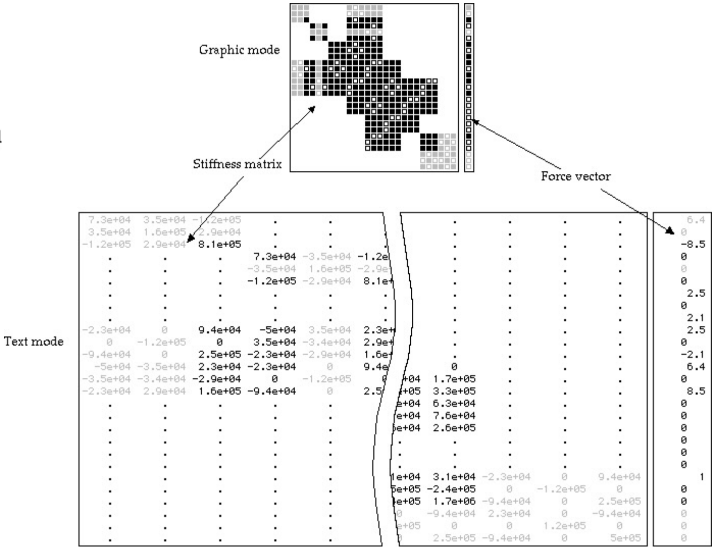
- The potential energy is a functional of the displacement field. The body force is prescribed over the volume of the body, and the traction is prescribed on the surface Γ. The first two integral extends over the volume of the body. The third integral extends over the boundary.
- The principle of minimum potential energy states that the displacement field that represents the solution of the problem fulfils the displacement boundary conditions and minimizes the total potential energy:

V(u_sol) = min(U – W)

FEM theory

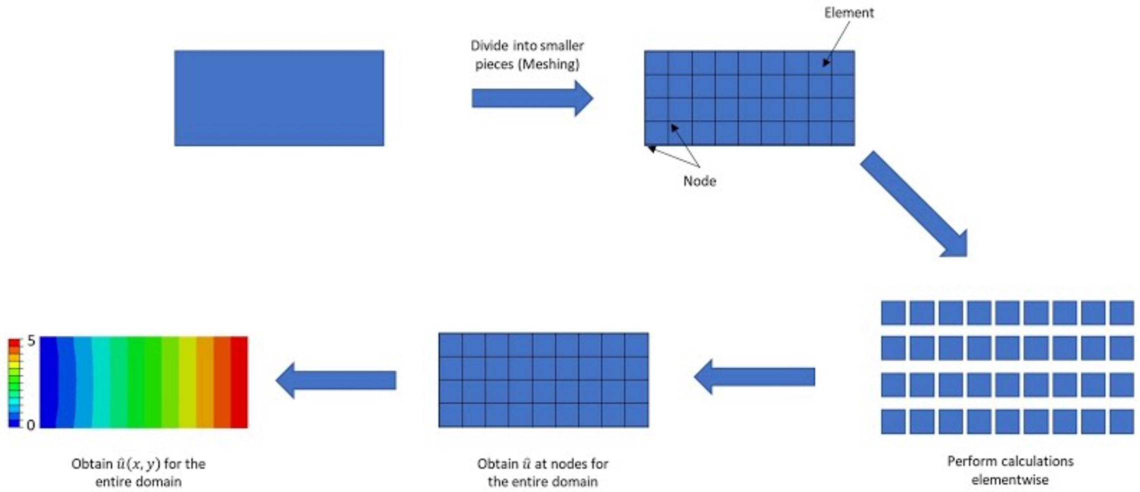
- **Examples of stiffness matrices**
- **Size of stiffness matrix** : ((number of nodes) x (DOFs per node)) ^ 2

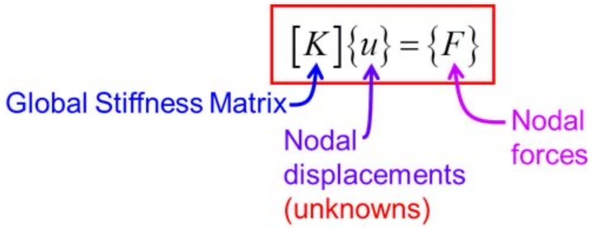




visualfea.com

FEM theory: calculation process



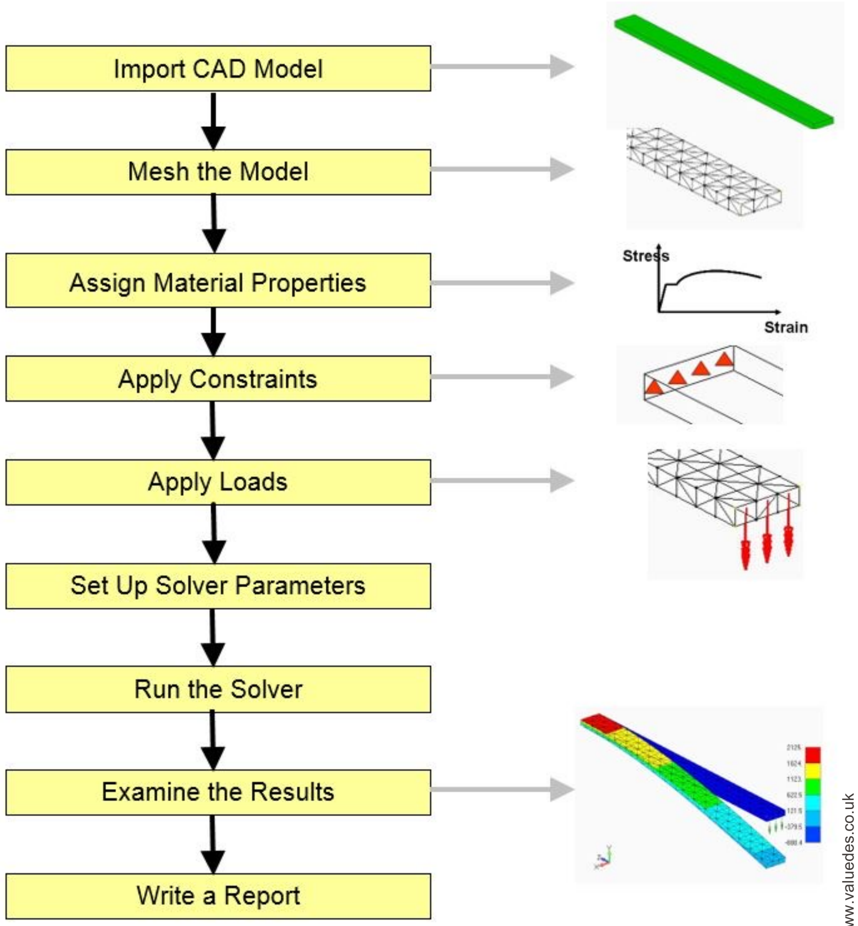


$F = KU$ → $\varepsilon = \frac{\Delta U}{U_0}$ → $\sigma = E\varepsilon$

- F: External force
- K: stiffness matrix of element
- U: displacement of each element
- ε: strain of each element
- σ: stress of each element
- E: Young modulus of material

FEA procedure

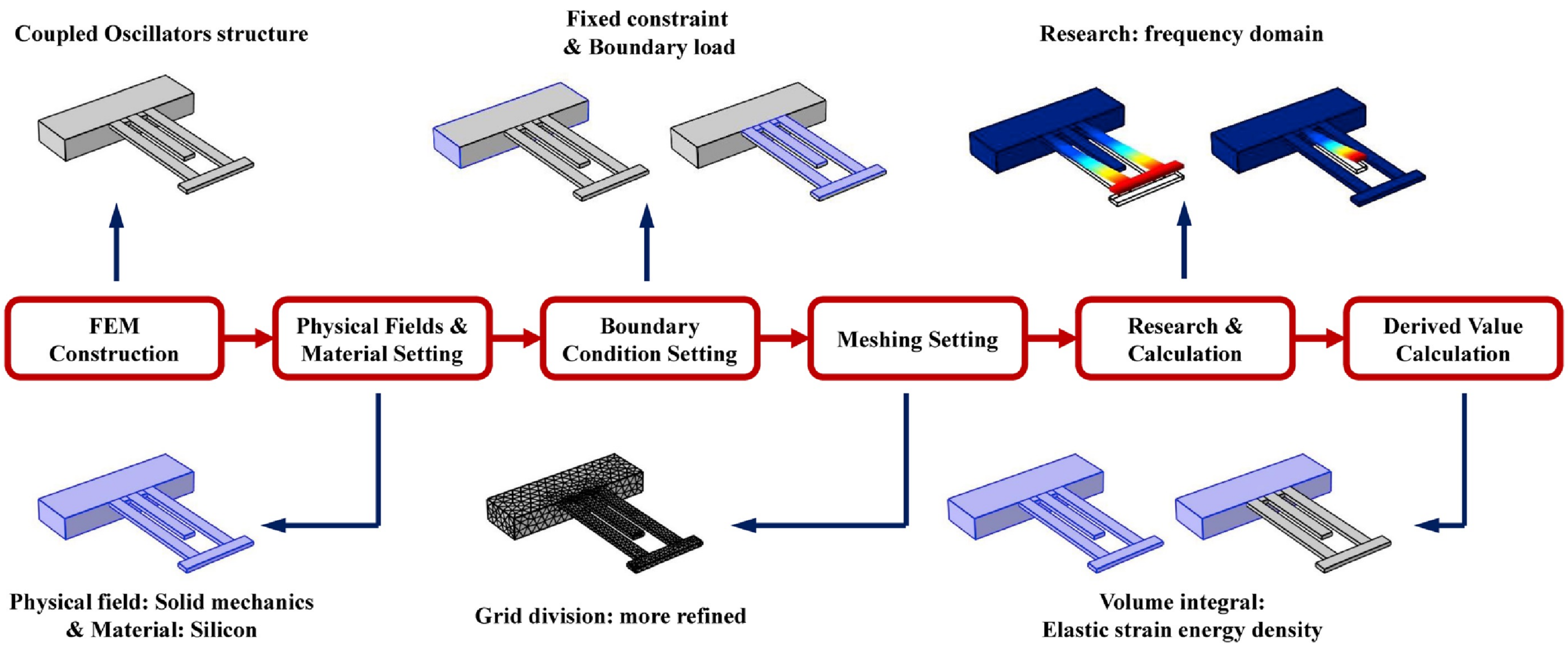
- Identify the **problem**, sketch the **structure** and **loads**
- Create the **geometry** with the FE package solid modeler or a CAD system
- Apply **material** properties
- **Mesh** the model
- Apply **boundary conditions** (constraints and loads) on the model
- **Solve** numerical equations
- **Evaluate** the results



www.valuedes.co.uk

FEA procedure

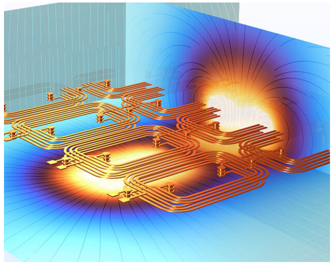
The flow diagram of specific FEM simulation method

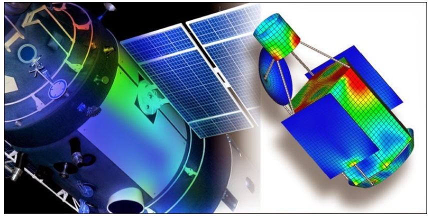


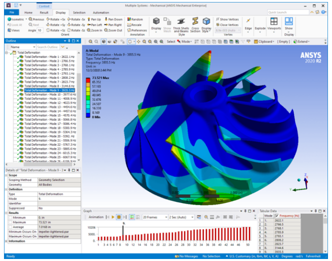
Wang, Rong & Wang, Dong & Xia, Cao & Feng, Haonan & Wang, Xin. (2022). Coupling element effect on anchor-limited Q in a synchronized oscillator with a π-shaped cantilever design. Microsystem Technologies. 28. 10.1007/s00542-022-05251-6.

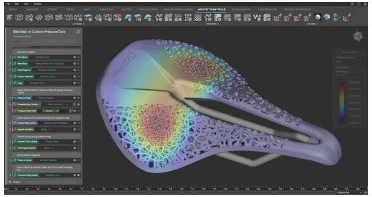
FEA softwares

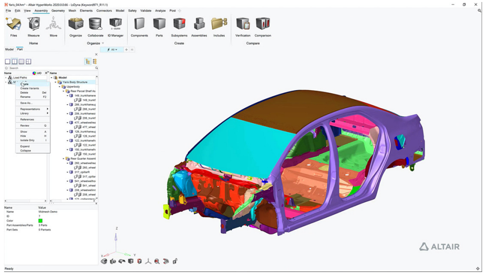
- COMSOL Multiphysics
- NASTRAN
- ANSYS
- NTOPOLOGY
- ALTAIR HyperWorks
- LS-DYNA
- ABAQUS
- …

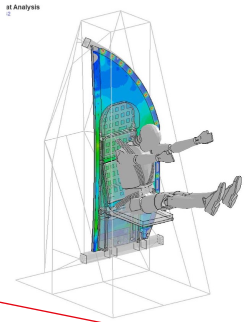


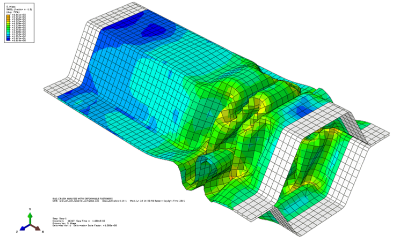






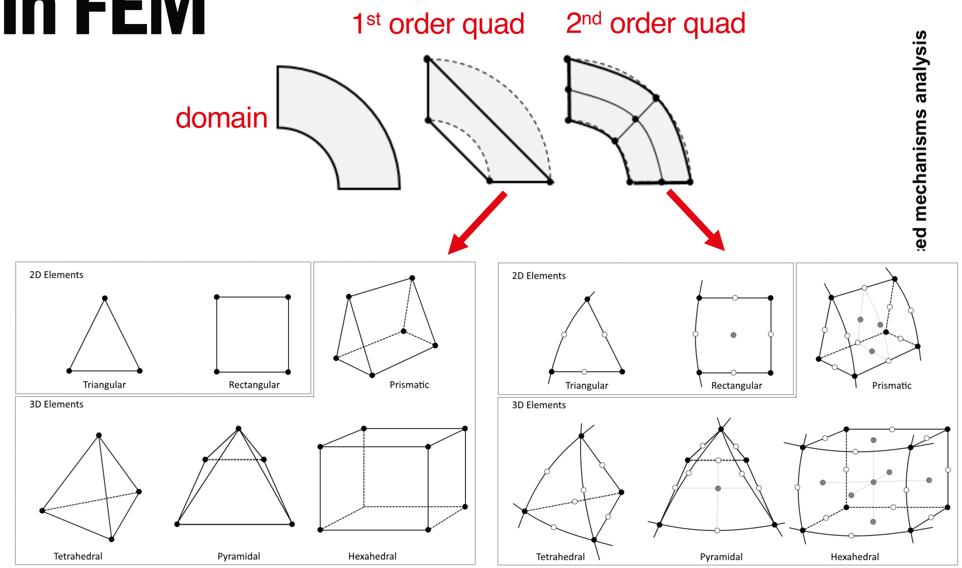


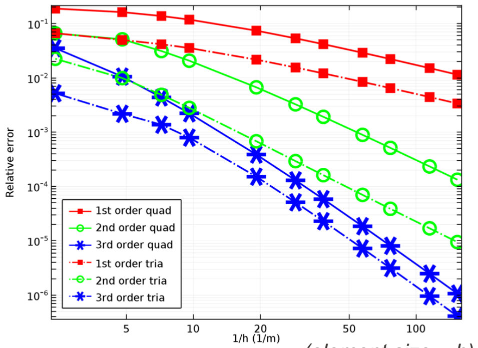




Types of elements used in FEM

- The domain is meshed into elements:
  - 2D or 3D elements
  - TRI / RECT / PRIS / TETRA / PYR / HEXA elements
  - 1st order or 2nd order
- The edges and surfaces facing a domain boundary are frequently curved
- The edges and surfaces facing the internal portion of the domain are lines or flat surfaces
- The higher the element order, the smaller the error of solution



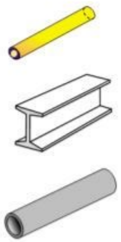
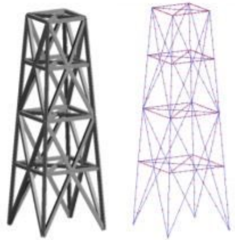
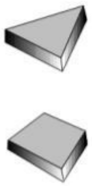
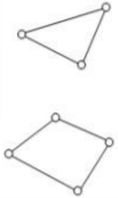
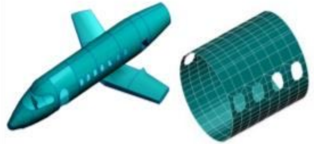
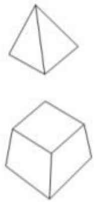
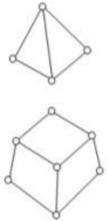
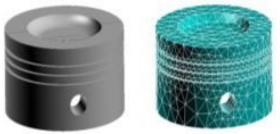


(element size = h)

comsol.com

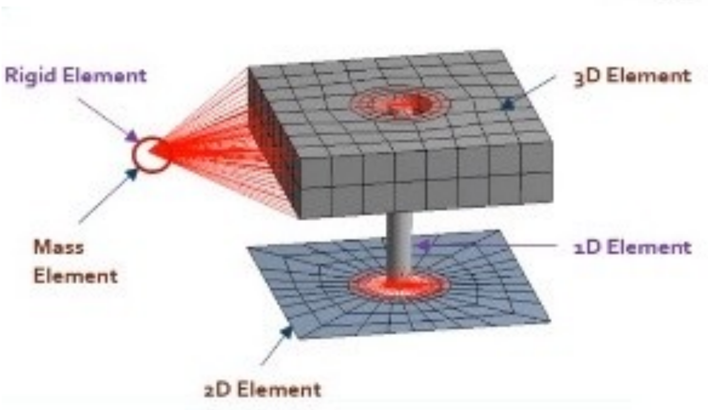
Other types of elements

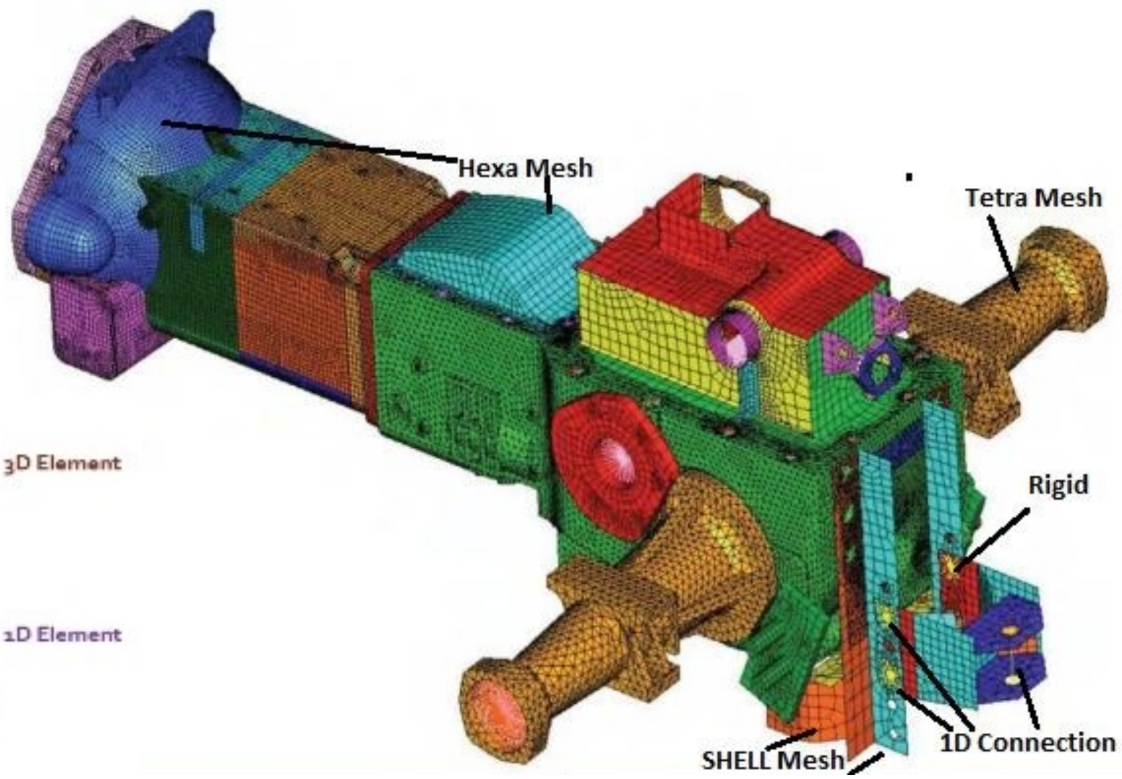
- **Flexures** are often modeled with **solid** elements or **shell** elements
- The thickness of a shell can be varied (parametrized) along the blade's axis
- **Each element** has a **specific number of DOFs**.
- **Shell** elements are used to model thin, flat or curved members with significant bending stiffness. They can be subjected to **both in-plane and out-of-plane loads.**
- **Plate** elements are used to model thin flat members with significant bending stiffness, loaded in **out-of-plane direction.**
- **Membrane** elements can transmit only in-plane forces (not moments) and have **no bending stiffness.**

	Geometry	Model name	Finite element	Example
1D		Bar/Truss Beam Tube/Pipe		
2D		Shell Plate Membrane		
3D		Solid		

feaforall.com

Examples of mesh containing several types of elements





Type of mesh

Structured meshes

- **Regular connectivity**.
- Element choices are **quadrilateral in 2D** and **hexahedra in 3D**.
- **Highly space efficient**, since the neighbourhood relationships are defined by storage arrangement.

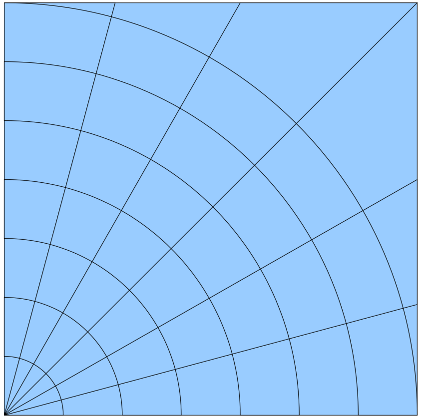
Unstructured meshes

- **Irregular connectivity**. cannot easily be expressed as a two-dimensional or three-dimensional array in computer memory.
- **Highly space inefficient** since it calls for explicit storage of neighborhood relationships
- These grids typically employ **triangles in 2D** and **tetrahedral in 3D.**

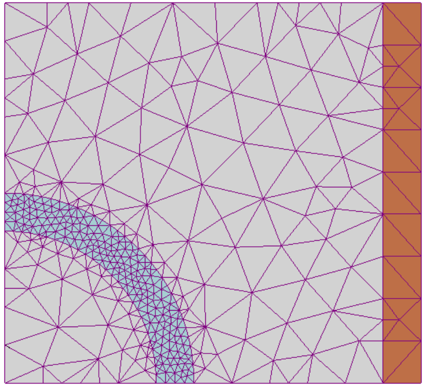
Hybrid meshes

- Contains a **mixture of structured portions** and **unstructured portions.**

Structured mesh



Unstructured mesh

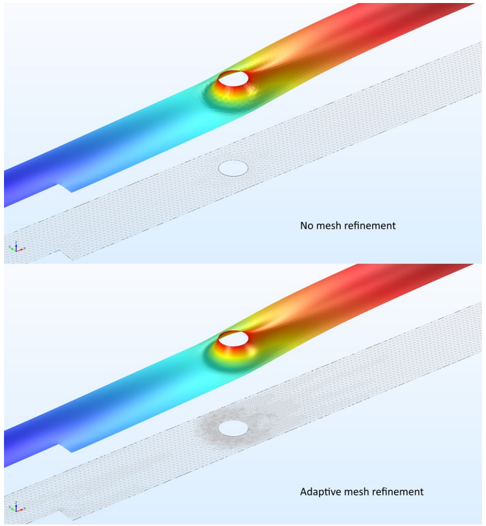


wikipedia.org

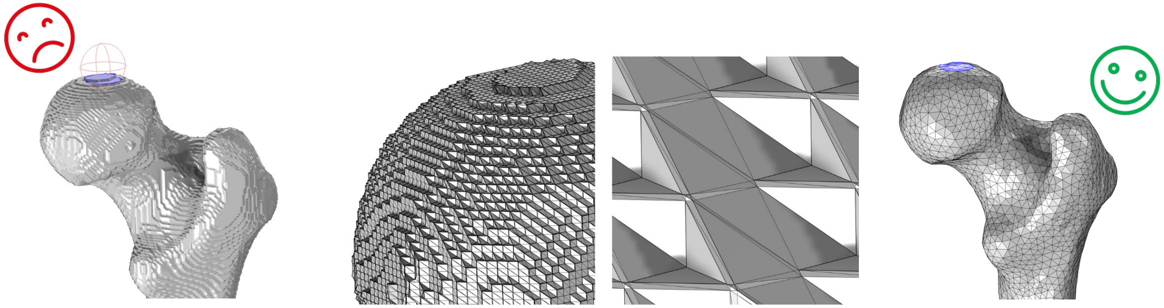
EPFL

Mesh and elements quality

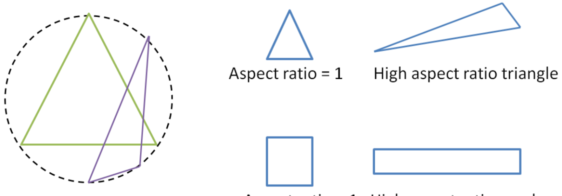
- **Hexahedral mesh** is the best mesh for **accuracy.**
- The **density of the mesh** is required to be sufficiently high to capture all the flow features.
- It **should not be so high** that it **captures unnecessary details** of the flow, thus burdening the CPU and **wasting** more **time**.
- Based on the **skewness**, **smoothness**, and **aspect ratio**, the suitability of the mesh can be decided
- The **skewness** of a grid is a good indicator of the mesh quality and suitability. **Large skewness compromises the accuracy** of the interpolated regions.
- The **change in size** should also be **smooth**. There should not be sudden jumps in the size of the cell because this may cause erroneous results at nearby nodes.

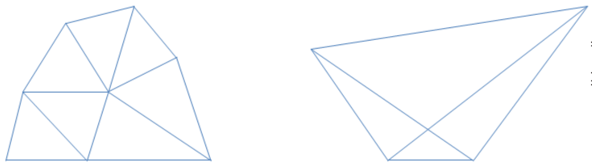


Temperature field around a heated cylinder that is subjected to a flow computed without mesh refinement (top) and with mesh refinement (bottom).



comsol.com

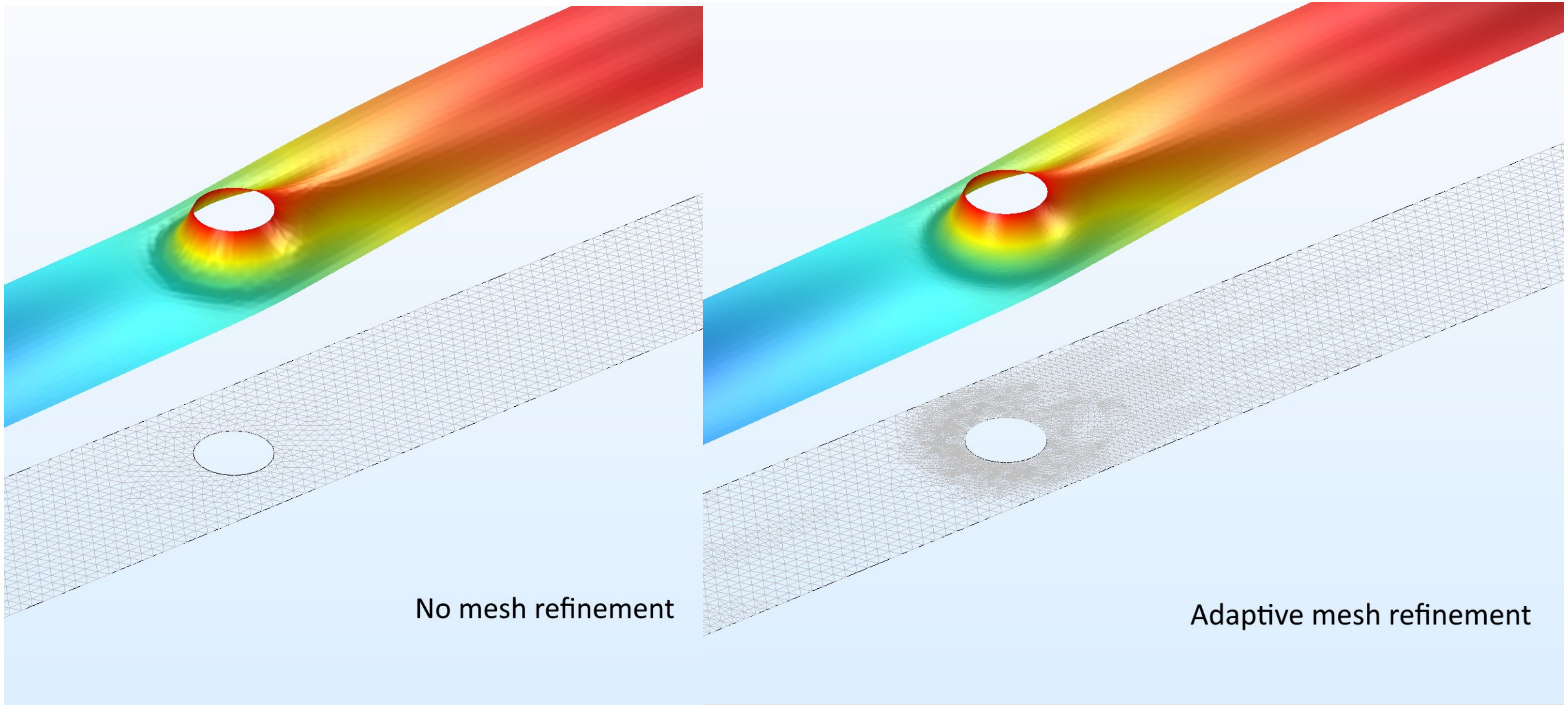




wikipedia.org

Mesh and elements quality

Temperature field around a heated cylinder that is subjected to a flow computed without mesh refinement (left) and with mesh refinement (right).



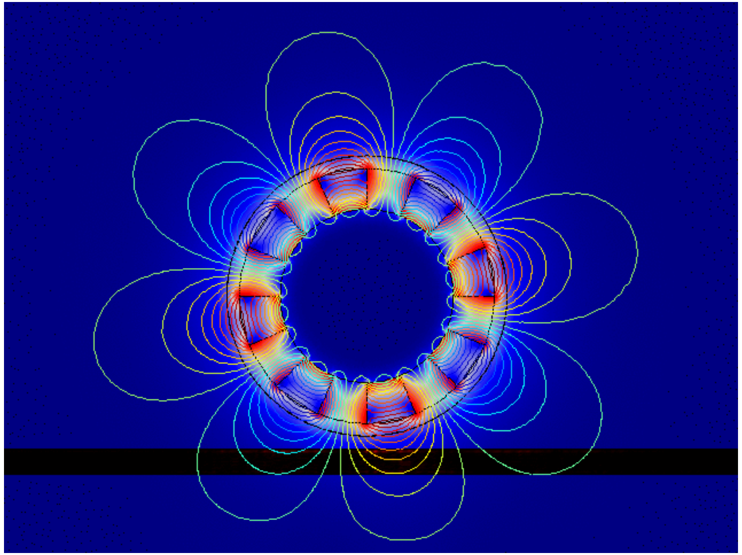
comsol.com

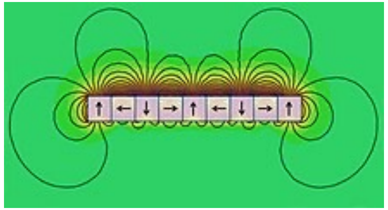
What the FEA?

Guess the following:

- What **object** is simulated?
- What **physics** is simulated?
- What kind of **study** is performed?
- What kind of **elements** are used?
- What is the **dimension** of the study?
- What **software** is used?
- Is any **symmetry** property used?
- Are **contacts** considered?
- What physical **result** is displayed?

What the FEA?

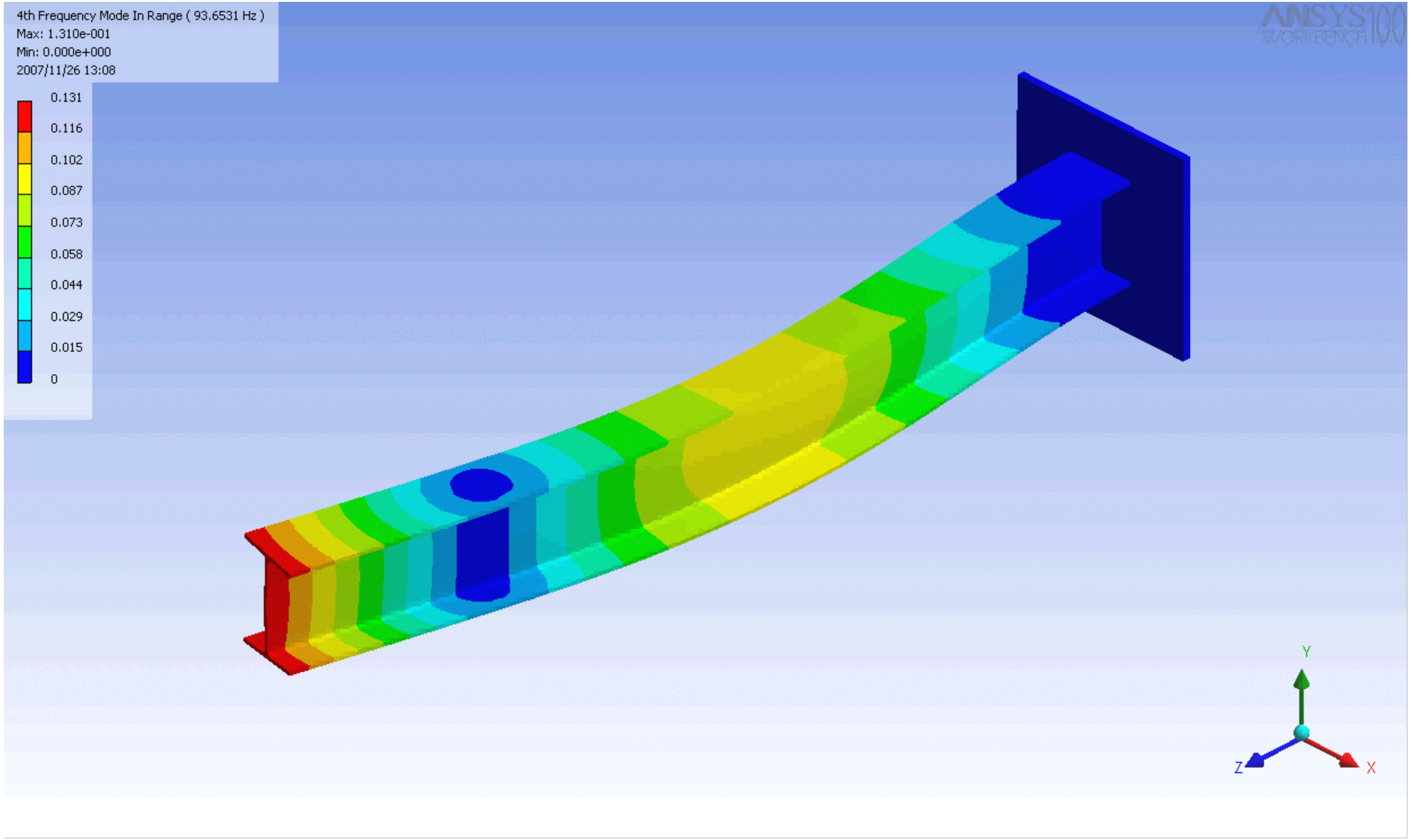




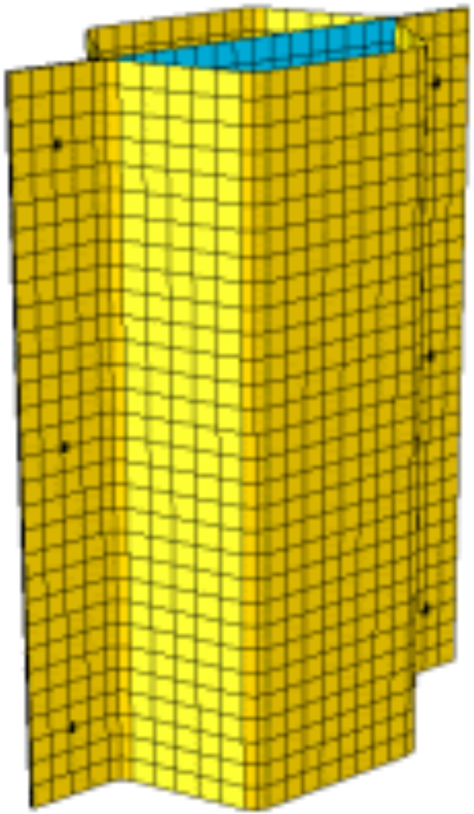
The flux diagram of a Halbach array

doc.comsol.com

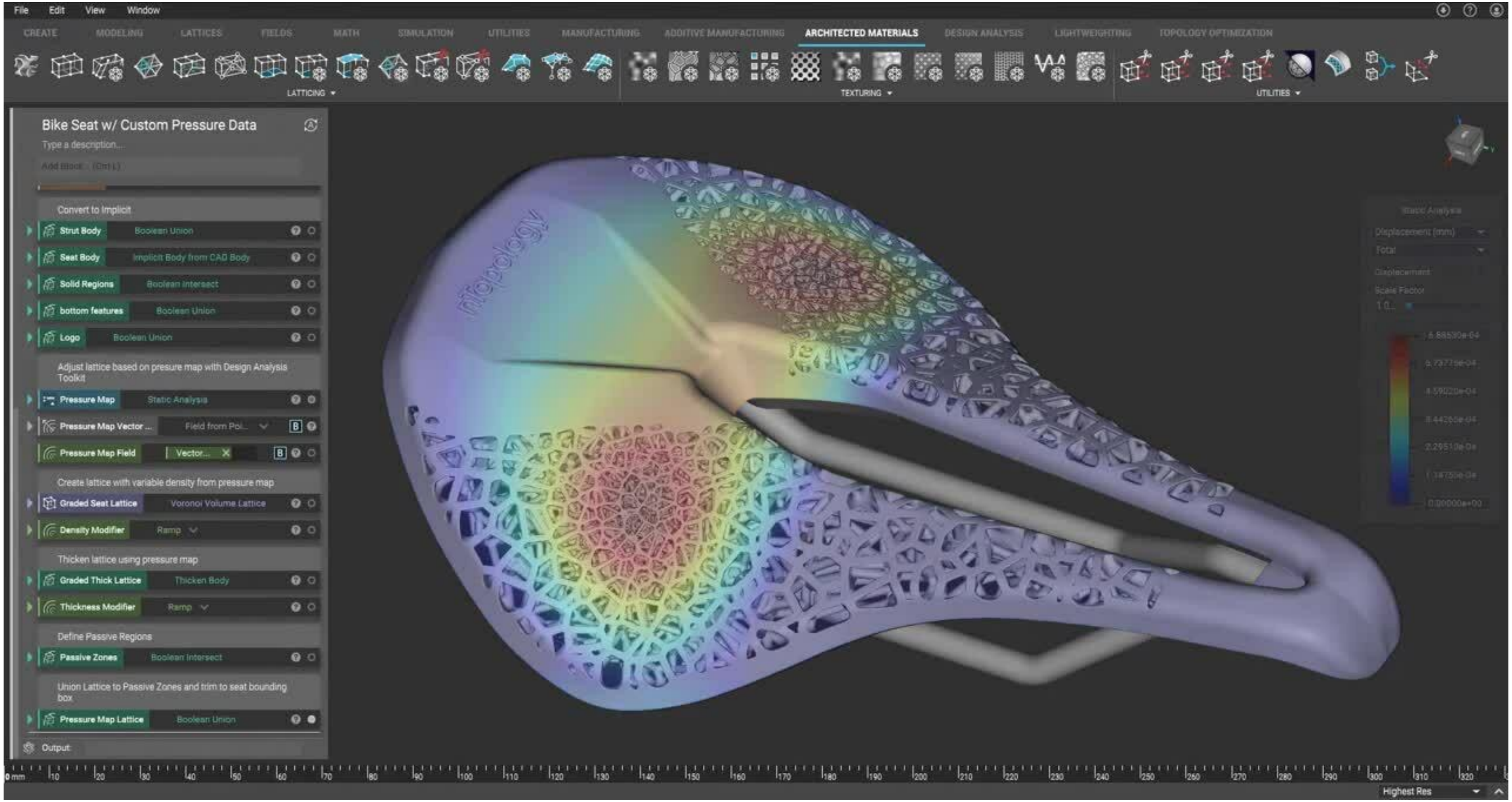
What the FEA?


4th Frequency Mode In Range (93.6531 Hz)
Max: 1.310e-001
Min: 0.000e+000
2007/11/26 13:08
0.131
0.116
0.102
0.087
0.073
0.058
0.044
0.029
0.015
0
ANSYS 10.0
WORKBENCH
Y
Z
X

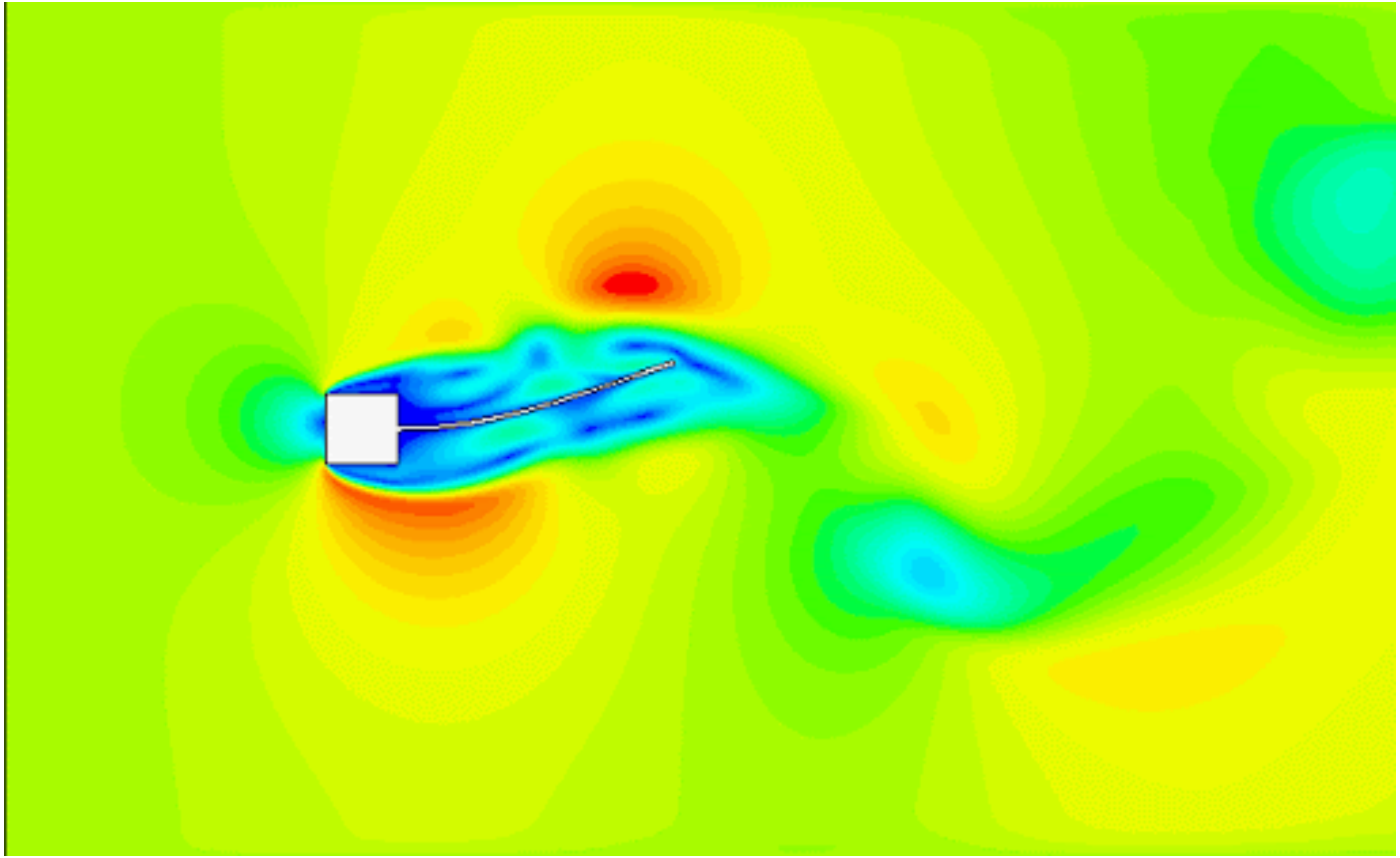
What the FEA?



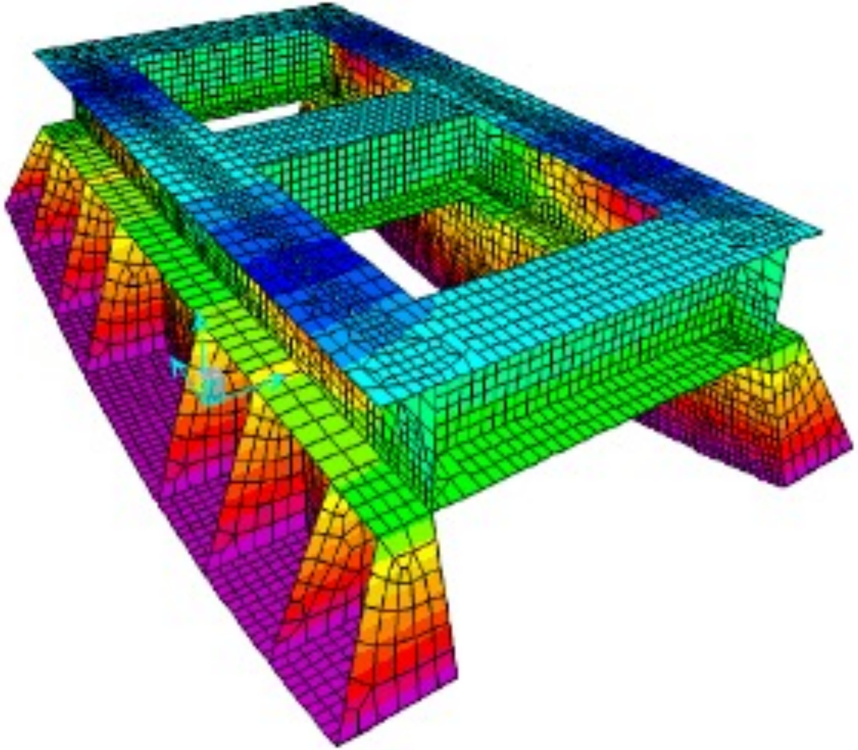
What the FEA?


File
Edit
View
Window
CREATE
MODELING
LATTICES
FIELDS
MATH
SIMULATION
UTILITIES
MANUFACTURING
ADDITIVE MANUFACTURING
ARCHITECTED MATERIALS
DESIGN ANALYSIS
LIGHTWEIGHTING
TOPOLOGY OPTIMIZATION
LATTICING
TEXTURING
UTILITIES
Bike Seat w/ Custom Pressure Data
Type a description...
Convert to Implicit
Strut Body
Boolean Union
Seat Body
Implicit Body from CAD Body
Solid Regions
Boolean Intersect
bottom features
Boolean Union
Logo
Boolean Union
Adjust lattice based on presure map with Design Analysis Toolkit
Pressure Map
Static Analysis
Pressure Map Vector ...
Field from Poi...
Pressure Map Field
Vector...
Create lattice with variable density from pressure map
Graded Seat Lattice
Voronoi Volume Lattice
Density Modifier
Ramp
Thicken lattice using pressure map
Graded Thick Lattice
Thicken Body
Thickness Modifier
Ramp
Define Passive Regions
Passive Zones
Boolean Intersect
Union Lattice to Passive Zones and trim to seat bounding box
Pressure Map Lattice
Boolean Union
Output
Highest Res

What the FEA?

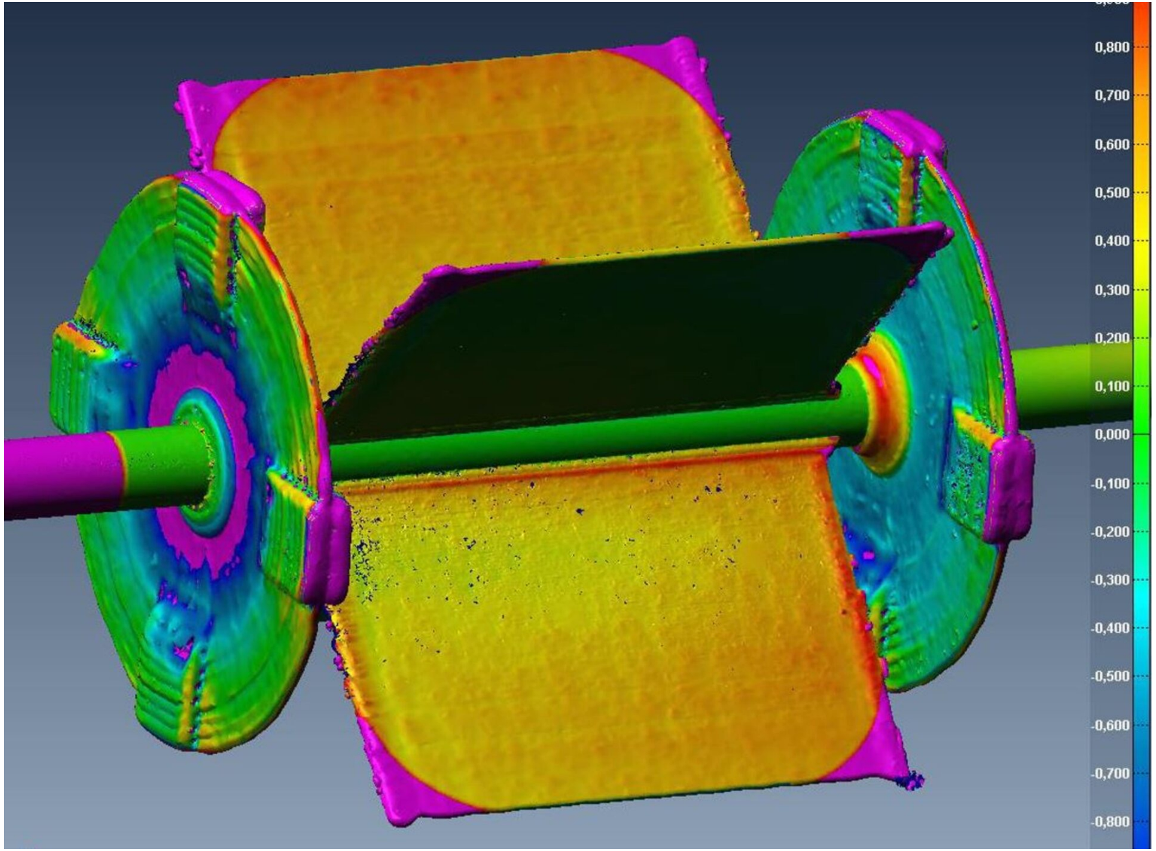


What the FEA?



pasofal.com

What the FEA?

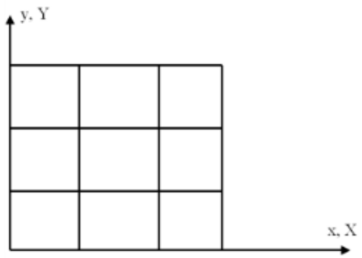

0,800
0,700
0,600
0,500
0,400
0,300
0,200
0,100
0,000
-0,100
-0,200
-0,300
-0,400
-0,500
-0,600
-0,700
-0,800

Frames in COMSOL

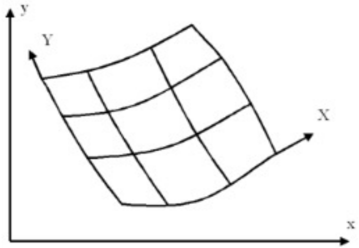
- **Spatial** frame coordinates x, y, z
- **Material** frame coordinates X, Y, Z
- **Geometry** frame coordinates Xg, Yg, Zg
- **Mesh** frame coordinates Xm, Ym, Zm
- **Displacement** field u, v, w

Spatial frame = Material frame + displacement field

$$(x, y, z) = (X, Y, Z) + (u, v, w)$$



An undeformed mesh. In the initial configuration, the spatial frame (x, y) and the material frame (X, Y) coincide.



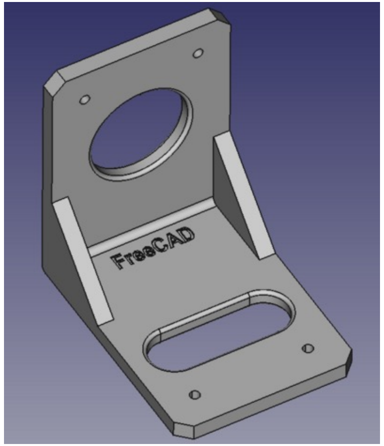
After deformation of the material, the spatial frame (x, y) remains the same, while the material coordinate system (X, Y) has been deformed, following the material.

Meanwhile, the material coordinates of each material point remain the same but its spatial coordinates have changed.

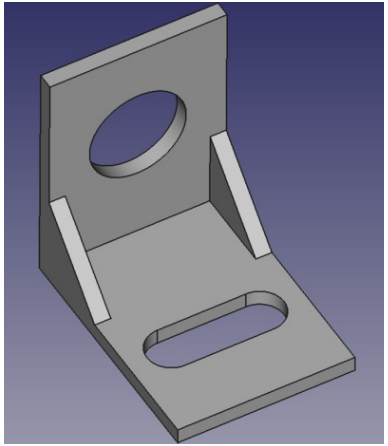
CAD preparation before FEA

- CAD designs are typically too detailed to be suitable for FEM simulations.
- It can be hard to obtain a good mesh when the part is too detailed and even if such a mesh is obtained eventually, it might be very dense, leading to unreasonable solving times.
- Thus, one should always try to simplify the design as much as possible, leaving only those geometric features that may have a significant impact on the results (strength/stiffness) and thus can't be ignored.
- The following features are typically omitted:
  - small fillets and chamfers
  - small holes
  - other small details
  - welds
  - bolts, threads
  - decorative elements (logos, engravings)

Original bracket geometry

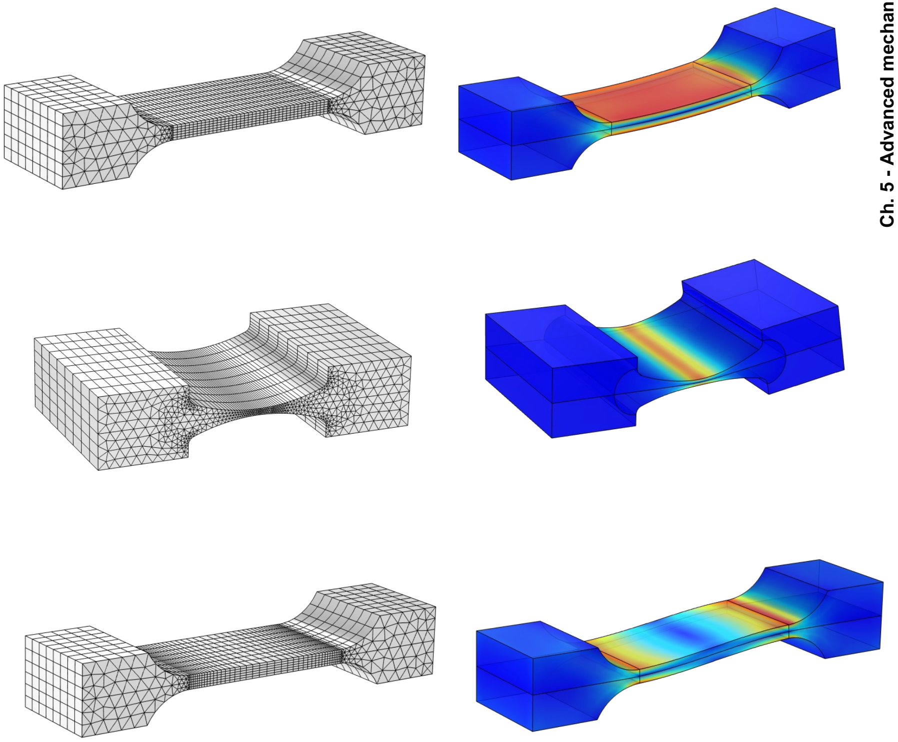


Simplified bracket geometry



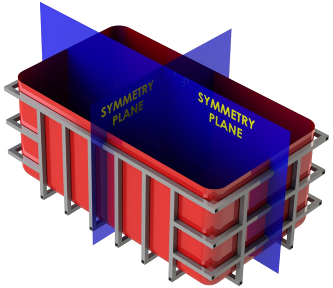
Meshing flexures

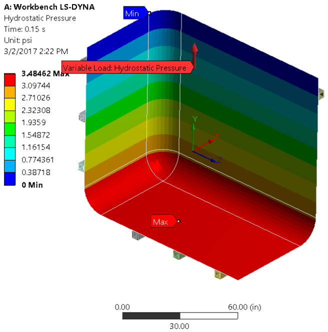
- For detailed simulations, split the flexure in two along the thickness to guarantee a symmetrical distribution
- Try to account for the bending mode and the expected stress distribution
- Increase the mesh density were the stress (and stress gradient) will be higher
- For detailed simulations, ensure to have at least 4 elements in the thickness
- For preliminary simulations, 2 elements are enough



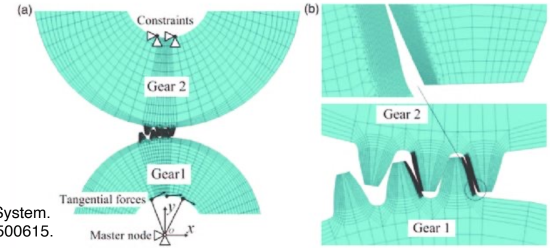
FEM types of boundary constraints

- Fixed Constraint (points, edges, surfaces)
- Prescribed displacement, velocity, acceleration
- Rigid connector (add virtual stiffness!!)
  - Impose displacement / angle
  - Apply force / torque
  - Attribute mass /inertia
  - Set rotation point manually
- Periodicity
- Symmetry
- Lumped elements(spring, damper)
- Contact (with and without friction)
- Rotating frames
- …



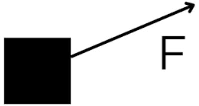


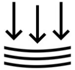
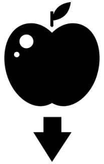
xceed-eng.com

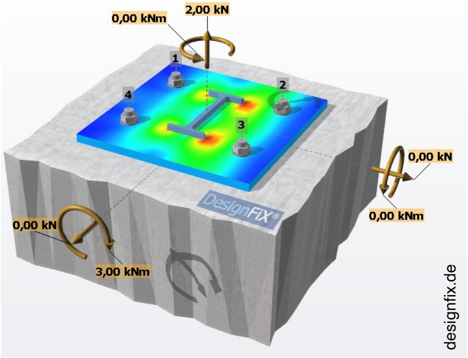


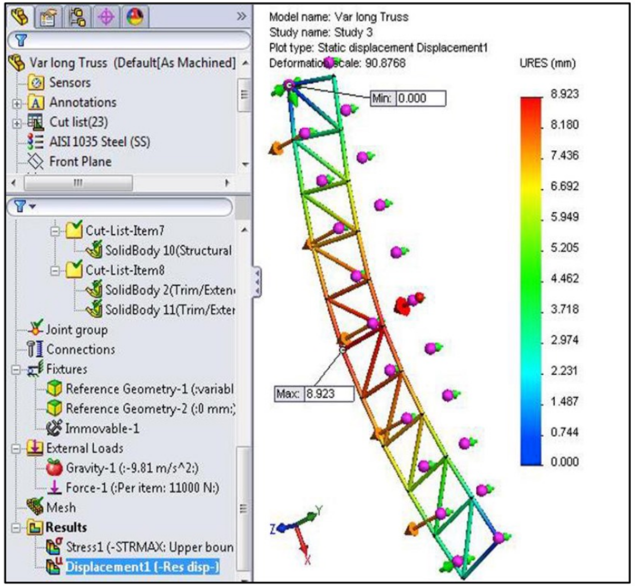
Ma, Hui & Yang, Jian & Song, Rongze & Zhang, Suyan & Wen, Bangchun. (2014). Effects of Tip Relief on Vibration Responses of a Geared Rotor System. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science. 228. 1132-1154. 10.1177/0954406213500615.

FEM types of loads

- Distributed load 
- Force (on points, on edges, on surfaces, on rigid connectors) 

- Torque (on rigid connectors) 
- Pressure 
- Gravity load 
- Inertial load 
- Thermal load 
- …





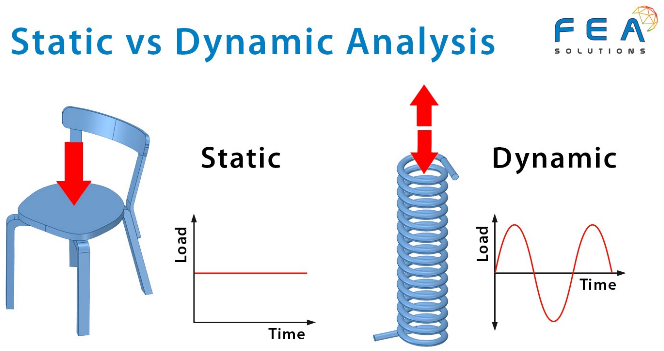
M Urdea, Static linear analysis for trusses structure for supporting pipes, IOP Conf. Series: Materials Science and Engineering 399 (2018)

FEM types of analysis

- Static
- Transient (or dynamic, or temporal)
- Eigenfrequency
- Frequency and modal
- Buckling
- Multibody dynamic
- Environmental vibration analysis

FEM types of analysis: static versus dynamic analysis

- **Static analysis** is performed if the **system does not depend on time**, and if the **loads** being applied are **constant**.
- In a **dynamic analysis**, the **system** itself, the **load** application, or **both might change with time**.
- **Static analyses** don't **consider inertia**.
- In a **dynamic analysis**, the **inertial loads** developed by the system **due to acceleration are taken into account**.
- Mathematically, the difference between static and dynamic analysis is that in a **static analysis, only the stiffness matrix** of the FEA model **is solved**.
- In a **dynamic analysis**, in addition to the **stiffness matrix**, the **mass matrix** (and **damping matrix**, if not zero) is solved as well.
- For a **dynamic analysis**, the loads can be in either the **time domain** or in the **frequency domain**.



fea-solutions.co.uk

In linear problems, the PDEs reduce to a matrix equation as:
[K]{x} = {f}

and for non-linear static problems as:
[K(x)]{x} = {f}

For dynamic problems, the matrix equations come down to:
[M]{x´´} + [C]{x´} + [K]{x} = {f}

simscale.com

FEM types of analysis: static analysis

Linear or nonlinear analysis:

- Static analysis can be performed using linear or nonlinear material and geometric models.
- Linear analysis assumes linear material behavior and small deformations.
- Nonlinear analysis considers material nonlinearities (such as plasticity, large deformations) and geometric nonlinearities (such as large displacements and rotations).

Equilibrium:

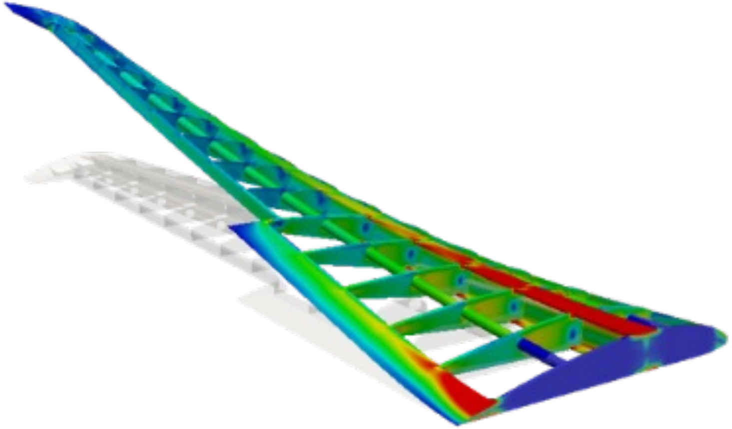
- The static analysis in FEM aims to determine the equilibrium state of the structure under applied loads.
- This involves solving the equilibrium equations, typically derived from the principle of virtual work or the minimum potential energy principle.

Solver techniques:

- Static analysis problems are typically solved using iterative solvers such as the direct stiffness method or the matrix displacement method.
- Iterative solvers are employed to solve the large system of equations resulting from the discretization of the structure into finite elements.

FEM types of analysis: static analysis

- The **large deformations** (geometric non-linearity) are generally **activated**, when simulating **flexures**
- When **stiffness** is evaluated at (or near) the equilibrium position, no need to activate the geometric non-linearity.
- Can be in **parametric study**, by **sweeping over positions** and reusing each step to calculate the next when large deformations are made.
- **Results from static study**
  - Reaction forces
  - Stiffness and stiffness non-linearity
  - Stress
  - Limit stroke
  - Buckling loads
  - Center shift
  - Parasitic motion
  - Mechanism output path



www.aa.washington.edu/

FEM types of analysis: linear versus non-linear

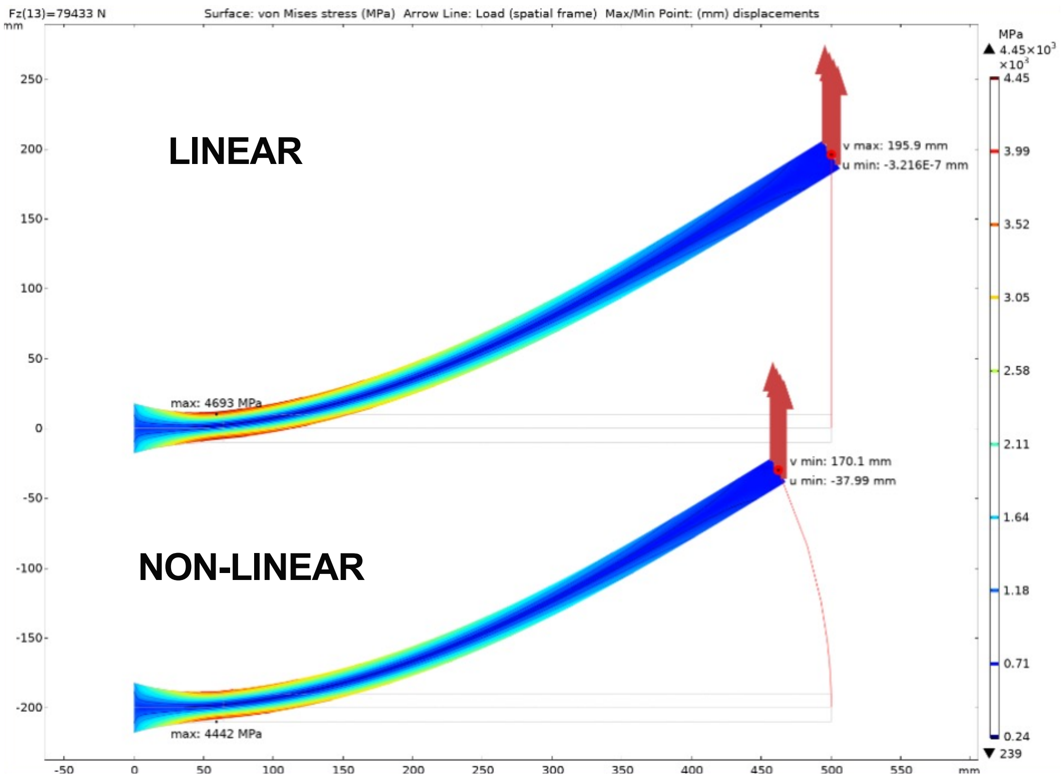
Linear analysis:

- Typically assumes small displacements and small rotations.
- Boundary conditions are often prescribed as fixed displacements or applied forces without considering geometric nonlinearities.
- Linear relationships between applied loads and structural response.

Nonlinear analysis:

- Allows for large displacements, rotations, and deformation gradients.
- Boundary conditions may include sliding contacts, frictional interactions, or nonlinear constraints, which can significantly affect the structural response.
- Considers more realistic material behaviour, boundary conditions, and deformation effects compared to linear analysis, making it suitable for analysing structures subjected to large deformations

FEM types of analysis: linear versus non-linear



www.aa.washington.edu/

Week 7 exercises and homework

- Exercise on MOODLE:
    - EXO_7_fatigue.pdf
    - EXO_7.xlsx
    - EXO_7.m
- Homework (on moodle):
    - Read through *introfem.pdf*

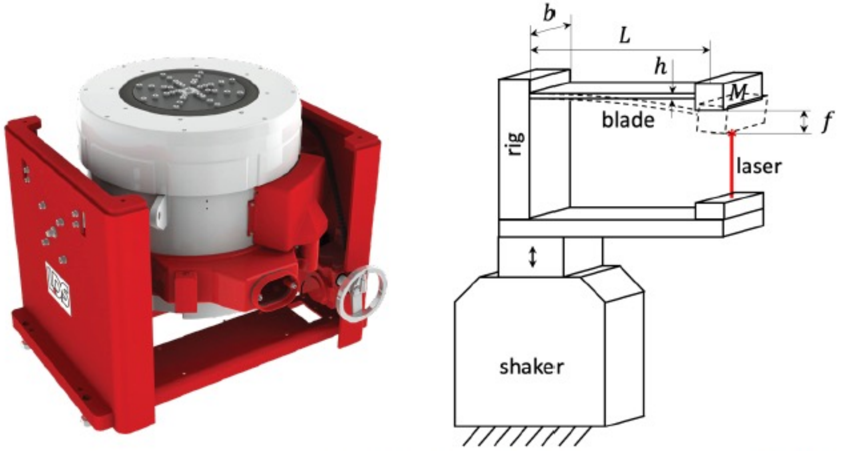


Figure 1: Left: Electromagnetic vibrating shaker. Right: Experimental setup schema of a blade mounted on top of a vibrating shaker.