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

MICRO 372 - Advanced Mechanisms for Extreme Environments

Chapter 5b

Advanced mechanisms analysis

Florent Cosandier

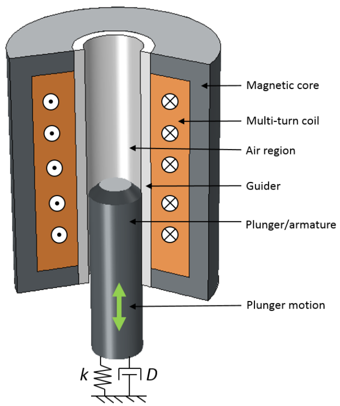
École polytechnique fédérale de Lausanne

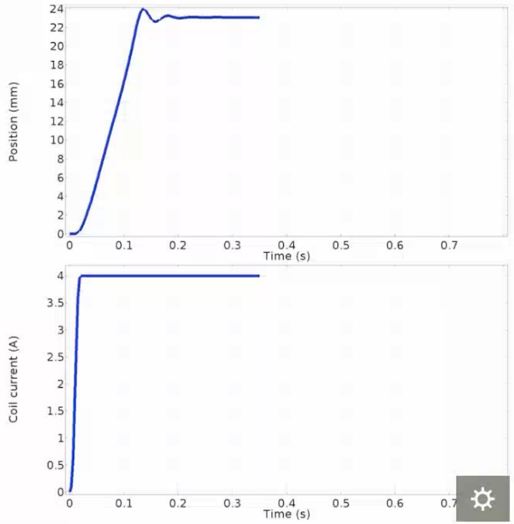
FEM types of analysis: transient (or dynamic) analysis

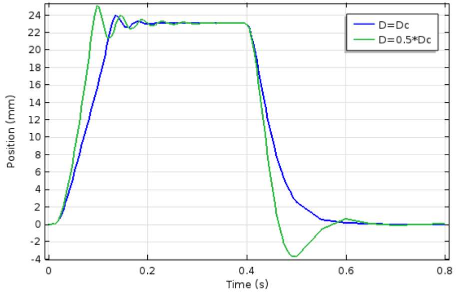
- Transient analysis requires **time integration methods** to solve the equations of motion.
- Common time integration schemes include **explicit methods** (forward euler, runge-kutta) and **implicit methods** (backward euler, newmark).
- **Time-varying loading conditions**, such as: **impulsive loads**, **harmonic loads**, **time-dependent forces**, or **prescribed displacements over time**.
- **Initial conditions**, such as initial displacements and velocities, **are required** to start the transient analysis.
- The **accuracy** and **stability** of transient analysis depend on the **selection of an appropriate time step size**.
- **Avoid to strong transient phenomenon**, otherwise algorithm can calculate solutions on very small time steps
- **Inertial terms** matter

FEM types of analysis: transient (or dynamic) analysis

- The animation below demonstrates the dynamics of a plunger for the applied transient current in the coil.
- On the left: 3D visualization of an electromagnetic plunger with the magnetic flux density norm plot,
- On the right: the corresponding plot for the plunger position and the current through the solenoidal coil.
- Far right: the damping coefficient is changed

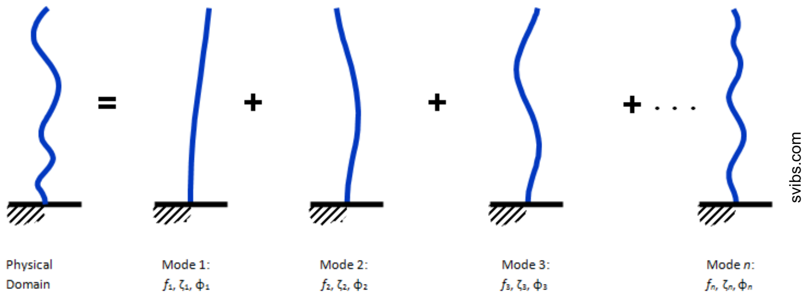






FEM types of analysis: transient (or dynamic) analysis

- Transient analysis involves simulating the behavior of the system over a **specified time duration**.
- **Time steps** have generally to be specified
- The **duration of the analysis** depends on **the nature of the dynamic** loading and **the time scale** of interest for the response of the structure or system.
- **Damping** may be included in the analysis **to account for energy dissipation** and **to stabilize the numerical solution**, especially for systems with high-frequency components.
- **Modal analysis** can be employed **to reduce the computational cost** of transient analysis by considering **only the dominant modes of vibration** that significantly contribute to the dynamic response of the structure.



Numerical analysis: transient (or dynamic) analysis

- Numerical solving of **ODE** with **MATLAB**: Van der Pol oscillator example

$$y_1'' - \mu(1 - y_1^2)y_1' + y_1 = 0$$

Rewrite this equation (with $\mu = 1$) as a system of first-order ODEs by making the substitution $y_1' = y_2$:

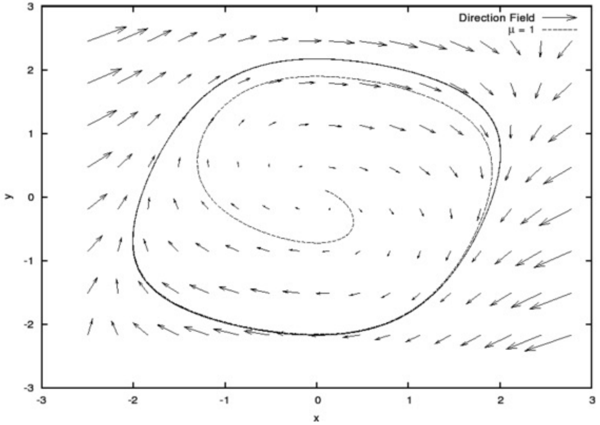
$$y_1' = y_2$$
$$y'_2 = \mu(1 - y_1^2)y_2 - y_1$$

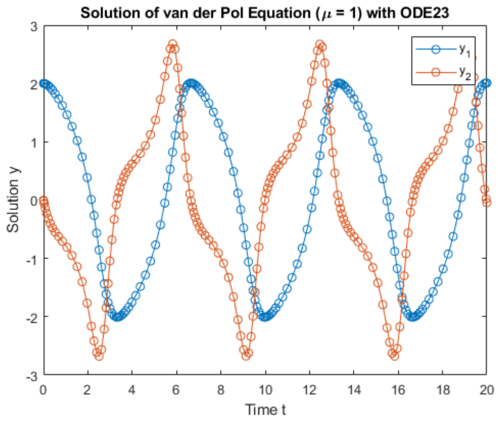
```
function dydt = vdp1(t,y)
dydt = [y(2); (1-y(1)^2)*y(2)-y(1)];
```

diff. equation

```
[t,y] = ode23(@vdp1,[0 20],[2; 0]);
```

solver — function — time — solution — initial cond.

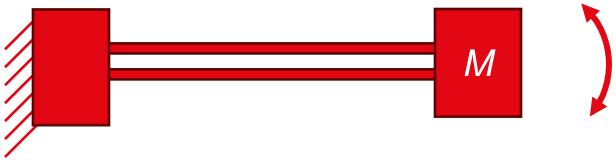




EPFL

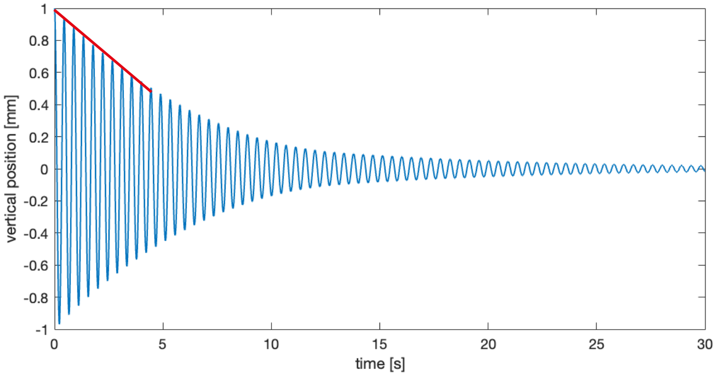
Numerical analysis: transient (or dynamic) analysis

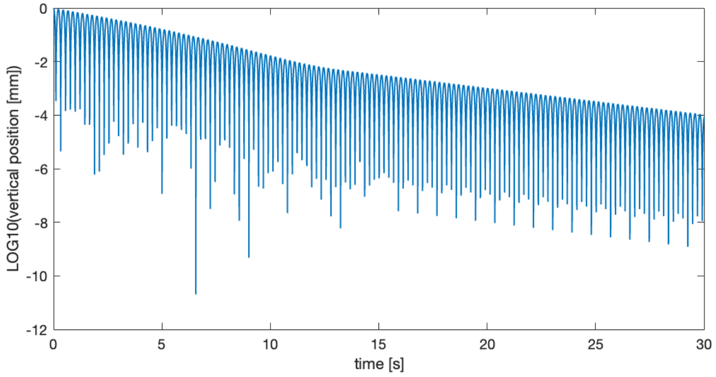
- Numerical solving of ODE with MATLAB: oscillating blades with friction



$$M\ddot{x} + c\dot{x} + kx + F = 0$$

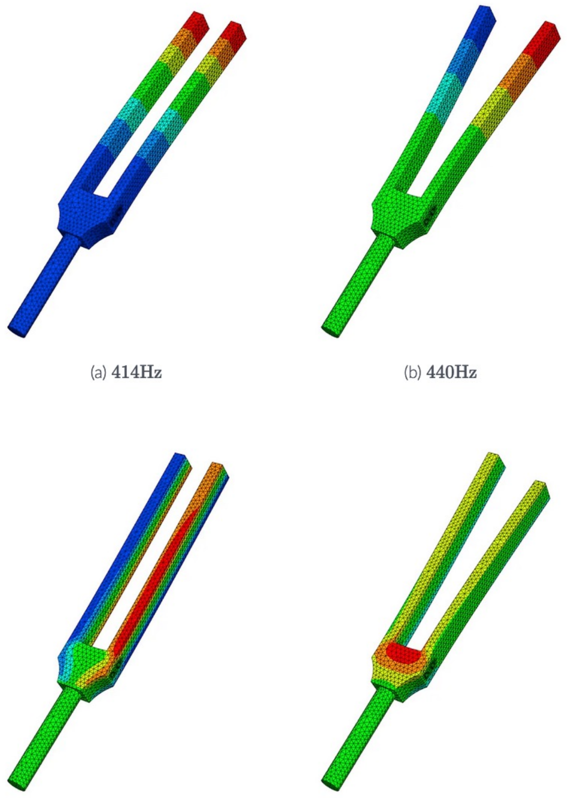
```
[t,y] = ode23(@obwf2,[0 30],[1, 0]);

function dydt = obwf(t,y)
    dydt = zeros(2,1);

    k = 200;
    c = 0.2;
    M = 1;
    F = 0;
    if abs(y(1)) > 0.1
        F = 1*sign(y(2));
    end

    dydt(1) = y(2);
    dydt(2) = 1/M*(-F-c*y(2)-k*y(1));
end
```





FEM types of analysis: eigenfrequency analysis

- Eigenfrequency analysis calculates the **natural frequencies of vibration** at which a structure or **system tends to oscillate without external excitation**.
- These natural frequencies are determined by **solving the eigenvalue problem** associated with the dynamic equations of motion.
- Eigenfrequency analysis also provides the corresponding **mode shapes** or **mode vectors** associated with each natural frequency.
- **Mode shapes** represent the **spatial distribution of displacements** and describe the vibrational behavior of the structure or system **at a particular natural frequency**.

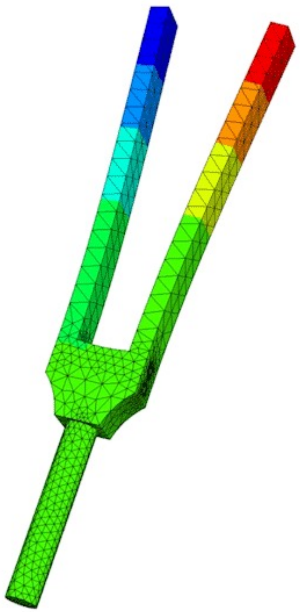


(a) 414Hz (b) 440Hz

(c) 637Hz (d) 798Hz

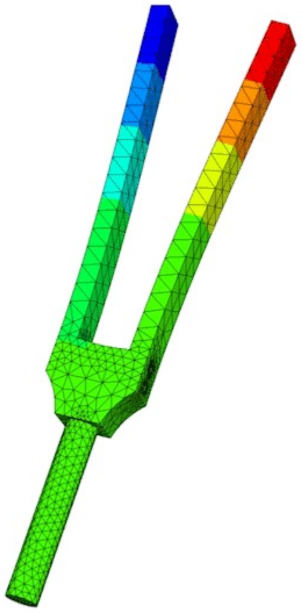
FEM types of analysis: eigenfrequency analysis

- **Eigenfrequency analysis** is typically performed assuming **linear material behavior** and **small deformations**.
- This assumption simplifies the dynamic equations of motion and allows for **efficient computation** of natural frequencies and mode shapes.
- Proper **boundary conditions** must be applied to simulate the structural constraints accurately.



FEM types of analysis: eigenfrequency analysis

- **Eigenfrequency analysis** involves **solving the generalized eigenvalue problem**, where the dynamic **stiffness matrix** of the structure is **multiplied by the mode shape vector** and **set equal to the mass matrix multiplied by the mode shape vector times the natural frequency squared**.



$$\boldsymbol{K} \cdot \varphi_i = \lambda_i \cdot \boldsymbol{M} \cdot \varphi_i$$

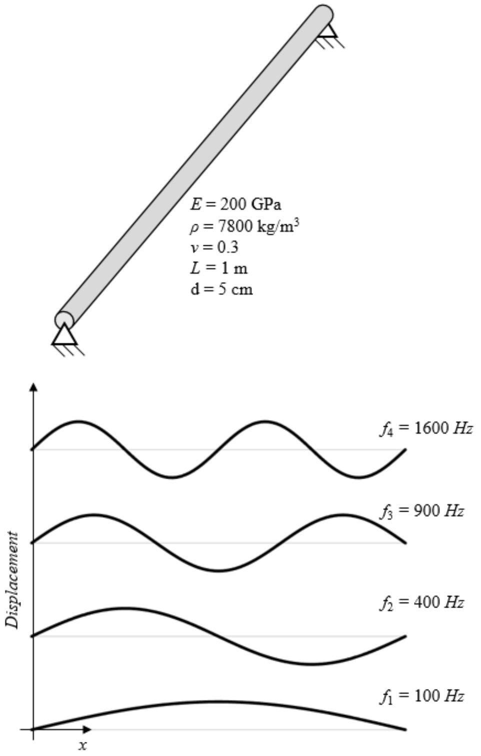
$\boldsymbol{K}$	Stiffness matrix	$\boldsymbol{M}$	Mass matrix
φ_i	i^{th} eigenmode	λ_i	i^{th} eigenvalue
		$(\lambda_i = \omega_i^2)$	

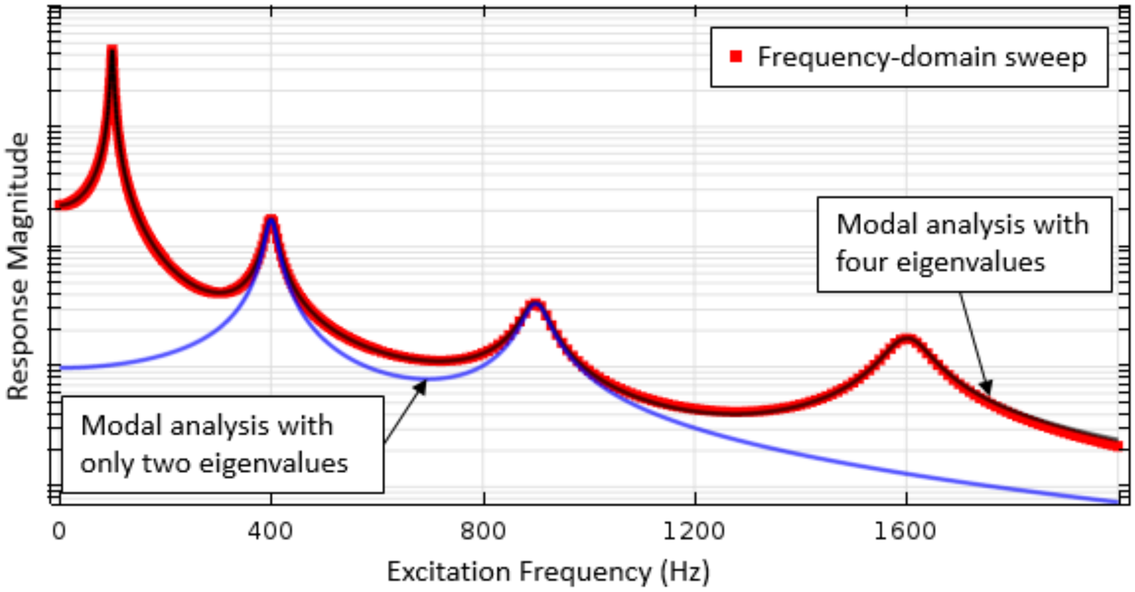
FEM types of analysis: frequency (modal) analysis

- **Frequency analysis** enables the prediction of the **spectral response** of the **mechanical system** to **dynamic (harmonic) loading conditions**. This includes determining the **amplitude response**, **phase response**, and **frequency response curves** over the specified frequency range.
- **Frequency modal analysis** allows for **analyzing the spectral response of the mechanical system** over a specified frequency range, while **only considering predetermined eigenfrequencies**. This allows to reduce the calculations required to perform the analysis.

FEM types of analysis: frequency (modal) analysis

- The first **four resonant frequencies** and **mode shapes** of a **pinned circular beam**. And a comparison of a **frequency-domain sweep** and **frequency-domain modal** study using a smaller number of eigenvalues.



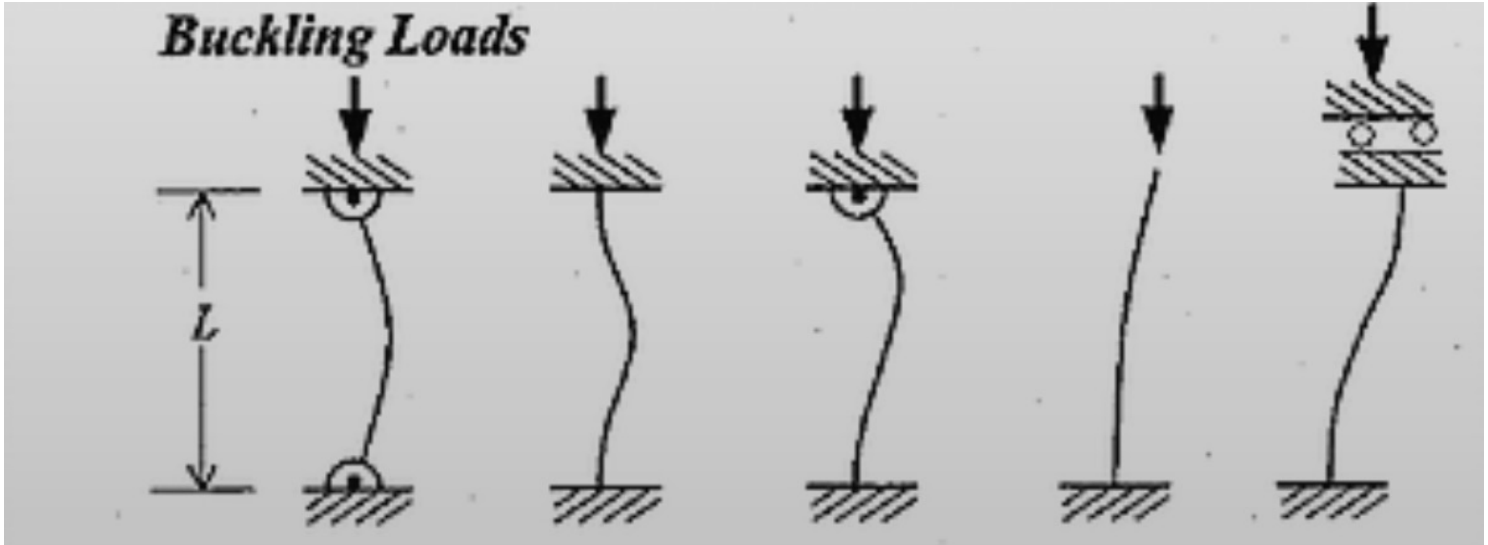


comsol.com

FEM types of analysis: buckling analysis

- Buckling analysis calculates the **critical load** at which the structure experiences buckling instability.
- Buckling analysis involves **solving the eigenvalue problem** associated with the **linearized stability equations of the structure**. This problem results in determining the **eigenvalues** (buckling load factors) and **eigenvectors** (buckling modes) that represent the buckling behavior of the system.
- Buckling analysis is typically performed assuming **linear material behavior** and **small deformations**. This assumption simplifies the stability equations and allows for the efficient computation of the buckling load.
- Buckling analysis may **consider initial geometric imperfections** in the structure. Imperfections can affect the buckling behavior and may lead to earlier onset of buckling compared to idealized, perfectly straight structures.

FEM types of analysis: buckling analysis



Buckling Load	$\frac{\pi^2 EI}{L^2}$	$\frac{4\pi^2 EI}{L^2}$	$\frac{2.045\pi^2 EI}{L^2}$	$\frac{\pi^2 EI}{4L^2}$	$\frac{\pi^2 EI}{L^2}$
Effective Length	L	$0.5L$	$0.699L$	$2L$	L

FEM types of analysis: buckling analysis

- In an eigenvalue buckling **general problem** we look for the **loads** for which the model **stiffness matrix becomes singular**, so that the problem:

$$Kv = 0$$

has nontrivial solutions.

- K is the tangent stiffness matrix when the loads are applied, and the
- v are nontrivial displacement solutions.
- The applied loads can consist of
  - pressures,
  - concentrated forces,
  - nonzero prescribed displacements,
  - and/or thermal loading.

FEM types of analysis: buckling analysis

- In an eigenvalue buckling **specific problem** we look for the **loads** for which the model **stiffness (including stress stiffening) matrix becomes singular**, so that the problem:

$$(K_0 + \lambda_i K_\Delta) v_i = 0$$

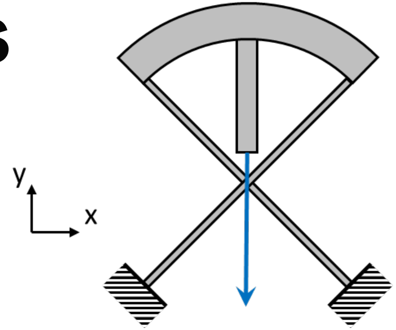
has nontrivial solutions.

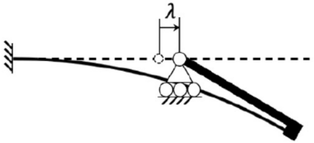
- K_0 is the tangent stiffness matrix when the loads are applied
- λ_i is the i^{th} eigenvalue (used to multiply the loads which generated K_Δ)
- K_Δ is the stress stiffening matrix (or stiffness increasing due to the buckling load)
- v_i is the i^{th} nontrivial displacement solutions.

FEM types of analysis: buckling analysis

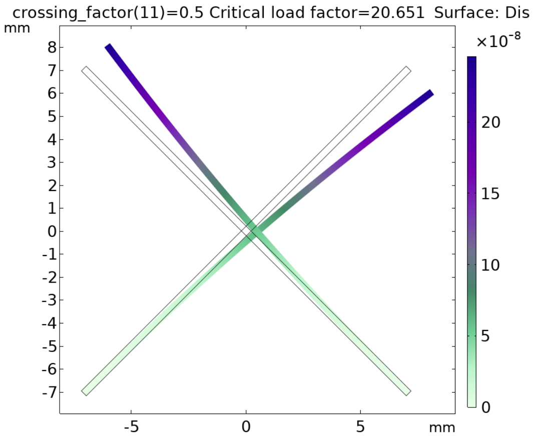
Example of a **separated crossed blades pivot**:

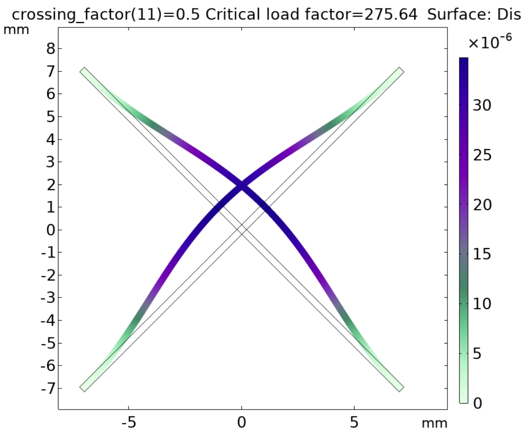
- First mode: 20 N → buckling of the pivot, $K_\theta \rightarrow 0$
- Second mode: 275 N → buckling of the blades
- Third mode: -40N → buckling of the pivot, due to the displacement of the centre of rotation, does not appear on the linear buckling analysis!

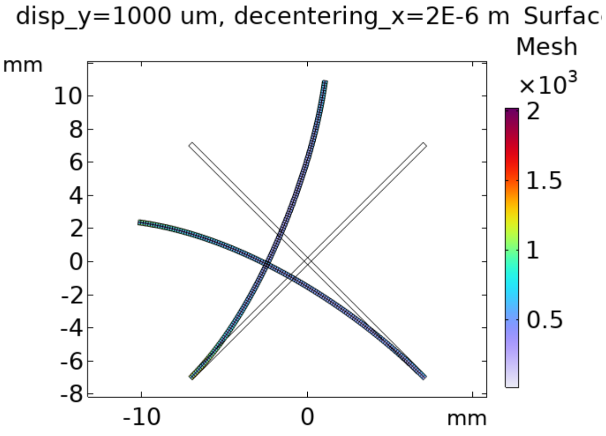




E	210E9	Pa
b	4.00E-03	m
h	3.00E-04	m
L	2.00E-02	m





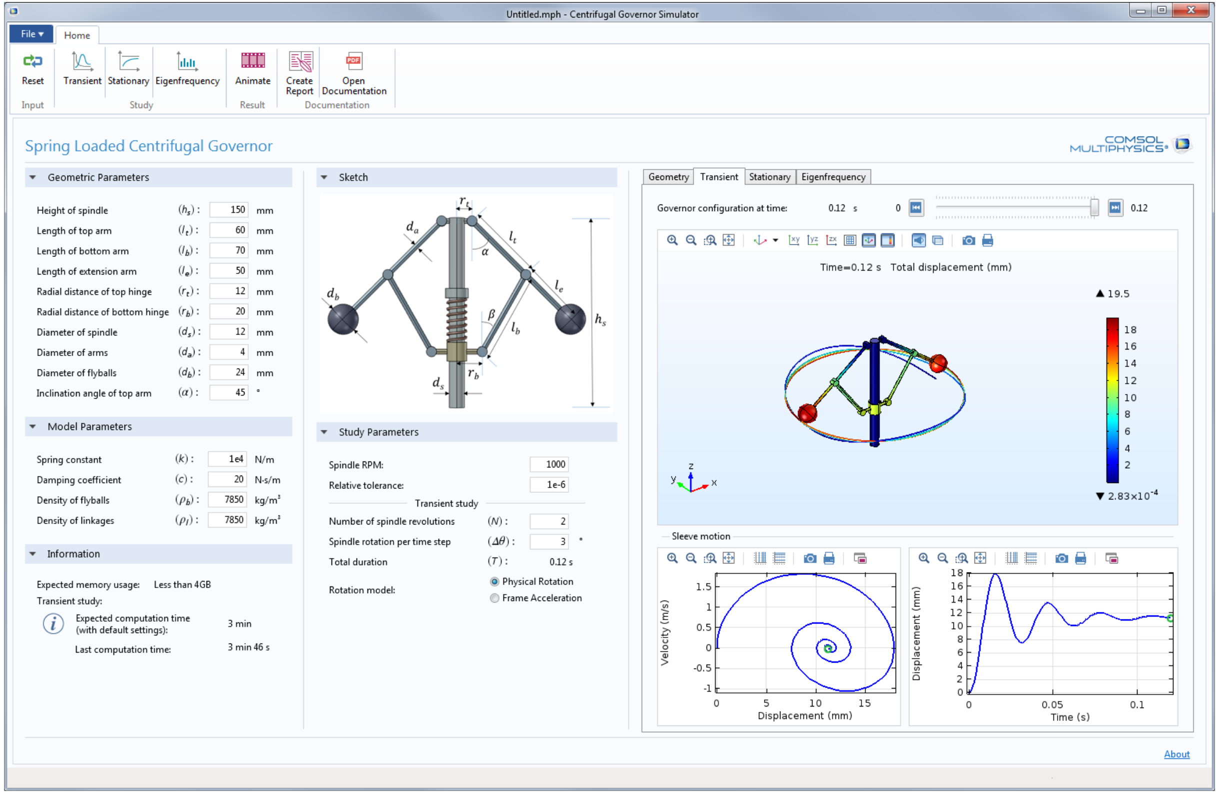


FEM types of analysis : multibody dynamic analysis

- Multibody Dynamic Module provides an advanced set of tools for designing and optimizing 2D and 3D multibody systems using finite element analysis (FEA). The module has the ability to **simulate mixed systems of flexible and rigid bodies** to find the critical components in a system
- The following **joint types** are available:
  - Prismatic, hinge, cylindrical, screw, planar, ball, slot, reduced slot, fixed, distance, universal
- **Types of analyses**:
  - Static, time dependent, eigenfrequency, frequency domain, mode superposition, random vibration

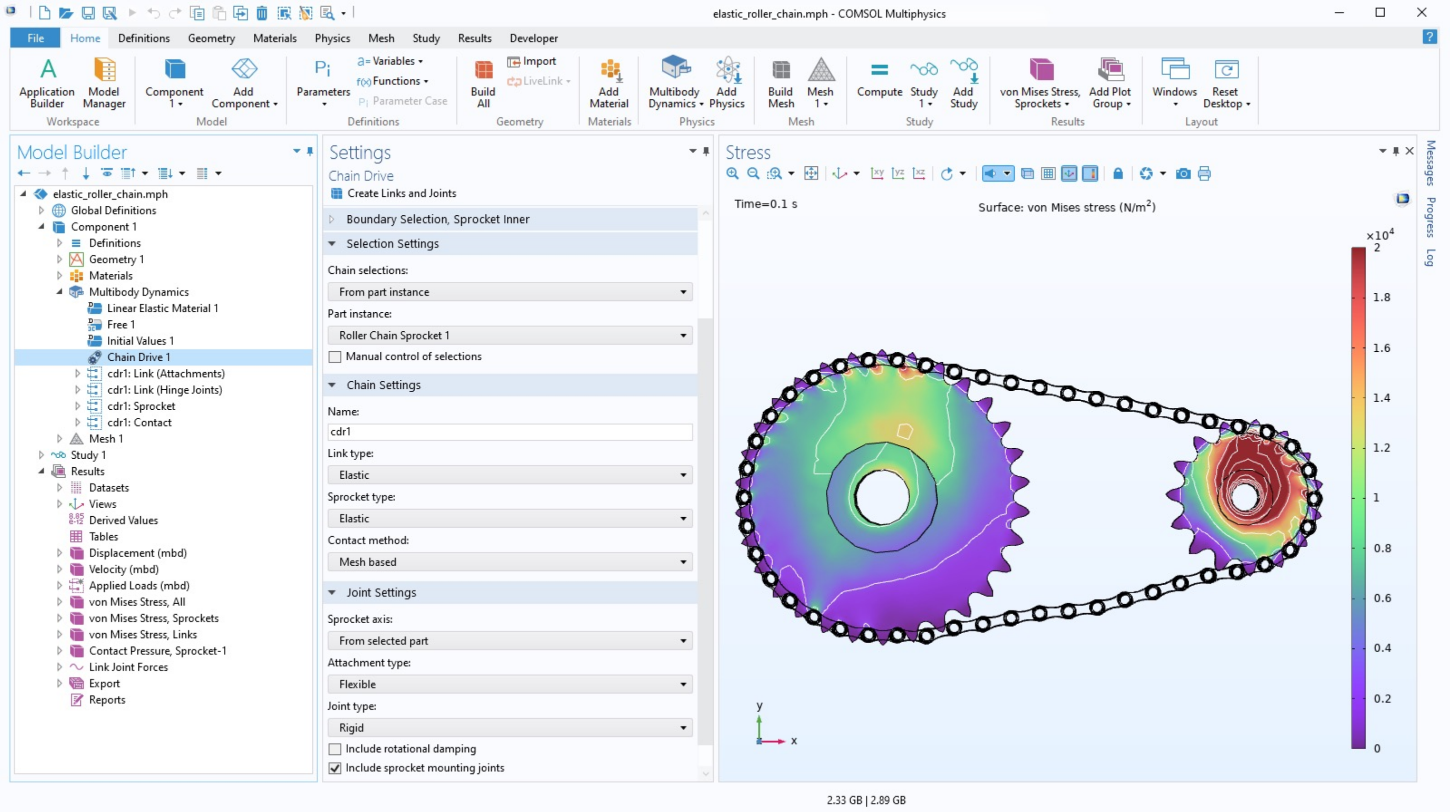
FEM types of analysis : multibody dynamic analysis

- User interface of the "**Simulation of a Centrifugal Governor**" app – the results of the **transient simulation** of a centrifugal governor that has reached the steady state are shown.



FEM types of analysis : multibody dynamic analysis

- An elastic **chain sprocket assembly** consisting of a **roller chain wrapped around two sprockets in 3D**. The Chain Drive feature is used to quickly set up a chain drive system. Results show the stress distribution in the sprockets.

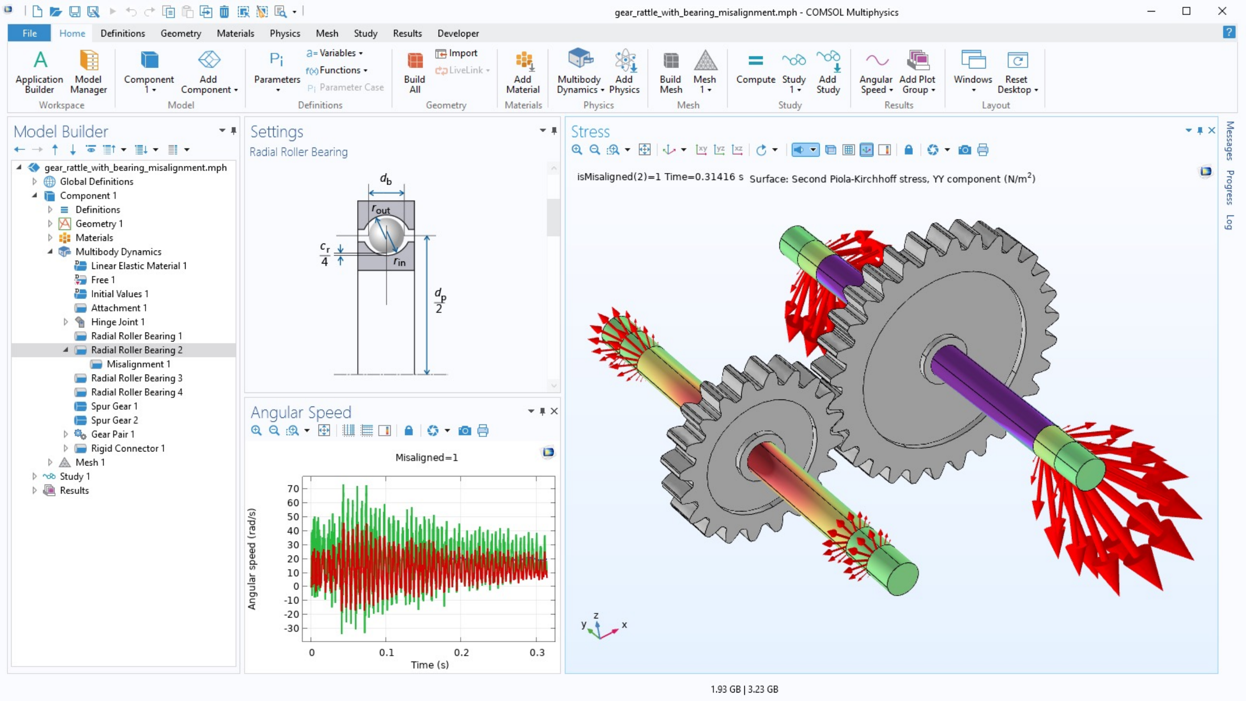


comsol.com

FEM types of analysis : multibody dynamic analysis

The **Multibody Dynamics Module**, together with the **Rotordynamics Module**, provides the following different types of **predefined radial roller bearings in 3D**:

- Deep groove ball
- Angular contact ball
- Self-aligning ball
- Spherical roller
- Cylindrical roller
- Tapered roller

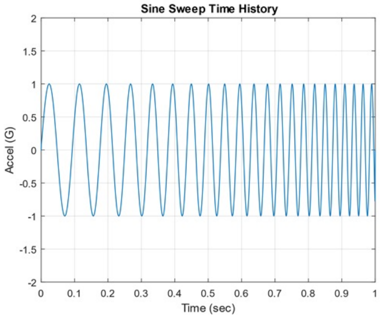


comsol.com

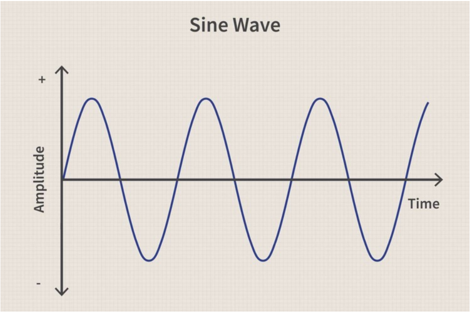
FEM types of analysis: environmental vibration analysis

Types of analyses:

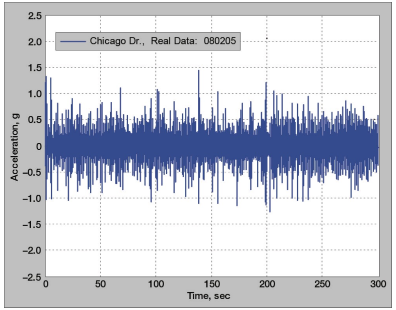
- (A) Sine: low to high power
- (B) Sine sweep: low power to check the structure integrity
- **(C) Random vibrations:** low to high power
  - **Miles' formula is analytical approximation**
- (D) Shocks



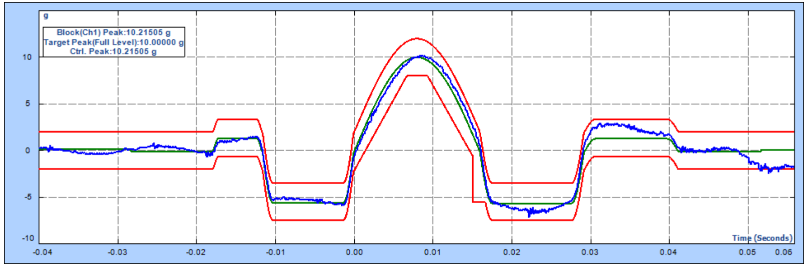
B



A



C

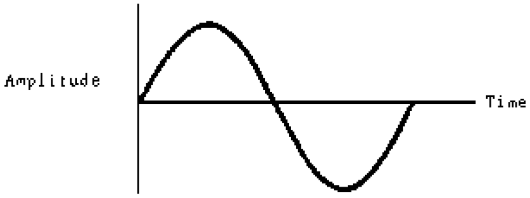


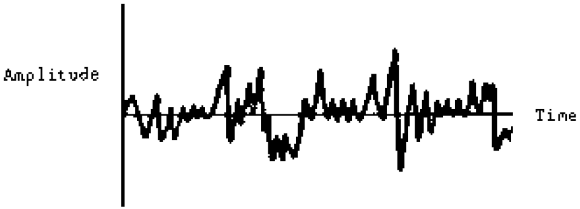
D

FEM types of analysis: environmental vibration analysis

A random vibration:

- is assumed to be a stationary process.
- is one whose absolute value is not predictable at any point in time there is no well-defined periodicity (as opposed to sinusoidal vibration).
- may excite numerous frequencies at the same time → structural resonances of different components can be excited simultaneously, **the interaction of which could be vastly different from sinusoidal vibration**, wherein each resonance would be excited separately.





Amplitude-Time History of (left) Sinusoidal Vibration and (right) Random Vibration following a Normal distribution.

Controls, B. and Hopkinton, M.A., *Random Vibration-An Overview*

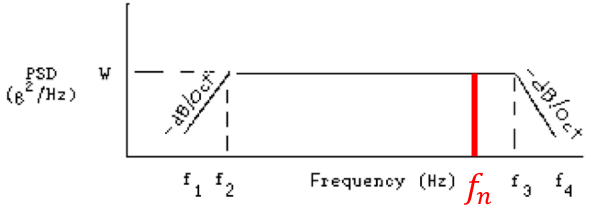
FEM types of analysis: environmental vibration analysis

The **structure response at a particular frequency** may be of prior concern. **The Miles formula** is used to find the structure acceleration G_{rms}:

$$G_{rms} = \sqrt{\frac{\pi}{2} \cdot f_n \cdot Q \cdot ASD_{f_n}}$$

Where:

G_{rms} is the structure acceleration (1g = 9,81 $\mathrm{m/s^2}$)
f_n is the considered natural frequency of the structure
Q is amplification factor
ASD_f is the Acceleration Spectral Density evaluated at frequency f_n



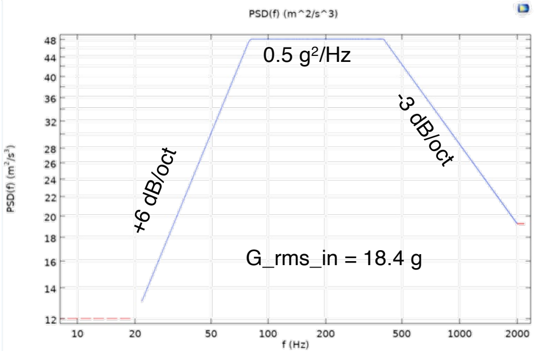
The structure acceleration (G_{rms}) is then used to find the 3-sigma stress through inertia loading computation:

$$F_{3\sigma} = 3 \cdot M \cdot G_{rms}$$

FEM types of analysis: environmental vibration analysis

Stress calculation for the lateral mode of the flexleg of MVIS planar joint through the Miles formula:

$$G_{RMS_{out}} = \sqrt{\frac{\pi}{2} f_n Q \cdot ASD_f} = 105 \text{ g}$$

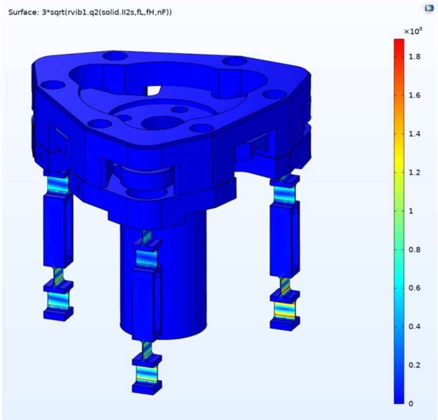


With

- $f_n = 1204 \text{ Hz}$
- $Q = 150$
- $ASD_f = 0.039 \text{ g}$

Then the 3-sigma stress is:

$$\sigma_f = \frac{3F_{3\sigma}L}{2bh^2} = 194 \text{ MPa}$$



With

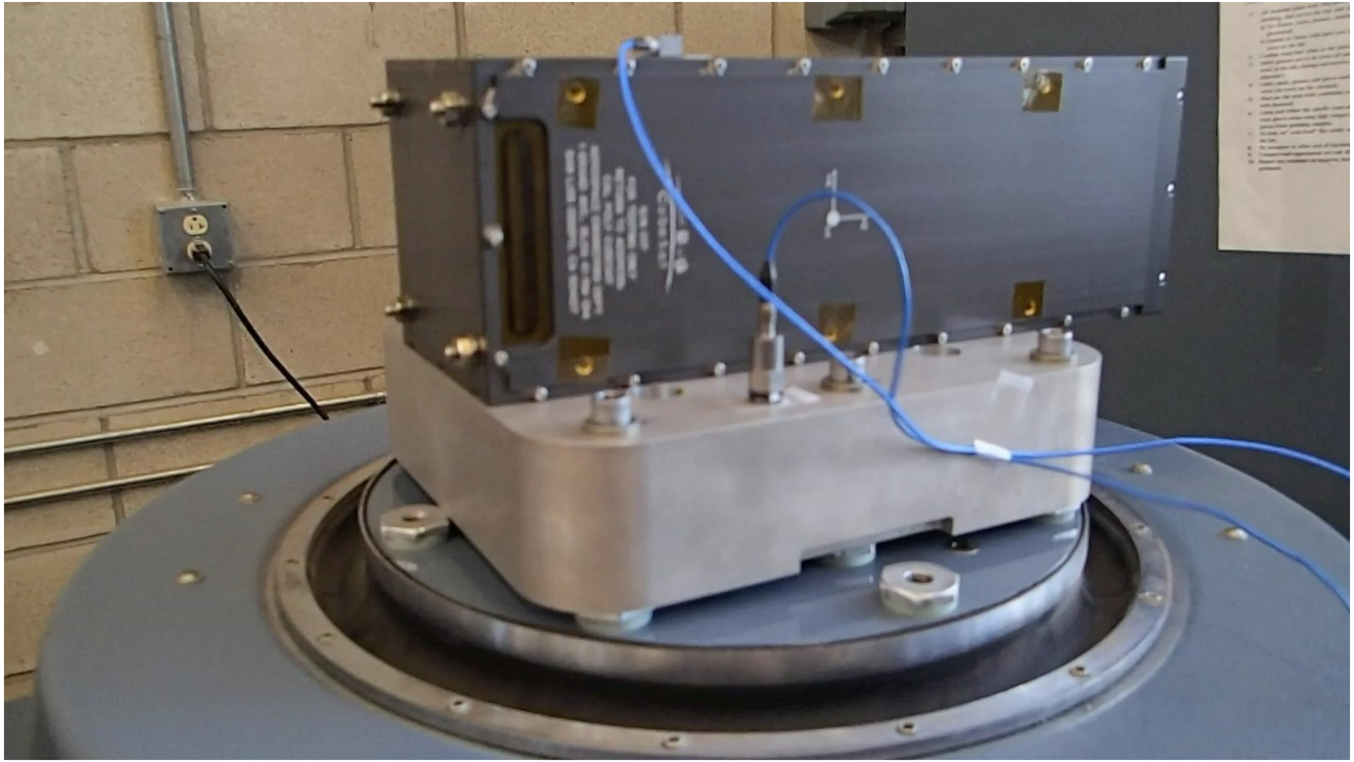
- $F_{3\sigma} = 3G_{RMS}M_{arm} = 6.5 \text{ N}$
- $L = 4.33 \text{ mm}$ (including a third of the radius)
- $b = 6 \text{ mm}$
- $h = 190 \text{ µm}$
- $M_{arm} = 7 \, g$

NB: $1 \frac{g^2}{Hz} = (9.81)^2 \frac{m^2}{s^3}$

→ EXERCISE 9

FEM types of analysis: environmental vibration analysis

- ***LightSail-A undergoes random vibration testing** at Cal Poly San Luis Obispo on Oct. 24, 2014. **The test simulates the stresses of an Atlas V launch**. Video credit: Riki Munakata*



FEM types of analysis: environmental vibration analysis

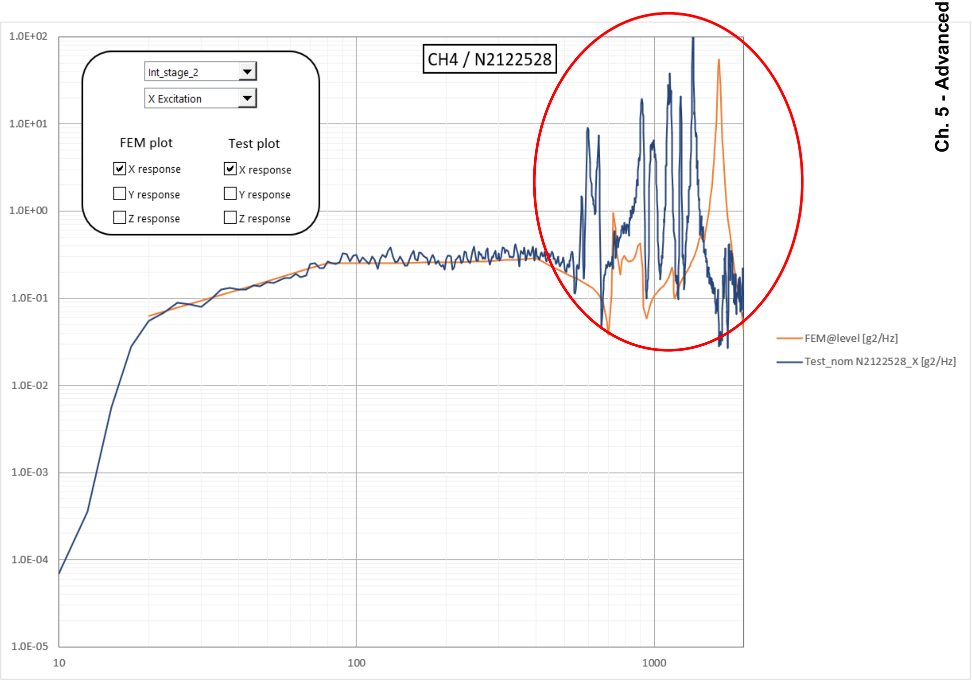
Microvibration platform isolation environmental tests: succeed

FEM types of analysis: environmental vibration analysis

Random vibrations FEA versus measurement

Issues for **compliant mechanism vibration**:

- Single resonant frequency in the FEM can give to multiple peaks in the tests
- Interaction with **end stops**
  - multiple shocks,
  - amplification changes
  - frequency shift
- **Prediction and correlation** is difficult
- At first approximation compare overall G_{rms}



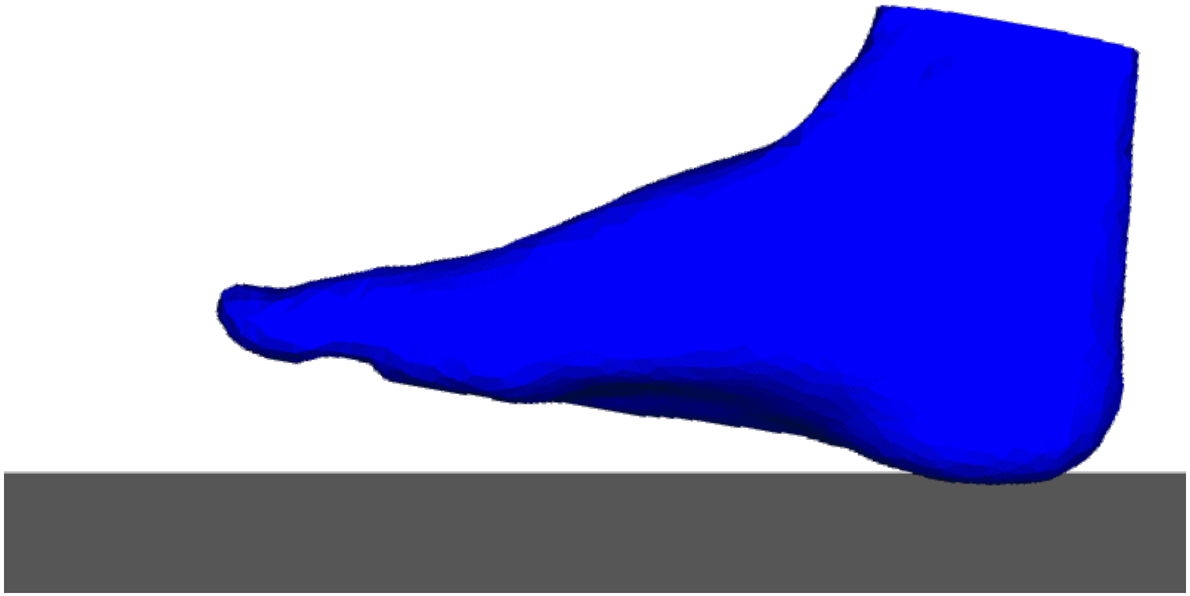
What the FEA?

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?

EPFL

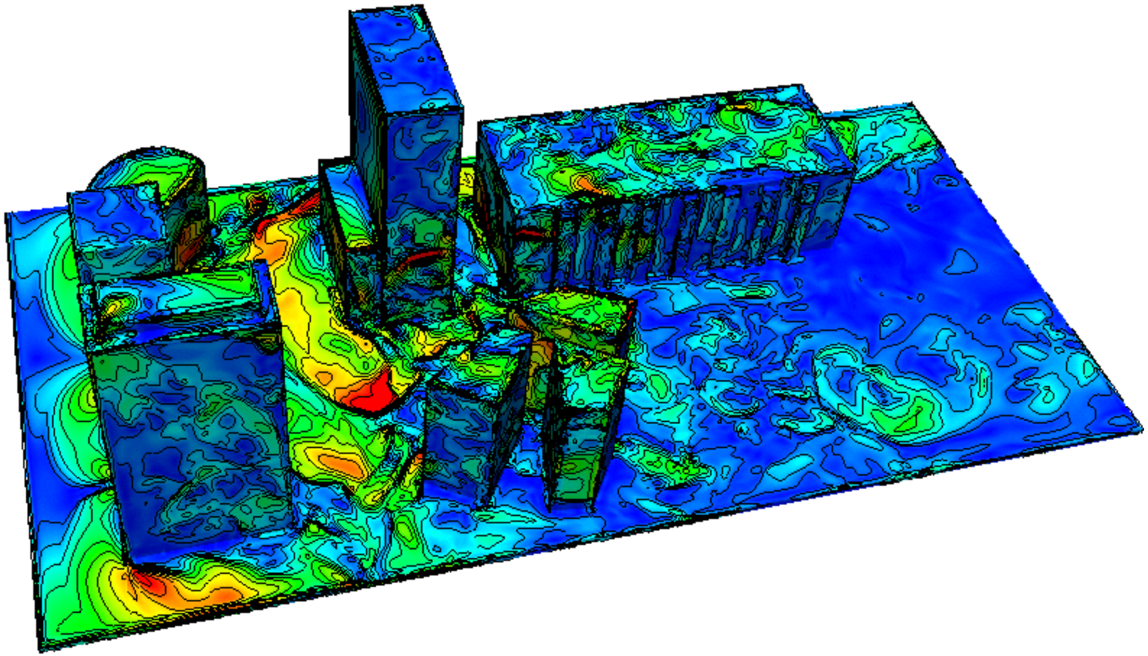
What the FEA?



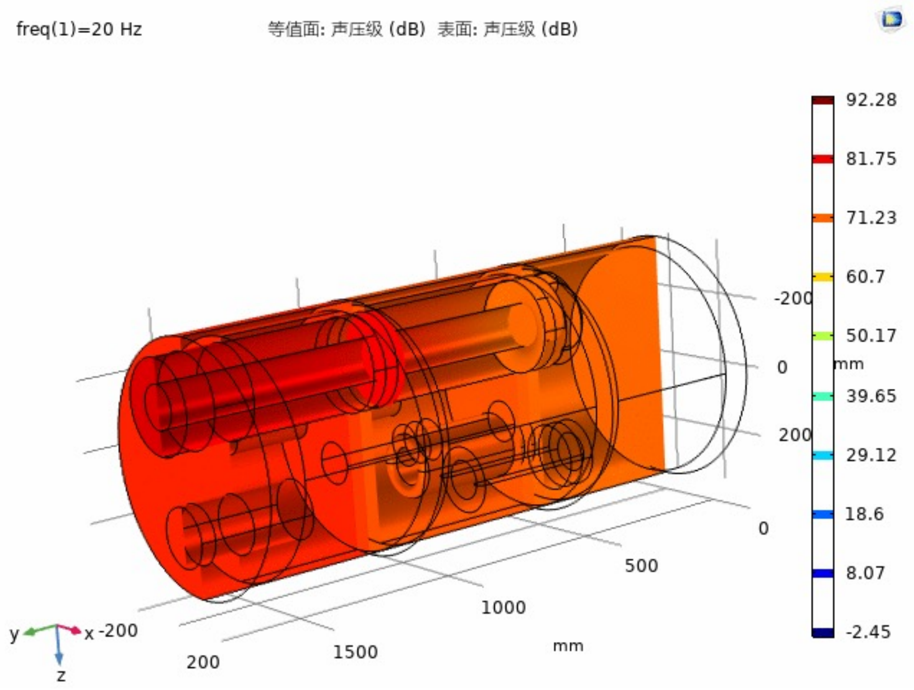
What the FEA?

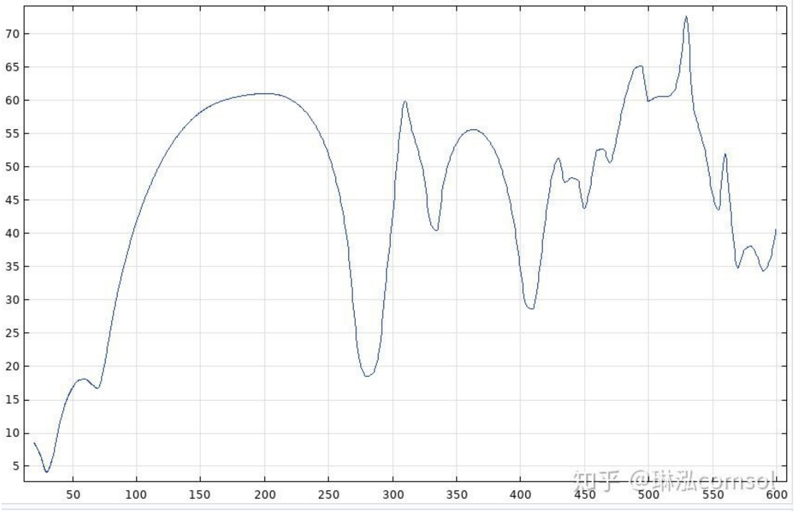

MSC/PATRAN Version 9.0 18-Sep-00 15:37:18
Deform: Results #1, Cycle 0, Time 0.000000, Displacement, (NON-LAYERED)
Z
X
Y
default_Deformation
Max 0. @Nd 1
Frame: 1

What the FEA?



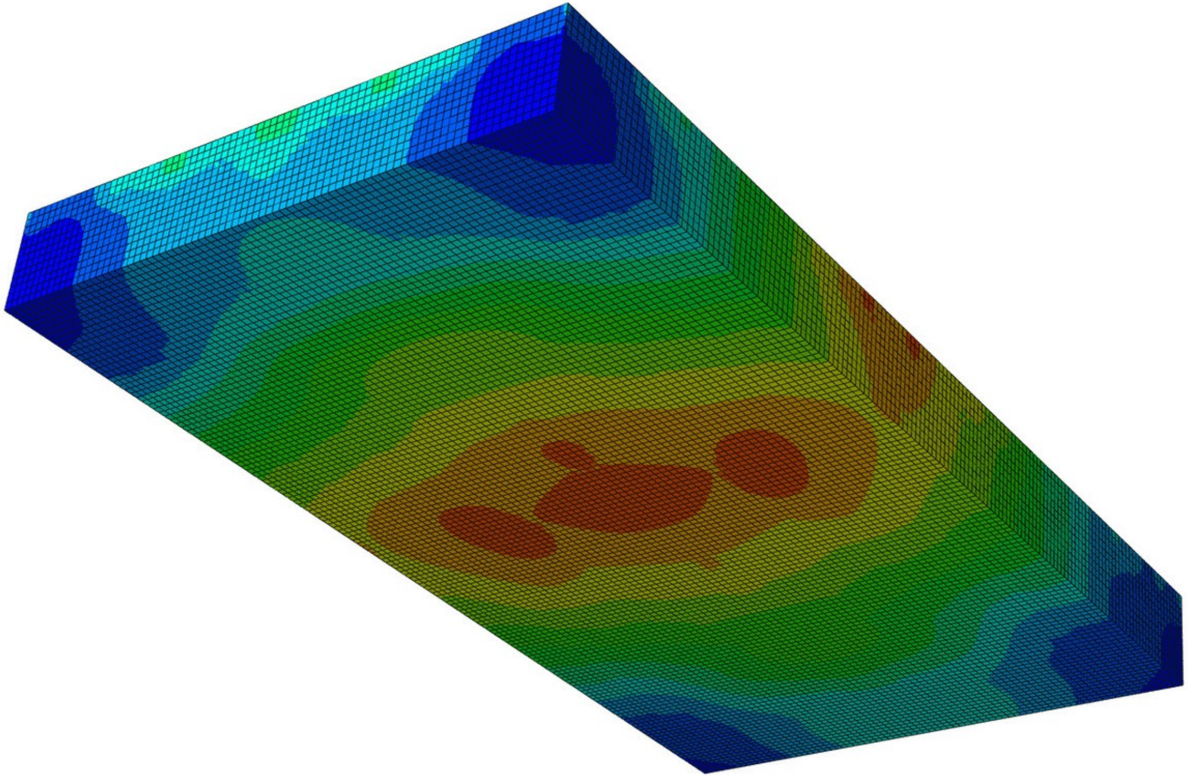
What the FEA?





doc.comsol.com

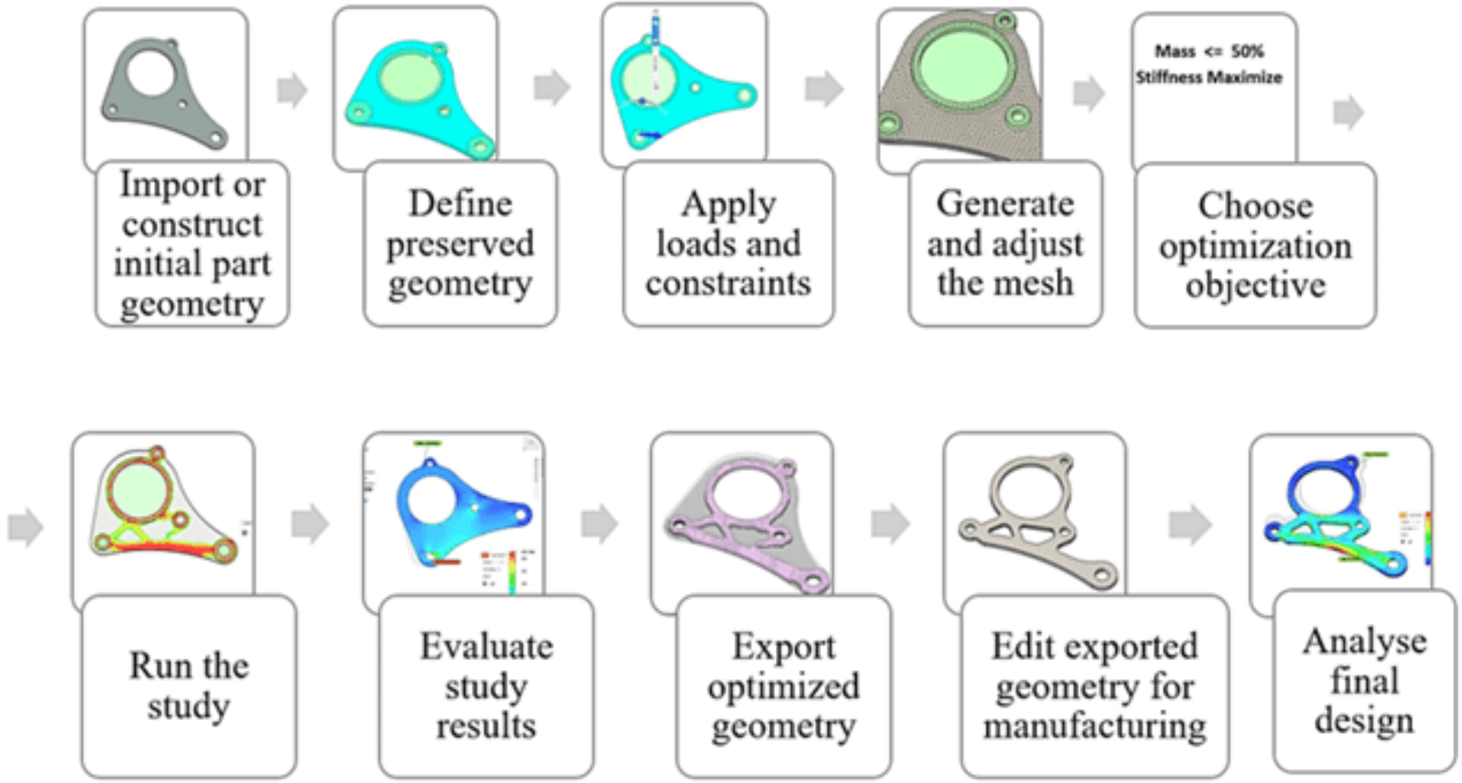
What the FEA?



Topology optimization

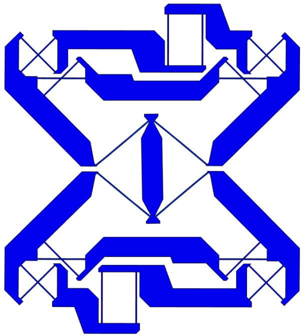
1. **Define the design problem**, including **objectives** (maximizing stiffness, minimizing weight, maximizing first eigenfrequency) and **constraints** (stress limits, manufacturing constraints).
2. Specify the **design space**, which represents the volume or region within which the material layout will be optimized.
3. **Discretize the design domain** into a finite element **mesh**, dividing it into smaller elements to facilitate mathematical analysis.
4. **Assign material** properties and **boundary conditions** to the mesh, such as **loads and supports**
5. Run **FEA**.
6. Define the **objective function**, which quantifies the performance to be optimized
7. Select an appropriate **optimization algorithm** to solve the topology optimization problem.
8. Iteratively update the **material distribution** within the design domain to **improve the objective function while satisfying constraints.**
9. **Post-process** the optimized design to refine the material layout, remove small features, and ensure manufacturability.
10. **Evaluate the final optimized design** using additional analyses, such as finite element analysis (fea), to validate its performance under different loading conditions.

Topology optimization

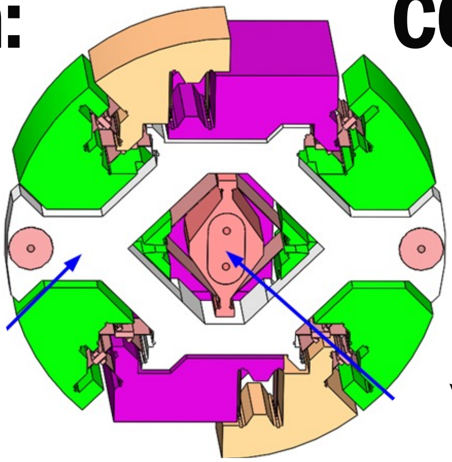


EPFL

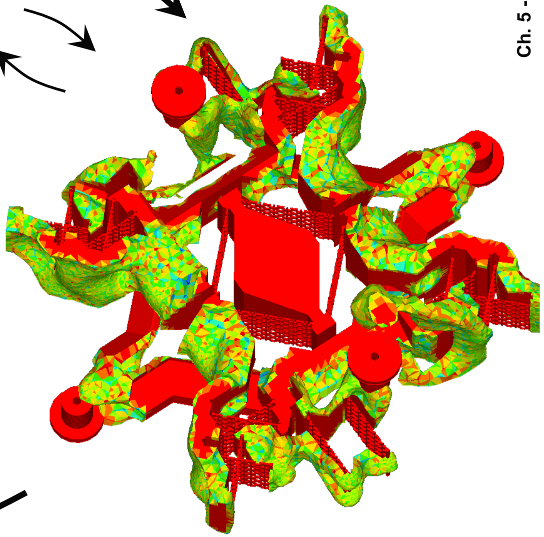
Topology optimization: COMAM example



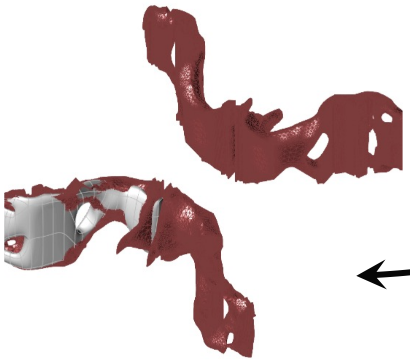
Preliminary design



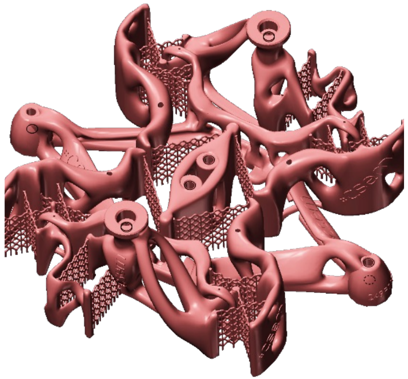
Design space and load cases definition



Optimized density design (iterative process)



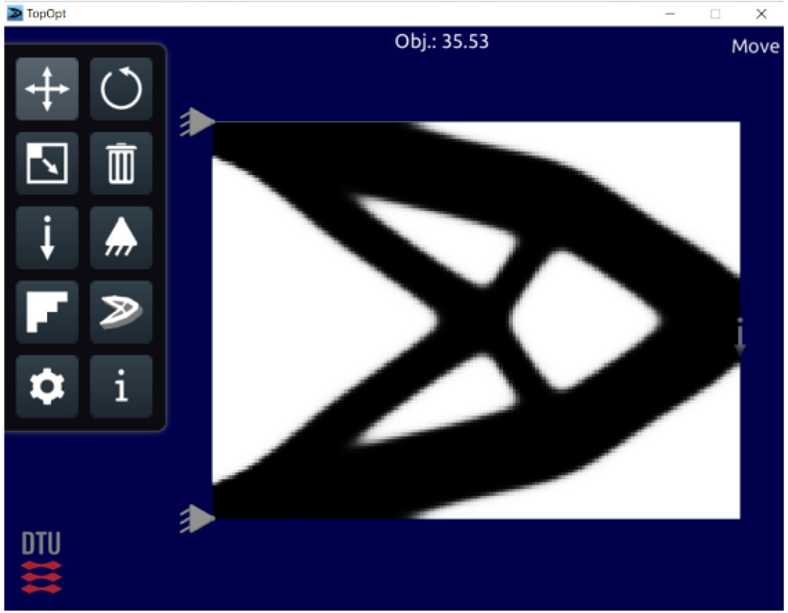
Smoothing of 3D surfaces



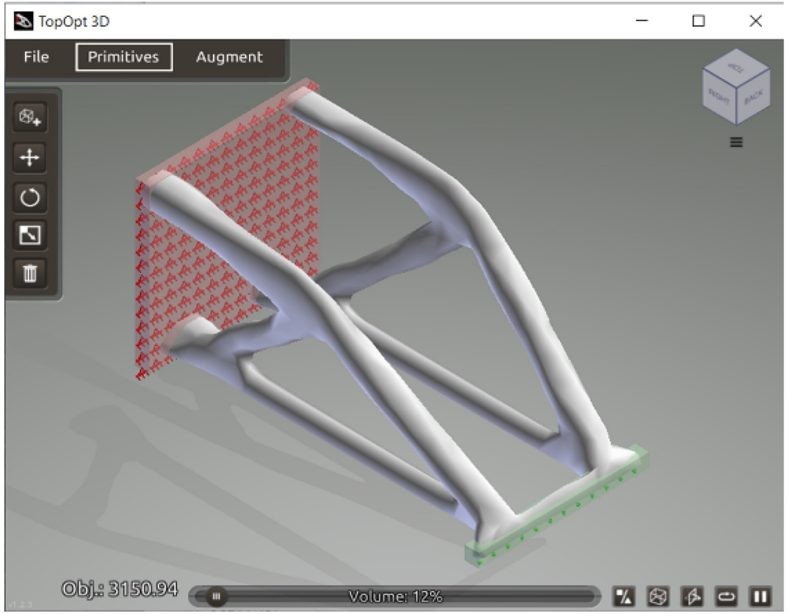
Final design with lattices and interfaces added

Topology optimization: DTU Interactive 2D/3D TopOpt App

link



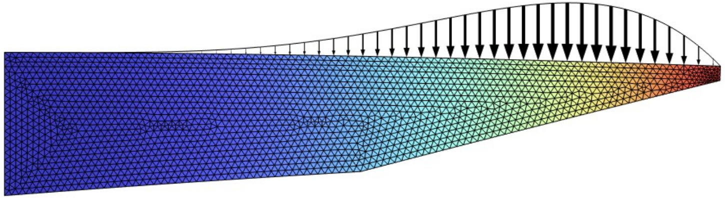
(a)



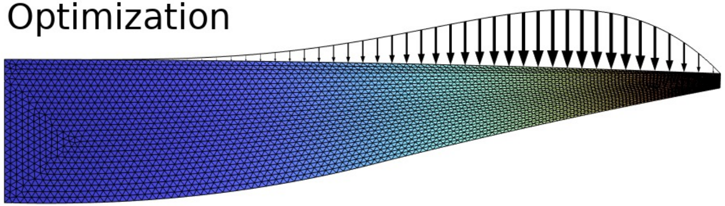
(b)

Topology optimization: alternatives

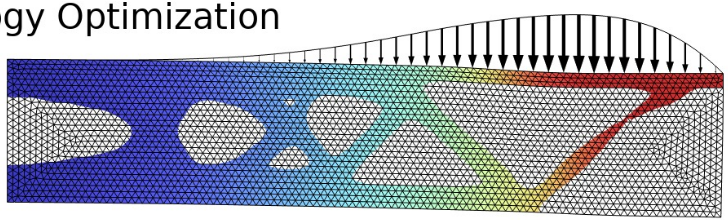
Parameter Optimization



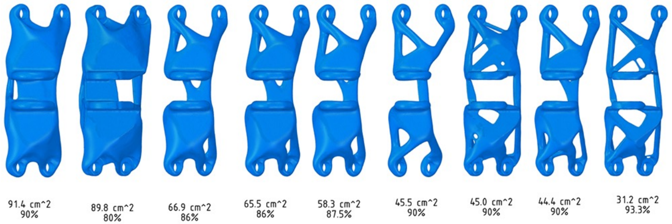
Shape Optimization



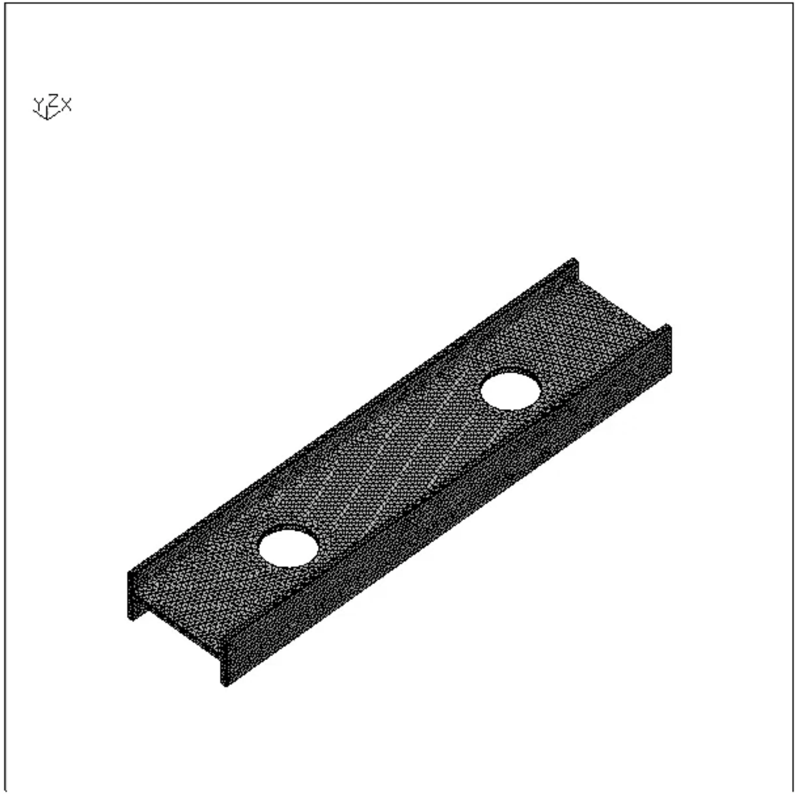
Topology Optimization



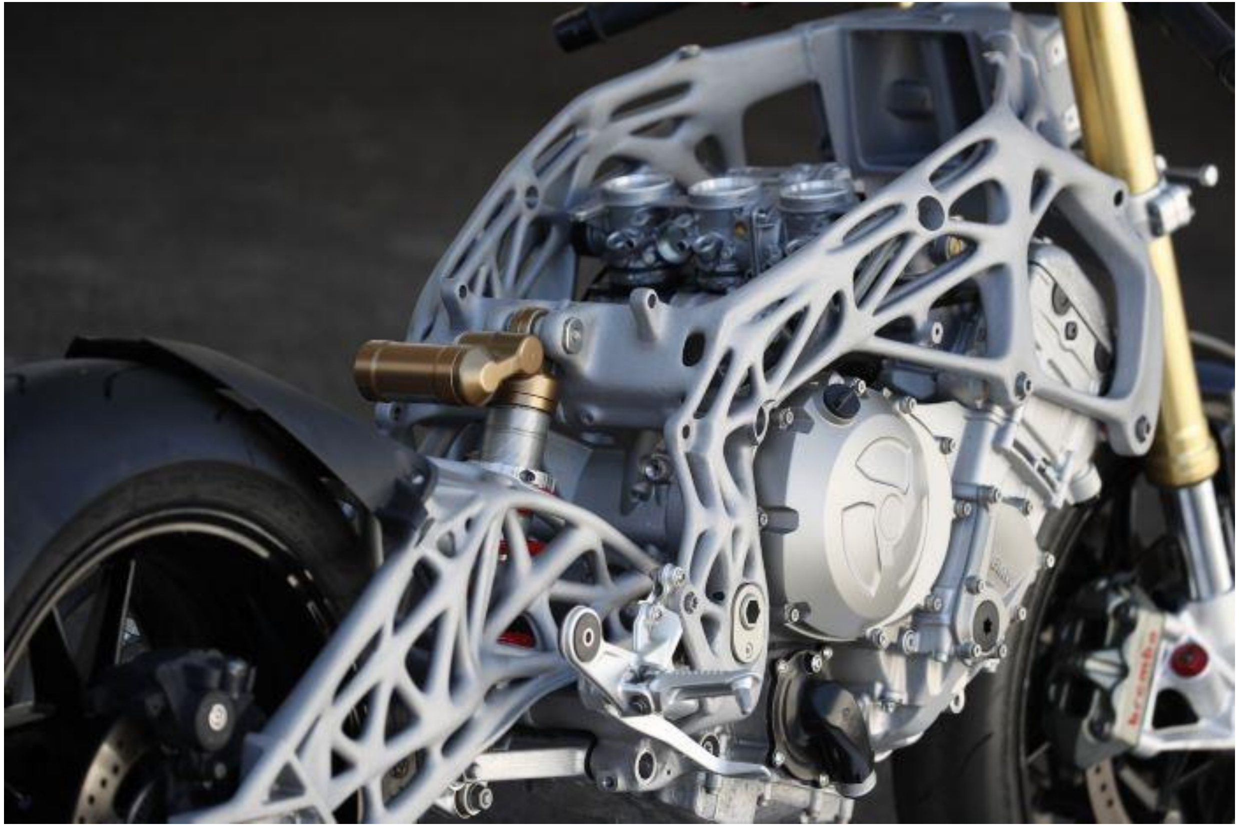
Generative Design



Topology optimization

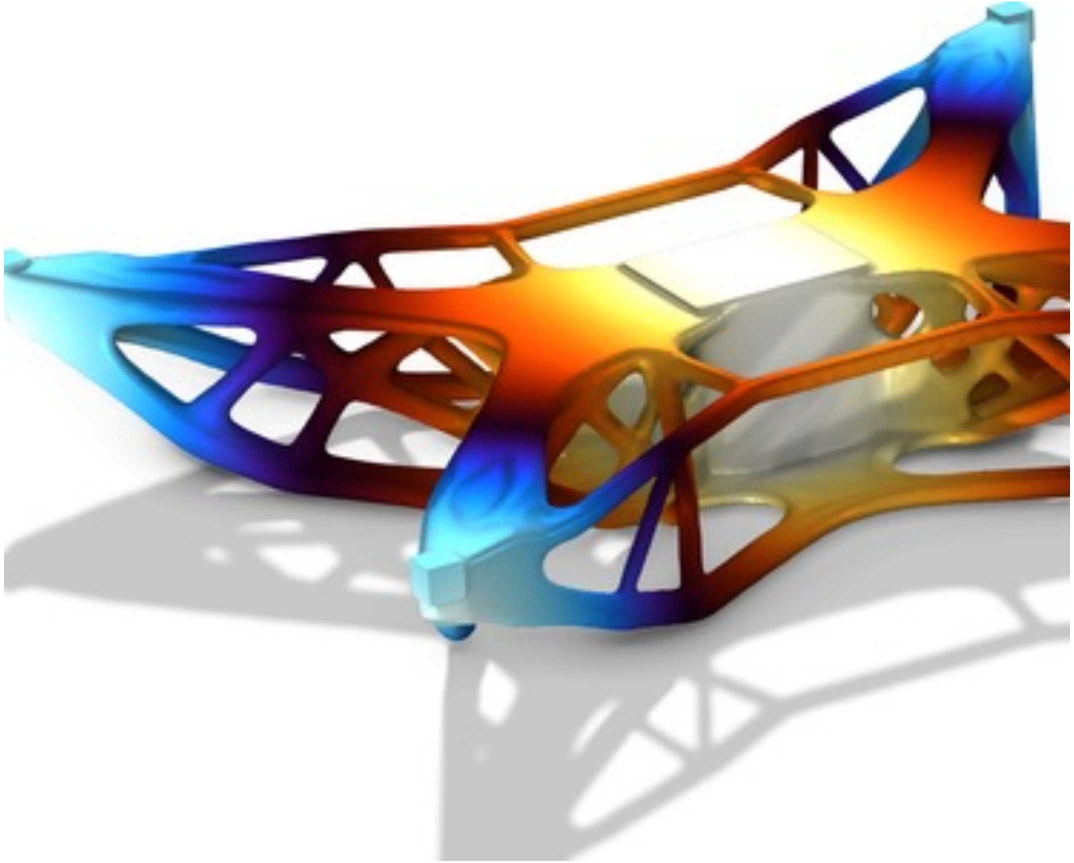


Topology optimization



Source - BMW

Topology optimization



Source - COMSOL

Topology optimization

Source - AMFG

Week 8 exercises and homework

Exercise on MOODLE:

- EXO8_lambda.pdf
- EXO8_lambda.m
- EXO8_lambda.mph

Homework:

- no homework

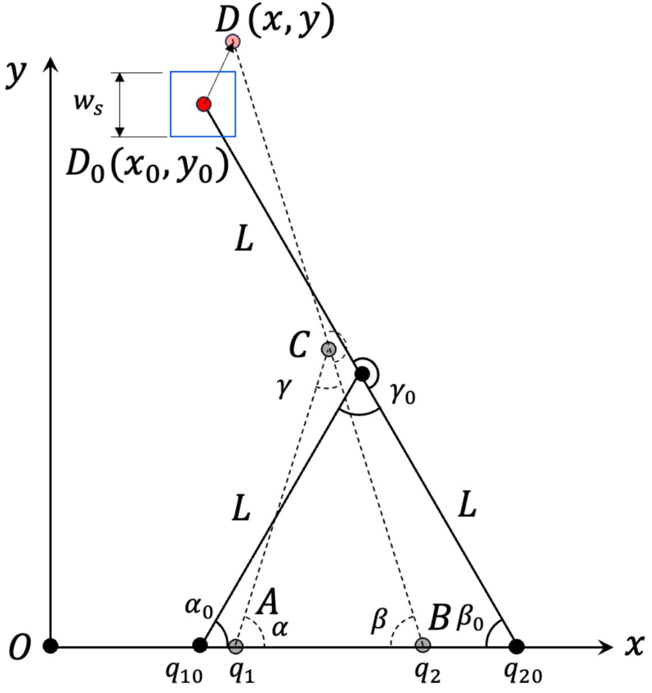


Figure 1: Schema of the parallel kinematic structure of a lambda planar robot