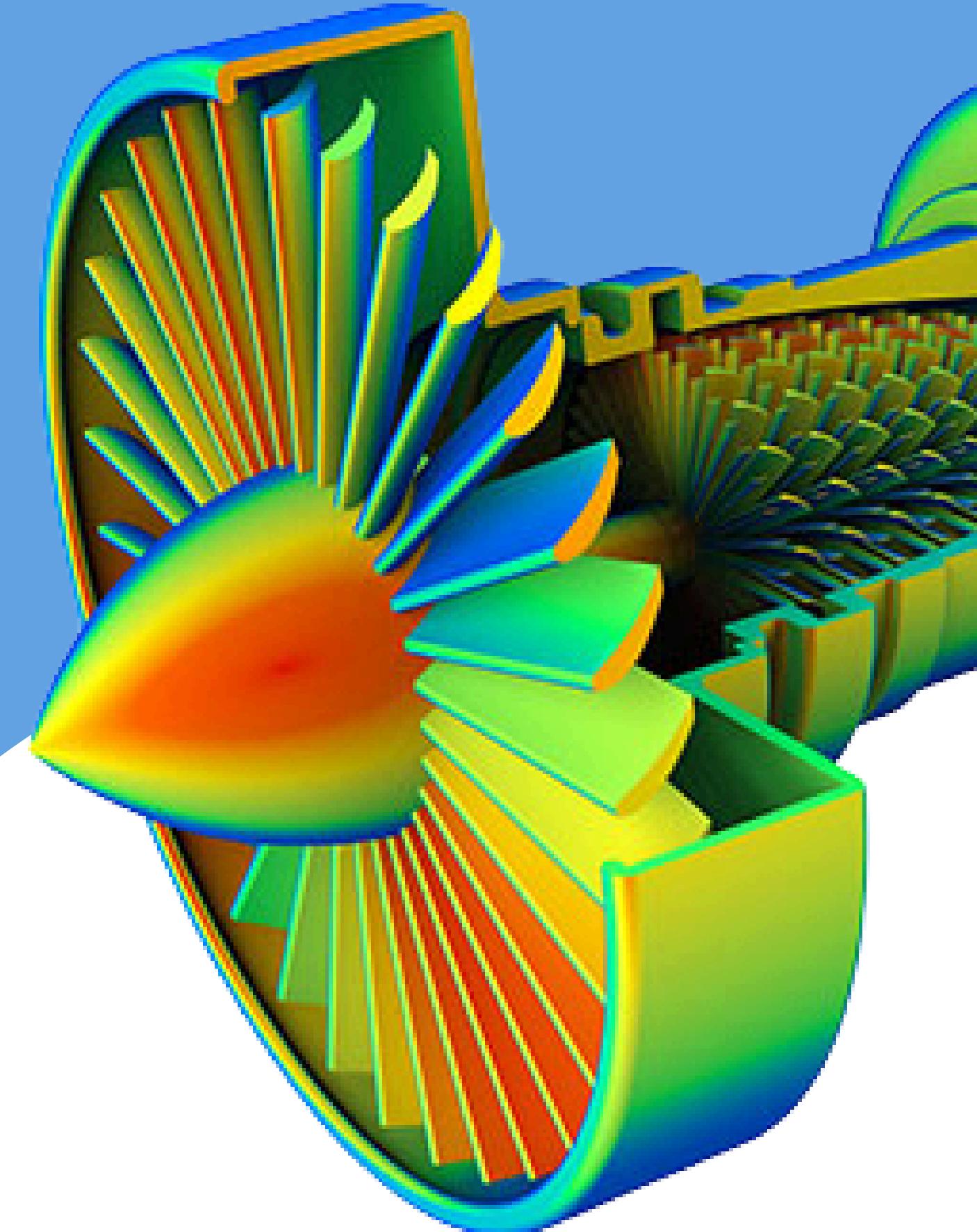


INTRODUCTION TO FINITE ELEMENT ANALYSIS (FEA)

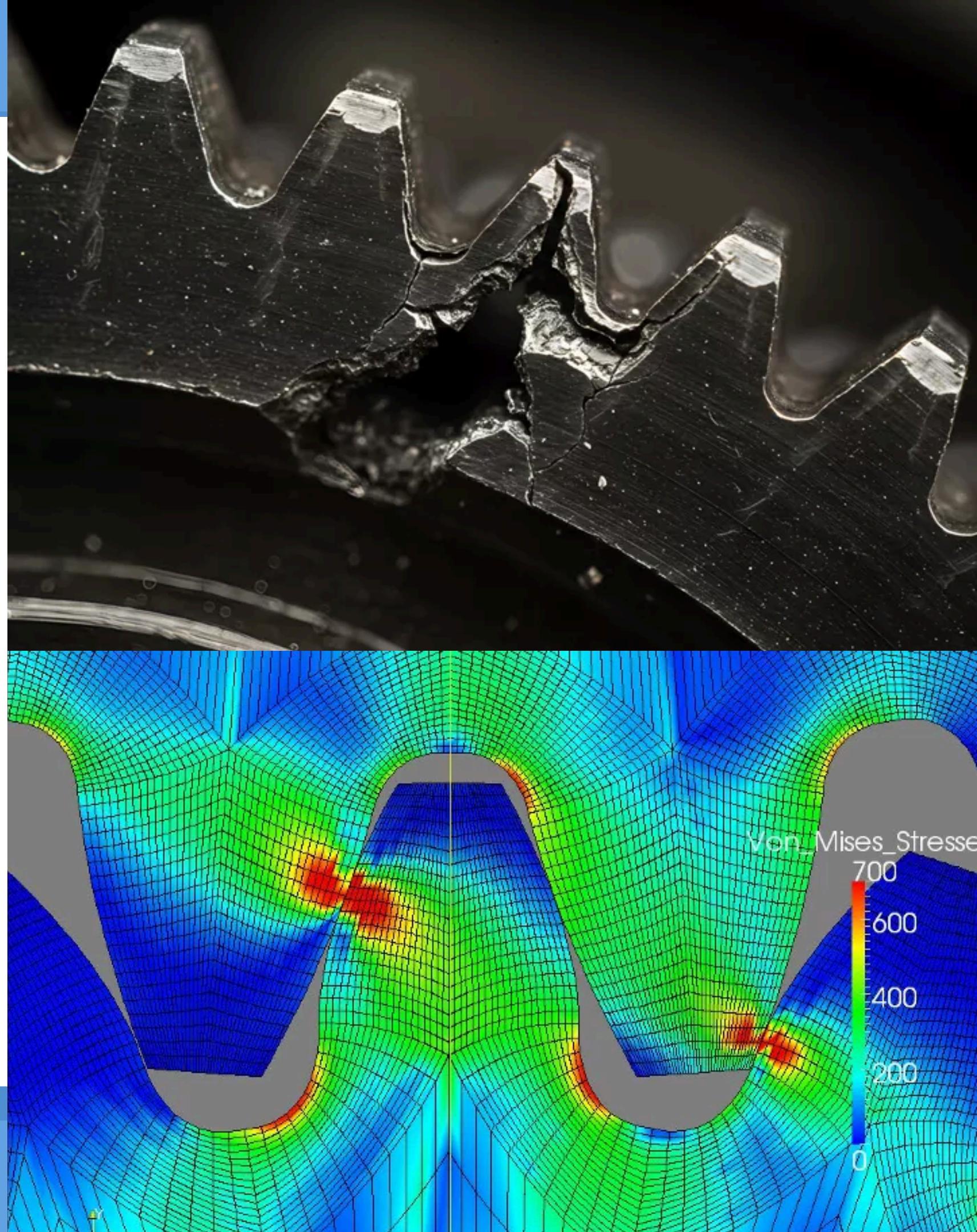
With applications in materials
characterization



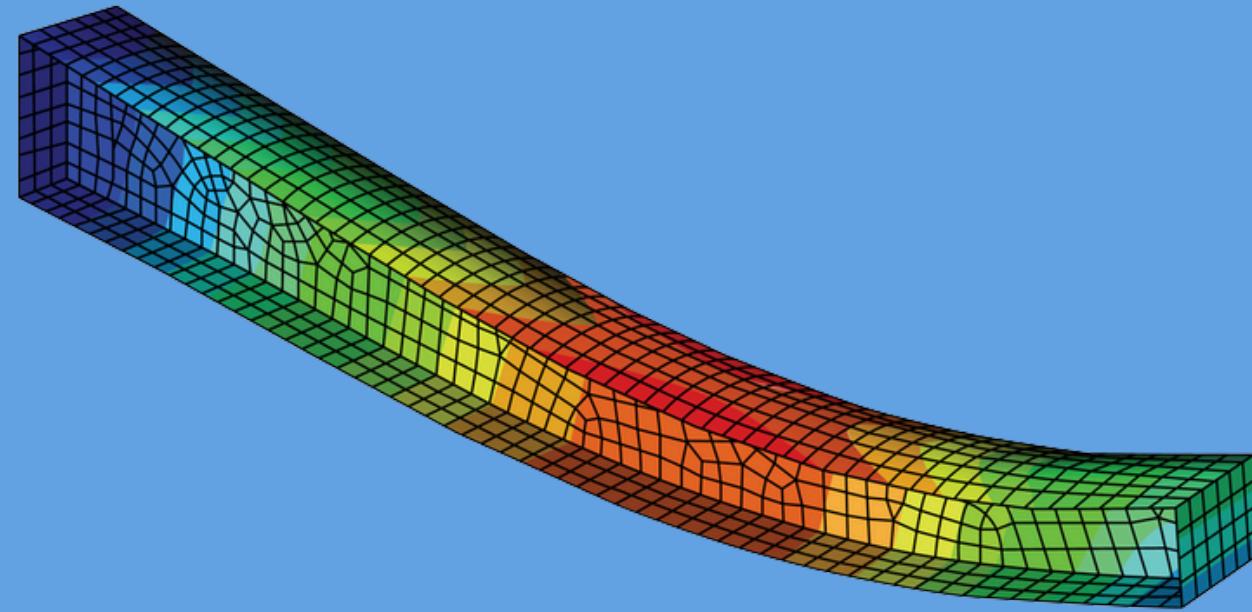
WHAT IS FEA?

Finite Element Analysis is a numerical method to predict how structures or materials behave under loads, heat, or other conditions.

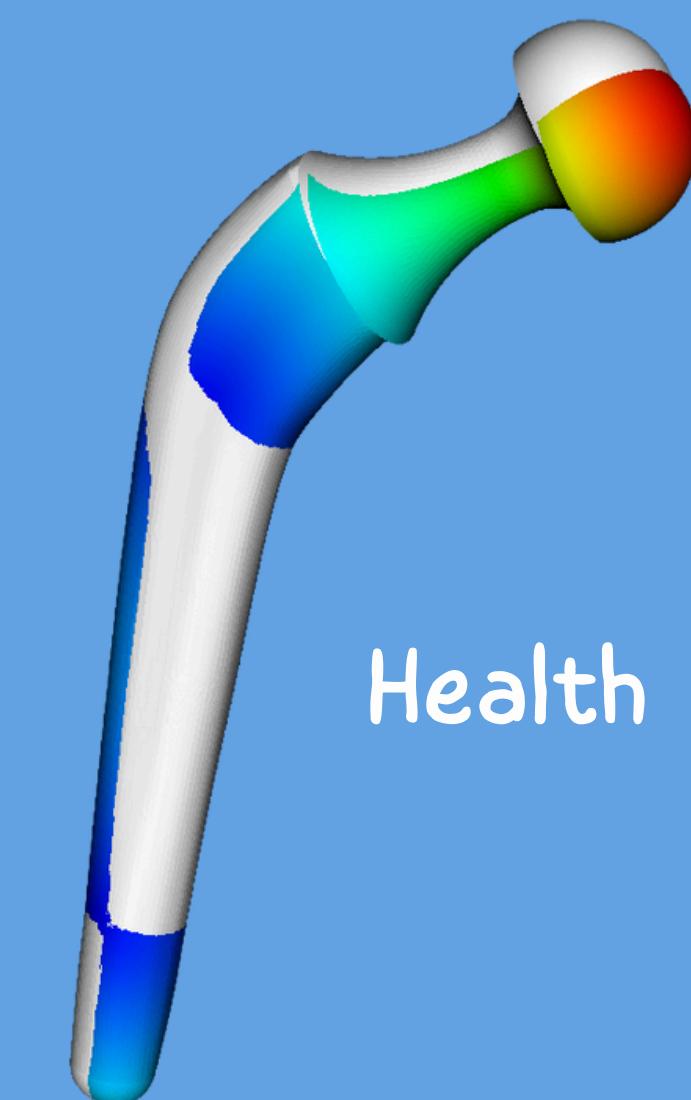
- Why it matters for engineers and scientists?



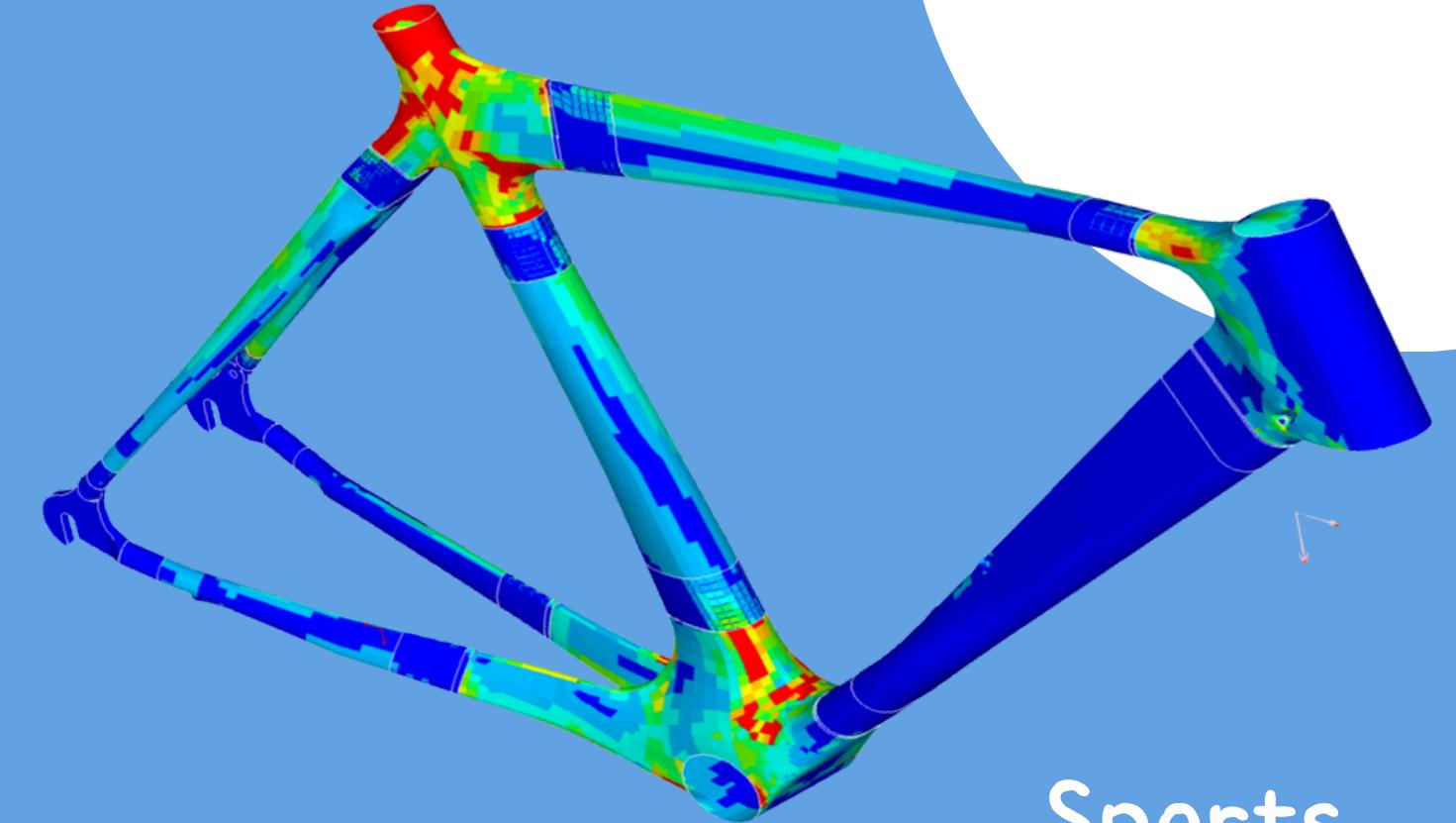
Construction



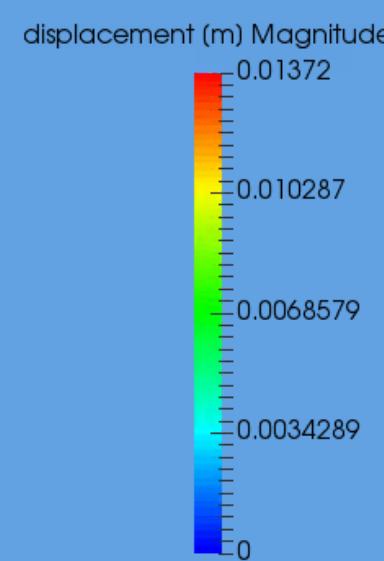
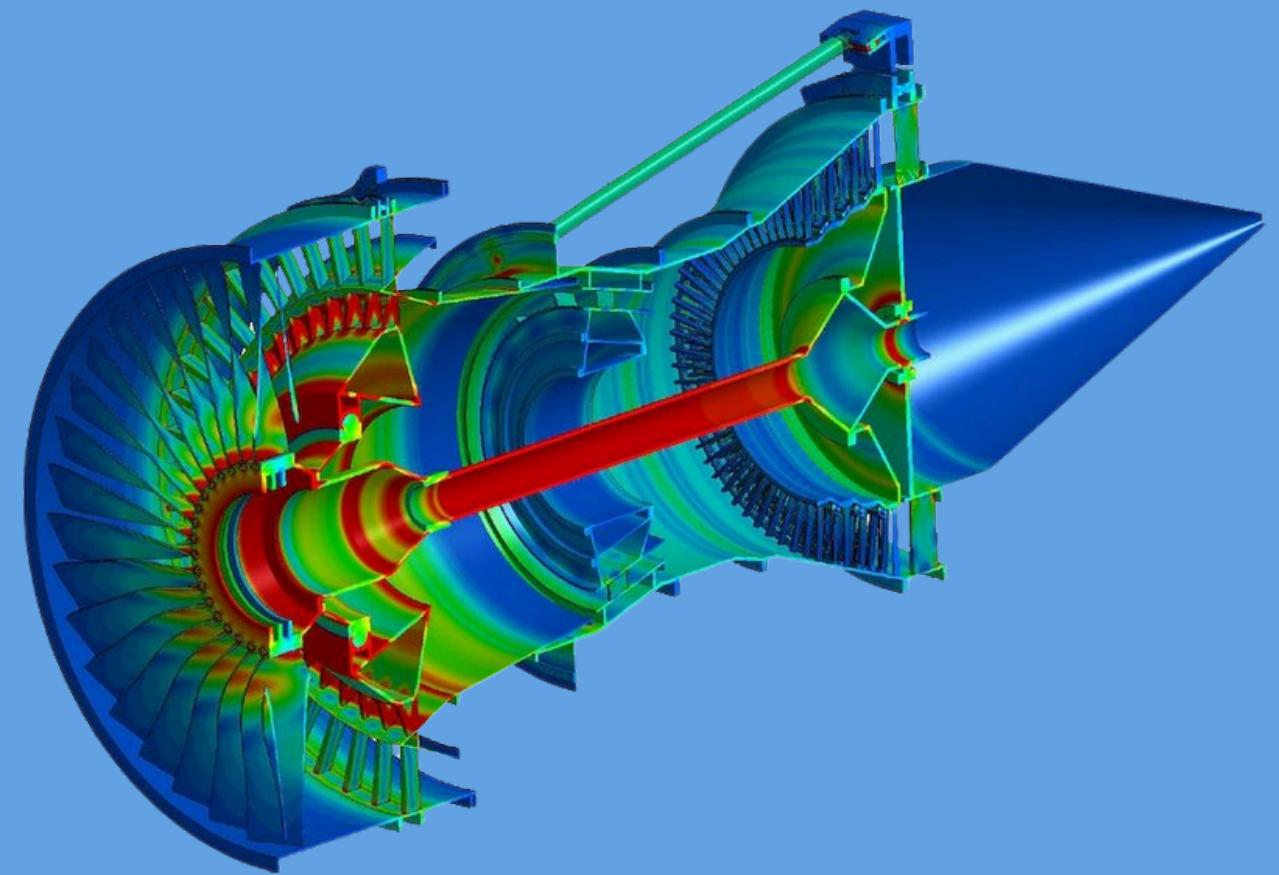
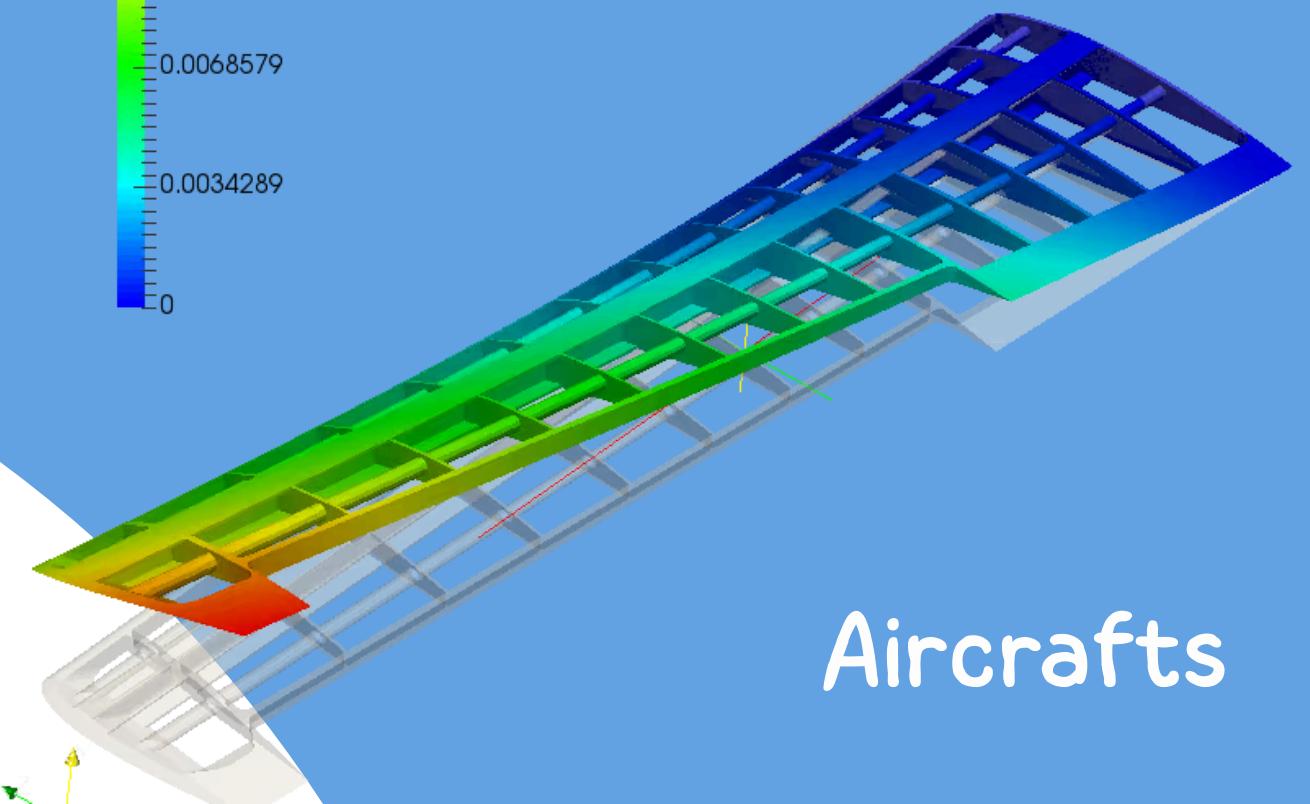
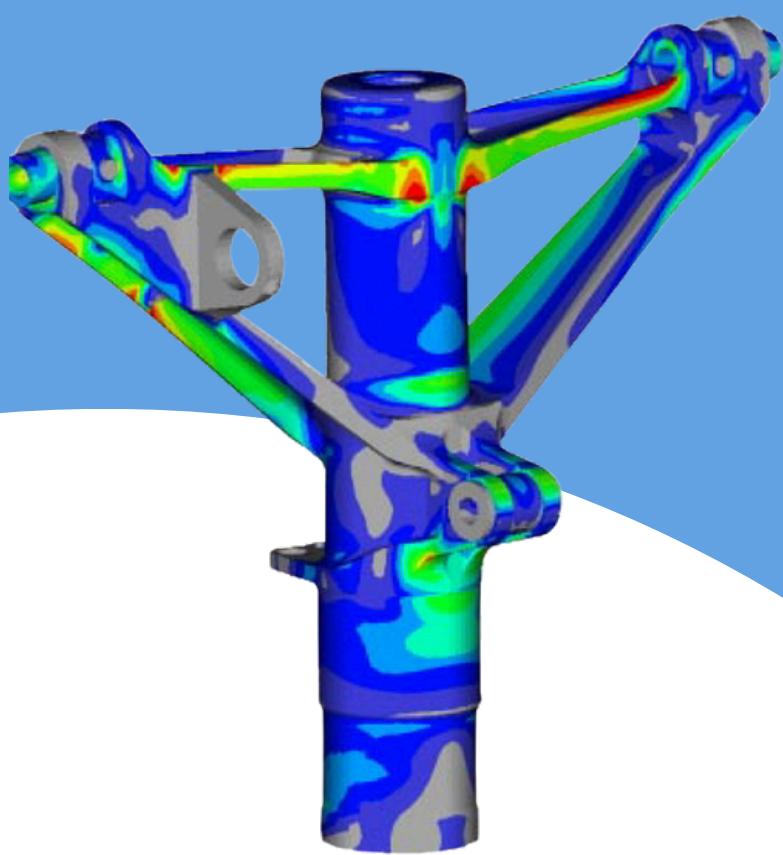
Health



Sports



Aircrafts

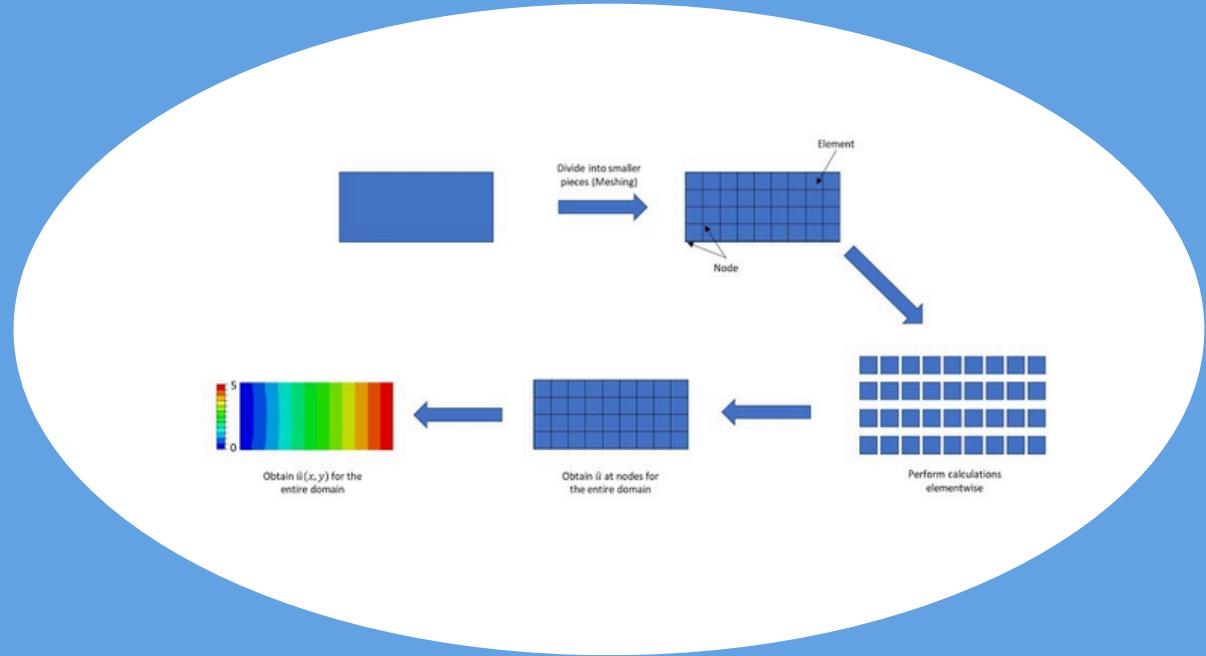


BASICS OF FEA

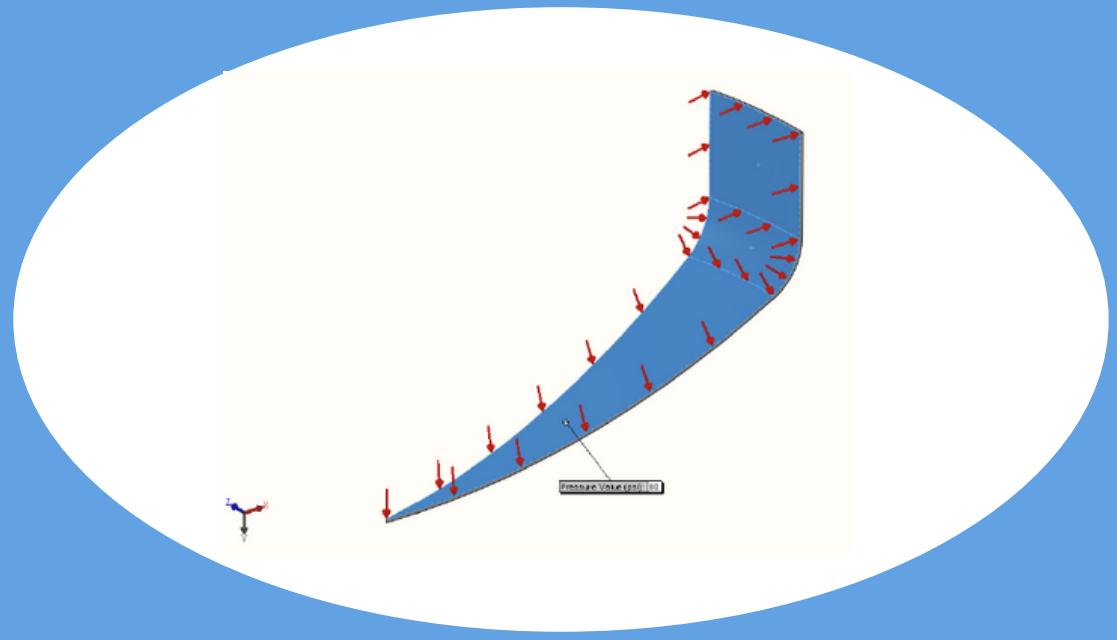


Process

How does it work?

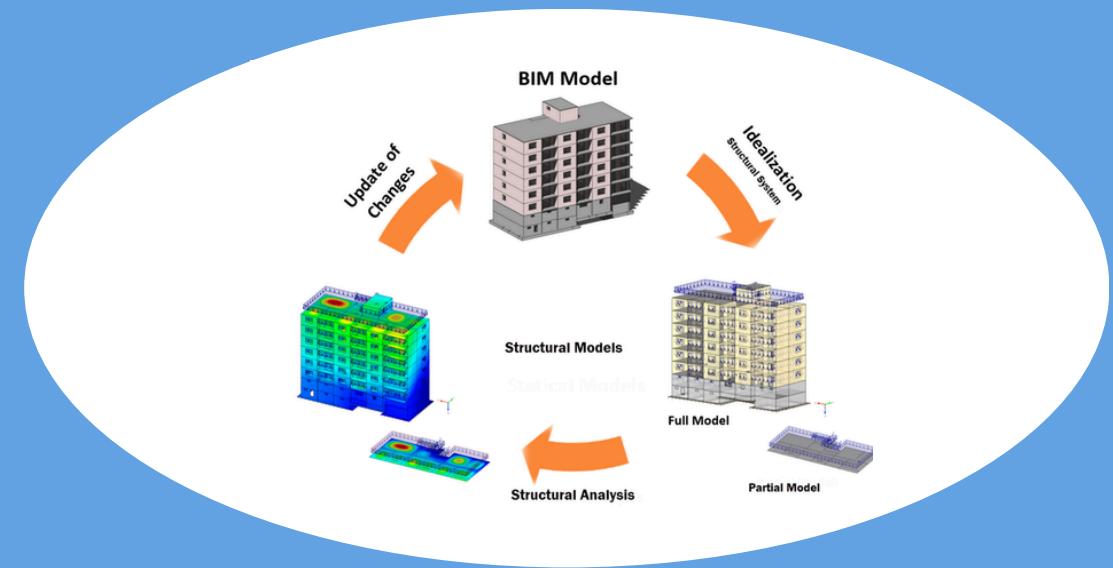


Think of it like breaking down a large, complicated structure into many small, simple pieces, which we call finite elements. These elements are connected at points called nodes, and together they form a mesh.



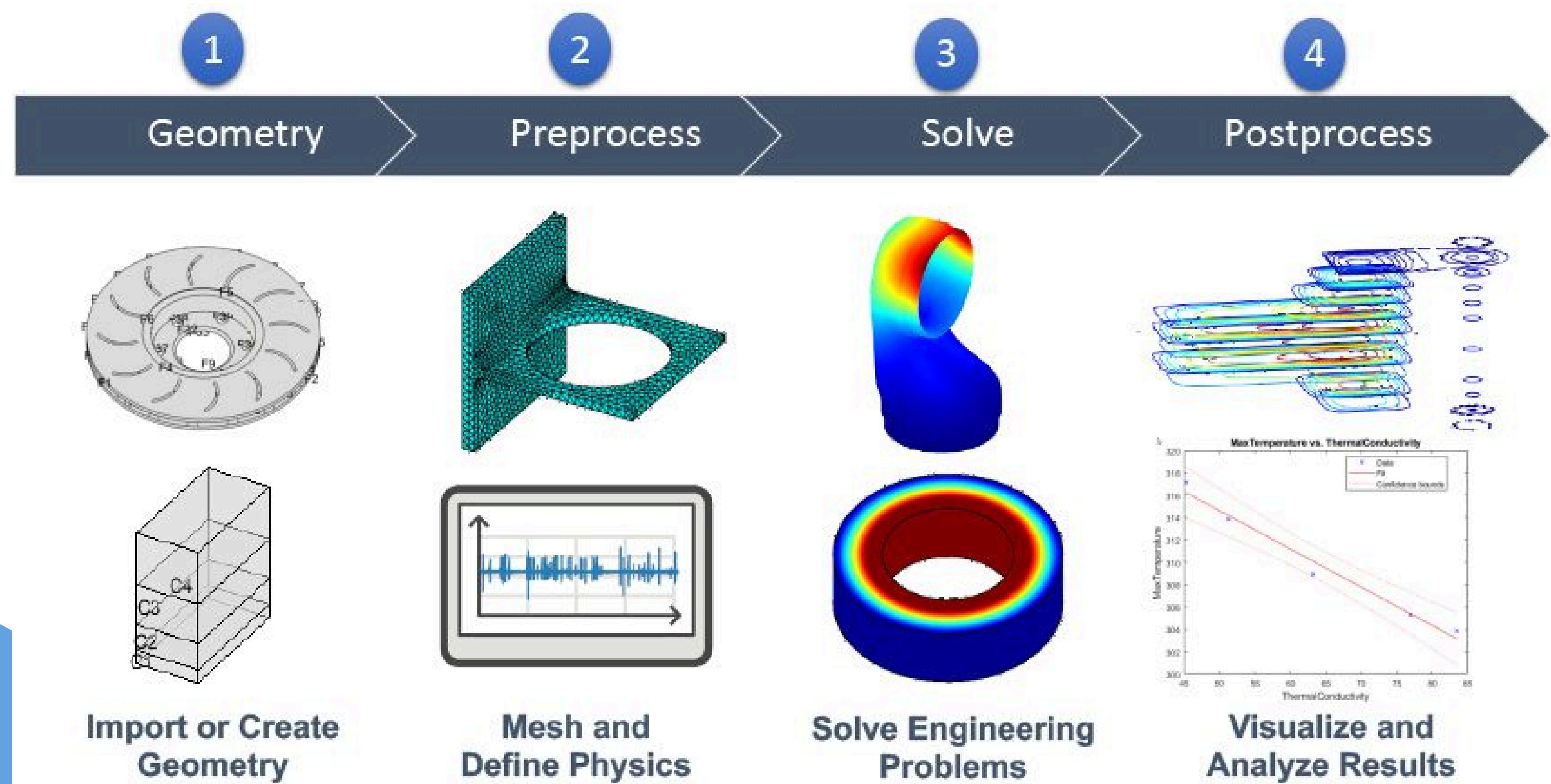
When we apply loads, forces, or constraints, the computer calculates how each element responds, then combines those results to predict the overall behavior of the material or structure.

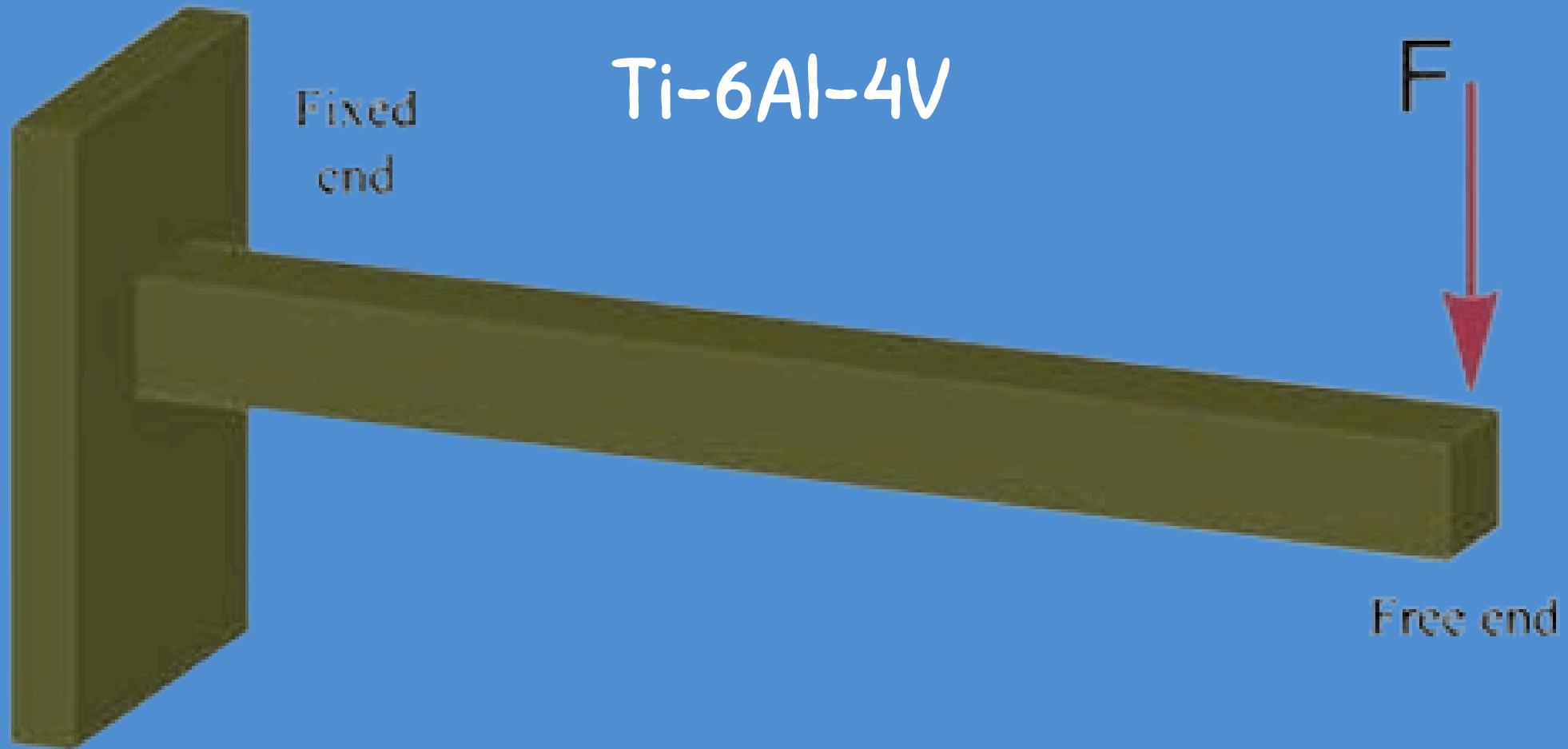
Advantages



It saves time and money, provides detailed insight into material behavior, and allows us to test conditions that might be impossible in real life.

FINITE ELEMENT ANALYSIS PROCESS

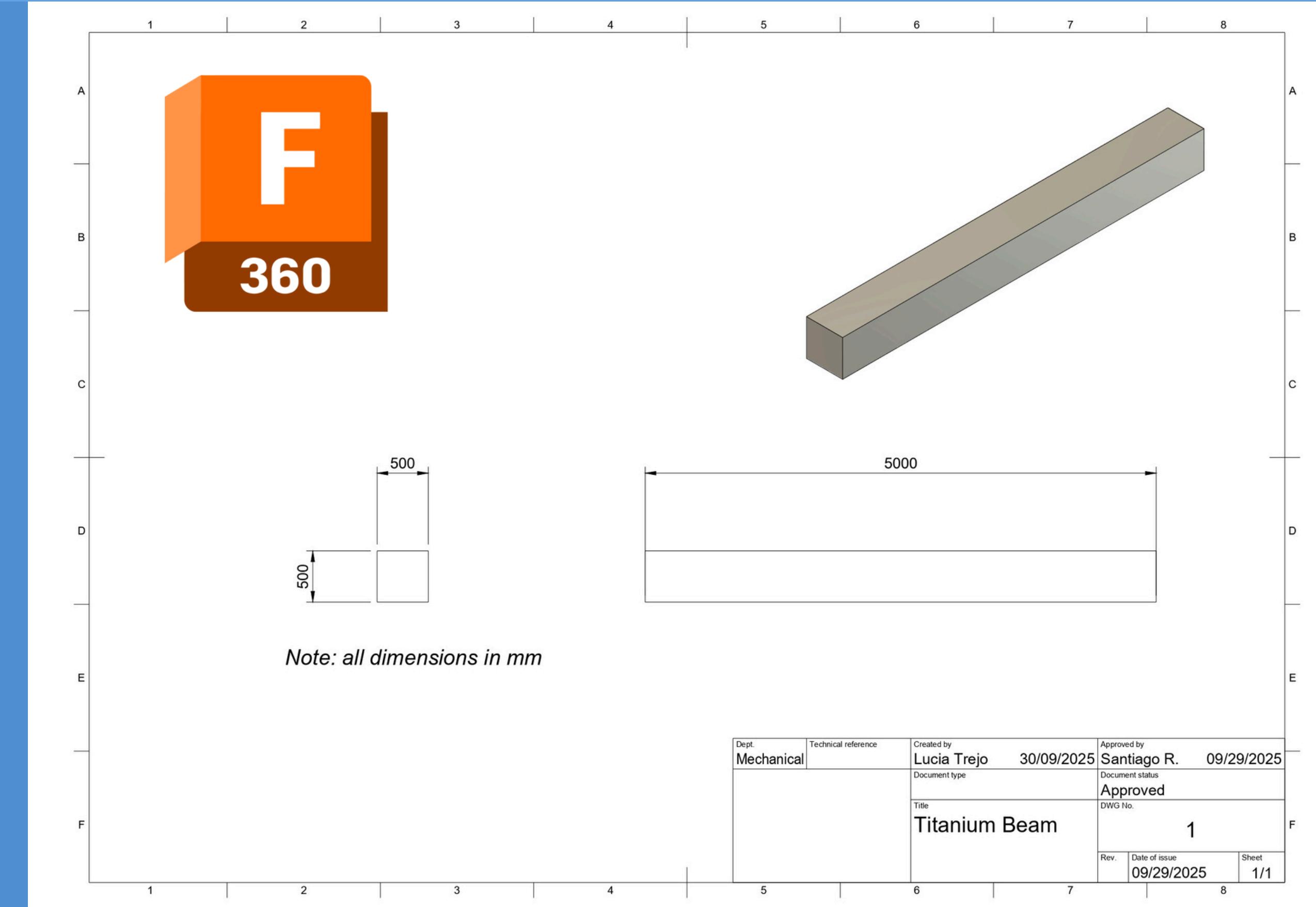




CASE OF STUDY EXAMPLE



1. Model simple beam geometry



The screenshot displays two side-by-side windows of the Workbench software, both titled "Unsaved Project - Workbench".

Left Window (Project Schematic):

- Toolbar:** File, View, Tools, Units, optiSLang, Extensions, Jobs, Help.
- Toolbox:** Hydrodynamic Diffraction, Hydrodynamic Response, LS-DYNA, LS-DYNA Acoustics, LS-DYNA Restart, Magnetostatic, Modal, Modal Acoustics, Particle Dynamics (Rocky), Random Vibration, Response Spectrum, Rigid Dynamics, Speos, Static Acoustics, Static Structural, Steady-State Thermal, Structural Optimization, Substructure Generation, Thermal-Electric, Transient Structural.
- Project Schematic:** A hierarchical tree view under section A:
 - 1 Static Structural
 - 2 Engineering Data (checked)
 - 3 Geometry
 - 4 Model
 - 5 Setup
 - 6 Solution
 - 7 Results

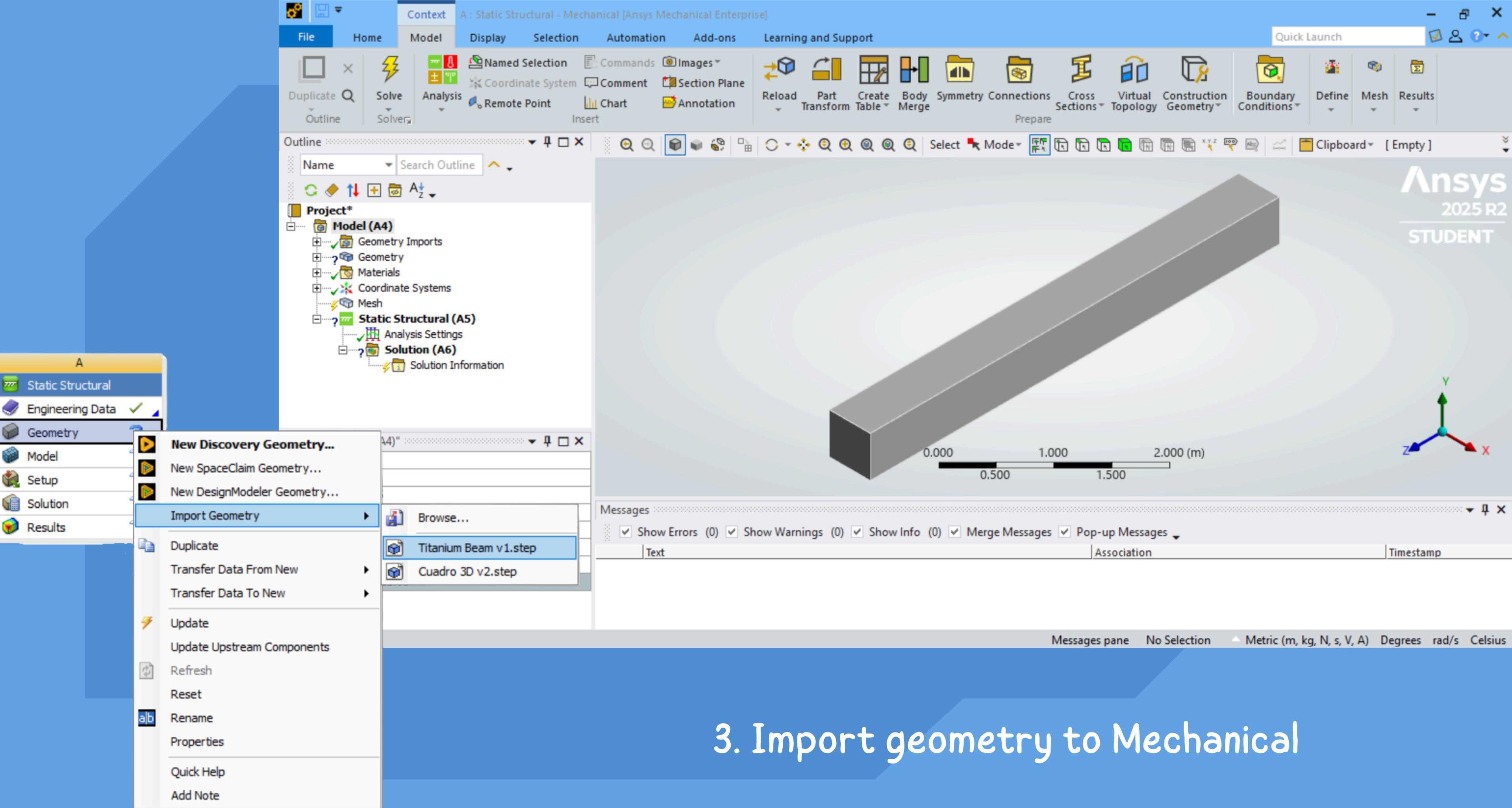
Right Window (Engineering Data):

- Toolbar:** File, View, Tools, Units, optiSLang, Extensions, Jobs, Help.
- Toolbox:** Physical Properties, Linear Elastic, Hyperelastic Experimental Data, Hyperelastic, Chaboche Test Data, Plasticity, Creep, Life, Strength, Gasket, Viscoelastic Test Data, Viscoelastic, Shape Memory Alloy, Geomechanical, Damage, Cohesive Zone, Fracture Criteria, Crack Growth Laws, Three Network Model, Hyperviscoelasticity.
- Outline of Schematic A2: Engineering Data:** A table showing the contents of engineering data for a Titanium alloy, Ti-6Al-4V, annealed.

	A	B	C	D	E
1	Contents of Engineering Data	(color icon)	(checkmark icon)	Source	Description
3	Titanium alloy, Ti-6Al-4V, annealed	(link icon)	(checkbox icon)	(link icon)	Titanium, alpha-beta alloy, Ti-6Al-4V, annealed Data compiled by Ansys Granta, incorporating various sources including JAHM and MagWeb. ANSYS, Inc. provides no warranty for this data.
- Properties of Outline Row 3: Titanium alloy, Ti-6Al-4V, annealed:** A detailed table of material properties.

	A	B	C	D	E
1	Property	Value	Unit	(checkmark icon)	(link icon)
2	Material Field Variables	Table			
3	Density	4429	kg m ⁻³	(checkbox icon)	(checkbox icon)
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
12	Tensile Yield Strength	Tabular			
13	Tensile Ultimate Strength	Tabular			
14	Isotropic Thermal Conductivity	7.187	W m ⁻³ K ⁻¹	(checkbox icon)	(checkbox icon)
15	Specific Heat Constant Pressure, C _p	522.6	J kg ⁻¹ K ⁻¹	(checkbox icon)	(checkbox icon)
16	Isotropic Resistivity	1.69E-06	ohm m	(checkbox icon)	(checkbox icon)

2. Define the material properties.



3. Import geometry to Mechanical

Screenshot of Ansys Mechanical Enterprise 2025 R2 Student Edition interface showing a 3D model of a titanium beam and its mesh settings.

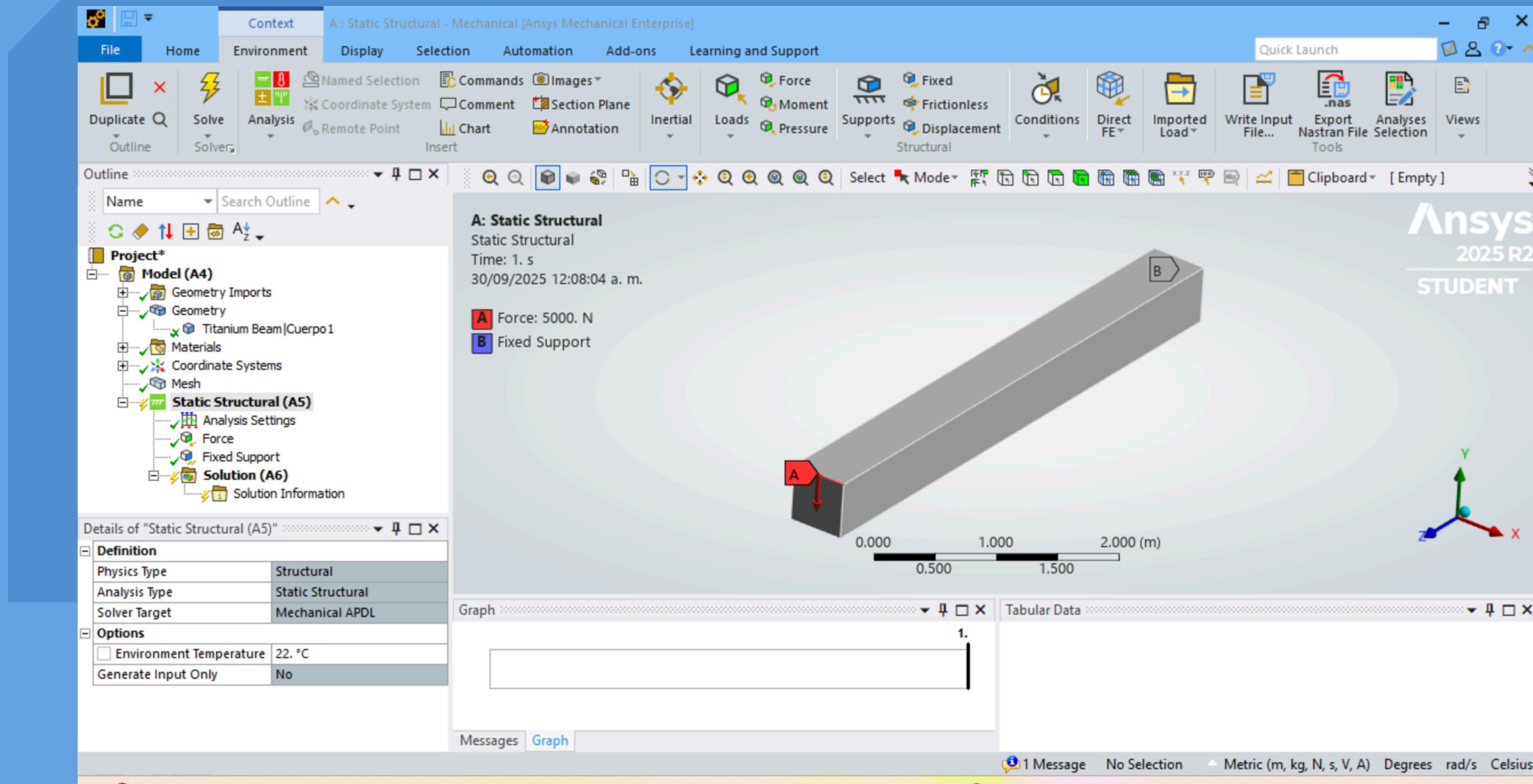
The interface includes:

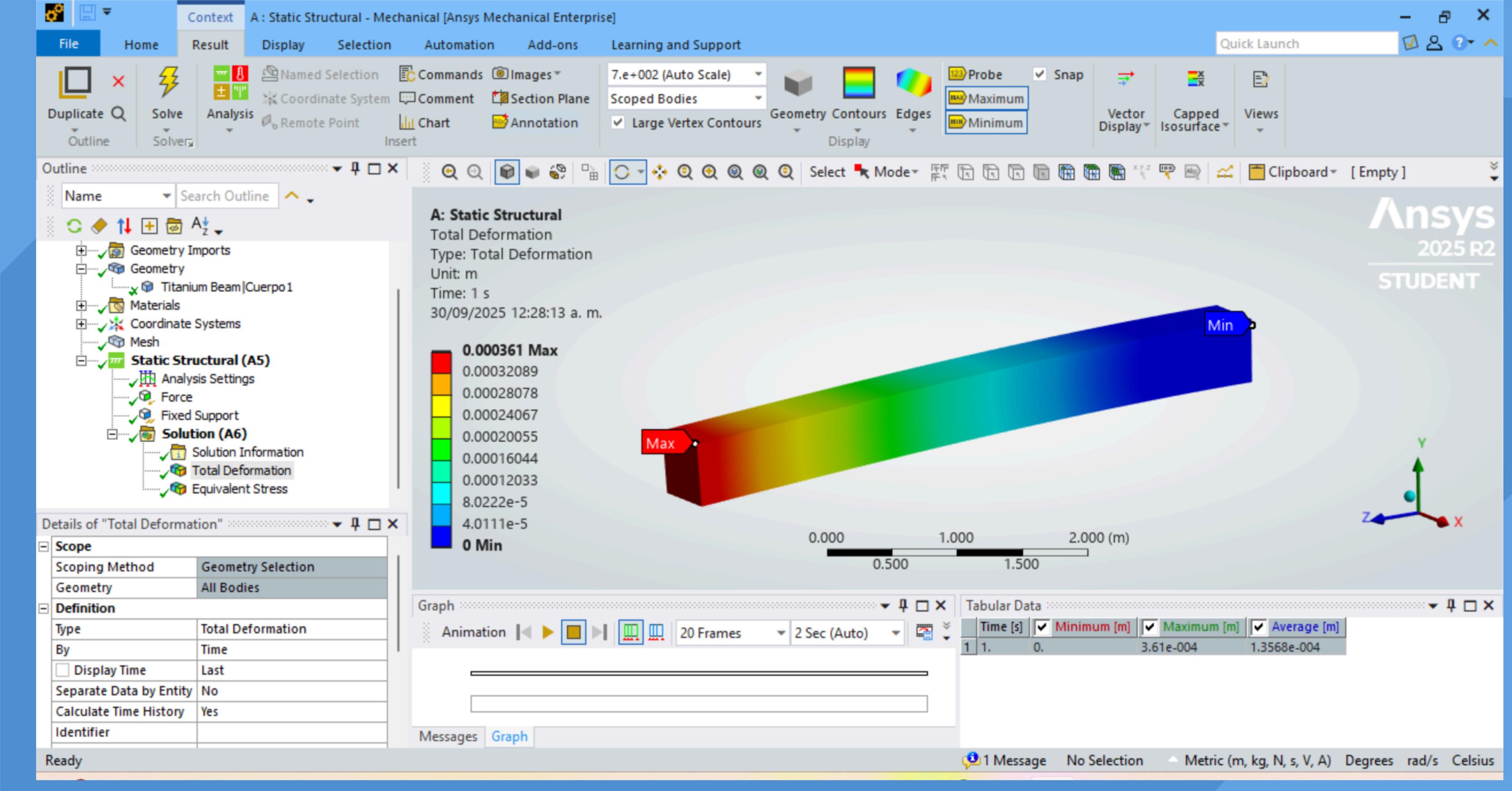
- Top Bar:** File, Home, Mesh (selected), Display, Selection, Automation, Add-ons, Learning and Support, Quick Launch.
- Toolbar:** Mesh Workflows, Feature Detection, Update, Generate, Mesh Worksheet, Surface Mesh, Source/Target, Method, Sizing, Face Meshing, Mesh Copy, Match Control, Contact Sizing, Inflation, Refinement, Gasket, Pinch, Feature Suppress, Connect, Geometry Fidelity, Deviation, Weld, Quad Layer, Mesh Edit, Quality Worksheet, Metrics Display.
- Outline View:** Shows the Project structure: Model (A4) containing Geometry Imports, Geometry (with Titanium Beam | Cuerpo1), Materials, Coordinate Systems, Mesh, and Static Structural (A5).
- Details of "Mesh" Dialog:** Settings for Display Style (Use Geome...), Element Order (Mechanical), Element Size (5.e-002 m), Sizing (Use Adaptive Sizing Yes, Resolution Default (2), Mesh Defeaturing Yes).
- 3D View:** A 3D model of a rectangular beam with a cross-hatched mesh. Dimensions shown are 0.600 (m) length, 0.150 width, and 0.150 height. A coordinate system (X, Y, Z) is also visible.
- Messages View:** Shows a message: "When you change the Color Theme, Mechanical automatically uses the background color".

4. Generate a structured mesh

At least 8-12 elements across gauge thickness/width

5. Apply Boundary Conditions: Fixed Support and Force (F)



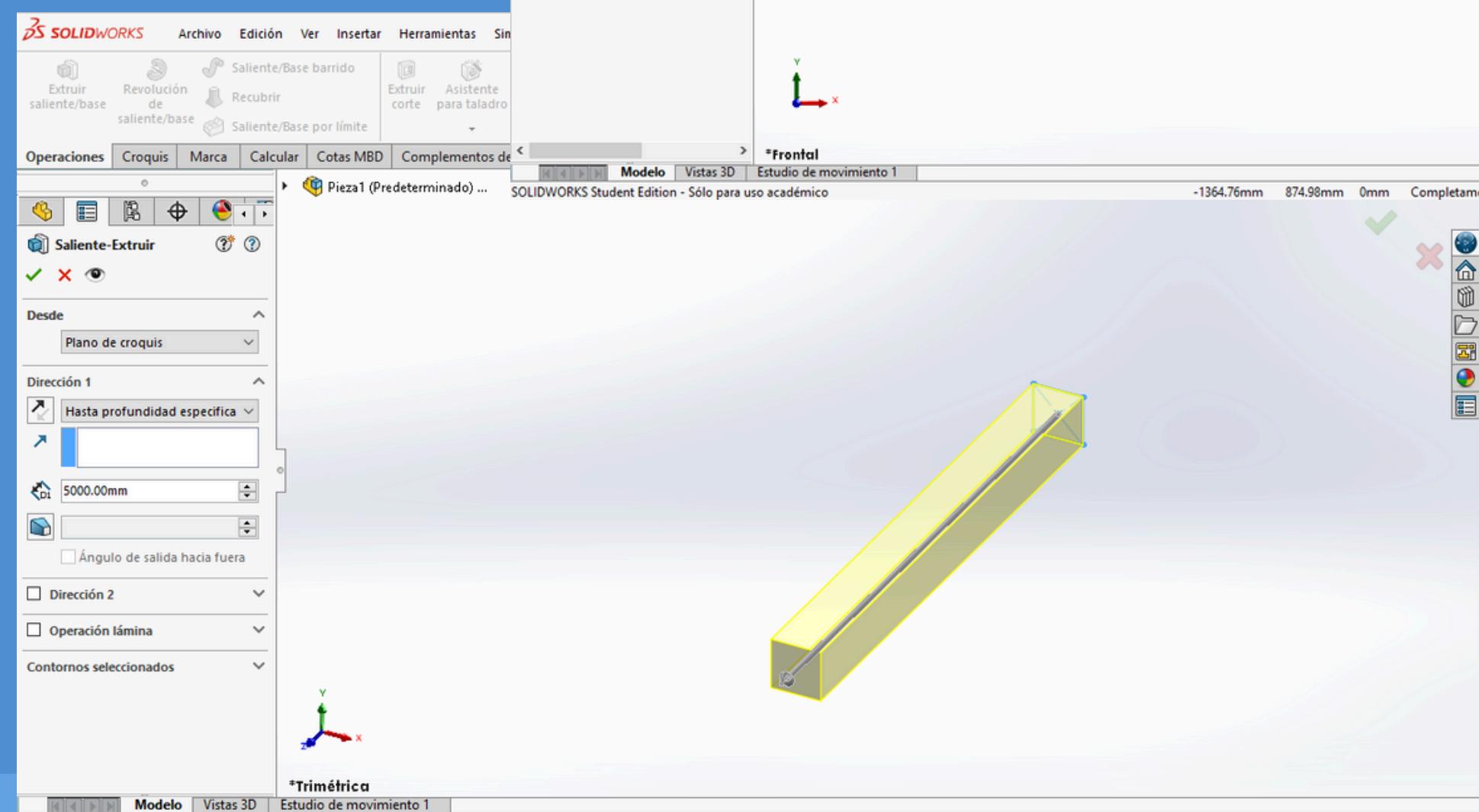


6. Solve the model.

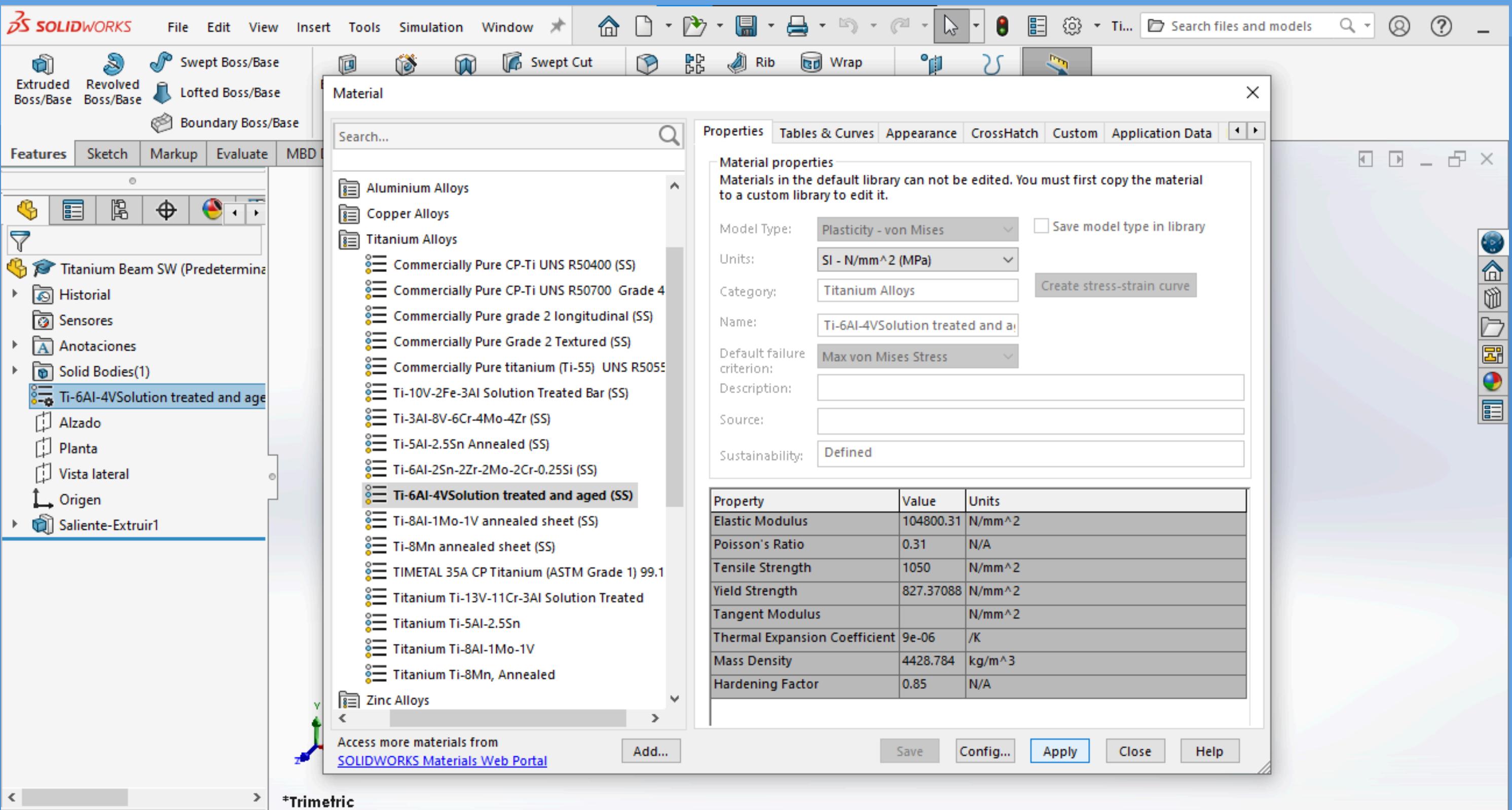
Extract Total Deformation, Equivalent (von Mises) Stress, and Strain results.



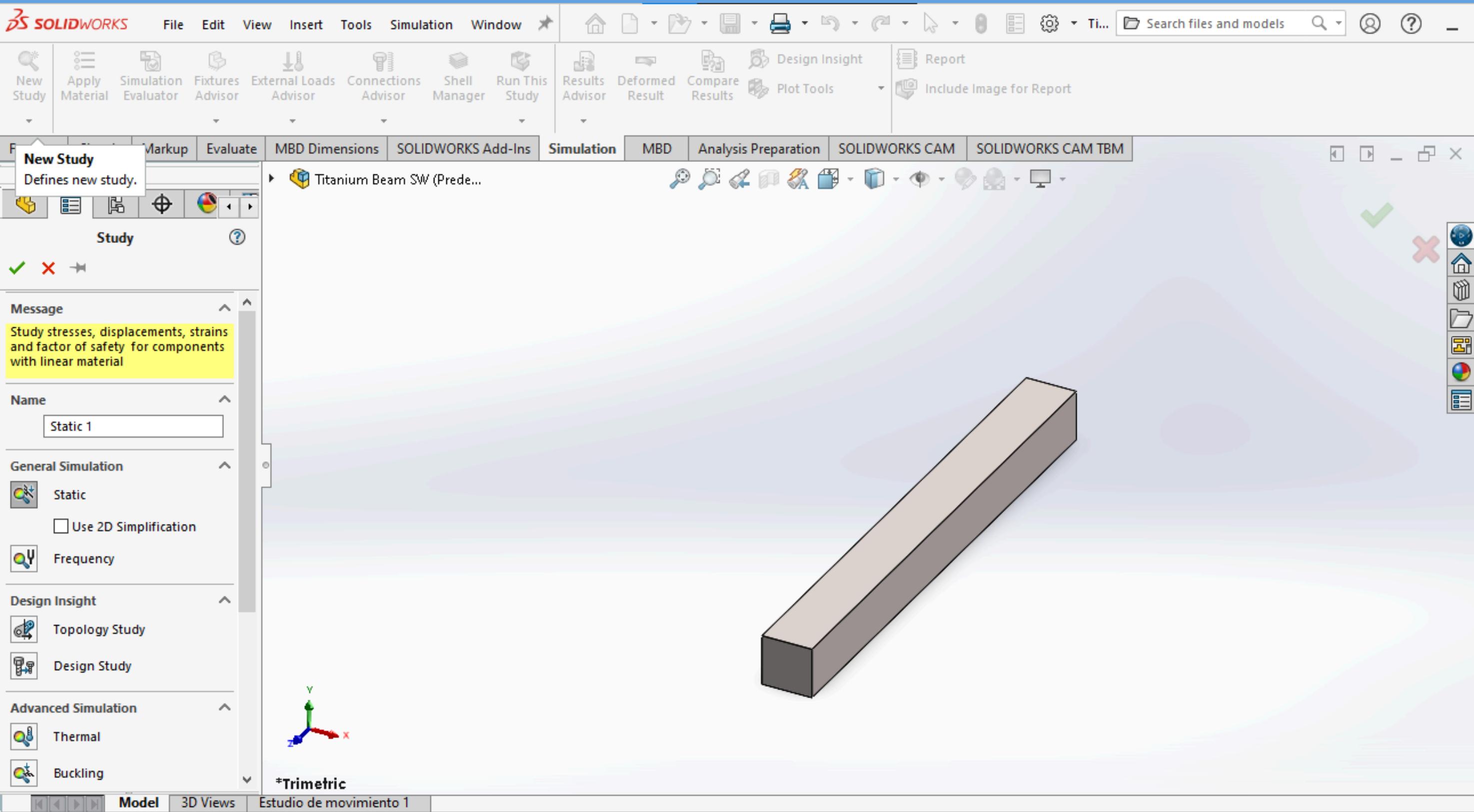
1. Model simple beam geometry



Actual
disponib
Hay
una
nueva
actual
de
3DEX
Market
para
el
compl
de
SOLID
dispon
para
su
descar
6.33.4

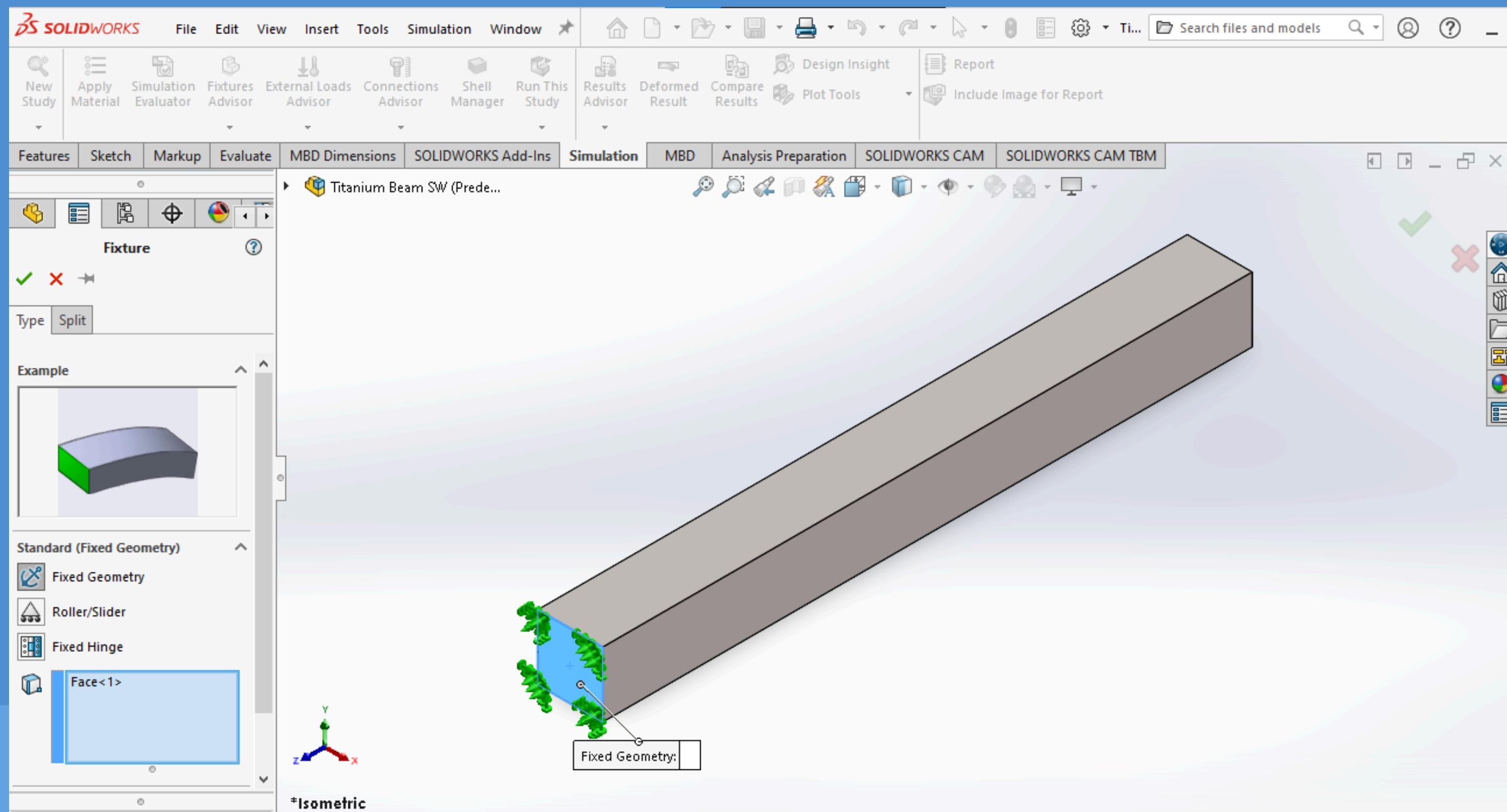


2. Define the material properties.

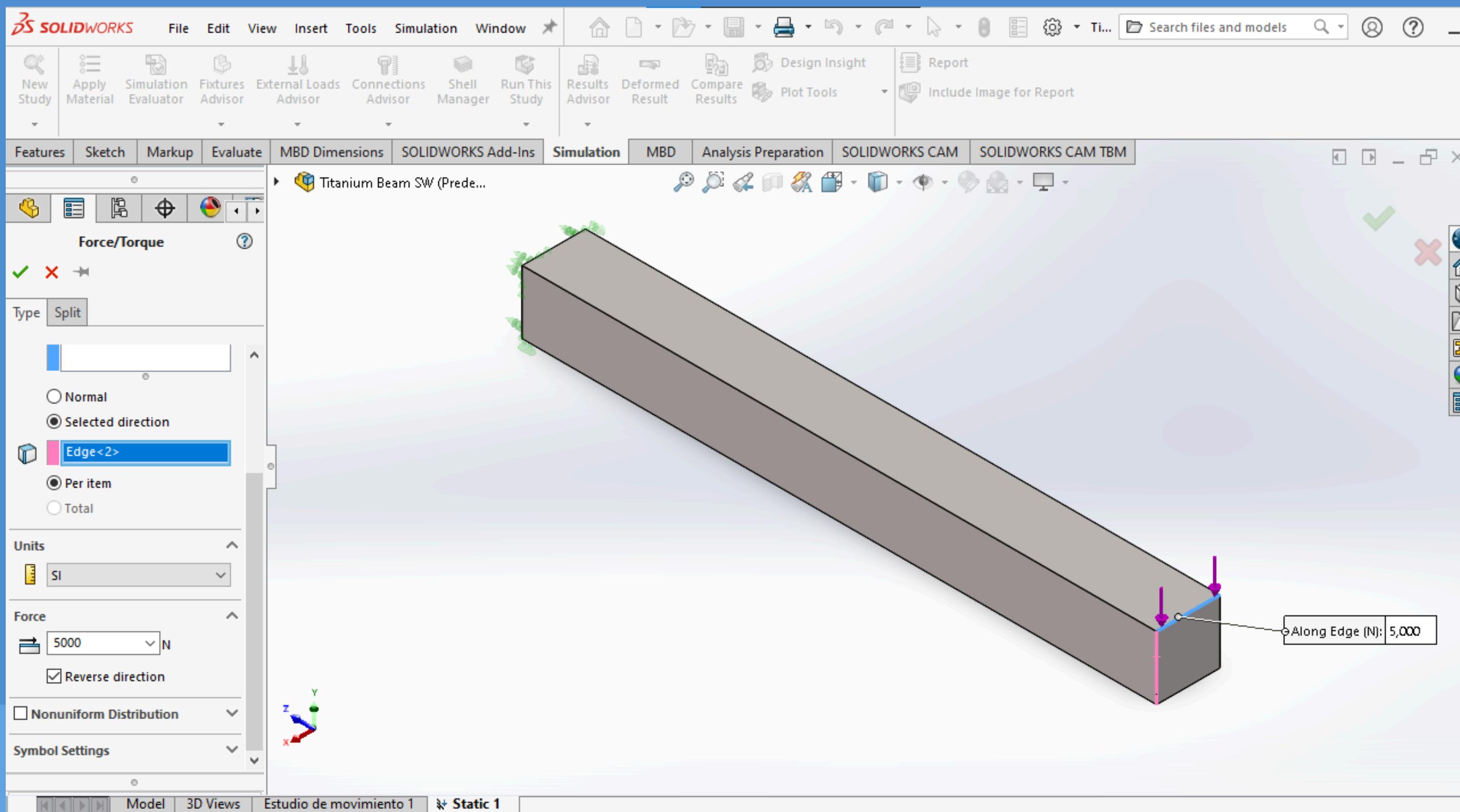


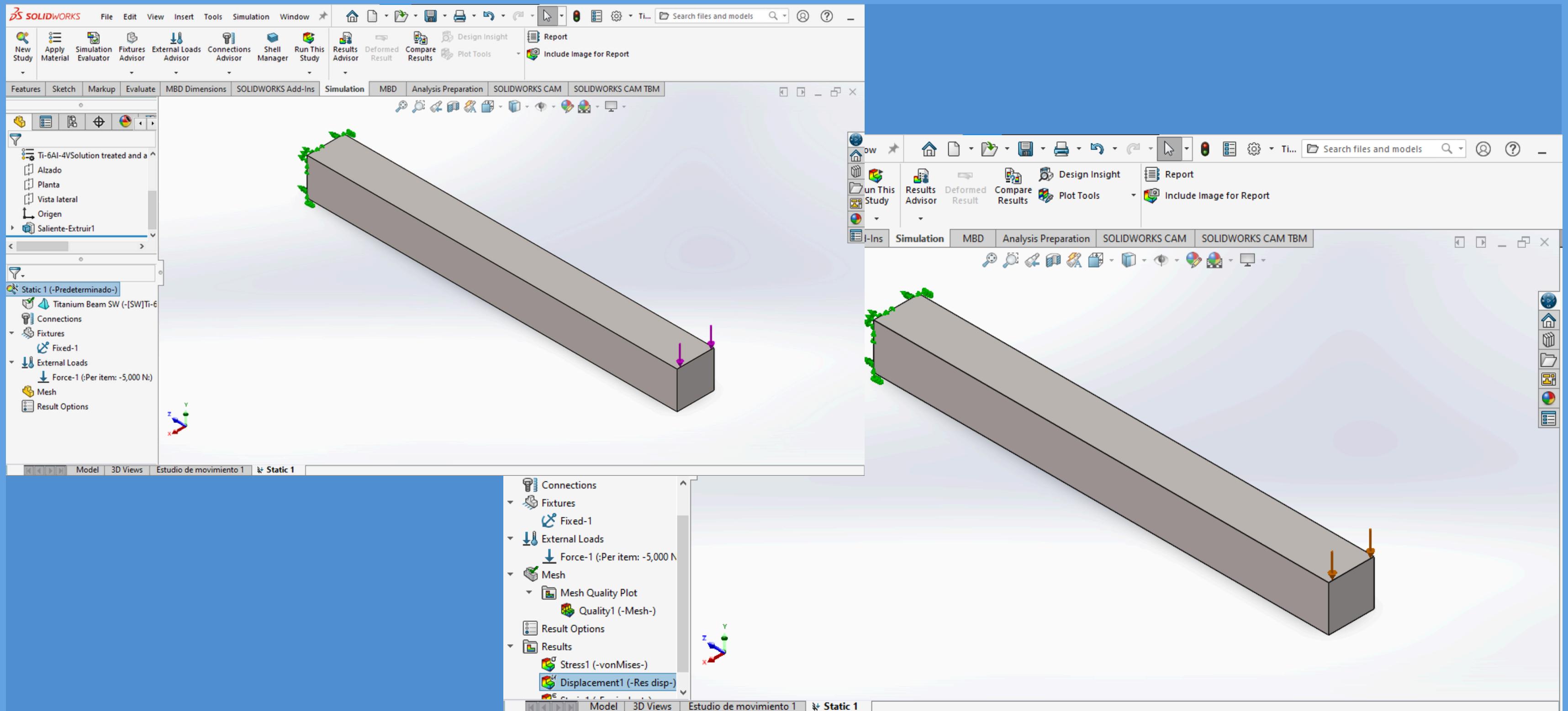
3. Start New Static Study

5. Apply Boundary Conditions: Fixed Support



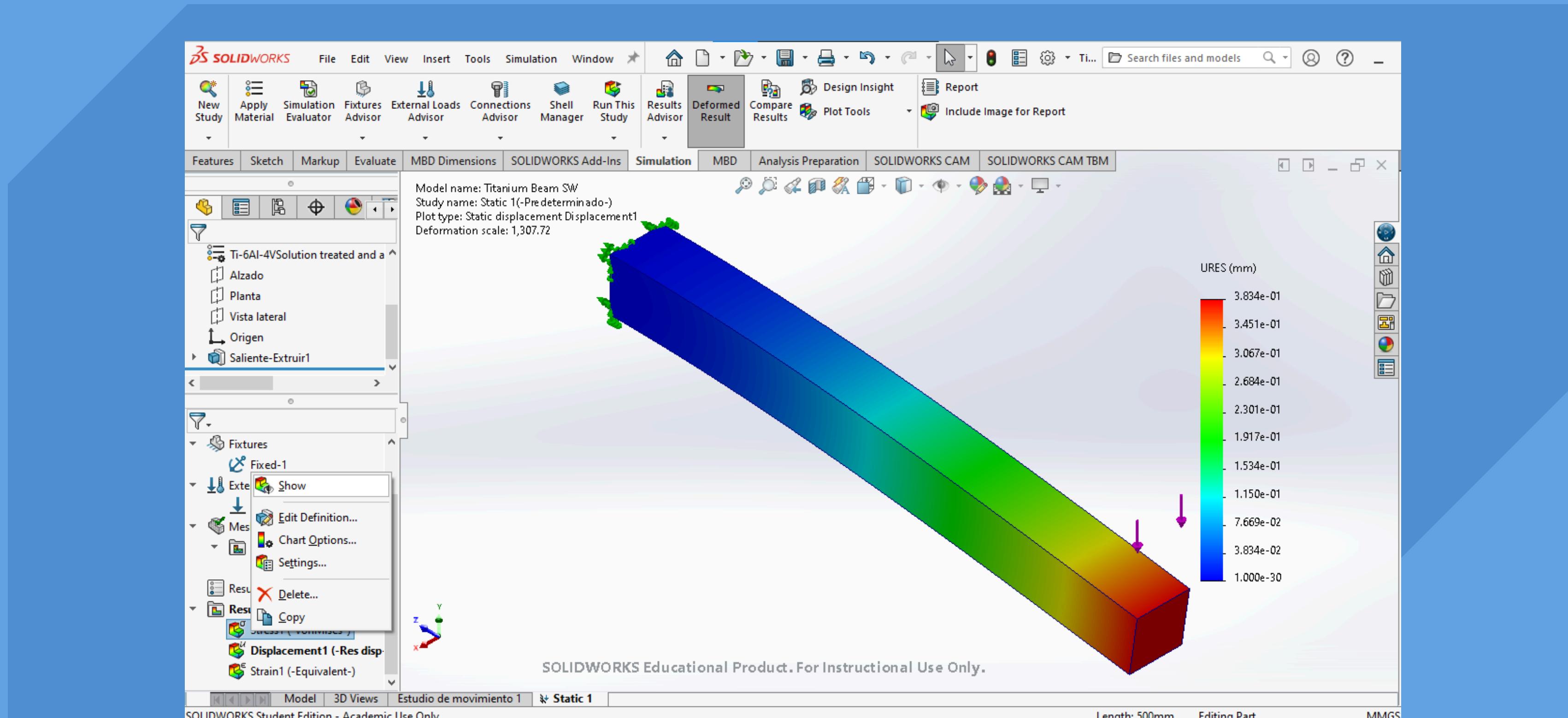
5. Apply Boundary Conditions: Force (F)



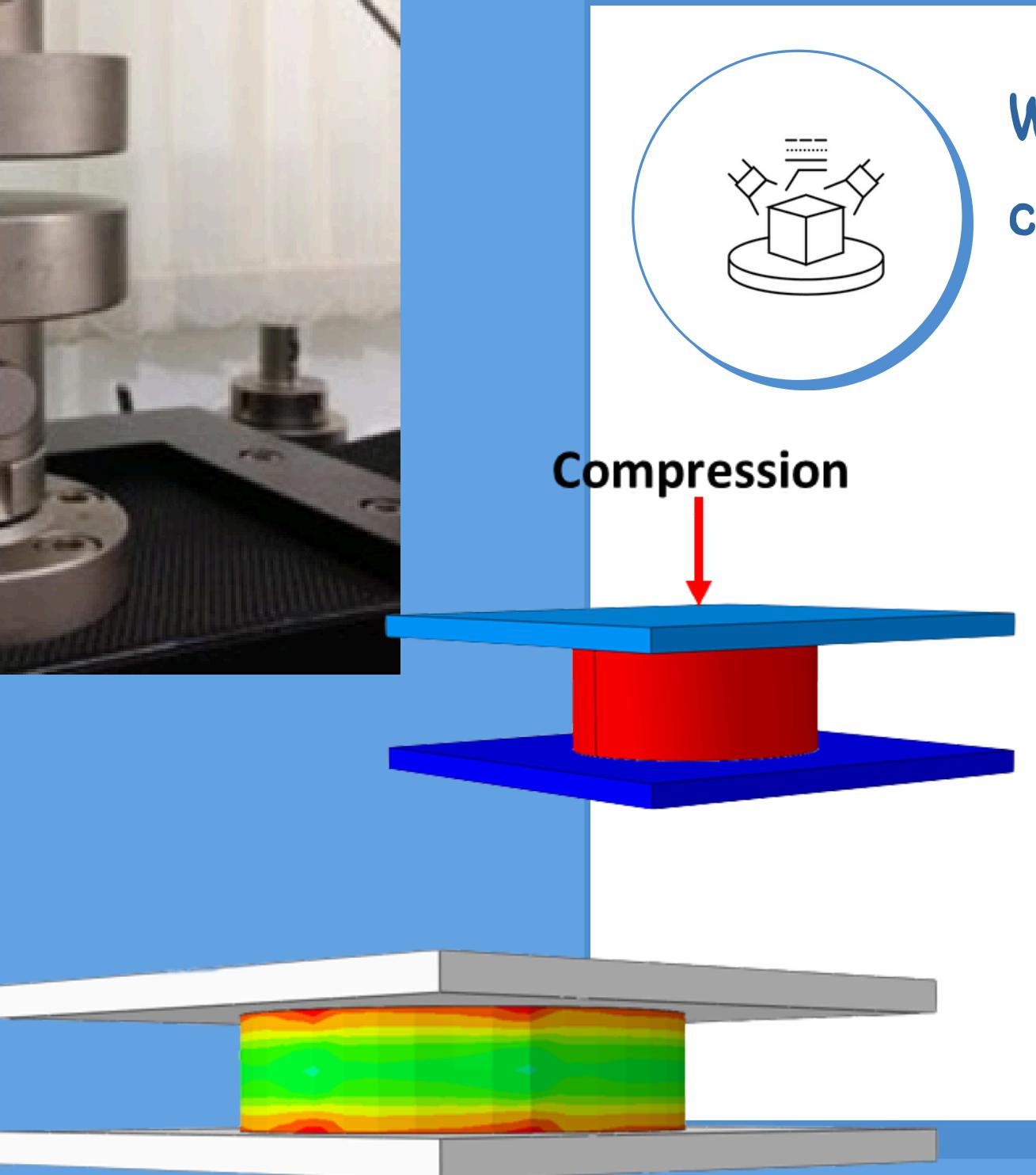


6. Solve the model.

Extract Total Deformation, Equivalent (von Mises) Stress, and Strain results.

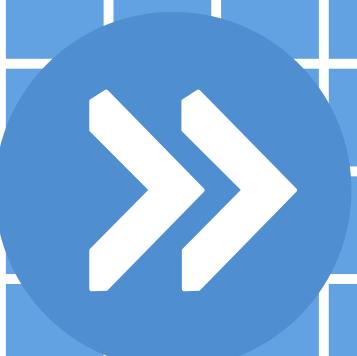


7. Displacement, Stress (von Mises), and Strain results.

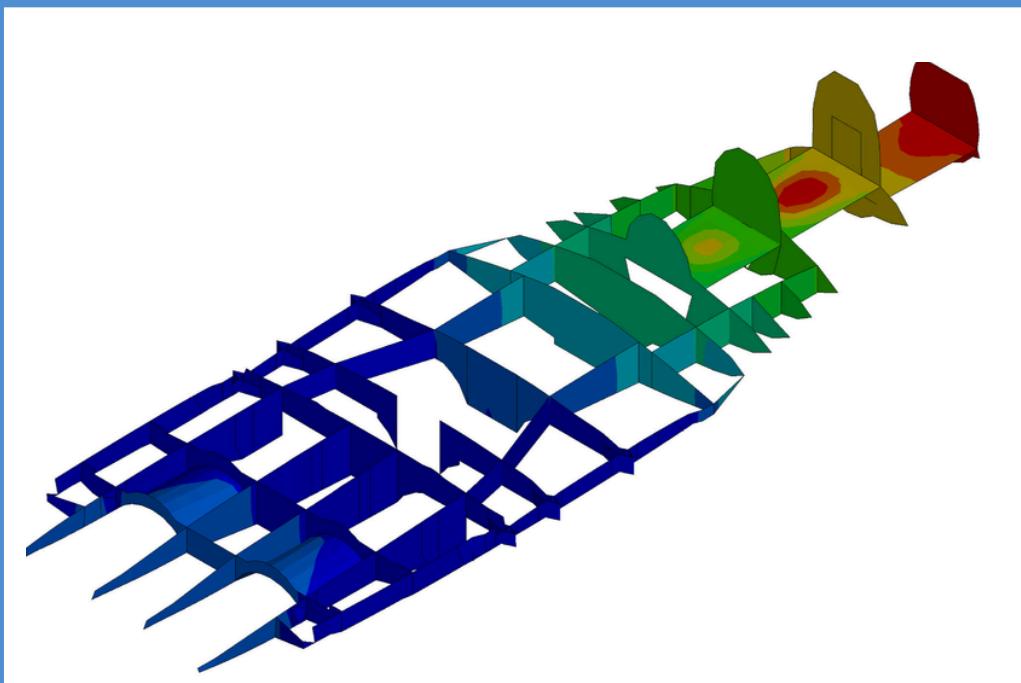


Why does this matter for materials characterization?

FEA allows us to simulate mechanical tests, such as tensile or compression tests, and predict how a material will behave under stress. For example, if we design a new composite for aerospace, FEA can show us where it might deform, crack, or fail, before we even build a prototype. It helps us compare different material properties, optimize designs, and better understand microscopic behaviors like stress concentration or fatigue life.

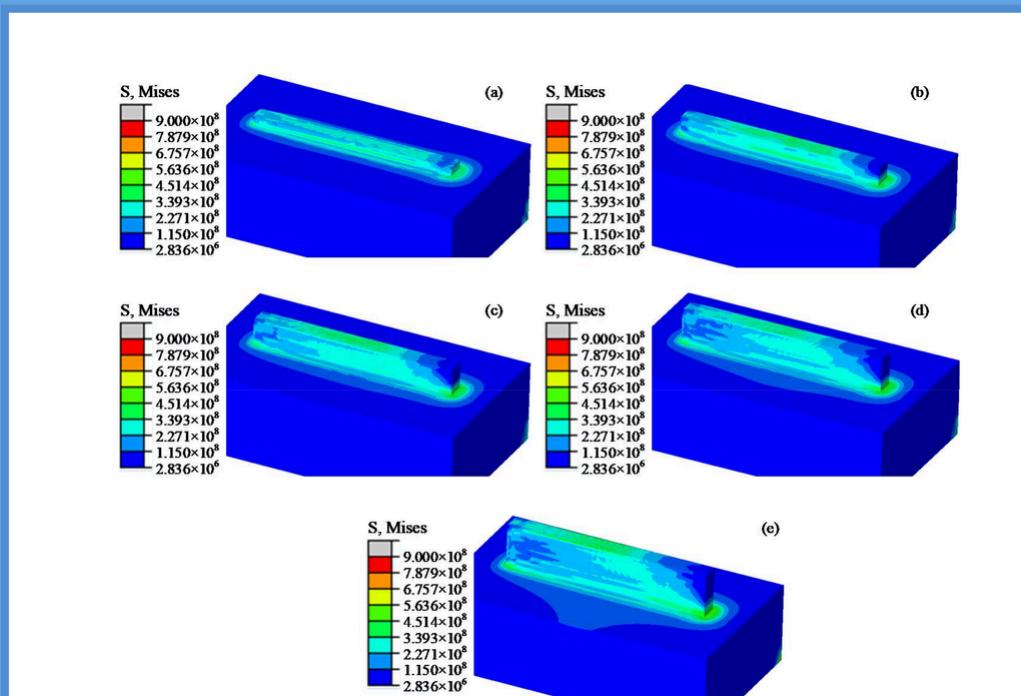


IMPORTANCE IN RESEARCH FIELD



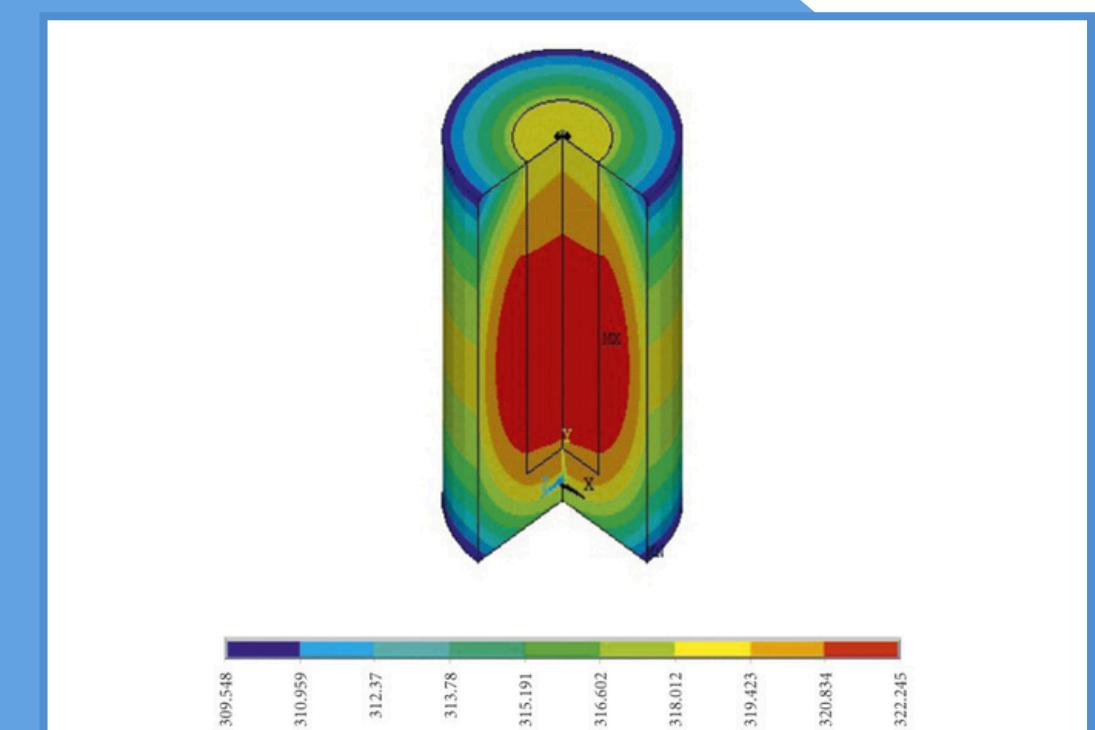
Aerospace

In aerospace engineering, we use FEA to evaluate lightweight materials, making sure they're strong enough to handle flight conditions.



Advanced manufacture

In additive manufacturing, it helps us predict how 3D-printed parts might show defects, residual stresses, and anisotropy ensuring better quality and consistency.



Materials

In materials research, it can complement mechanical testing by validating experimental data with simulations, giving us more confidence in the results.

RECAP

Limitations

FEA is only as good as the model we build, so it depends on accurate material properties, correct boundary conditions, and careful validation against experiments.

Conclusion

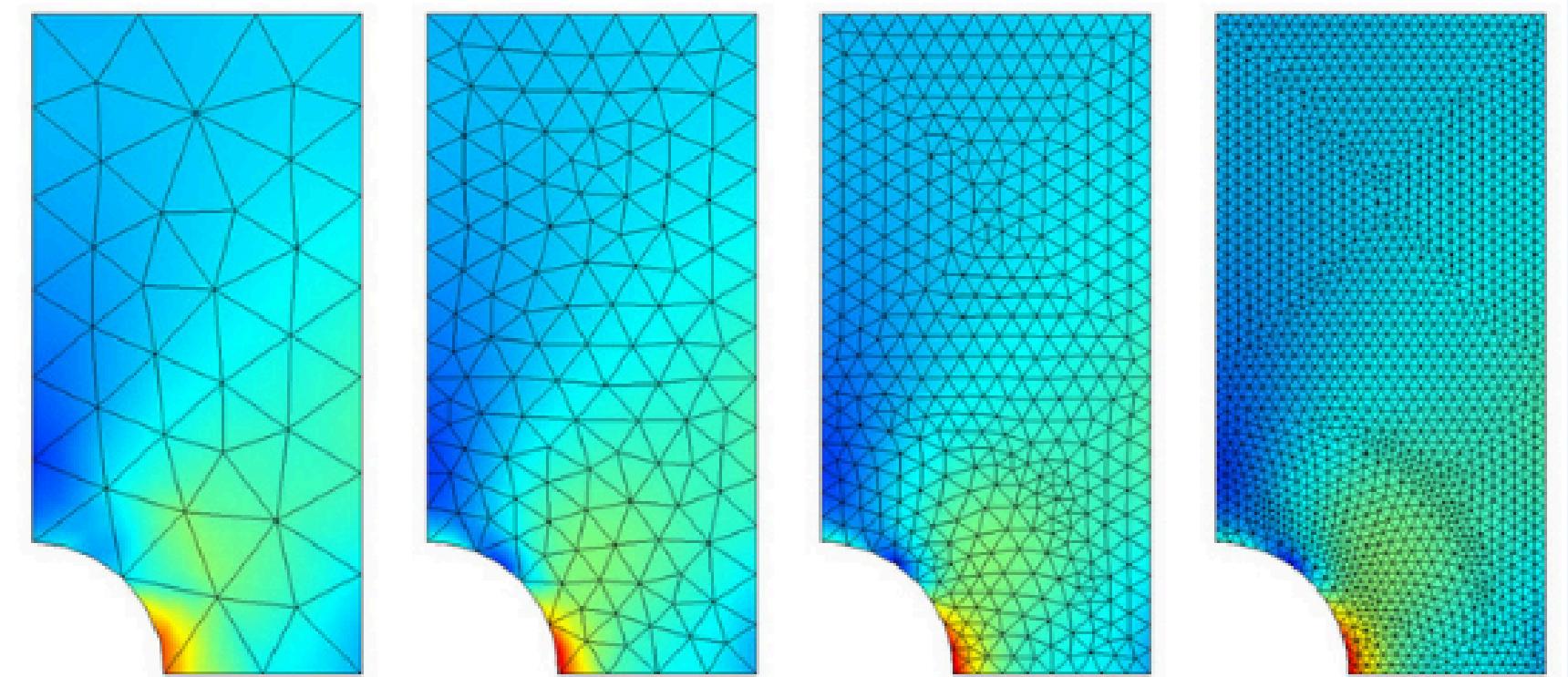
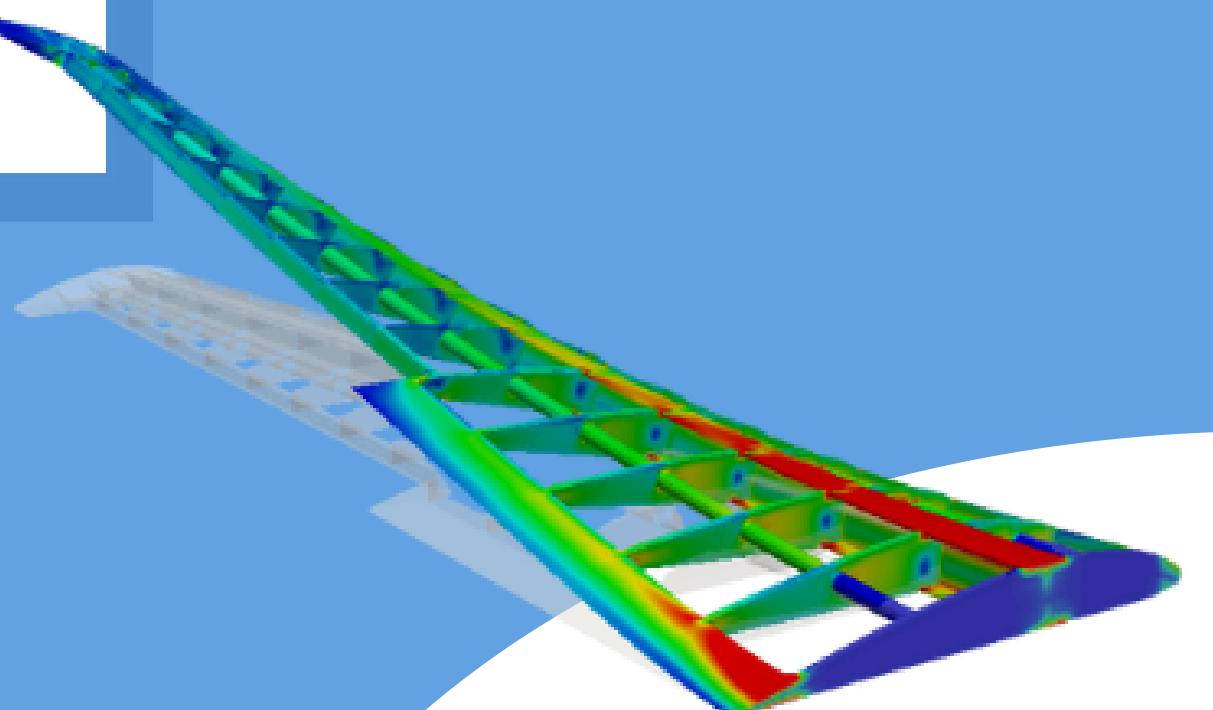
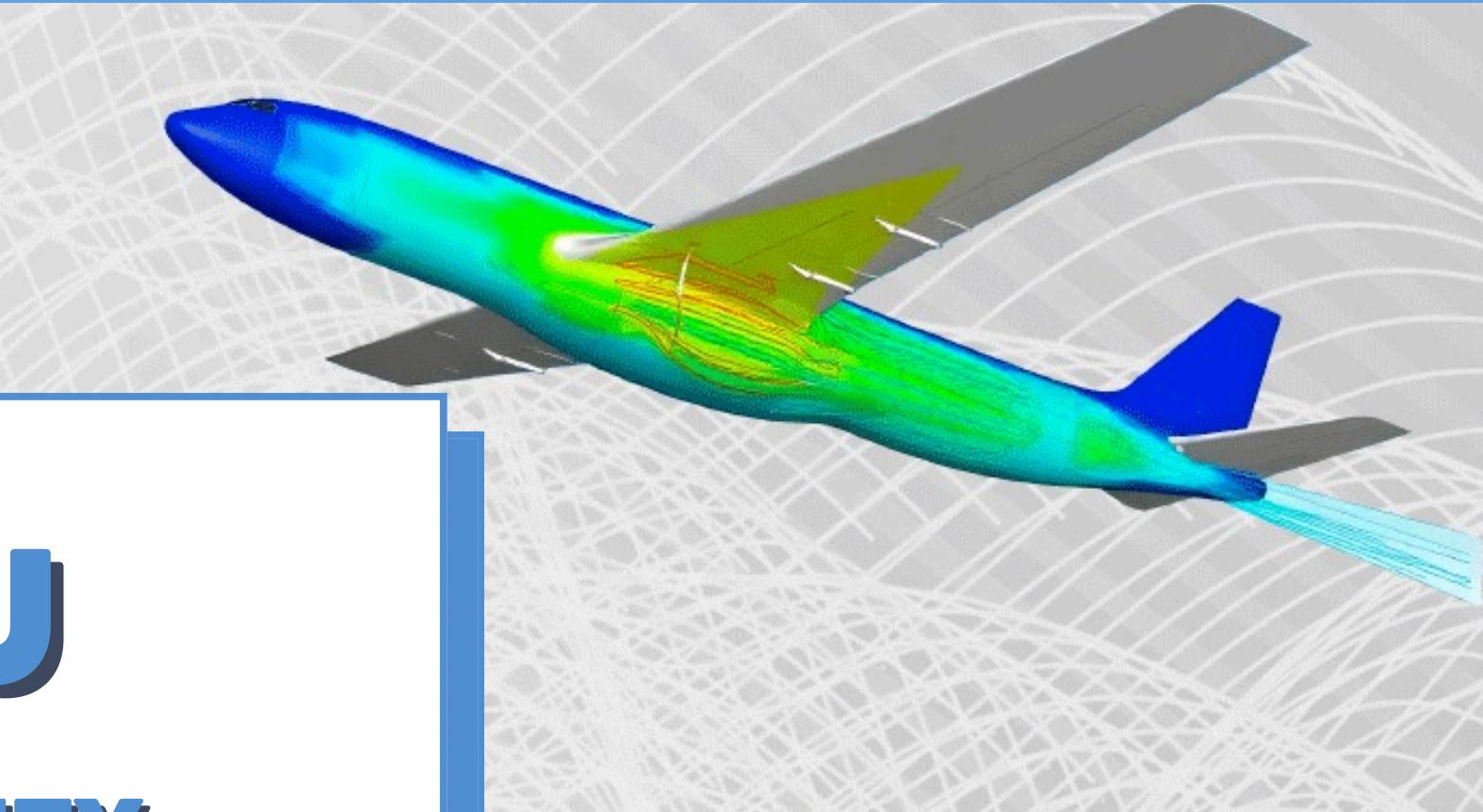
FEA is not just a simulation tool; it's a bridge between theory and experiment. By combining physical testing with numerical modeling, we can deepen our understanding of materials and make better engineering decisions.

Implementation

FEA does not replace experiments, but complements them. Together, simulation and experimental data provide a deeper and more reliable understanding of materials.



THANK YOU
IVRT – YORK UNIVERSITY



REFERENCES

- Finite element analysis (FEA) software. (2025, September 8). Autodesk.com.
<https://www.autodesk.com/solutions/simulation/finite-element-analysis>
- Koutromanos, I. (2020). Fundamentals of finite element analysis: Nonlinear finite element analysis. Wiley-Blackwell.
- Moatamed, M., & Khawaja, H. (2021). Finite element analysis. CRC Press.
- Nicholson, D. W. (2019). Finite element analysis: Thermomechanics of solids, second edition (2nd ed.). CRC Press.
- Szabó, B., & Babuška, I. (2021). Finite element analysis: Method, verification and validation (2nd ed.). Standards Information Network.
- What is FEA. (2018, December 15). SimScale. <https://www.simscale.com/docs/simwiki/fea-finite-element-analysis/what-isfea-finite-element-analysis/>
- (N.d.-a). Uvic.Ca. Retrieved September 30, 2025, from
https://www.engr.uvic.ca/~mech410/lectures/FEA_Theory.pdf
- (N.d.-b). Ansys.com. Retrieved September 30, 2025, from [https://www.ansys.com/simulation-topics/what-is-finite-element-analysis#:~:text=Finite%20element%20analysis%20\(FEA\)%20is,of%20the%20results%20FEM%20provides.](https://www.ansys.com/simulation-topics/what-is-finite-element-analysis#:~:text=Finite%20element%20analysis%20(FEA)%20is,of%20the%20results%20FEM%20provides.)