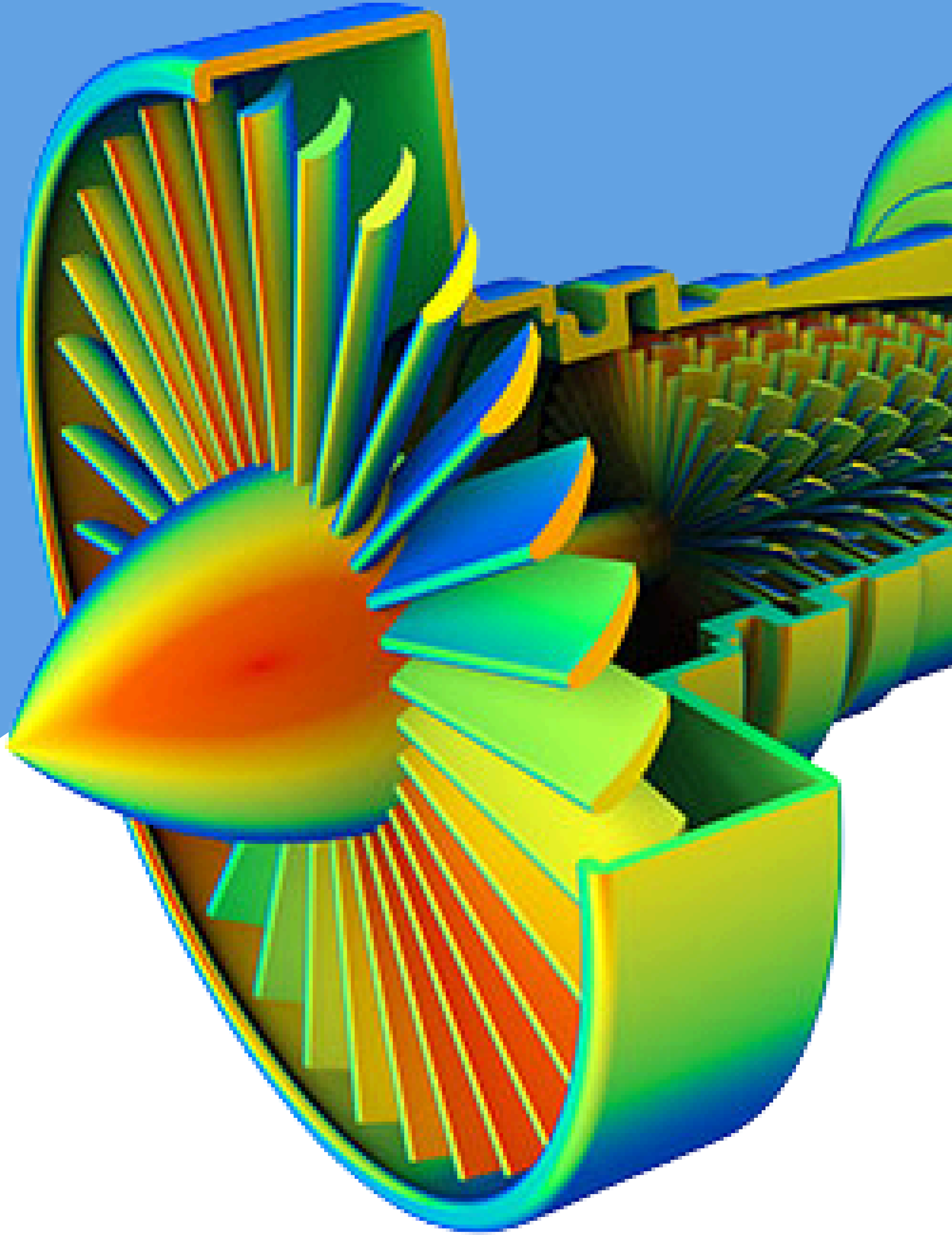
INTRODUCTION TO FINITE ELEMENT ANALYSIS (FEA)

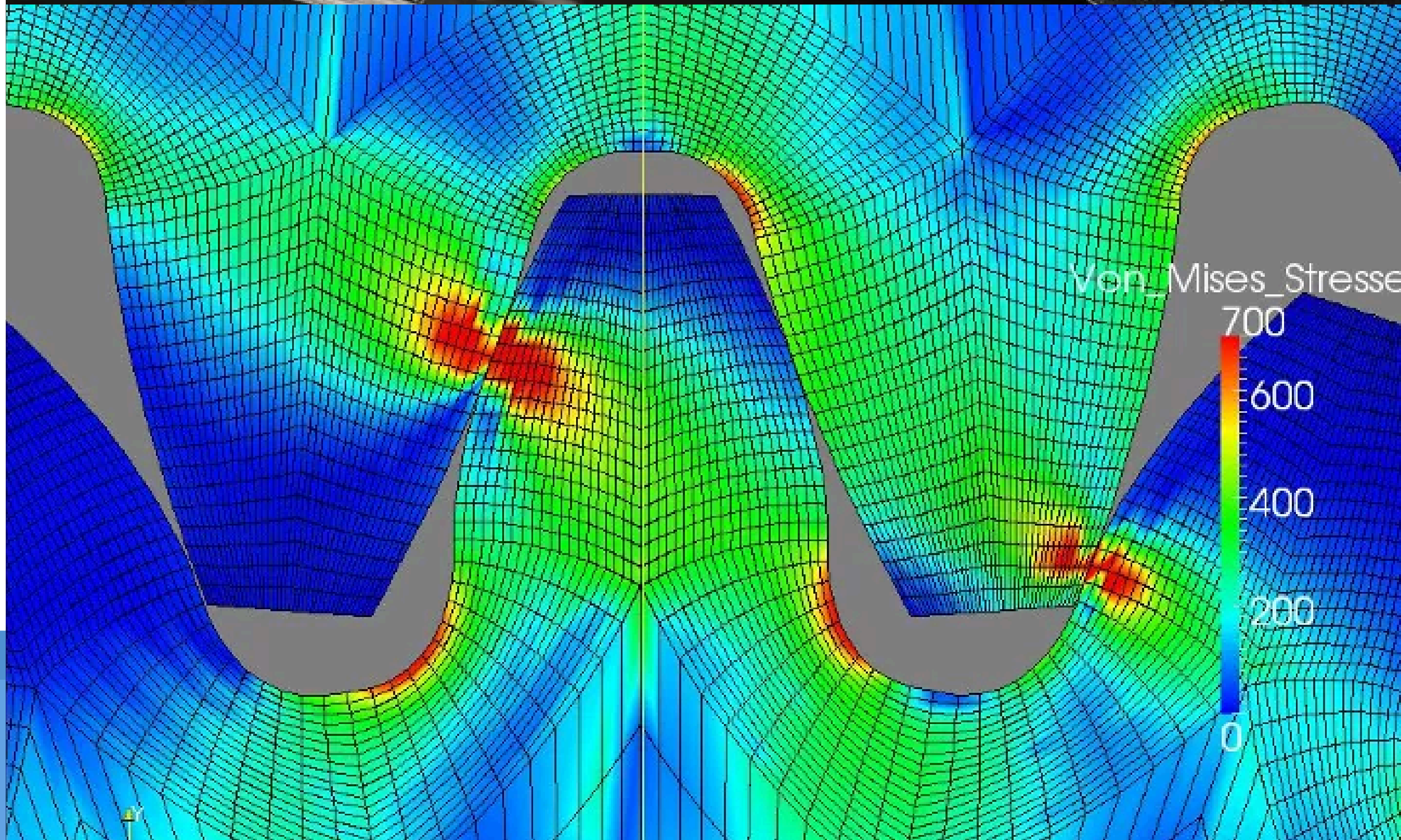
With applications in materials characterization



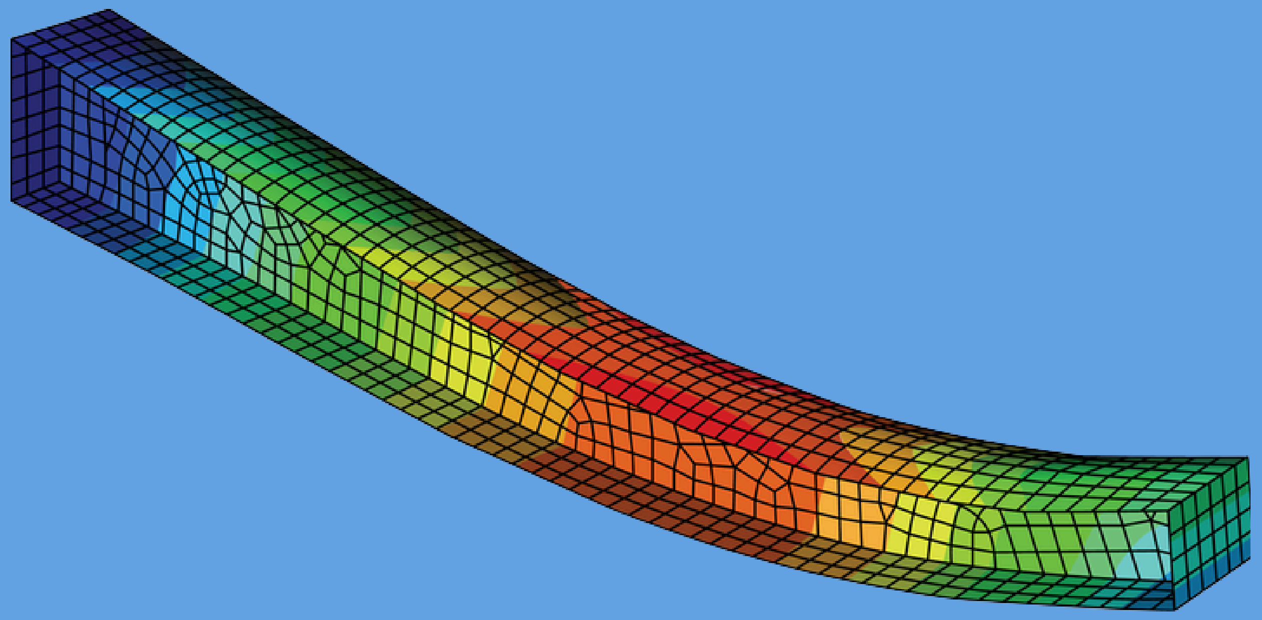
WHAT IS FEA?

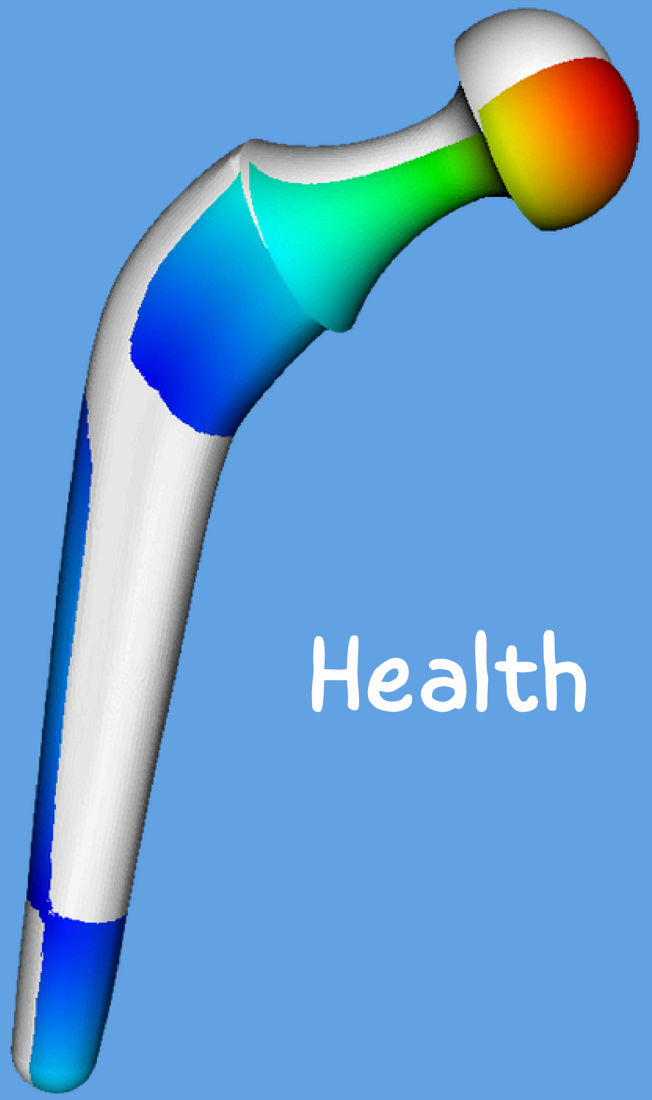
Finite Element Analysis is a numerical method to predict how structures or materials behave under loads, heat, or other conditions.

- Why it matters for engineers and scientists?

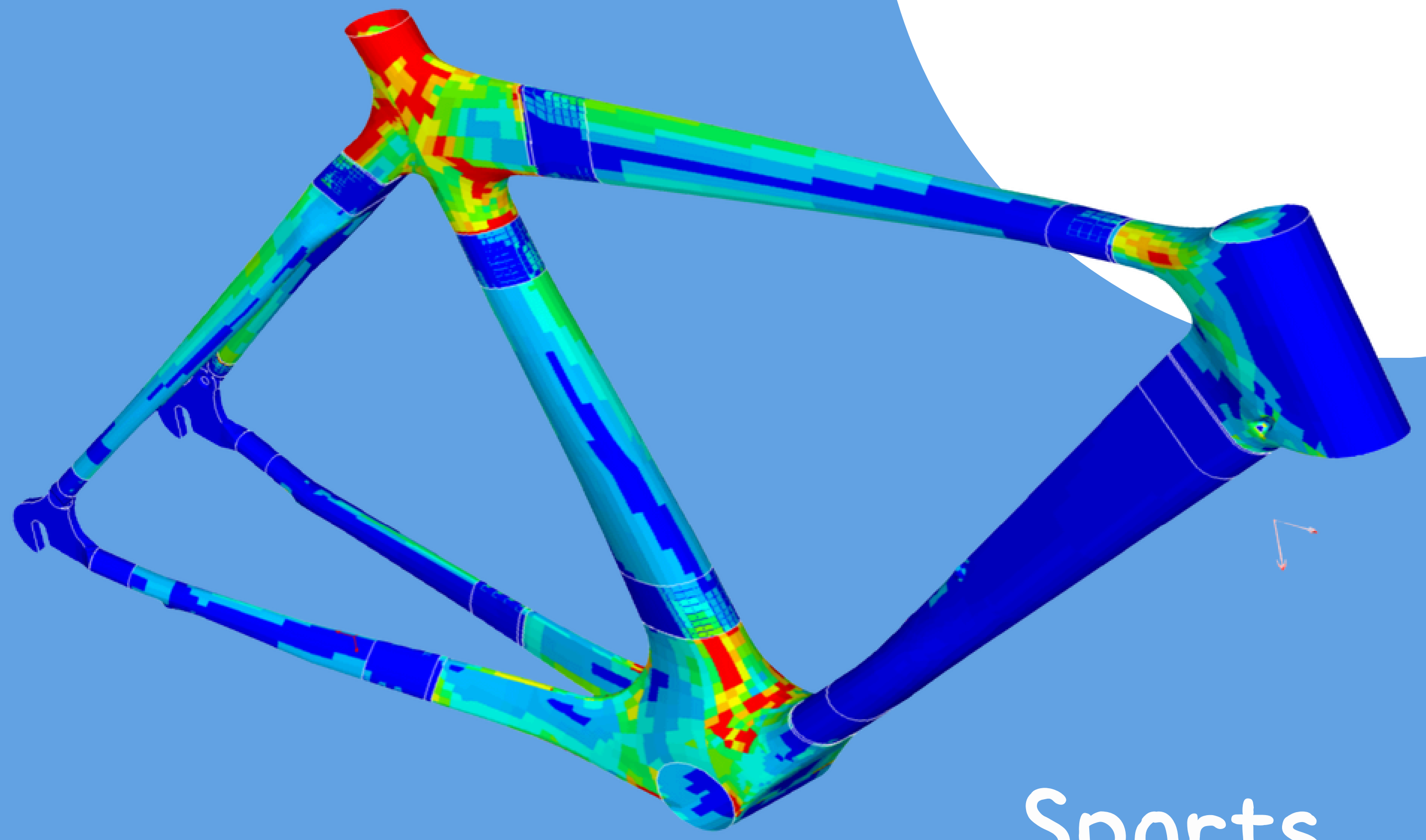


Construction

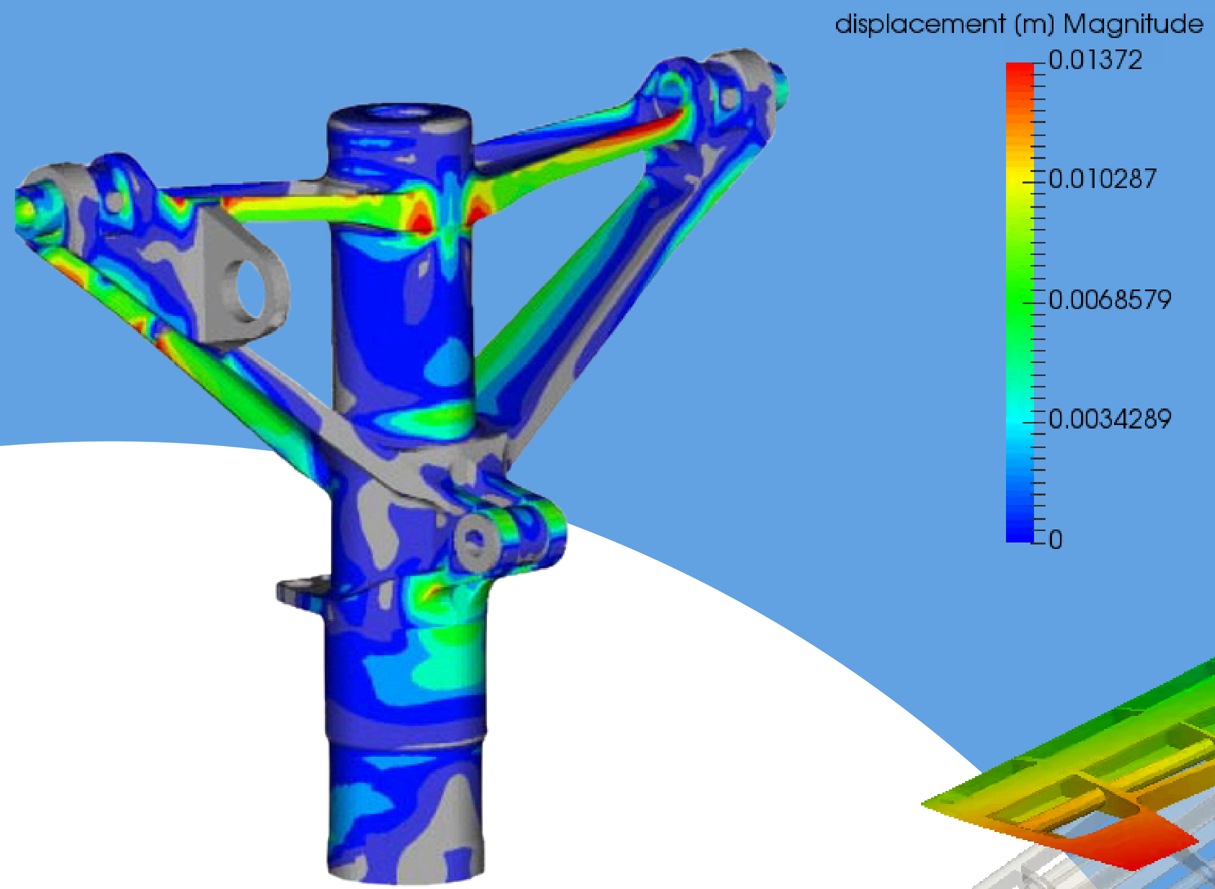


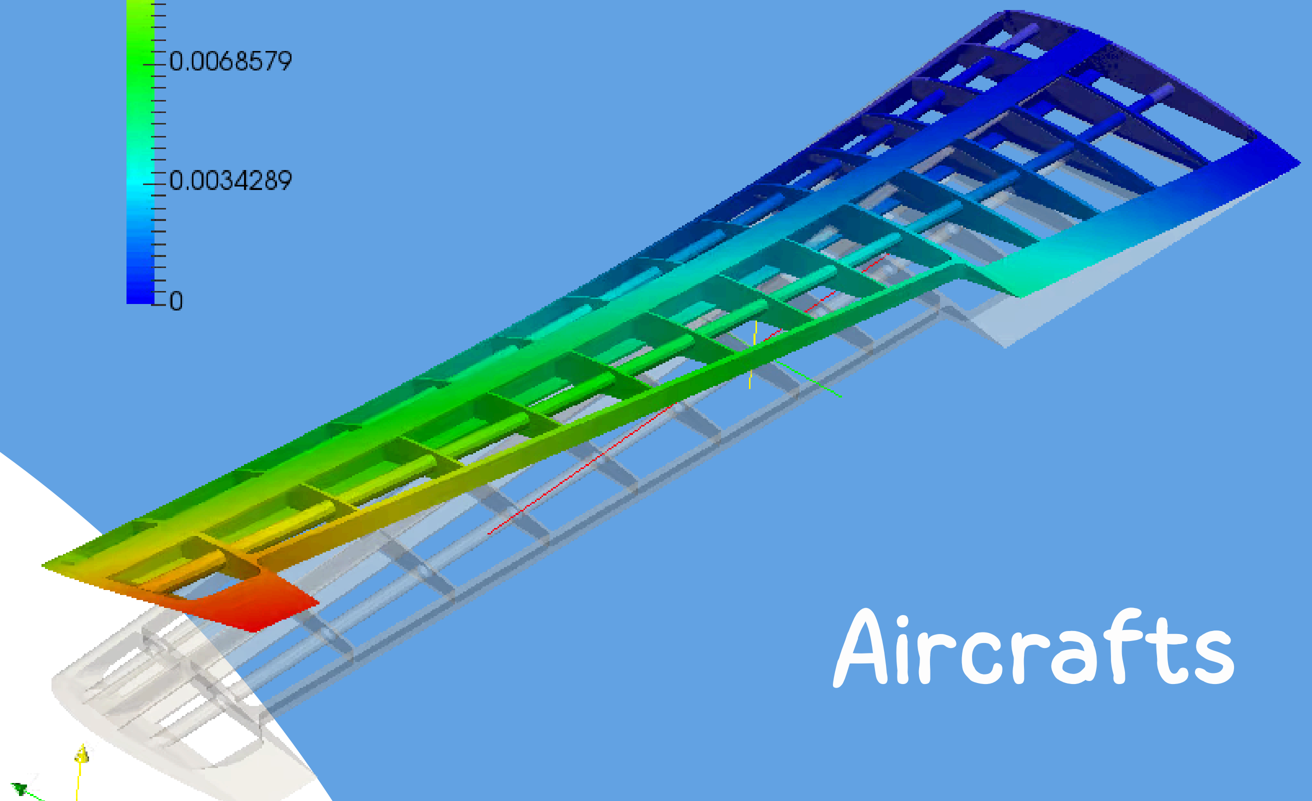


Health

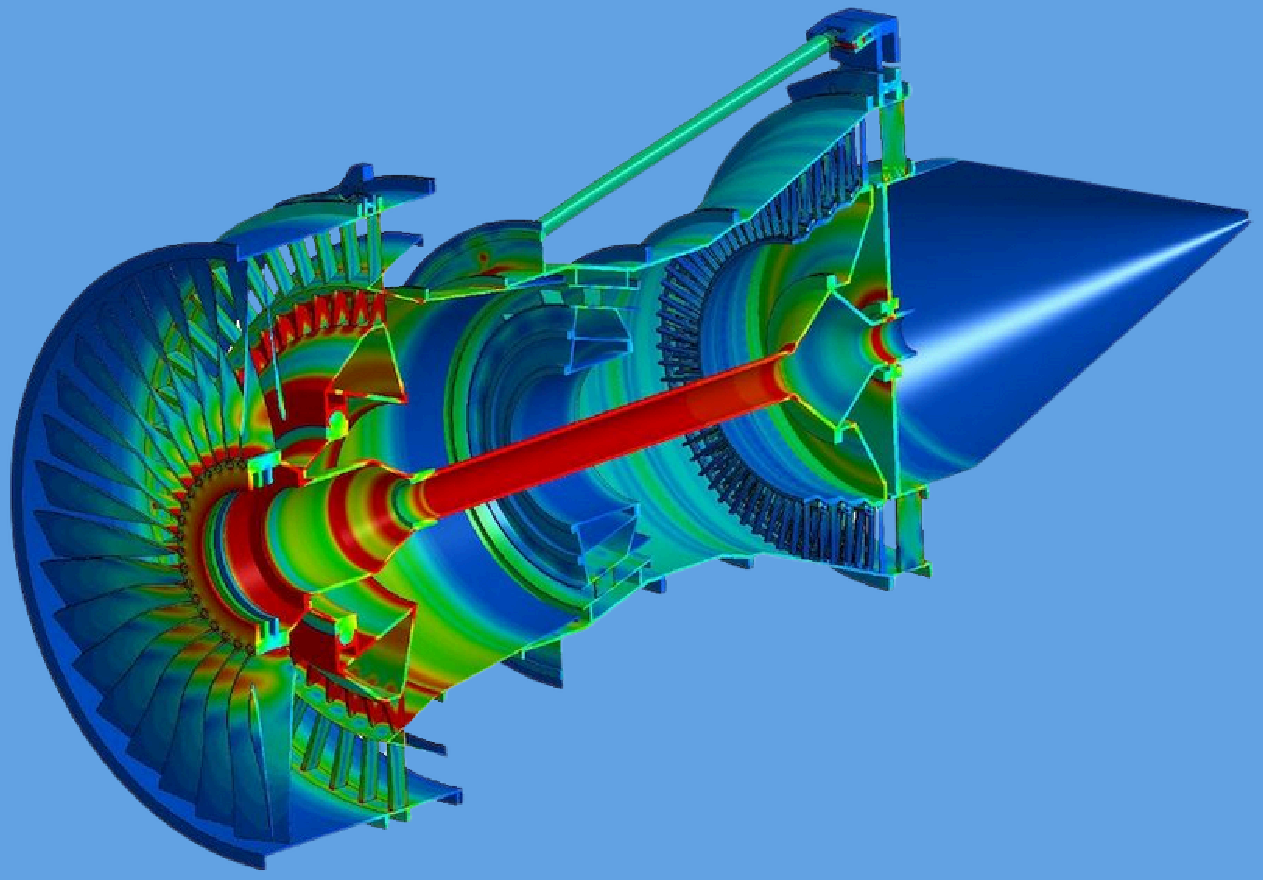


Sports



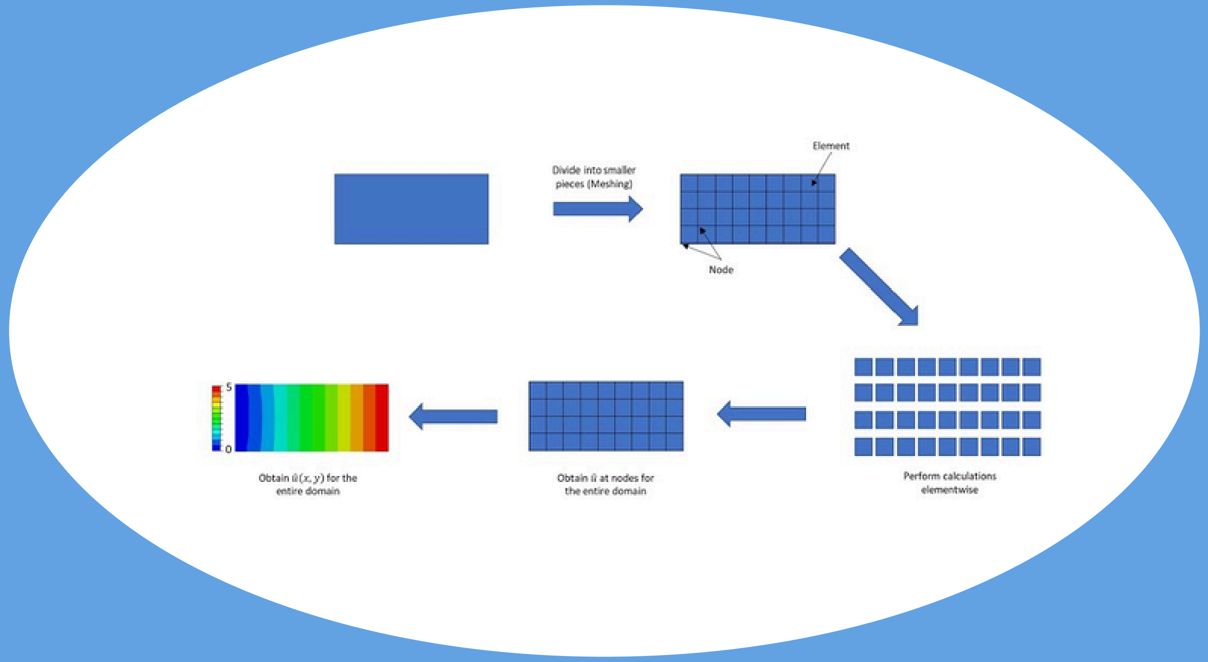


Aircrafts



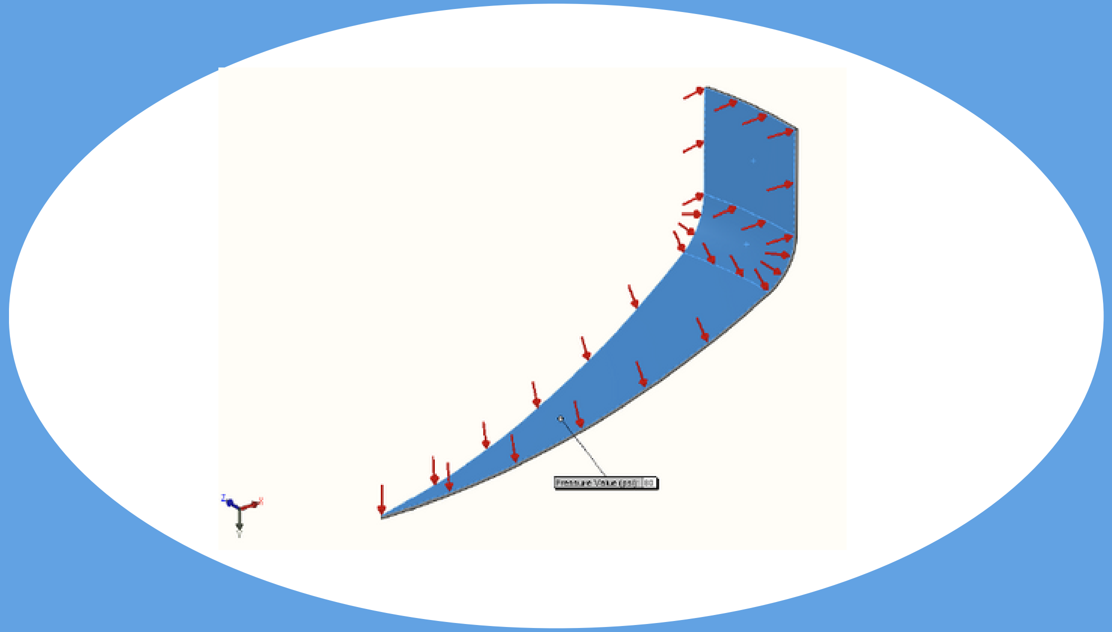
BASICS OF FEA

How does it work?



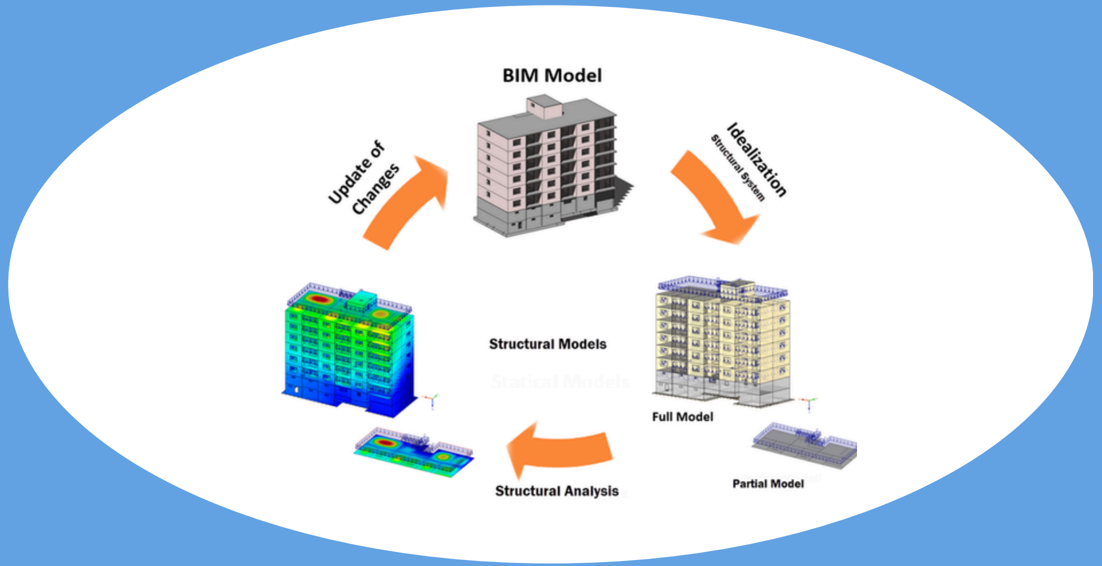
Think of it like breaking down a large, complicated structure into many small, simple pieces, which we call finite elements. These elements are connected at points called nodes, and together they form a mesh.

Process



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.

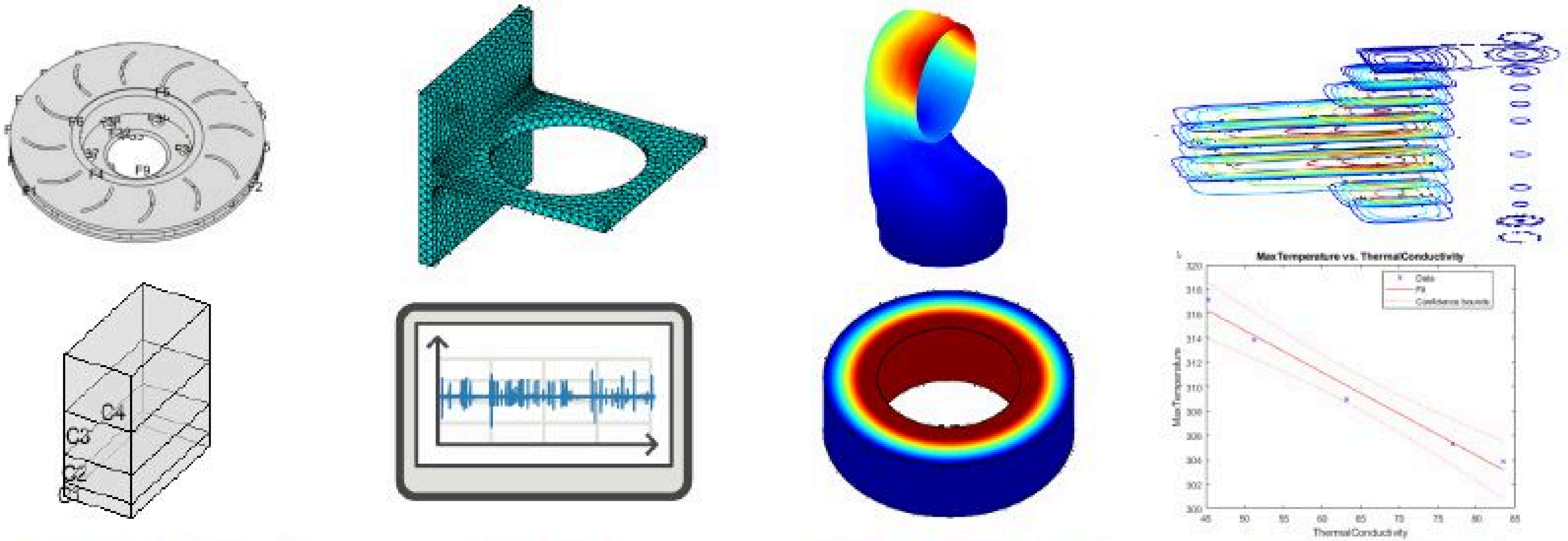
Advantages



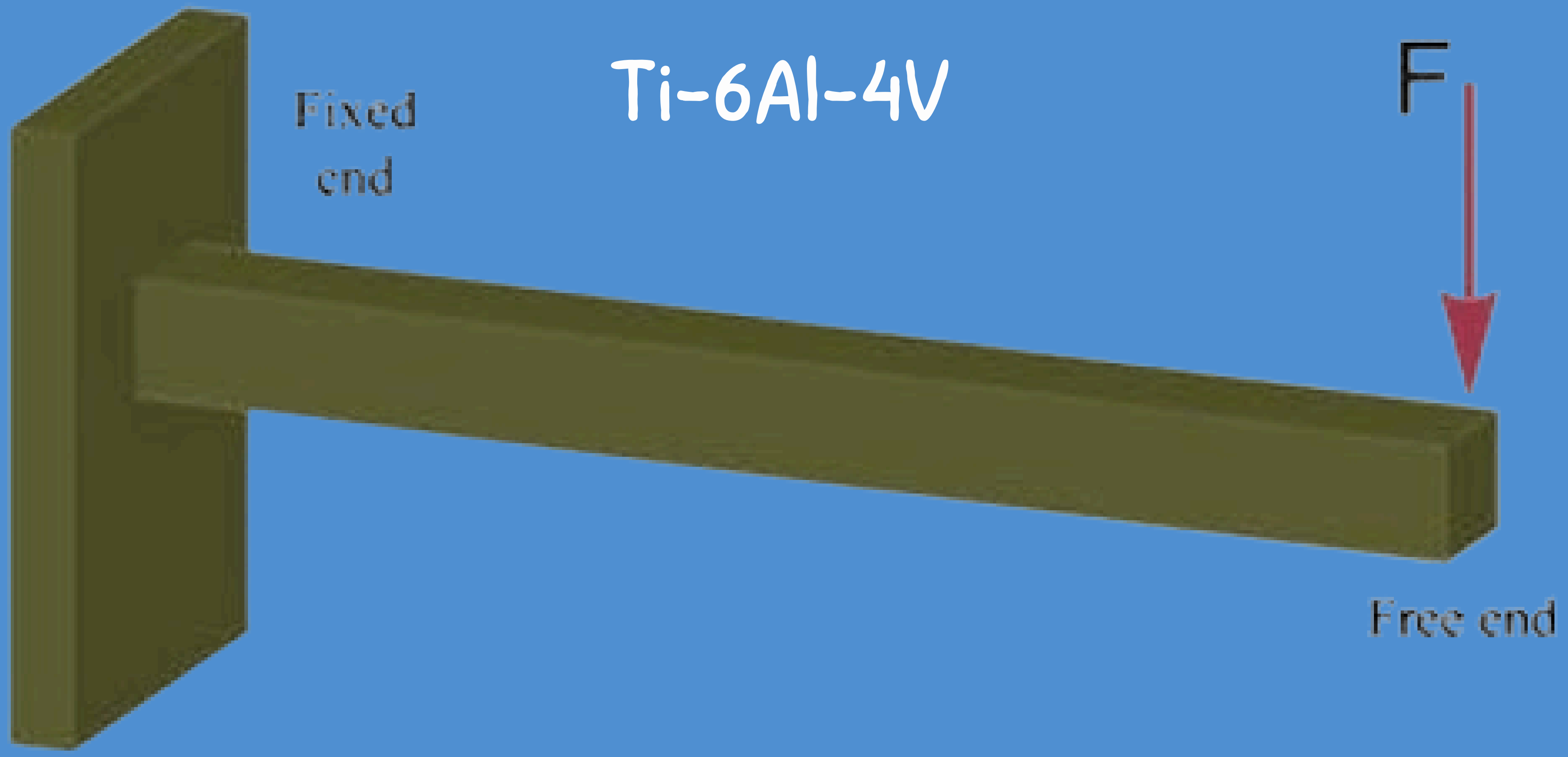
It saves time and money, provides detailed insight into material behavior, and allows us to test conditions that might be impossible in real life.

FINITE ELEMENT ANALYSIS PROCESS

1 Geometry | 2 Preprocess | 3 Solve | 4 Postprocess



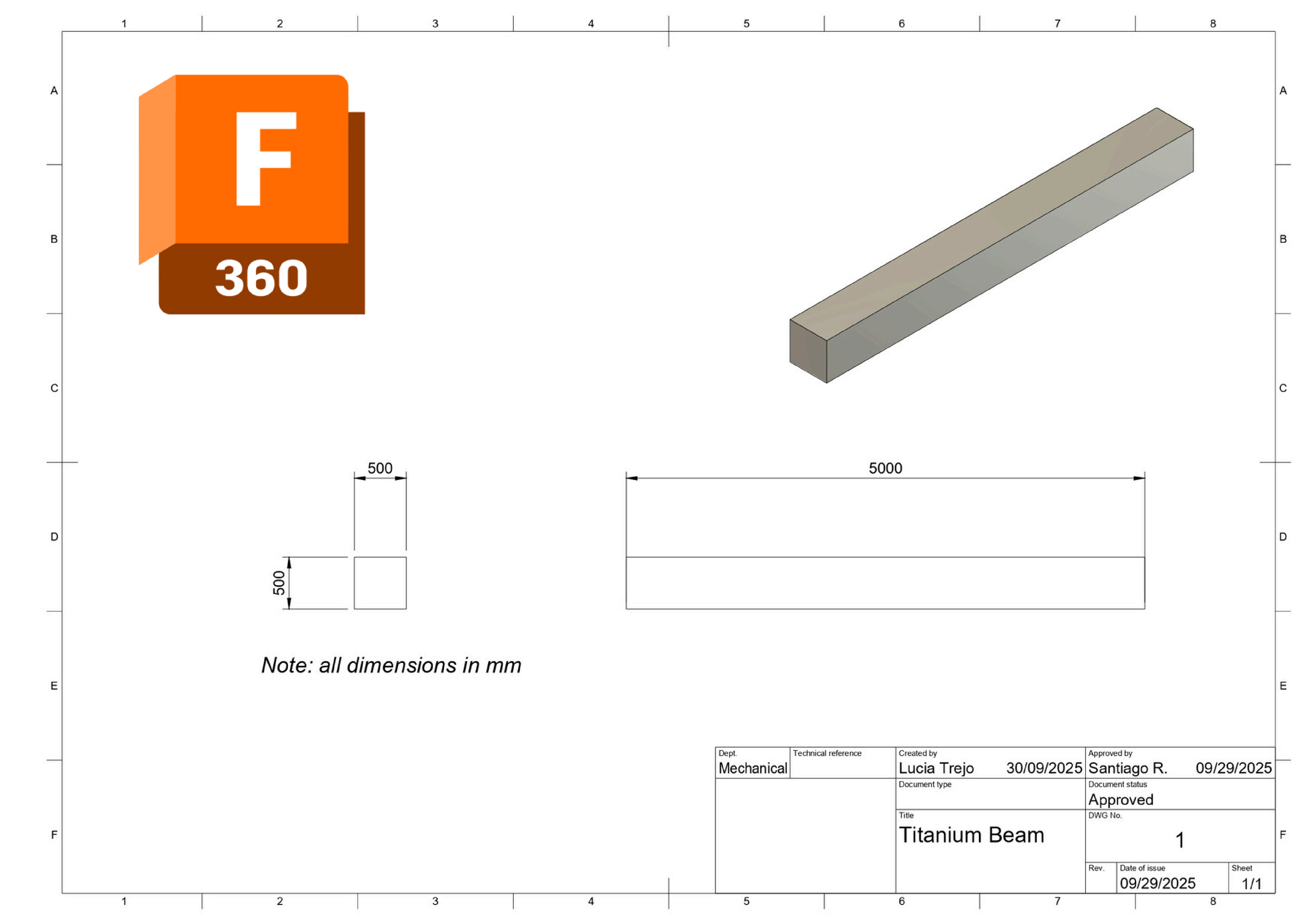
Import or Create Geometry	Mesh and Define Physics	Solve Engineering Problems	Visualize and Analyze Results

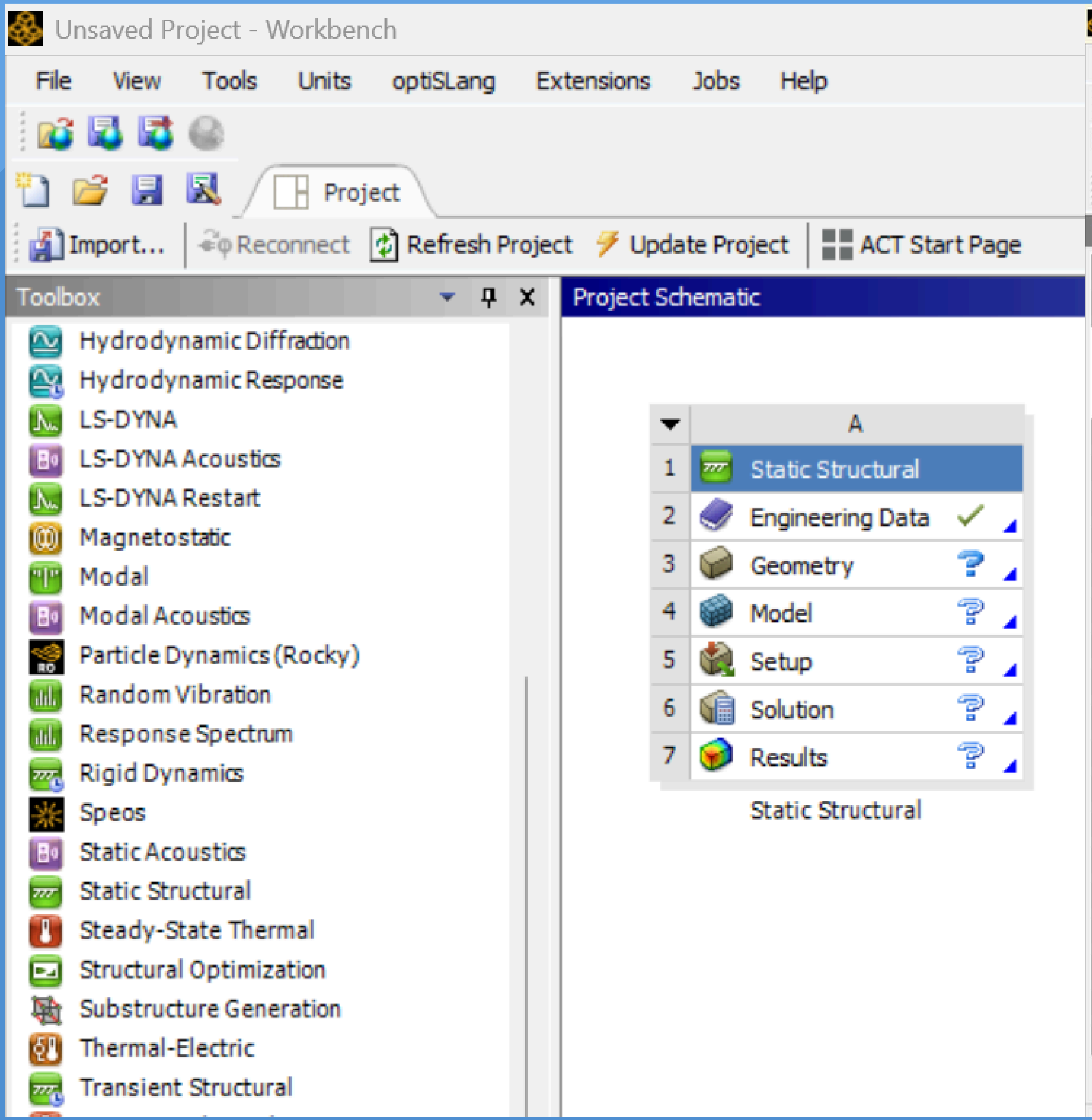


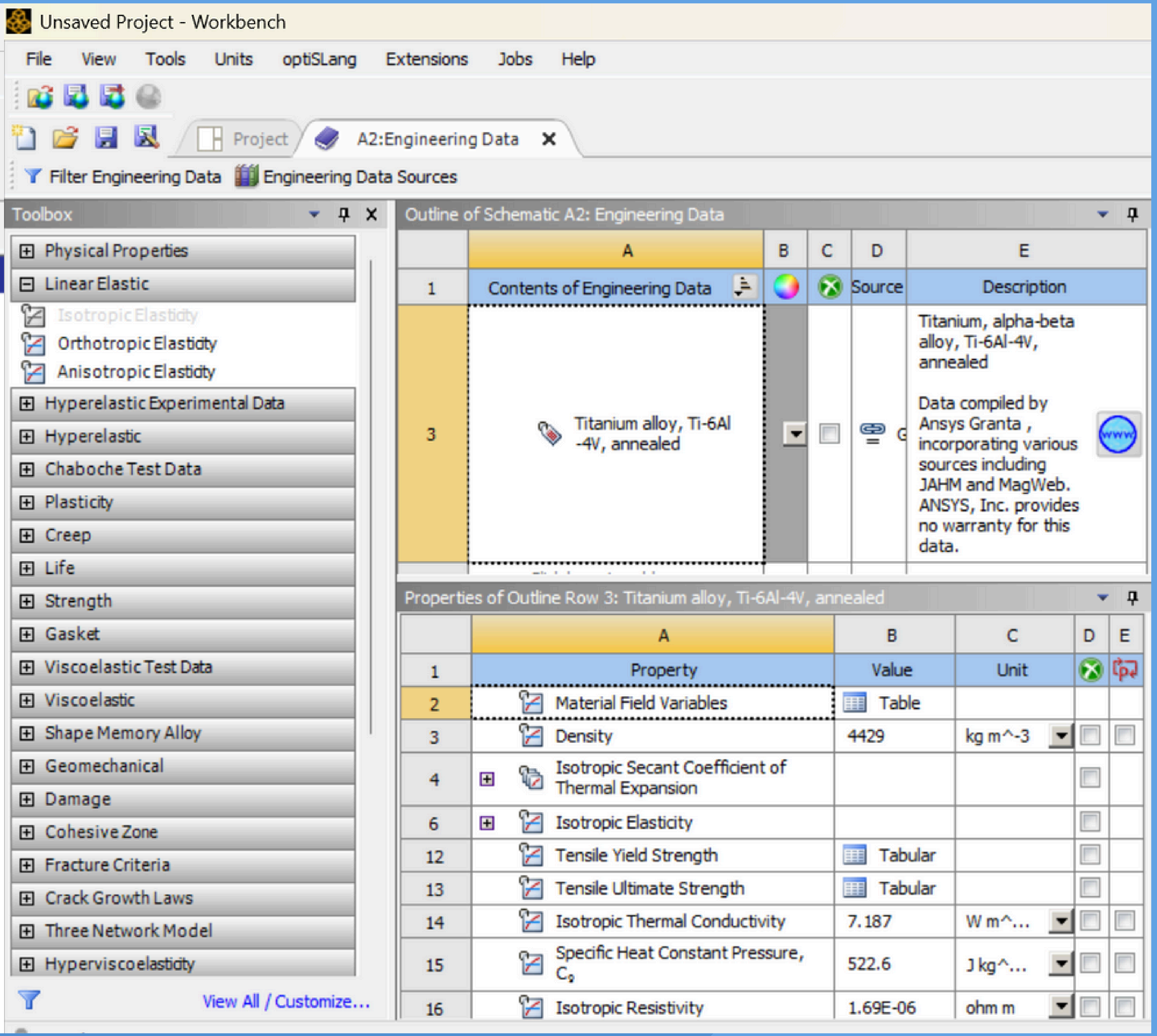
CASE OF STUDY EXAMPLE

Ansys

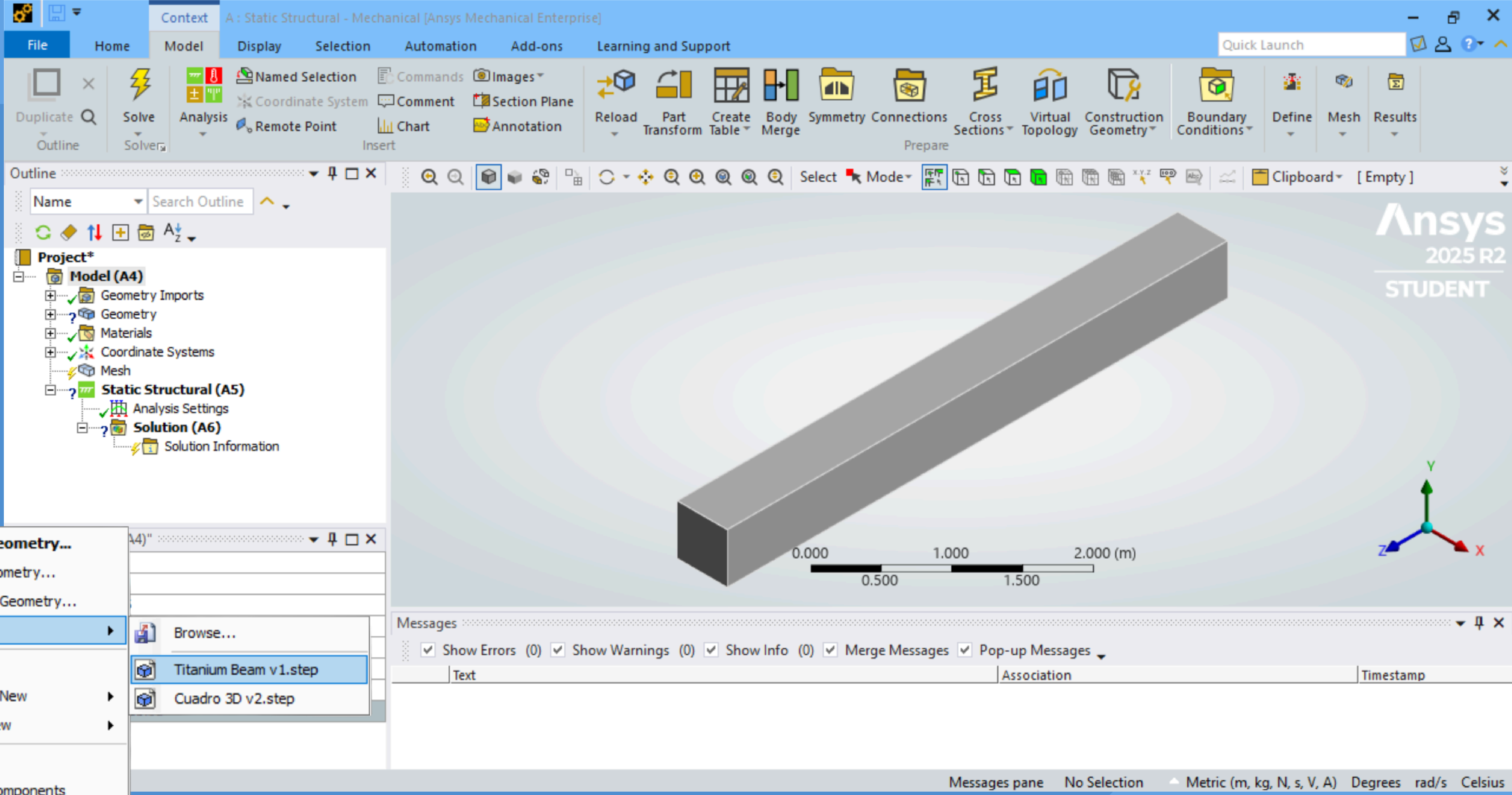
1. Model simple beam geometry

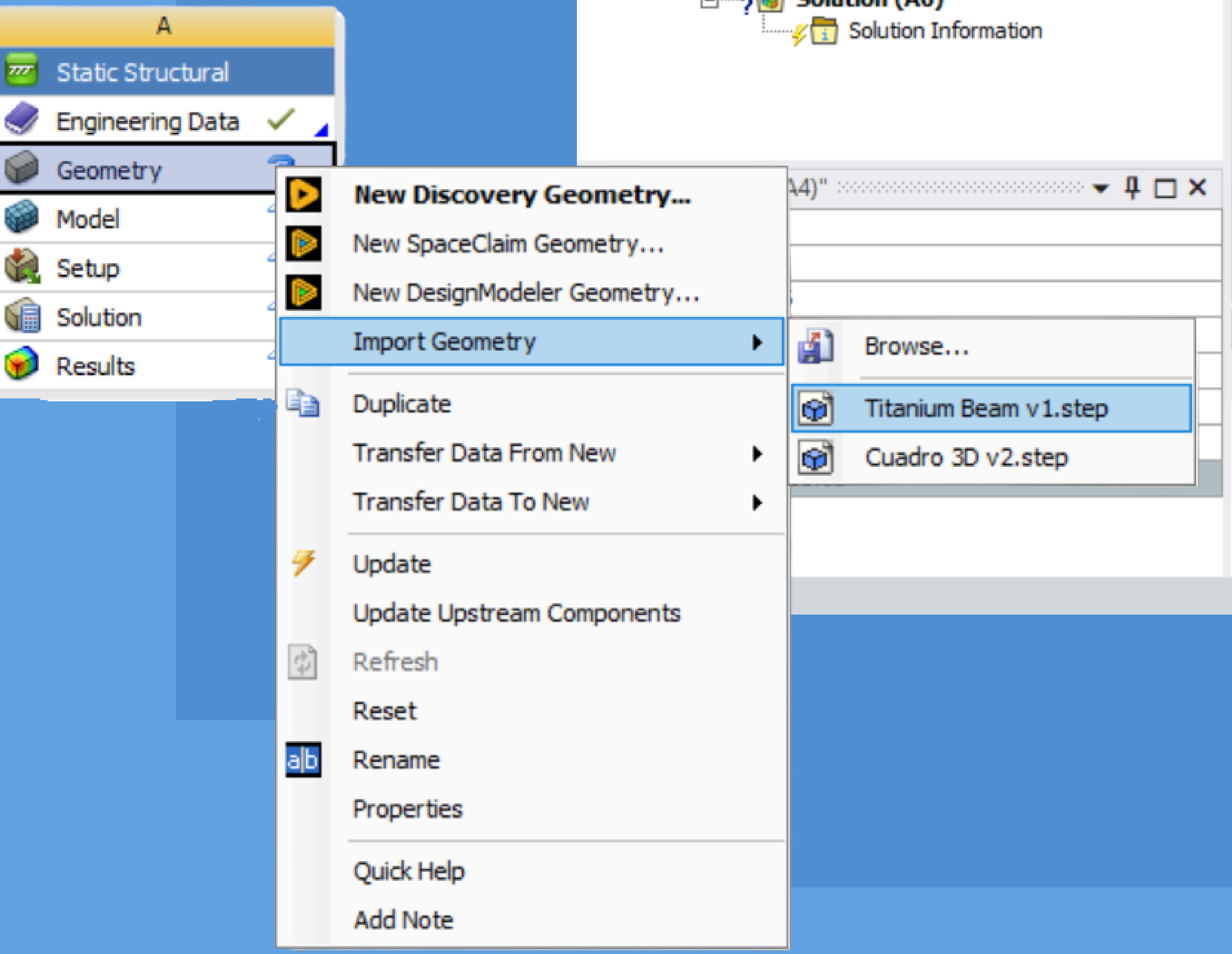




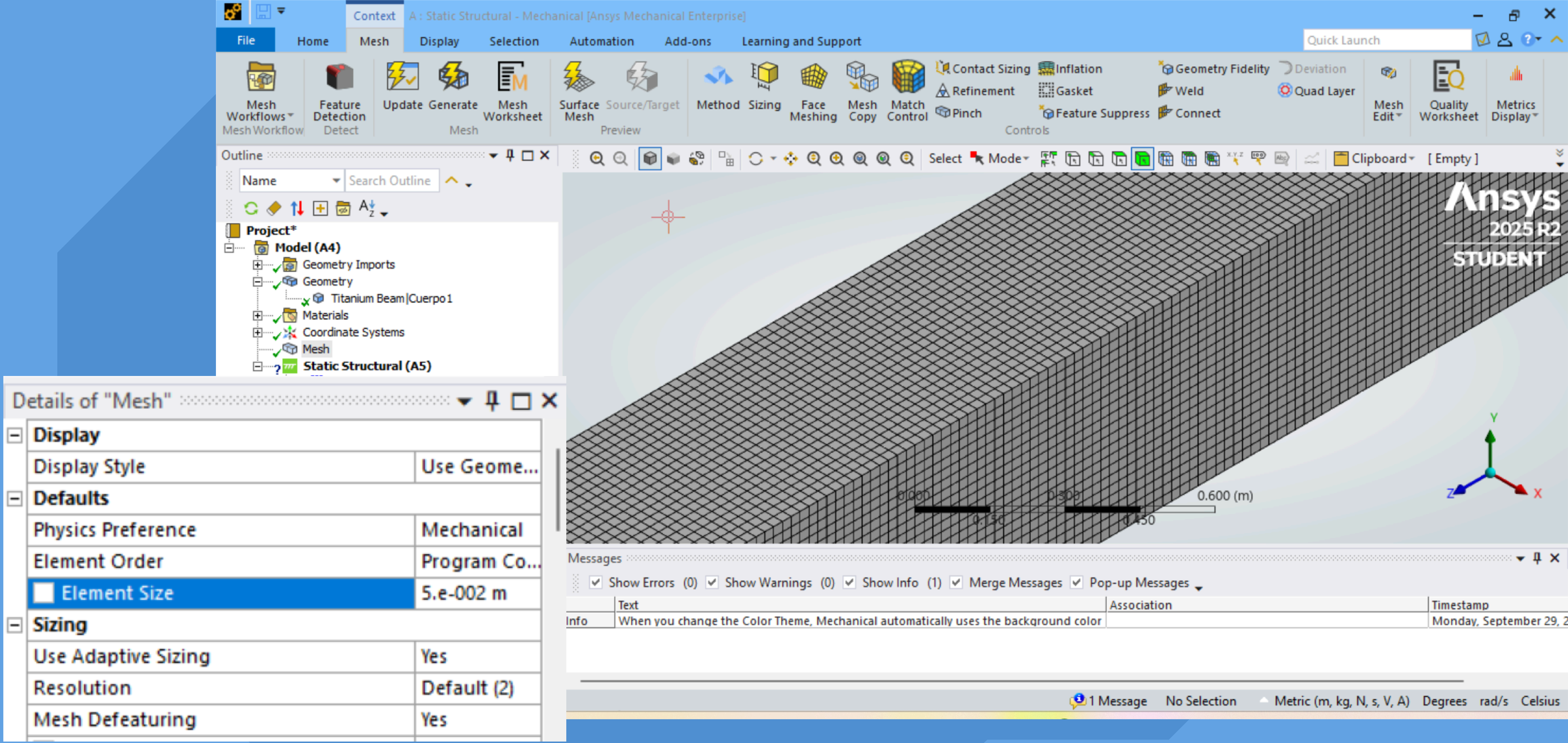


2. Define the material properties.





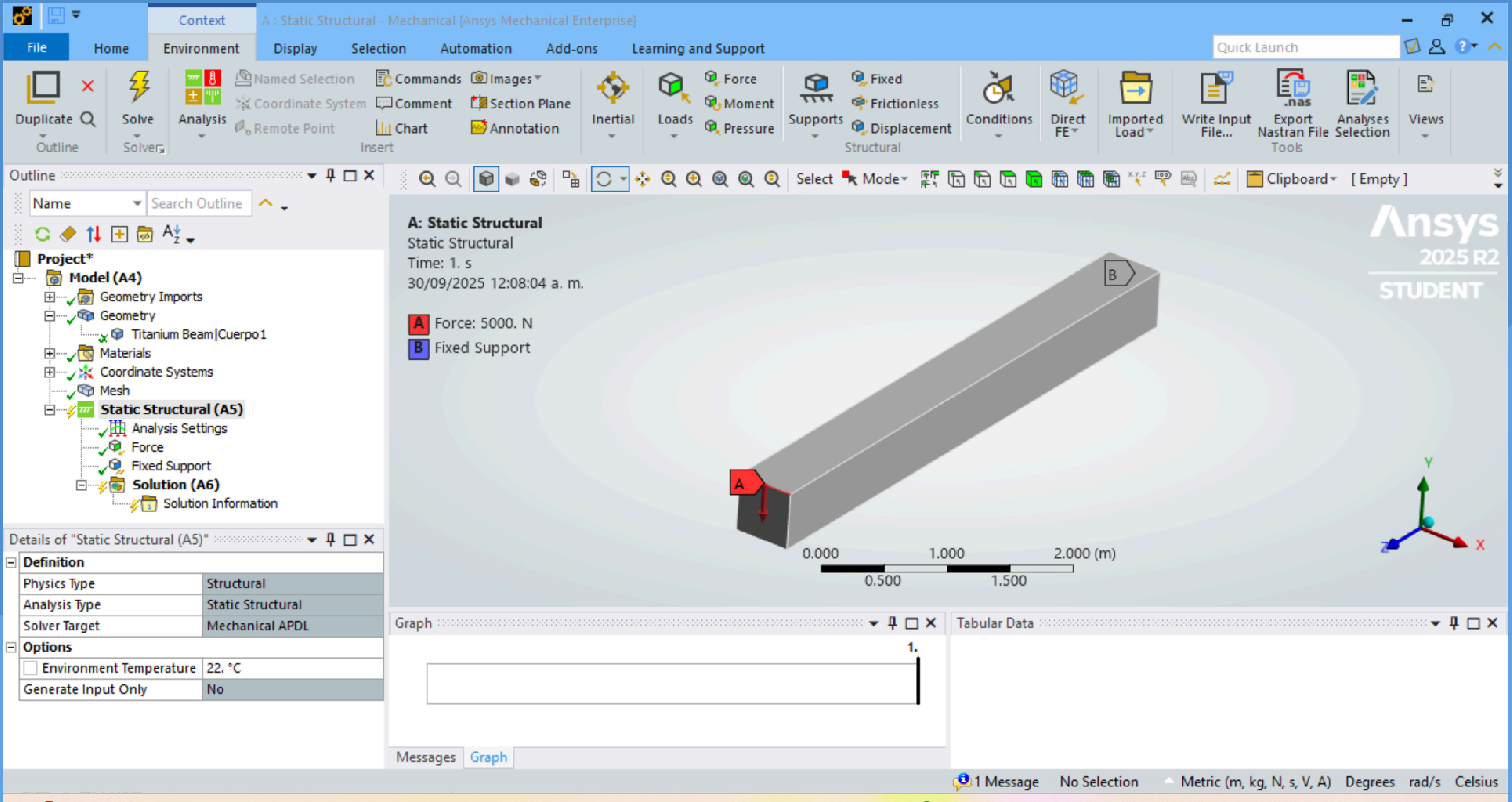
3. Import geometry to Mechanical

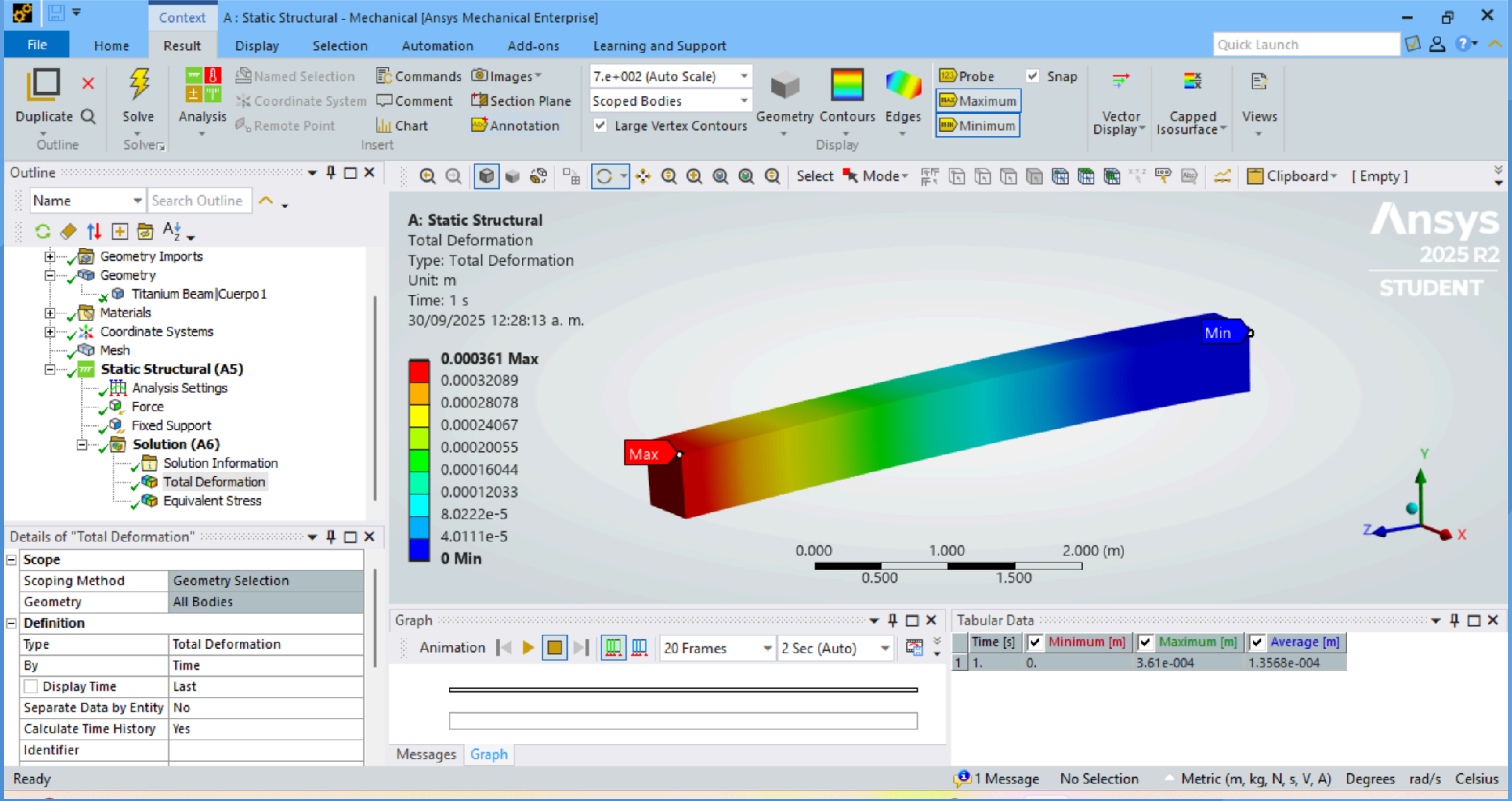


4. Generate a structured mesh

At least 8–12 elements across gauge thickness/width

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



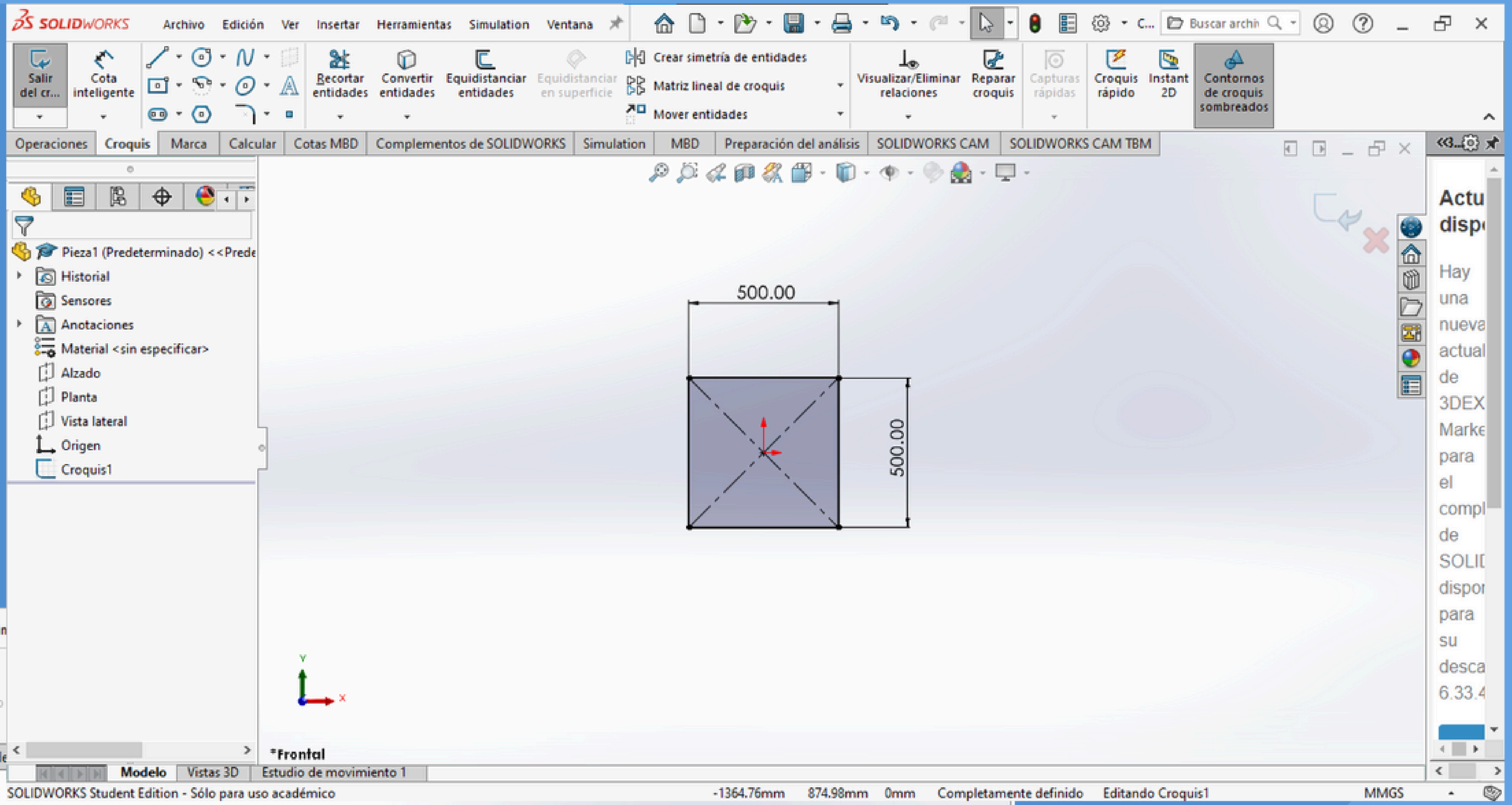


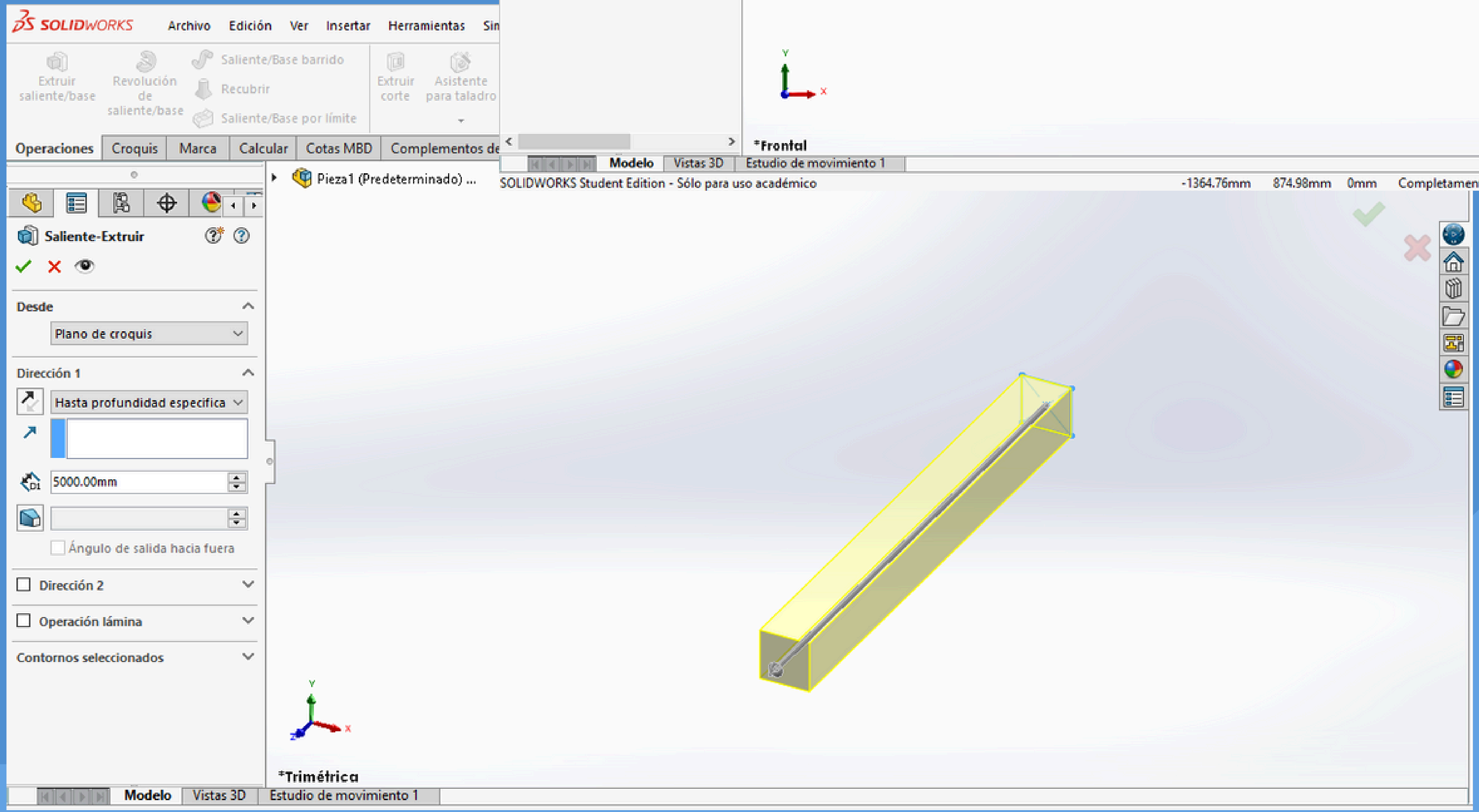
6. Solve the model.

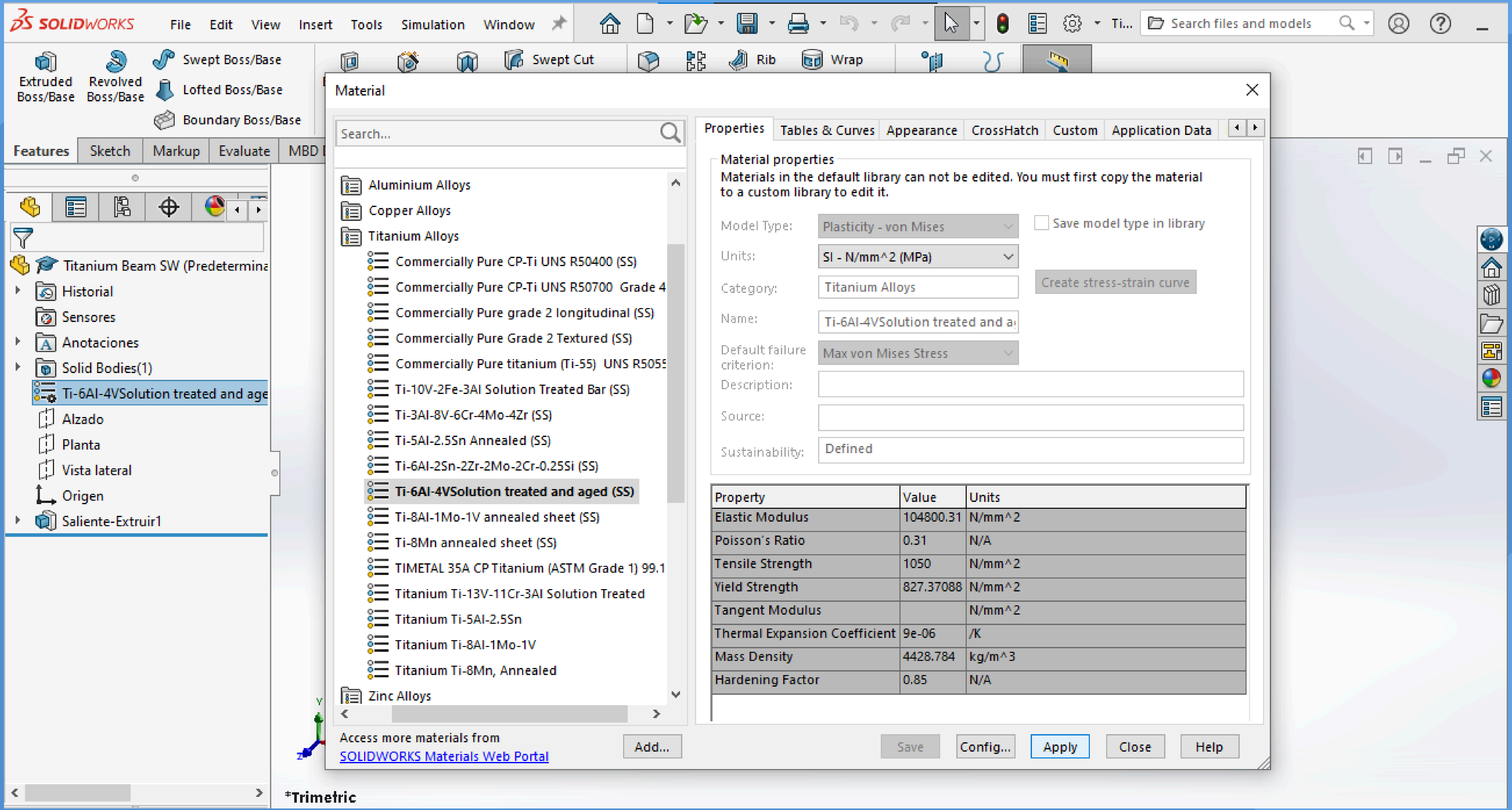
Extract Total Deformation, Equivalent (von Mises) Stress, and Strain results.

SOLIDWORKS

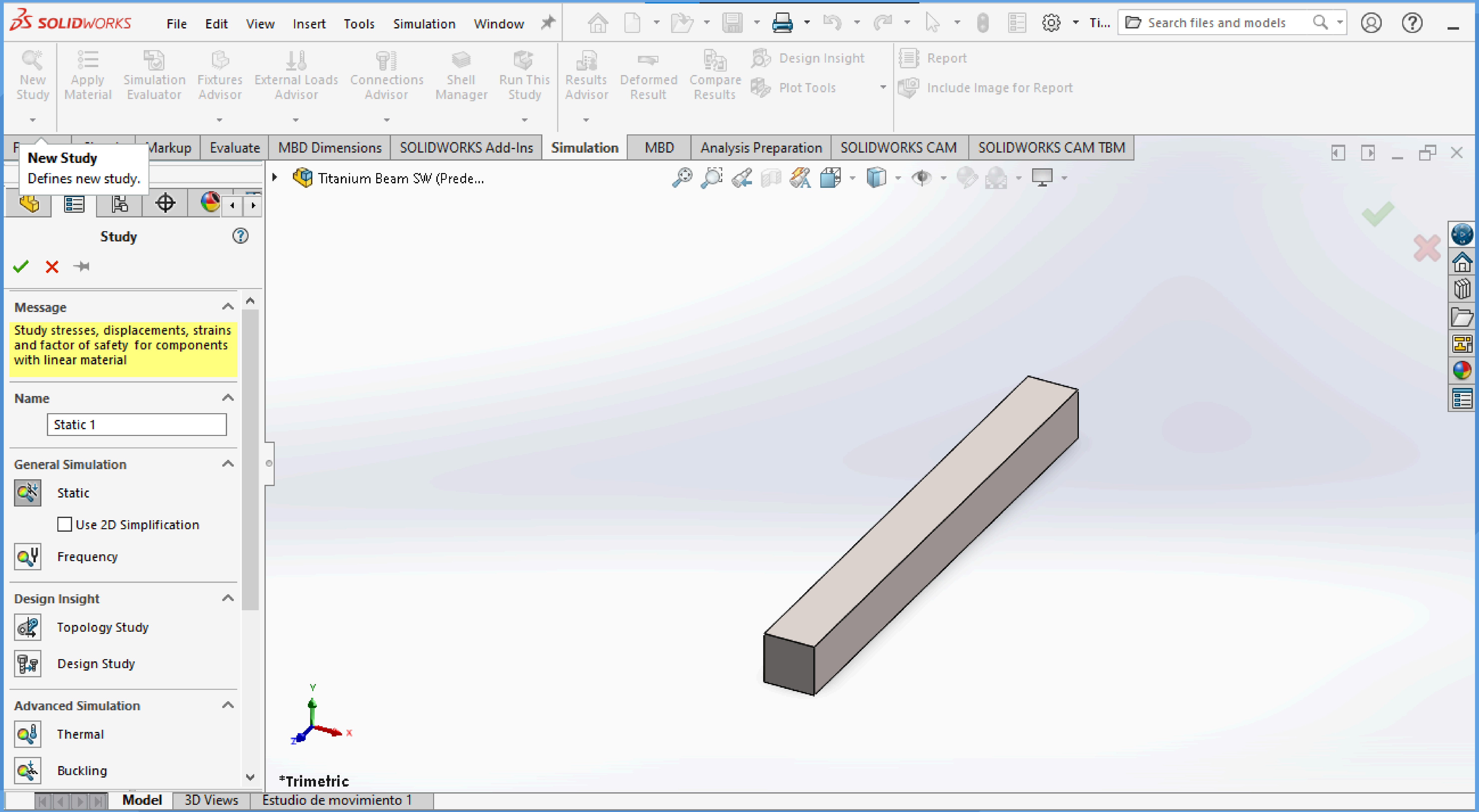
1. Model simple beam geometry





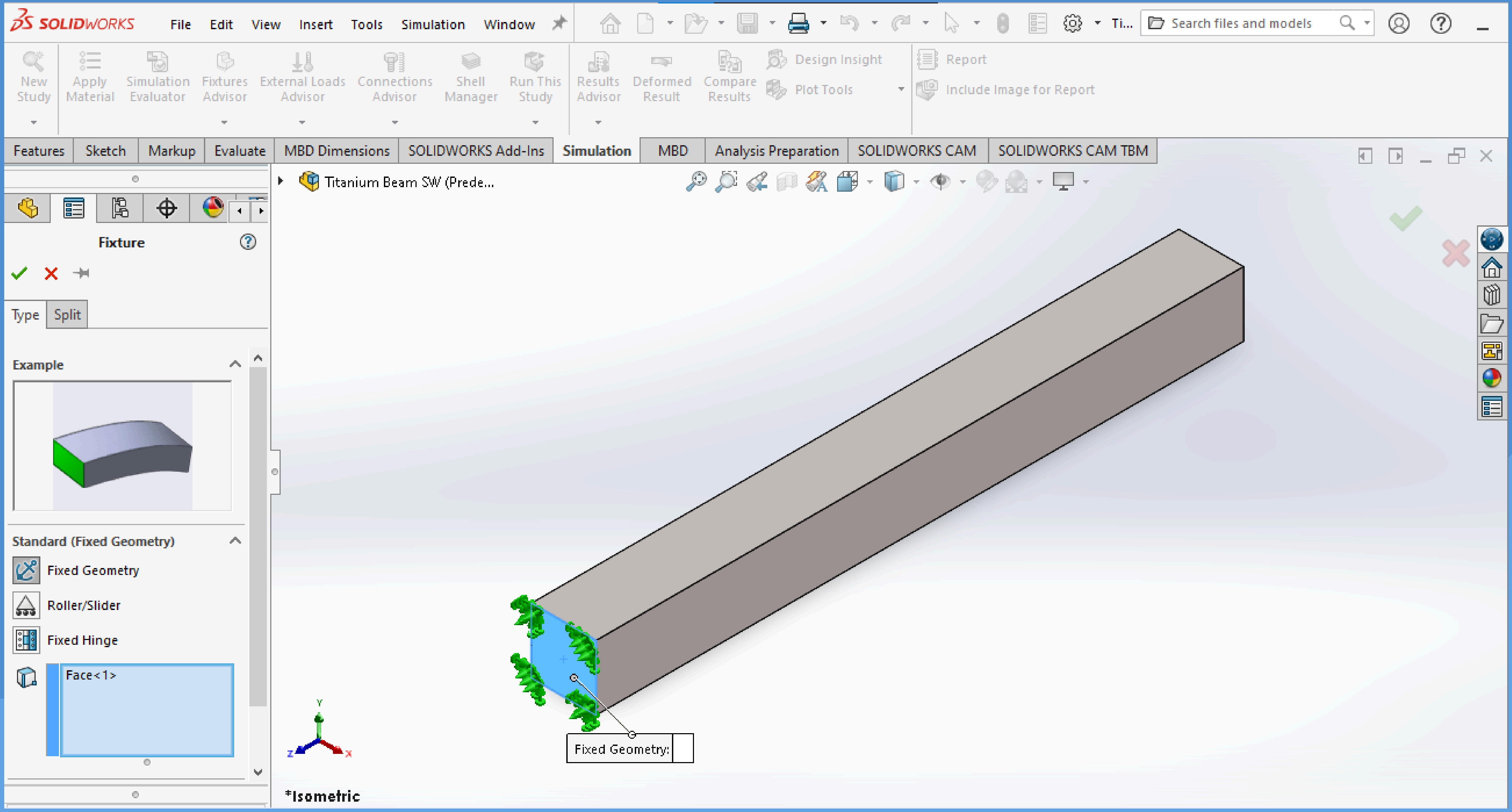


2. Define the material properties.

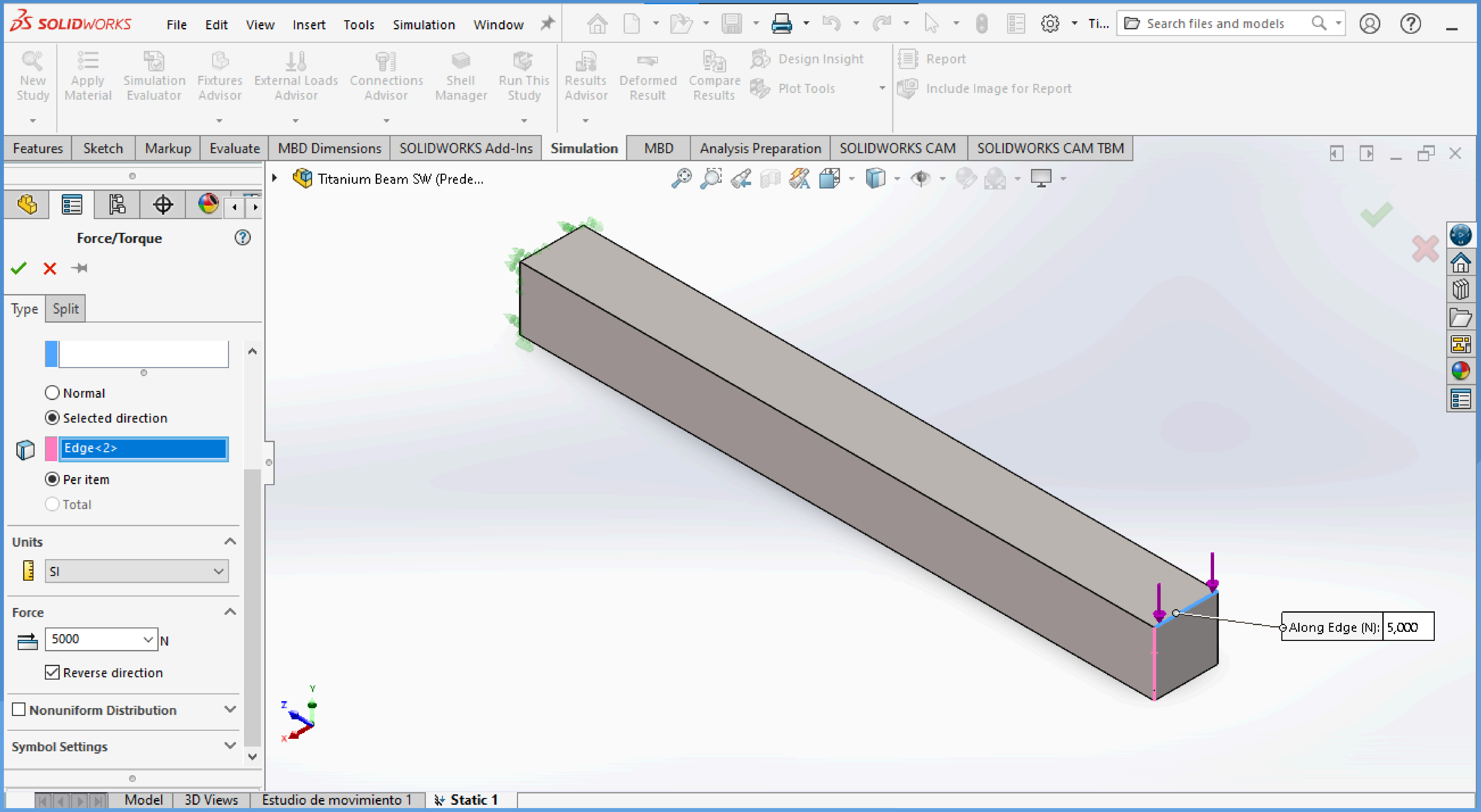


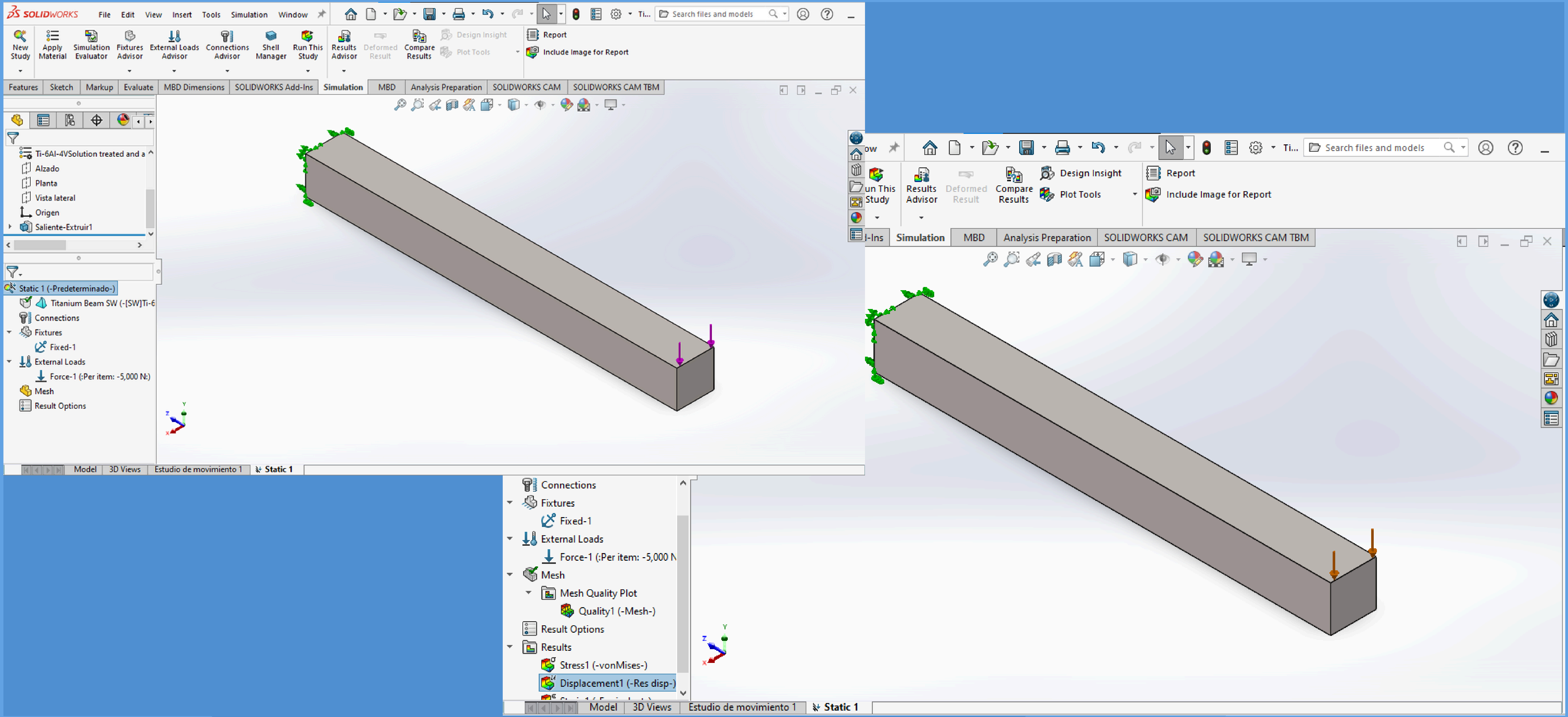
3. Start New Static Study

5. Apply Boundary Conditions: Fixed Support



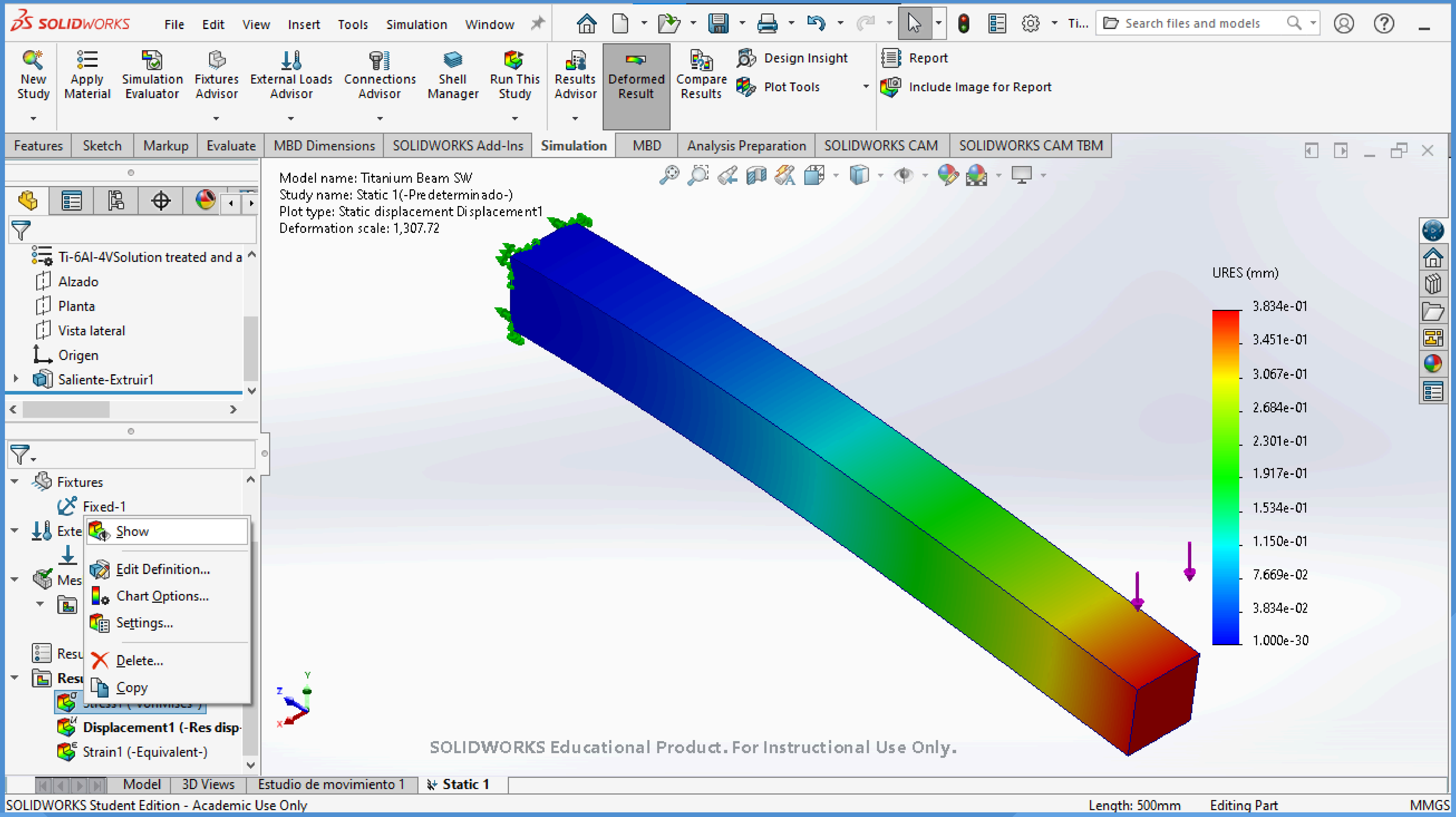
5. Apply Boundary Conditions: Force (F)





6. Solve the model.

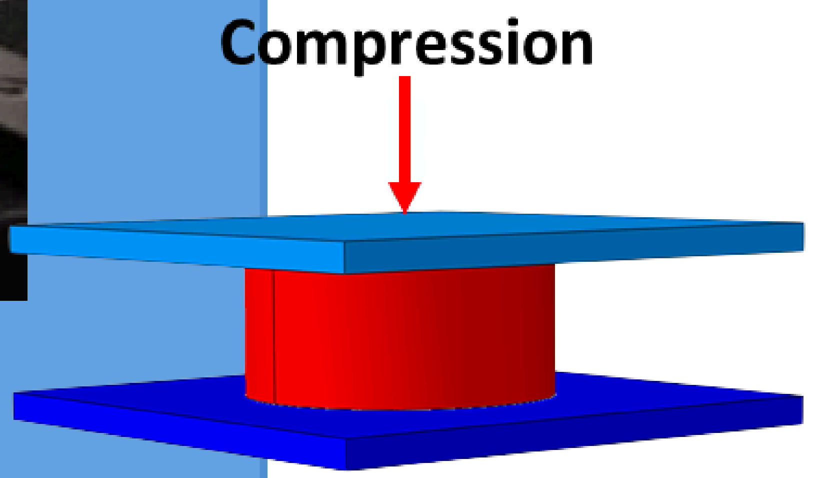
Extract Total Deformation, Equivalent (von Mises) Stress, and Strain results.

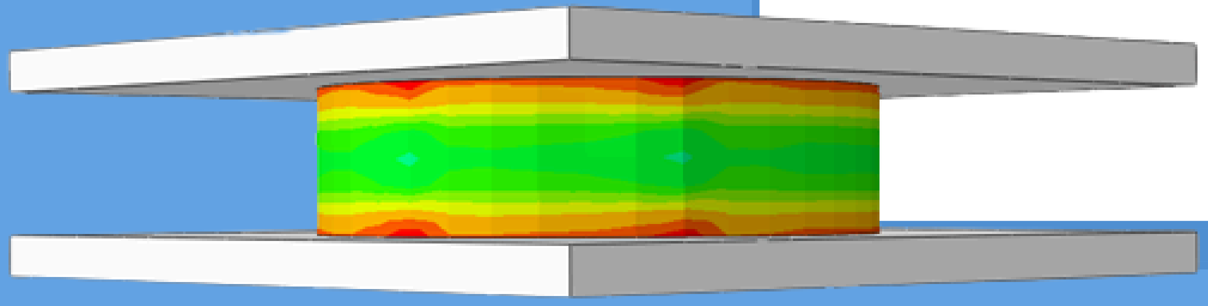


7. Displacement, Stress (von Mises), and Strain results.

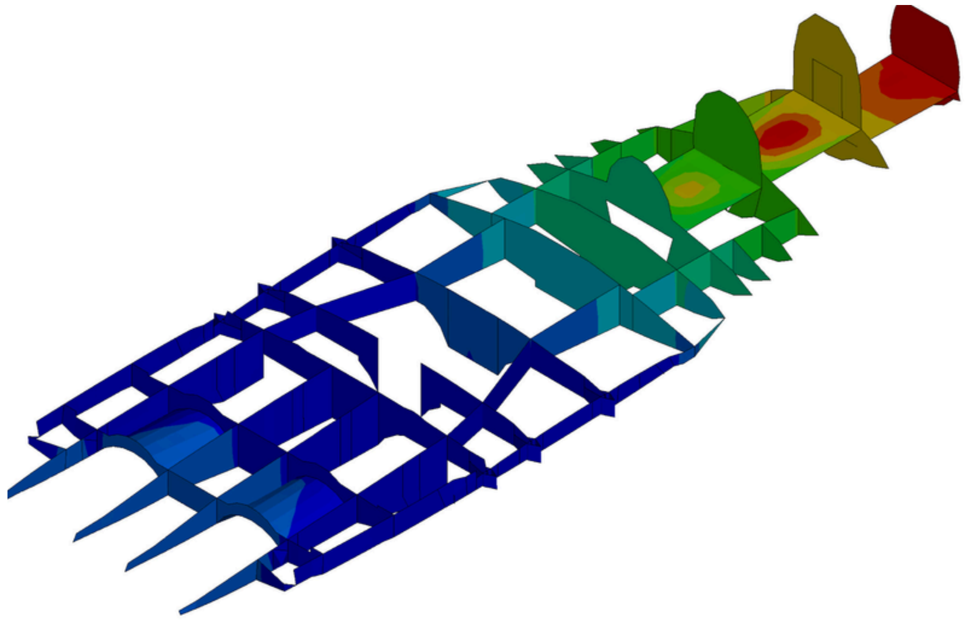
Why does this matter for materials characterization?

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.



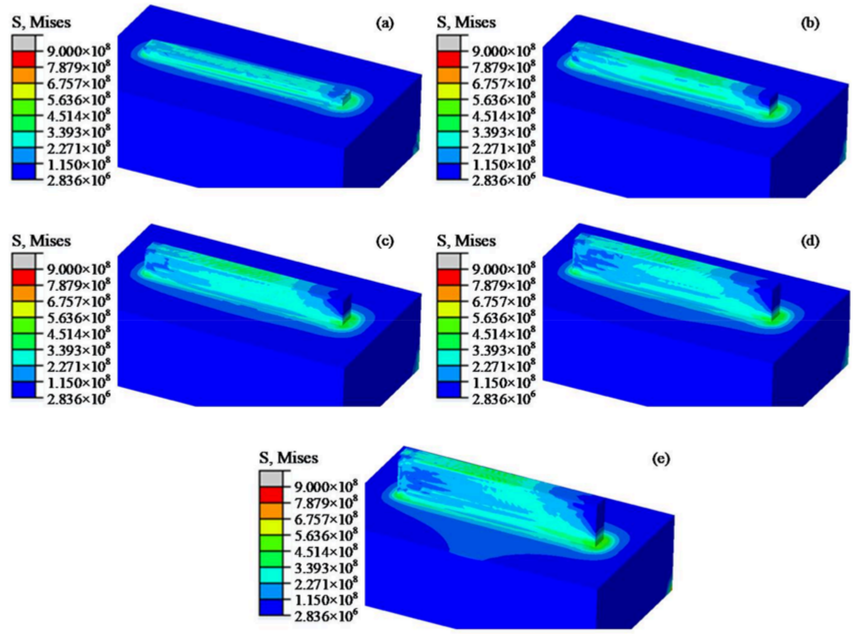


IMPORTANCE IN RESEARCH FIELD



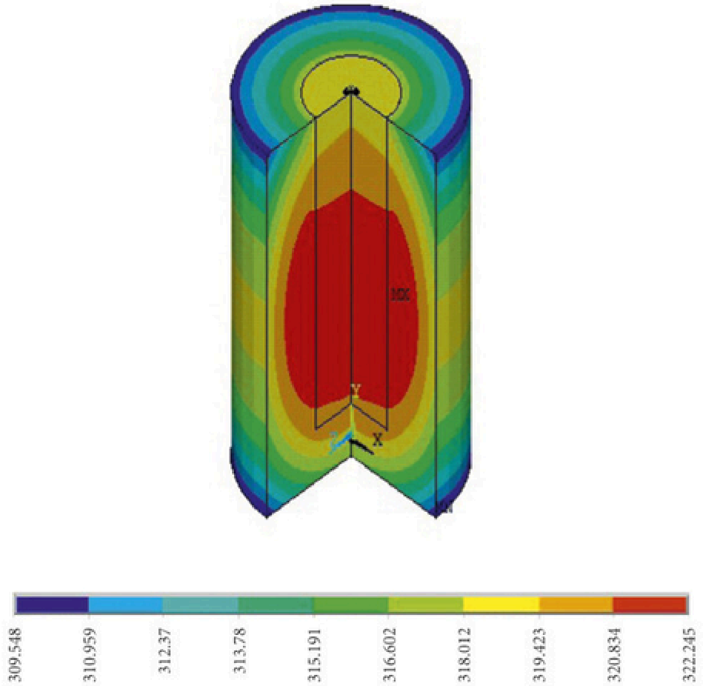
Aerospace

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



Advanced manufacture

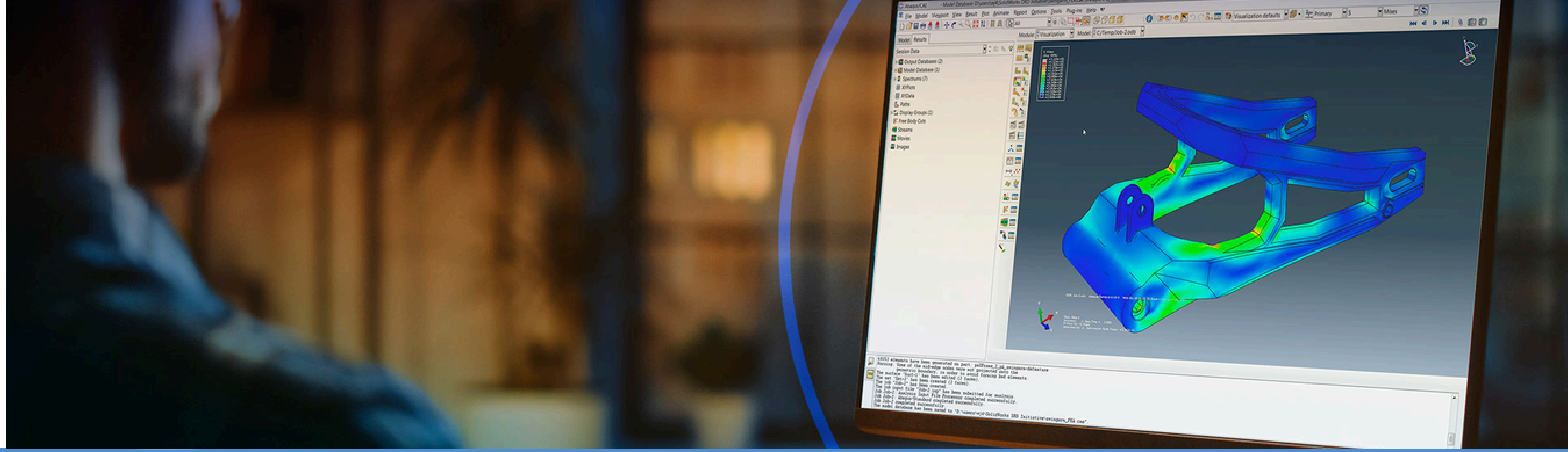
In additive manufacturing, it helps us predict how 3D-printed parts might show defects, residual stresses, and anisotropy ensuring better quality and consistency.



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

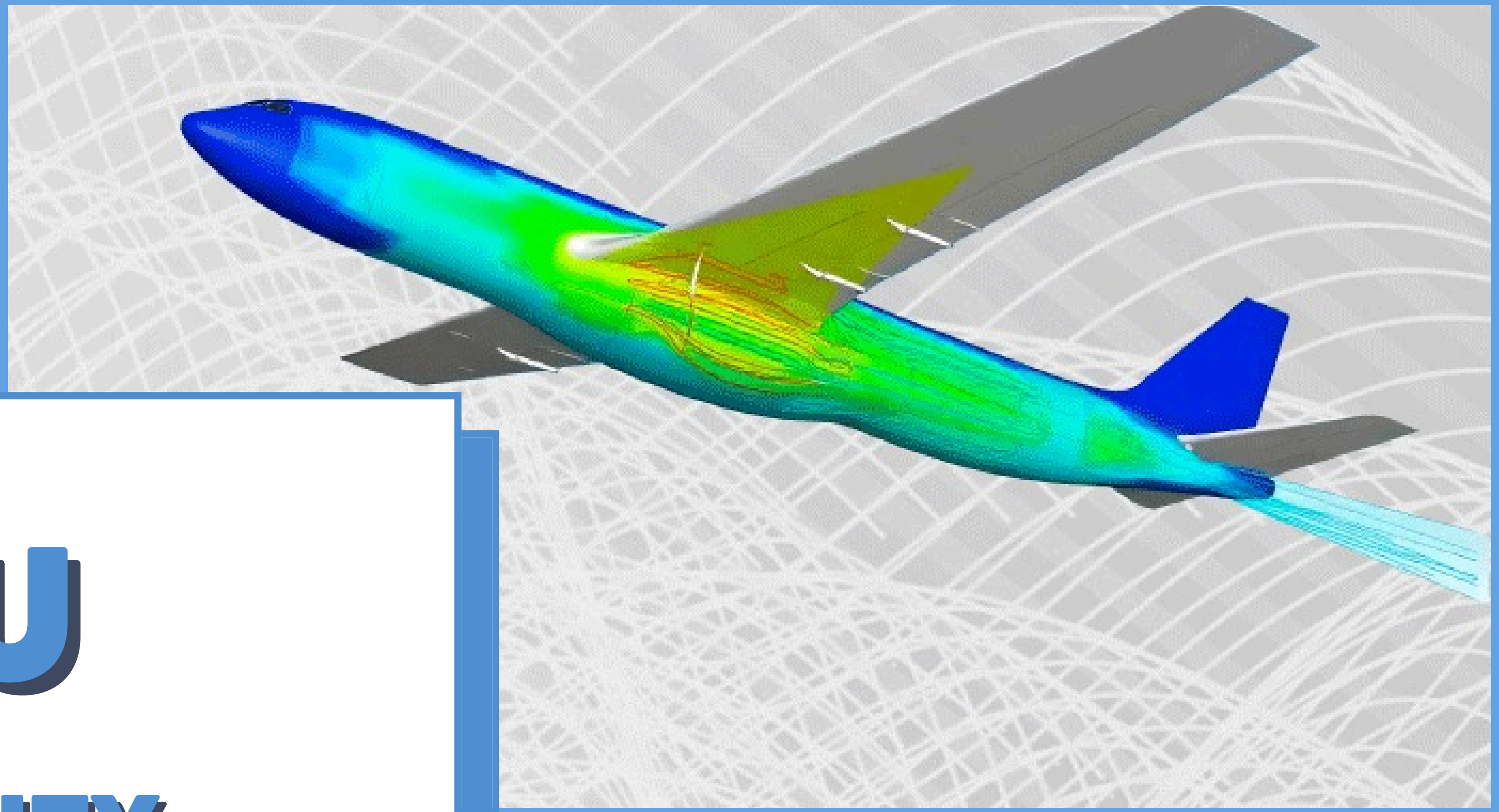
FEA is not just a simulation tool; it's a bridge between theory and experiment. By combining physical testing with numerical modeling, we can deepen our understanding of materials and make better engineering decisions.

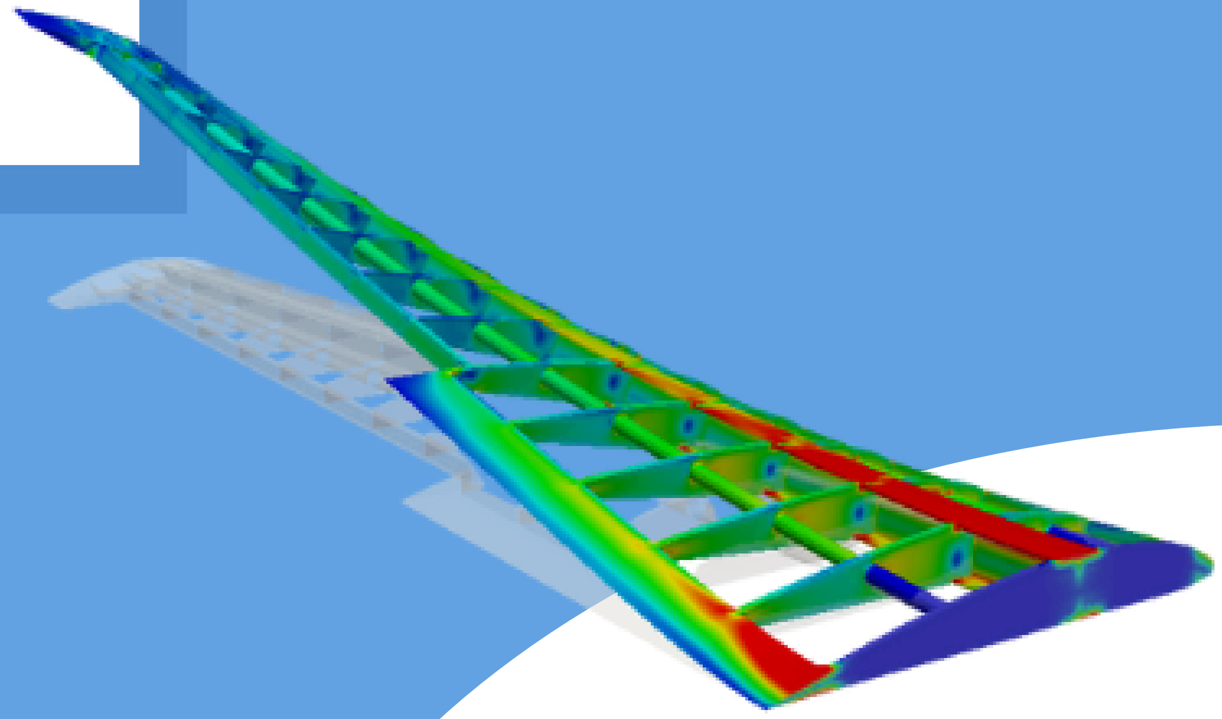
Implementation

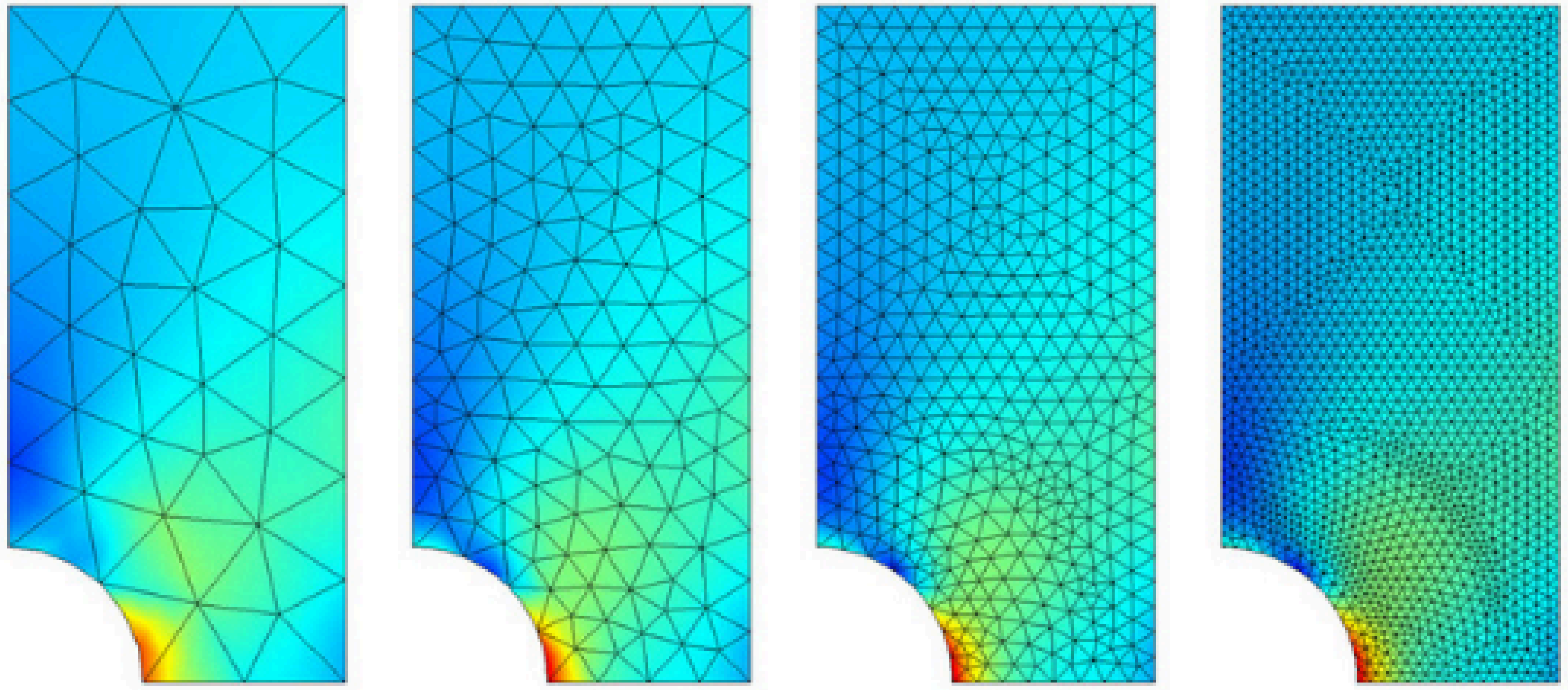
FEA does not replace experiments, but complements them. Together, simulation and experimental data provide a deeper and more reliable understanding of materials.

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.

Moatamedi, 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-is-fea-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.