UNIVERSITA' degli STUDI di ROMA
TOR VERGATA

MULTIPHYSICS SIMULATION BY THE FINITE ELEMENT METHOD

Luca Marinangeli

Supervisor : Prof. Giuseppe Tomassetti

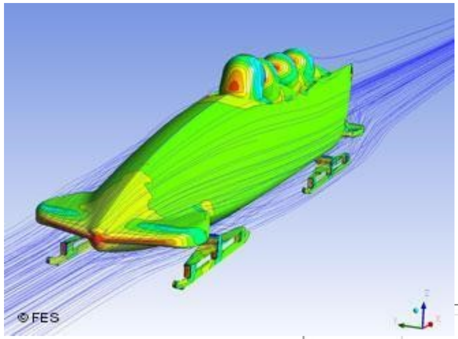
Bachelor's Degree in Engineering Sciences
University of Rome Tor Vergata

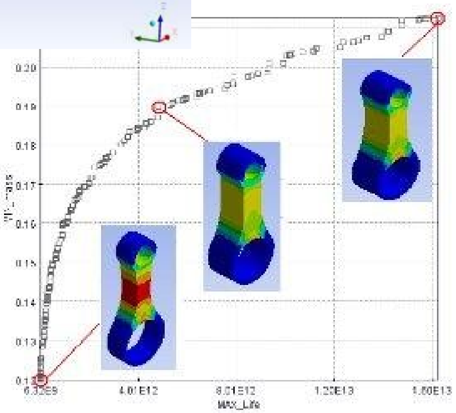
A.Y. 2013/2014

Context

Computer Simulation has become an essential part of science and engineering for :

- Developing new products
- Optimizing designs
- Describing physical phenomena



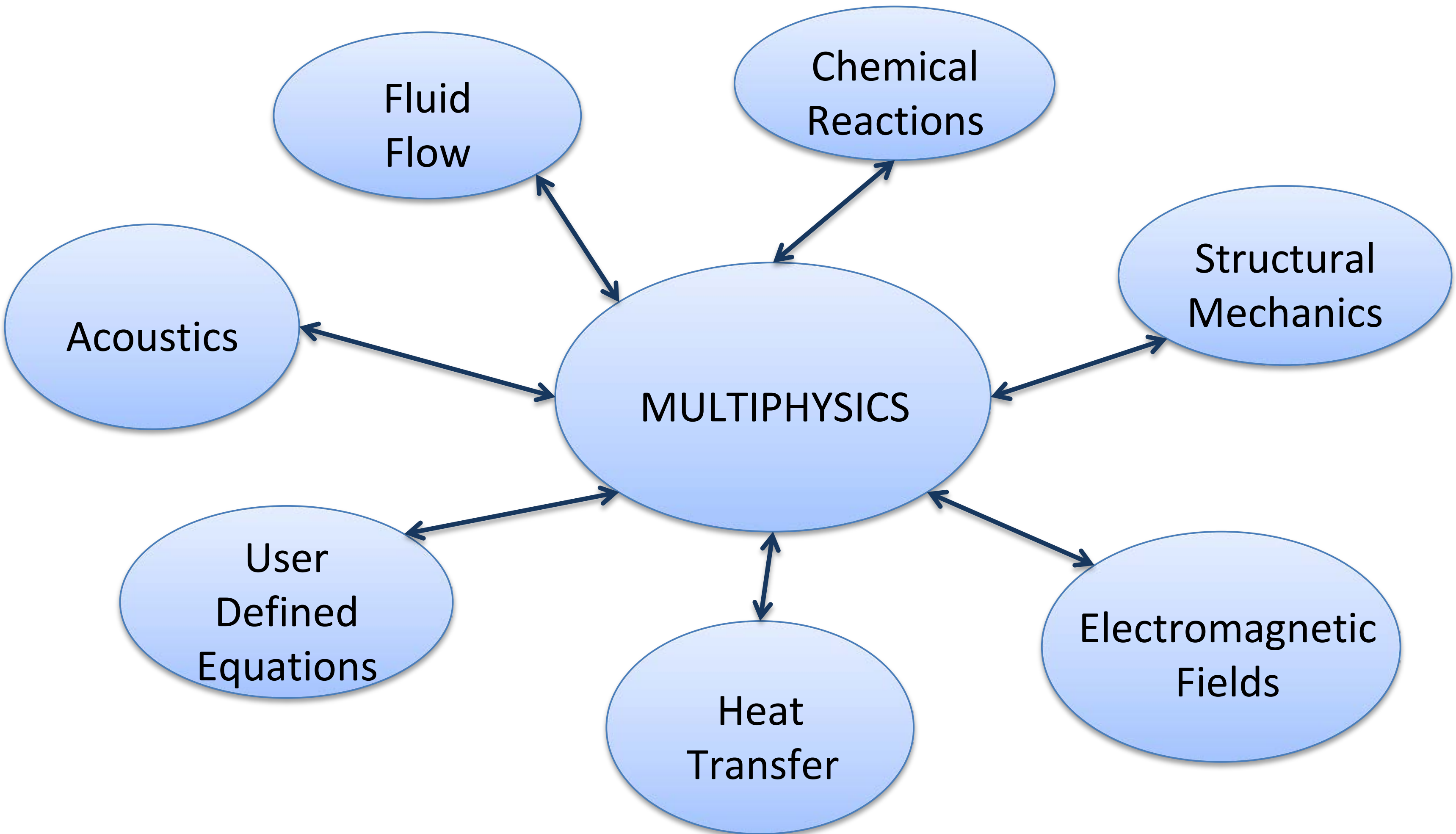


But a common concern is represented by :

- Modelling **Accuracy**
- Results **Reliability**

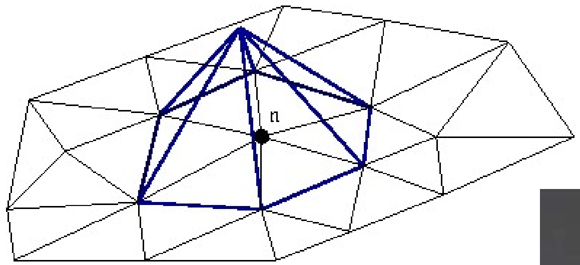
Multiphysics Simulations

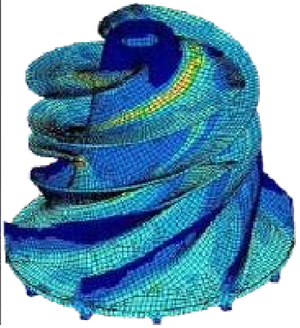
Multiphysics treats simulations that involve multiple physical models or multiple simultaneous physical phenomena

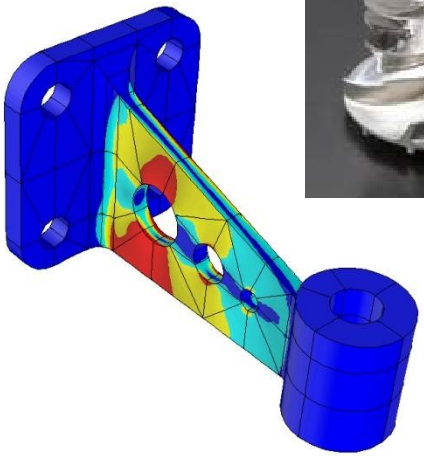


Finite Element Method (FEM)

The **Finite Element Method** (FEM) is a numerical approximation technique that divides a component into discrete regions (the finite elements) where the response of a given physical problem is evaluated.







FEM requires :

- Formulation

- Solution Process
- Material Representation
- Geometry
- Boundary Conditions and Loadings

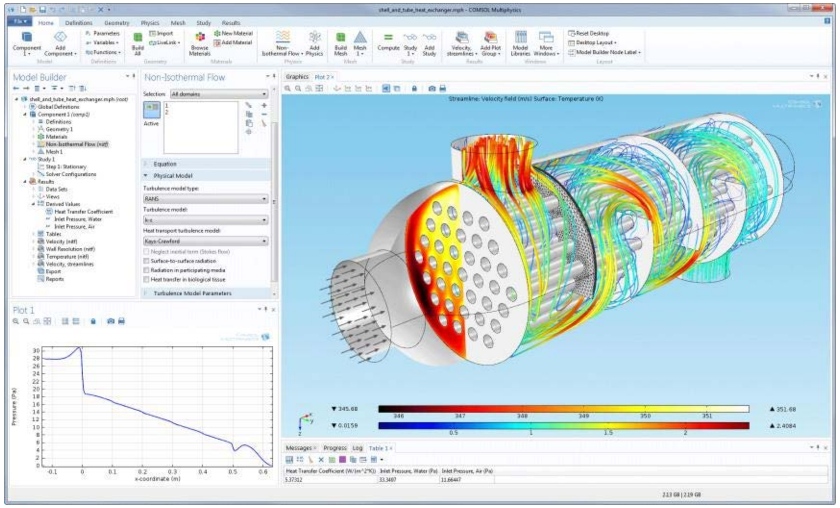
COMSOL Multiphysics

COMSOL Multiphysics is a simulation environment designed with real-world applications in mind. The idea is to mimic as closely as possible effects that are observed in reality. To do this, there is the need to consider multiphysics.

COMSOL allows to :

- Translate real-world physical laws into their virtual form
- Add any physical effect to your model
- Perform reliable simulations

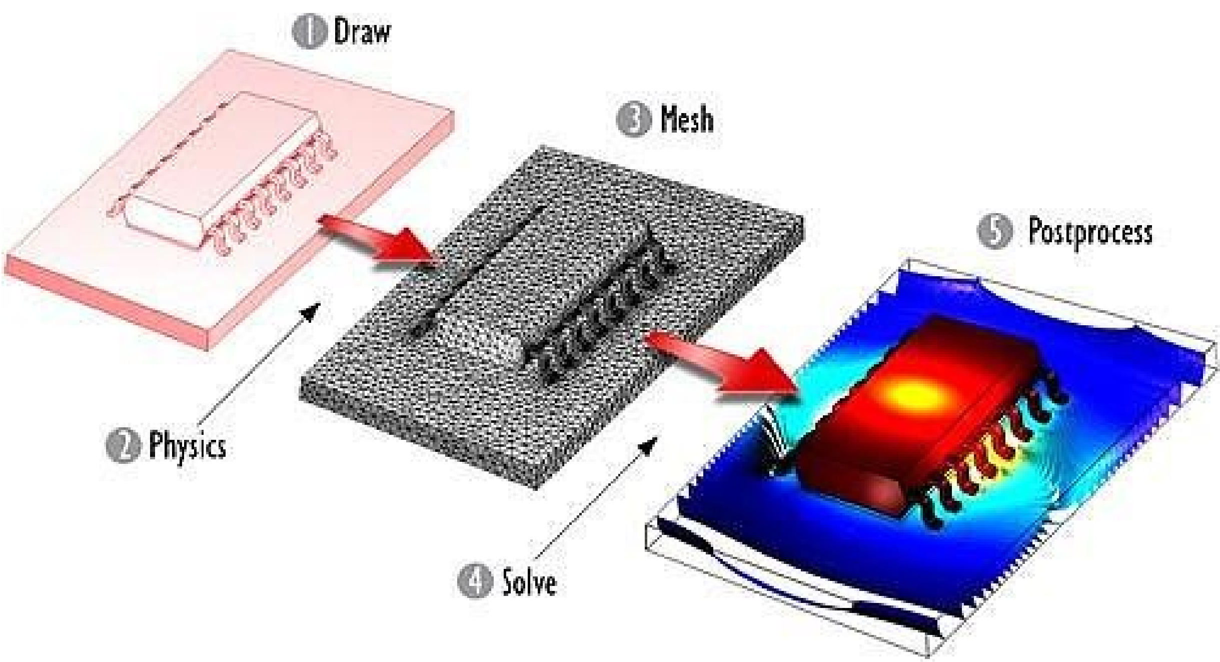
COMSOL MULTIPHYSICS®



COMSOL Multiphysics

Step Analysis Approach

1) Drawing **Geometry** and defining **Materials**
2) Adding **Physics**
3) Defining **Mesh**
4) Performing **Simulation**



Content of the work

➢ Basic Example

Structural Analysis of a Cantilever Beam with an applied distributed load

- Analytical Solution using Beam Theory
- COMSOL Solution

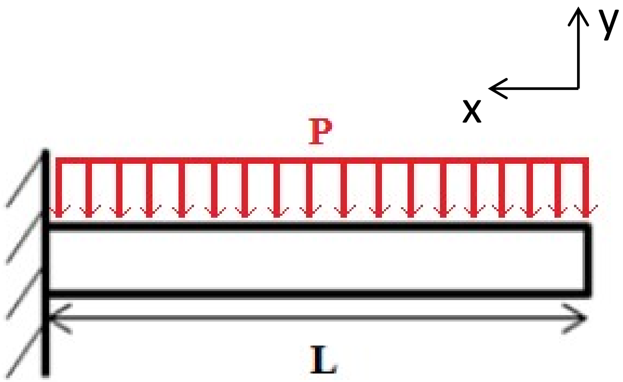
➢ Thorough Example

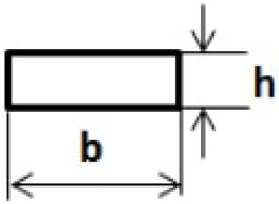
Multiphysics Simulation of a Busbar

- Electrical Heating Analysis
- Structural Deformation Analysis

Basic Example : Cantilever Beam

Beam Theory Solution





Data Set :

L = 3 m
Lenght of the Beam

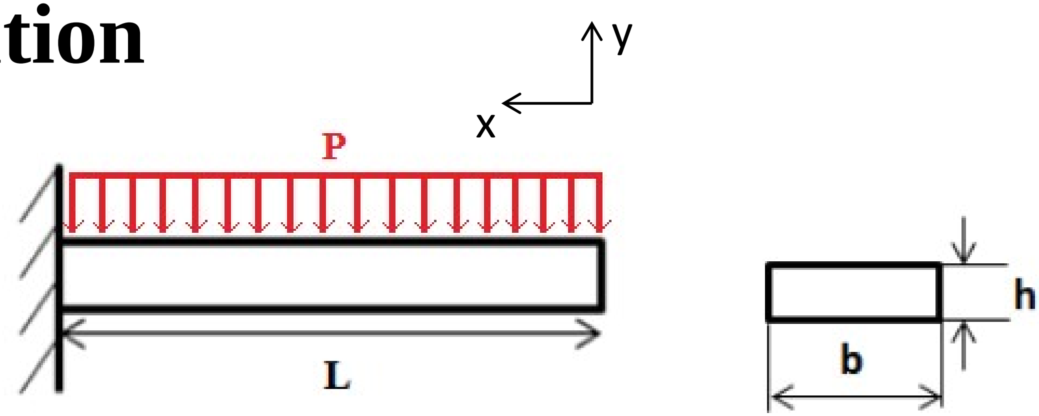
b = 0.1 m	Cross Section Base
h = 0.1 m	Cross Section Height
P = 1022.22 N/m	Distributed Load
E = 68.9 GPa	Young's Modulus of Aluminum 6061-O
ν = 0.33	Poisson's Ratio

ρ = 2700 kg/m^3 Density

Y = 55.2 MPa Yield Strenght

Basic Example : Cantilever Beam

Beam Theory Solution



$$M(x) \square \frac{P}{2} x$$

$$EI \frac{d^2 y}{dx^2} \square \frac{P}{2} x^2$$

$$EI \square(x) \square \int \frac{P}{2} x^2 dx \square \frac{P}{6} x^3 \square C_1 \Rightarrow \square(L) \square 0 \quad , \quad C_1 \square \square \frac{P}{6} L^3$$

$$P\ ^3 dx \square \quad P6 \quad L^3 dx \square 24P\ x^4 \square P6\ L^3 x \square C_2$$

Basic Example : Cantilever Beam

$EIy(x) \square \; 6\,x$

$0\,,\; C_2 \square \frac{}{} L$

$P\;^4 y(L) \square$

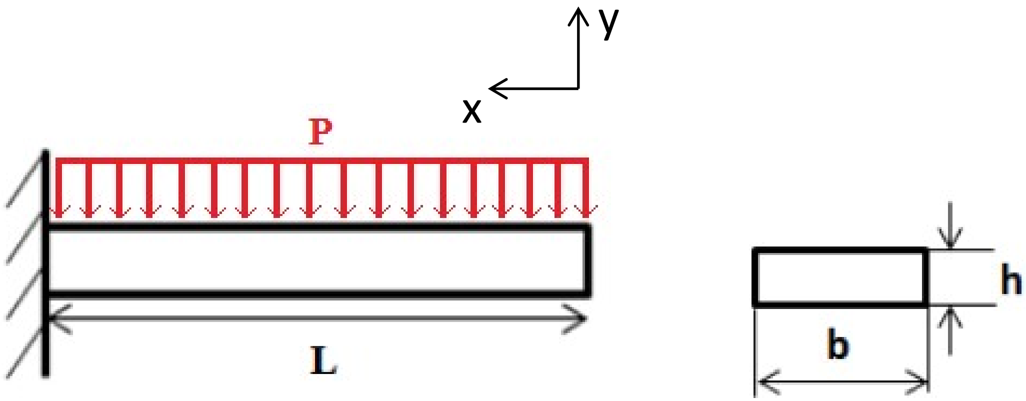
8

Basic Example : Cantilever Beam

$$y(x) \square \frac{P}{24EI}x^4 \square \frac{P}{6EI}L^3x \square \frac{P}{8EI}L^4$$

Basic Example : Cantilever Beam

Beam Theory Solution



$$y_{max} = y(0) = \frac{P}{8EI} L^4 = 0.018026\ m$$

$$\sigma_{max} = \frac{Mh}{2I} = \frac{3P}{bh^2} L^2 = 27.60\ MPa$$

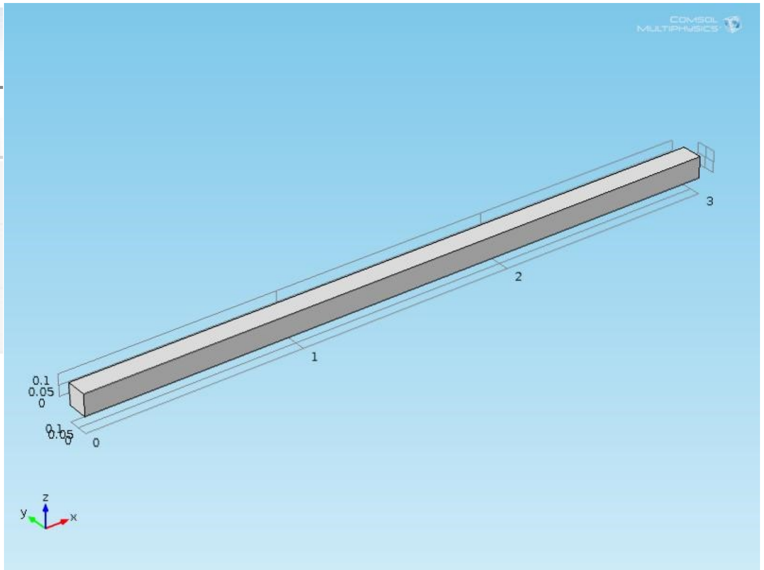
Basic Example : Cantilever Beam

COMSOL Solution

- **Geometry and Material Definition**

Material Contents

	Property	Name	Value	Unit
✓	Young's modulus	E	689000...	Pa
✓	Poisson's ratio	nu	0.33	1
✓	Density	rho	2700	kg/...

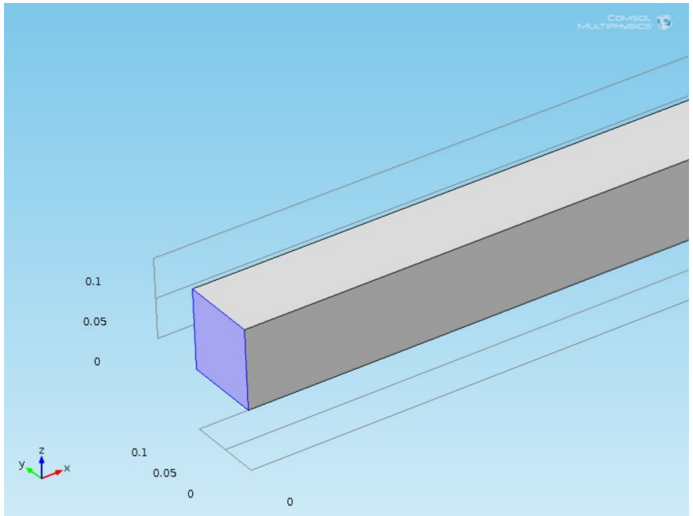


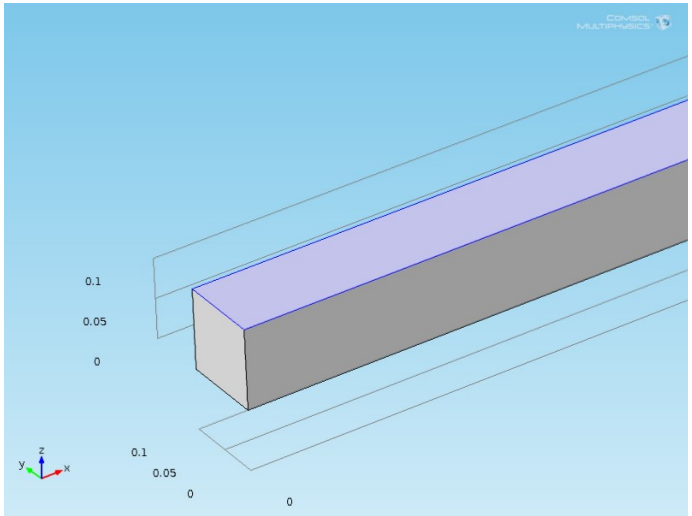
COMSOL Solution

Basic Example : Cantilever Beam

- **Physics & Boundary Conditions**

  "Solid Mechanics"module is selected and a <u>fixed constraint</u> together with a <u>boundary load</u> are defined





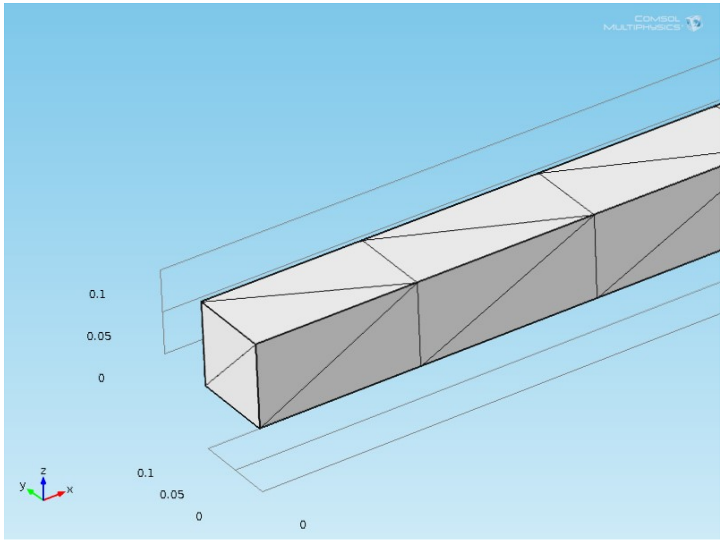
Basic Example : Cantilever Beam

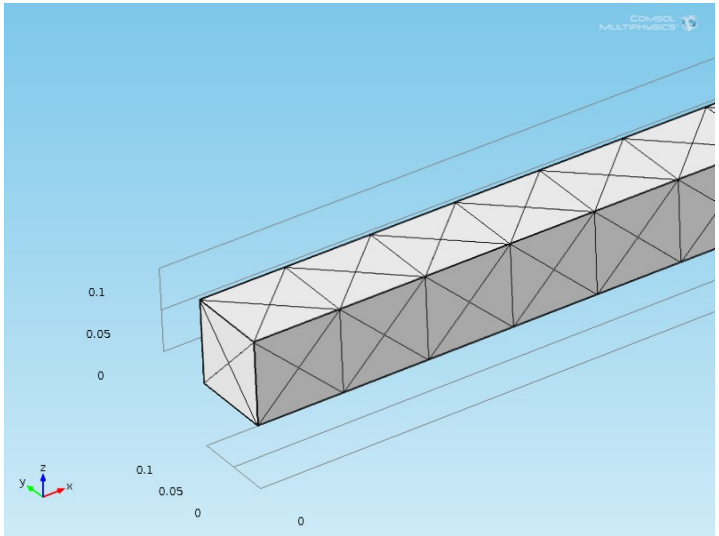
COMSOL Solution

- **Mesh Definition**

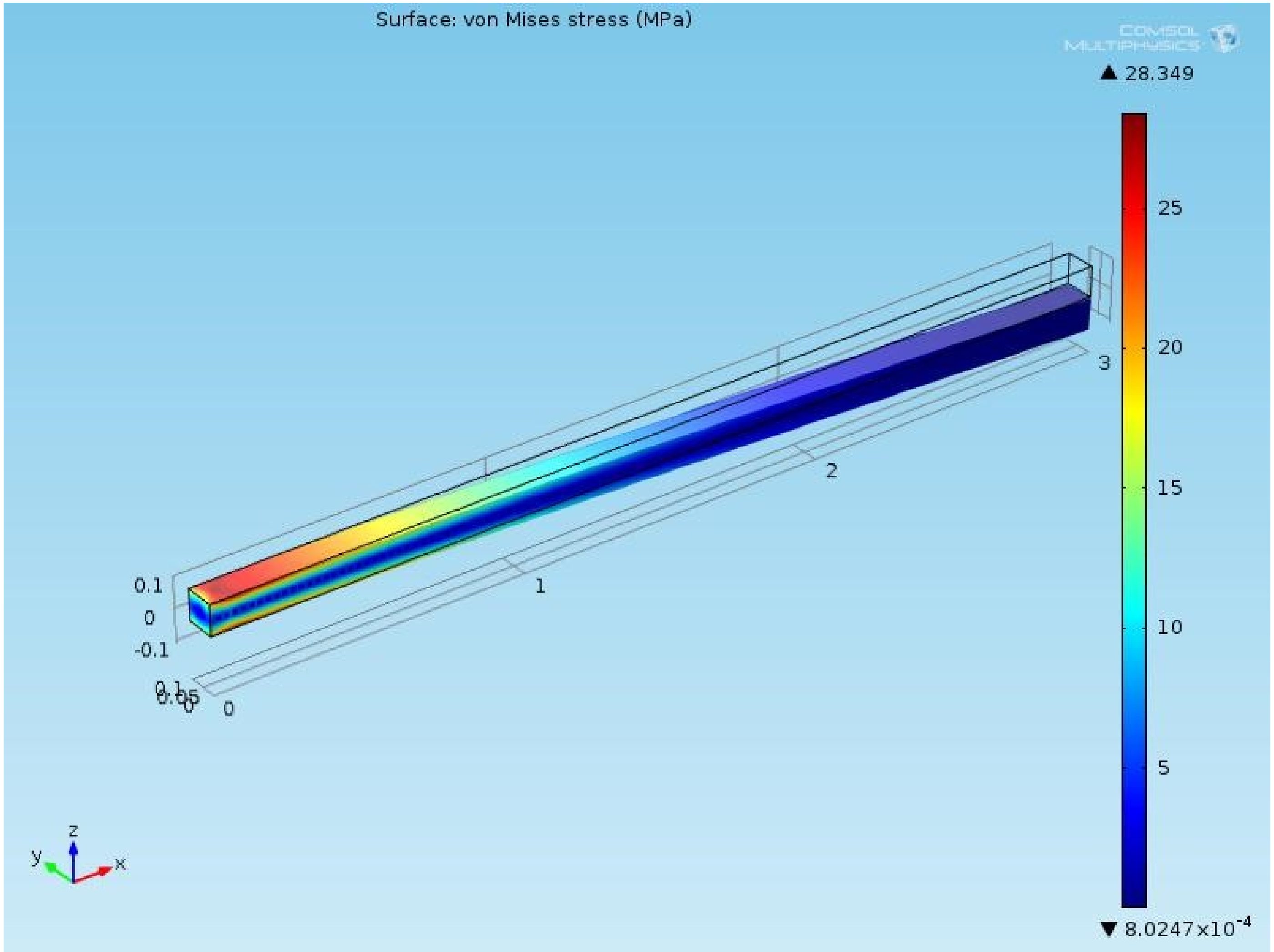
A coarse mesh is firstly added to the model, then a finer mesh is chosen to perform a second simulation

Basic Example : Cantilever Beam





Basic Example : Cantilever Beam



• B

Basic Example : Cantilever Beam

Beam Theory

$y_{max} = 0.018026 \ m$
$\sigma_{max} = 27.60 \ MPa$

COMSOL Coarse Mesh $y_{max} = 0.017601 \ m$ $\sigma_{max} = 25.15 \ MPa$

COMSOL Fine Mesh $y_{max} = 0.017922 \ m$ $\sigma_{max} = 28.35 \ MPa$

The percent error (%E) in our model max deflection/stress can be defined as:

$$\%E = abs\left(\frac{x_{theoretical} - x_{model}}{x_{theoretical}}\right) \times 100$$

Coarse Mesh

$\%E_y = 2.36 \%$

Basic Example : Cantilever Beam

$\%E_\sigma = 8.88$

% **Fine Mesh**

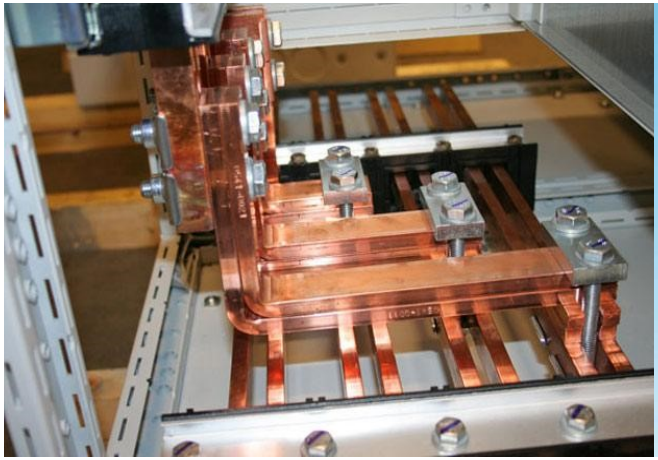
$\%E_y = 0.58\ \%$

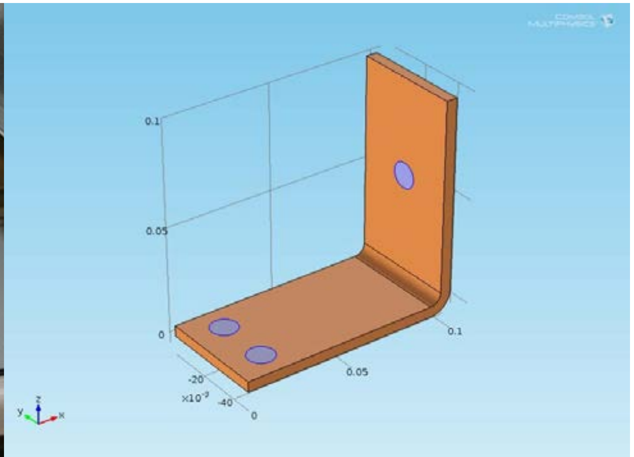
$\%E_\sigma = 2.70\ \%$

Thorough Example : Busbar

In electrical power distribution, a **busbar** is a strip or bar of copper, brass or aluminum that conducts electricity within an electrical apparatus.

The cross-sectional size of the busbar determines the maximum amount of current that can be safely carried.





Busbar : Joule Heating

The current conducted in the busbar, from the top bolt to the bolts on the bottom, produces heat due to resistive losses, a phenomenon referred to as **Joule Heating**.

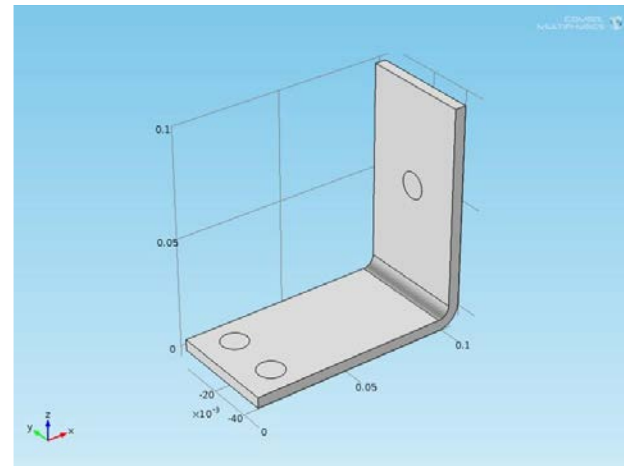
- **Geometry and Material Definition**

Busbar : Joule Heating

Geometry

Parameters

Name	Expression	Value	Description
L	9[cm]	0.090000 m	lenght of busbar
rad_1	6[mm]	0.0060000 m	radius of bolts
tbb	5[mm]	0.0050000 m	thickness of busbar
wbb	5[cm]	0.050000 m	width of busbar



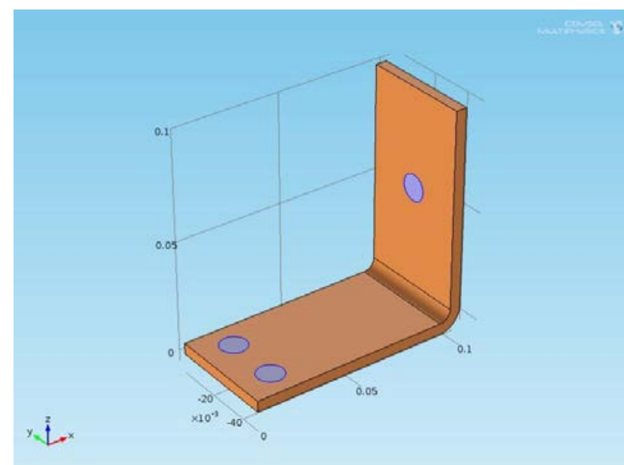
Materials

<u>Busbar</u> Material : **Copper**

Electrical Conductivity : $\sigma_{Cu} = 5.998*10^7$ S/m

<u>Bolts</u> Material : **Titanium beta-21S**

Electrical Conductivity : $\sigma_{Ti} = 7.407*10^5$ S/m



Busbar : Joule Heating

- **Physics & Boundary Conditions**

“Joule Heating” module is selected and the boundary conditions for the heat transfer problem and the conduction of electric current are set as follows. **Boundary Conditions**

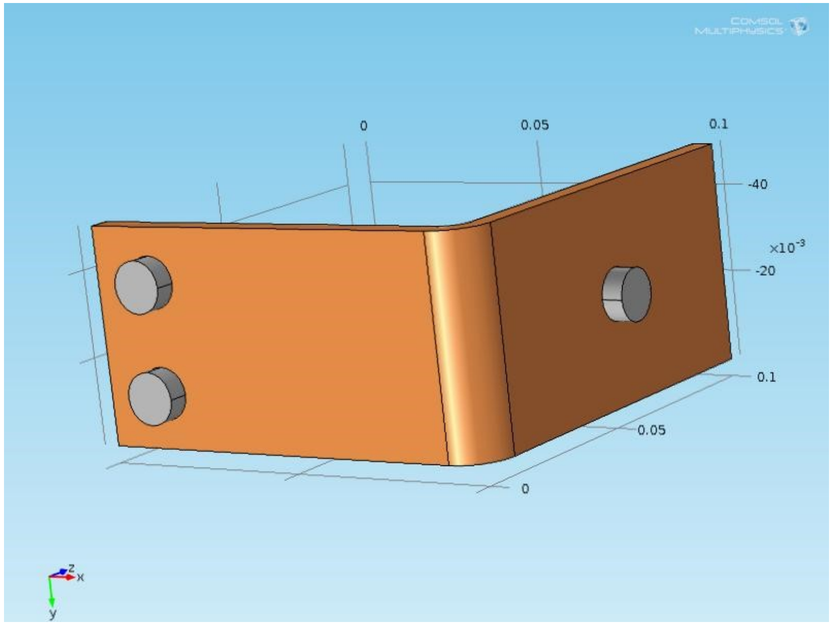
Heat Transfer : Circular bolt

boundariesneither heated are

norassumed cooled to by be the

surroundings.

Electric Current : The single Titanium bolt is set to an electric potential of 20 mV , whereas the

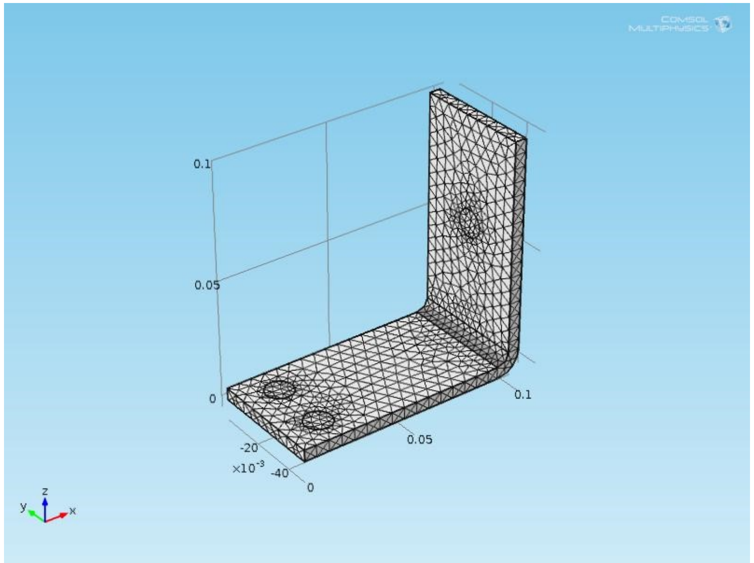


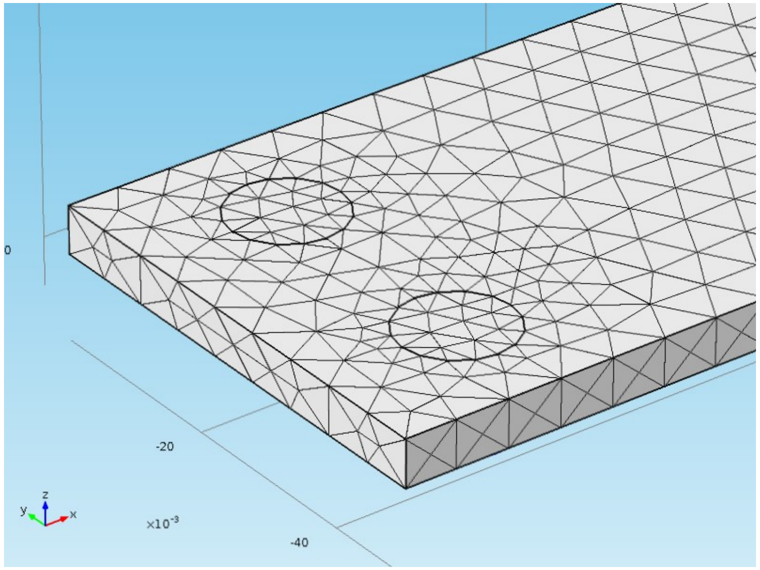
Busbar : Joule Heating

two remaining bolts are set to ground.

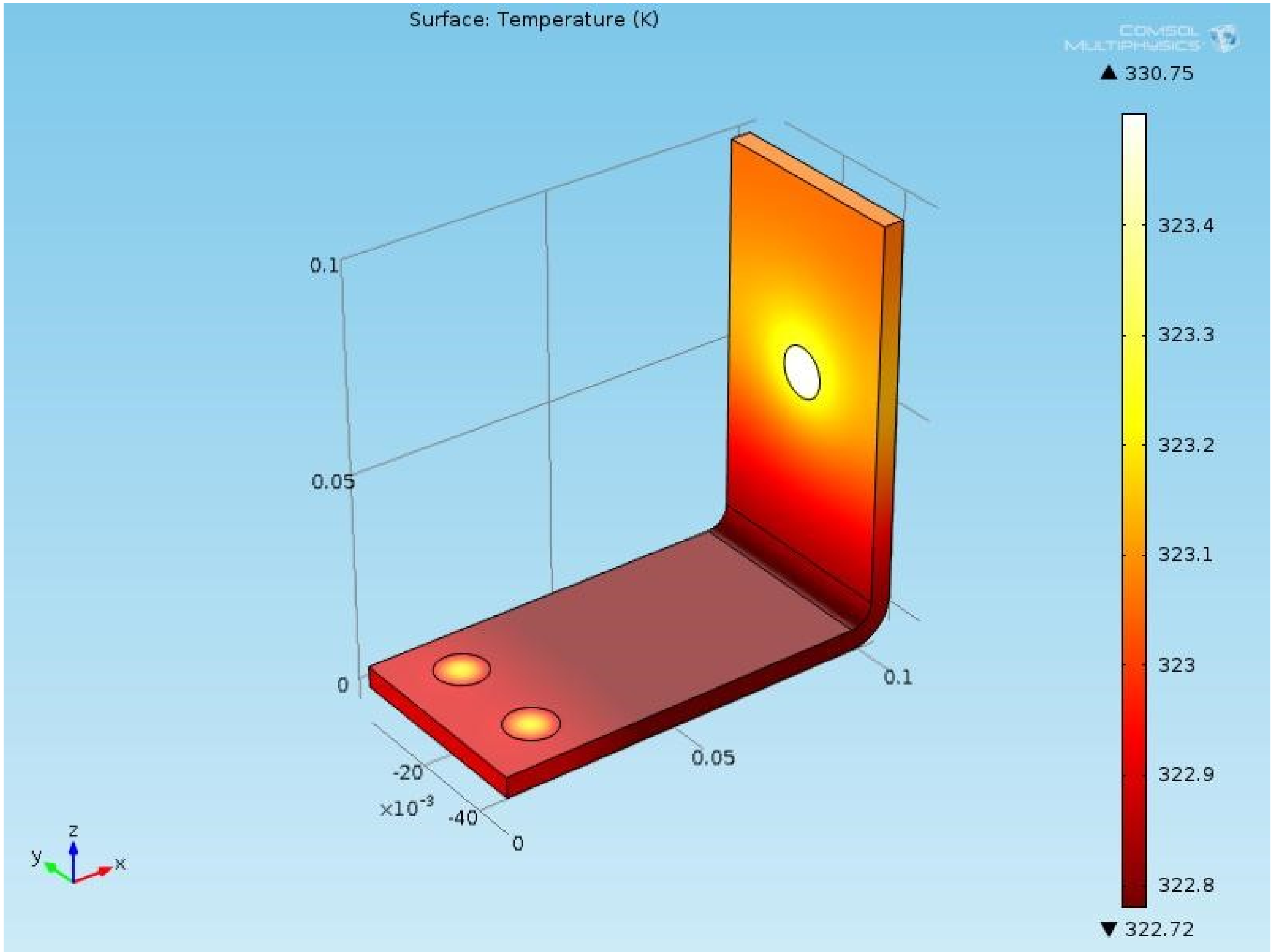
- **Mesh Definition**

A physics-controlled mesh is created by default. It is important to notice that the number of elements on curved boundaries is higher.

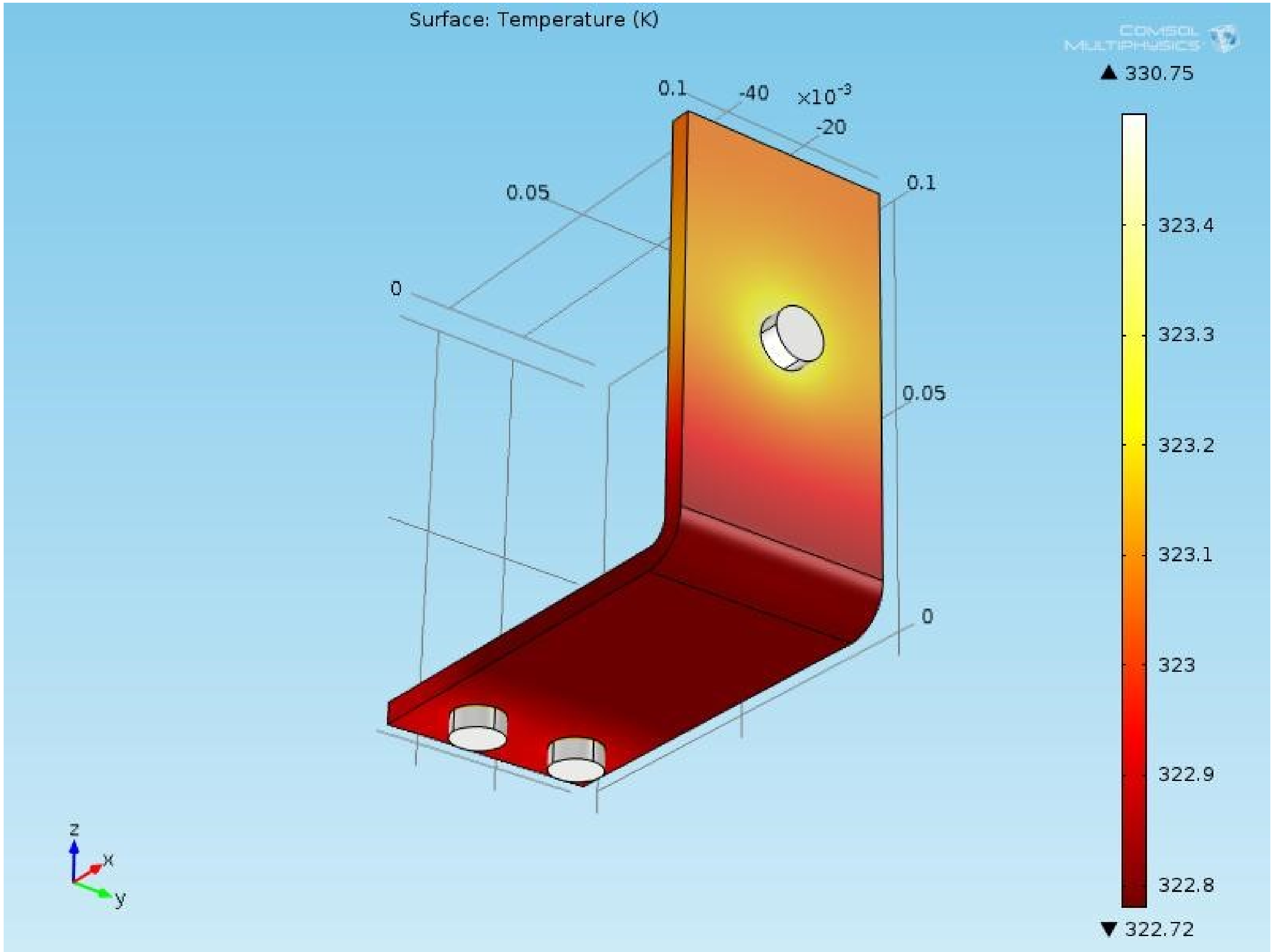




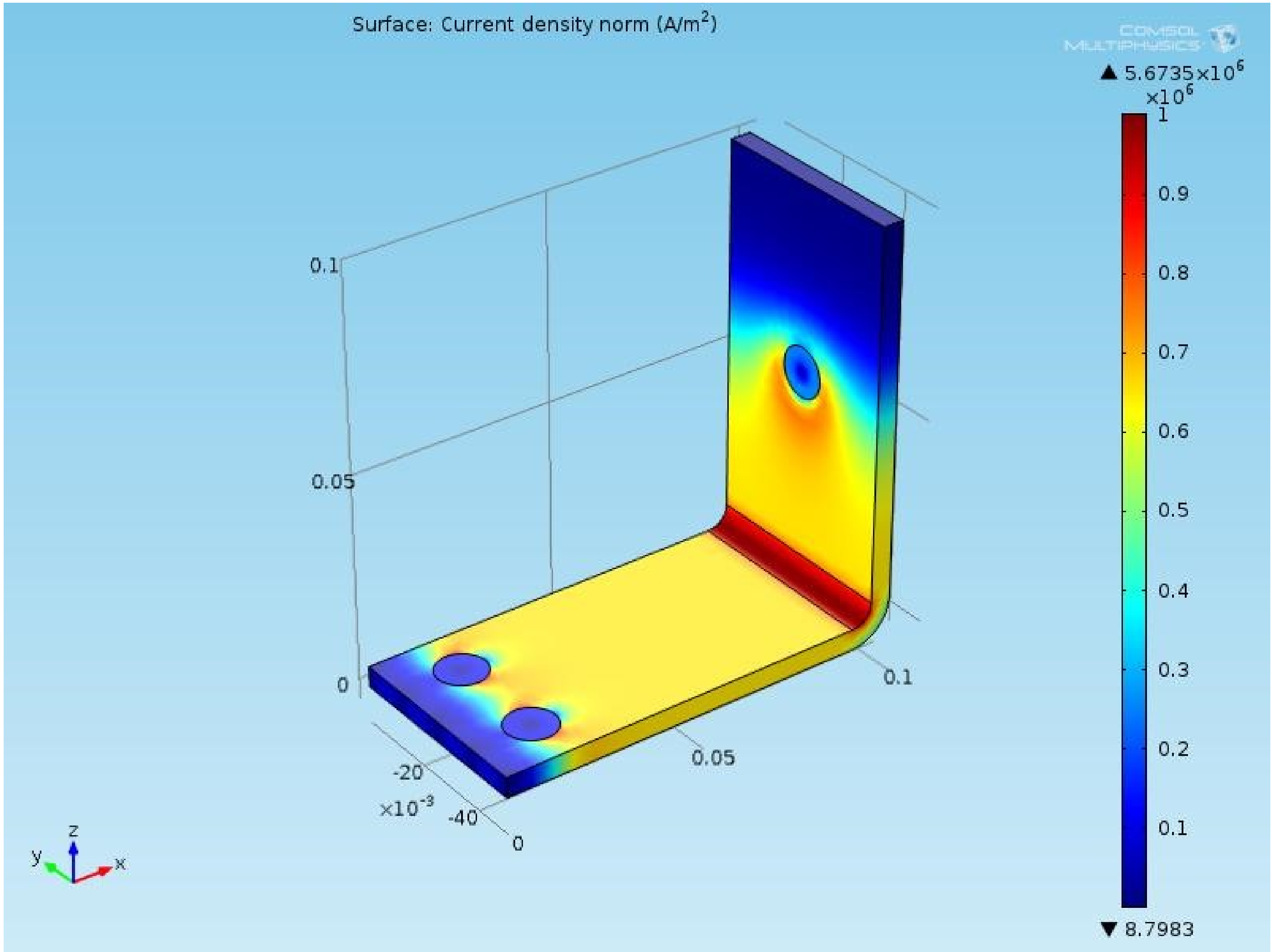
Busbar : Joule Heating



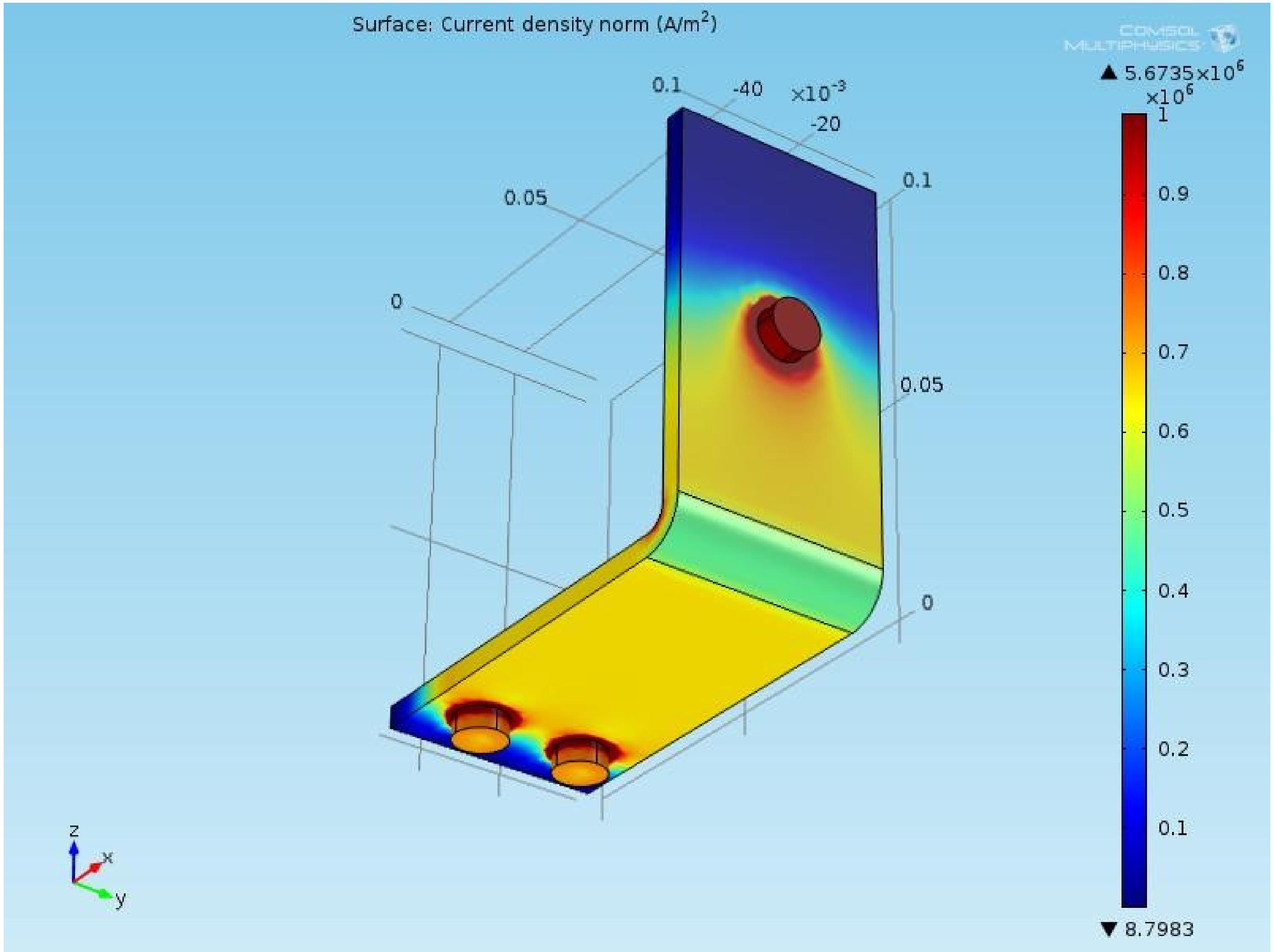
Busbar : Joule Heating



Busbar : Joule Heating



Busbar : Joule Heating



Busbar : Joule Heating

Adding Physics to the Model

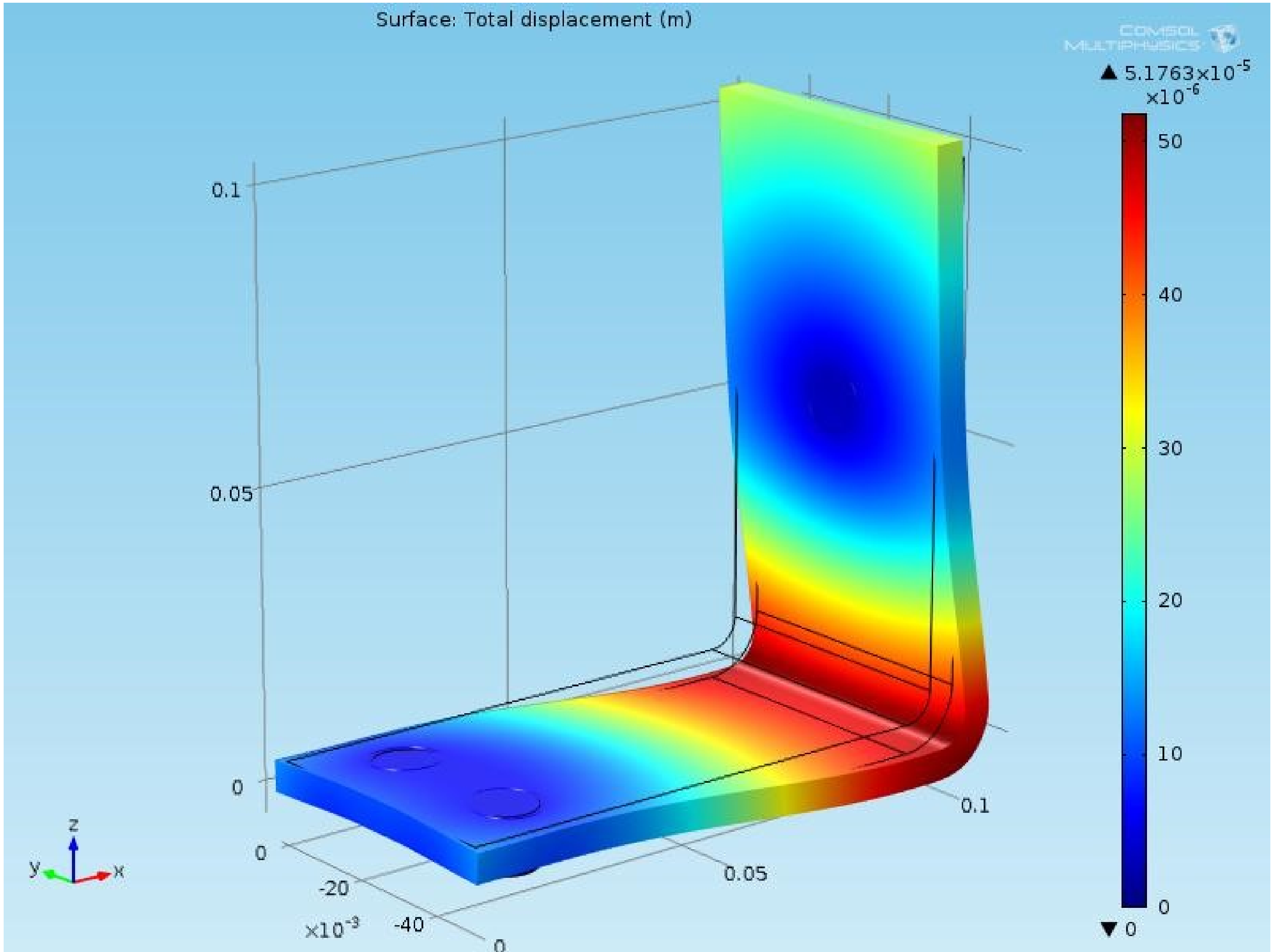
The Joule Heating simulation has just shown that there is a temperature rise in the busbar. So it is logical to ask :

What kind of mechanical stress is induced by thermal expansion?

To answer this question we can **simply expand the existing model** to include the physics associated **with "Structural Mechanics"**.

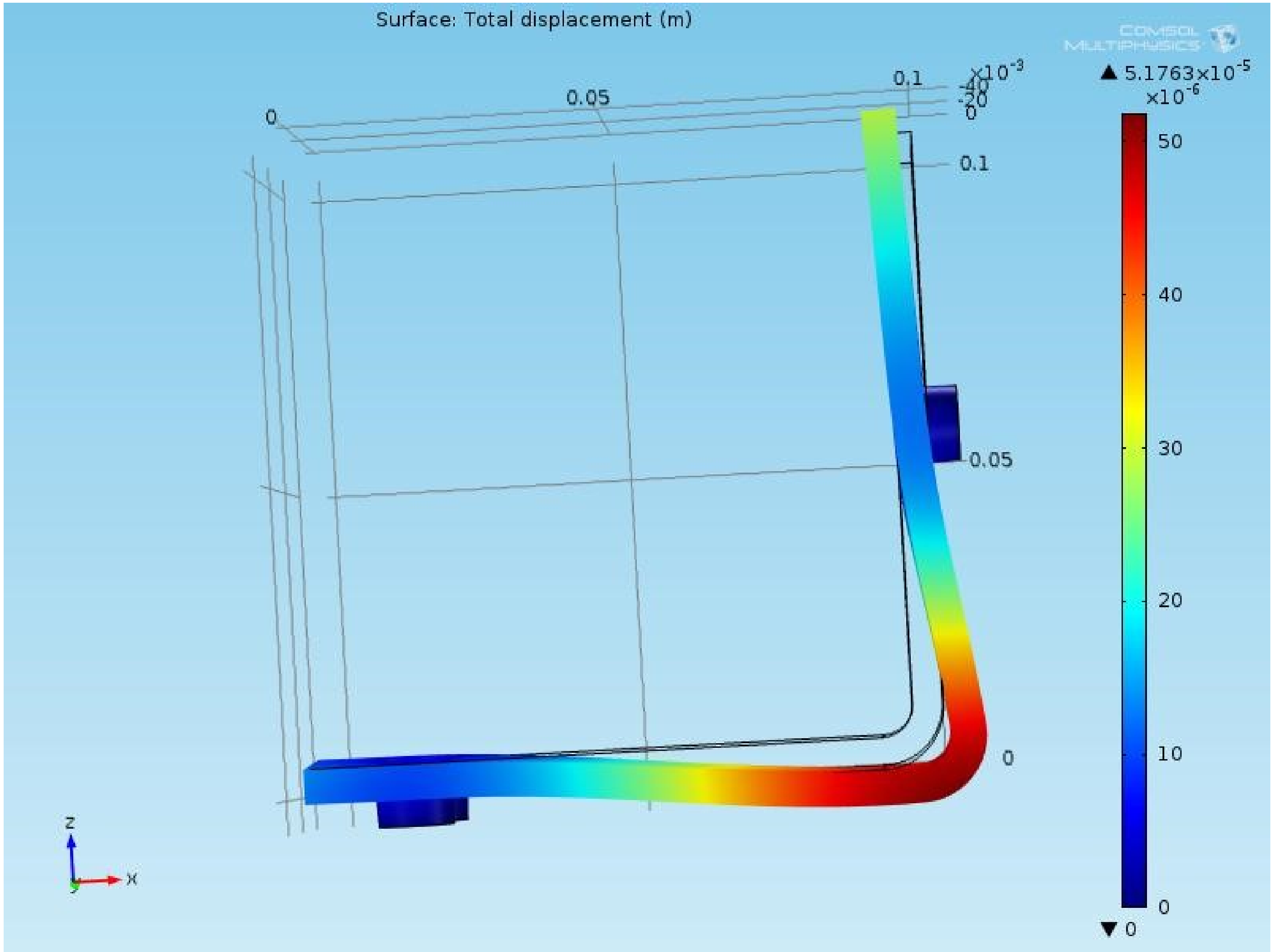
The Joule Heating module and the Structural Mechanics module are then coupled to perform an overall simulation which includes both the effects. **We can now perform a true Multiphysics Simulation!**

Busbar : Structural Deformation



- **Results : Total Displacement**

Busbar : Structural Deformation

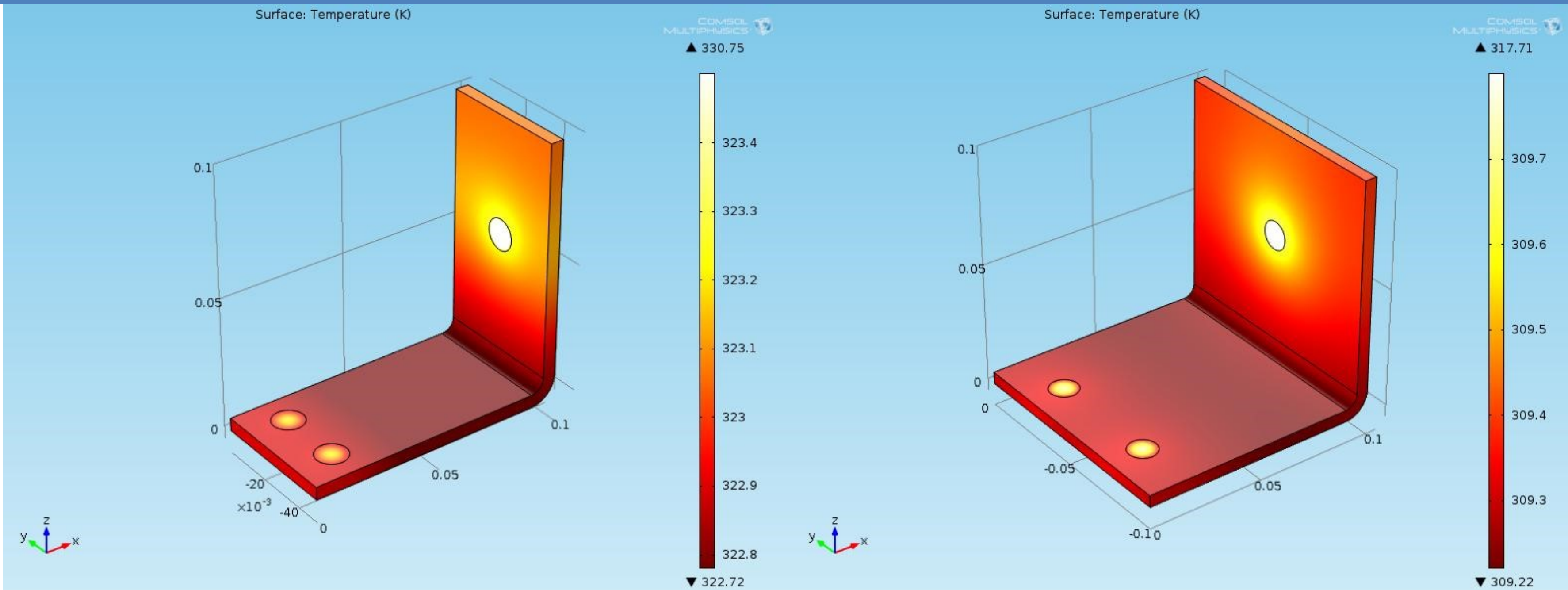


- **Results : Total Displacement**

Sweeping a Geometric Parameter

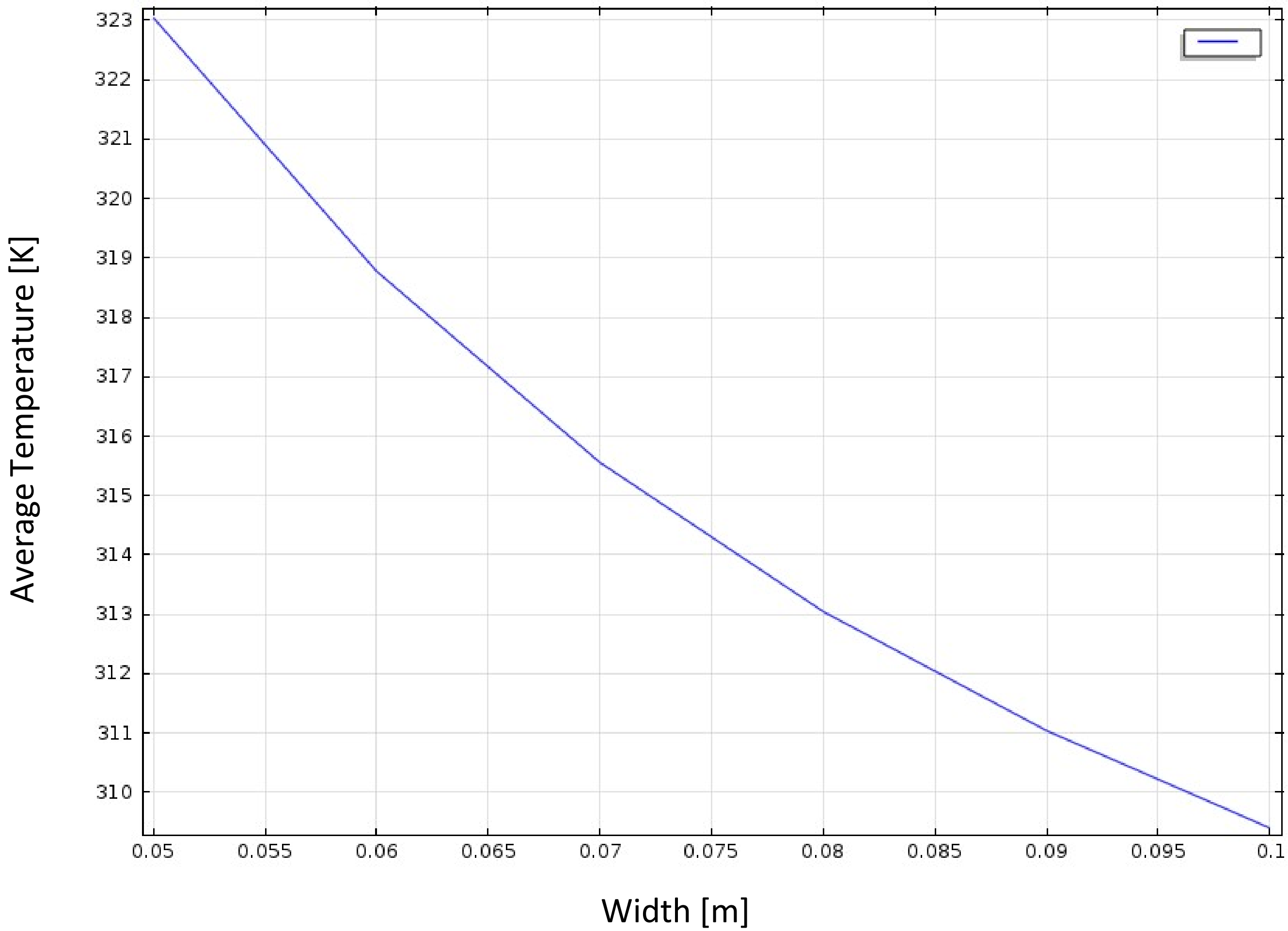
Often it is interesting to generate multiple instances of a design to meet specific constraints. For the busbar, a design goal might be to lower the operating temperature by varying its geometry.

- Initial busbar width = 5 cm , T_{max} = 330 K
- New busbar width = 10 cm , T_{max} = 317 K



Sweeping a Geometric Parameter

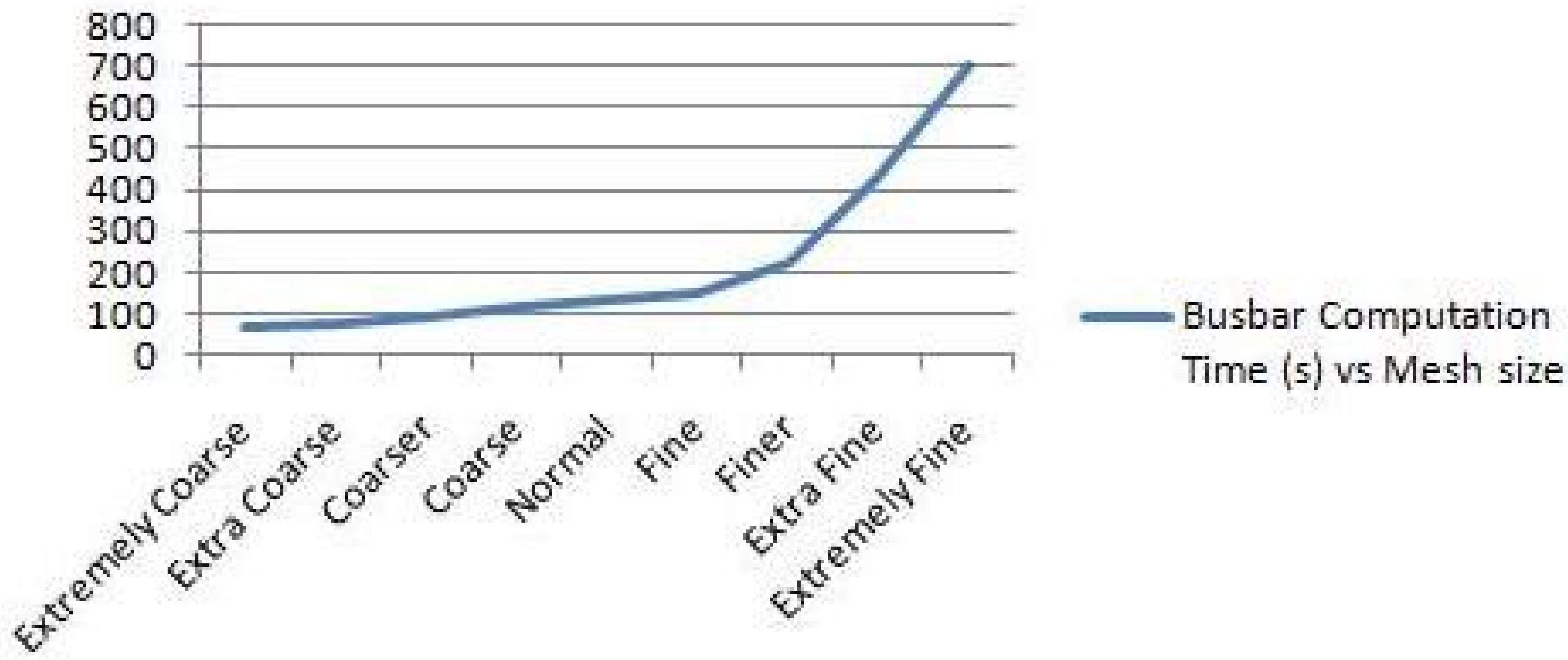
The plot shows that the average temperature decreases as the width increases.



Average Temperature [K]
323
322
321
320
319
318
317
316
315
314
313
312
311
310
0.05
0.055
0.06
0.065
0.07
0.075
0.08
0.085
0.09
0.095
0.1
Width [m]

Busbar Computation Time vs Mesh Size

The computational time decreases with the coarseness of the mesh size, as shown by the plot.



Conclusions

- ✓ The Cantilever Beam analysis shows that computer simulation can provide reliable results (as predicted in theory)
- ✓ It is possible to add several physical domains without the need of creating a disconnected model each time
- ✓ It is possible to test various options of a product more quickly and efficiently
- ✓ It is important to find a common balance between

computation speed and precision of results
(“Parallel Computing” for very time consuming simulations)

Thank you for your attention!