

ENR145 Computational Methods: COMSOL multi-physics speed run

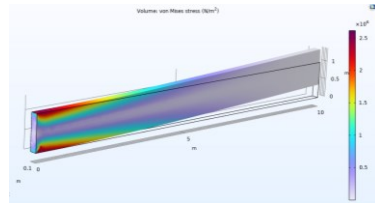
Xiang Li
Spring 2026

COMSOL Multiphysics is easy if you don't math

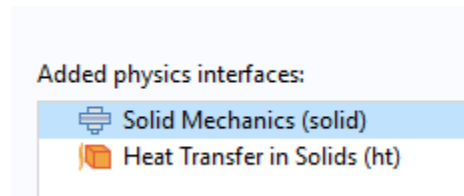
Step 1: start COMSOL



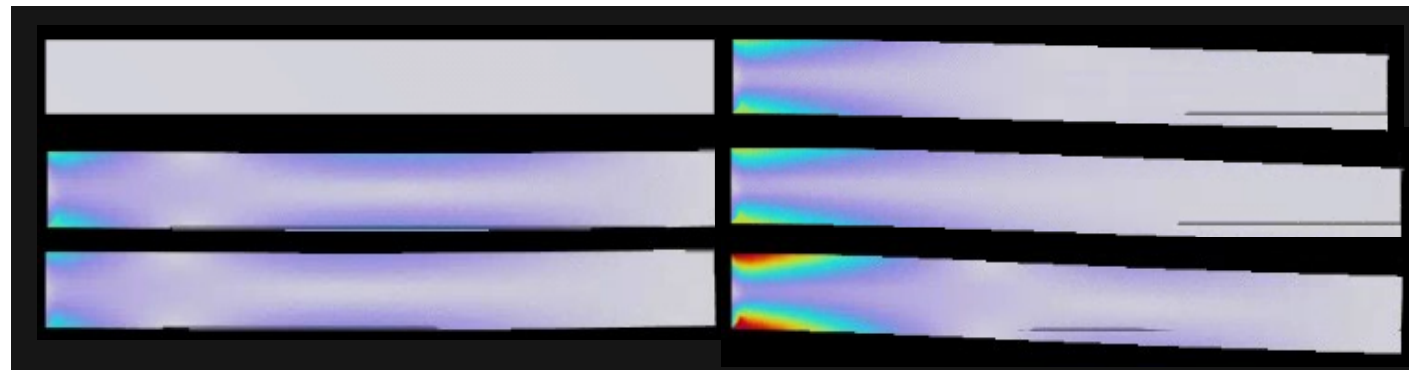
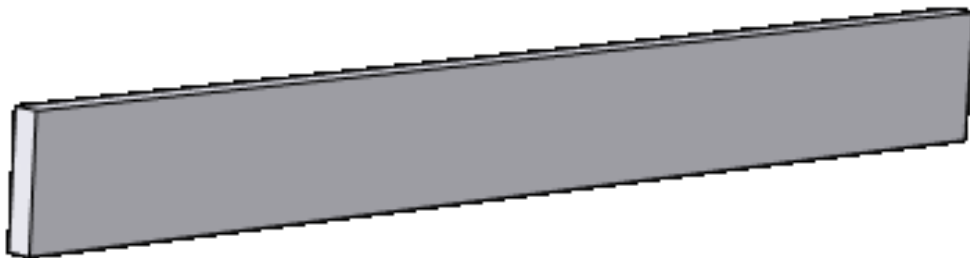
Step 2: run one model



Step 3: run another model at the same time



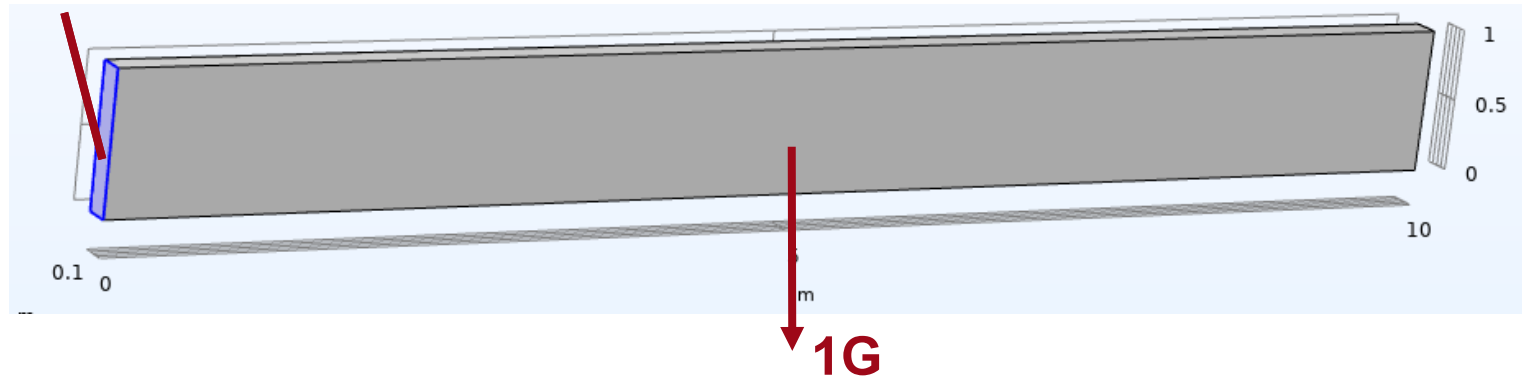
Step 4: have educated guess about everything multiphysics



The wobbling effect when E is T dependent in the 1st second for a wax beam

This is our model: a wax beam

Constrained here on this face



Settings

Block

Build Selected Build All Objects

Type: Solid

Size and Shape

Width: 10 m

Depth: 0.2 m

Height: 1 m

Position

Base: Corner

x: 0 m

y: 0 m

z: 0 m

Axis

Axis type: z-axis

Rotation Angle

Rotation: 0 deg

Linear Elastic Material

Material symmetry: Isotropic

Specify: Young's modulus and Poisson's ratio

Young's modulus: E User defined 39E6 Pa

Poisson's ratio: ν User defined 0.45 1

Density: ρ User defined rho kg/m³

Why this?

Introducing: global parameters:

Global Definitions

Parameters 1

Settings

Parameters

Label: Parameters 1

Parameters

Name	Expression	Value	Description
Vol	$10 \cdot 0.2 \cdot 1$ [m ³]	2 m ³	volume of the beam
g	9.81 [m/s ²]	9.81 m/s ²	gravity on earth
rho	900 [kg/m ³]	900 kg/m ³	density of the beam
gravi_1	Vol*rho*g	17658 N	total gravitational force

COMSOL syntax is matlab syntax

Value [unit]

power is ^

Scientific value is: E



COE COLLEGE

And gravity will be easy to adjust:

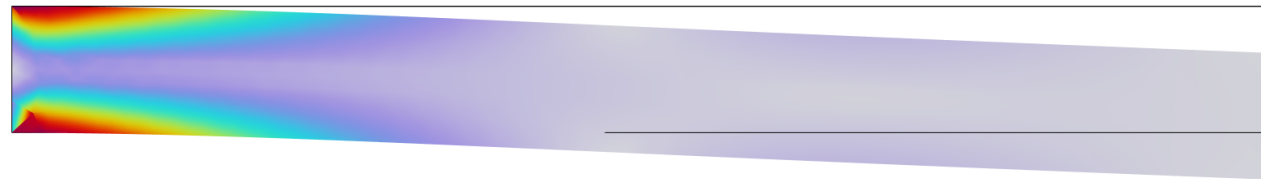
Force

Load type:
Total force

F_{tot} User defined

0	x
0	y
-gravi_1	z

N



Wax beam is bending under its own weight.

Introducing: heat transfer

- Heat Transfer in Solids (*ht*)

 - Solid 1

 - Initial Values 1

 - Thermal Insulation 1

 - Heat Flux 1

 - Heat Flux 2

- Solid Mechanics (*solid*)

 - Linear Elastic Material 1

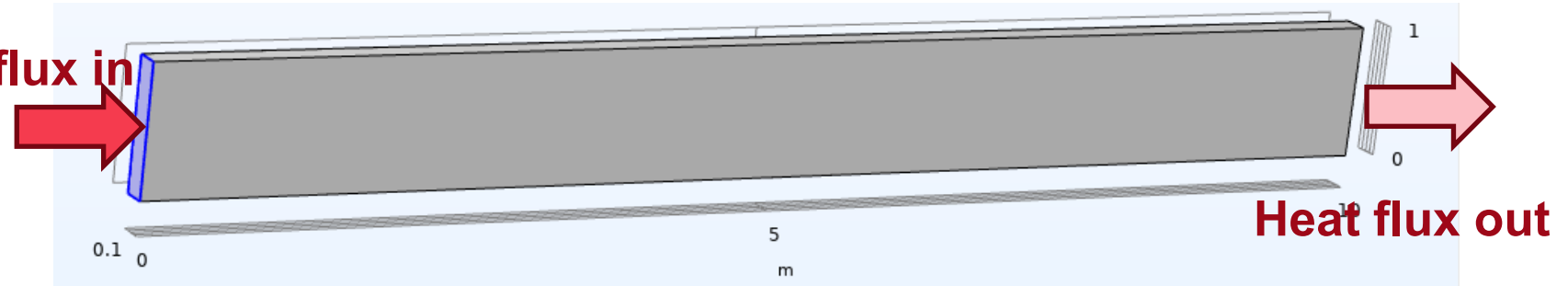
 - Free 1

 - Initial Values 1

 - Fixed Constraint 1

 - Body Load 1

Heat flux in



Heat flux out



COE COLLEGE

Introducing: heat transfer

Heat Transfer in Solids (ht)

- Solid 1
- Initial Values 1
- Thermal Insulation 1
- Heat Flux 1
- Heat Flux 2

Solid Mechanics (solid)

- Linear Elastic Material 1
- Free 1
- Initial Values 1
- Fixed Constraint 1
- Body Load 1

Settings
Solid

Label: Solid 1

Domain Selection

Selection: All domains

1

Override and Contribution

Equation

Model Input

Coordinate System Selection

Coordinate system: Global coordinate system

Heat Conduction, Solid

Thermal conductivity:

k User defined

0.2 W/(m·K)

Isotropic

Thermodynamics, Solid

Density:

ρ User defined

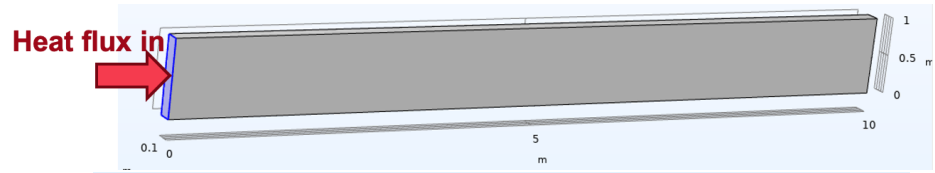
rho kg/m³

Heat capacity at constant pressure:

C_p User defined

2.5 J/(kg·K)

Introducing: heat transfer



- Heat Transfer in Solids (*ht*)
 - Solid 1
 - Initial Values 1
 - Thermal Insulation 1
 - Heat Flux 1
 - Heat Flux 2
- Solid Mechanics (*solid*)
 - Linear Elastic Material 1
 - Free 1
 - Initial Values 1
 - Fixed Constraint 1
 - Body Load 1

Settings

Heat Flux

Label: Heat Flux 1

Boundary Selection

Selection: Manual

1

Override and Contribution

Equation

Material Type

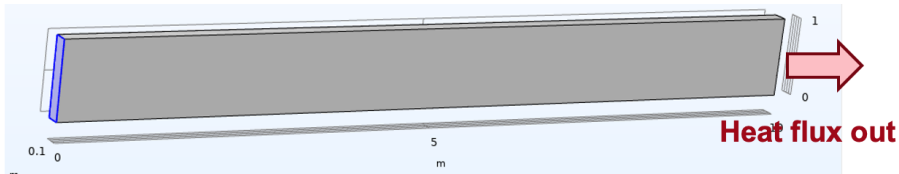
Material type: Nonsolid

Heat Flux

Flux type: General inward heat flux

q_0 55e3 W/m²

Introducing: heat transfer



Heat Transfer in Solids (*ht*)

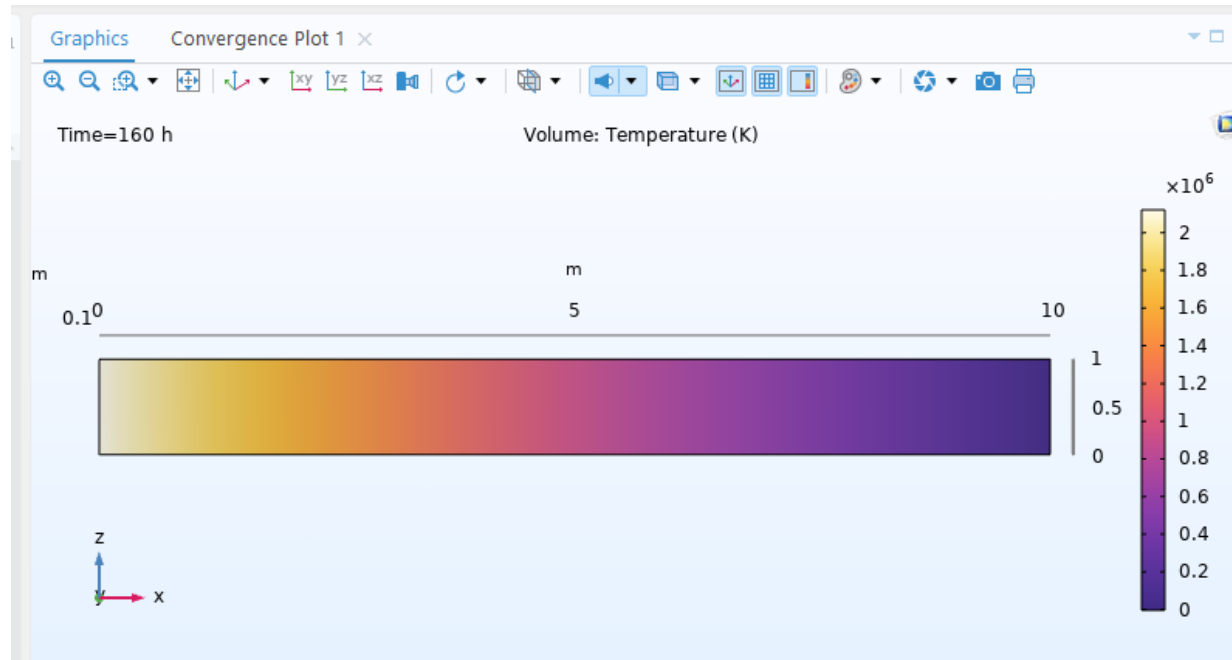
- Solid 1
- Initial Values 1
- Thermal Insulation 1
- Heat Flux 1
- Heat Flux 2

Solid Mechanics (*solid*)

- Linear Elastic Material 1
- Free 1
- Initial Values 1
- Fixed Constraint 1
- Body Load 1

The screenshot shows the "Settings" panel for a "Heat Flux" boundary condition. The "Label" is "Heat Flux 2". The "Boundary Selection" is set to "Manual" and shows a selection of "6". The "Material Type" is "Nonsolid". The "Heat Flux" section is expanded, showing "Flux type" set to "Convective heat flux", "Heat transfer coefficient" h set to -10 W/(m²·K), and "External temperature" T_{ext} set to "User defined" with a value of 293.15[K]. Red boxes highlight the "Flux type" and "External temperature" settings.

But heat transfer is time-dependent, right?




But heat transfer is time-dependent, right?




Just add a time-dependent study, duh!

Added study:

 Time Dependent

Settings

Time Dependent

 Compute  Update Solution

Label: Time Dependent

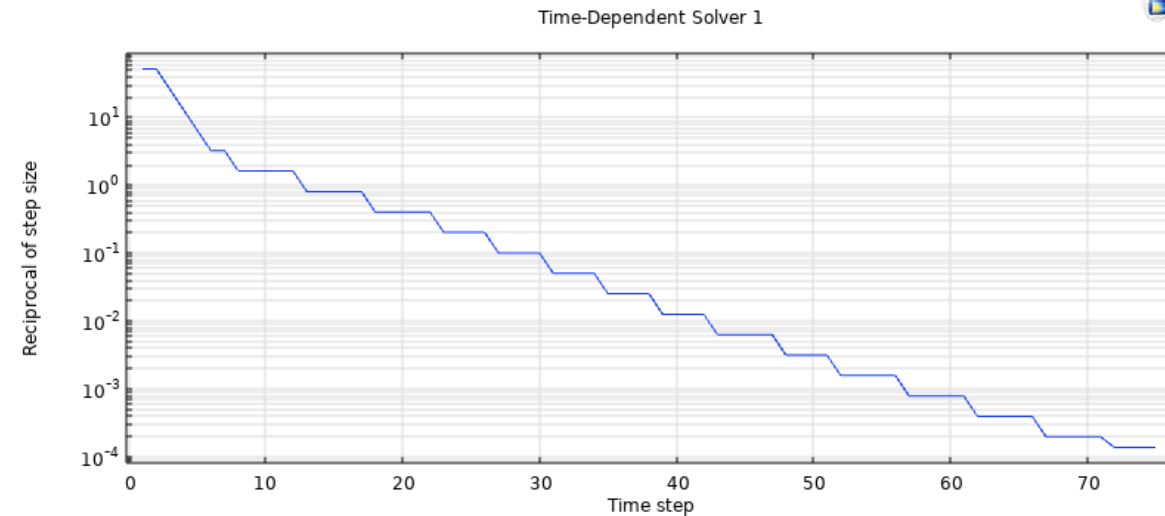
Study Settings

Time unit: s

Output times: range(0,0.1,1) s

Tolerance: Physics controlled

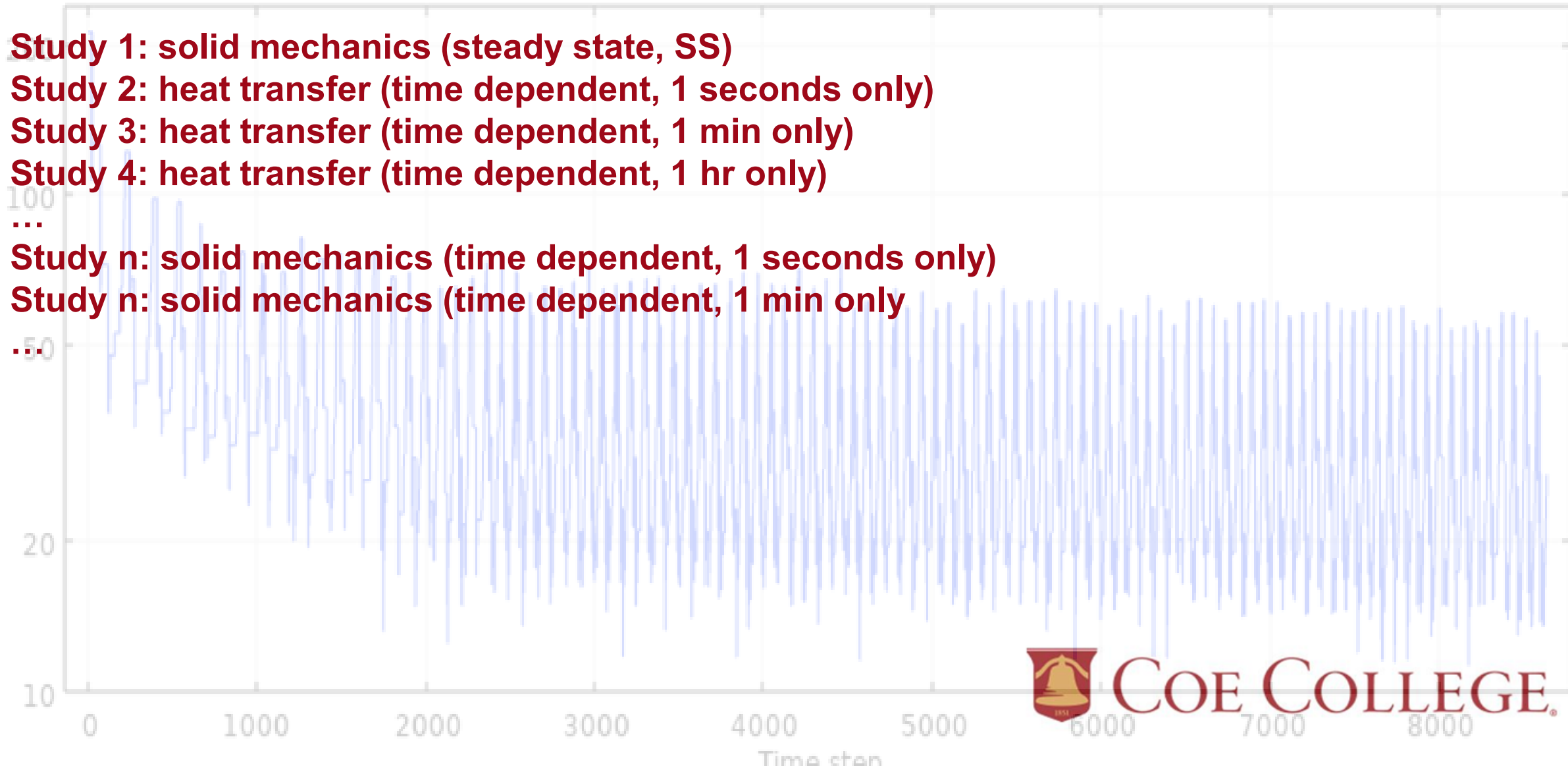
Looks familiar? I knew that your python skill could pay-off.



At this stage, you should NOT run a model that takes ~10 min to compute. IT WON'T WORK.

Take baby steps, one model a time

- Study 1: solid mechanics (steady state, SS)**
- Study 2: heat transfer (time dependent, 1 seconds only)**
- Study 3: heat transfer (time dependent, 1 min only)**
- Study 4: heat transfer (time dependent, 1 hr only)**
- ...
- Study n: solid mechanics (time dependent, 1 seconds only)**
- Study n: solid mechanics (time dependent, 1 min only)**



Then, you add “Multiphysics”

Young's modulus and Poisson's ratio

Young's modulus:

E User defined

39E6-1000*T What is the meaning of this? Pa

Again, this is NOT a good idea.

Physics and Variables Selection

Modify model configuration for study step

Node	Solve for	Equation form
Component 1 (comp1)	<input checked="" type="checkbox"/>	
Solid Mechanics (solid)	<input checked="" type="checkbox"/>	Automatic (Stationary)
Heat Transfer in Solids (ht)	<input checked="" type="checkbox"/>	Automatic (Stationary)

This IS a good idea.

Physics and Variables Selection

Modify model configuration for study step

Node	Solve for	Equation form
Component 1 (comp1)	<input checked="" type="checkbox"/>	
Solid Mechanics (solid)	<input type="checkbox"/>	Automatic (Stationary)
Heat Transfer in Solids (ht)	<input checked="" type="checkbox"/>	Automatic (Stationary)



Then, you add “Multiphysics”

When E is a constant:



When E is temperature dependent:



Then, you add “Multiphysics”

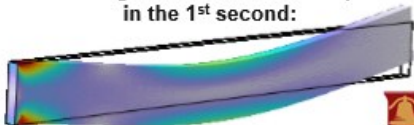
When E is a constant:



When E is temperature dependent:



The wobbling effect when E is T dependent
in the 1st second:

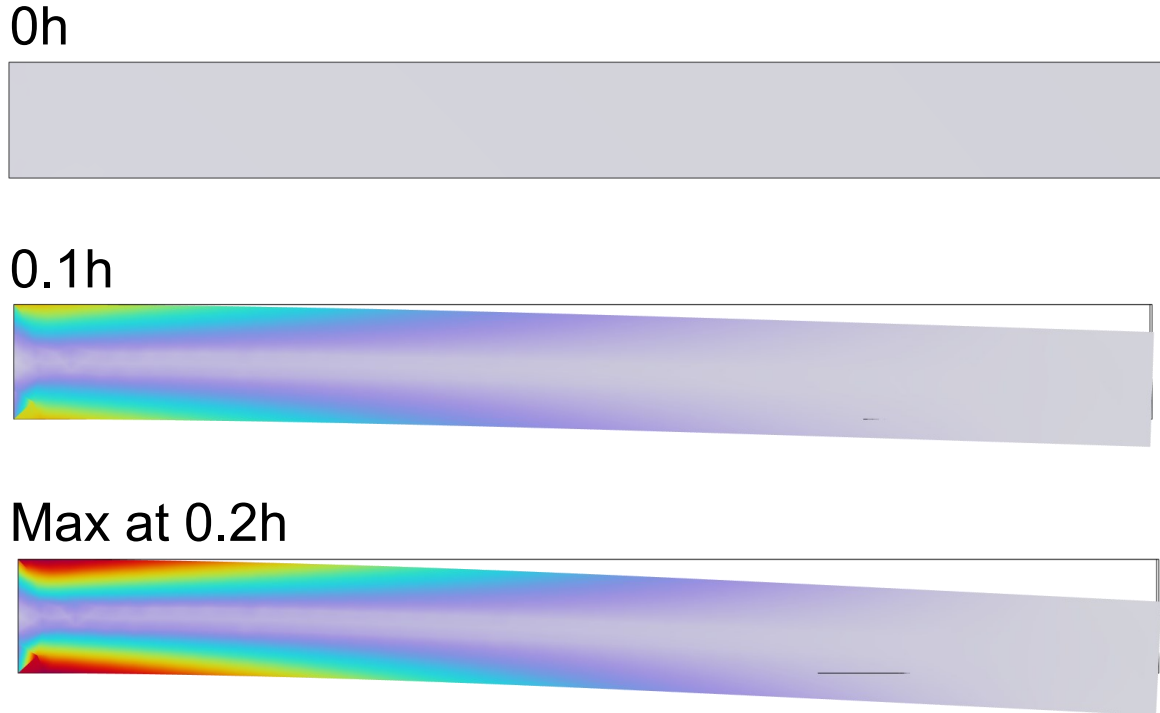


The wobbling effect when E is T dependent
in the 1st second:

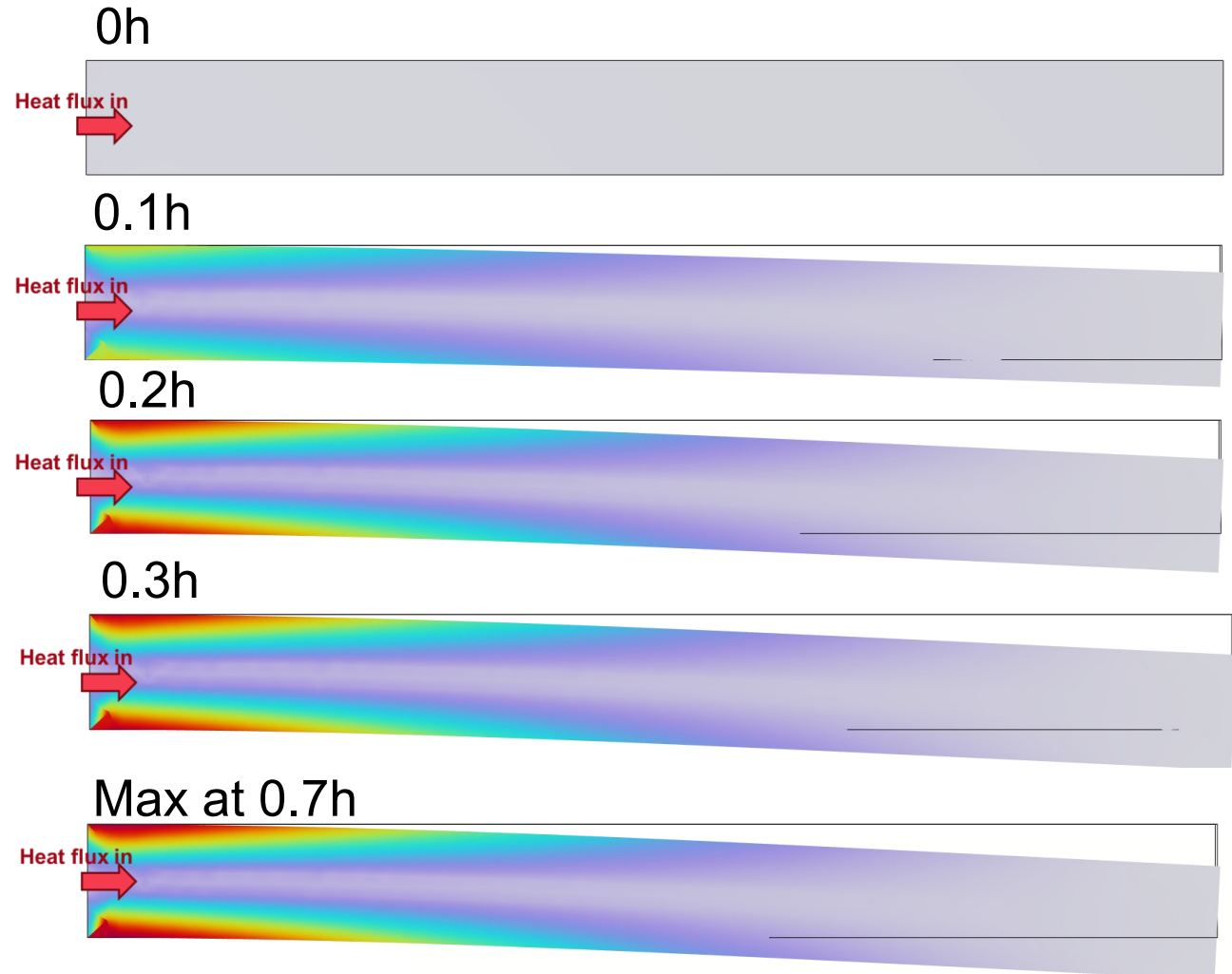


Then, you add “Multiphysics”

When E is a constant:



When E is temperature dependent:



Eventually we need to consider phase change

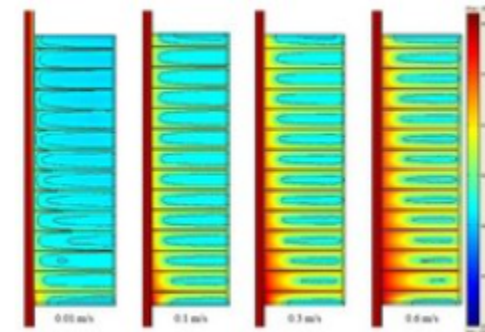
Solid-Liquid Phase Change Simulation Applied to a Cylindrical Latent Heat Energy Storage System

D. Groulx[1], and W. Ogoh[1]

Published in 2009

*[1]Mechanical Engineering Department, Dalhousie University,
Halifax, Nova Scotia, Canada*

One way of storing thermal energy is through the use of latent heat energy storage systems. One such system, composed of a cylindrical container filled with paraffin wax, through which a copper pipe carrying hot water is inserted, is presented in this paper. It is shown that the physical processes encountered in the flow of water, the heat transfer by conduction and convection, and the phase change behavior of the material can be modeled numerically using COMSOL Multiphysics.



COE COLLEGE