ABAQUS

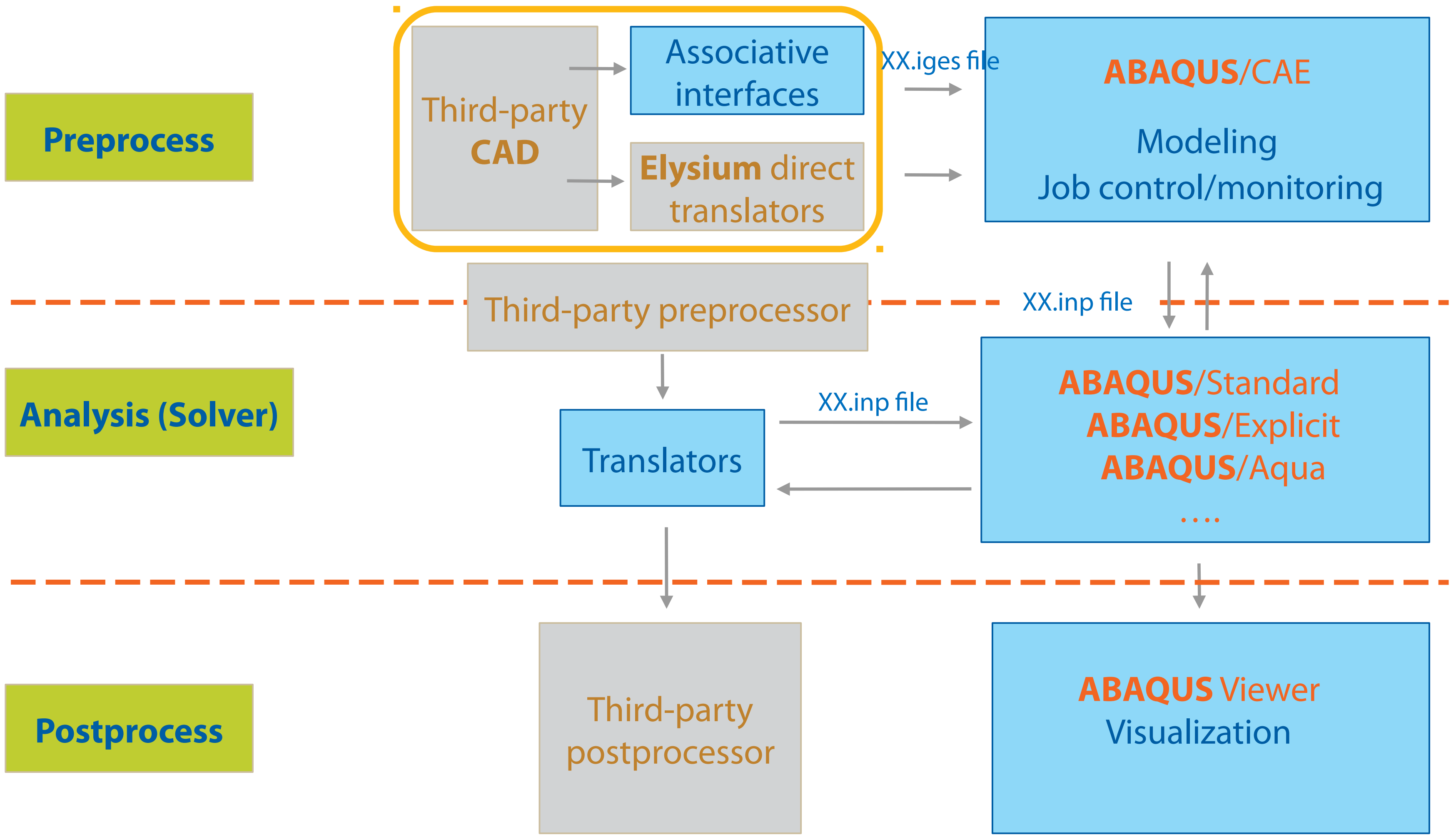
ME 498CM Fall 2015

Introduction & Workflow

Outline of Module

1. FEA Workflow (Redux), Postprocessing
2. Meshing
3. Loading & Analysis
4. Coupling Physics
5. Materials & Modeling
6. Fracture & Contact FEA
7. Dynamic FEA (Standard *v.* Explicit)
8. Batch Jobs & Scripting

ABAQUS Ecosystem

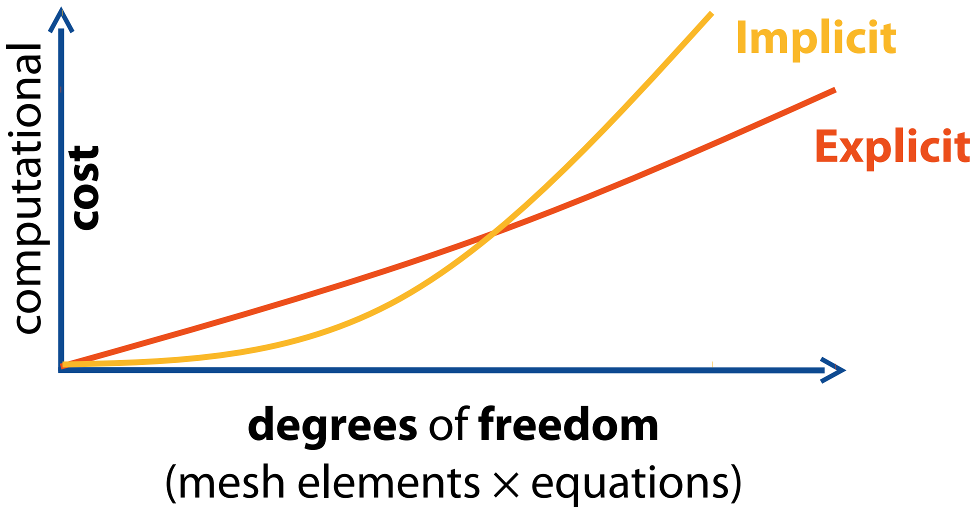


Comparison of **Implicit** and **Explicit**

Quantity	ABAQUS/Standard	ABAQUS/Explicit
Element library	Extensive	Subset
Analysis procedures	General & linear perturbation	General
Material models	Wide range of material models	Wide range + failure material models
Contact formulation	contact problems	complex contact problems
Solution technique	unconditionally stable stiffness-based solution technique	conditionally stable explicit integration solution technique
Disk space & memory	large with many iterations	small
Ideal Problem	smooth nonlinear problems etc.	brief transient dynamic events

Comparison of **Implicit** and **Explicit**

Cost of Degrees of Freedom Refinement



Implicit: computational cost proportional to square of degrees of freedom (actually f(connectivity))

Explicit: computational cost proportional to number of elements, inversely proportional to smallest element dimension

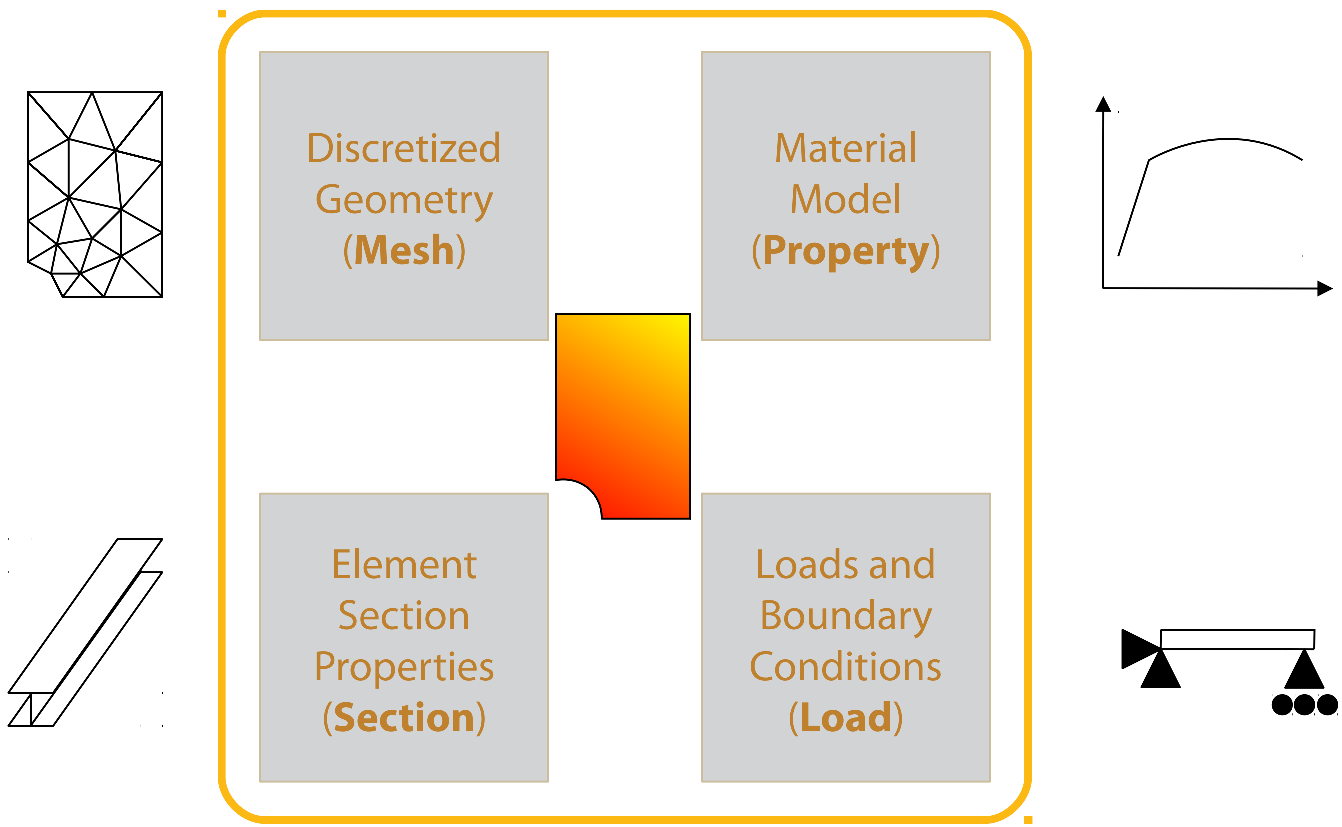
ABAQUS Workflow

Preprocessing	**ABAQUS**/CAE **Modules:** *Part, Property, Assemble, Step, Interaction, Load, Mesh*
	`inp` file
Simulation	**ABAQUS**/Standard **ABAQUS**/Explicit **Module:** *Job*
	odb, `fil`, `dat`, `res` files
Postprocessing	**ABAQUS**/CAE **Module:** *Visualization*
	png, `txt`, `csv` files, etc.

ABAQUS Workflow

1. Draw 2D **sketch** and create 3D **part**s.
2. Assign **Material** and **Section** property.
3. **Assemble** the model; give **interaction**s in form.
4. **Mesh** the frame.
5. Apply **Load** and boundary conditions.
6. Create **job** and configure output requests.
7. Submit it for **analysis** (Standard/Explicit).
8. **Visualize** the results of analysis.

ABAQUS Preprocessing



ABAQUS Solvers

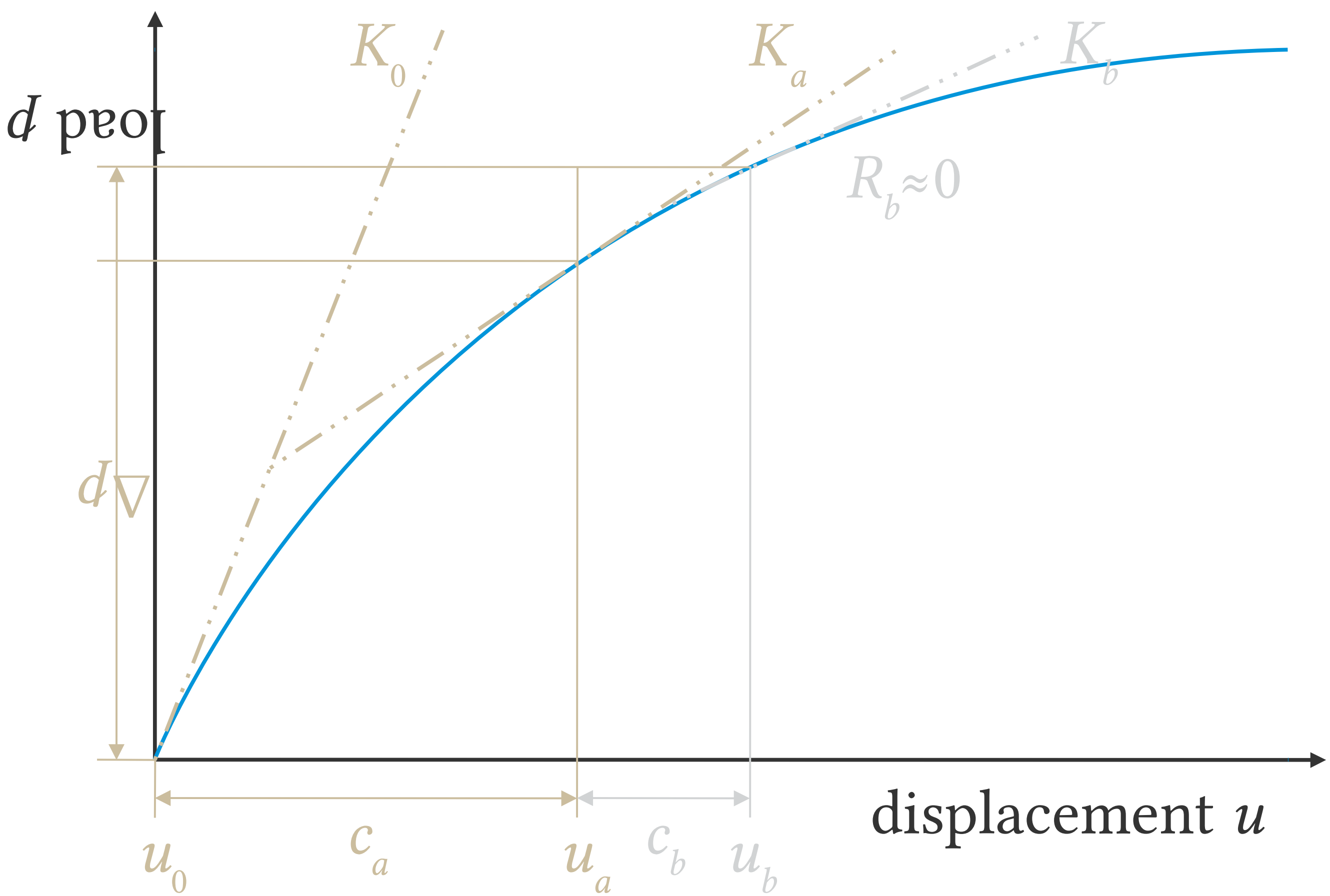
- **ABAQUS/Standard**

  Solves system of equations **implicitly** at each solution “increment”.

- **ABAQUS/Explicit**

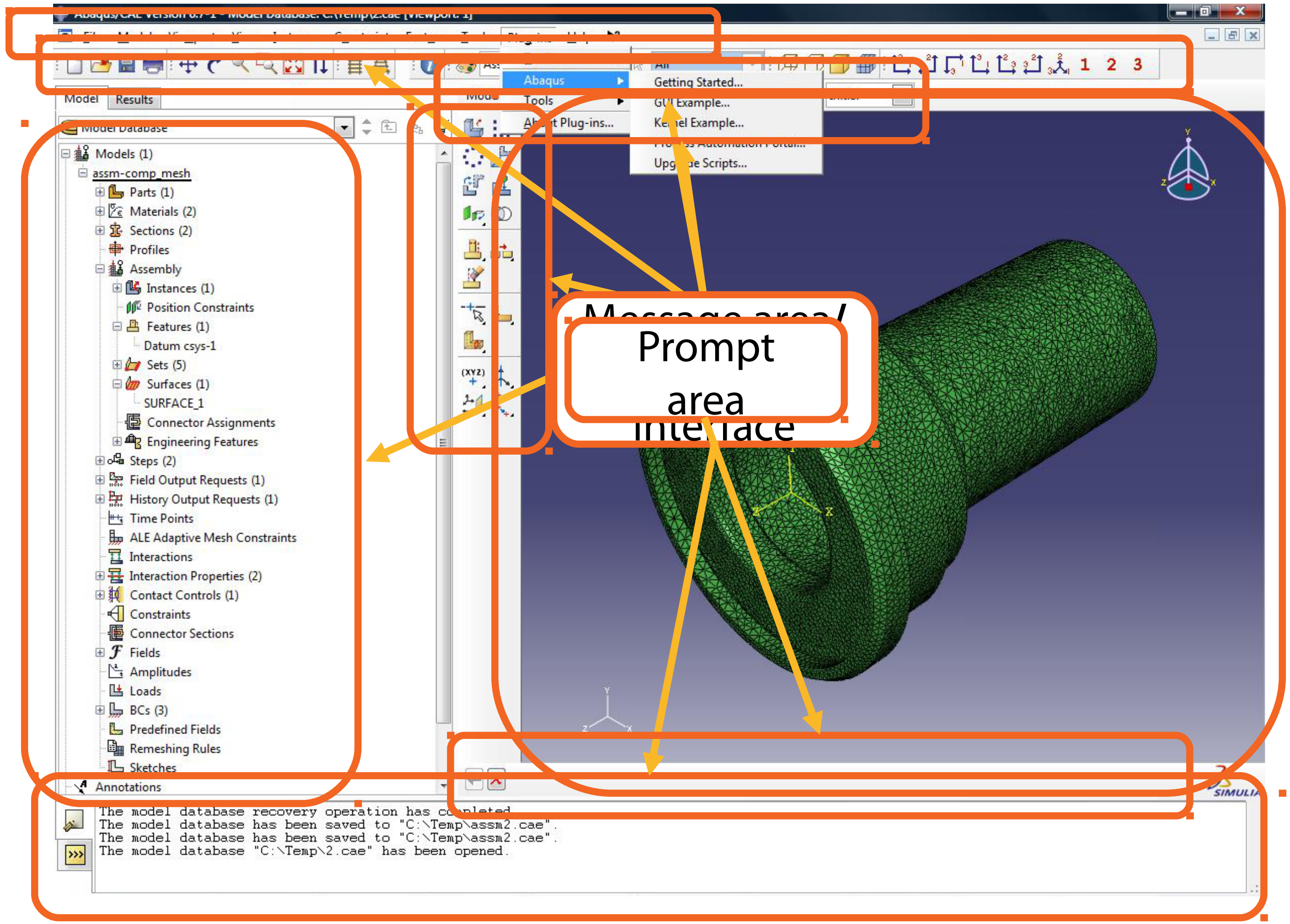
  Marches solution forward through time **explicitly** in small time increments *without* solving coupled system of equations at each increment.

ABAQUS Solvers—Standard



ABAQUS/CAE

- Modeling
  - Geometry
  - Material Property
  - Mesh
  - Load & BC
  - Job manage
- Result Viewing



ABAQUS Units

- ABAQUS has no built-in units
- Specify all input data in consistent units

m	l	t	F	σ	E
kg	m	s	N	Pa	J
kg	cm	s	10^{-2} N		
kg	cm	ms	10^{4} N		
kg	cm	µs	10^{10} N		
kg	mm	ms	kN	GPa	kN·mm
g	cm	s	dyne	dyne·cm^{-2}	erg
g	cm	µs	10^{7} N	Mbar	10^{7} N·cm
g	mm	s	10^{-6} N	Pa	
g	mm	ms	N	MPa	N·mm
ton	mm	s	N	MPa	N·mm
$lb_f \cdot s^2 \cdot in^{-1}$	in	s	lb_f	psi	$lb_f \cdot in$
slug	ft	s	lb_f	psf	$lb_f \cdot ft$
$kg_f \cdot s^2 \cdot mm^{-1}$	mm	s	kg_f	$kg_f \cdot mm^{-2}$	$kg_f \cdot mm$
kg	mm	s	mN	kPa	
g	cm	ms	10^{1} N	10^{5} Pa	

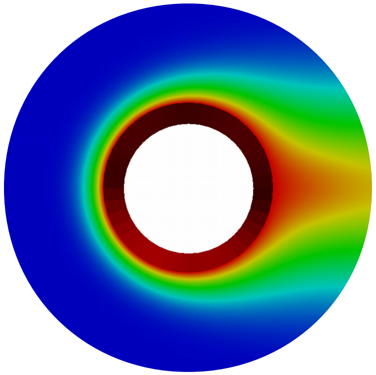
Suggested FEM **Courses**

ME 471—Introduction to Finite Element Analysis
ME 570—Nonlinear Solid Mechanical Design

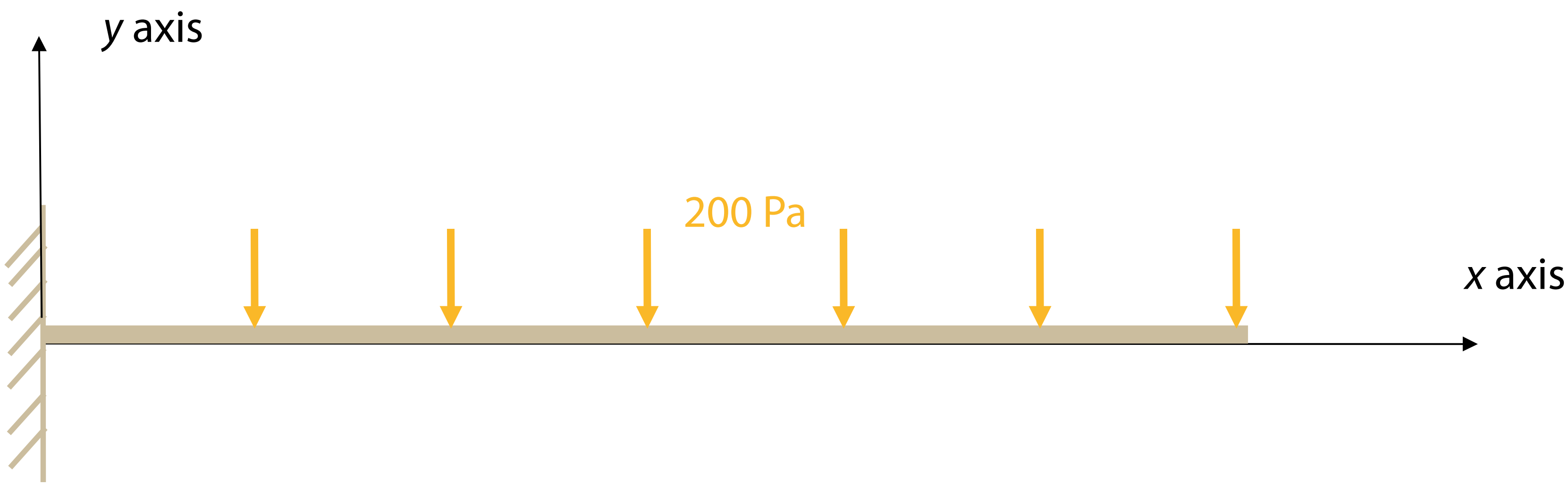
CEE 470—Structural Analysis
CEE 570—Finite Element Methods
CEE 576—Nonlinear Finite Elements

CS 555—Numerical Methods for PDES

TAM 574—Advanced Finite Element Methods



Beam Exercise #1



Steel beam with continuous loading of $\boldsymbol{F} = 200$ Pa

$\boldsymbol{E} = 2 \times 10^{11}$ Pa — Young's modulus

$\boldsymbol{\nu} = 0.26$ — Poisson's ratio

$\boldsymbol{L} \times \boldsymbol{W} \times \boldsymbol{H} = (200 \text{ cm}) \times (20 \text{ cm}) \times (20 \text{ cm})$ — Dimensions

Beam Exercise #2

Steel beam with end load of 20,000 Pa at $\boldsymbol{x}_L$ = 200 cm

$\boldsymbol{E} = 2 \times 10^{11}$ Pa	Young's modulus
$\boldsymbol{\nu} = 0.26$	Poisson ratio
$\boldsymbol{\rho} = 7.8 \times 10^{3}$	Density
$\boldsymbol{L}$ = 200 cm	Length
⌀ 20 cm