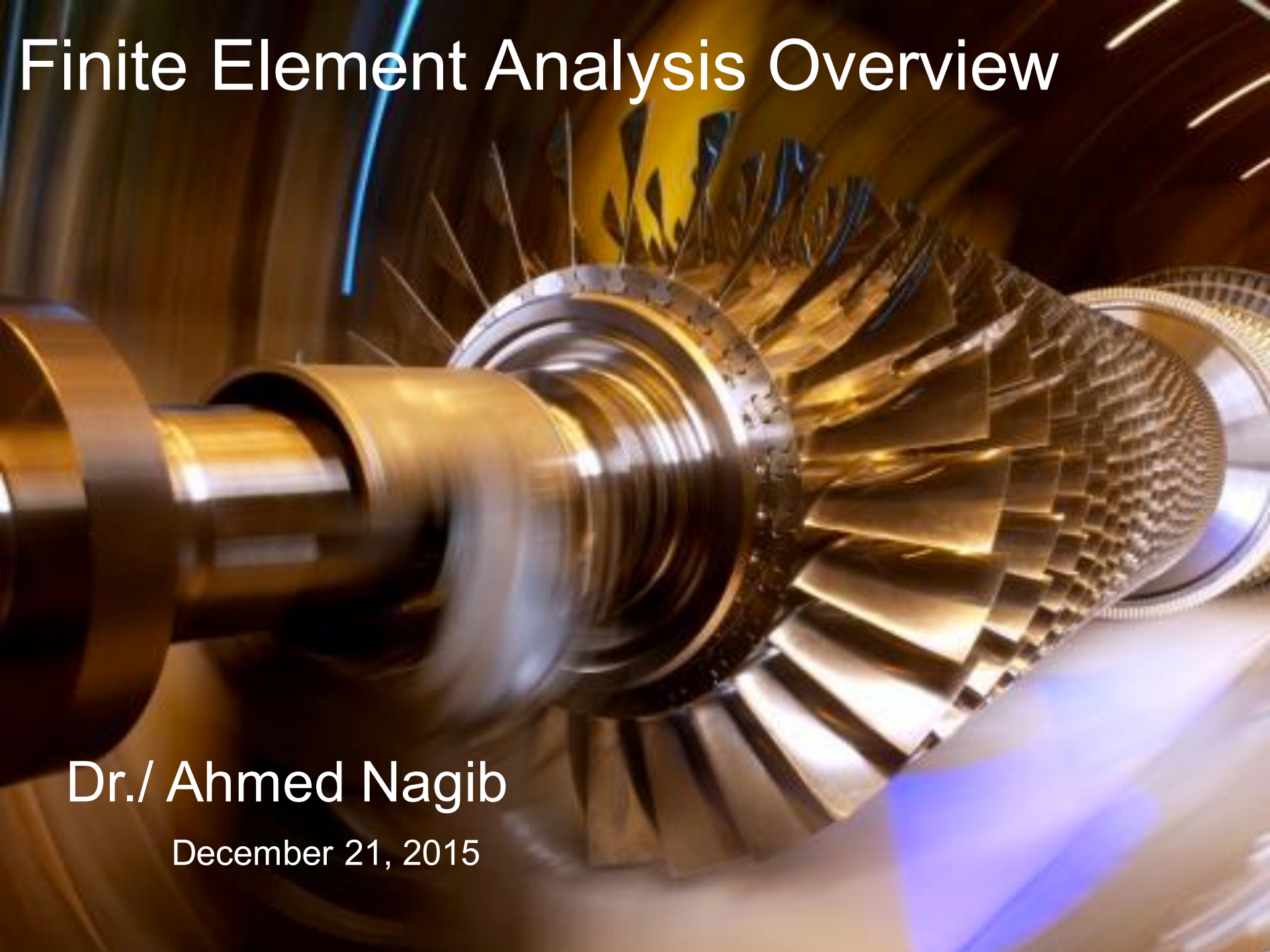
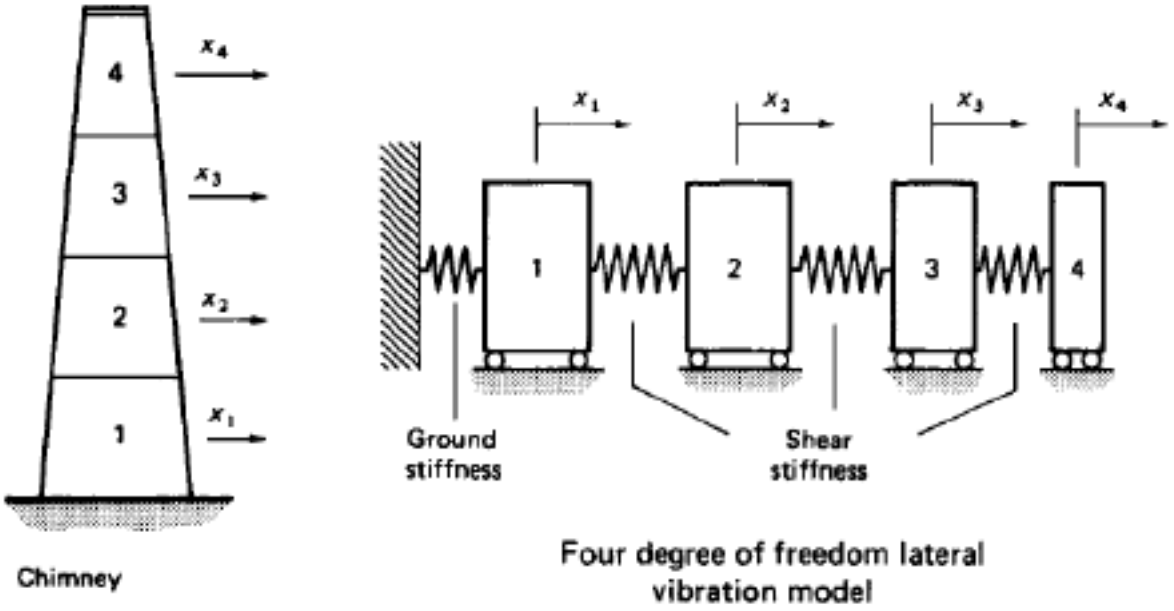
# Finite Element Analysis Overview



Dr./ Ahmed Nagib

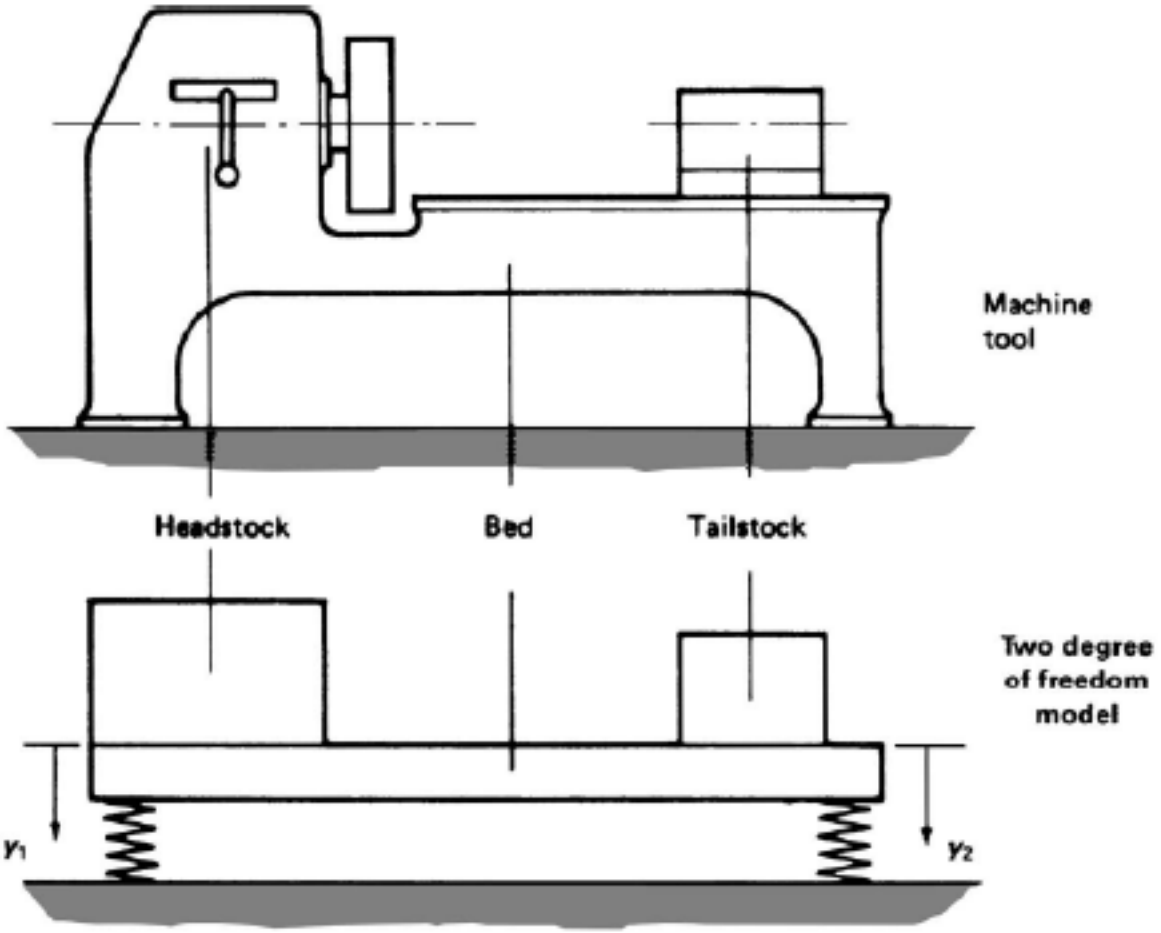
December 21, 2015

# Finite Element Analysis



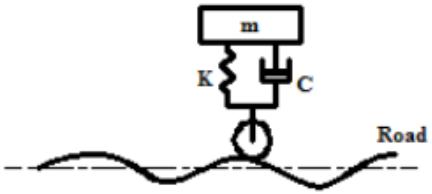

Chimney vibration analysis model.

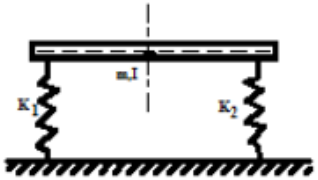
# Finite Element Analysis

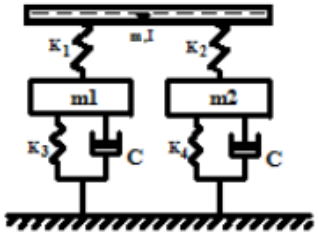


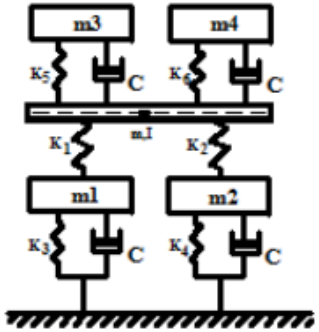

Machine tool vibration analysis model.

# Finite Element Analysis




Mathematical models of a car

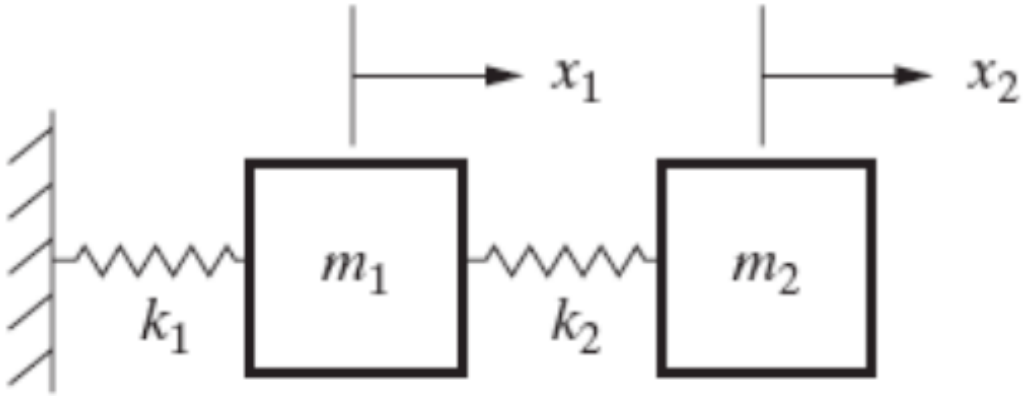
# Finite Element Analysis

**The linear equation of motion for vibration is**

$$[M]\{\ddot{x}\} + [C]\{\dot{x}\} + [K]\{x\} = \{F\}$$

- $\{x\}$ , which is the displacement vector
- $\{\ddot{x}\}$ , which is the acceleration vector
- $[M]$, which is the Mass matrix
- $[K]$, which is the stiffness matrix
- $[C]$, which is the damping matrix
- $\{F\}$ , which is the load vector

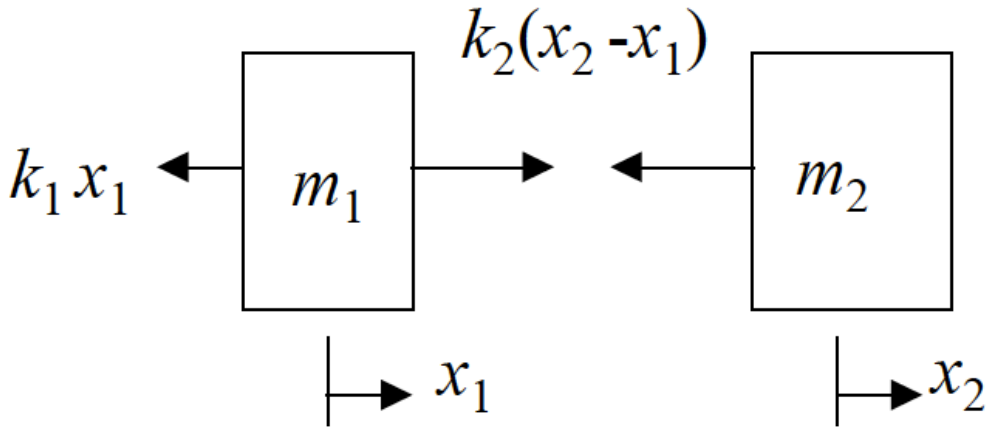
# Finite Element Analysis




A 2 degree of freedom system used to base much of the analysis and conceptual development of MDOF systems on.

# Finite Element Analysis

## Free-Body Diagram of each mass



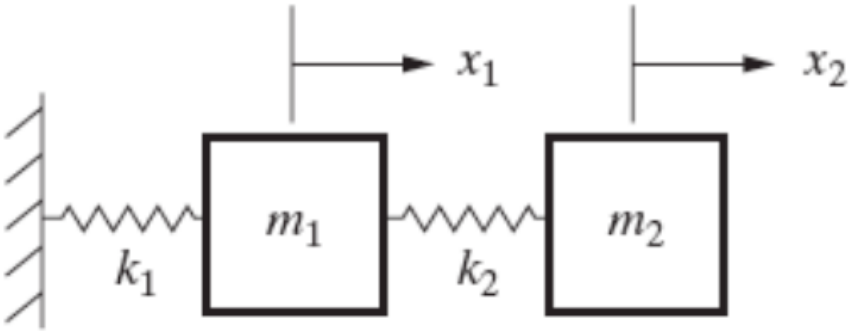
# Finite Element Analysis

**The linear equation of motion for vibration is**

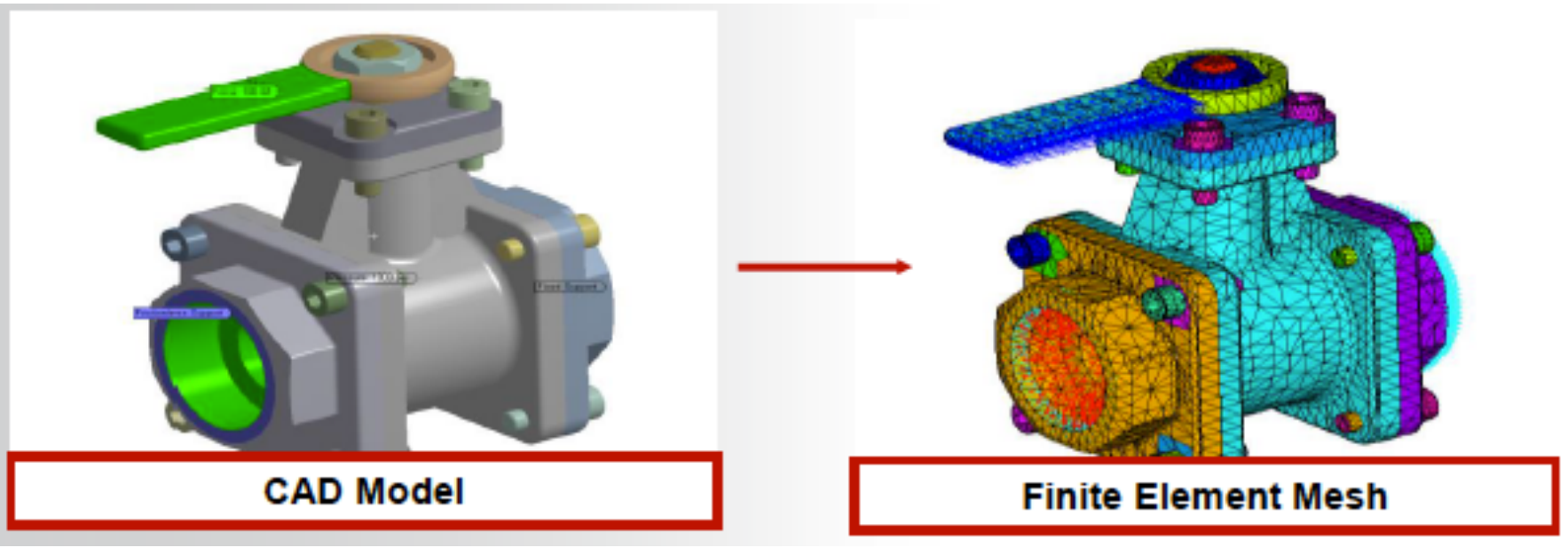
$$[\boldsymbol{M}]\{\ddot{\boldsymbol{x}}\} + [\boldsymbol{C}]\{\dot{\boldsymbol{x}}\} + [\boldsymbol{K}]\{\boldsymbol{x}\} = \{\boldsymbol{F}\}$$

$$\begin{bmatrix} m_1 & 0 \\ 0 & m_2 \end{bmatrix}\{\ddot{\boldsymbol{x}}\} + \begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}\{\boldsymbol{x}\} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

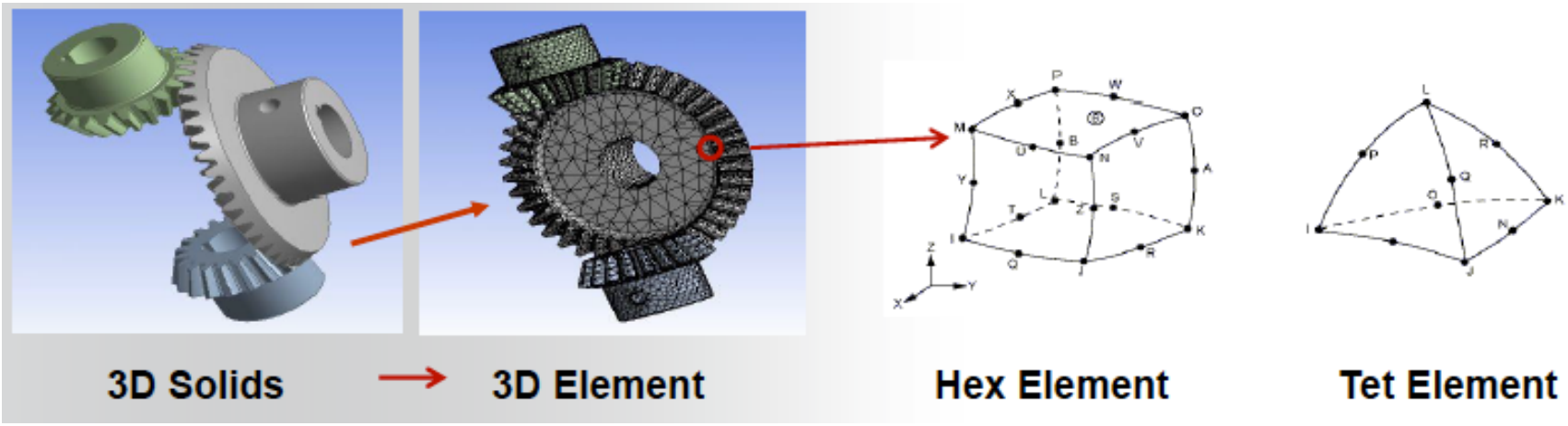
$$\begin{bmatrix} \frac{1}{3} & \frac{1}{3} \\ 1 & -1 \end{bmatrix}\{\ddot{\boldsymbol{x}}\} + \begin{bmatrix} \frac{1}{3} & \frac{1}{3} \\ 1 & -1 \end{bmatrix}\{\boldsymbol{x}\} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$



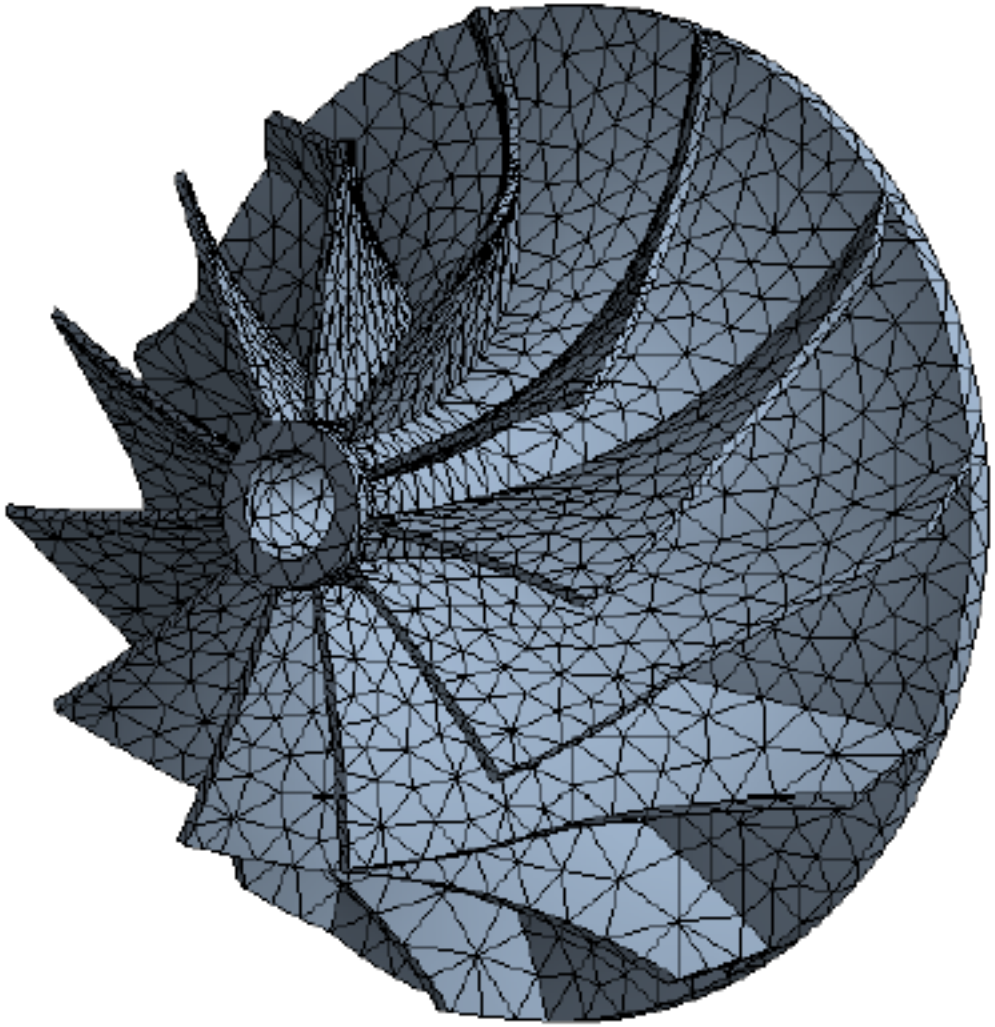
# Finite Element Analysis



# Finite Element Analysis



# Finite Element Analysis



# Static Analysis

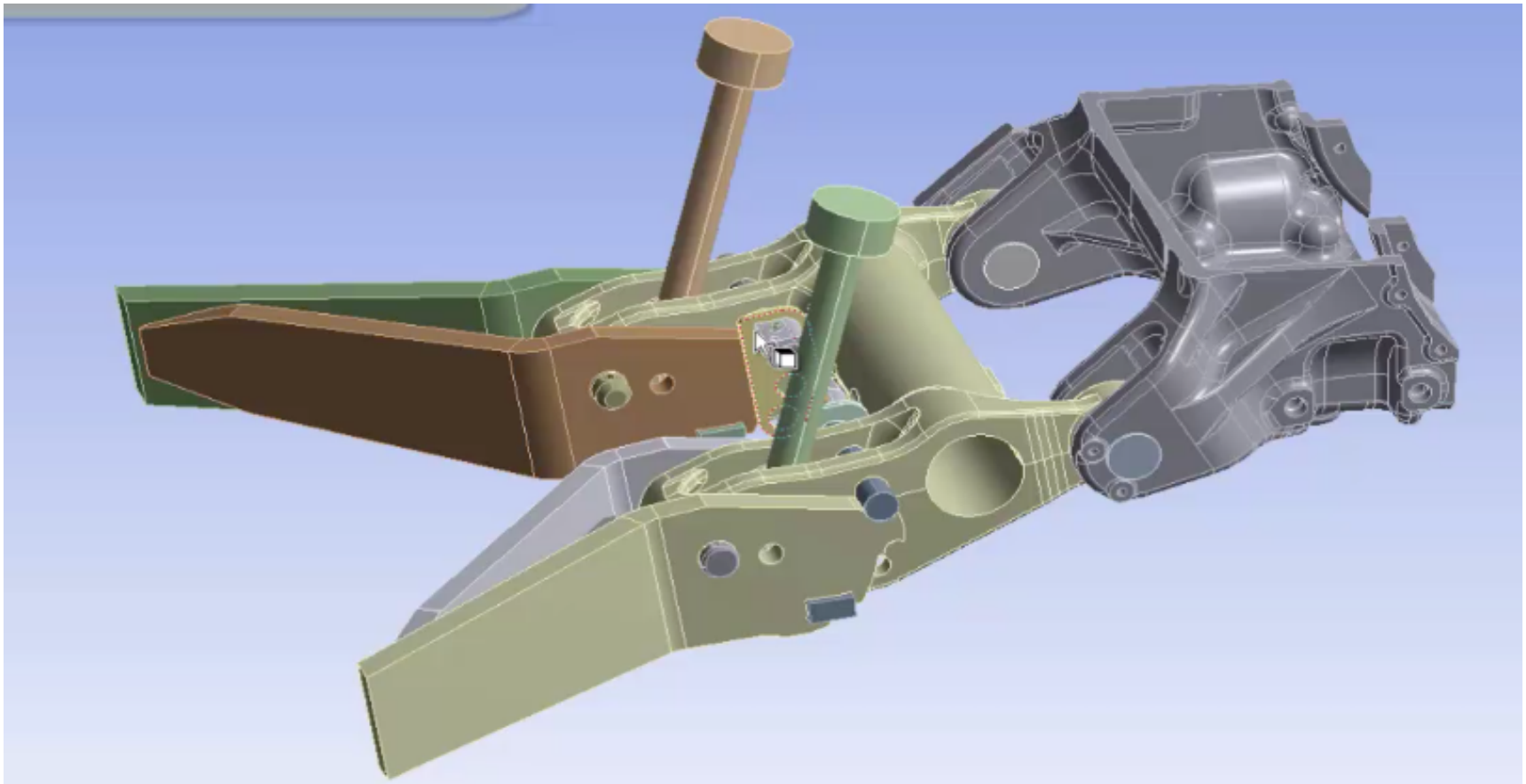
**For a linear static structural analysis, the global displacement vector $\{x\}$ is solved for in the matrix equation below:**

$$[K]\{x\} = \{F\}$$

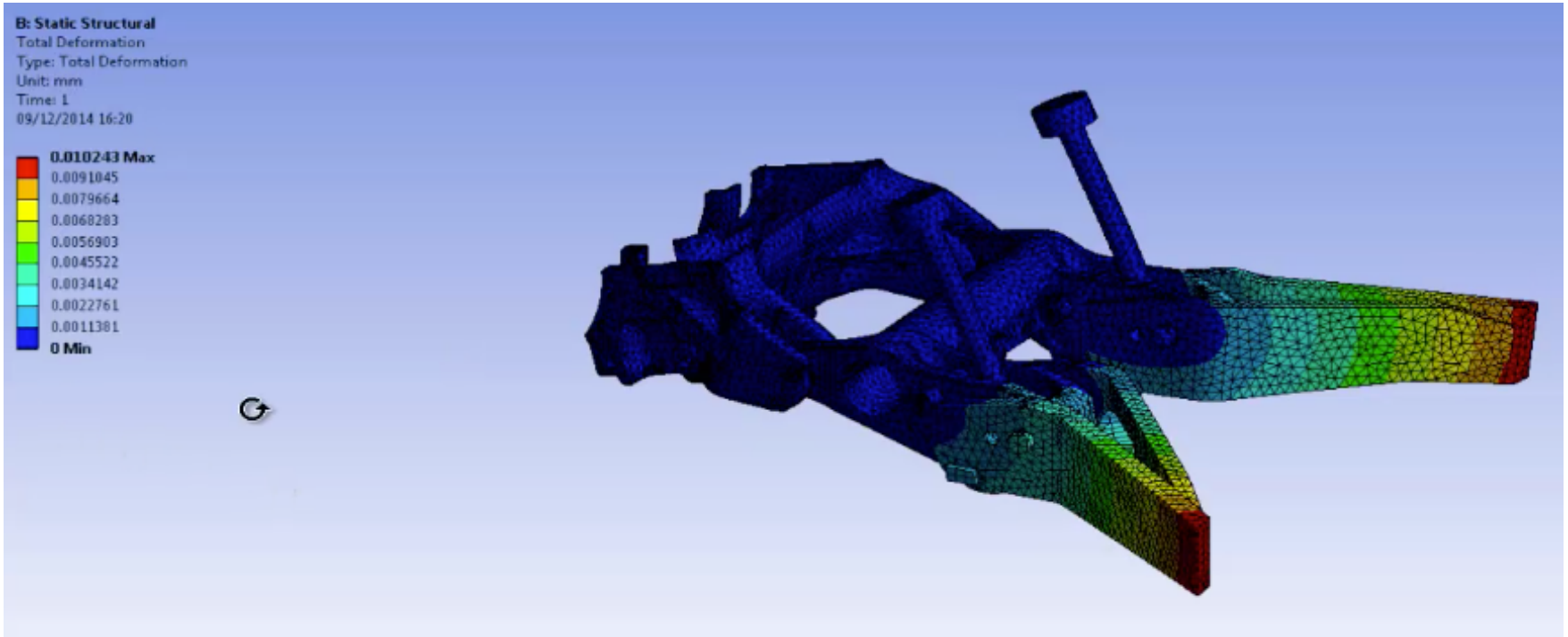
**Assumptions made for linear static structural analysis are:**

- **$[K]$, which is the global stiffness matrix, is constant**
  - **Linear elastic material behavior is assumed**
  - **Small deflection theory is used**
- **$\{F\}$ , which is the global load vector, is statically applied**
  - **No time-varying forces are considered**
  - **No damping effects**

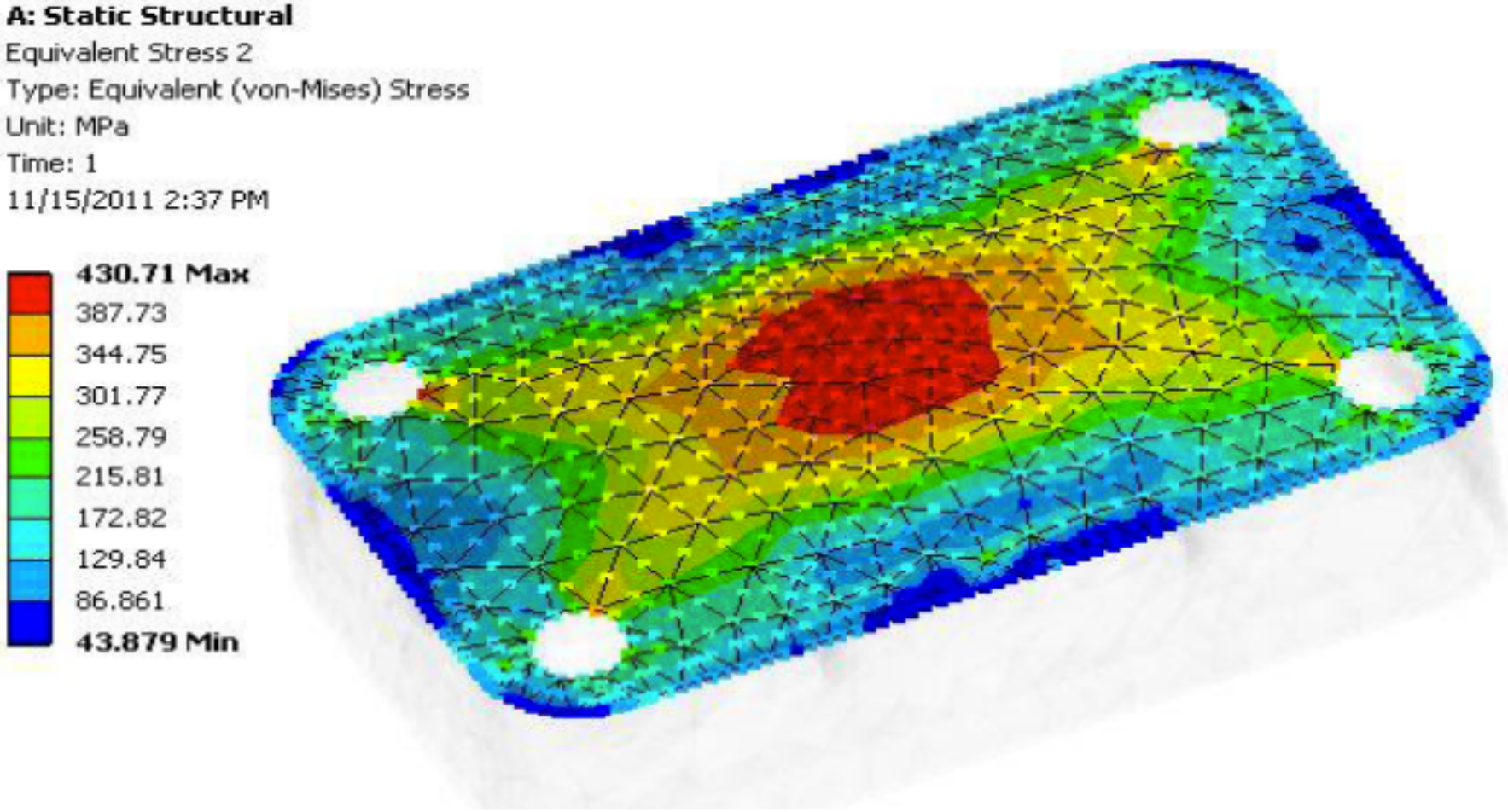
# Finite Element Analysis



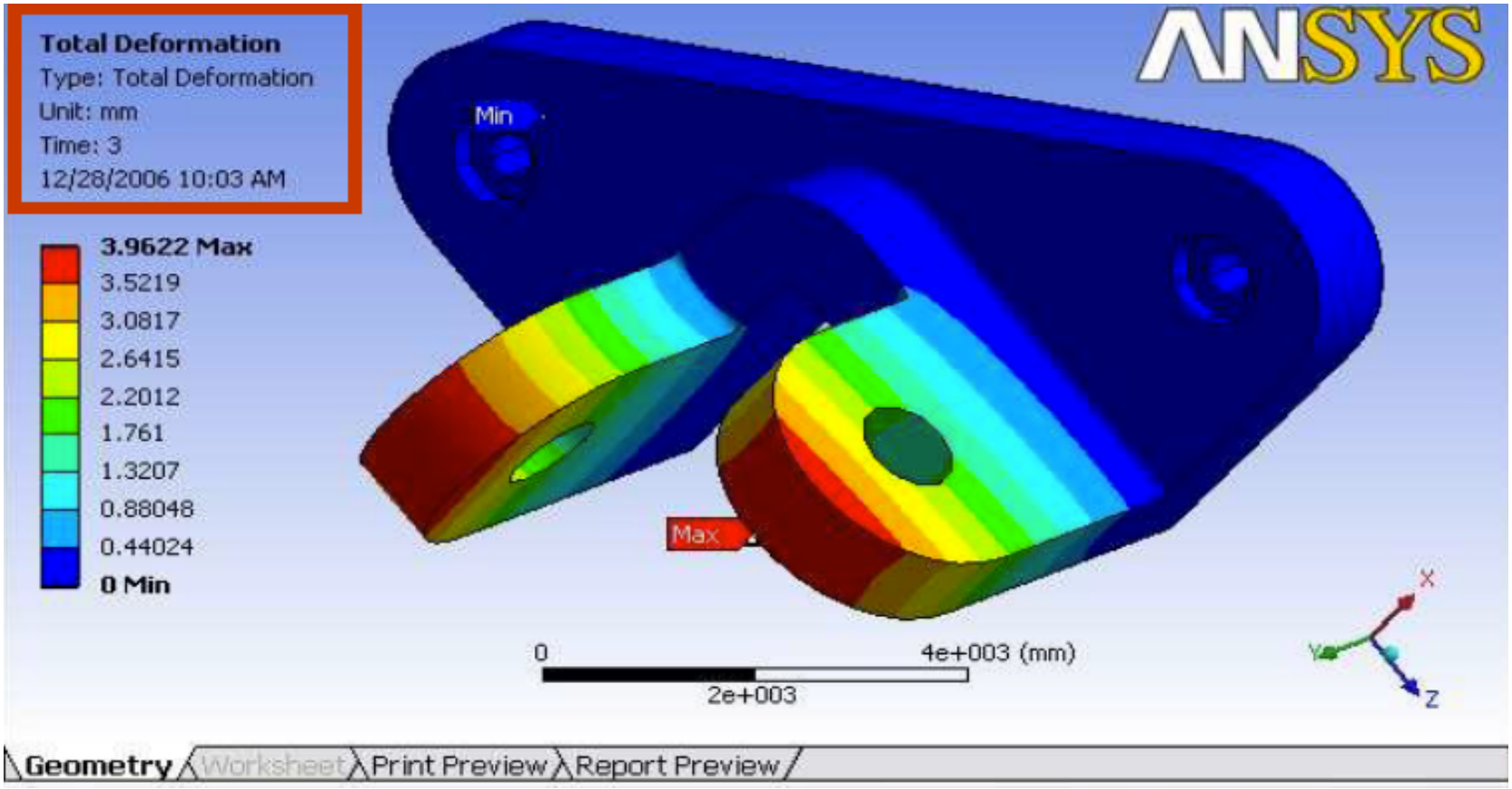
# Finite Element Analysis

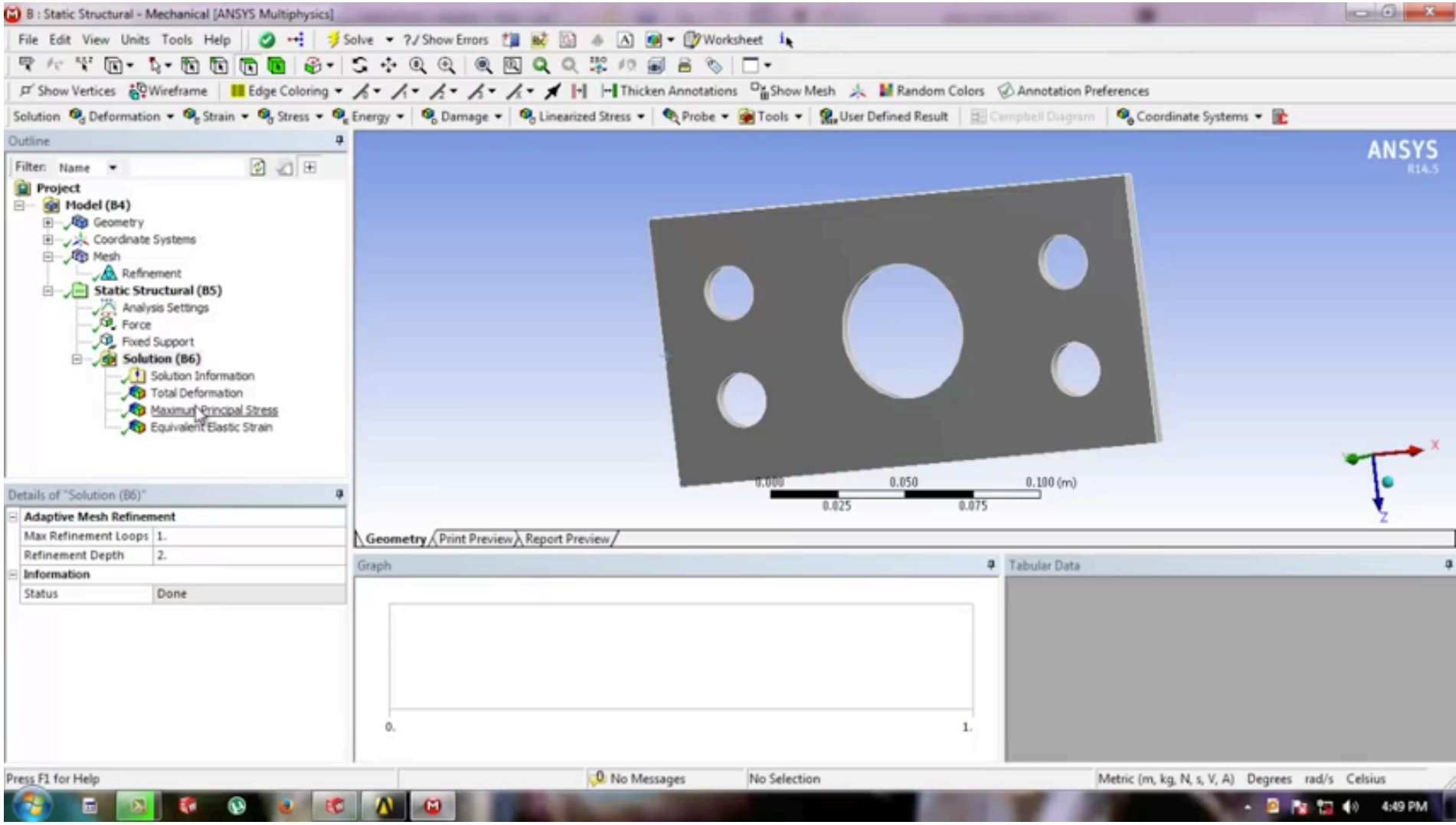


# Static Analysis

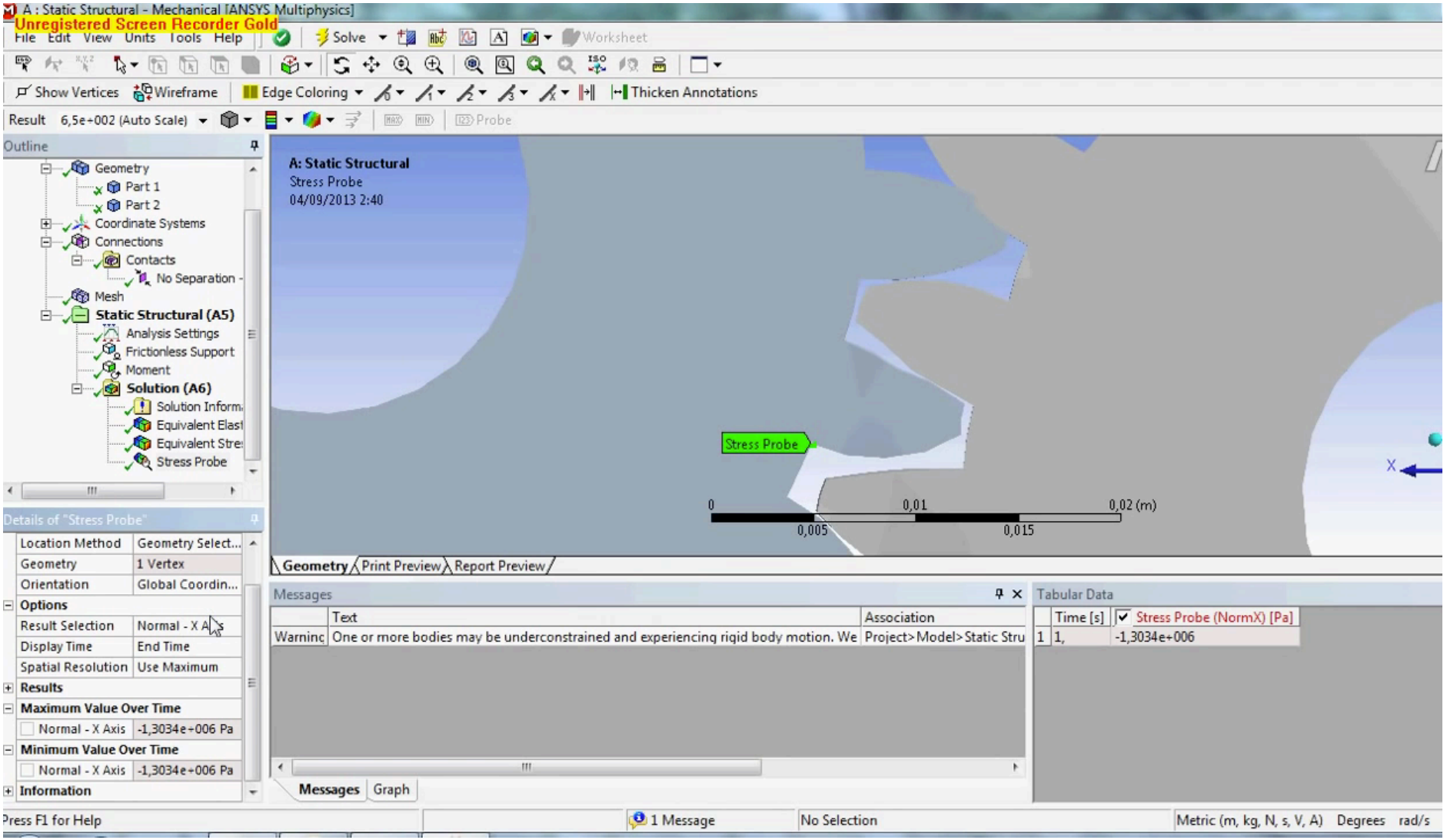


# Static Analysis

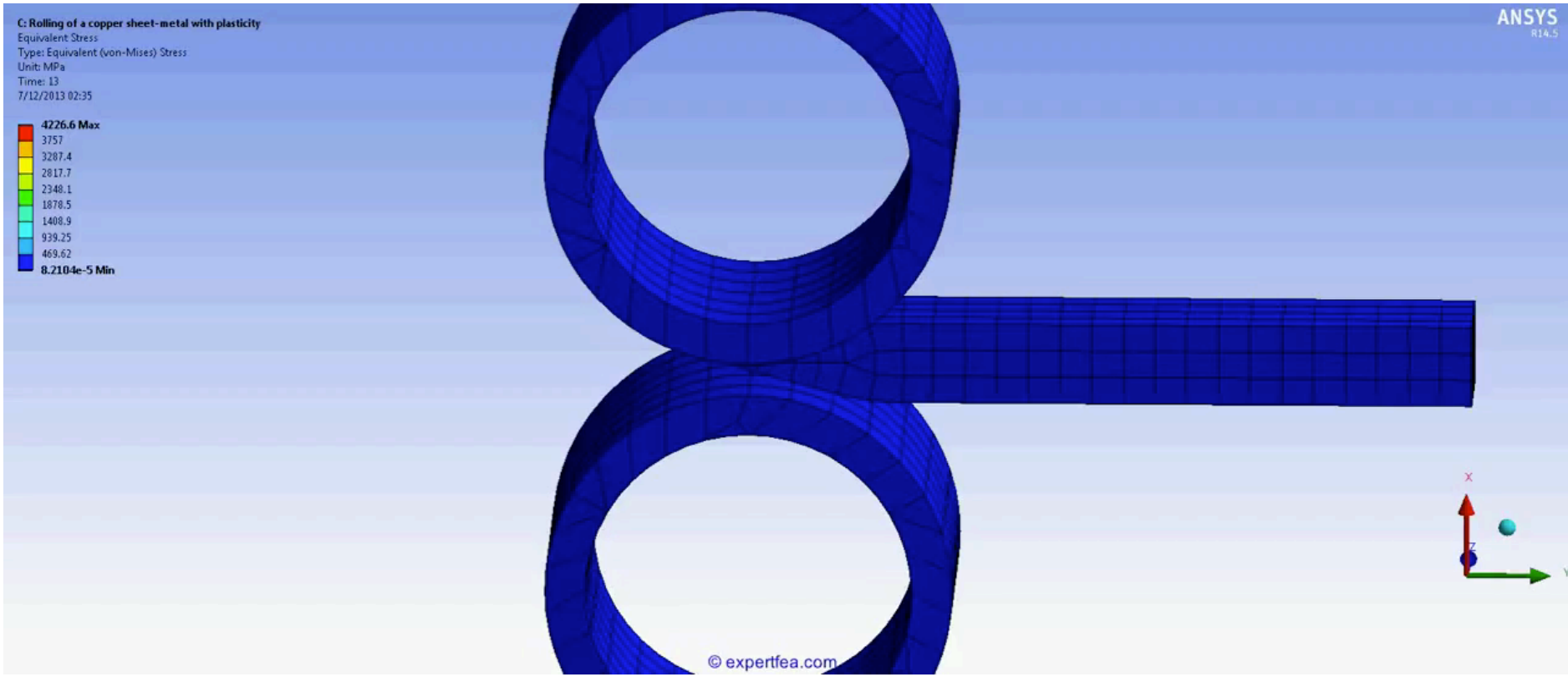



B : Static Structural - Mechanical [ANSYS Multiphysics]
File Edit View Units Tools Help
Solve
Show Errors
Worksheet
Show Vertices
Wireframe
Edge Coloring
Thicken Annotations
Show Mesh
Random Colors
Annotation Preferences
Solution
Deformation
Strain
Stress
Energy
Damage
Linearized Stress
Probe
Tools
User Defined Result
Campbell Diagram
Coordinate Systems
Outline
Filter: Name
Project
Model (B4)
Geometry
Coordinate Systems
Mesh
Refinement
Static Structural (B5)
Analysis Settings
Force
Fixed Support
Solution (B6)
Solution Information
Total Deformation
Maximum Principal Stress
Equivalent Elastic Strain
Details of "Solution (B6)"
Adaptive Mesh Refinement
Max Refinement Loops 1.
Refinement Depth 2.
Information
Status Done
ANSYS
R14.5
0.000
0.050
0.100 (m)
0.025
0.075
X
Z
Geometry
Print Preview
Report Preview
Graph
0.
1.
Tabular Data
Press F1 for Help
No Messages
No Selection
Metric (m, kg, N, s, V, A) Degrees rad/s Celsius
4:49 PM

# Static Analysis



# Static Analysis

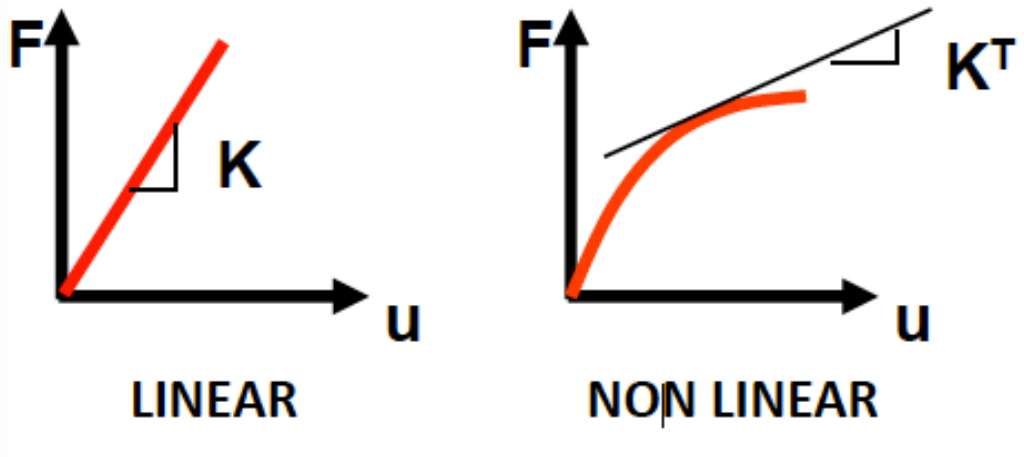


# Static Analysis

## Linear vs Non Linear solve

In a linear analysis, the matrix equation [K]{x}={F} is solved in one iteration. That means the model stiffness does not change during solve : [K] is constant. A non linear solve allow stiffness changes and uses an iterative process to solve the problem. In a static structural analysis, ANSYS runs a non linear solve automatically when the model contains :

- Non linear material laws : Plasticity, Creep, Gasket, Viscoelasticity …
- Non linear contact : Frictionless, Rough, Frictional
- Large deflection turned <<ON>>
- Joints
- Bolt pretension



# Modal Analysis

**The linear equation of motion for free, un-damped vibration is**

$$[\boldsymbol{M}]\{\ddot{\boldsymbol{x}}\} + [\boldsymbol{K}]\{\boldsymbol{x}\} = \{\boldsymbol{0}\}$$

**Assume harmonic motion:**

$$\{\boldsymbol{x}\} = \{\boldsymbol{\phi}\}_{\boldsymbol{i}} \sin(\omega_i t + \theta_i)$$
$$\{\ddot{\boldsymbol{x}}\} = -\omega_{\boldsymbol{i}}^{\boldsymbol{2}}\{\boldsymbol{\phi}\}_{\boldsymbol{i}} \sin(\omega_i t + \theta_i)$$

**Substituting $\{x\}$ and $\{\ddot{x}\}$ in the governing equation gives an eigenvalue equation:**

$$\left(-\omega_{\boldsymbol{i}}^{\boldsymbol{2}}[\boldsymbol{M}] + [\boldsymbol{K}]\right)\{\boldsymbol{\phi}\}_{\boldsymbol{i}} = \{\boldsymbol{0}\}$$

**where**

**$\omega_i$: Natural Frequencies**

**$\{\boldsymbol{\phi}\}_{\boldsymbol{i}}$ : Mode Shapes**

# Modal Analysis

## Assumptions for Modal Analysis

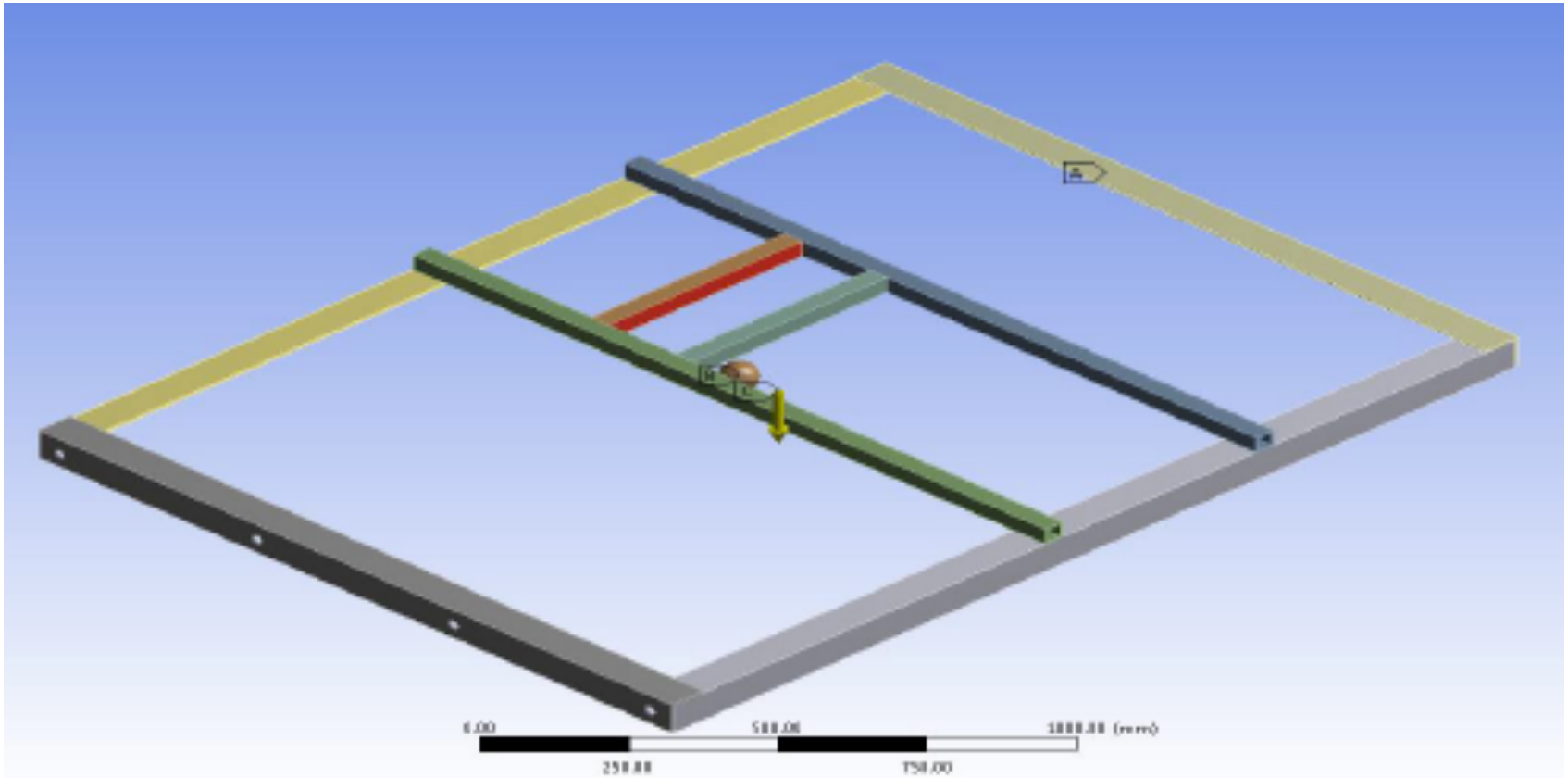
- [K] and [M] are constant:
  - Linear elastic material behavior is assumed
  - Small deflection theory is used, and *no* nonlinearities included
  - [C] is not present, so damping is not included
  - {F} is not present, so no excitation of the structure is assumed
  - Mode shapes $\{\phi\}_i$ are *relative* values, not absolute

# Modal Analysis

**Modal Results:**

- **Because there is no *excitation* applied to the structure the mode shapes are *relative* values not actual ones.**
  - **Because a modal result is based on the model's properties and not a particular input, we can interpret where the maximum or minimum results will occur for a particular mode shape but not the actual value.**

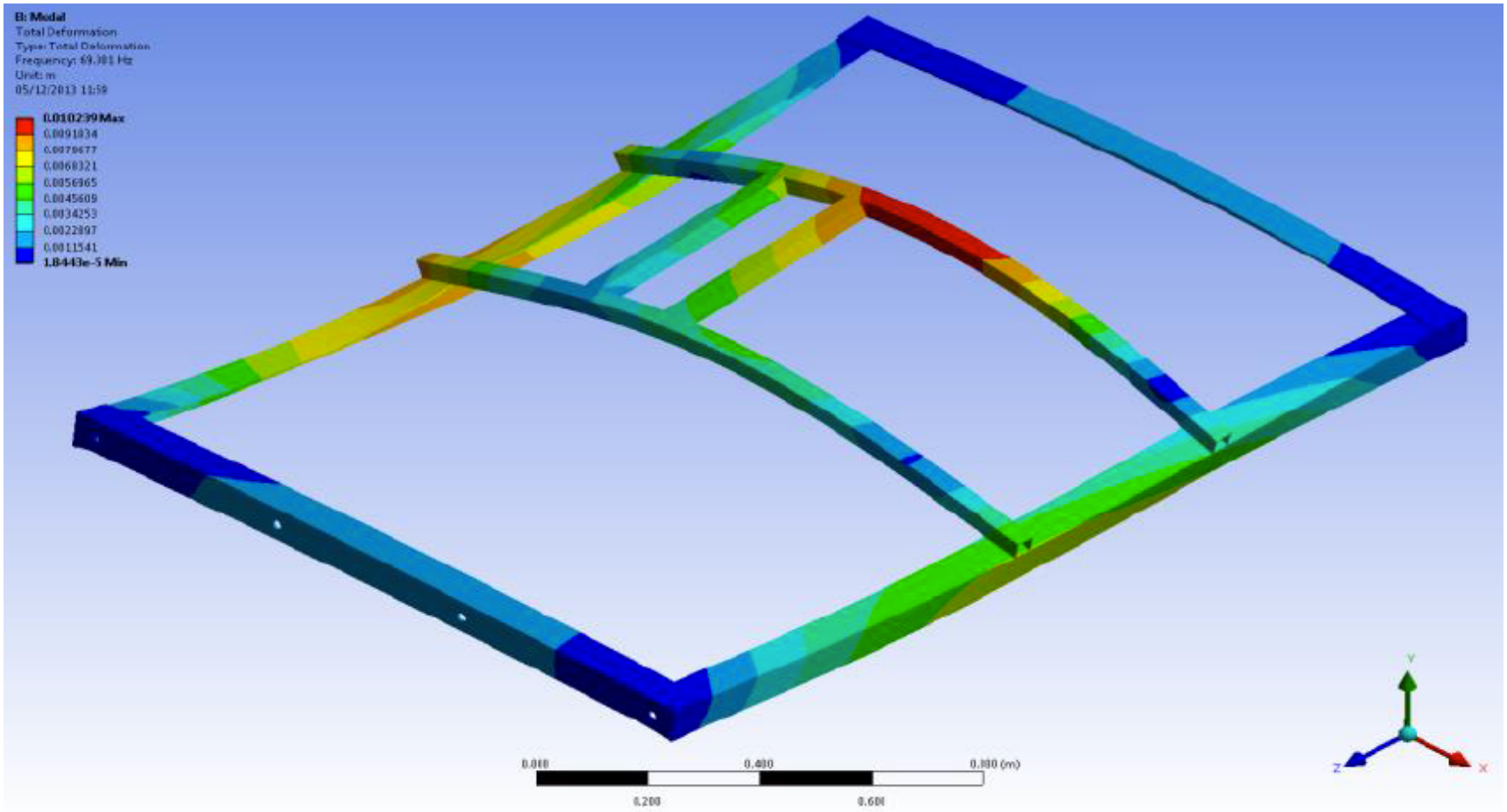
# Modal Analysis



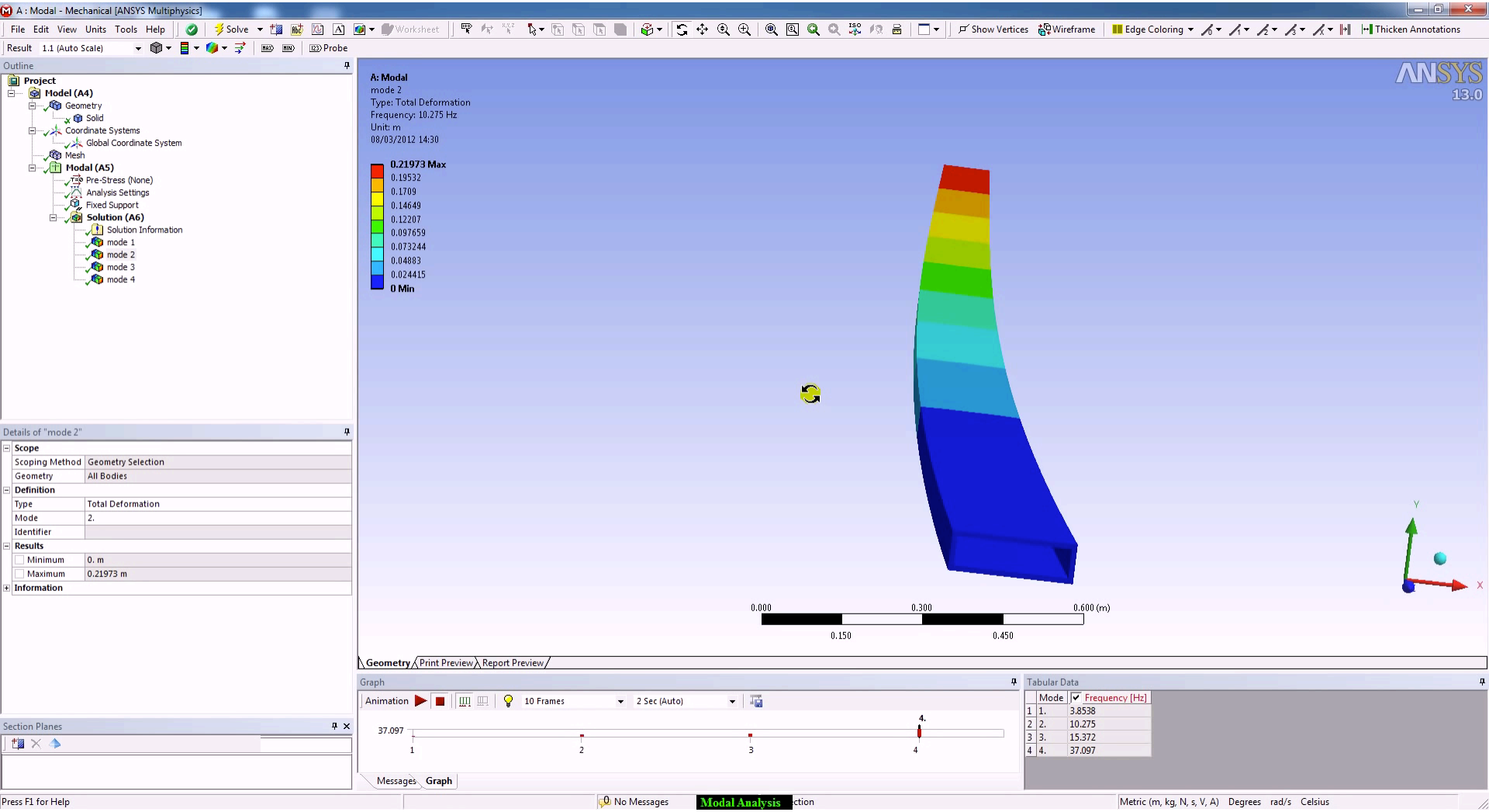
# Modal Analysis

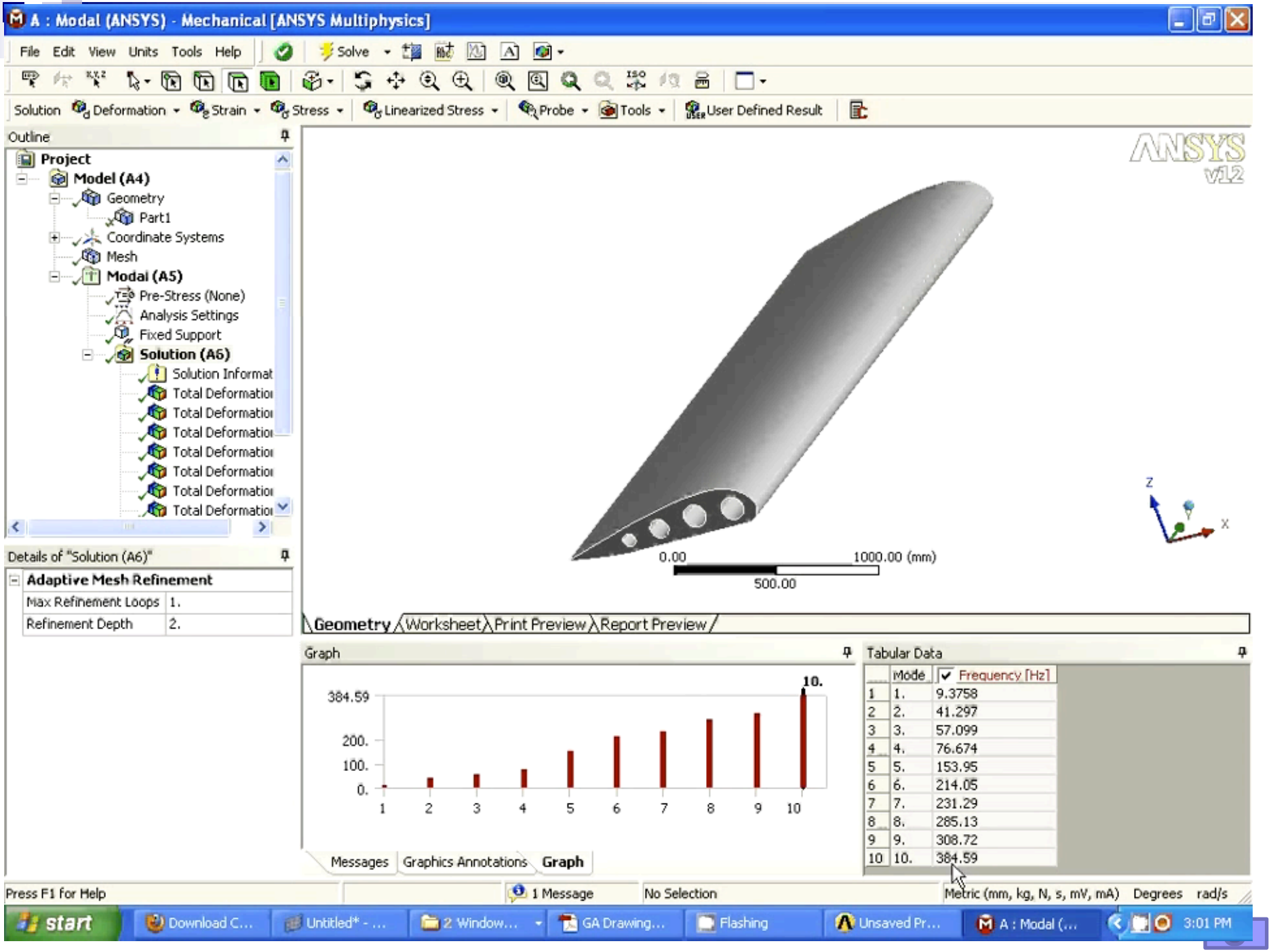
| | Mode | Frequency [Hz] |
|---|---|---|
| 1 | 1. | 31.582 |
| 2 | 2. | 39.69 |
| 3 | 3. | 65.477 |
| 4 | 4. | 69.164 |
| 5 | 5. | 80.171 |
| 6 | 6. | 111.06 |

# Modal Analysis



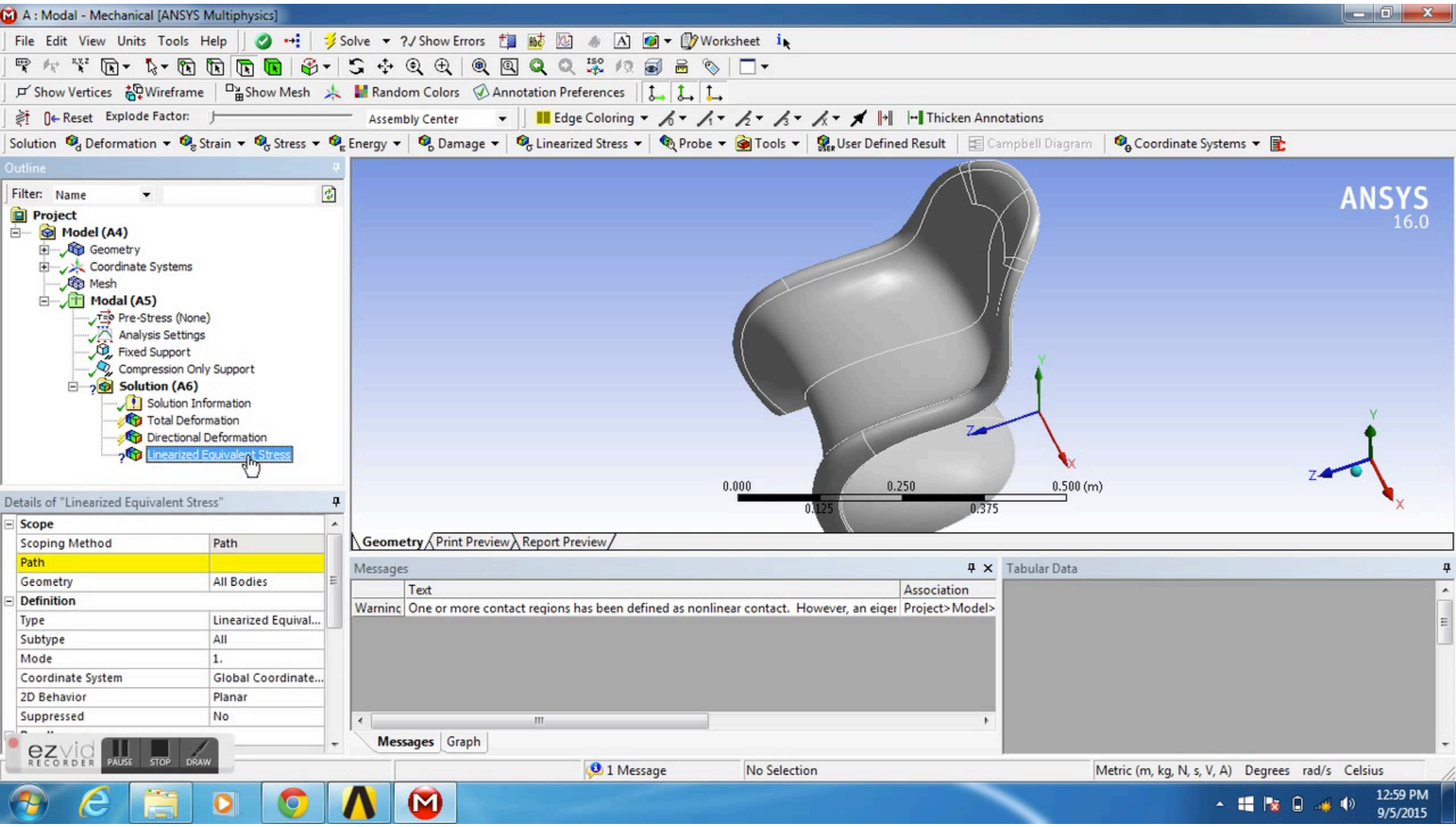
# Modal Analysis




A : Modal (ANSYS) - Mechanical [ANSYS Multiphysics]
File Edit View Units Tools Help
Solve
Solution Deformation Strain Stress Linearized Stress Probe Tools User Defined Result
Outline
Project
Model (A4)
Geometry
Part1
Coordinate Systems
Mesh
Modal (A5)
Pre-Stress (None)
Analysis Settings
Fixed Support
Solution (A6)
Solution Informat
Total Deformatio
Total Deformatio
Total Deformatio
Total Deformatio
Total Deformatio
Total Deformatio
Total Deformatio
ANSYS
v12
Z
X
0.00
1000.00 (mm)
500.00
Details of "Solution (A6)"
Adaptive Mesh Refinement
Max Refinement Loops 1.
Refinement Depth 2.
Geometry Worksheet Print Preview Report Preview
Graph
10.
384.59
200.
100.
0.
1 2 3 4 5 6 7 8 9 10
Messages Graphics Annotations Graph
Tabular Data
Mode Frequency [Hz]
1 1. 9.3758
2 2. 41.297
3 3. 57.099
4 4. 76.674
5 5. 153.95
6 6. 214.05
7 7. 231.29
8 8. 285.13
9 9. 308.72
10 10. 384.59
Press F1 for Help
1 Message
No Selection
Metric (mm, kg, N, s, mV, mA) Degrees rad/s
start
Download C...
Untitled* - ...
2 Window...
GA Drawing...
Flashing
Unsaved Pr...
A : Modal (...
3:01 PM

# Modal Analysis

# Modal Analysis



# Dynamic Analysis

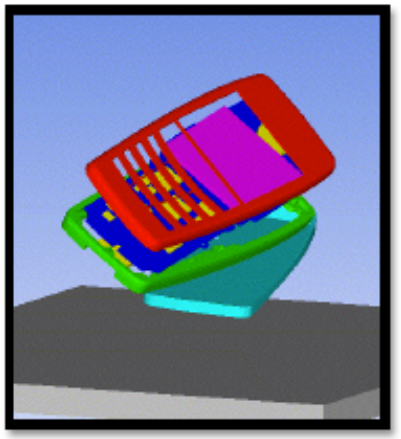
**Typical Solid Response Behavior at Different VELOCITY**

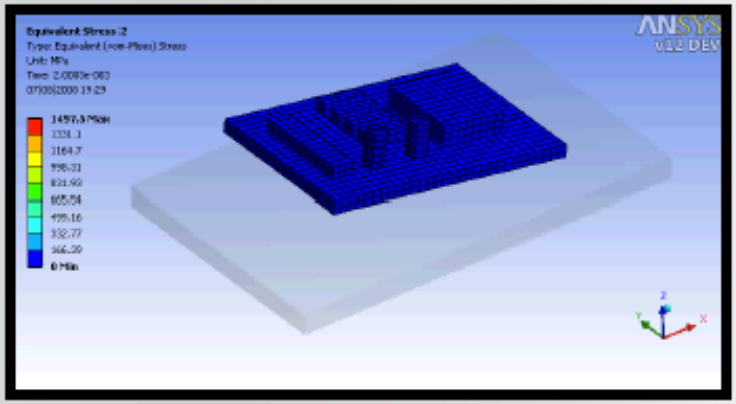
| VELOCITY | LOW | HIGH |
|---|---|---|
| Deformation | Global | Local |
| Response Time | ms - s | µs - ms |
| Strain | <10% | >50% |
| Strain Rate | < 10 $s^{-1}$ | > 10000 $s^{-1}$ |
| Pressure | < Yield Stress | 10-100 x Yield Stress |

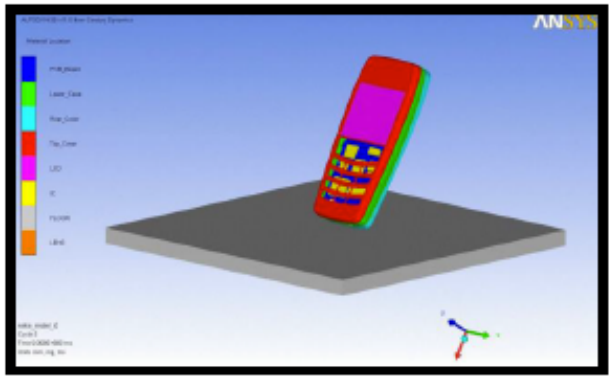
# Dynamic Analysis

Electronics Applications

- Quantify the interaction/load transfer between components
- Quantify the failure event during impact



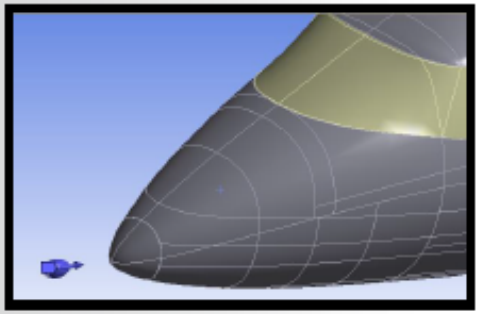


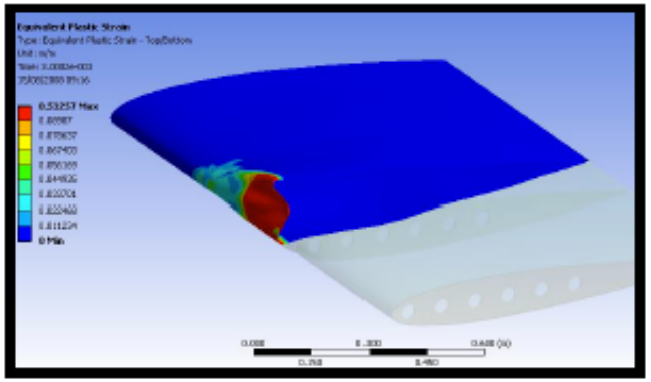



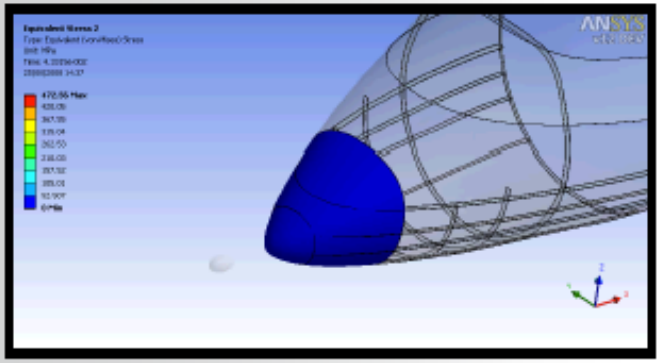
# Dynamic Analysis

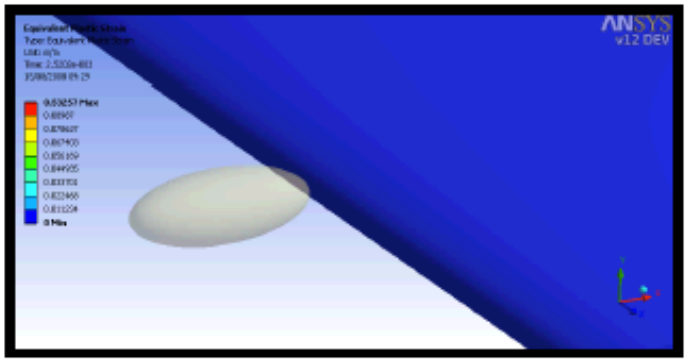
## Aerospace Applications

- **Bird strike:**
  - Structural response
  - Failure and damage footprint





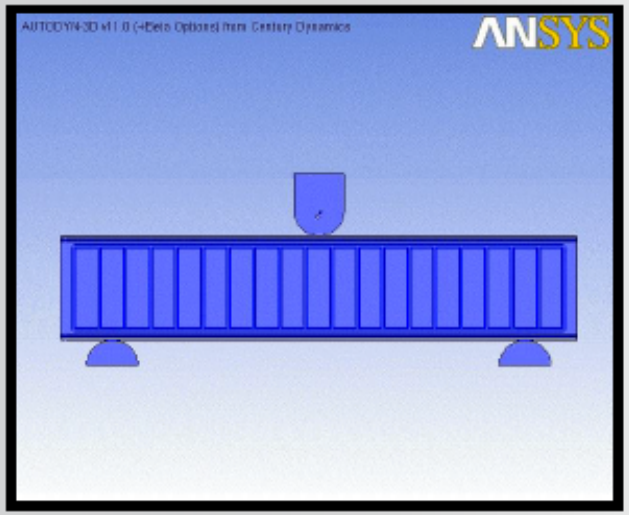


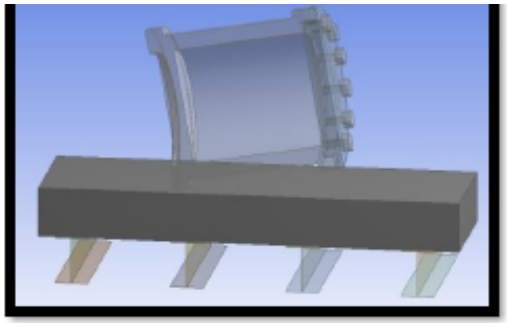
# Dynamic Analysis

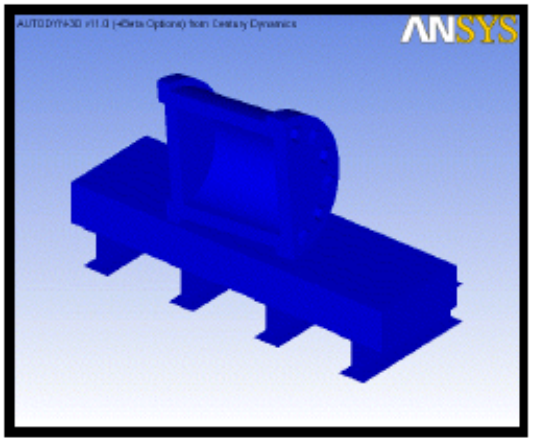
Applications in Nuclear Power safety

- Quantify the damage extent and structural survivability
- Quantify energy absorption




Characterization/component level



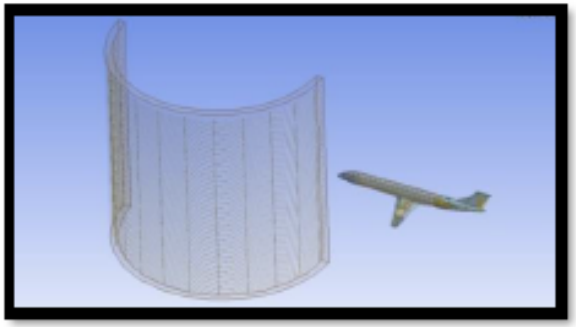


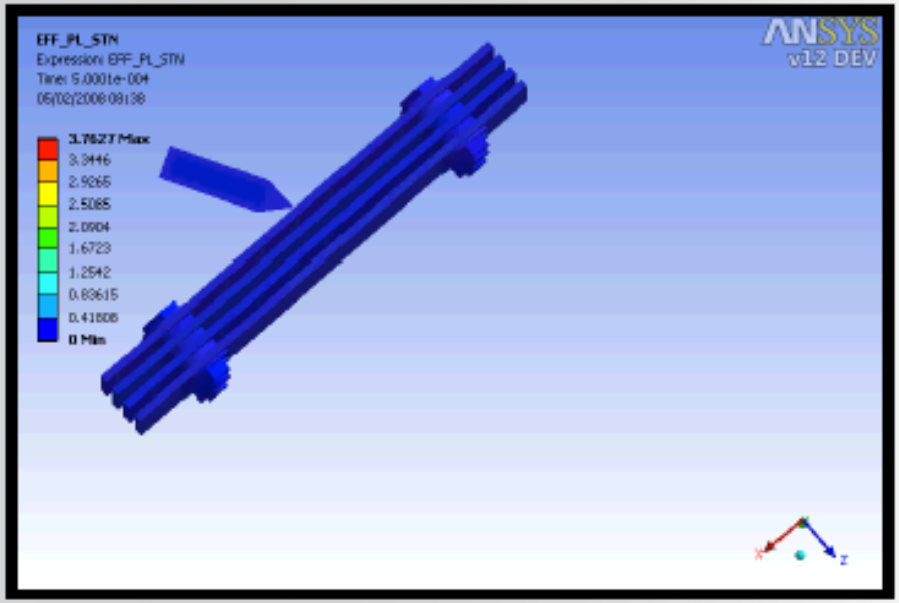

Performance Simulation

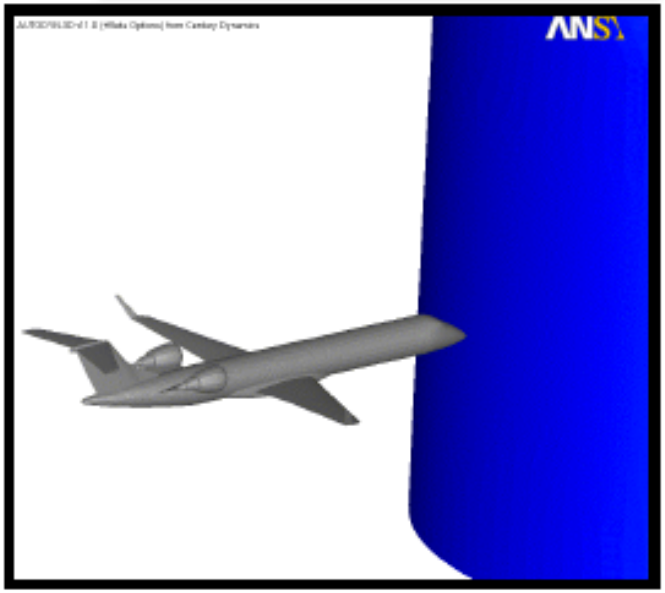
# Dynamic Analysis

## Applications in Homeland Security

- Energy absorption
- Lightweight design
- Survivability



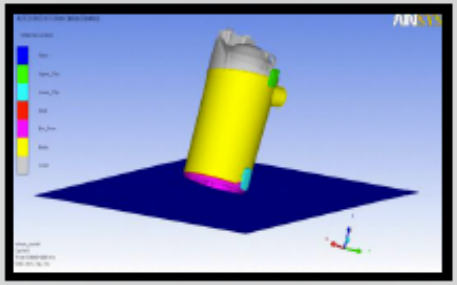


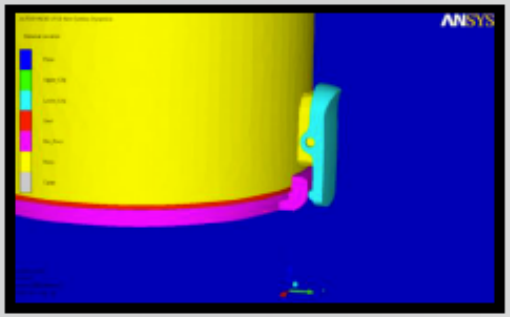



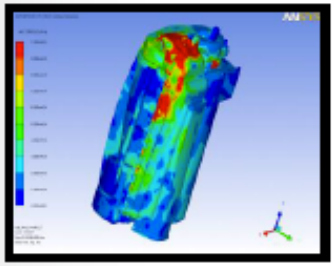
# Dynamic Analysis

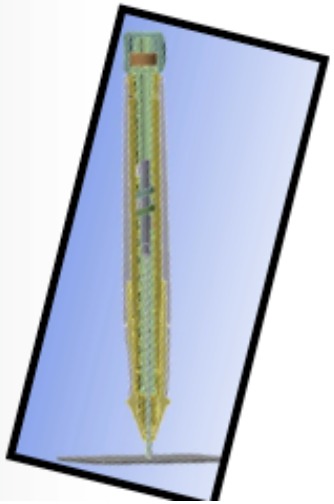
## Consumer Products Applications

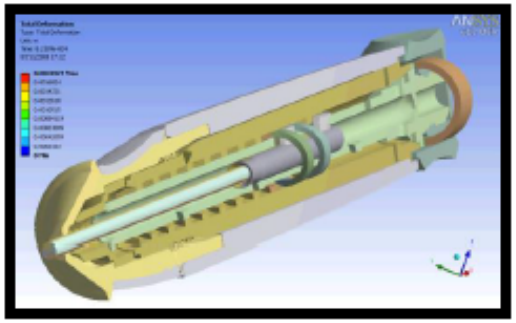
- Impact behavior
  - Component interaction
  - Acceleration response

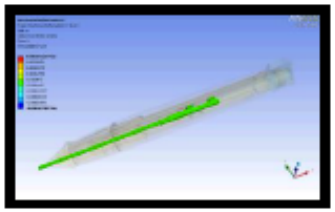


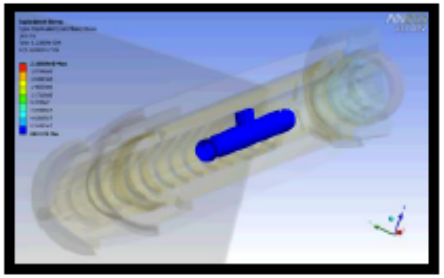








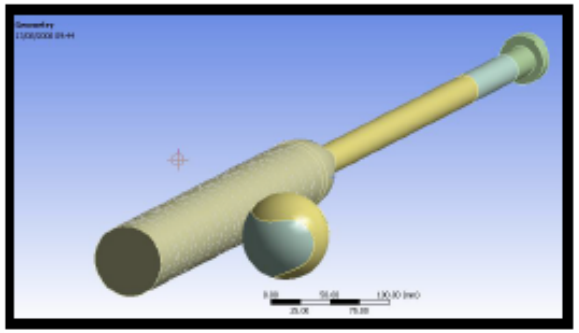


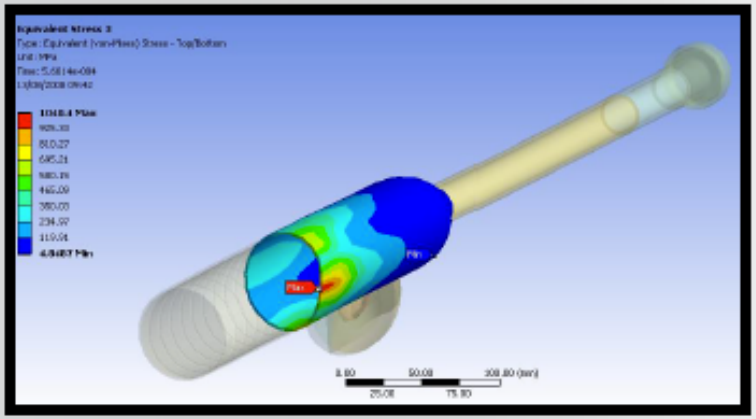


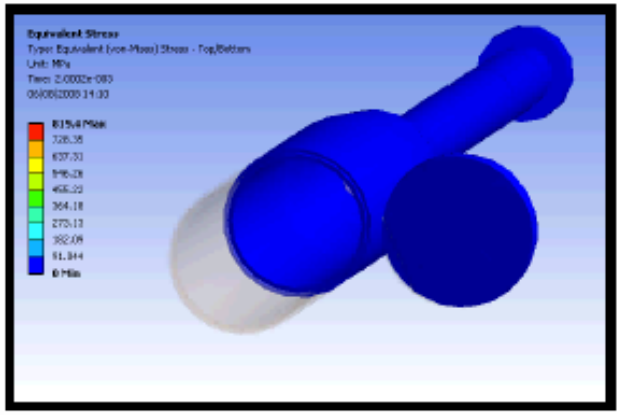
# Dynamic Analysis

## Sporting Goods Application

- Energy absorption
- Deformation and reaction force study







# Dynamic Analysis

Governing equation in Dynamics

$$m\ddot{x} + \dot{c}\dot{x} + kx = F(t)$$

***m*** = mass matrix,

***c*** = damping matrix

***k*** = stiffness matrix

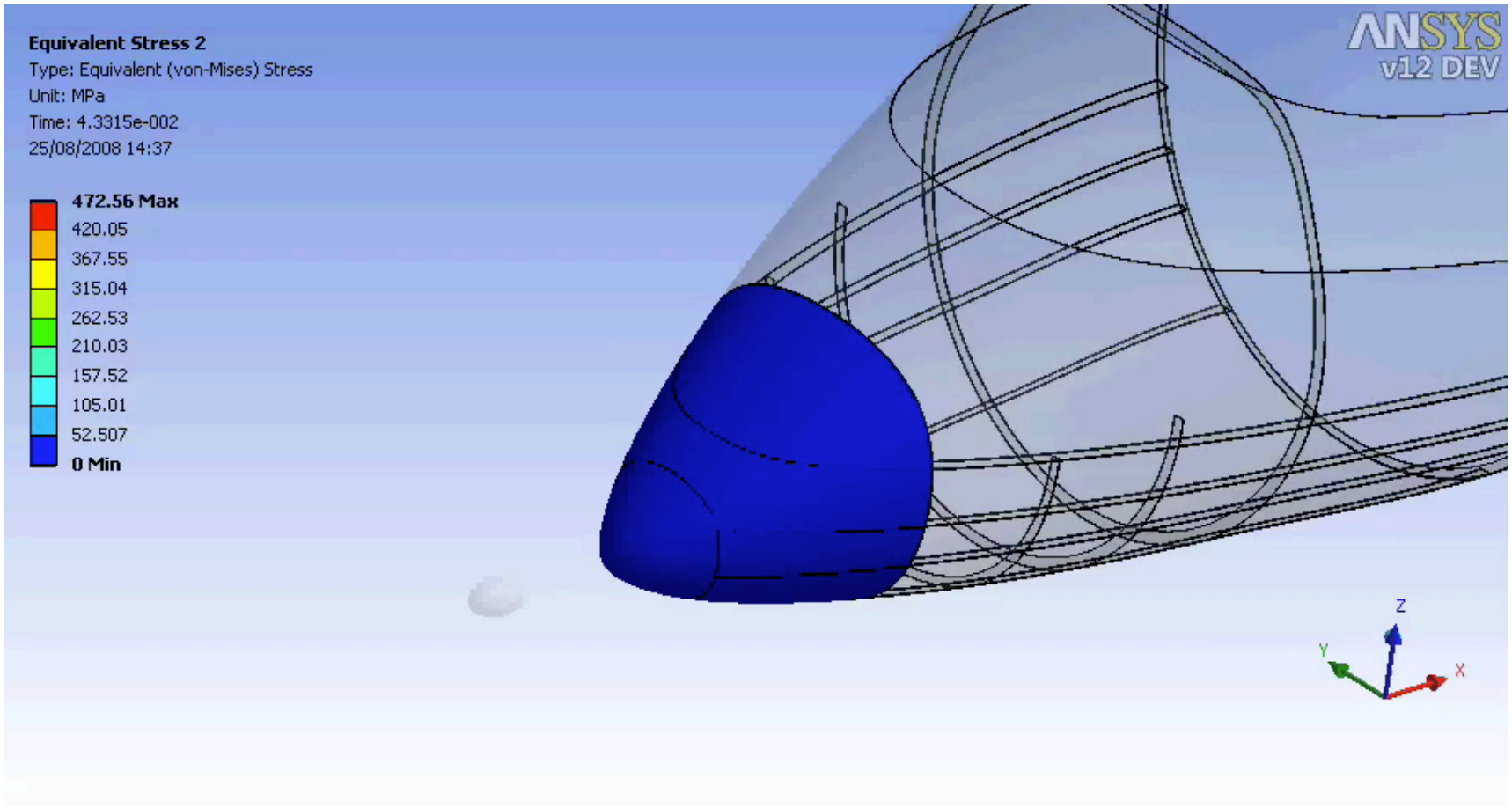
**F(t)** = load vector

At any **t**, this equation is a set ***"static" equilibrium*** equations with ***Inertia forces*** and ***Damping forces***.

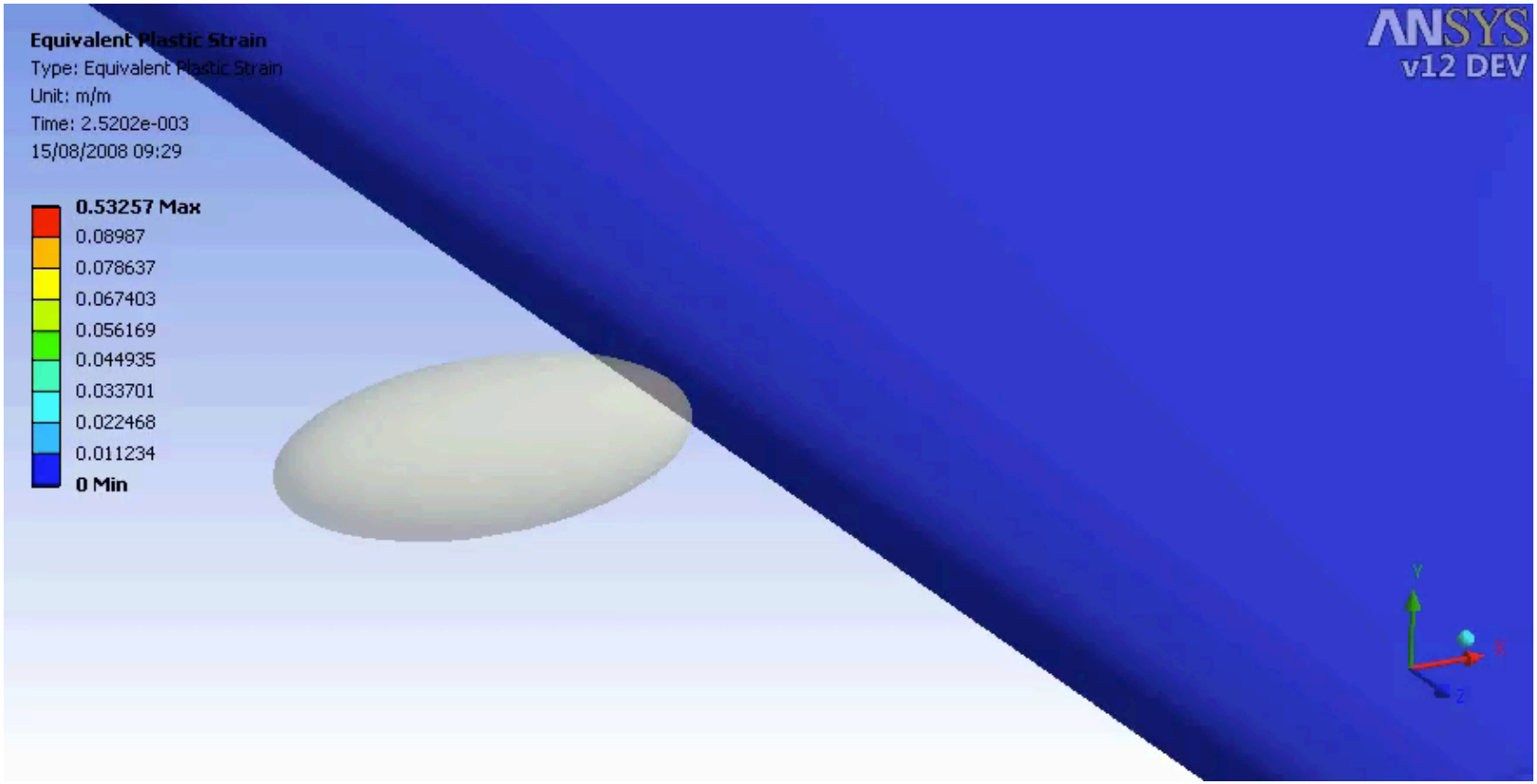
Solution Method: The Newmark or HHT (Hilber-Hughes-Taylor) at discrete time points

The time increment between successive time points is called the integration time step

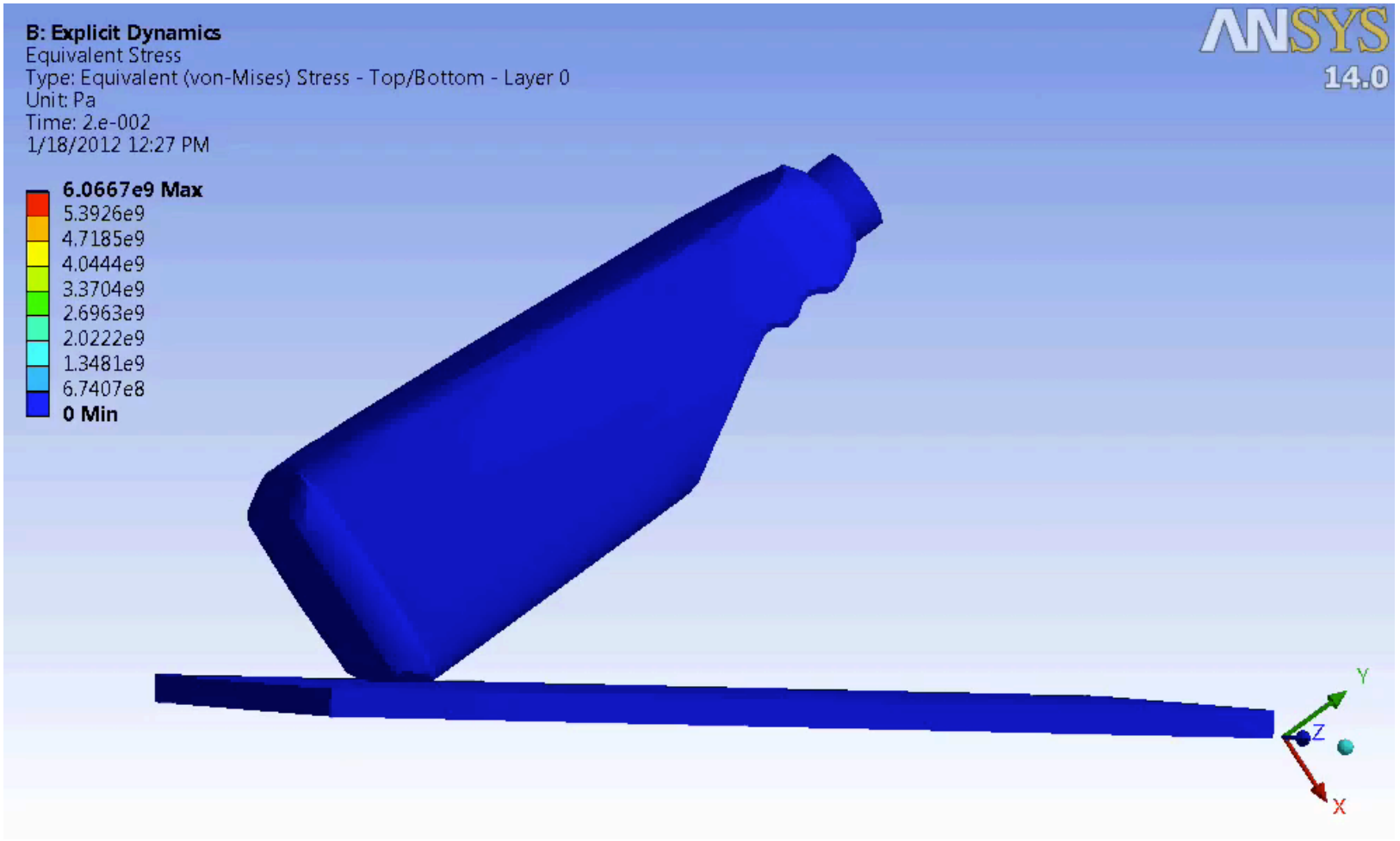
# Dynamic Analysis



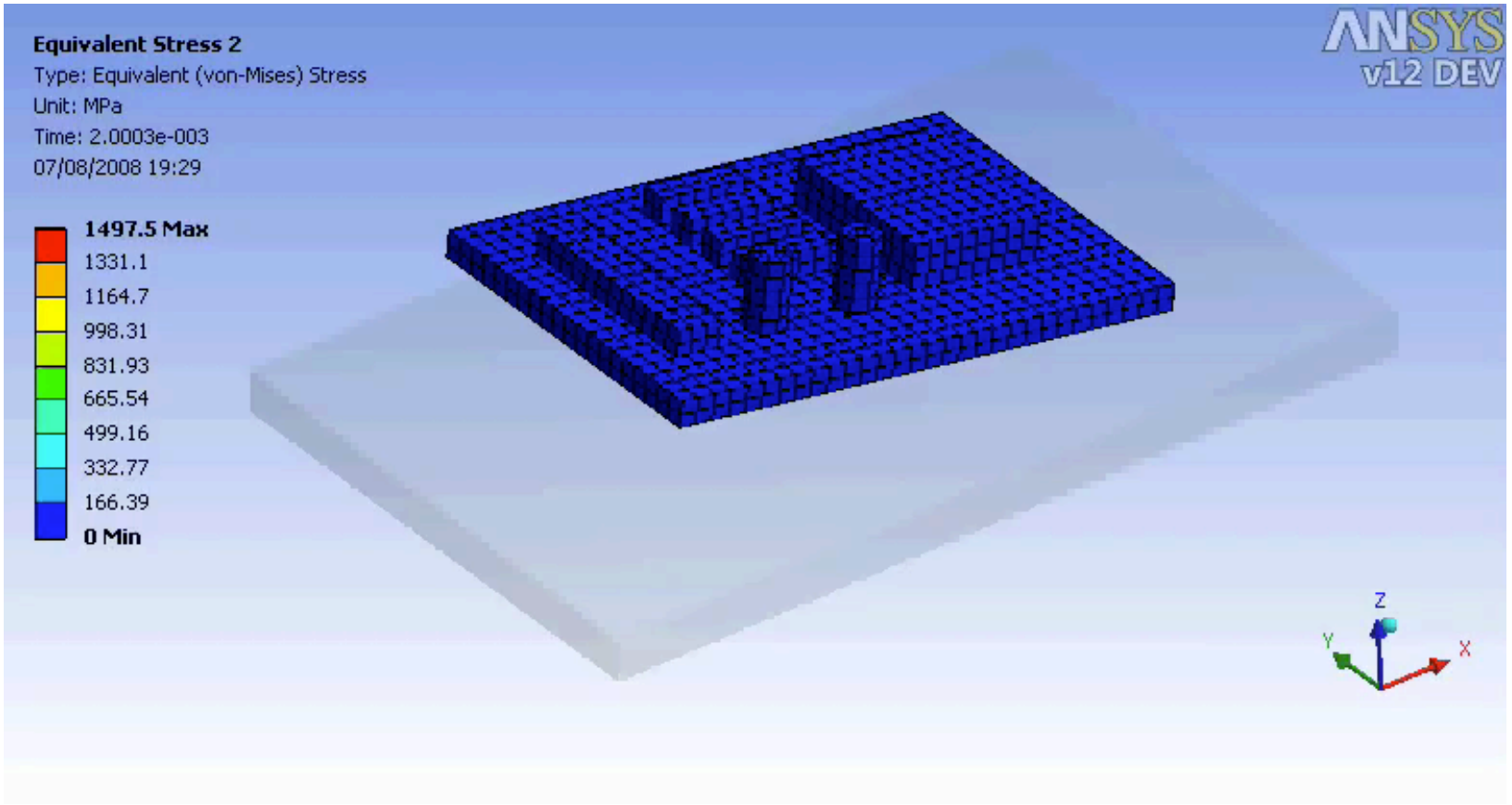
# Dynamic Analysis



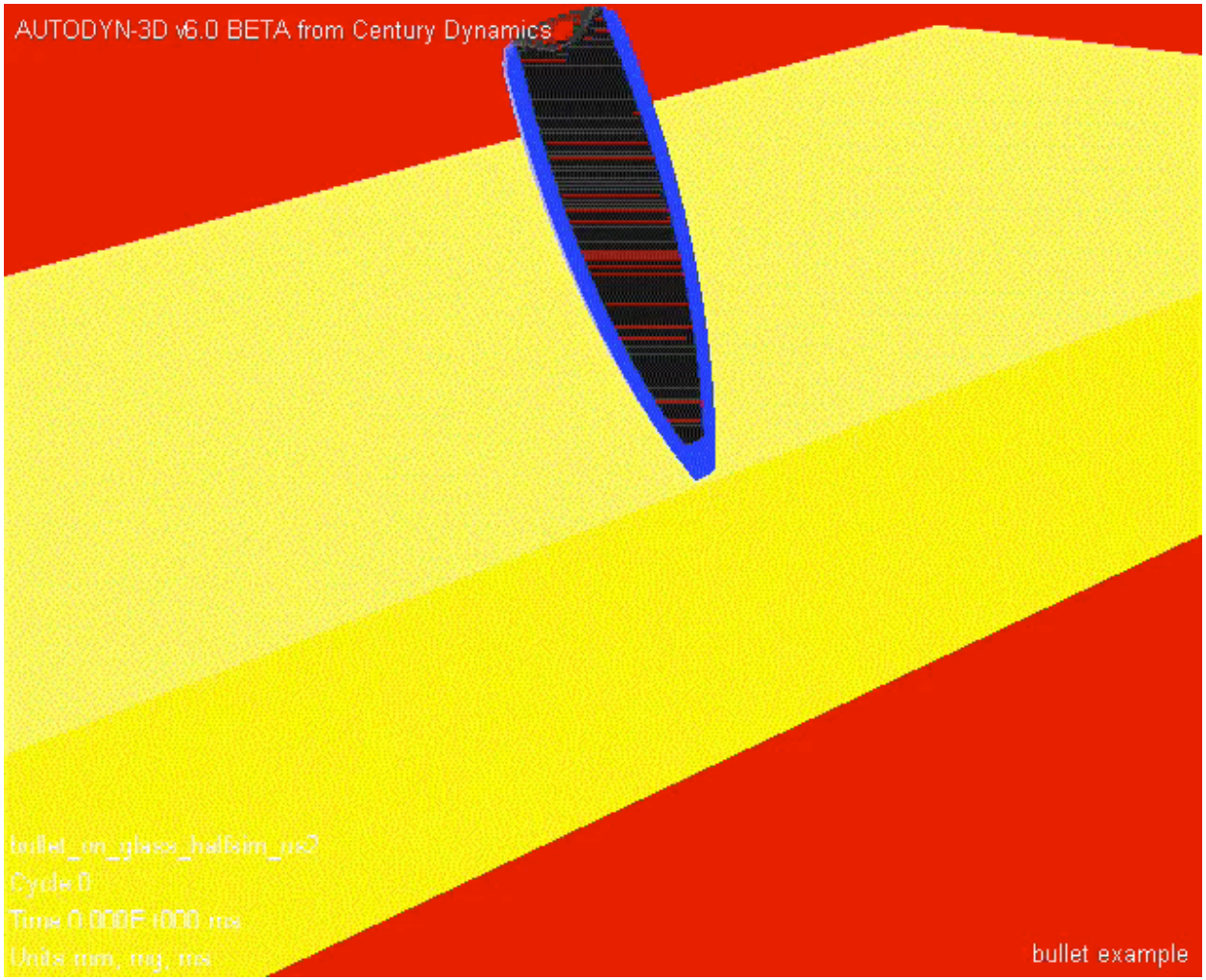
# Dynamic Analysis



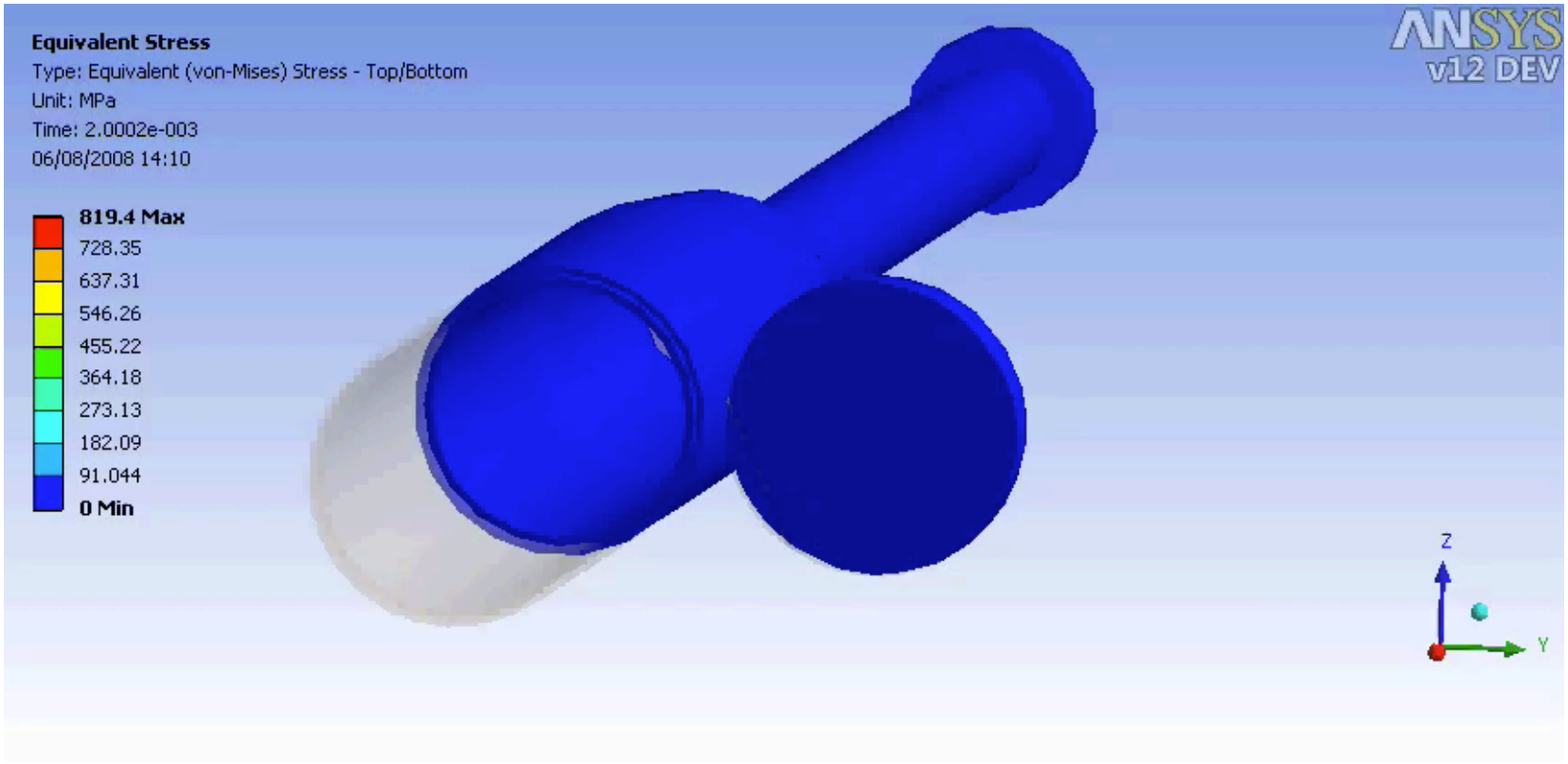
# Dynamic Analysis



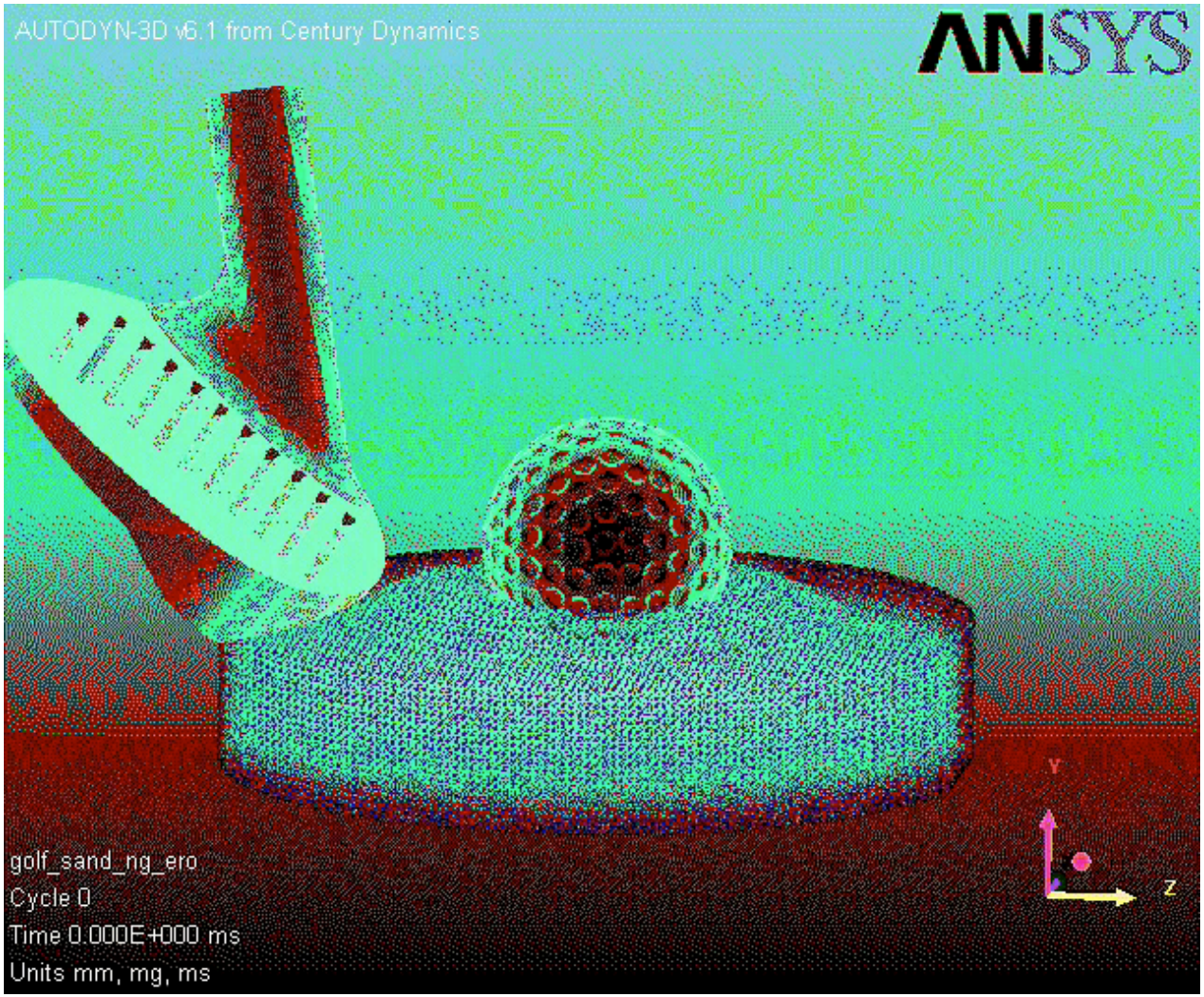
# Dynamic Analysis



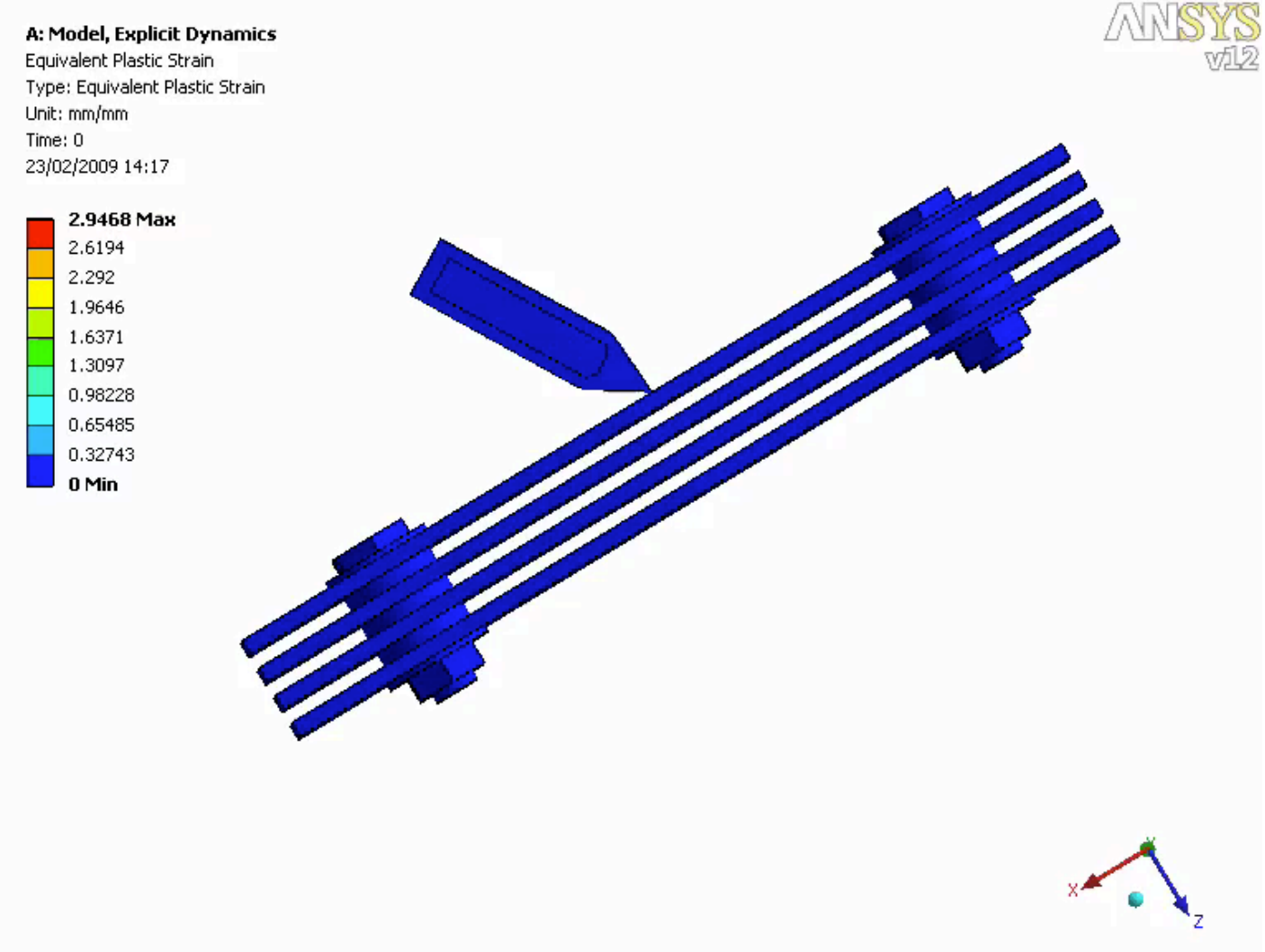
# Dynamic Analysis



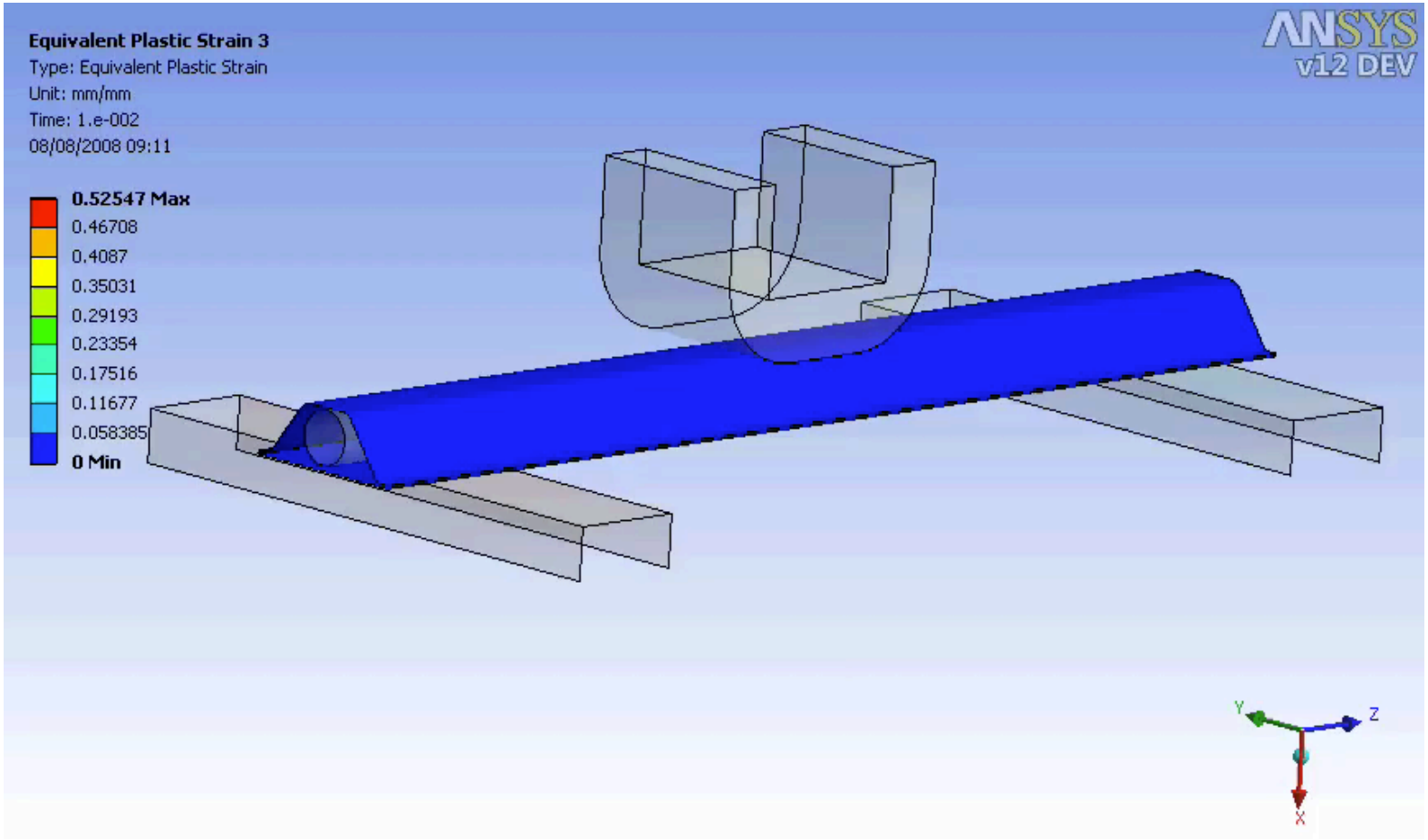
# Dynamic Analysis



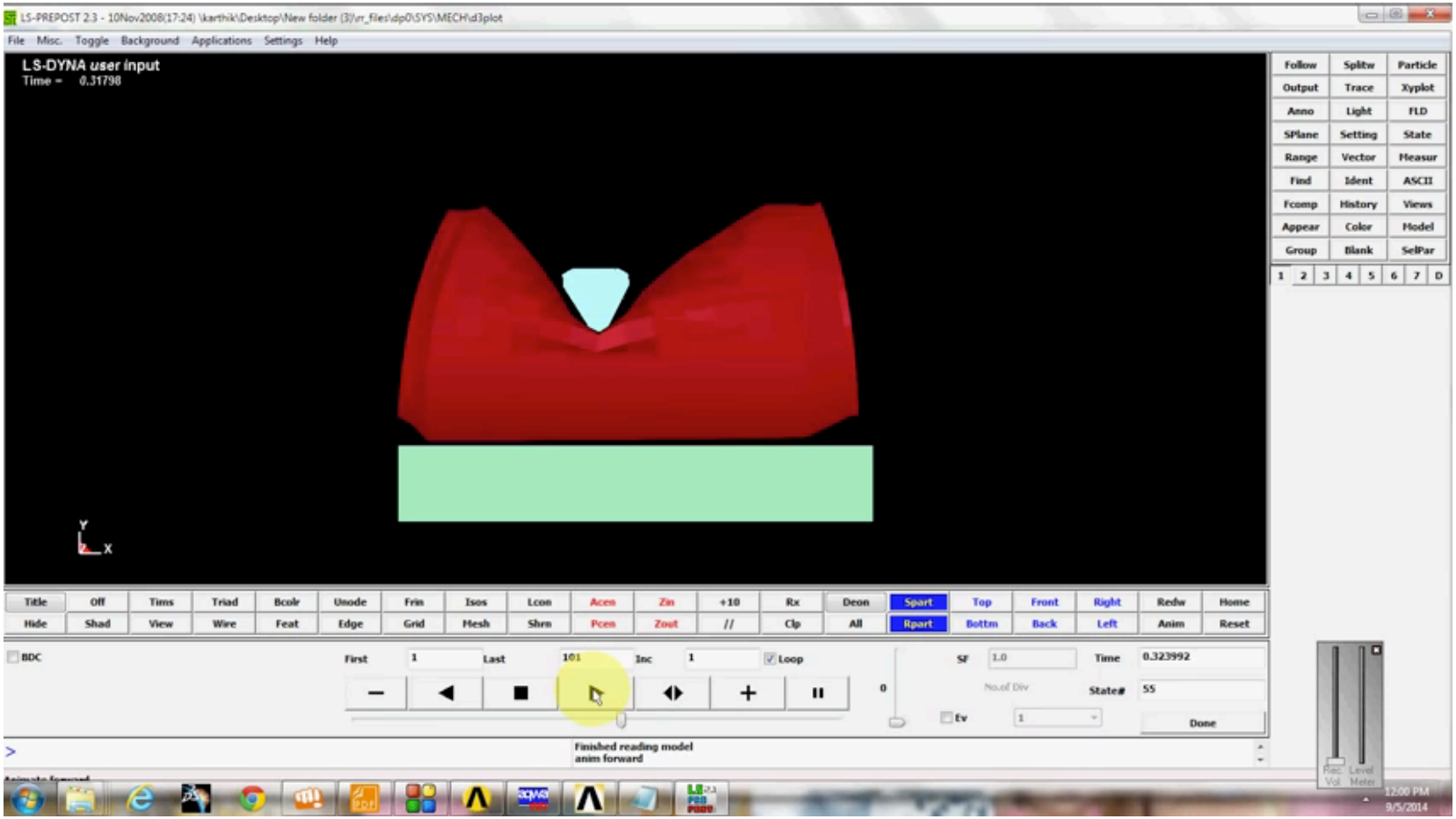
# Dynamic Analysis



# Dynamic Analysis

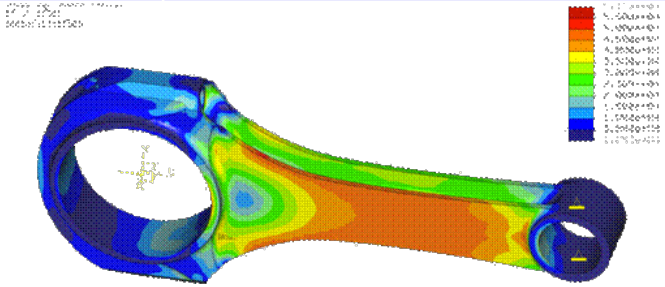


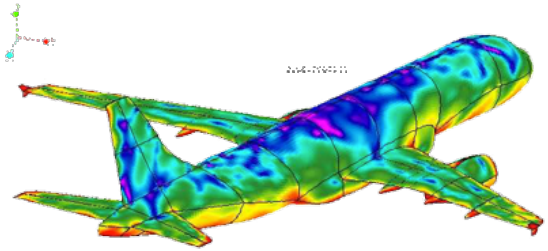
# Dynamic Analysis

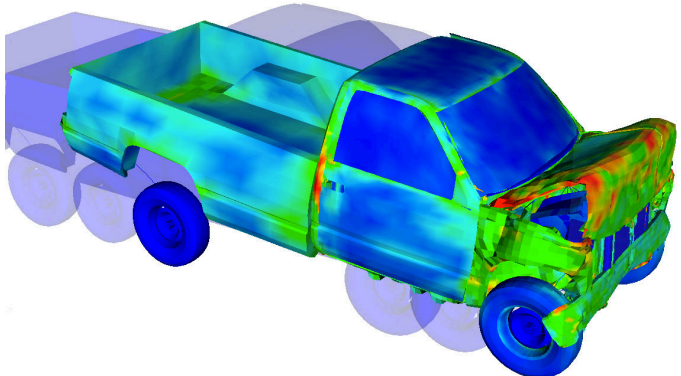


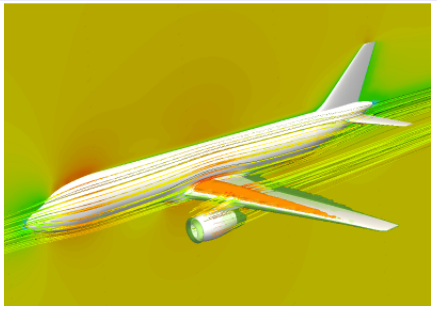
# Fluid-Structure Interaction

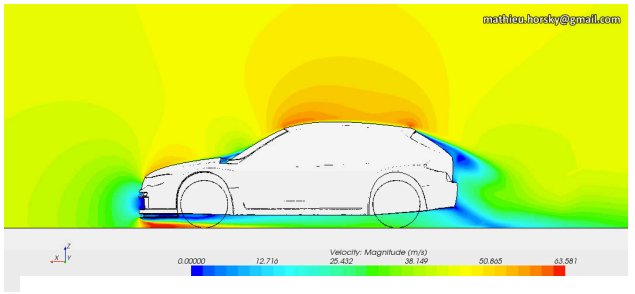
| | Solid Mechanics-Structural Analysis | Fluid Dynamics |
|---|---|---|
| Solved by | Finite Element Analysis | Computational Fluid Dynamics (CFD) |

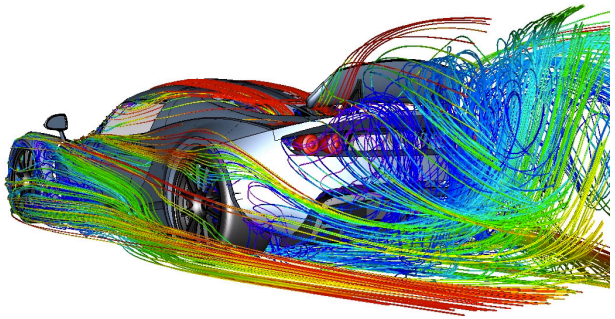




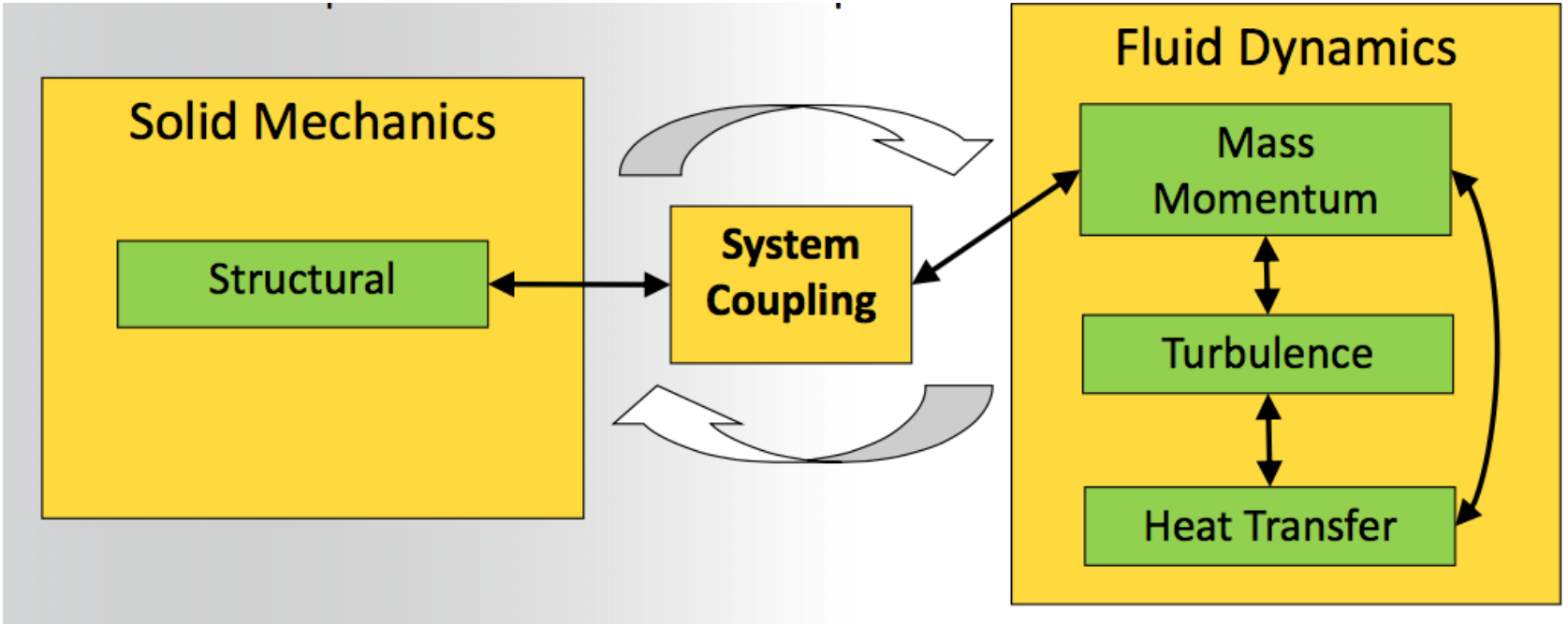






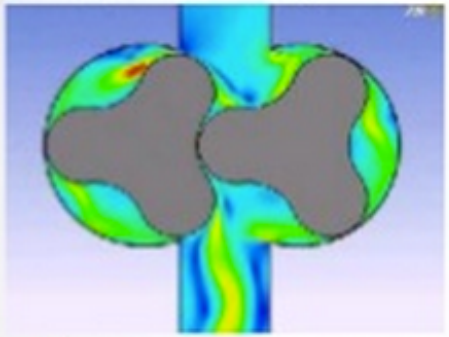
# Recent Computational Methodology



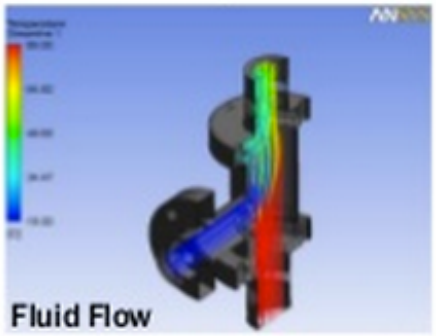

| | Finite Element Analysis | Computational Fluid Dynamics (CFD) |
|---|---|---|
| Commercial Software | Ansys Mechanical, Abaqus | Ansys Fluent, Ansys CFX, Open-foam |

# Recent Computational Methodology

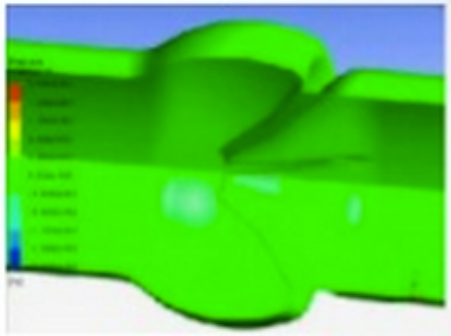
Rigid Body FSI

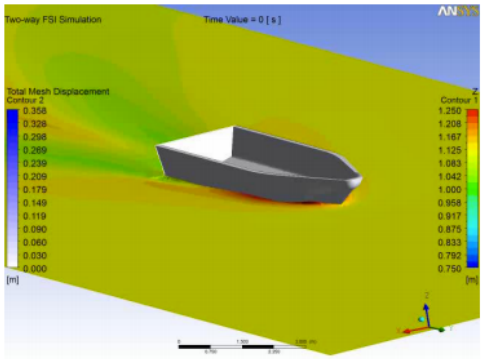


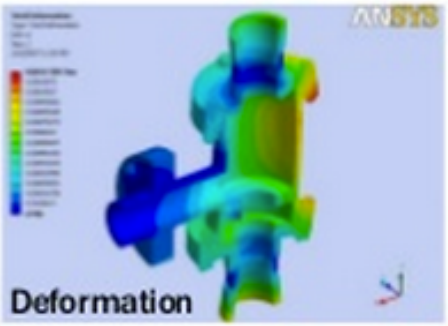
1-way FSI

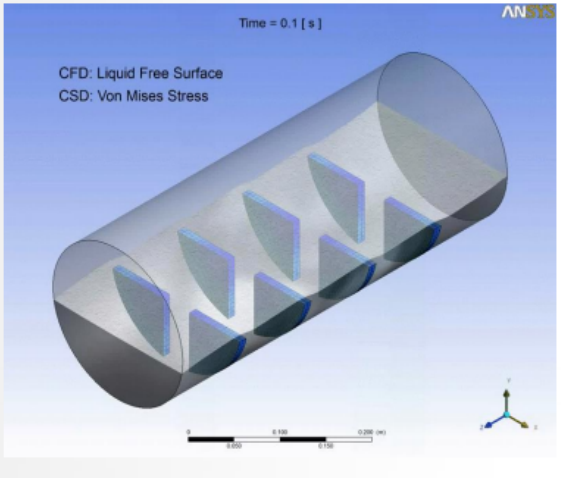



2-way FSI

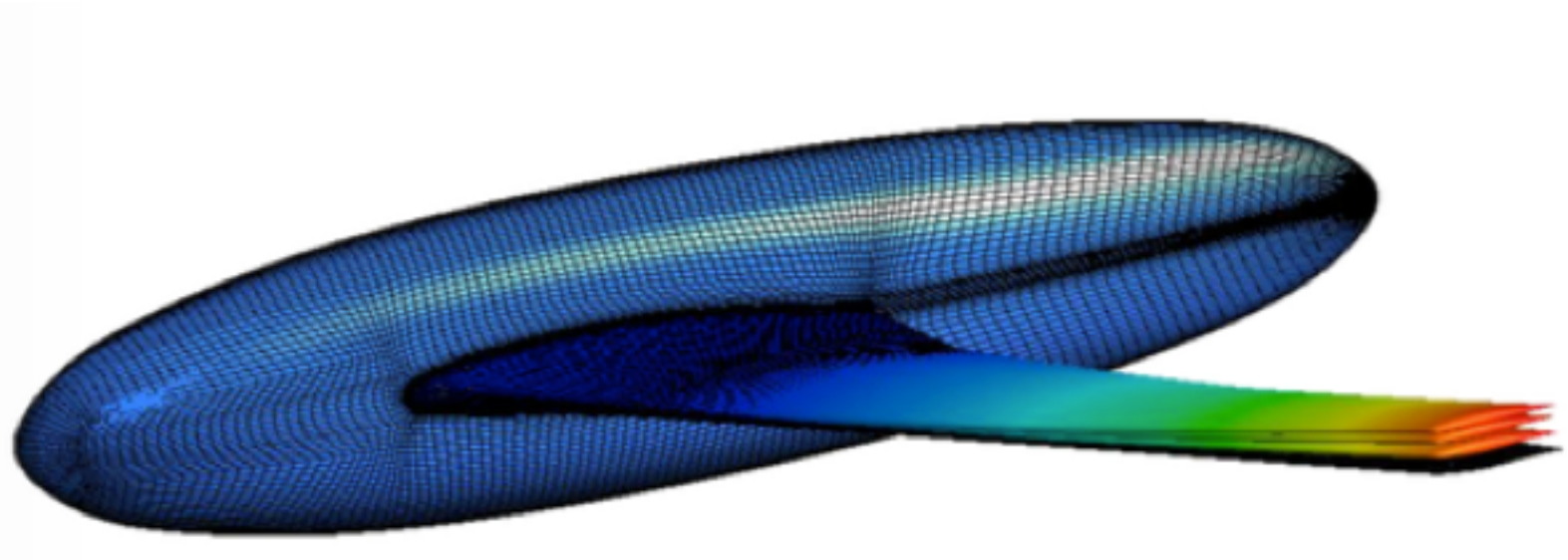




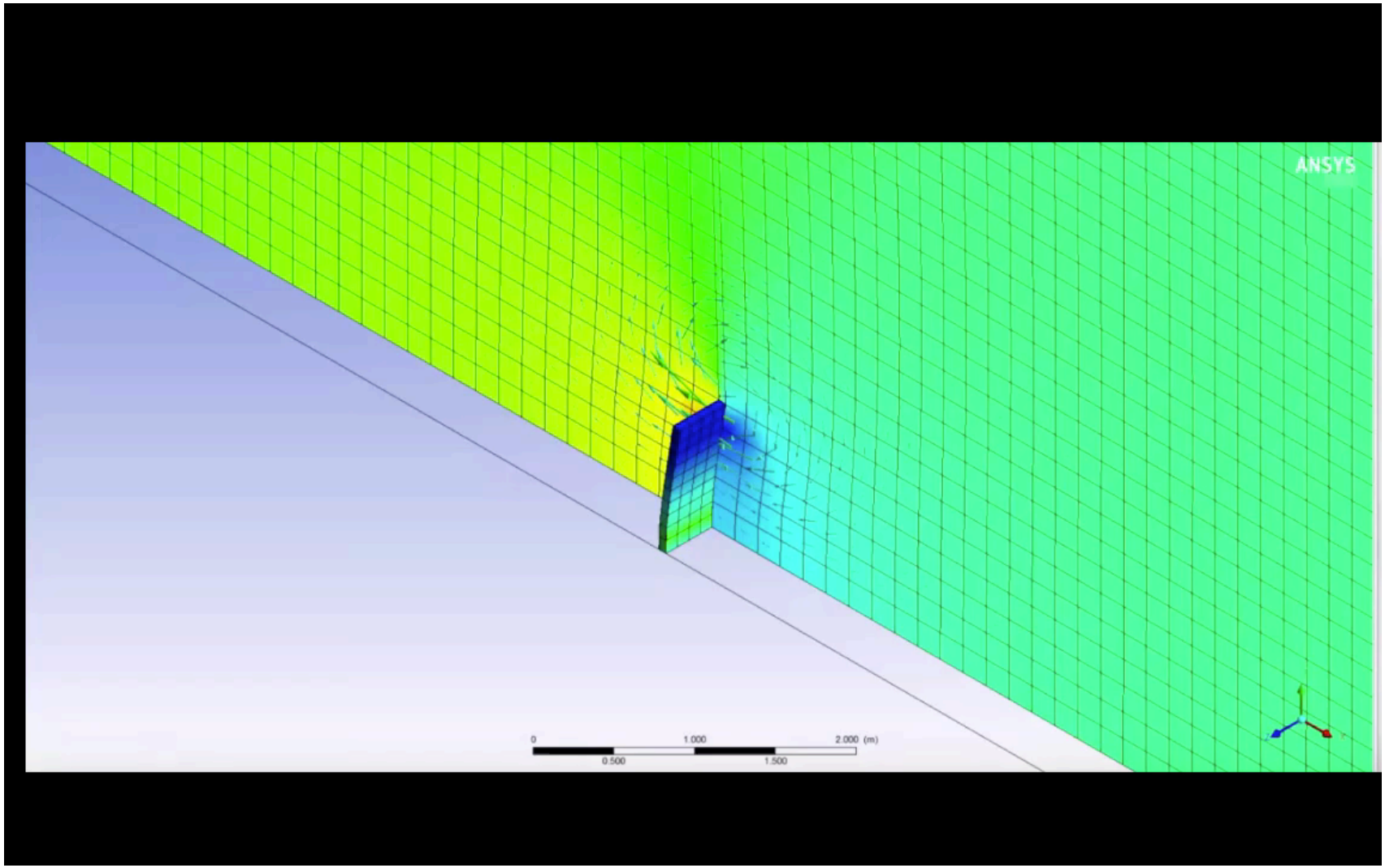



# Recent Computational Methodology

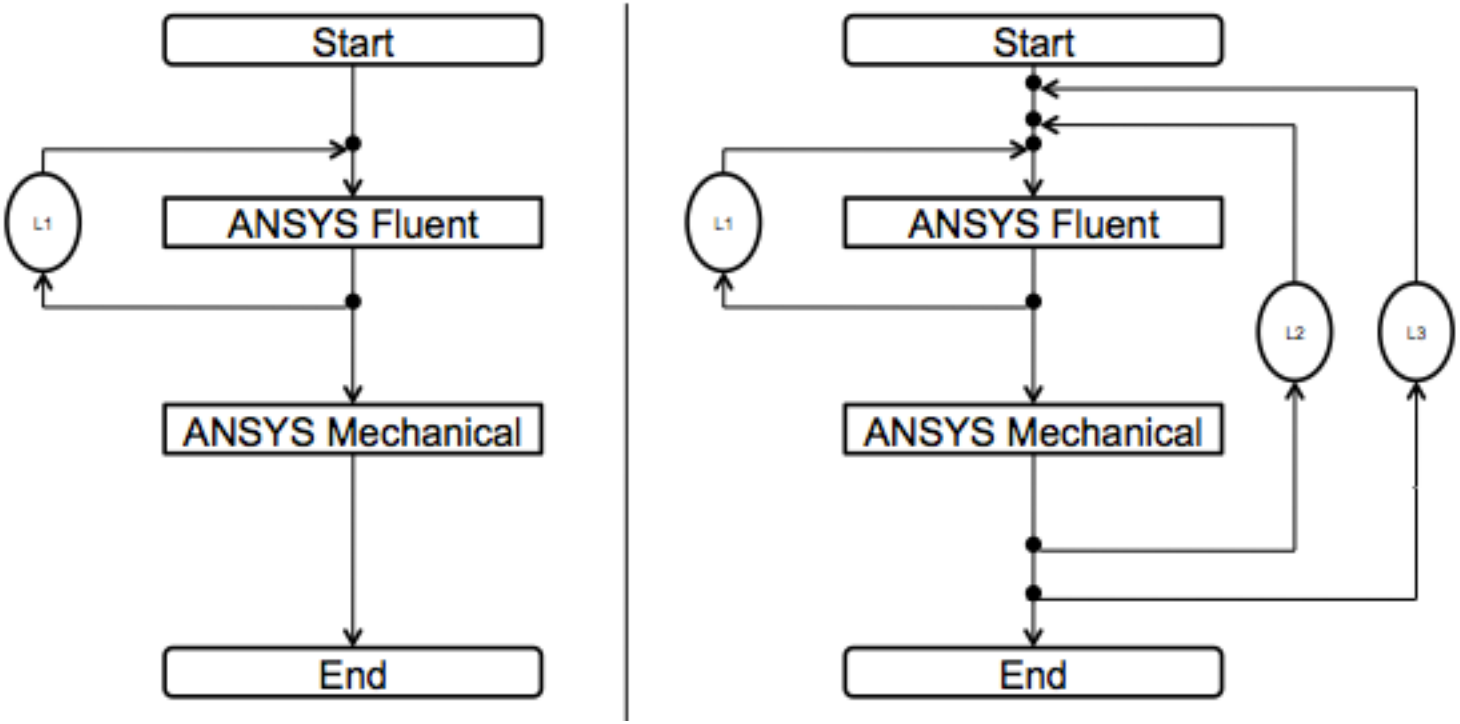


# Fluid-Structure Interaction

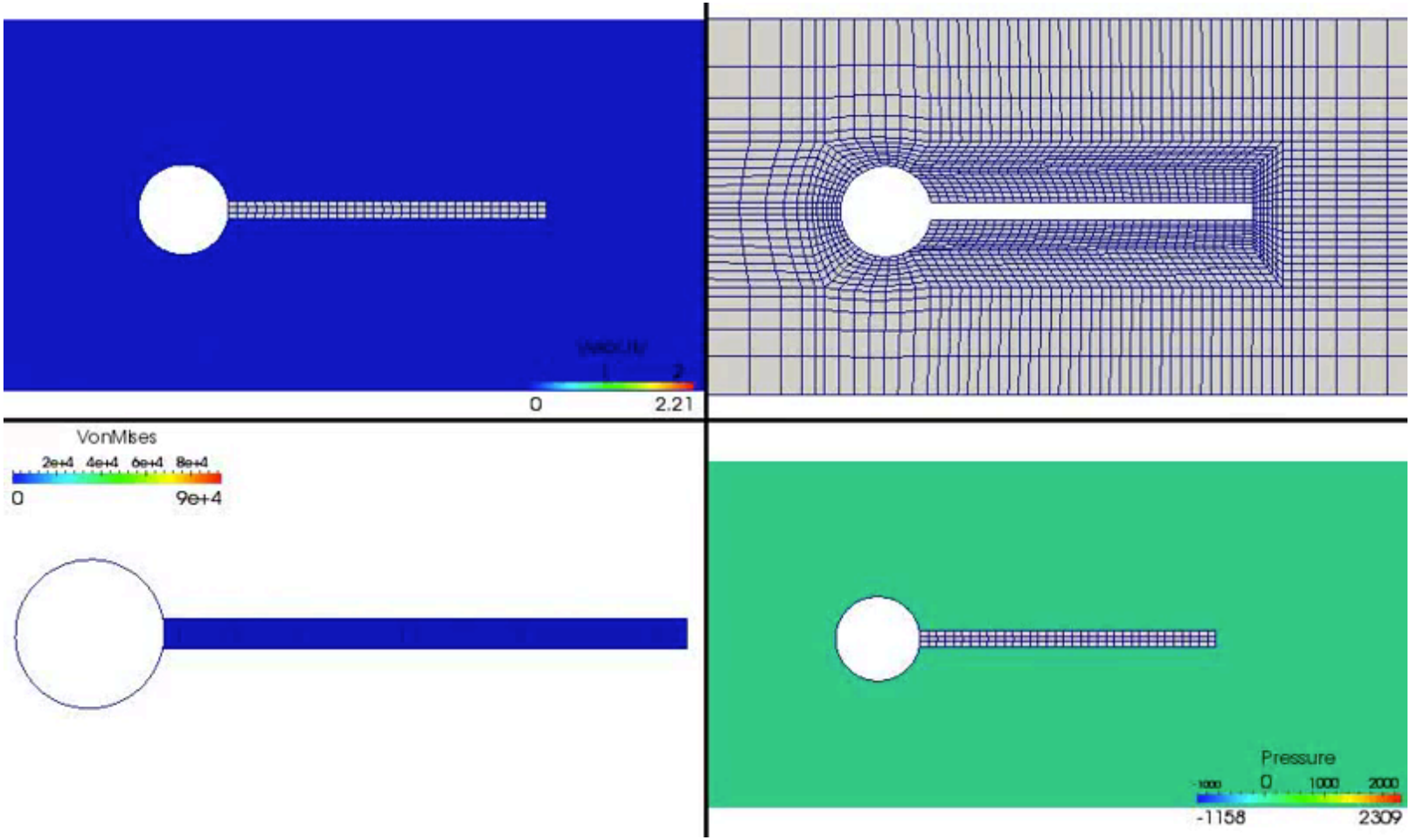


# Recent Computational Methodology

1 way FSI vs Two way FSI



# Fluid-Structure Interaction



# Fluid Structure Interaction