# **Cantilever Beam Modal Analysis - Physics Setup**

Author: John Singleton, Cornell University Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation Exercises Comments

# **Physics Setup**

**Material Assignment** 

At this point, we will tell ANSYS to assign the Aluminum material properties that we specified earlier to the geometry. First, (expand) Geometry then (click) Line Body, as shown below.



Then, (expand) Material in the "Details of Line Body" table and set Assignment to Aluminum, as shown below.

Graphics Properties		
Definition		
Suppressed	No	
Coordinate System	Default Coordinate	
Reference Temperature	By Environment	
Offset Mode	Refresh on Update	
Offset Type	Centroid	
- Material		
Assignment	Structural Steel	
Nonlinear Effects	Yes	Alumnum
Thermal Strain Effects	Yes	-0
• Bounding Box		
Properties		
Statistics		

At this point your "Details of Line Body" table, should look comparable to the following image.

Details of "Line Body" 4				
Ŧ	Graphics Properties			
Ξ	Definition			
	Suppressed	No		
	Coordinate System	Default Coordinate		
	Reference Temperature	By Environment		
	Offset Mode	Refresh on Update		
	Offset Type	Centroid		
	Material			
	Assignment	Aluminum		
	Nonlinear Effects	Yes		
	Thermal Strain Effects	Yes		
Ŧ	Bounding Box			
Ŧ	Properties			
Ŧ	Statistics			

**Fixed Support** 

First, (right click) Modal > Insert > Fixed Support, as shown below.



Next, click the vertex selection filter button, Then, click on the left end of the beam and apply it as the Geometry in the "Details of Fixed Support" table.

#### **Constrain Beam to XY Plane**

In this section the beam's motion will be constricted to the xy plane.

First, (right click) Modal > Insert > Displacement, as shown below.

File Edit View Units Tools Help   Environment   Invironment   Invironment <td< th=""><th>🔀 C : Cantilever Modal Confined to XY Plane - I</th><th>Mechanical [ANSYS Academic Teachi</th></td<>	🔀 C : Cantilever Modal Confined to XY Plane - I	Mechanical [ANSYS Academic Teachi
Environment 🔍 Inertial 🔹 🔍 Loads 🔹 🔍 Supports 🔹 💽 Outline Project Project Model (C4) C C C Model (C4) C C C C C C Model (C4) C C C C C C Model (C4) C C C C Model (C4) C C C C C C C C	File Edit View Units Tools Help	🦻 🏓 Solve 🔻 🏥 👪 🚺 🔺
Project     Project       Project     Geometry       Coordinate Systems     4/19       Mesh     Mesh       Mesh     Fixed Support       Fixed Solut     Fixed Support       Solut     Clean       Pre-Stic     Clean       Pre-Stic     Support       Solut     Clean       Pre-Stic     Support       Solut     Frictionless Support       Support     Simply Supported       Fixed Rotation     Fixed Rotation	Environment 🍕 Inertial 👻 😪 Loads 👻 😪 Si	upports 🔻 👔
Elastic Support     Constraint Equation	Outline Project Medel (C4) Coordinate Systems Mesh Modal (C5) Insert Solve Solve Clean D Rename	C: Can Moda Frequ 4/19/2 Fixed Support Fixed Support Fixed Support Fixed Support Fixed Support Cylindrical Support Simply Supported Fixed Rotation Fixed Rotation Elastic Support Constraint Equation
Commands		Commands

Click Here for Higher Resolution

Next, click the *edge selection filter* button, **b**. Then, click on the geometry and apply it as the *Geometry* in the "Details of Displacement" table. Lastly, set *Z Component* to 0, as shown below.

Details of "Displacement"		
=	Scope	
	Scoping Method	Geometry Selection
	Geometry	1 Edge
Ξ	Definition	
	Туре	Displacement
	Define By	Components
	Coordinate System	Global Coordinate System
	X Component	Free
	Y Component	Free
	Z Component	•
	Suppressed	No

### Save

## Go to Step 5: Numerical Solution

Go to all ANSYS Learning Modules