

# Modal Analysis of a Satellite - Numerical Solution

Author: Robert McBride, 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](#)

## Numerical Solution

In order to obtain modal shapes and their corresponding frequencies, we must insert a total deformation graphic for each mode we asked the solver to find in Analysis Settings. Do this by right clicking **Solution > Insert > Deformation > Total**.

Right click on the newly created **Total Deformation** tab and rename it **Mode 1**. We must assign each total deformation to a specific mode. Do this by selecting **Mode 1** and setting the **Mode** number under **Definition** to **1**. Repeat this process for modes 2 through 5. The details window for Mode 1 deformation is pictured below.

Details of "Mode 1"	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	All Bodies
<b>Definition</b>	
Type	Total Deformation
Mode	1.
Identifier	
<b>Results</b>	
<input type="checkbox"/> Minimum	0. m
<input type="checkbox"/> Maximum	5.1565 m
Minimum Occurs On	Leg
Maximum Occurs On	Large Box 2
<b>Information</b>	

[Go to Step 6: Numerical Results](#)

[Go to all ANSYS Learning Modules](#)