



谐响应分析

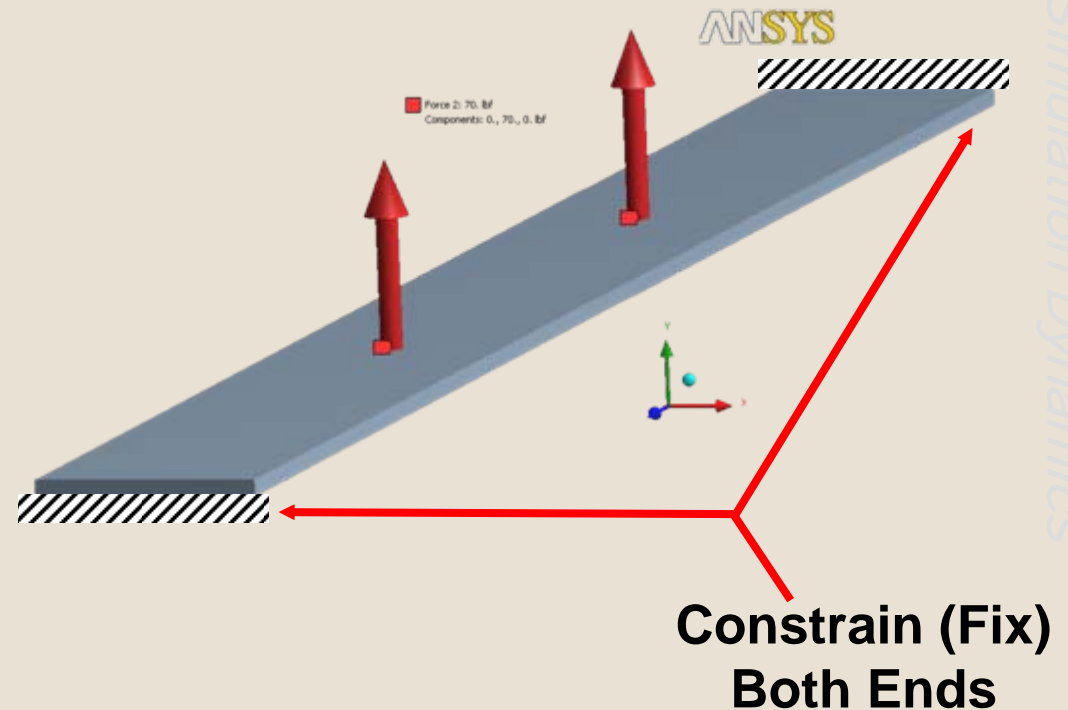
两端固定梁

Workshop 5



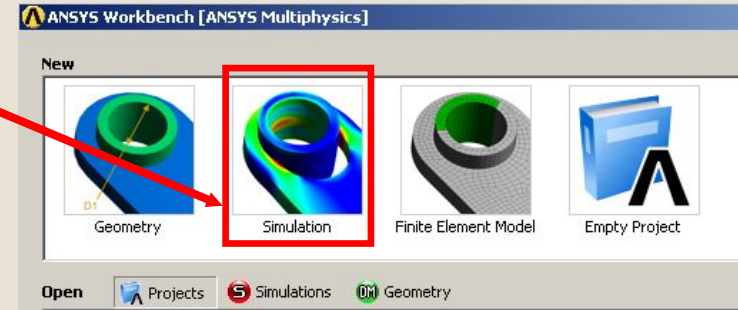
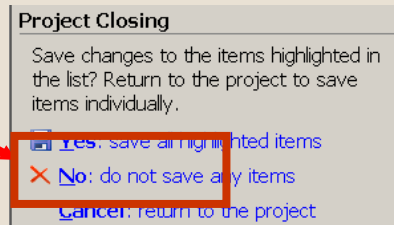
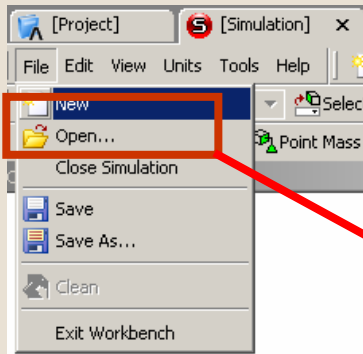
# Workshop 5 – 目的

- 目的在于分析两端固定梁在同相位的简谐力作用下的响应。
  - 施加的力表示为在梁在1/3位置处旋转机械的作用。
  - 旋转机械的转速为 300 -1800 每分钟。
- 梁的材料为钢材。
  - 横截面：10" x 3/4"
  - 长度：10'
- 假设梁的两端完全固定。
- 假设阻尼比为0.02.

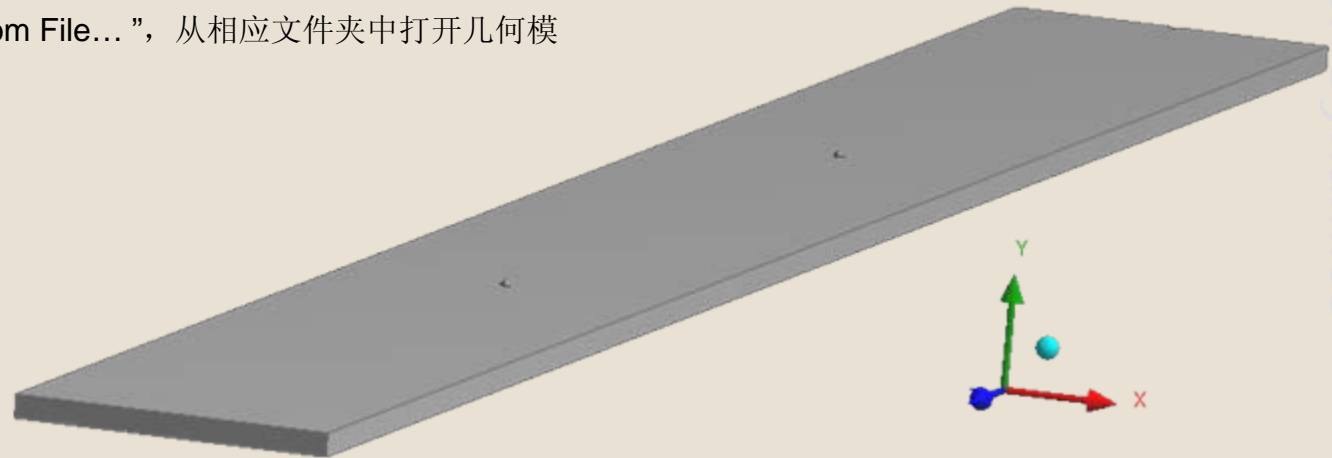


# Workshop 5 – 起始页

- 从“ WorkBench Project Launcher ”点击“ Simulation”。
- 如果进入“Simulation”，点击 >File>New
  - 基于培训，可以点击 “No: do not save any items”



- 点击 “ >Geometry>From File...” ，从相应文件夹中打开几何模型文件 beam.dsb



ANSYS Workbench - Simulation Dynamics

## Workshop 5 – 设置

1. 当几何模型导入以后，在“Map of Analysis Types...”选择“Harmonic Response”

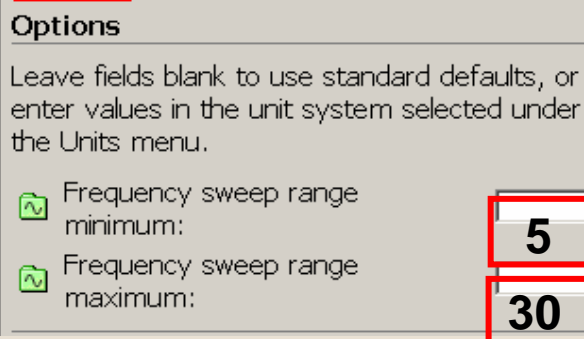
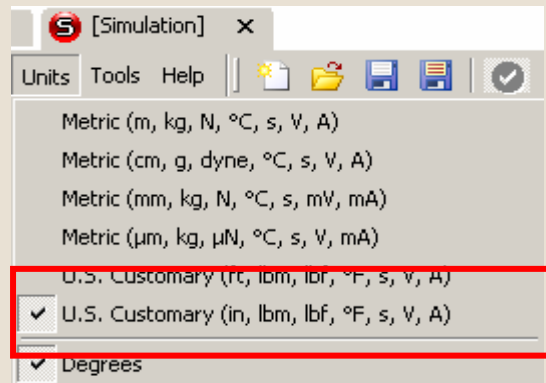
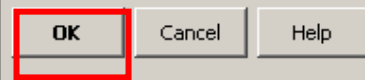
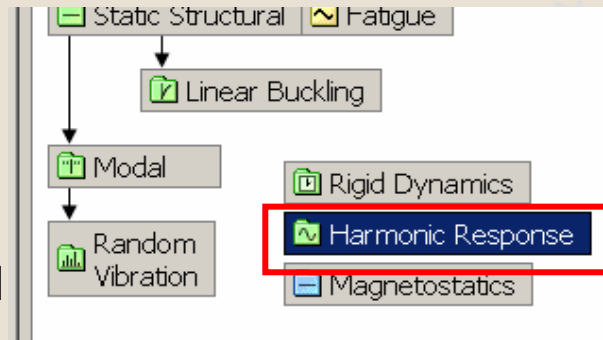
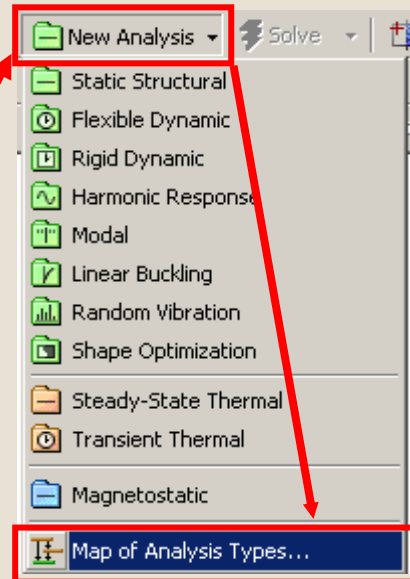
– 如果需要的话，点击 >New Analysis>Map of...

2. 输入扫频范围：5-30Hz (300 -1800 RPM)

3. 点击 OK

4. 选择单位系统为：

U.S. inch pound “Units > U.S. Customary (in, lbm, lbf...)”



# Workshop 5 – 前处理

- 前处理的第一步是指定材料为结构钢： Structural Steel.
- 5. 展开“Geometry”分支，点击相应的部件。
- 6. 在Details的“Material”位置,点击下拉目录然后校核钢材的材料属性。

5

6

Outline

- Project
  - Model
    - Geometry
      - Solid**
    - Mesh

---

Details of "Solid"

<b>Graphics Properties</b>	
<b>Definition</b>	
<input type="checkbox"/> Suppressed	No
Material	<b>Structural Steel</b>
Stiffness Behavior	Flexible
Nonlinear Material Effects	Yes
<b>Bounding Box</b>	
<b>Properties</b>	
<b>Statistics</b>	

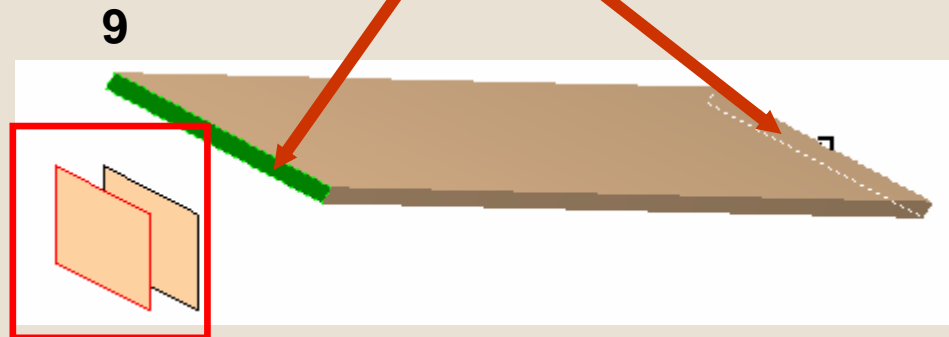
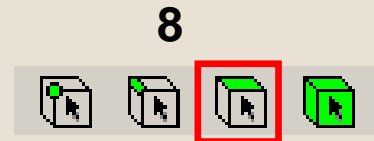
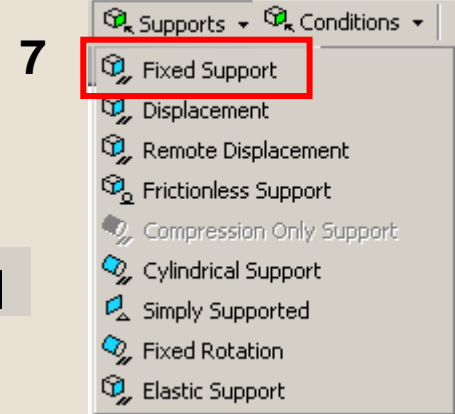
# Workshop 5 – 环境

7. 为了约束梁的两端，点击“ Harmonic Response”分支， 点击右键选择>Insert>Fixed Support.

8. 切换至面选择模式

9. 用左键选择左右两个面

- 利用“Depth Picking” 或者改变模型的视角。
- 按住 <CTRL>键实现多选。
- 在Details 窗口中点击“Apply”。



**Depth Picking**

# Workshop 5 – 环境

- 为了施加力，在梁的表面上已经存在WorkBench DM 关键点（便于集中力的施加）

10. 切换至点选模式

11. 选择: >Insert>Force

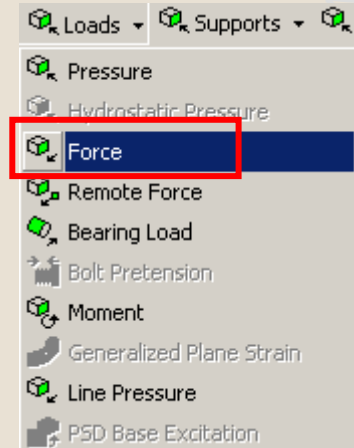
12. 用左键（LMB）点选出两个载荷施加点。  
 – 在Details窗口中点击 “Apply”

13. 在Details中，改变 “Defined By” 为“Components”  
 (i.e., XYZ).

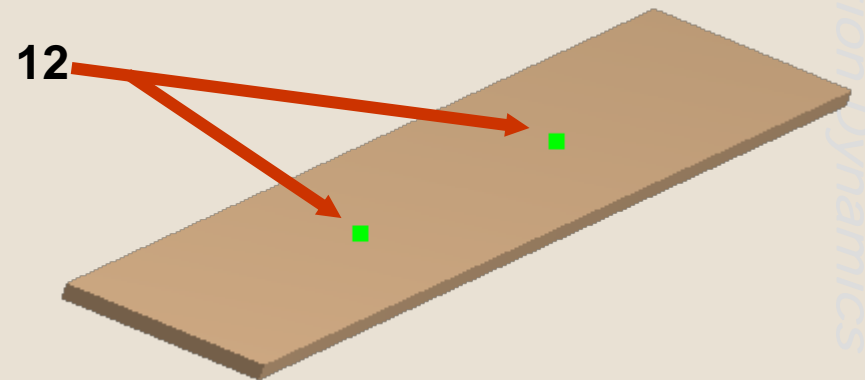
- 在Y Component 中输入-70， Phase Angle处为 0
- 注意：可以对每个点单独施加载荷，这样就可以施加不同相位的载荷。



10

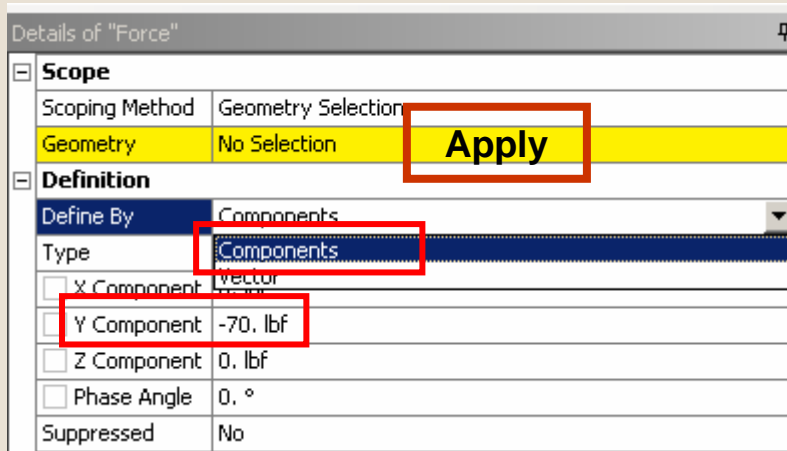


11



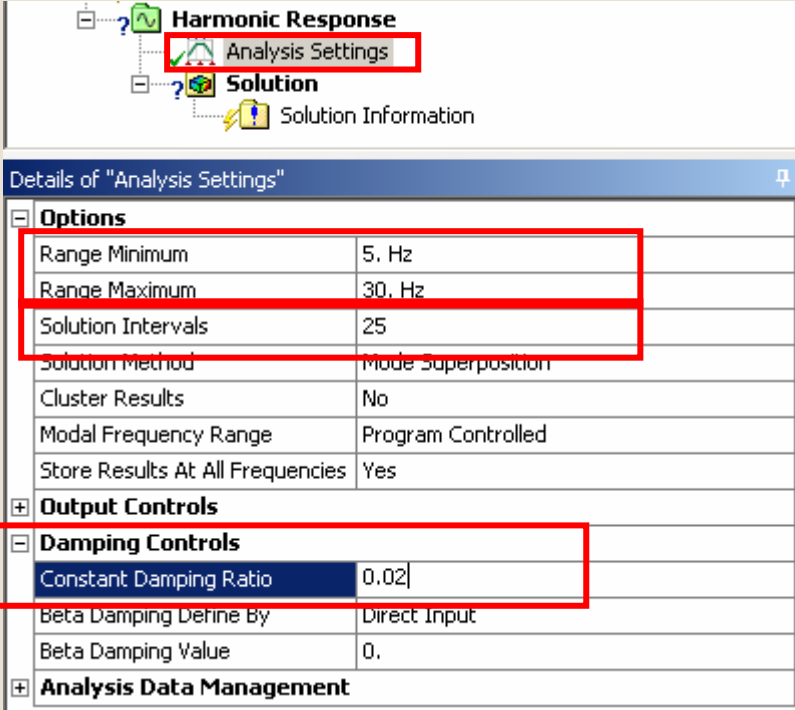
12

13



# Workshop 5 – 谐响应求解

- 在目录树中单击: Analysis Settings
- 
- 14. 在 Details 窗口中, 确认频率范围
- 15. 在“Solution Intervals”输入: 25
- 16. 在“Constant Damping Ratio”输入: 0.02



14

15

16

Details of "Analysis Settings"	
<b>Options</b>	
Range Minimum	5. Hz
Range Maximum	30. Hz
Solution Intervals	25
Solution Method	Mode Superposition
Cluster Results	No
Modal Frequency Range	Program Controlled
Store Results At All Frequencies	Yes
<b>Output Controls</b>	
<b>Damping Controls</b>	
Constant Damping Ratio	0.02
Beta Damping Define By	Direct Input
Beta Damping Value	0.
<b>Analysis Data Management</b>	



# Workshop 5 –谐响应求解

- 当 “Harmonic Response”分支数据准备完毕，即可求解谐响应分析。

- 经过最后的校核，所有的分支的符号必须是下面的一种：

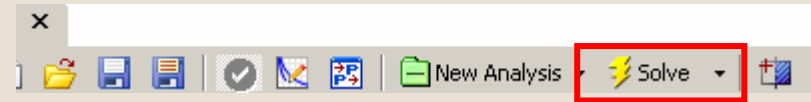
–  (准备好)

–  (完成)

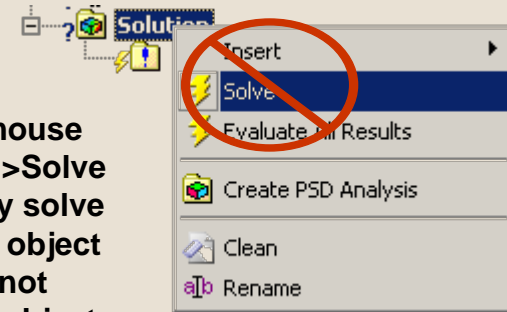
## 17. 求解.

– ToolBar Button >Solve

- 注意: 点击工具条的求解，将会求解所有的分支.
- 如果我们只希望求解一个分支的话，点击相应的分支进行求解即可.



17



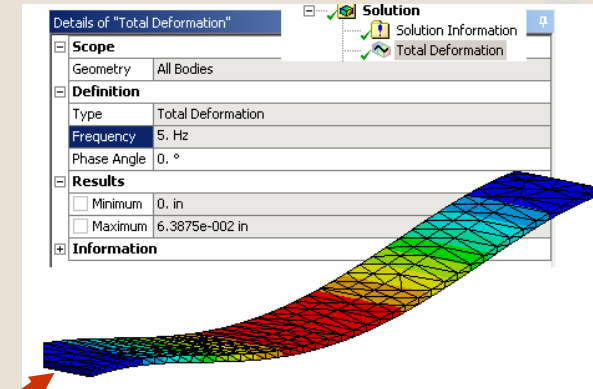
**The “mouse menu” >Solve may only solve a single object and not evaluate objects elsewhere in the Outline Tree**

# Workshop 5 – 结果

- 当谐响应分析求解完毕，查看相应的谐响应分析结果。
- 可以绘制在指定频率的云图。
- 可以绘制在频域内的曲线图。
  - 可以选择点、面等。
  - 可以绘制平均值、最小值、最大值等。

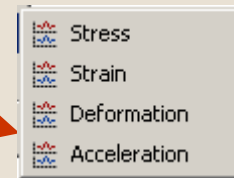
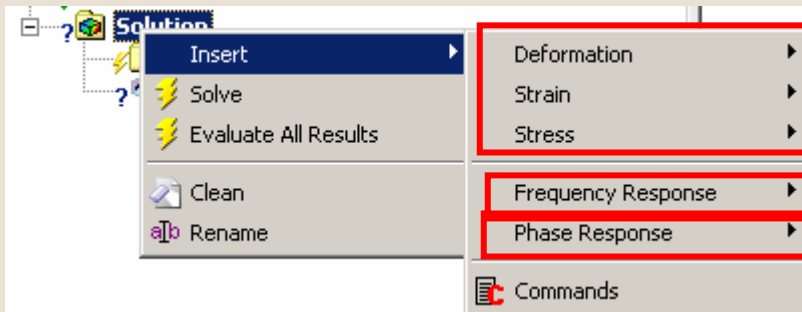
## 18. 在Solution分支下单击右键（RMB），选择“>Insert> Stress or Strain or Deformation”

- 将会插入结果对象
- 输入每一项结果所要求的几何或者其它输入

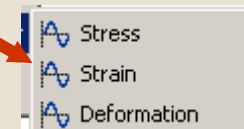


**Contours at a specific Frequency**

18



**Graphs**

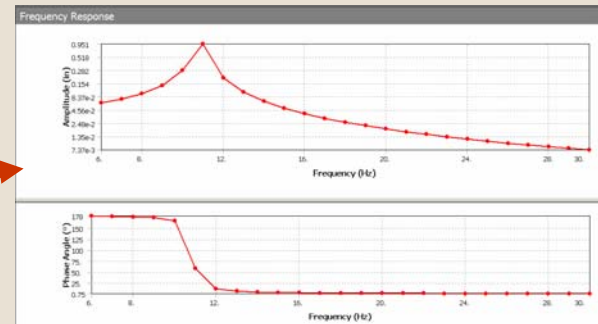
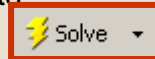


**Graphs**

# Workshop 5 – 谐响应分析结果

- 当查看频率与相位响应结果时，可以选择点、面等，可以绘制平均值、最小值或者最大值。
  - 比如：“>Insert>Frequency Response> Deformation”
19. 在Details中,对于Geometry，采用面选择选取梁的上顶面 然后点击 Apply
- 指定“Use Maximum”
20. 选择方向性变形：Y轴方向的变形
- Y方向是竖直方向
  - 采用 >Solve 或者单击右键 >Evaluate All Results

**Note: of course you can plan ahead and insert all the Harmonic Response Solution and Frequency Response Result objects when you originally inserted the Modal Solution. In that case you would >Solve them all at once, or you can review and add the objects incrementally as we have done here.**



Solution

- Solution Information
- Frequency Response

Details of "Frequency Response"

19 Scope

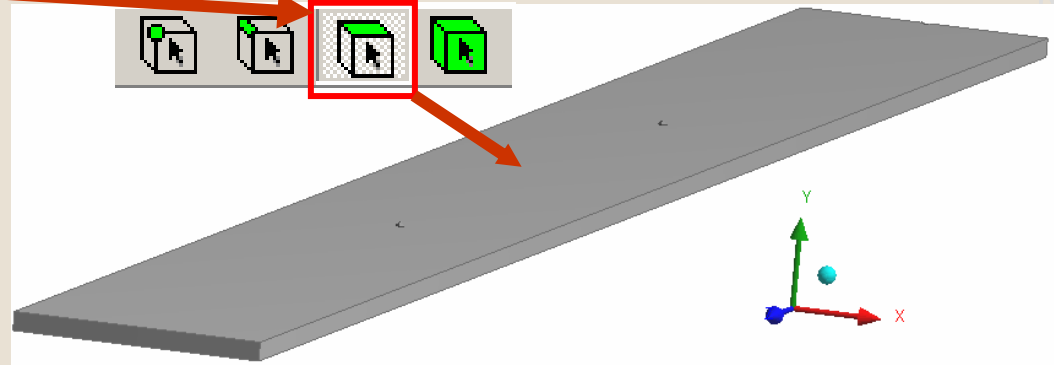
Geometry	Apply	Cancel
Spatial Resolution	Use Maximum	

20 Definition

Type	Directional Deformation
Orientation	Y

Options

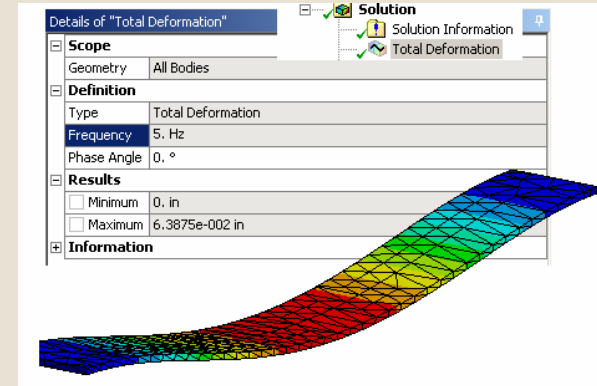
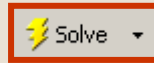
Frequency Range	Use Parent
-----------------	------------



# Workshop 5 - 谐响应分析结果

- 查看在不同指定频率下的变形和应力

21. 如果时间允许的话: 插入网格尺寸控制将网格细化, 改变扫频范围和求解频率间隔, 重新求解.

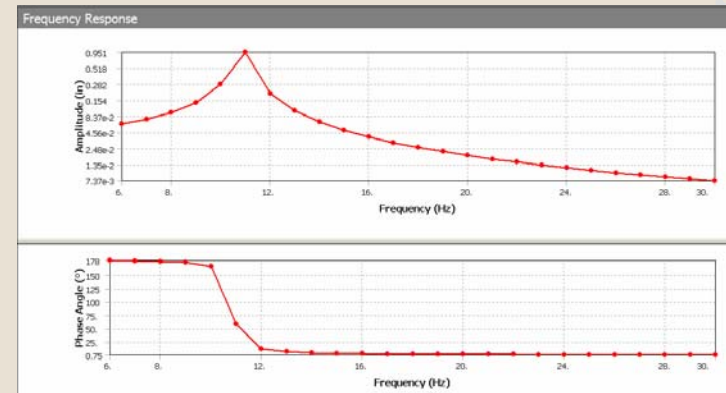


21

Details of "Analysis Settings"	
<b>Options</b>	
Range Minimum	0. Hz
<b>Range Maximum</b>	0. Hz
Solution Intervals	10

Outline for "beam"	
<b>Project</b>	
<b>Model</b>	
Geometry	
Mesh	
<b>Sizing</b>	
<b>Harmonic Response</b>	
Analysis Settings	



注意: 实际的求解结果会随网格、单位指定的不同等因素而稍有不同