ANSYS - Force Reaction Probe (Contact)
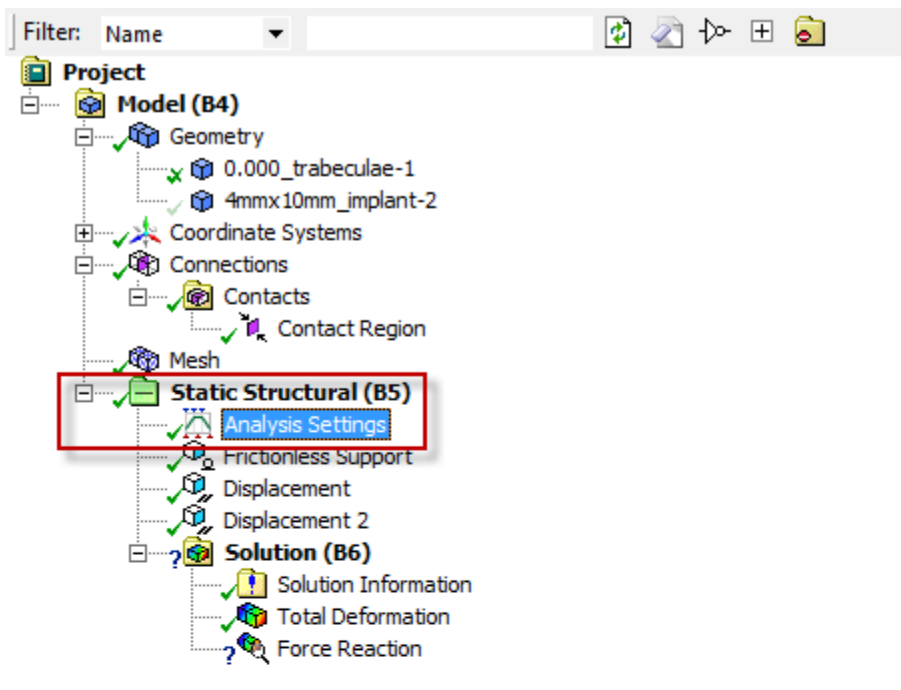
When selecting Force Reaction Probe, there are several options for location, as shown below;

The options, in order, are

- Boundary Condition
- Contact Region
- Remote Points
- Beam
- Spring
- Mesh Connection
- Surface

In order to include a reaction probe for a Contact region, you must first add an Output Control, under Analysis Settings back up under Static Structural. Set 'Nodal Forces' to 'Yes' as shown below;

Filter: Name ▼

- 📁 **Project**
  - 📦 **Model (B4)**
    - ✓ 🔷 Geometry
      - ✗ 🔷 0.000_trabeculae-1
      - ✓ 🔷 4mmx10mm_implant-2
    - ✓ ⚹ Coordinate Systems
    - ✓ 🔷 Connections
      - ✓ 📦 Contacts
        - ✓ 🔲 Contact Region
    - ✓ 🔷 Mesh
    - ✓ 🟩 **Static Structural (B5)**
      - ✓ 🔼 Analysis Settings
      - ✓ 🔲 Frictionless Support
      - 🔷 Displacement
      - 🔷 Displacement 2
      - ? 📦 **Solution (B6)**
        - ✓ 🔷 Solution Information
        - 🔷 Total Deformation
        - ? 🔍 Force Reaction

Details of "Analysis Settings" 📌

⊟ **Step Controls**	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
⊟ **Solver Controls**	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Solver Pivot Checking	Program Controlled
Large Deflection	Off
Inertia Relief	Off
⊞ **Restart Controls**	
⊞ **Nonlinear Controls**	
⊟ **Output Controls**	
Stress	Yes
Strain	Yes
Nodal Forces	Yes
Contact Miscellaneous	No
General Miscellaneous	No
Store Results At	All Time Points
⊞ **Analysis Data Management**	
⊞ **Visibility**	

⚠ **Also, set 'Contact Miscellaneous' to 'Yes' as well**

Then you must RE-SOLVE THE SOLUTION, or the new output control that you included will not be taken into effect. To do this, delete the probe and then rerun. Once complete, you may re-insert the reaction probe and select Contact, and you will no longer get an error from Output Controls.