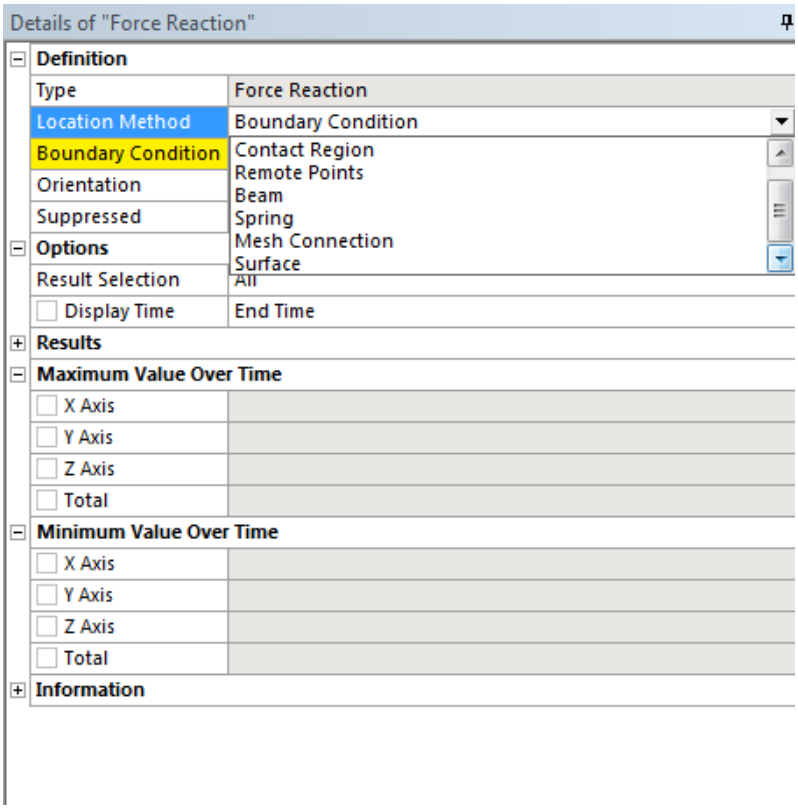
# ANSYS - Force Reaction Probe (Contact)

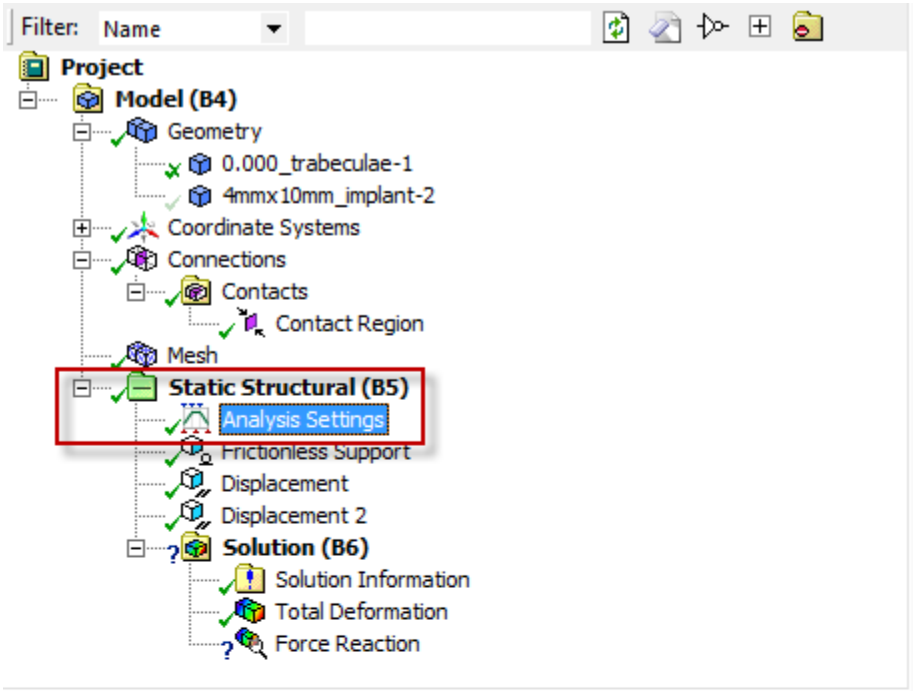
When selecting Force Reaction Probe, there are several options for location, as shown below;




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;




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.