ANSYS®

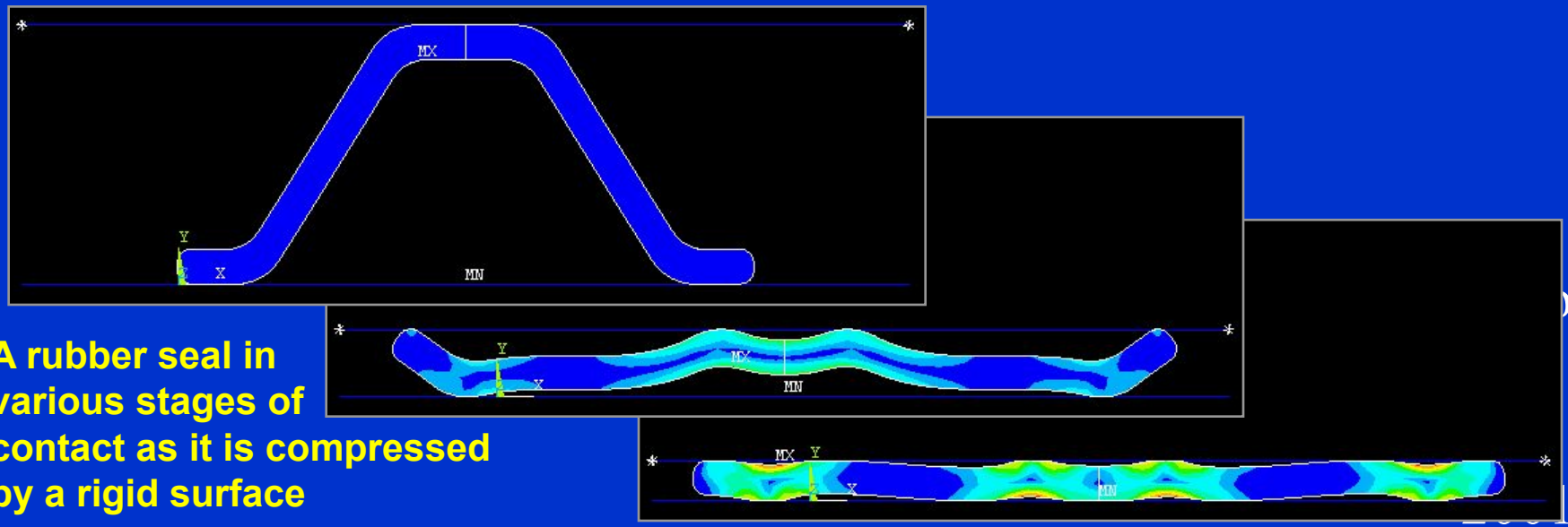
Module 9

Bonded Contact

9. Bonded Contact

ANSYS Training Manual

- Contact between two objects is one of the most frequently encountered phenomena in engineering analysis.
- It is also one of the most difficult nonlinearities to handle because the stiffness can suddenly disappear or reappear depending on whether the objects are out of contact or in contact.



A rubber seal in various stages of contact as it is compressed by a rigid surface

...Bonded Contact

ANSYS Training Manual

- Most contact analyses require advanced analytical techniques that are beyond the scope of this training course.

  However, one class of contact analysis, known as *assembly contact,* can often be performed without knowledge of advanced techniques.

- Assembly contact uses the "bonded" option of ANSYS contact elements and is also called ***bonded contact***.

- In this chapter, we will briefly describe how to set up and solve a bonded contact analysis:

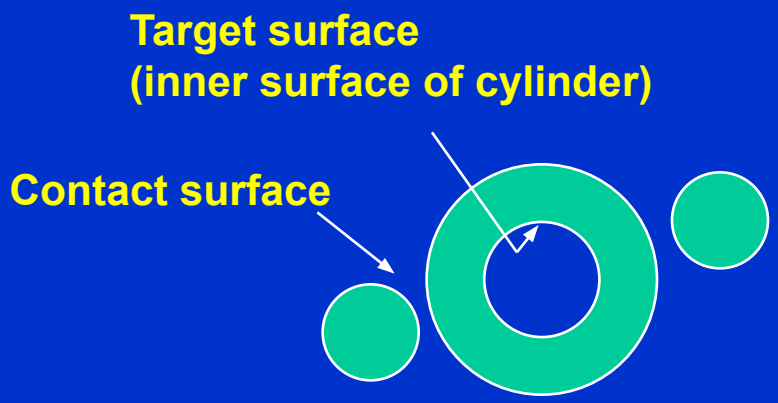
  A. Definitions

  B. Typical Procedure

  C. Workshop

A. Definitions

- *Bonded Contact* is a special case of contact analysis where the two contacting surfaces are assumed to be "glued" together throughout the analysis.
- The two contacting surfaces form a *contact pair.*
  - One of the surfaces is designated as the *target surface.*
  - And the other surface is called the *contact surface.*

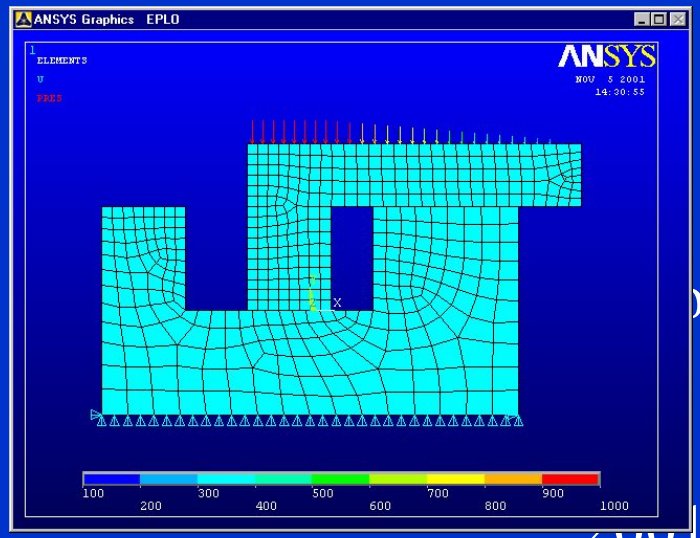


Advantages of bonded contact:

- **Faster solutions since there are no contact convergence issues. Convenient for a quick analysis of assemblies, for example.**
- **Small-deflection cases can be run as linear analyses with one substep and one equilibrium iteration.**
- **Also allows large-deflection (nonlinear) analyses. (Coupling and constraint equations are not recommended for nonlinear analyses.)**

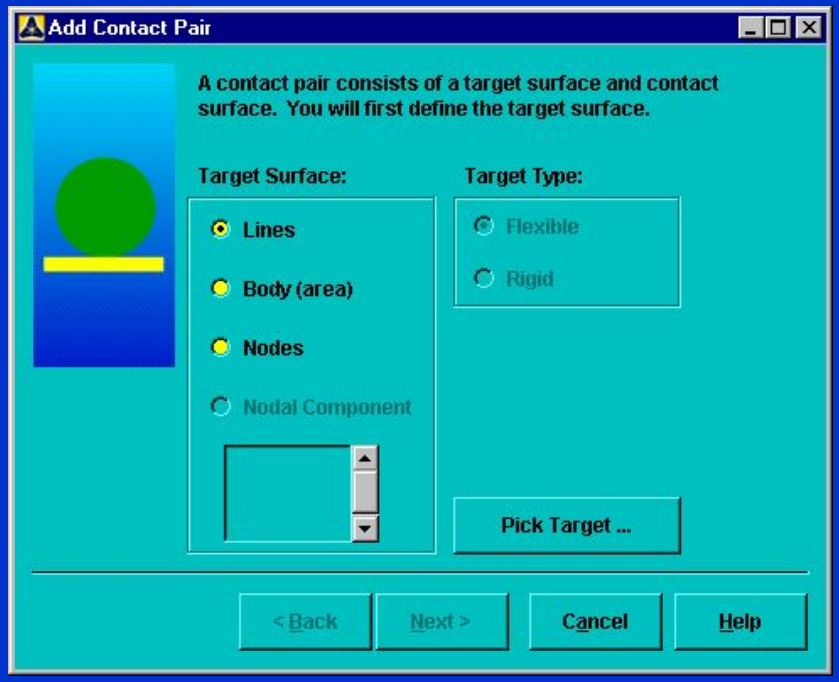
B. Typical Procedure

- **Seven main steps:**
  1. **Create or import the geometry.**
  2. **Mesh all of the contacting bodies. (Required for step 3.)**
  3. **Create the contact pair.**
  4. **Specify the analysis type and solution controls.**
  5. **Apply loads and boundary conditions.**
  6. **Save the database.**
  7. **Solve and review results.**
- **We will expand on steps 3 and 4 next.**



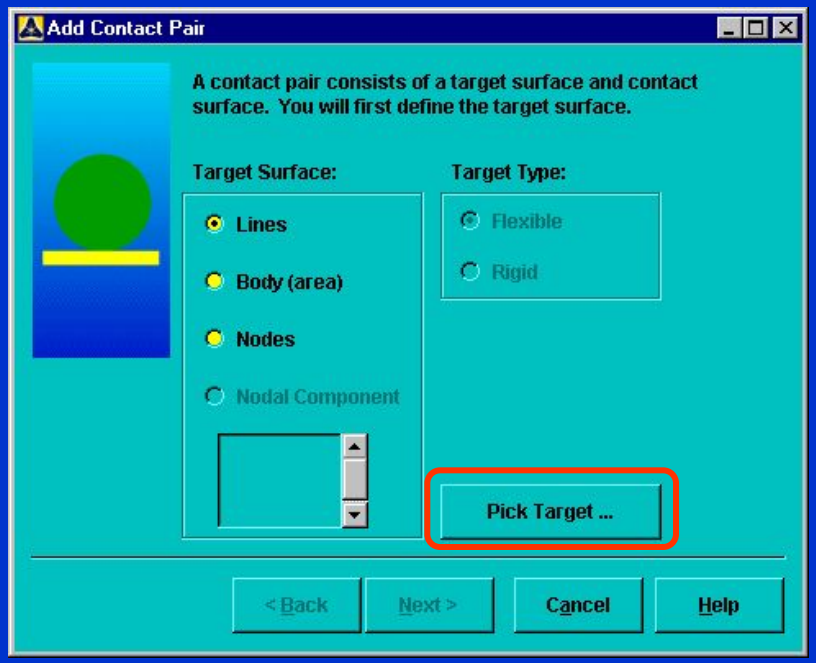
Creating the contact pair

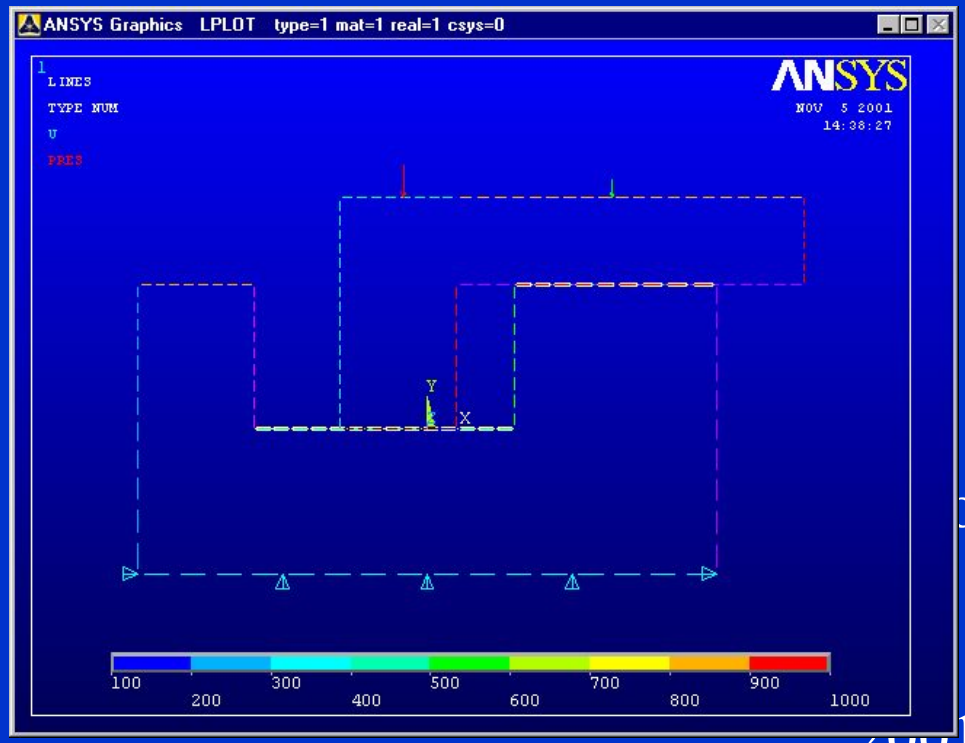
- **Once the contacting bodies have been meshed, the next step is to create the contact pair, which consists of *target surface* elements and *contact surface* elements.**
- **The *contact wizard* provides an easy way to do this.**
  - Preprocessor > Create > Contact Pair > Contact Wizard...



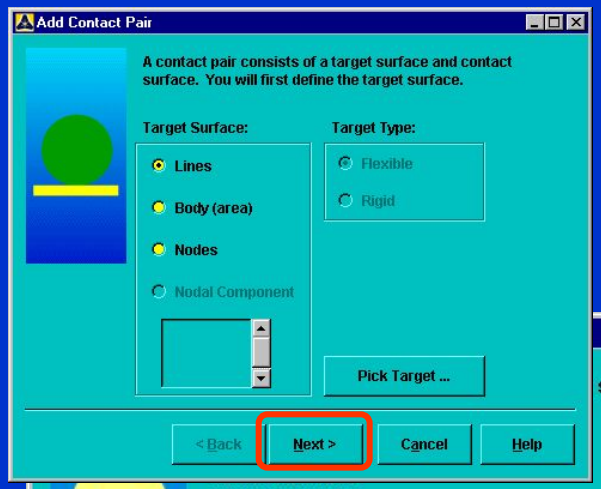
…Typical Procedure

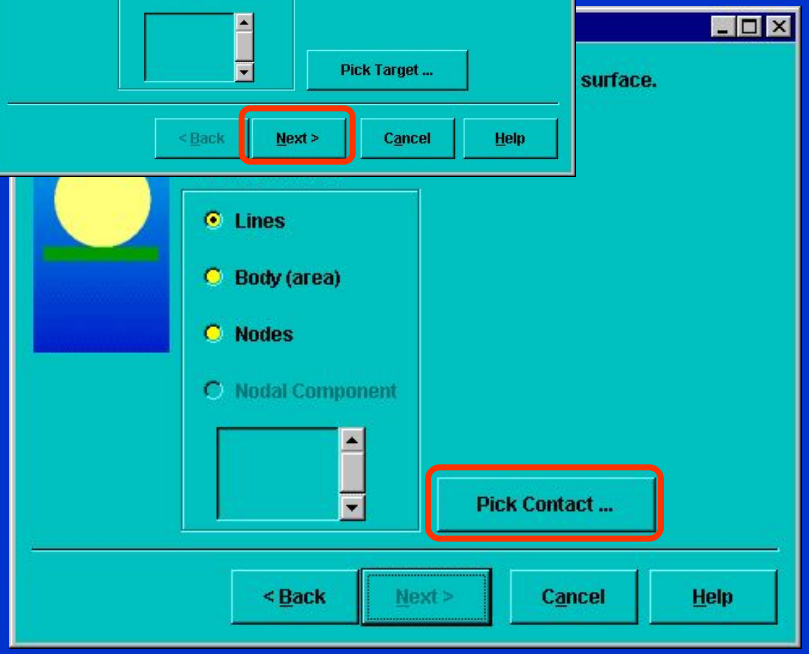
- First pick the target surface(s) on one part.

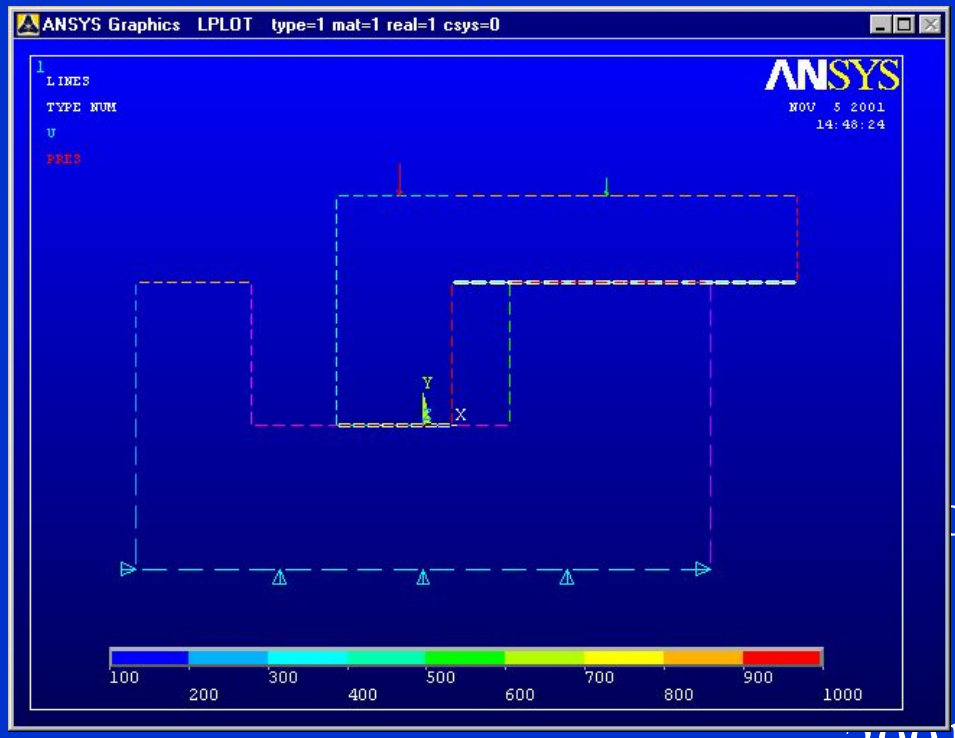




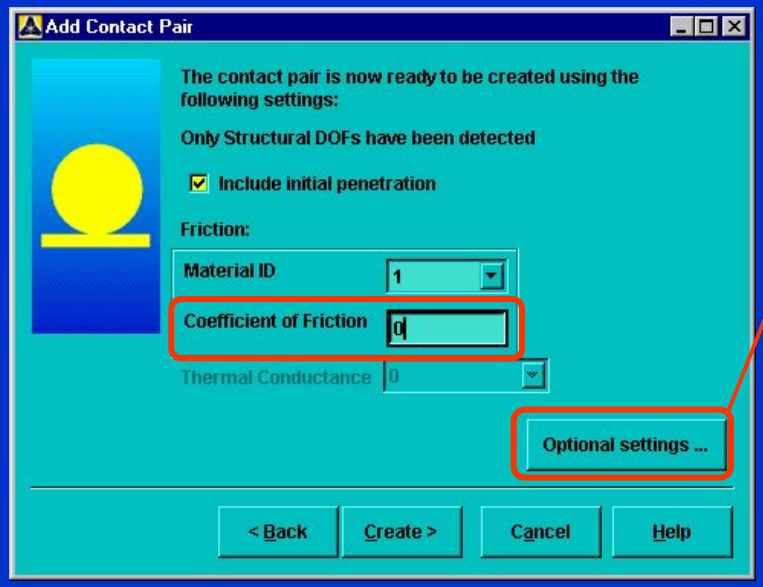
- Then pick the contact surface(s) on the other part.

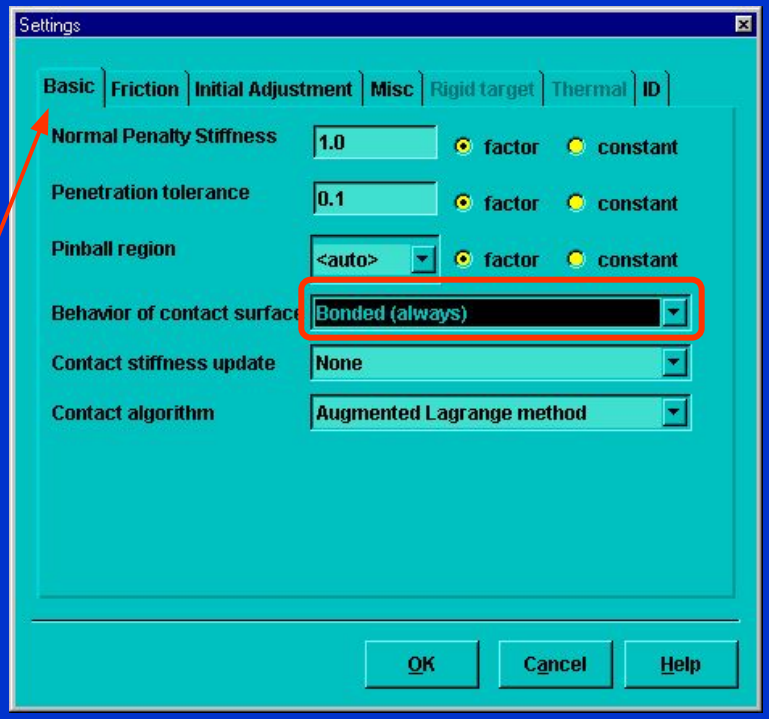




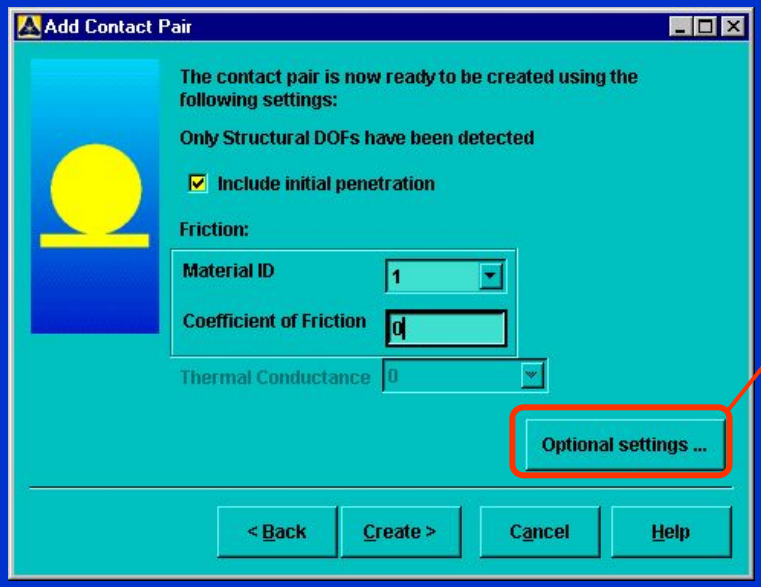


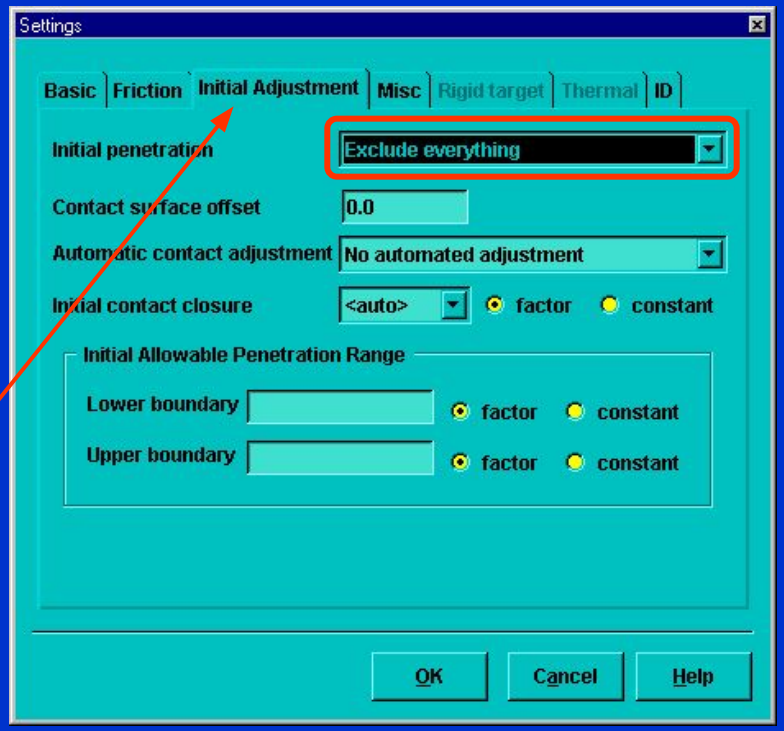
- Then establish contact settings. Many settings are available, but the common ones for bonded contact are:
  - Coefficient of friction = 0
  - Then under Optional Settings > Basic tab:
    - Behavior of contact surface = "Bonded (always)"



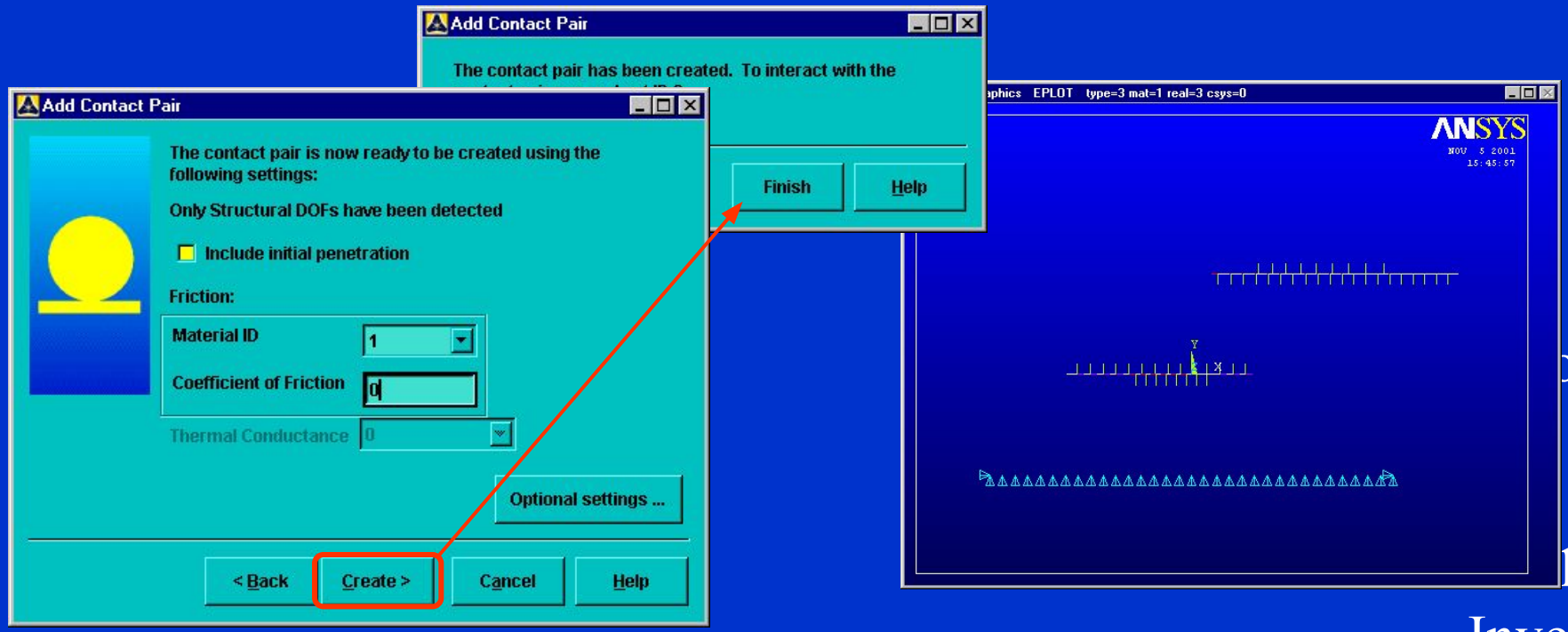


- Contact settings (cont'd):
  - Then under Optional Settings > Initial Adjustment tab:
    - Initial penetration = "Exclude everything"





- **Finally, generate the contact pair.**
  - **ANSYS will create the contact and target elements, and identify the contact pair with a real constant set number.**
  - **The contact pair is plotted with their element normals, which should be pointing toward each other. (If not, you can flip them using** Preprocessor > Create > Contact Pair > View and Edit ...)

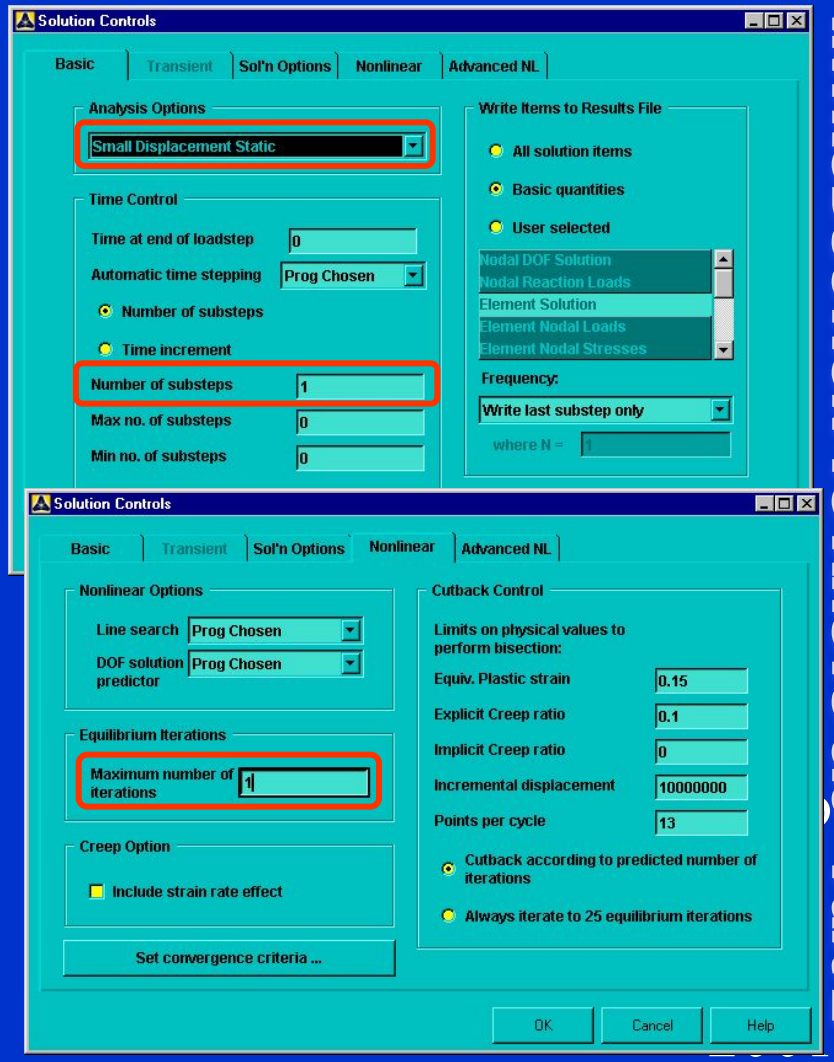


Analysis type and solution controls

- Both static and modal analyses can be performed.
- Typical solution control settings for static analysis:

  Solution > Sol'n Control…

  - Small displacement static.
  - One substep [nsubst,1], which is the default.
  - One equilibrium iteration [neqit,1]. This will cause a warning to be issued, but it is generally acceptable for bonded contact.



…Typical Procedure

- Demo:
  - Resume contact.db (contains two bodies made of aluminum, meshed with PLANE82 elements)
  - Bring up contact wizard and create contact pair using:
    - target surfaces on bottom part
    - contact surfaces on top part
    - MU = 0
    - initial penetration = exclude everything
    - contact behavior = bonded (always)
  - Enter Solution and issue the following commands (in order):
    - solc,off
    - neqit,1
  - Solve
  - Plot SEQV, then animate it. Also show UX and UY contours to demonstrate continuity due to bonded contact.

C. Workshop

- This workshop consists of the following problem:

  W8. Swaybar and Shaft Assembly

  Please refer to your *Workshop Supplement* for instructions.

October 30, 2001 Inventory