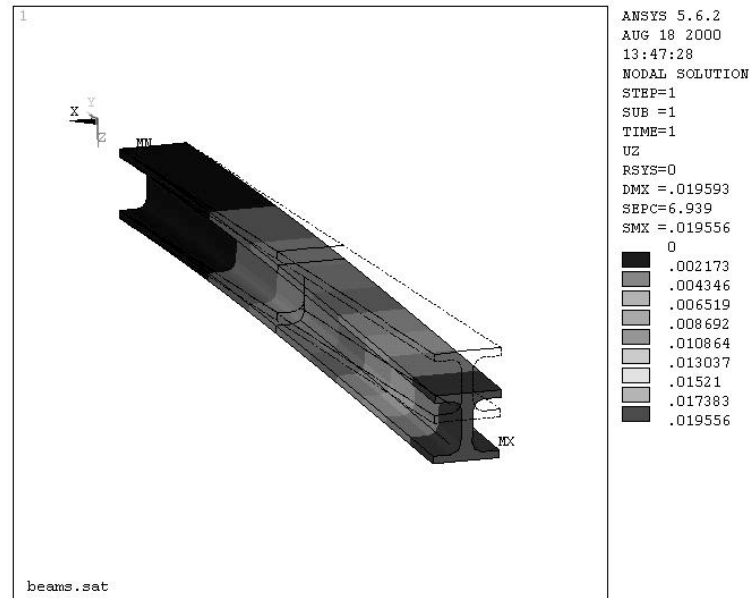


Connecting Parts - A study of Continuous Mesh versus Bonded Contact versus Constraint Equations



Sean Harvey

August 2000

CSI Tip of the Week

Introduction

Different ways of connecting parts

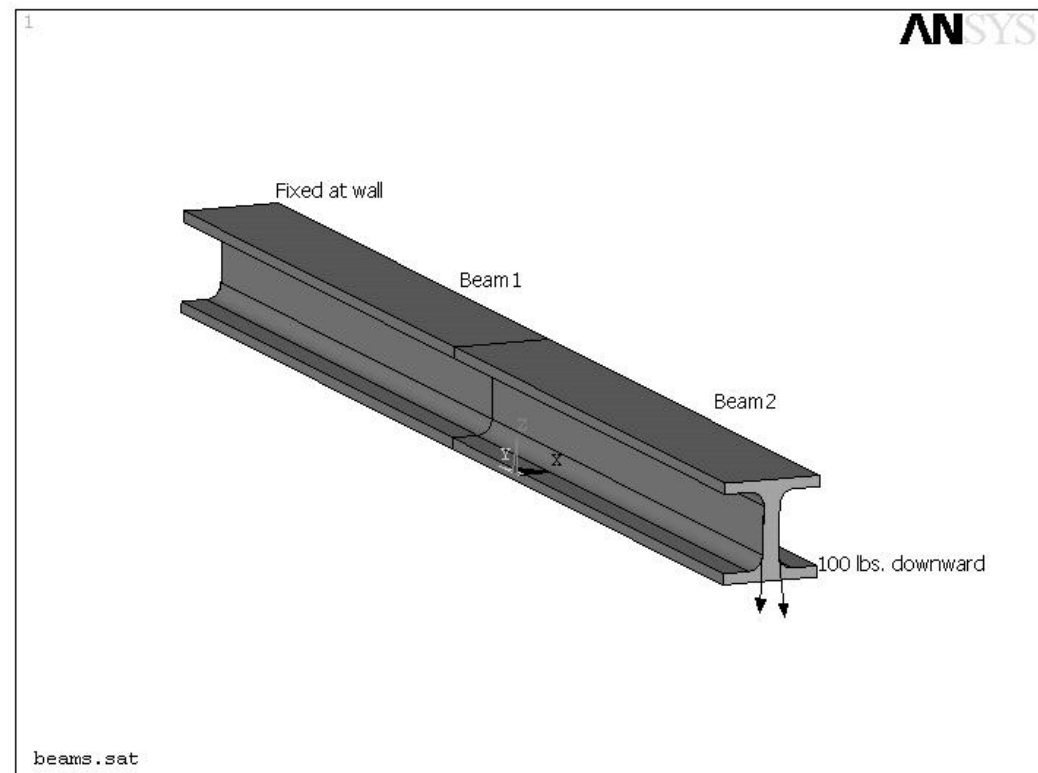
- There are several ways to connect two finite element parts together in ANSYS
- For this study, two solid meshed beams are connected together end to end with 3 different types of connectivity. Continuous mesh (shared common nodes), bonded contact, and constraint equations. The advantages, disadvantages, and results are discussed.
- These examples shown here just illustrate the techniques and a single particular set of results. *Your particular results may vary.*

CSI ANSYS Tip of the Week

Introduction

Model Description

- 2 Steel Beams connected end to end
- Fixed at wall
- 100 lbs. Evenly distributed over nodes on other end.
- Meshed with 8 noded brick elements (solid45)
- Section $I_{xx} = .847 \text{ in}^4$
- Section Area = 1.4877 in^2
- Modulus = $27.9 \text{e}6 \text{ psi}$
- Length = 24"



CSI ANSYS Tip of the Week

Introduction

Different ways of connecting parts

Continuous mesh - 2 Parts that share common nodes at their boundaries

- Advantages
 - Continuous stress between parts (in general)
 - No limitations on type of analysis, and large deflections are allowed
- Disadvantages
 - Mesh must be the same between parts. For mapped meshed assemblies, the node count can be large when mesh sizes can't change from part to part.
 - Once parts are glued, difficult to move parts w/o rebuilding the model.

CSI ANSYS Tip of the Week

Introduction

Different ways of connecting parts

Bonded Contact - 2 Parts that have surface to surface contact elements between solid element faces

- Advantages

- Dissimilar meshes allowed between pieces
- Large deflections allowed
- Can easily move parts around (chips on a board)
- Allow many heat transfer options (beta @5.6, Production @5.7)

- Disadvantages

- Contact must be manually created between parts
- Stresses between parts is discontinuous
- Low contact stiffness can adversely affect results.

CSI ANSYS Tip of the Week

Introduction

Different ways of connecting parts

Constraint Equations - 2 Parts with dissimilar meshes that have constrain equations written with the ceintf command. These equations "tie" two dissimilar meshes together.

- Advantages

- Dissimilar meshes allowed between pieces
- No additional elements or spring stiffnesses introduced

- Disadvantages

- Equations need to be generated for each interface
- Stresses between parts is discontinuous
- Large deflections are not allowed.
- Slower solutions and increased memory requirements
- Not allowed with new domain solver.

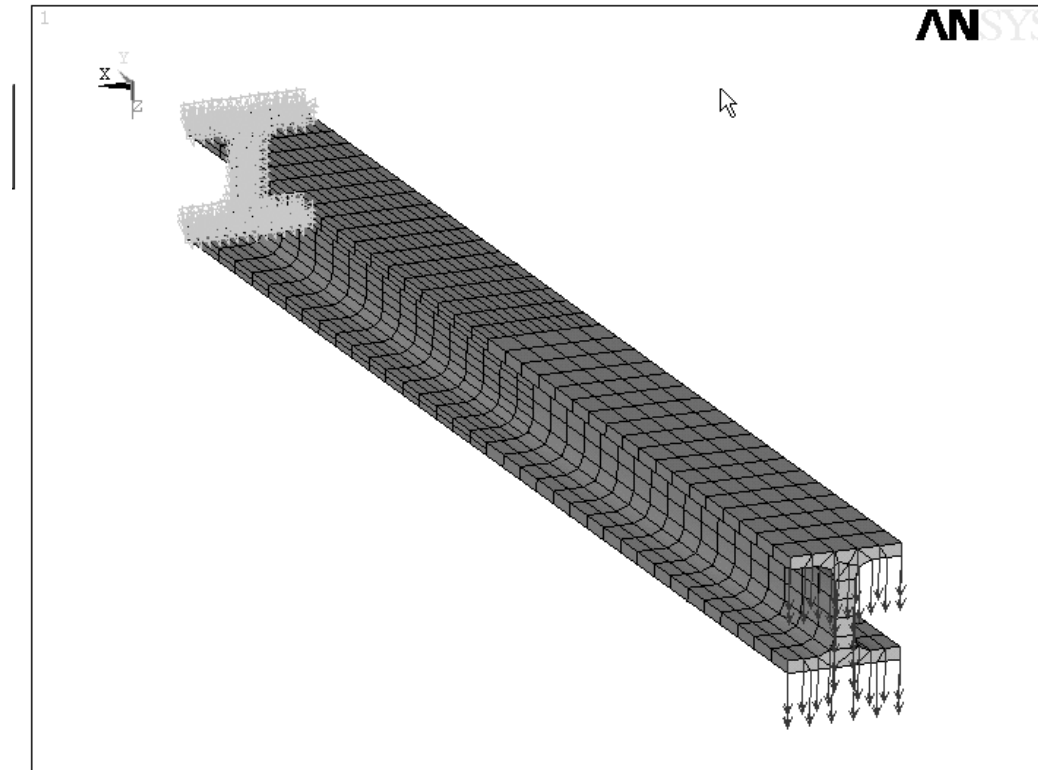
CSI ANSYS Tip of the Week

Sample Problem

- The following steps will detail creating bonded contact.

Sample Beam Problem with Bonded Contact

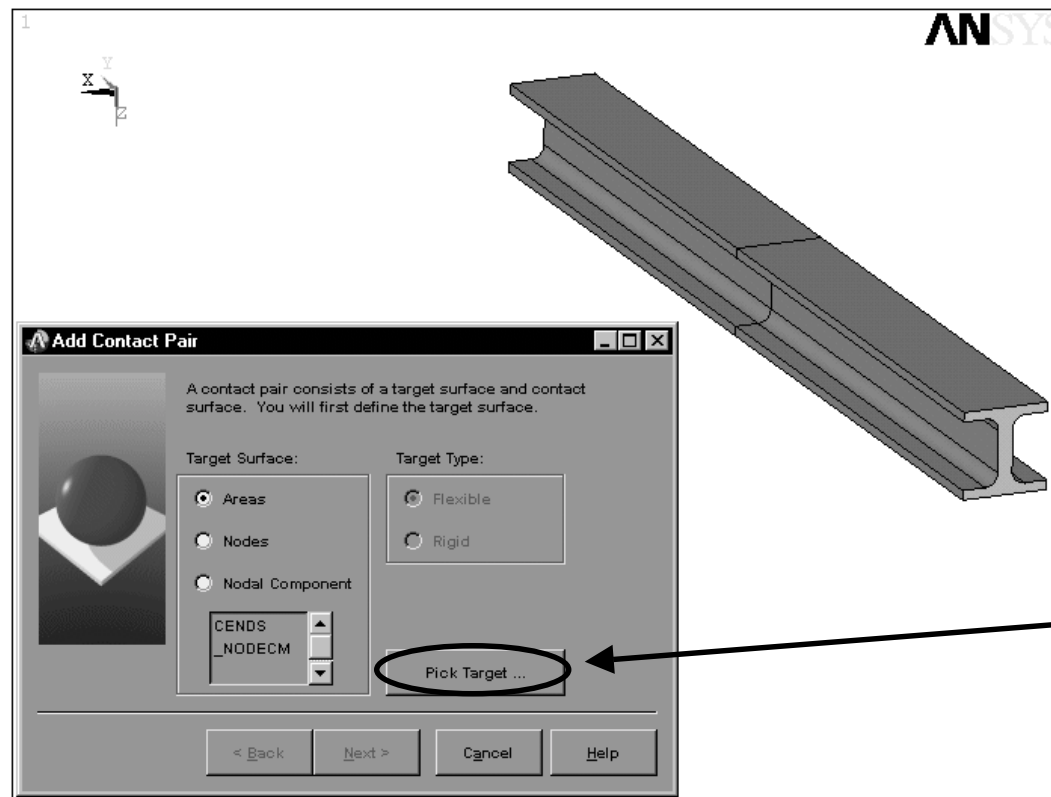
Model of 2 beams that do not share common nodes, meshed, and boundary conditions applied.



CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard

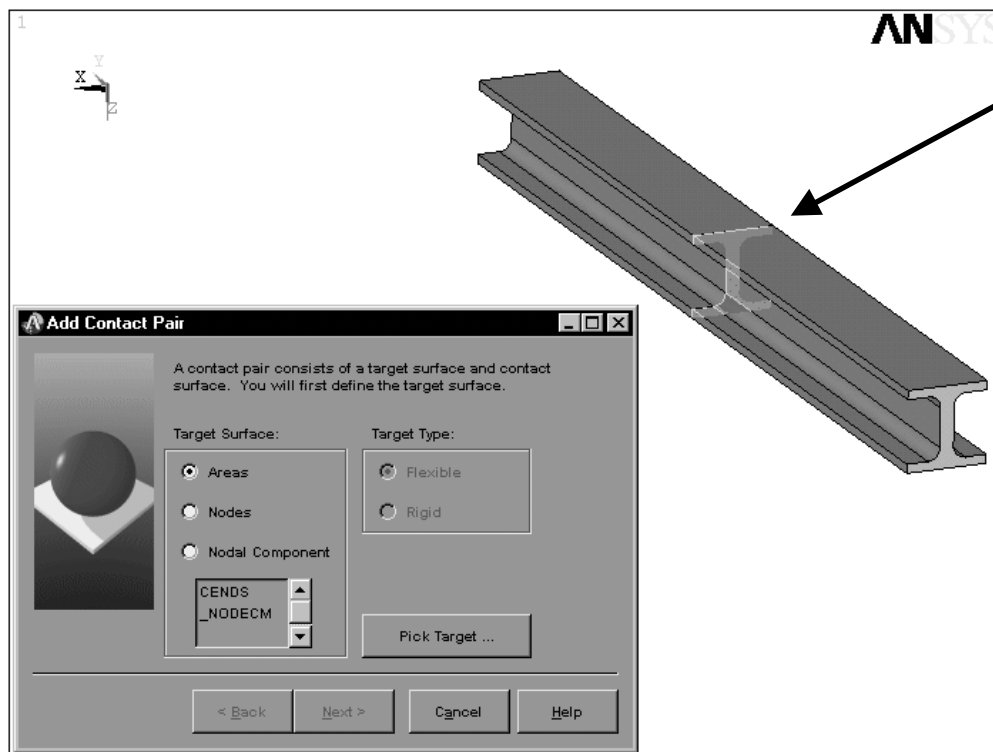


•Pick "Pick Target"

CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard

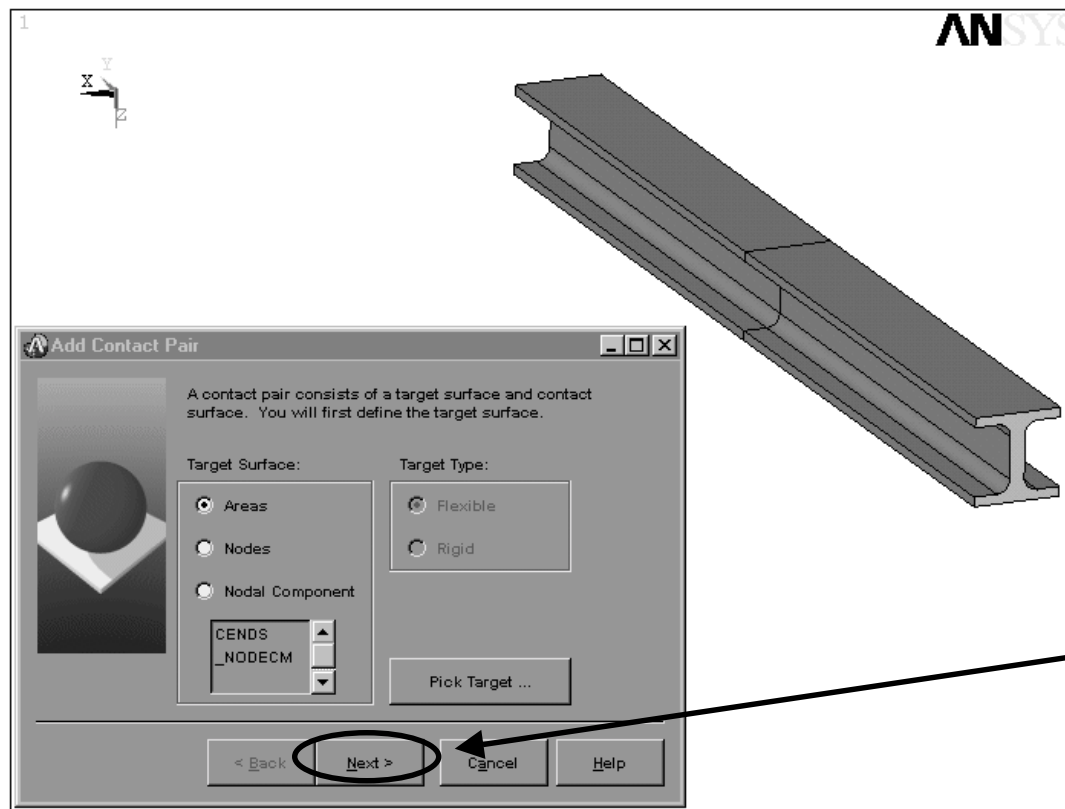


- Pick one of the two areas at the interface.
- In general the target area should be the coarser meshed area.

CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard

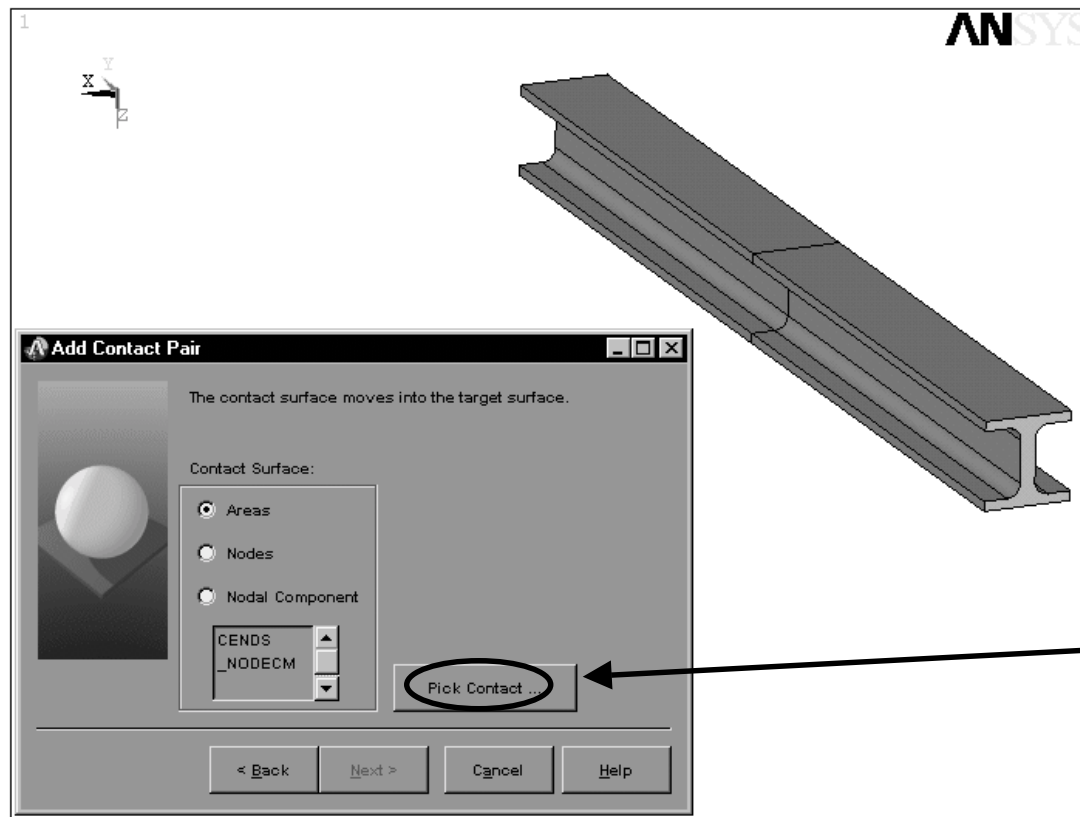


•Pick "Next"

CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard

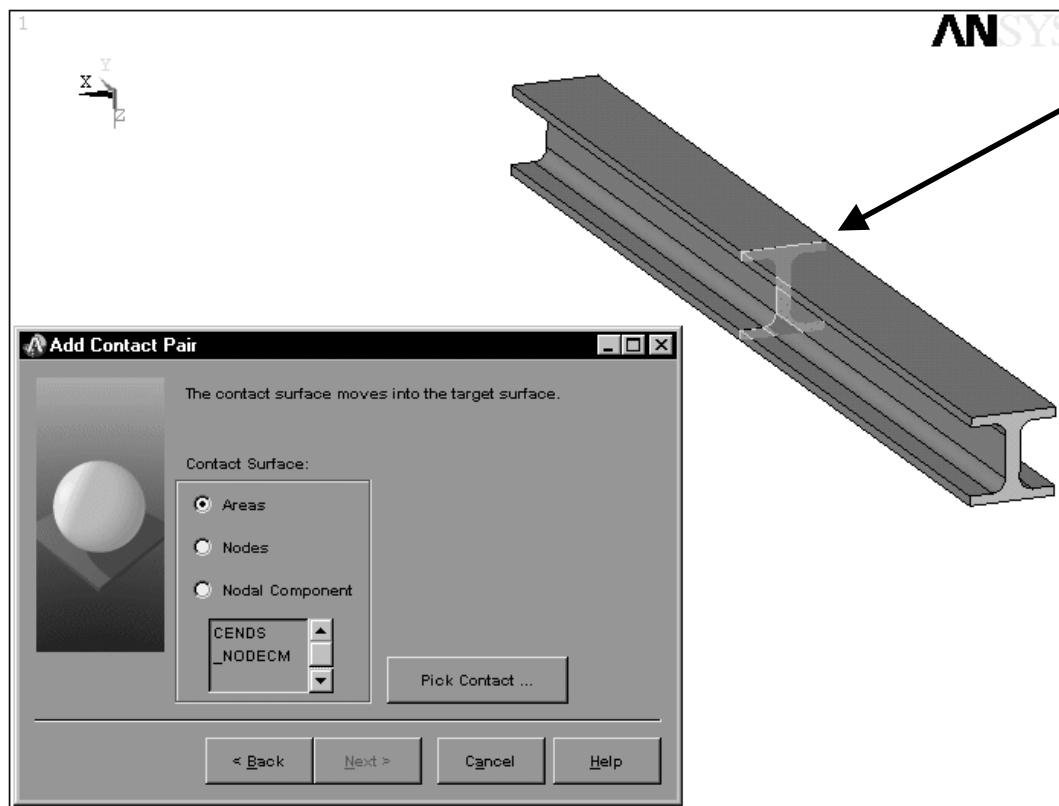


•Pick "Pick Contact"

CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard

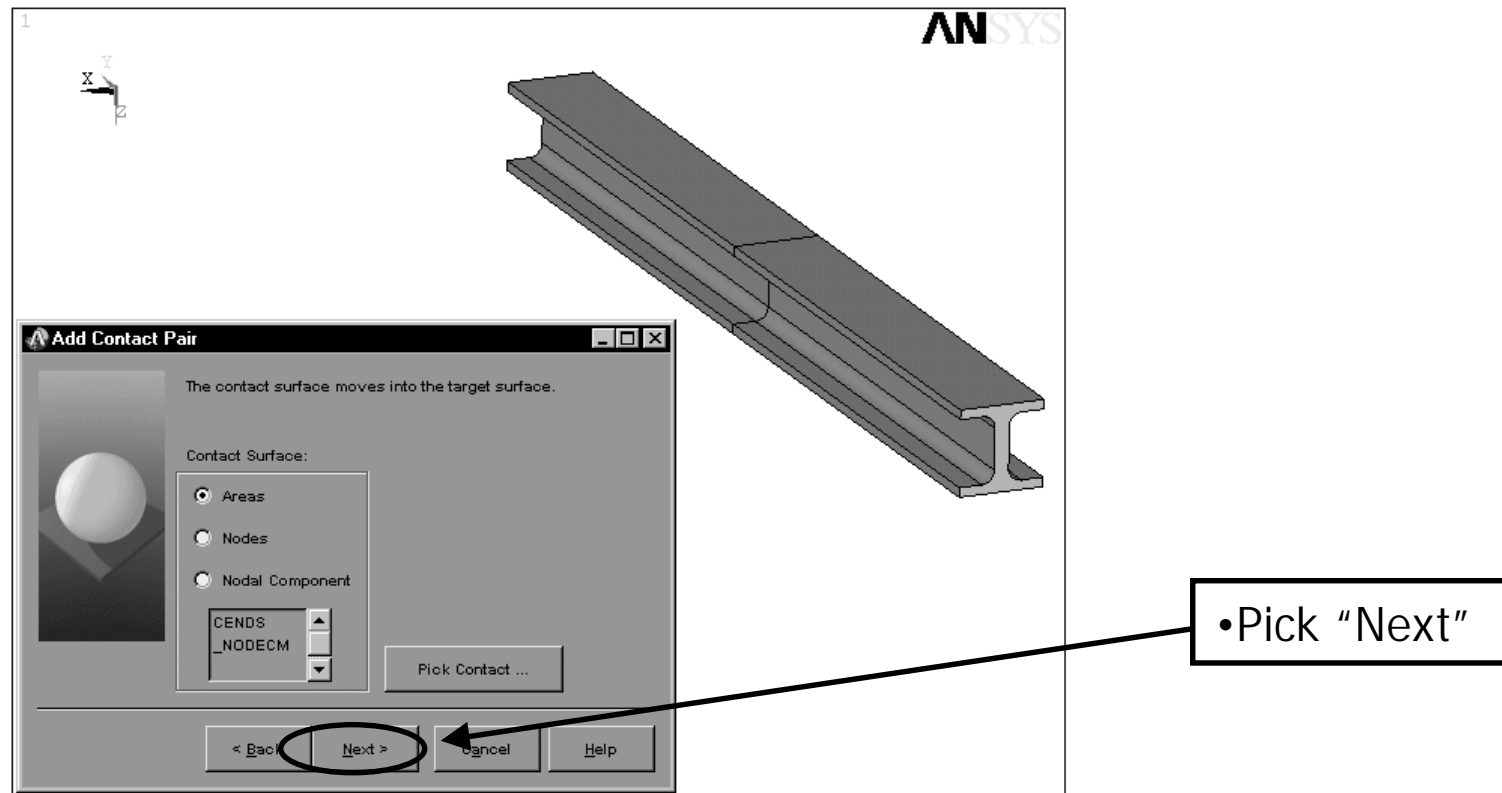


- Pick second of the two areas at the interface
- In general the contact area should be the finer meshed area.

CSI ANSYS Tip of the Week

Create Bonded Contact

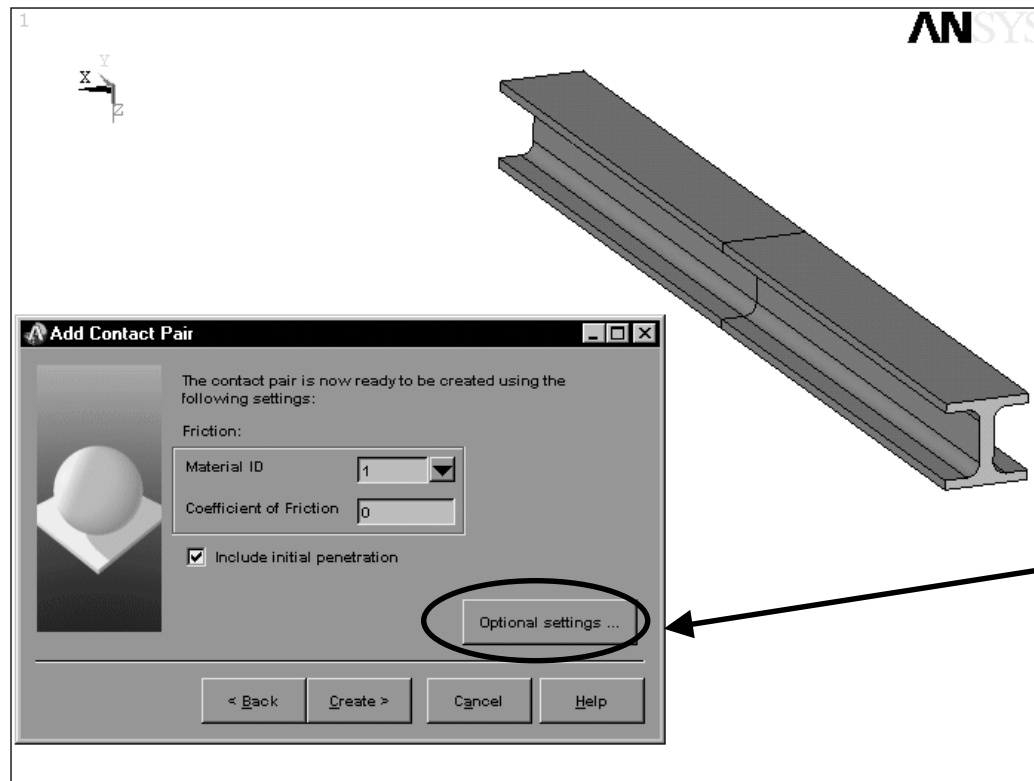
ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard



CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard

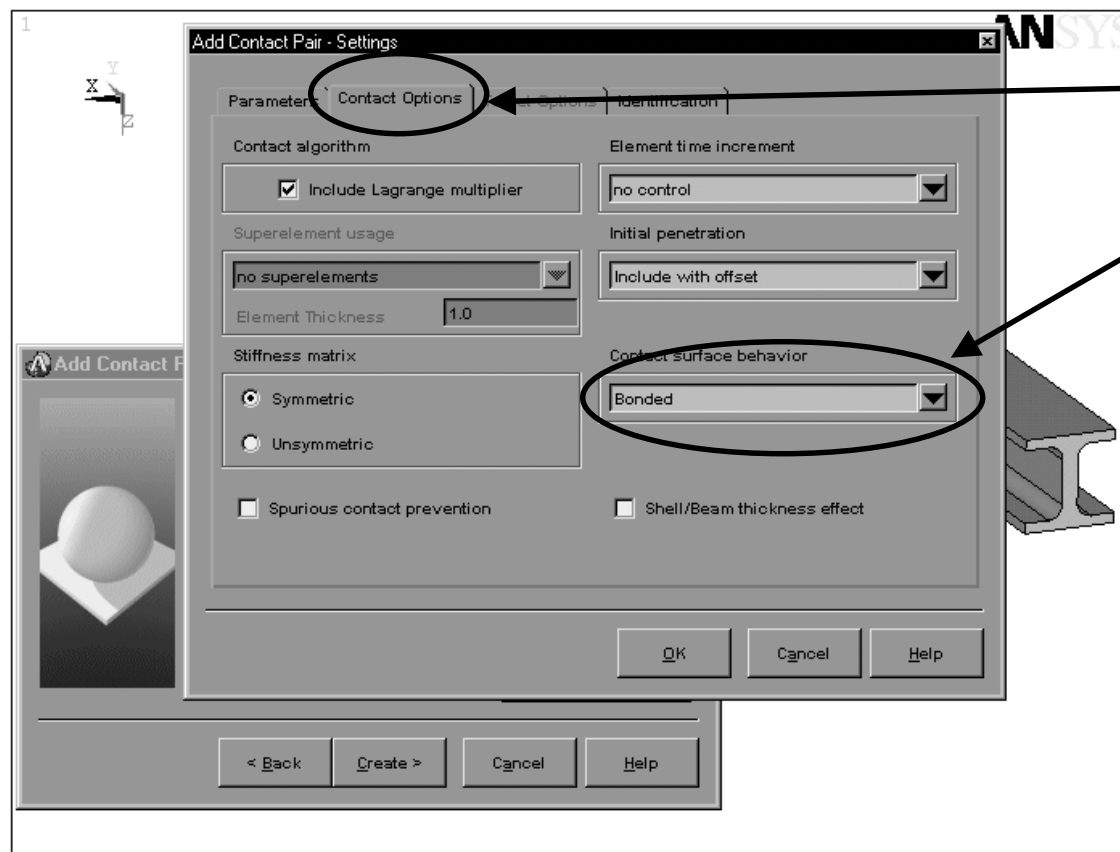


•Pick "Optional Settings"

CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard

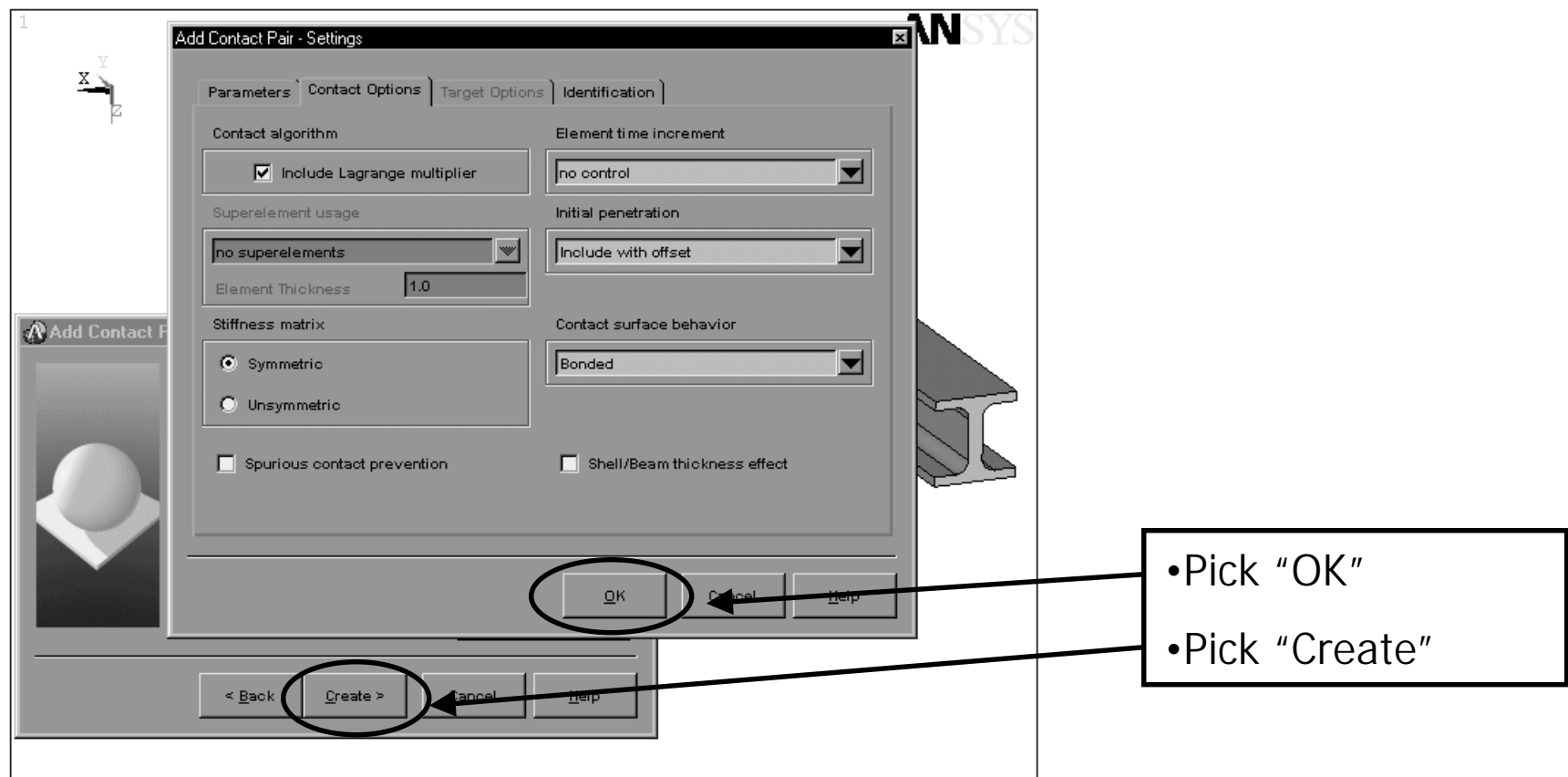


- Accept default parameters, i.e stiffness multiplier FKN, etc.
- Pick "Contact Options" then specify "Bonded"

CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard



CSI ANSYS Tip of the Week

Create Bonded Contact

ANSYS Main Menu > Preprocessor > Create > Create Contact Pair > Contact Wizard



- Contact Elements have been created
- Pick Finish

CSI ANSYS Tip of the Week

Bonded Contact Solution Options

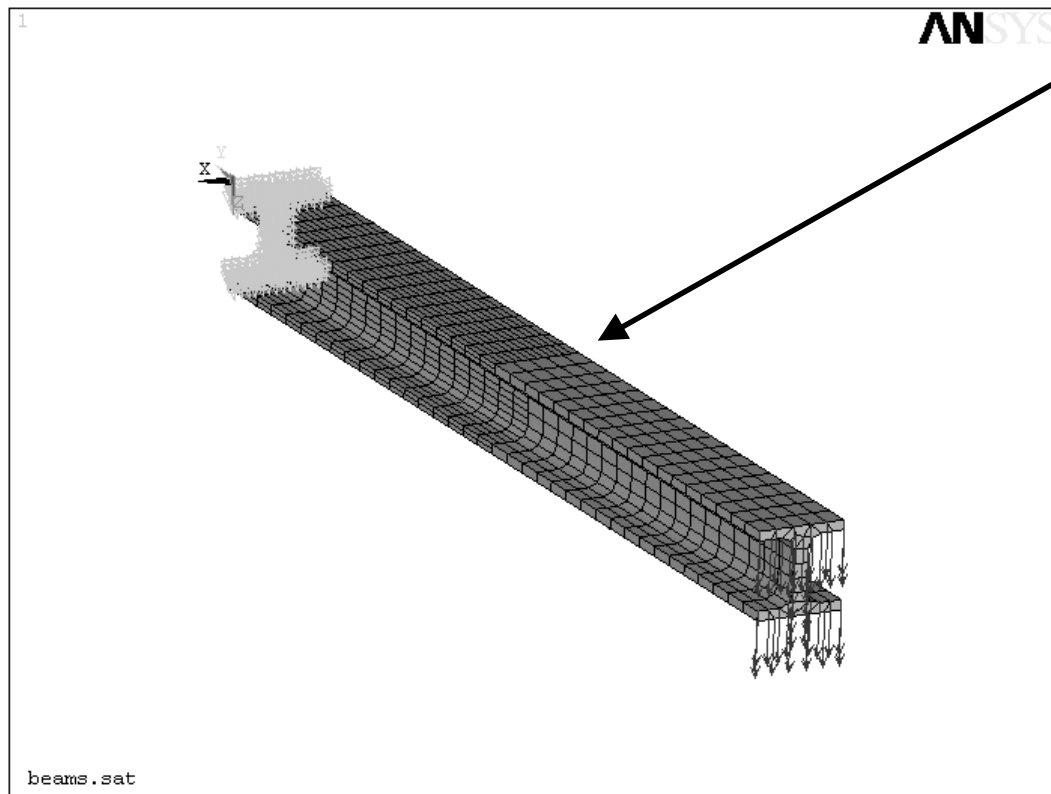
- With contact elements, solution is non-linear, but with the bonded option, the element status never changes i.e always in contact.
- Additionally, if solution is small deflection, the change in stiffness due to element rotations is negligible.
- In these cases, there is no need to perform multiple sub-steps, nor multiple iterations to solve this problem, but ANSYS will iterate by default.
- Specify **solcon,off** and **neqit,1** to solve this problem with 1 iteration. This is called linear bonded contact. This can save lots of time!

CSI ANSYS Tip of the Week

Sample Beam Problem with Constrain Equations

- The following steps will detail creating constraint equations to joint parts in a model.

Create Constraint Equations at interface

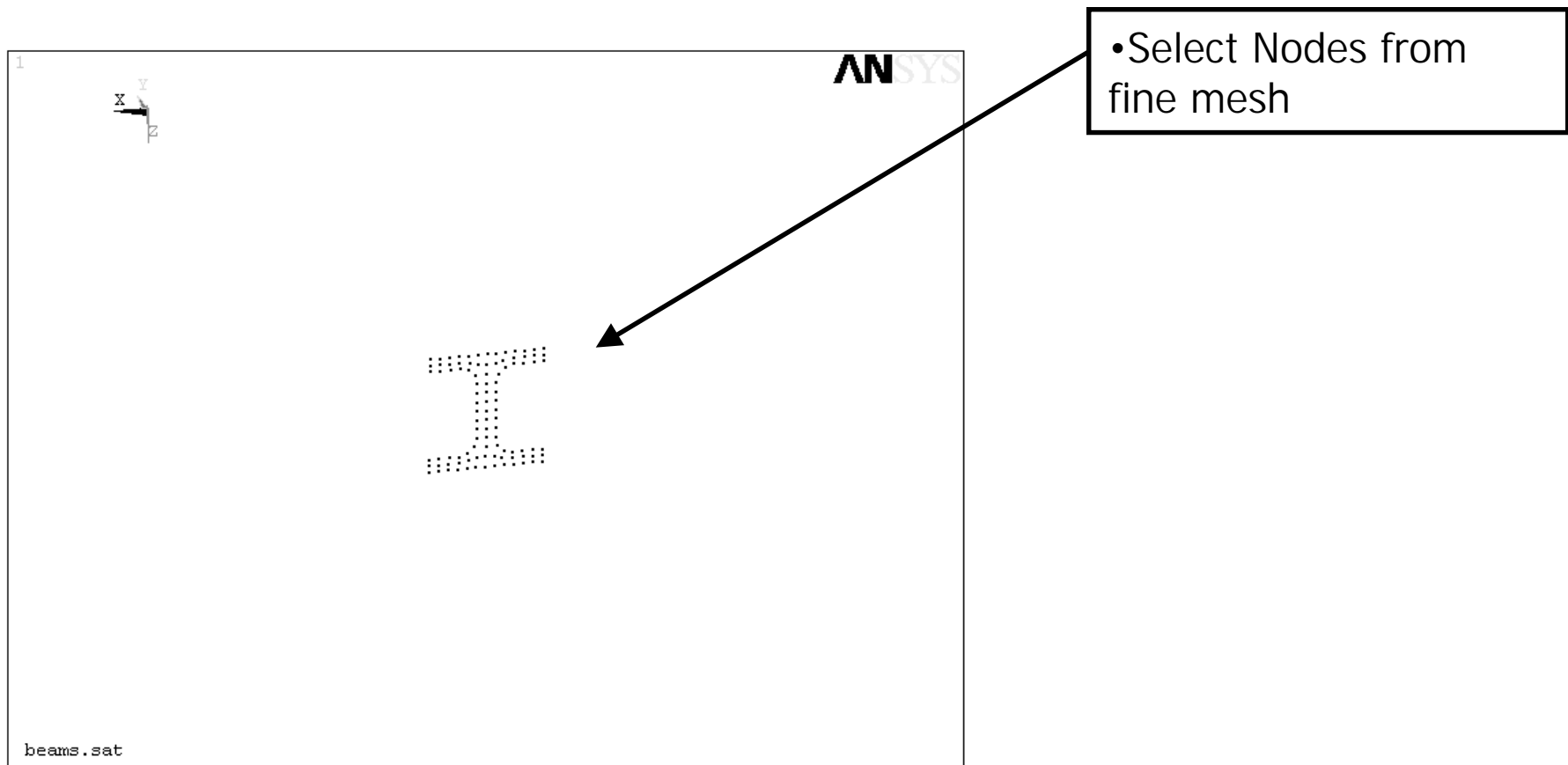


- Need to select nodes from the finer mesh and elements from the coarser mesh at the interface.

CSI ANSYS Tip of the Week

Create Constraint Equations at interface

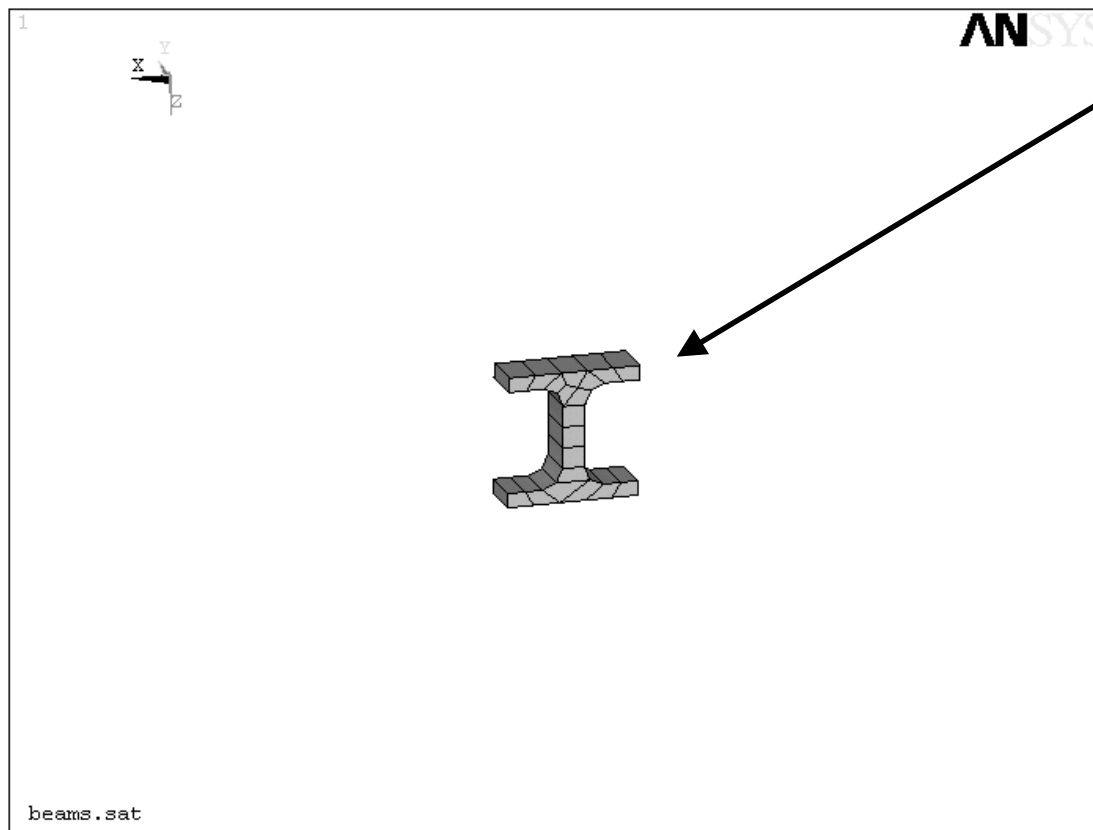
ANSYS Utility Menu > Select Entities



CSI ANSYS Tip of the Week

Create Constraint Equations at interface

ANSYS Utility Menu > Select Entities



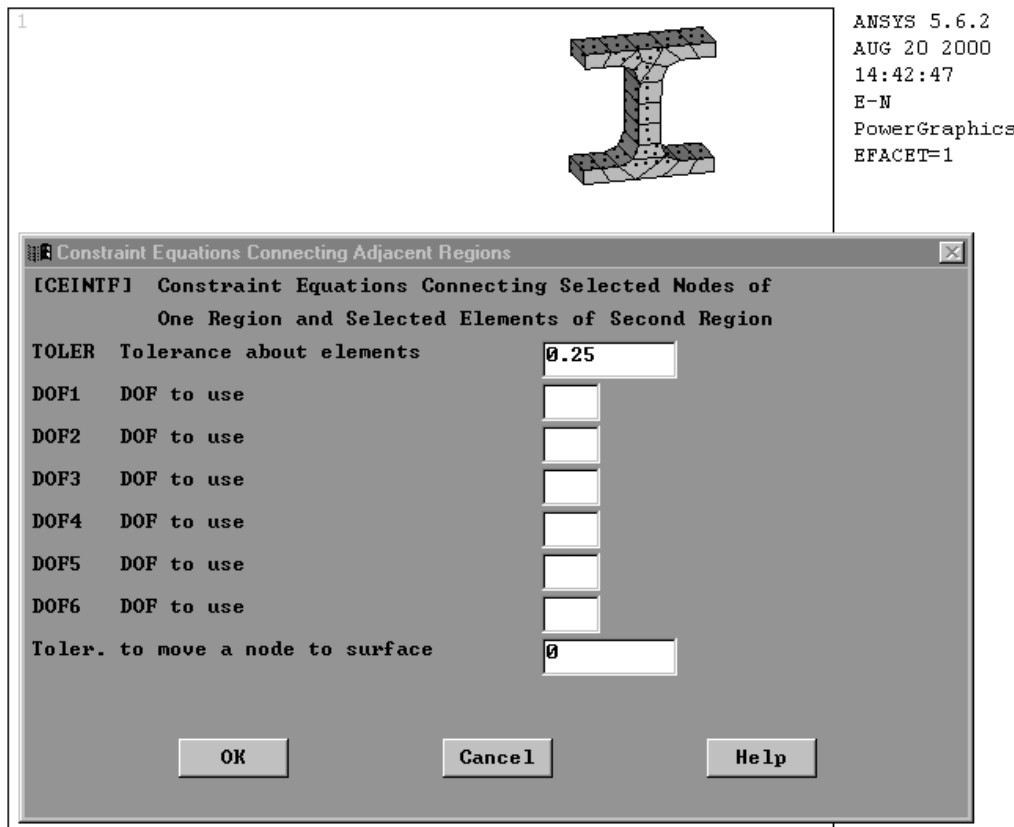
- Select Elements from Coarse Mesh

CSI ANSYS Tip of the Week

Create Constraint Equations at interface

ANSYS Main Menu > Preprocessor > Coupling / Ceqn > Adjacent Regions

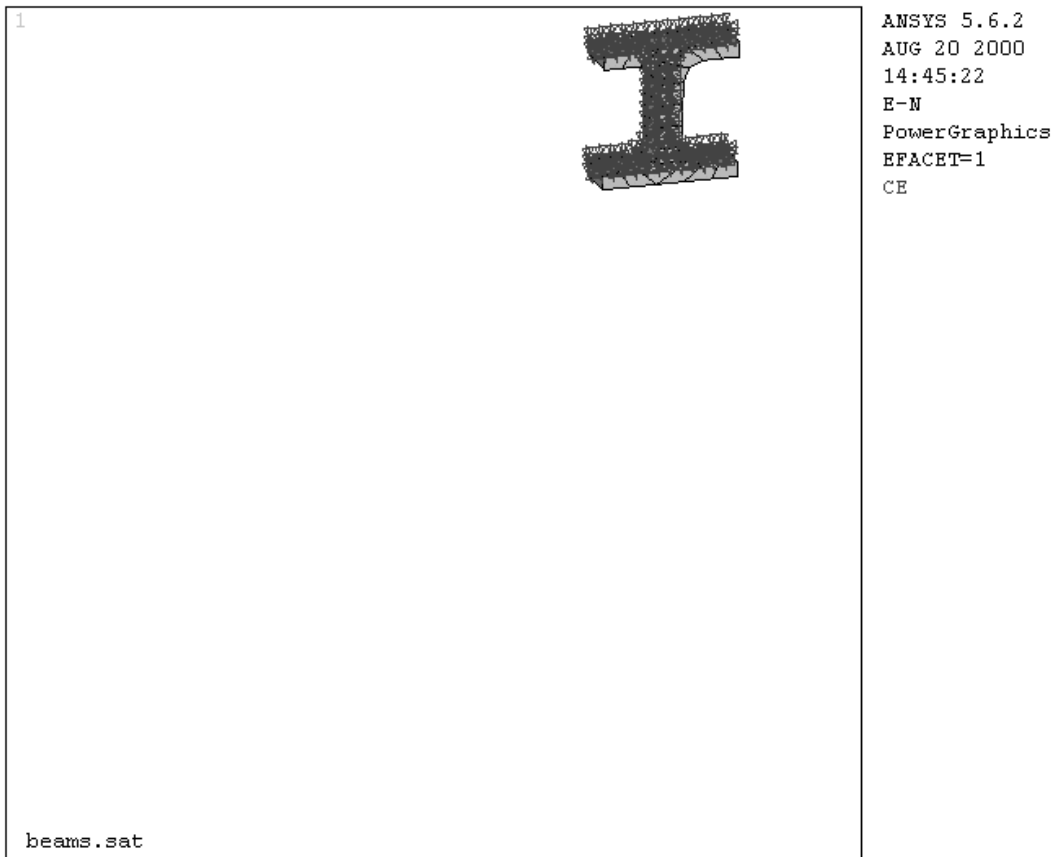
- Accept Defaults and Push OK



CSI ANSYS Tip of the Week

Create Constraint Equations at interface

ANSYS Main Menu > Preprocessor > Coupling / Ceqn > Adjacent Regions

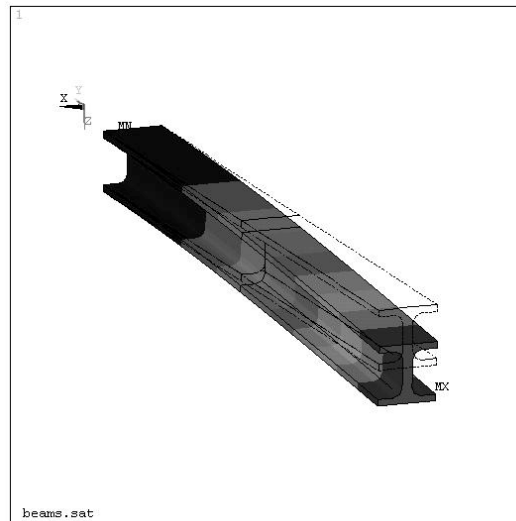


- Constraint Equations have been created.
- Turn symbols on with /pbc,ce,1 or goto Plotctrls > symbols

CSI ANSYS Tip of the Week

Compare Tip Displacements

- From Timoshenko tip deflection = $PL^3/(3EI) = .01949''$
- Continuous mesh tip deflection = $.01949''$
- Bonded Contact (4 iterations) tip deflection = $.01949''$
- Linear Bonded Contact (1 iteration) tip deflection = $.01949''$
- Constraint Equation tip deflection = $.01947''$

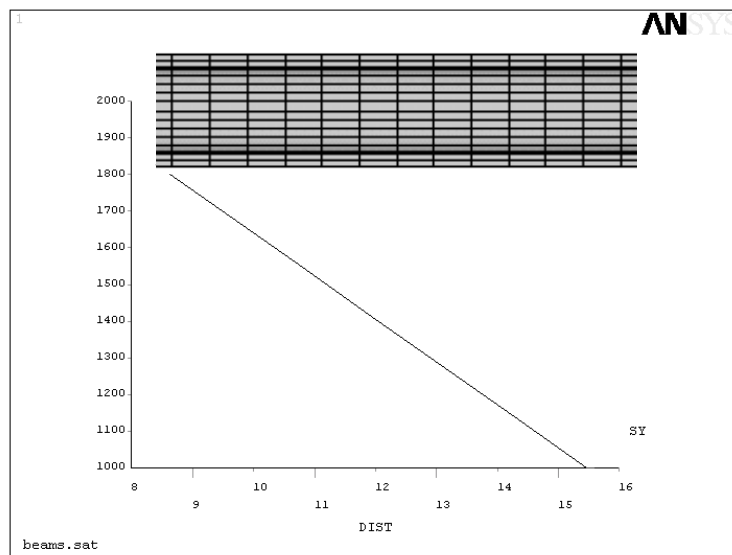


CSI ANSYS Tip of the Week

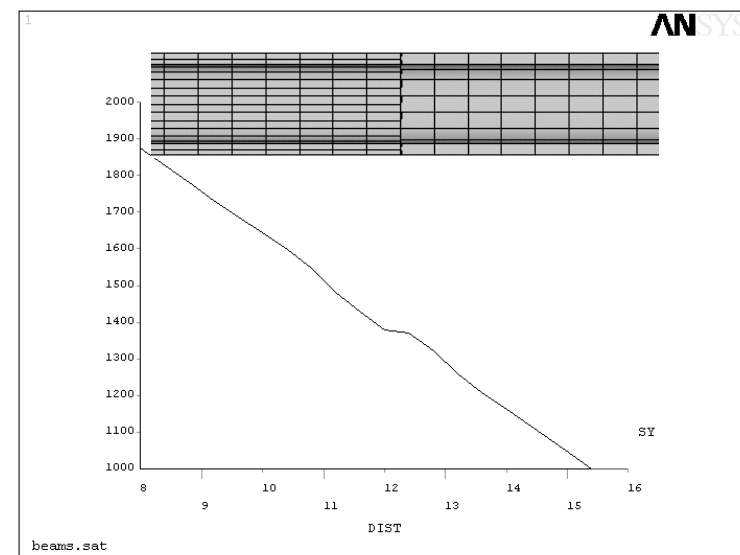
Compare Stress across interface

Using path plot, plot top side beam stress across interface

Continuous Mesh



Constraint Equations

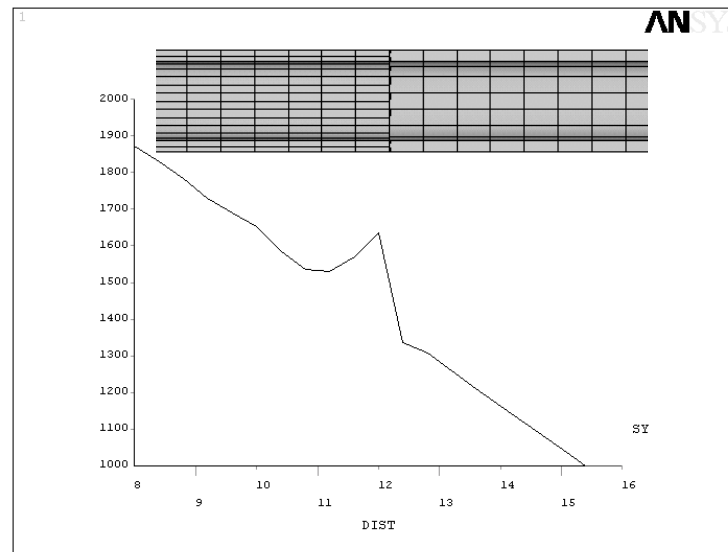


CSI ANSYS Tip of the Week

Compare Stress across interface

- Using path plot, plot top side beam stress across interface

Bonded Contact



CSI ANSYS Tip of the Week

Compare Natural Frequencies

| <u>Mode</u> | <u>Continuous Mesh</u> | <u>Constraint Equations</u> | <u>Bonded Contact</u> |
|-------------|------------------------|-----------------------------|-----------------------|
| 1 | 84.46 | 84.46 | 84.43 |
| 2 | 139.83 | 139.83 | 139.79 |
| 3 | 313.15 | 313.15 | 313.11 |
| 4 | 518.17 | 518.17 | 517.46 |
| 5 | 803.77 | 803.77 | 802.78 |
| 6 | 1020.20 | 1020.20 | 1019.94 |

CSI ANSYS Tip of the Week