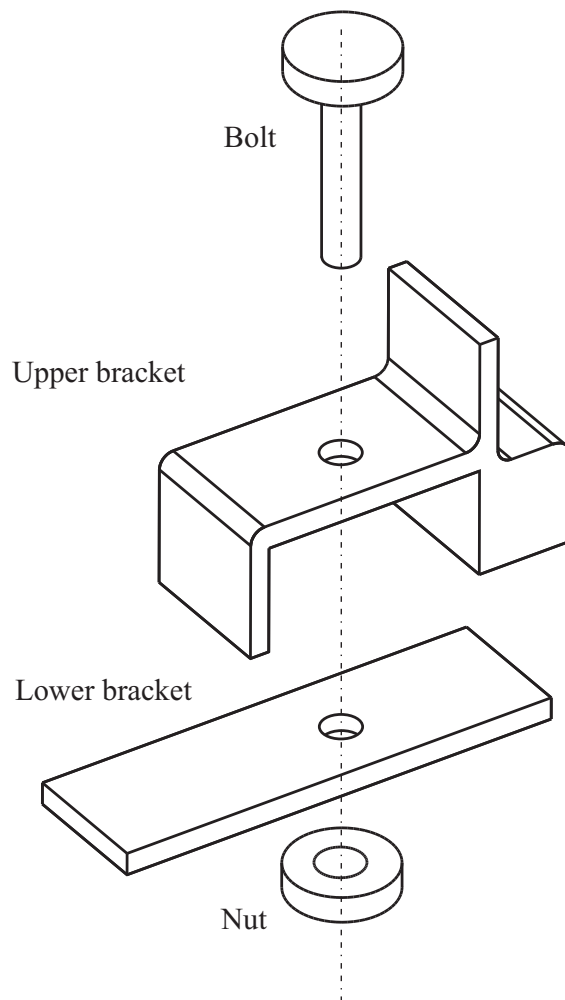


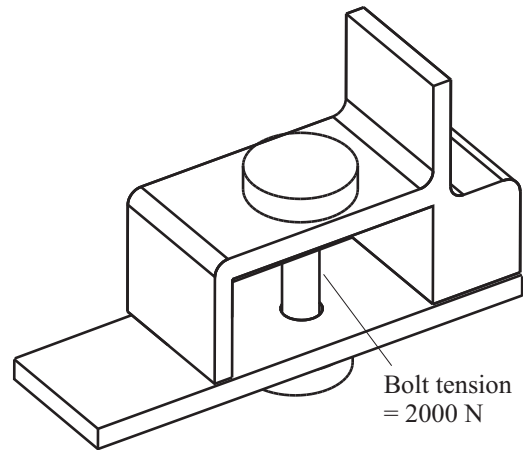
Problem description

A bracket assemblage is shown below in an exploded view:

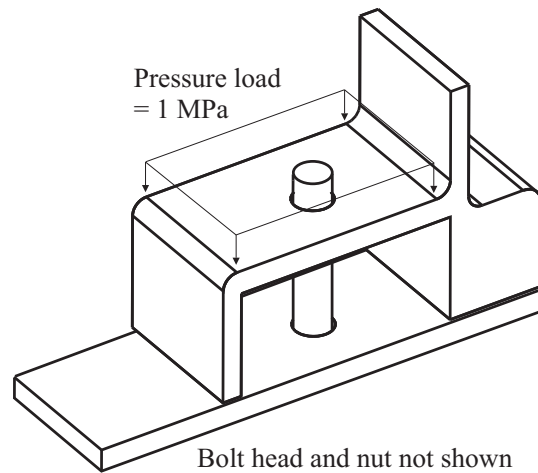


The two parts of the bracket are bolted together. After the parts are bolted together, the tension in the bolt is 2000 N and the assemblage appears as shown on the next page.

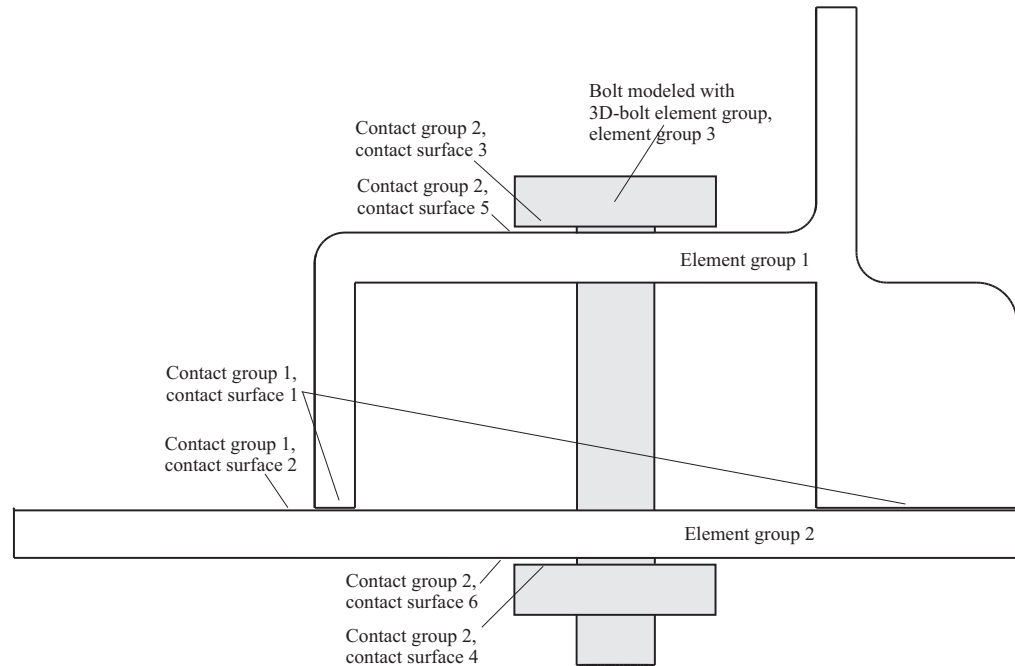
Problem 44: Analysis of a bracket assemblage with a bolt



Finally a pressure load of 1 MPa is applied to the top of the upper bracket as shown:



In this model, the bolt / nut combination is modeled using a 3D-bolt element group. The bolt element group and bracket element groups are allowed to come into contact, and contact surfaces are used to model the contact.



The bolt pretension is directly specified and is applied to the model in a bolt loading step.

In this problem solution, we will demonstrate the following topics that have not been presented in previous problems:

- Using 3D-bolt elements
- Displaying the contact gap in a band plot

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

Note that you must have an ADINA-M/PS license to do this problem.

This problem cannot be solved with the 900 nodes version of the ADINA System because the 900 nodes version of the ADINA System does not include ADINA-M/PS.

Much of the input for this problem is stored in file `prob44_1.in`. You need to copy file `prob44_1.in` from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Problem 44: Analysis of a bracket assemblage with a bolt

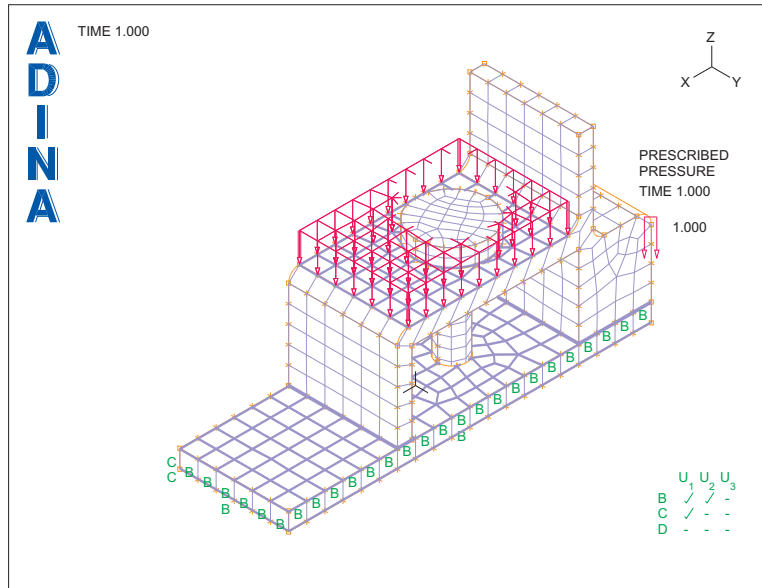
Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA Structures.

Model definition

We have prepared a batch file (prob44_1.in) which contains all of the model definition except for the bolt element definition.


Choose File→Open Batch, navigate to the working directory or folder, select the file prob44_1.in and click Open. The graphics window should look something like this:

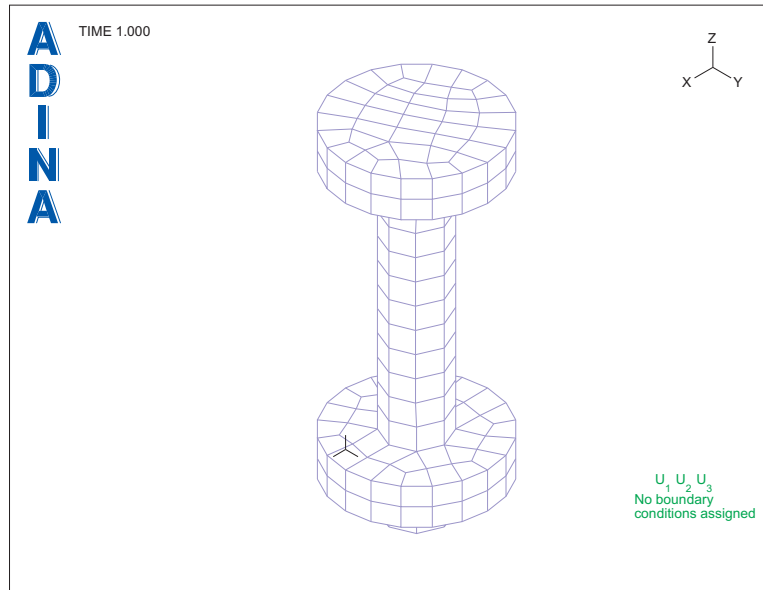


Defining the 3D-bolt element group

Element group 3 will be the 3D-bolt element group. In the Model Tree, expand the Zone entry, right-click on 12. EG3 and choose Display. The graphics window should look something like the figure on the next page.


It is seen that element group 3 contains both the bolt and nut.


Bolt element group: Click the Element Groups icon , make sure that the Group Number is 3 and set the Element Option to Bolt. In the Bolt box, set the Bolt # to 1, the Load to 1.0 and click OK.



Bolt loading: Choose Model→Bolt→Bolt Options, set the Bolt Loading Sequence Table to Yes and click the Bolt Table ... button. Add bolt table 1, enter (1, 1, 2000, Yes) in the first row of the table and click OK twice to close both dialog boxes.

Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

First click the Save icon  and save the database to file prob44. Click the Data

File/Solution icon , set the file name to prob44, make sure that the Run Solution button is checked and click Save.

In the Log Window, note the following text:

```
3D bolt group 3, bolt number = 1
Automatic bolt-plane calculation using inertial properties
Volume = 4.70302E+03
Min inertia = 1.63054E+05
Bolt direction = ( 0.00000E+00, 0.00000E+00, 1.00000E+00)
Bolt plane point = (-1.00000E+01, 1.53553E-15, 1.26637E+01)
Bolt length, cross-sectional area = 4.75000E+01, 4.30037E+01
```

(Your numbers might be slightly different than these numbers because free meshing produces different meshes on different platforms.)

Problem 44: Analysis of a bracket assemblage with a bolt

This text can be used to verify that the bolt is defined correctly. For example, the bolt direction for bolt number 1 is (0,0,1), which corresponds to the bolt axis. See the ADINA Structures Theory and Modeling Guide for more details.


In the ADINA Structures dialog box, in the Message window, notice that there is a line

```
Bolt iterations: step number =      1      bolt force factor =  2.0000000E+03
```


before the line


```
Step number =      1      step size = 1.0000000E+00      time = 1.0000000E+00
```

We see that, although there is only one solution step specified in the input, ADINA Structures performs bolt iterations before that solution step, in order to obtain the solution corresponding only to the bolt tension of 2000 N (before the pressure load is applied).


Close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob44.

Post-processing

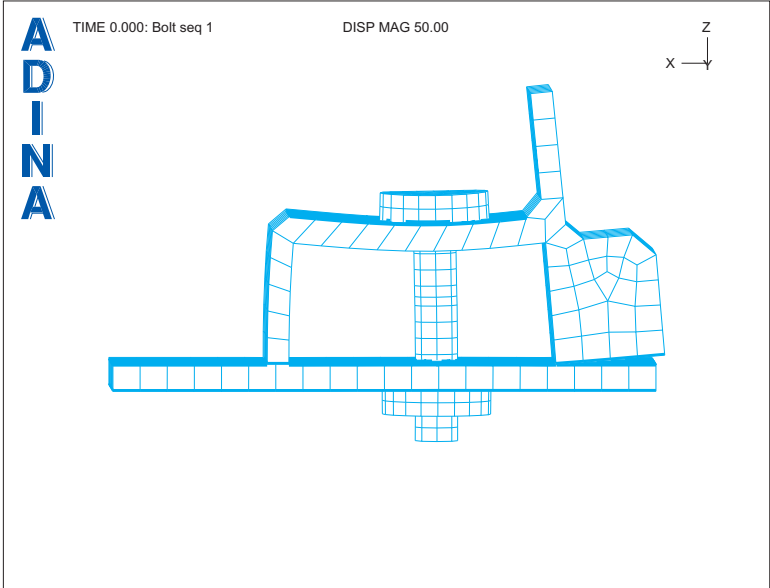
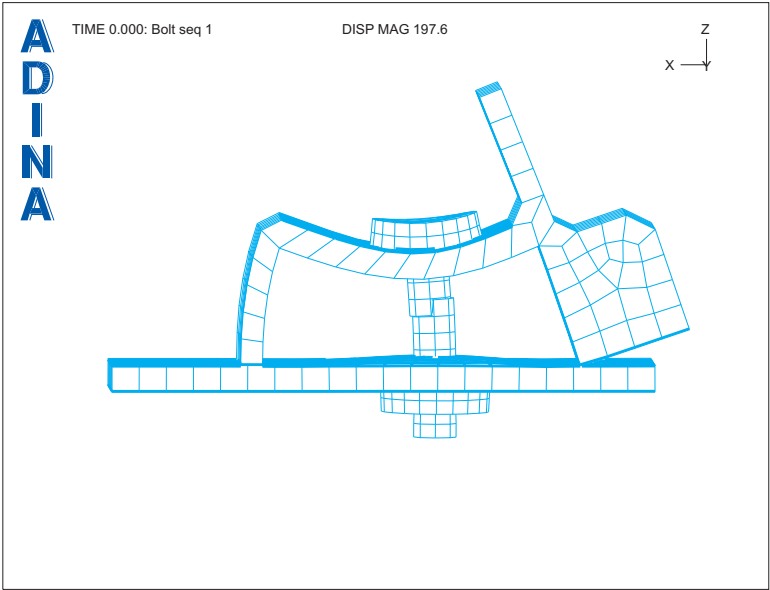
Bolt pretension: Click the Previous Solution icon  to display the solution for the bolt sequence step. Since no load other than the bolt pretension is applied in this step, this solution shows the effect of the bolt pretension.

Click the Scale Displacements icon  and use the mouse to rotate and scale the mesh plot until the graphics window looks something like the top figure on the next page.



The bolt pretension is seen to pull the two parts of the bracket assemblage together, and the two parts of the bracket assemblage are in contact. However, the displacements are magnified so much that the elements in the bolt group appear to move into each other. Click the Modify

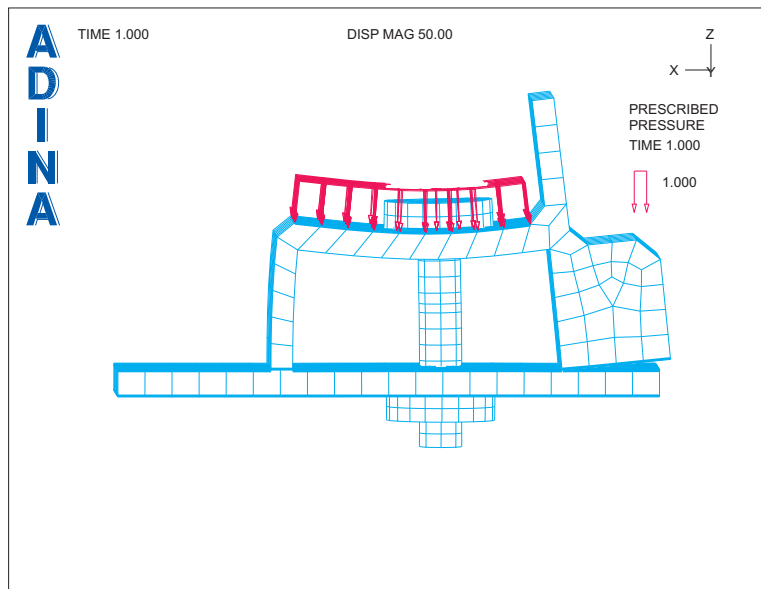
Mesh Plot icon , click the Model Depiction button, set the Magnification Factor to 50 and click OK twice to close both dialog boxes. The graphics window should look something like the bottom figure on the next page.

Problem 44: Analysis of a bracket assemblage with a bolt





Problem 44: Analysis of a bracket assemblage with a bolt

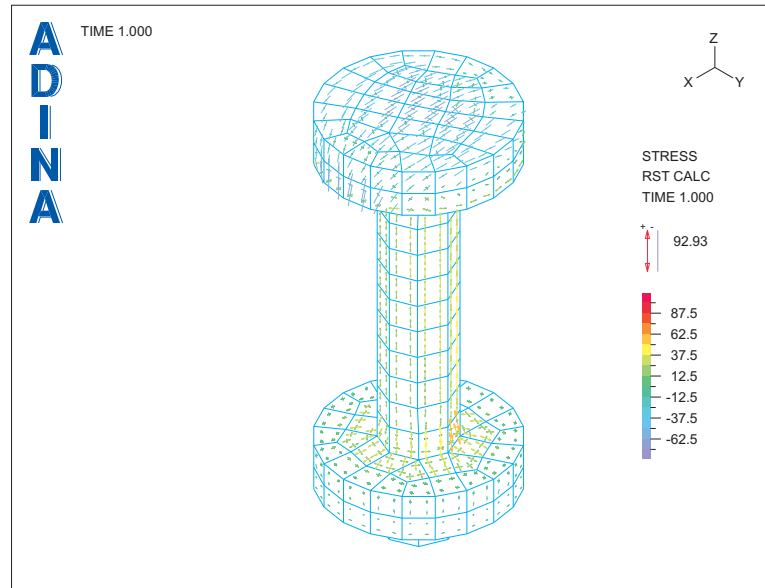
Bolt pretension and applied pressure: Click the Next Solution icon  and the Load Plot icon  to display the solution for time 1.0. The graphics window should look something like this:



The model displacements have increased due to the applied pressure.

Bolt results: Choose List→Value List→Zone, set Variable 1 to (Force: BOLT-FORCE), Variable 2 to (Displacement: BOLT-DISPLACEMENT) and click Apply. The results for "Bolt Sequence 1" are 2.00000E+03 (N), 4.59793E-02 (mm), and the results for "Time 1.00000" are 1.35826E+03, 4.59793E-02. We see that the specified pretension is applied to the bolt for "Bolt Sequence 1", and that the bolt length is shortened by 4.59793E-2 mm in order to achieve this bolt pretension. For "Time 1.00000", the applied pressure load reduces the bolt tension, and the bolt shortening is unchanged. (Your results might be slightly different than ours due to free meshing.)

Let's look at the stresses in element group 3. Click the Clear icon , and, in the Model Tree, expand the Zone entry, right click on 13. EG3 and choose Display. When you click the Quick Vector Plot icon , the graphics window should look something like the figure on the next page.





Now we calculate the axial force in the bolt from the stresses. Click the Cut Surface icon



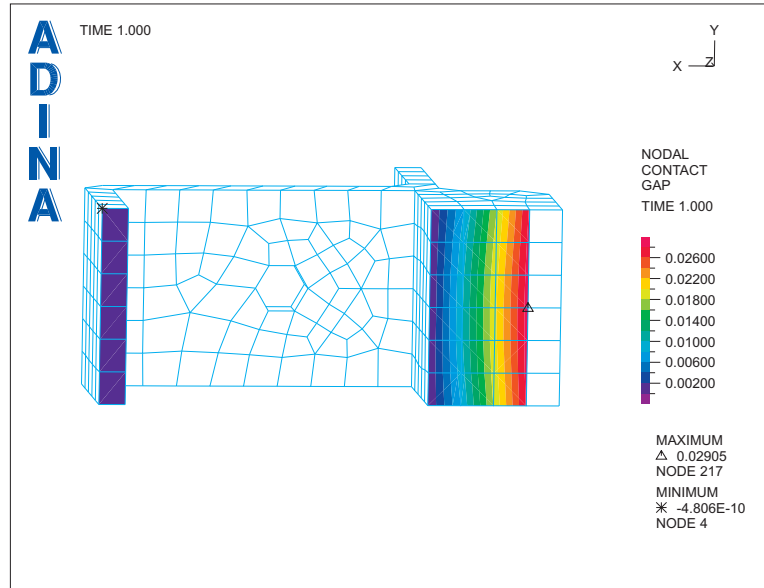
, set the Type to Cutting Plane, set "Defined by" to Z-Plane, set the Coordinate Value to 12 and click OK. The cutting surface corresponds to the bolt cross-section. Choose Definitions→Model Point (Special)→Mesh Integration, add point BOLT, make sure that the Integration Type is Integral and click OK. Choose List→Value List→Model Point, make sure that the Model Point Name is BOLT, set Variable 1 to (Stress: STRESS-ZZ) and click Apply. The value of STRESS-ZZ is first $1.99965E+03$ (N/mm²), then becomes $1.35795E+03$. Since model point BOLT represents the integral of the listed variable, these values correspond to axial forces, and these values are very close to the bolt forces obtained earlier.

Displaying the contact gap in a band plot

Now we look at the contact between the upper and lower brackets. Click the Clear icon  and, in the Model Tree, expand the Zone entry, right click on 11. EG1 and choose Display.

Then click the Create Band Plot icon , set the Band Plot Variable to (Contact: NODAL_CONTACT_GAP) and click OK. Use the mouse to rotate the mesh plot until the graphics window looks something like the figure on the next page.

Problem 44: Analysis of a bracket assemblage with a bolt



Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.